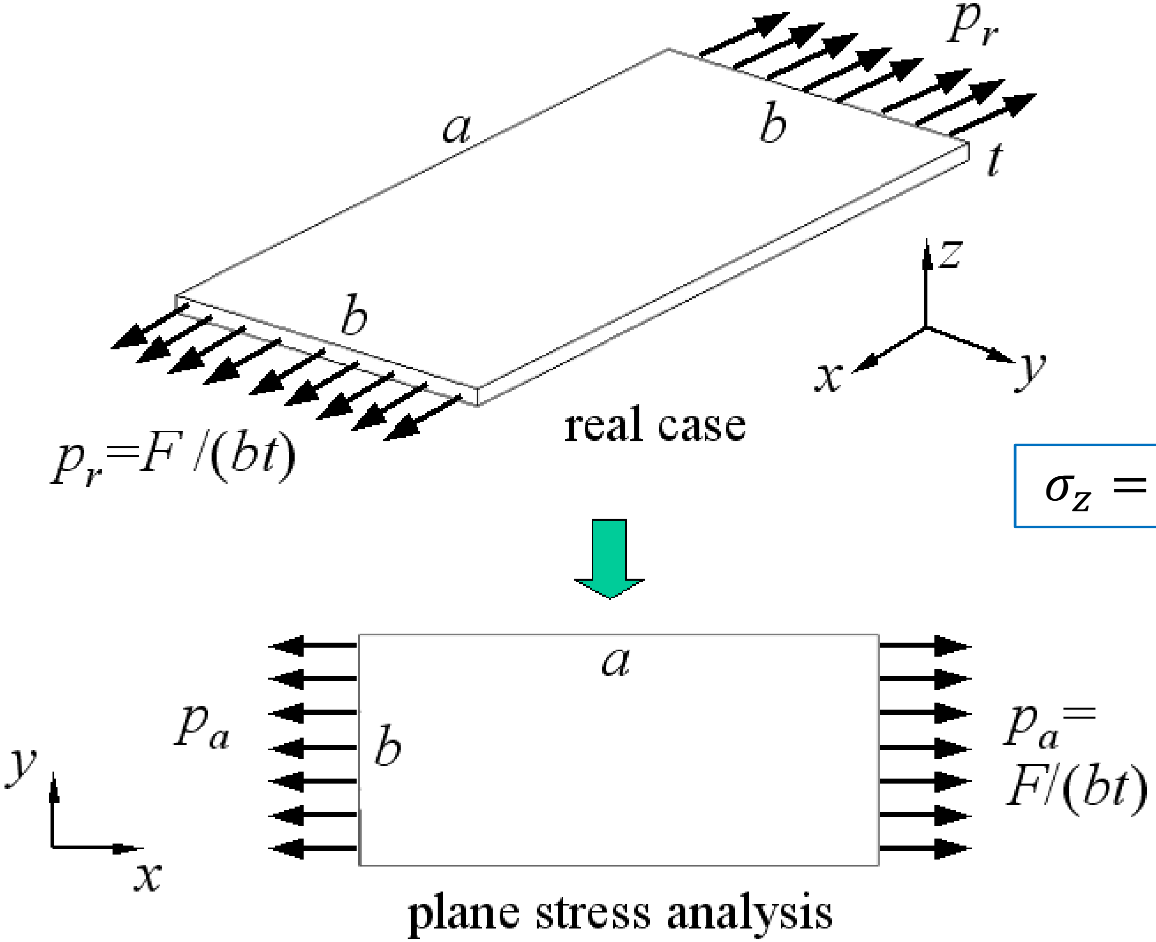
05

Advance Analysis

多步驟分析 / 平面應變&平面應力
非線性分析(接觸&材料設定) / 結構最佳化

2D Plane Problem

- **Plane stress, plane strain, axisymmetric, plane stress w/thk**
- **Plane stress**

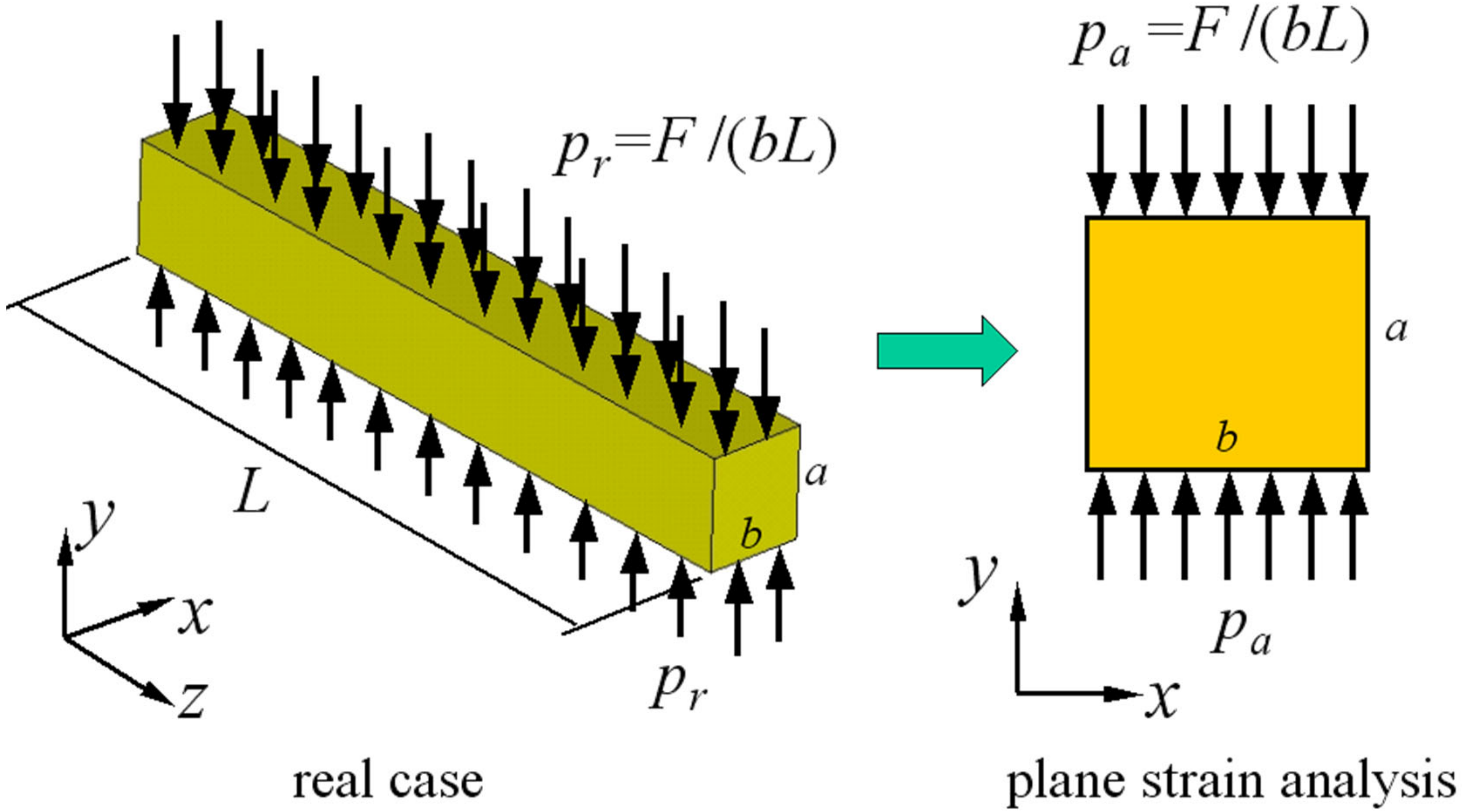


$$\sigma_z = \tau_{yz} = \tau_{xz} = 0$$

2D Plane Problem

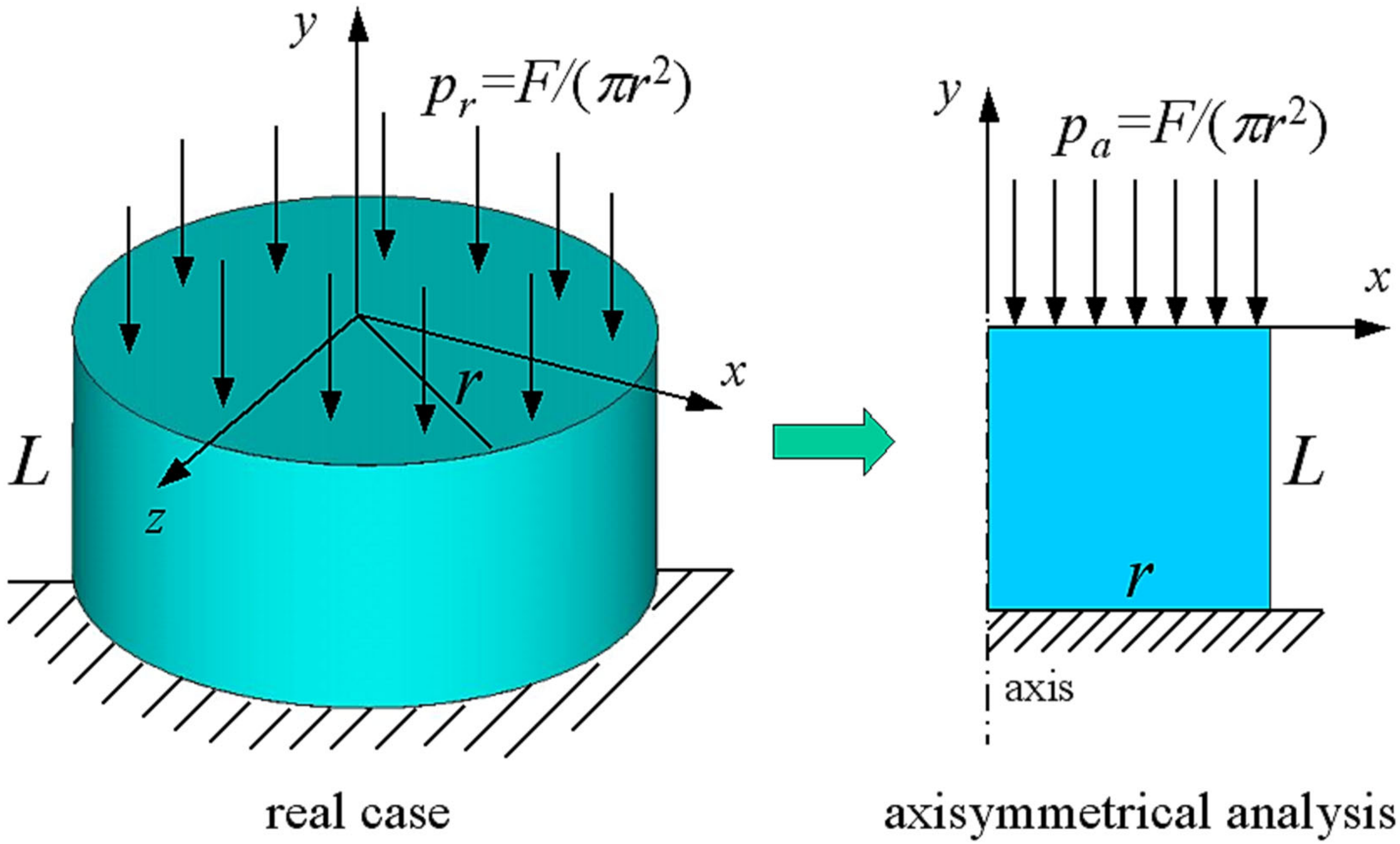
- **Plane strain**
  - **The cross-sections are along the thickness**

$$\varepsilon_z = \gamma_{yz} = \gamma_{xz} = 0$$



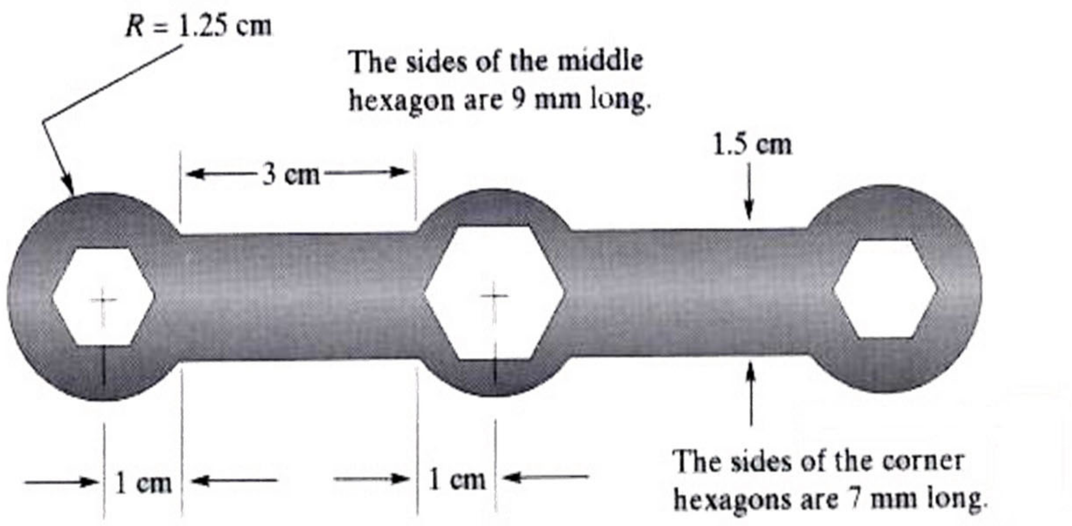
2D Plane Problem

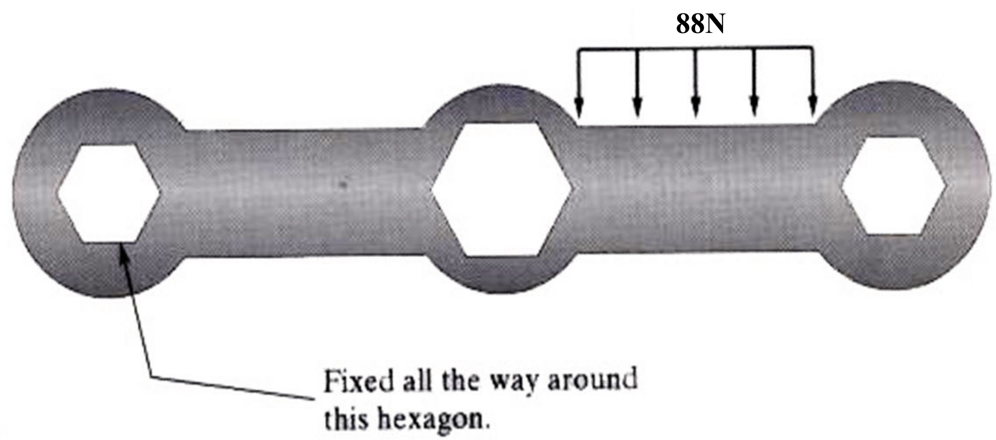
- **Axisymmetric**
  - **The cross-sections are along the radial directions**



Plane Stress – Ex.10

The bicycle wrench shown in figure is made of steel with 3 mm thick. Determine the von-Mises stresses under the given distributed load and boundary conditions.

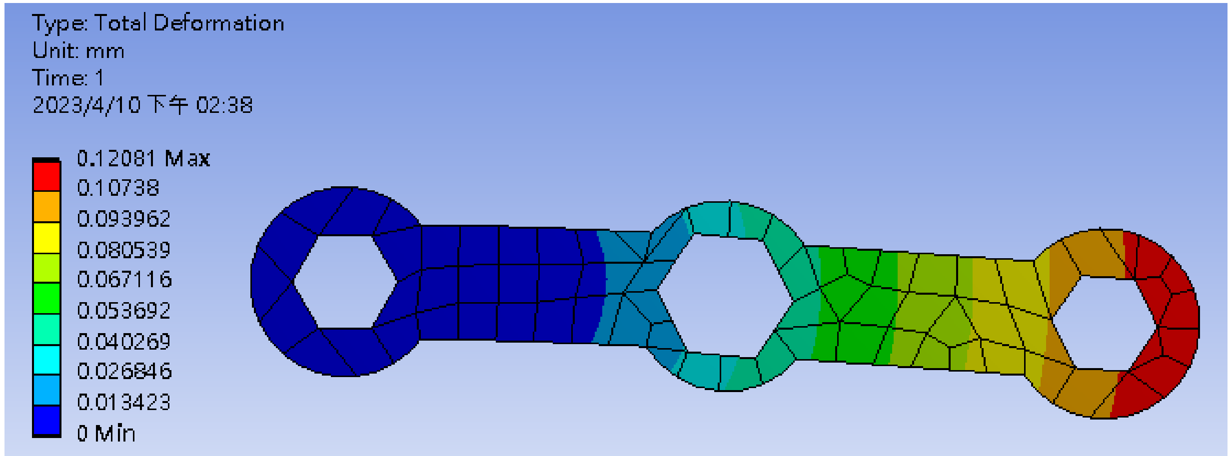




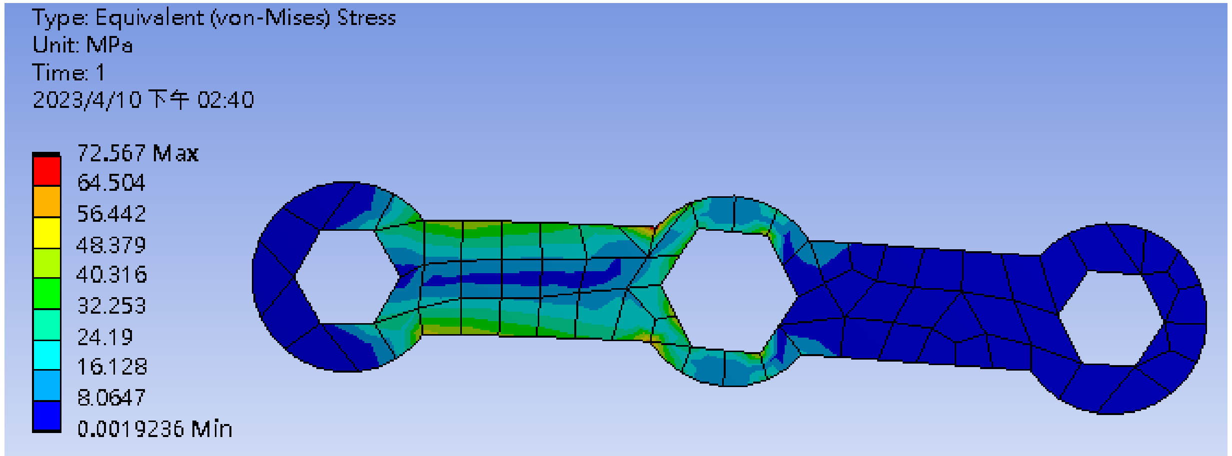
Plane Stress – Ex.10

學習目標
- 2D分析設定
- 單位轉換
- Polygon

The bicycle wrench shown in figure is made of steel with 3 mm thick. Determine the von-Mises stresses under the given distributed load and boundary conditions.



變形量
Total Deformation

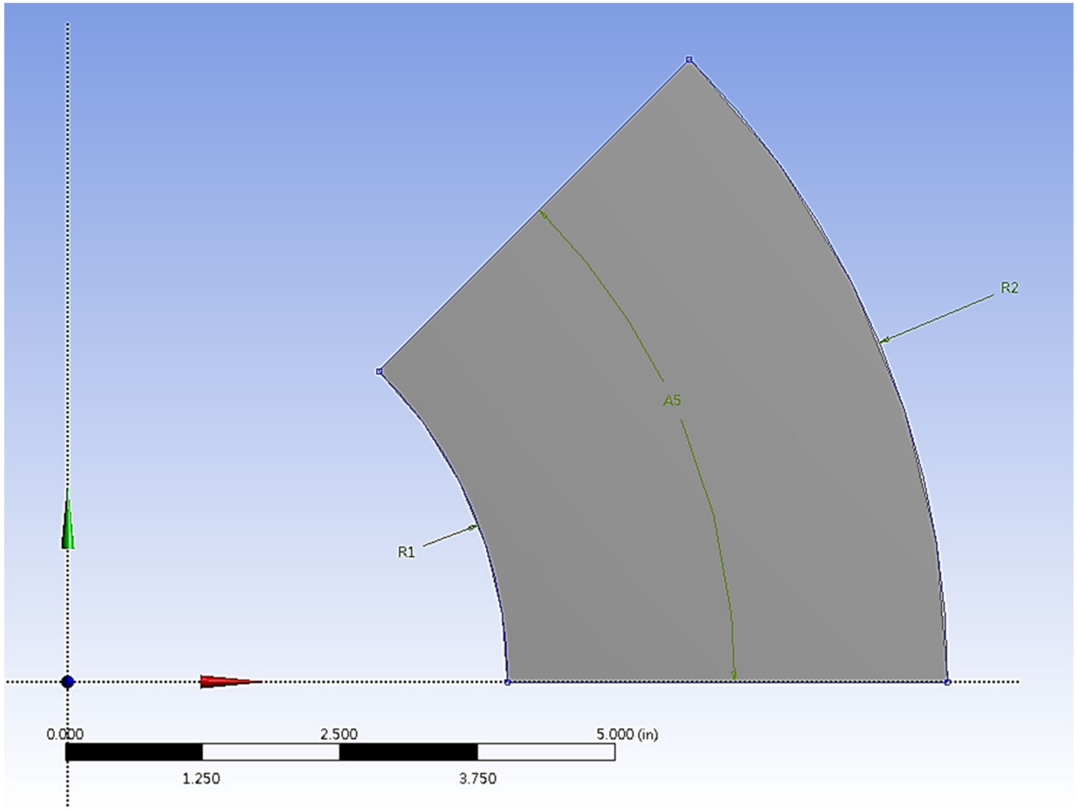


等效應力
Equivalent Stress

Plane Strain – Ex.11

Stresses in a Long Cylinder

A long thick-walled cylinder (with inner radius = 4 in. and outer radius = 8 in.) made of steel is initially subjected to an constant pressure (of 30000 psi). The pressure is then removed and the cylinder is subjected to a constant rotation (60000 rpm) about its center line. Find the radial displacement, radial stress, and hoop stress at the two load steps. (Face mesh size = 0.25 in)



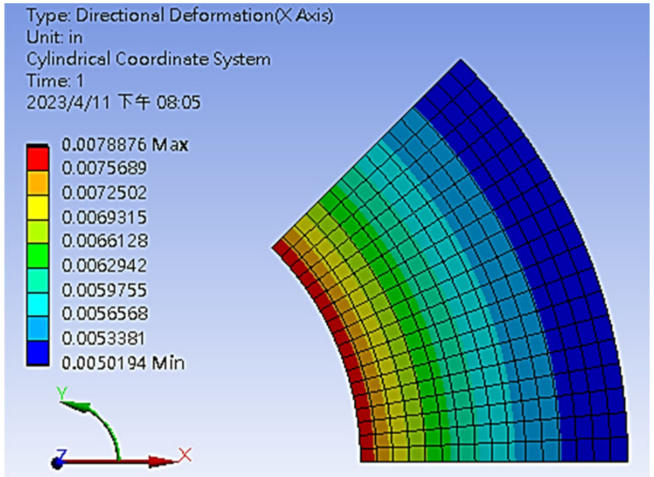
Plane Strain – Ex.11-1

學習目標
- 圓柱座標轉換
- *Face Meshing*
- 角速度設定
- 多步驟分析

Stresses in a Long Cylinder

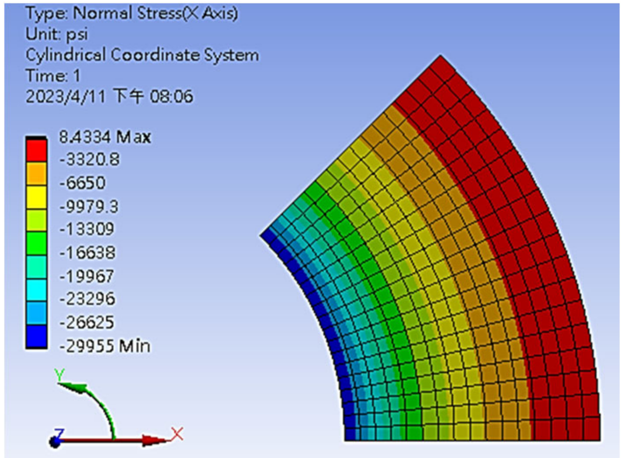
A long thick-walled cylinder (with inner radius = 4 in. and outer radius = 8 in.) made of steel is initially subjected to an constant pressure (of 30000 psi). The pressure is then removed and the cylinder is subjected to a constant rotation (60000 rpm) about its center line. Find the radial displacement, radial stress, and hoop stress at the two load steps. (Face mesh size = 0.25 in)

Step. 1



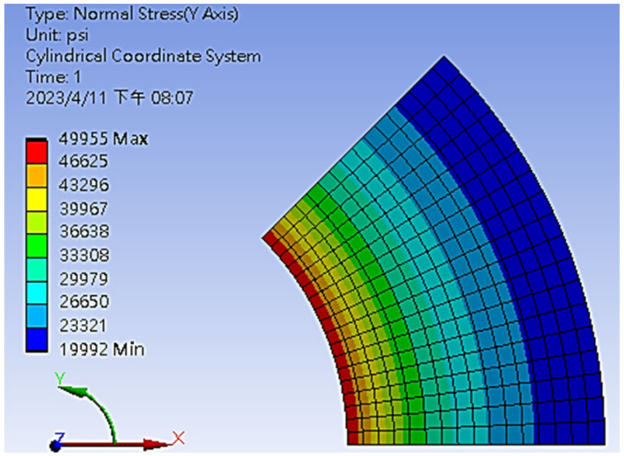
Directional Deformation

Radial displacement



Normal stress

Radial stress



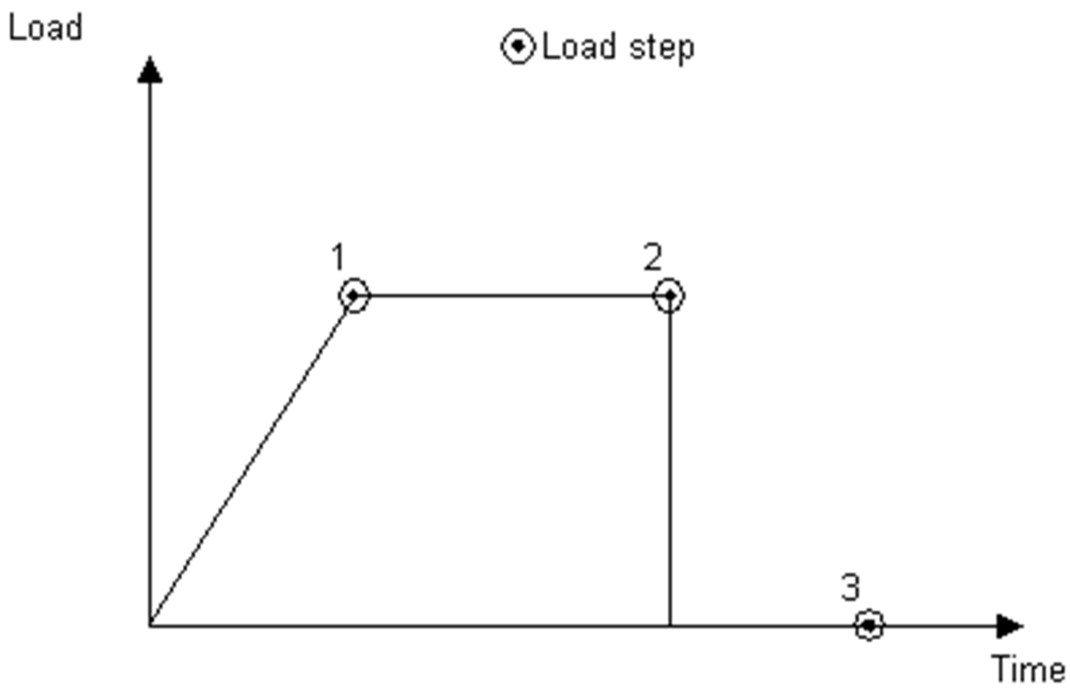
Normal stress

Hook stress

Introduction of ANSYS – Solution

Load Steps

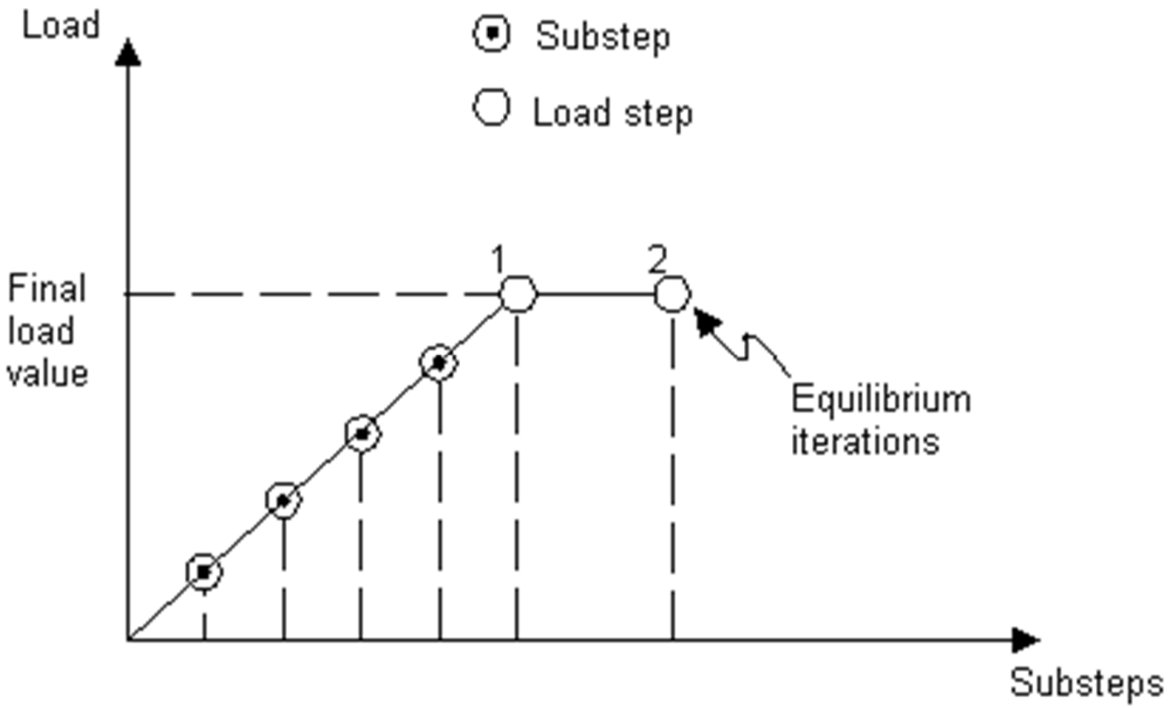
- In a linear static or steady-state analysis, you can use different load steps to apply different sets of loads--wind load in the first load step, gravity load in the second load step, both loads and a different support condition in the third load step, and so on.
- In a transient analysis, multiple load steps apply different segments of the load history curve.



Introduction of ANSYS – Solution

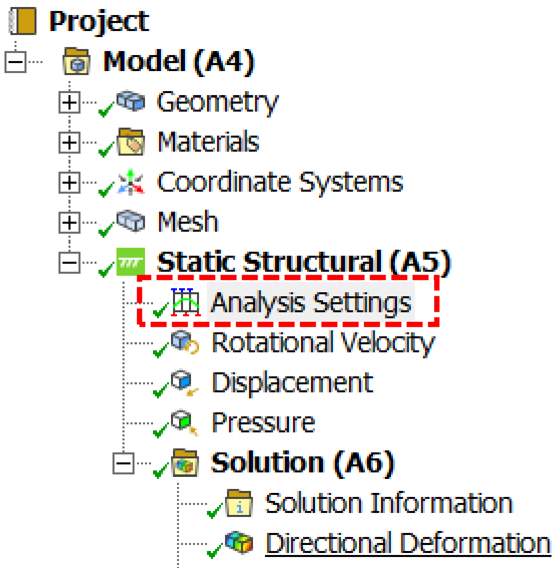
■ Substeps

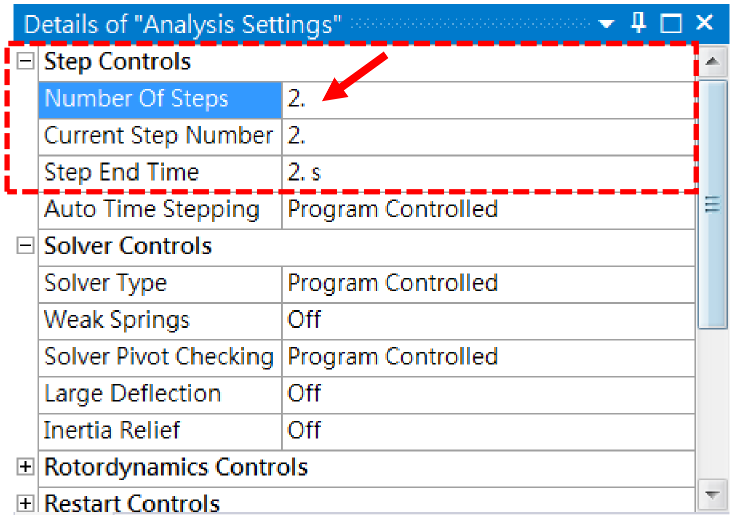
- Substeps are points within a load step at which solutions are calculated. You use them for different reasons:
  - ✓ In a linear or nonlinear transient analysis, use substeps to satisfy transient time integration rules
  - ✓ In a nonlinear static or steady-state analysis, use substeps to apply the loads gradually so that an accurate solution can be obtained.

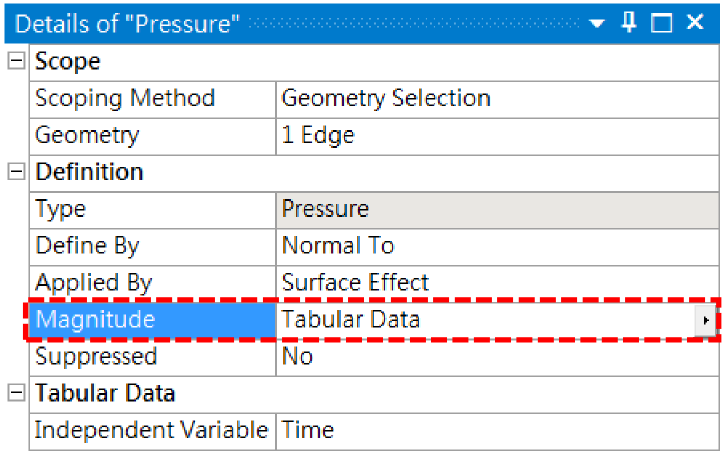


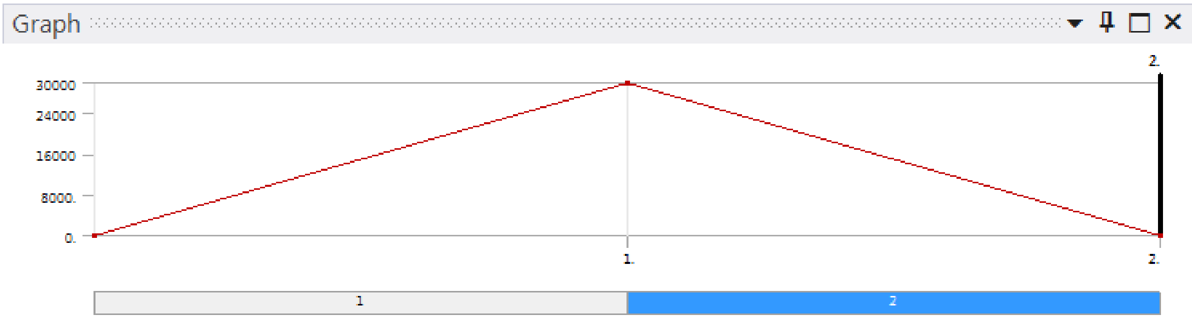
Introduction of ANSYS – Solution

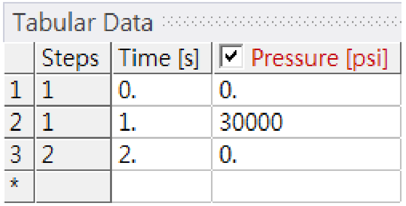
■ Step Controls











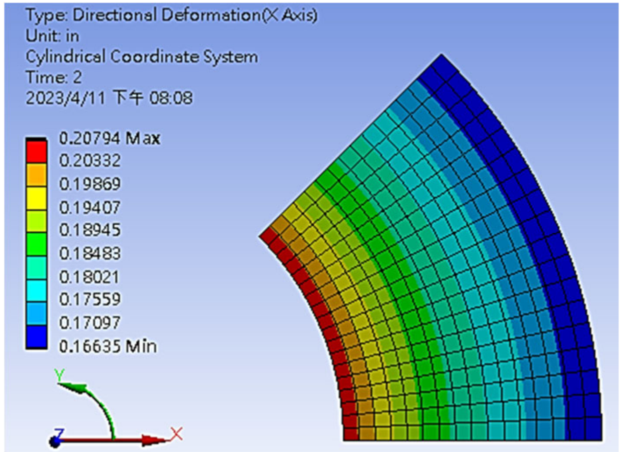
Plane Strain – Ex.11-2

學習目標
- 圓柱座標轉換
- *Face Meshing*
- 角速度設定
- 多步驟分析

Stresses in a Long Cylinder

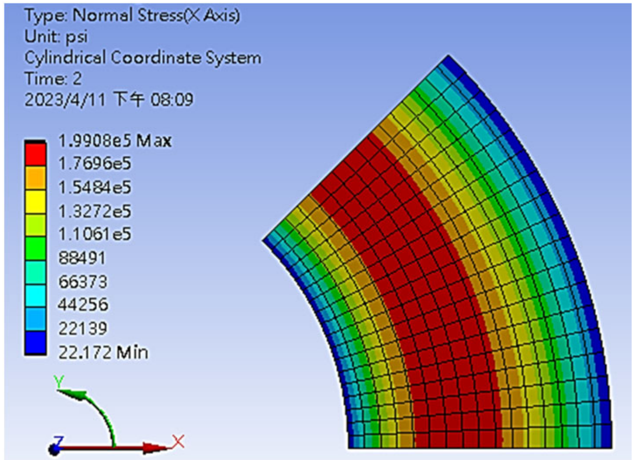
A long thick-walled cylinder (with inner radius = 4 in. and outer radius = 8 in.) made of steel is initially subjected to an constant pressure (of 30000 psi). The pressure is then removed and the cylinder is subjected to a constant rotation (60000 rpm) about its center line. Find the radial displacement, radial stress, and hoop stress at the two load steps. (Face mesh size = 0.25 in)

Step. 2



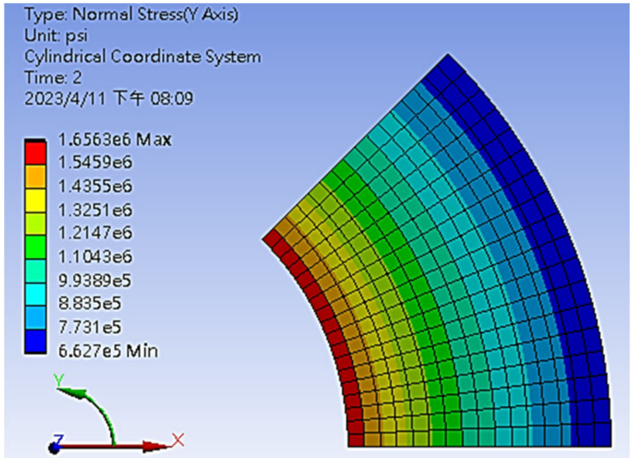
Directional Deformation

Radial displacement



Normal stress

Radial stress

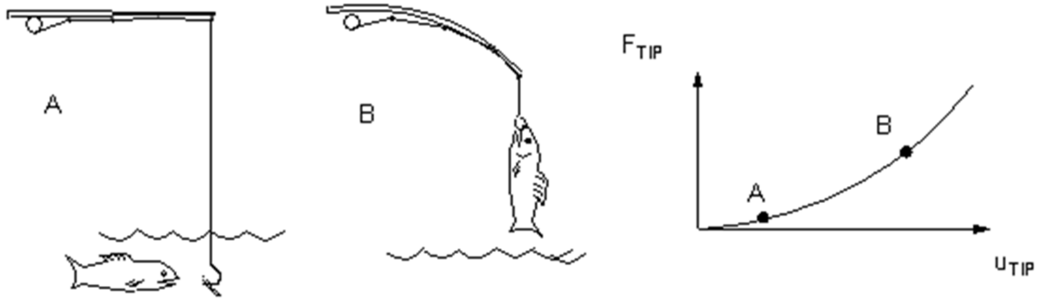


Normal stress

Hook stress

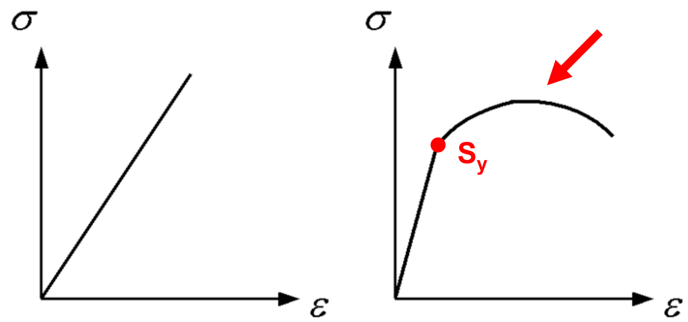
Nonlinear Structural Analysis

- 有限元素法中，常見的非線性(Nonlinear)問題主要可分為三類
  - 幾何非線性(Geometric nonlinearity)
    - ✓ 大變形、大位移



幾何非線性分析需開啟**Large Deflection**，當節點位置產生變化，將重新計算其剛度矩陣

  - 材料非線性(Material nonlinearity)
    - ✓ 非線性行為之材料特性，即應力應變曲線非線性關係

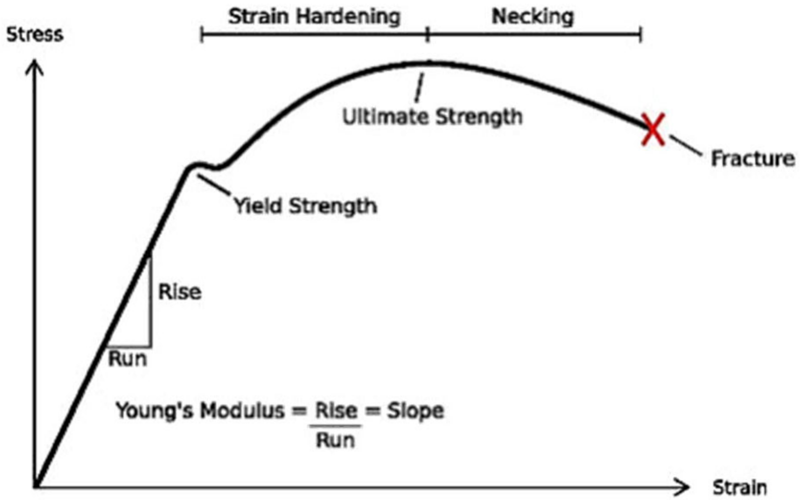


設定**Engineering Data**，指定為非線性材料性質

  - 邊界非線性(Contact analysis)
    - ✓ 非線性之接觸行為，如: **Frictional**、**Frictionless**、**Rough**

Material Nonlinearity

- 若應力分析範圍大於降伏強度，因應力與應變呈曲線關係，便成為**材料非線性分析**，這和遵循虎克定律之線彈性分析完全不同
- 在結構分析中，若負荷過程的應力包含了彈性部分與**超過強伏點的塑性部分**，可稱為**彈塑性(elastoplastic)分析**

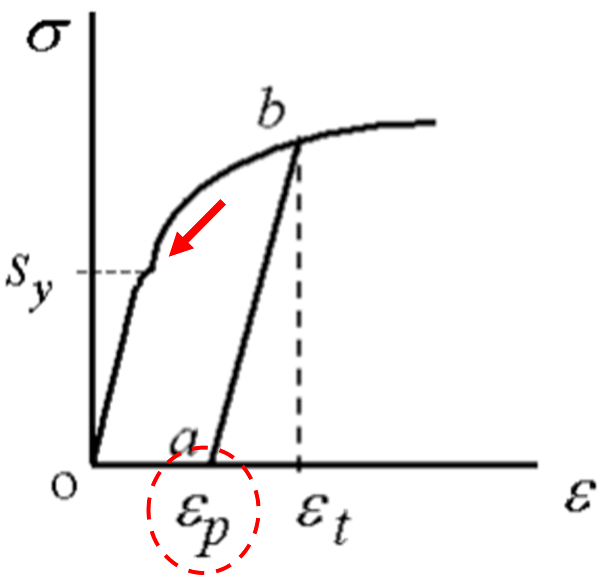


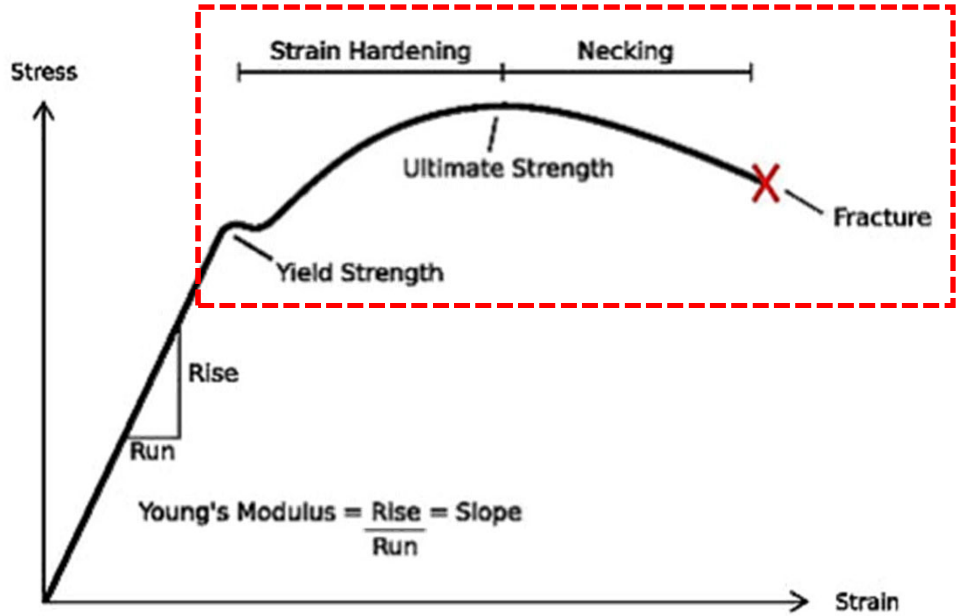
應力應變曲線

材料行為	變形型式	與應變率 (或時間) 之關係 (Rate dependency)	材料性質分類	材料模型 (Material laws)
線性	彈性	無關 (Rate-independent)	Linear elastic	Hooke's law
非線性	彈性	無關 (Rate-independent)	Hyperelastic	Mooney-Rivlin
				Arruda-Boyce
				Blatz-Ko
			Multilinear elastic	Multilinear elastic
		相關 (Rate-dependent)	Viscoelasticity	Viscoelasticity
	非彈性	無關 (Rate-independent)	Isotropic hardening plasticity	Bilinear isotropic
				Multilinear isotropic
				Voce's nonlinear isotropic
				Anisotropic
			Kinematic hardening plasticity	Bilinear kinematic
				Multilinear kinematic
				Chaboche
			Combined kinematic and isotropic hardening plasticity	Chaboche and bilinear isotropic
				Chaboche and multilinear isotropic
				Chaboche and Voce's
			Pressure-dependent plasticity	Druger-Prager
		相關 (Rate-dependent)	Viscoplasticity	Creep
				Anand
			Combined creep and isotropic hardening plasticity	Creep and bilinear isotropic
				Creep and multilinear isotropic
				Creep and Voce's

Material Nonlinearity – 彈塑性分析

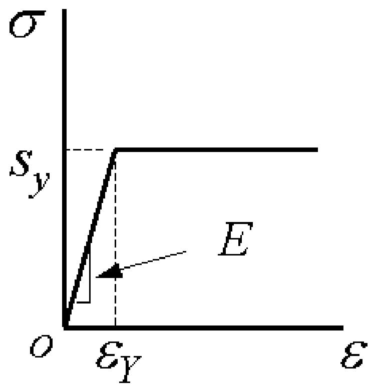
- 塑性力學(plasticity)所研究的對象是延性材料在降伏後(應力>S_y)所發生的塑性變形，屬於材料非線性分析的一種，由圖可看出結構受力過程包括了彈性與塑性變形，這類分析稱為彈塑性(elastoplastic)分析
- 材料達到塑性後，主要現象為產生應變硬化(strain hardening)與永久變形
- 若材料達到塑性變形(應力>S_y)，當外力移除後將留下永久變形，圖中之ε_p即為移除外力後所殘留之塑性應變(plastic strain)，且在許多情況下會有殘留應力(residual stress)存在於材料內部



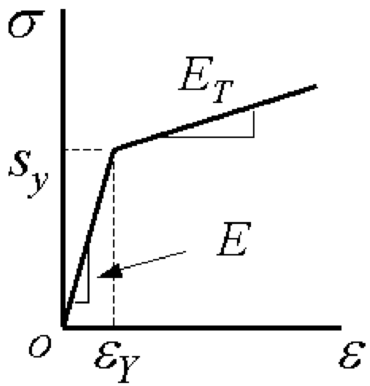


Material Nonlinearity – 彈塑性分析

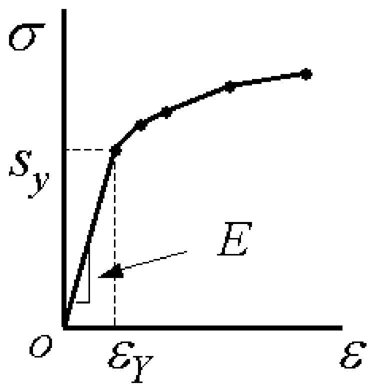
- 相對的在數學解析或有限元素分析上，有以下幾種應力應變曲線模式：
  1. 彈性-完全塑性(elastic perfectly-plastic)
  2. 雙線性(bi-linear)
  3. 多線段(multi-linear)
  4. 塑性曲線



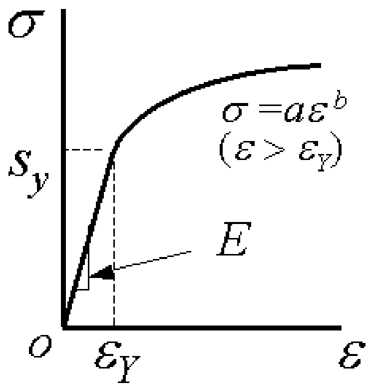
彈性-完全塑性



雙線性

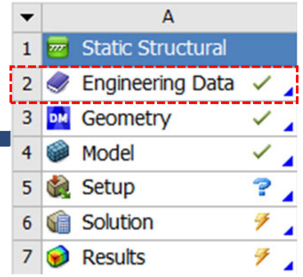


多線段

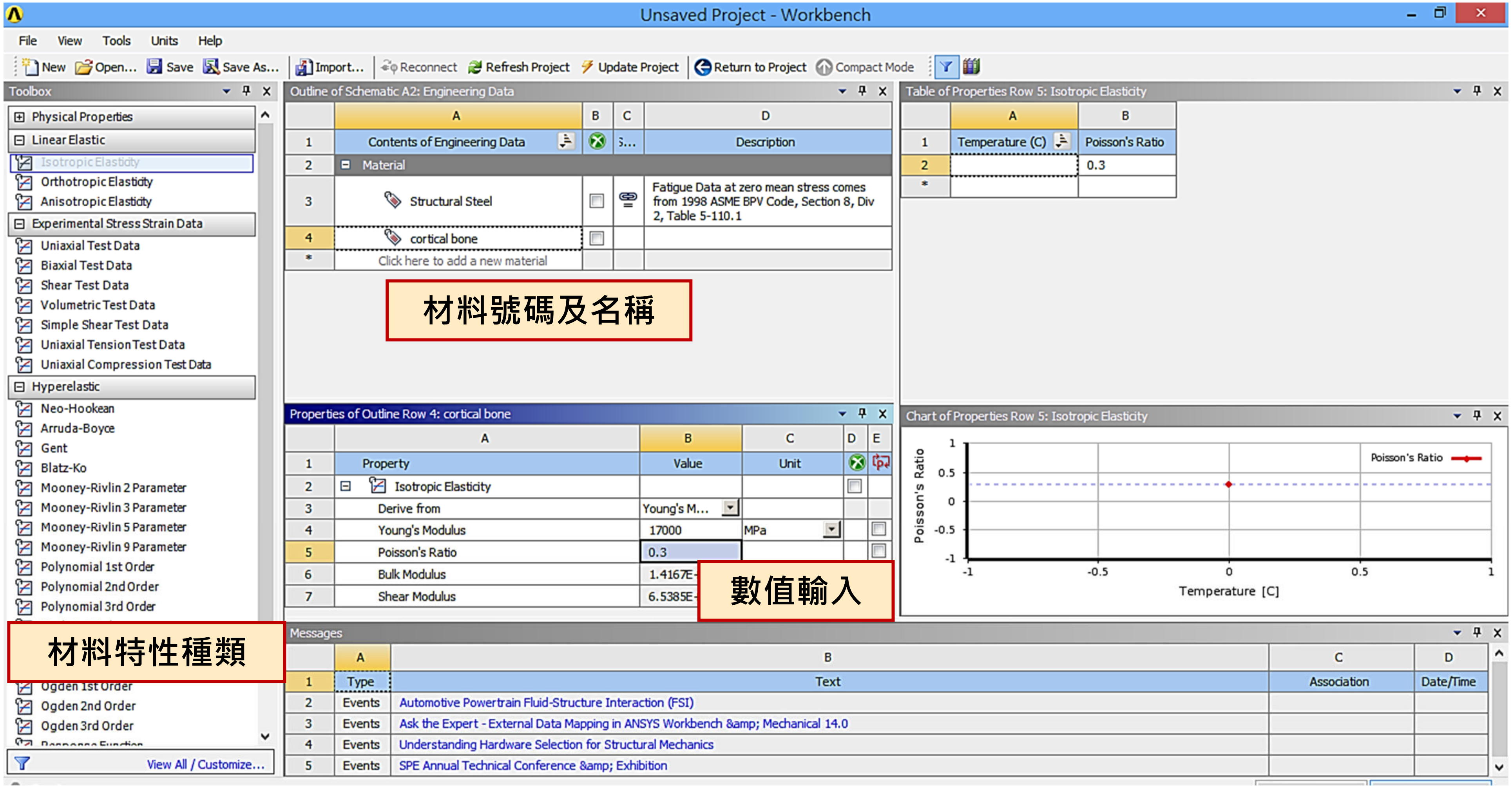


塑性曲線

Introduction of ANSYS Workbench



■ **Engineering Data(工程資料)**

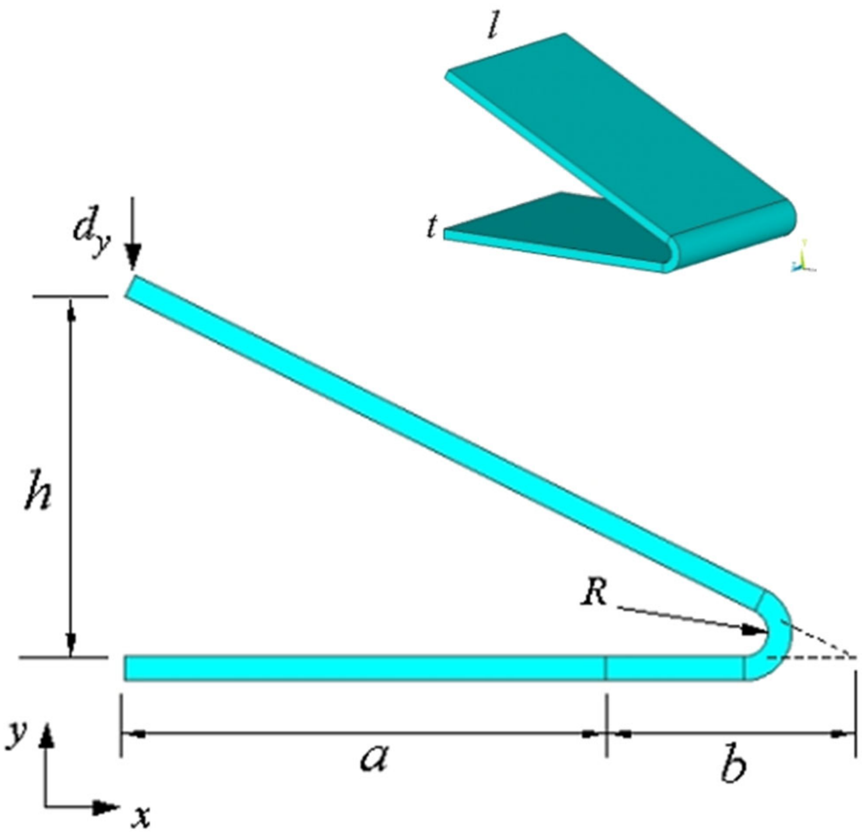


Nonlinear – Ex.12-1

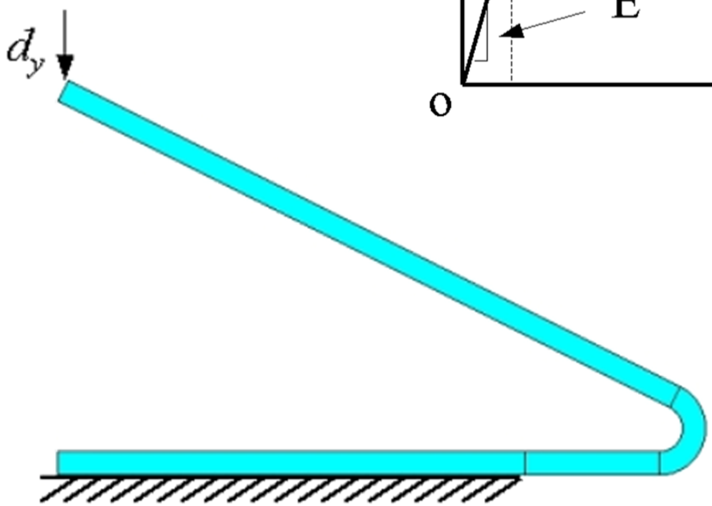
殘留應力分析

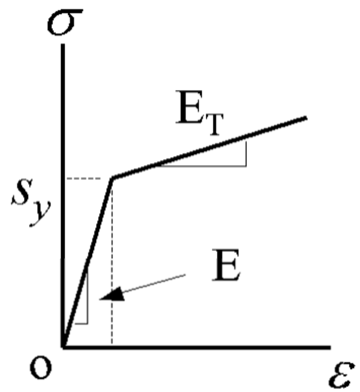
如圖所示類似彈簧片的結構，其尺寸為l=0.02m，h=0.015m，a=0.02m，b=0.01m，圓角R=0.001m，厚度t=0.001m。邊界條件如圖，長度a之底面部分全部拘束固定，當給定之y方向位移d_y=-0.002m時，該結構若有塑性變形，求出移除負荷後的殘留變形。

(1)材料為**bilinear material**，楊氏模數E=210000×10^6Pa，其真應力應變曲線如圖之BISO曲線，E_T=30000×10^6Pa，普松比v=0.3，降伏強度S_y=200×10^6Pa。



bilinear material

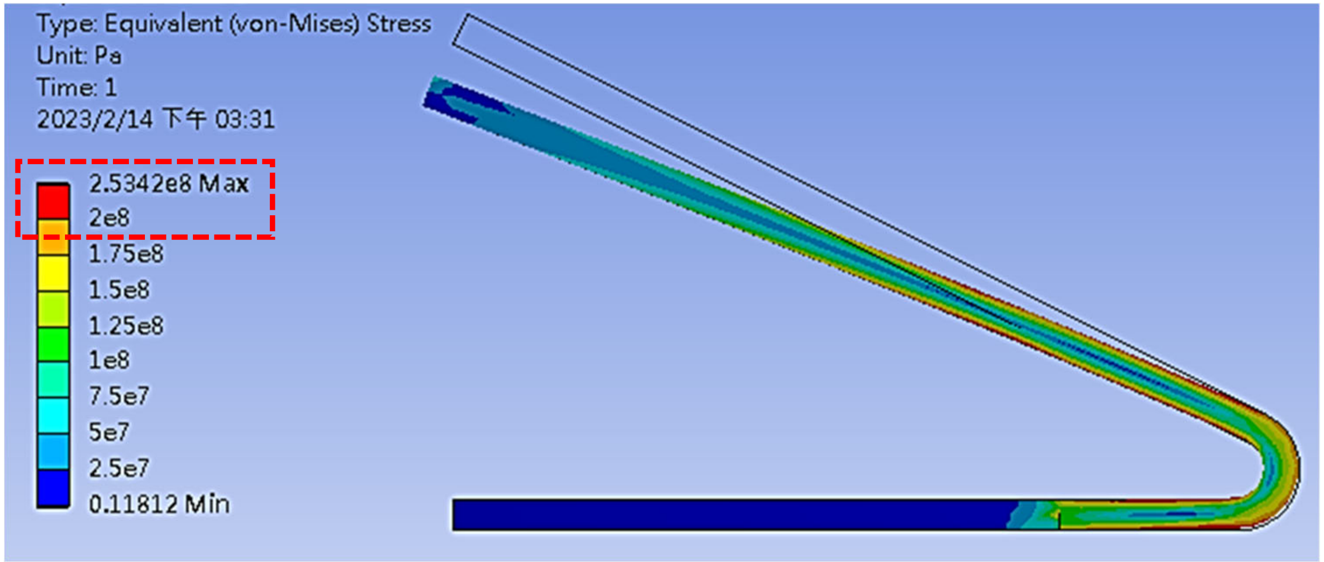




Nonlinear – Ex.12-1

學習目標
- 材料性質設定
- Form new part
- Mesh-*Sweep*
- 大變形

Step. 1 bilinear material



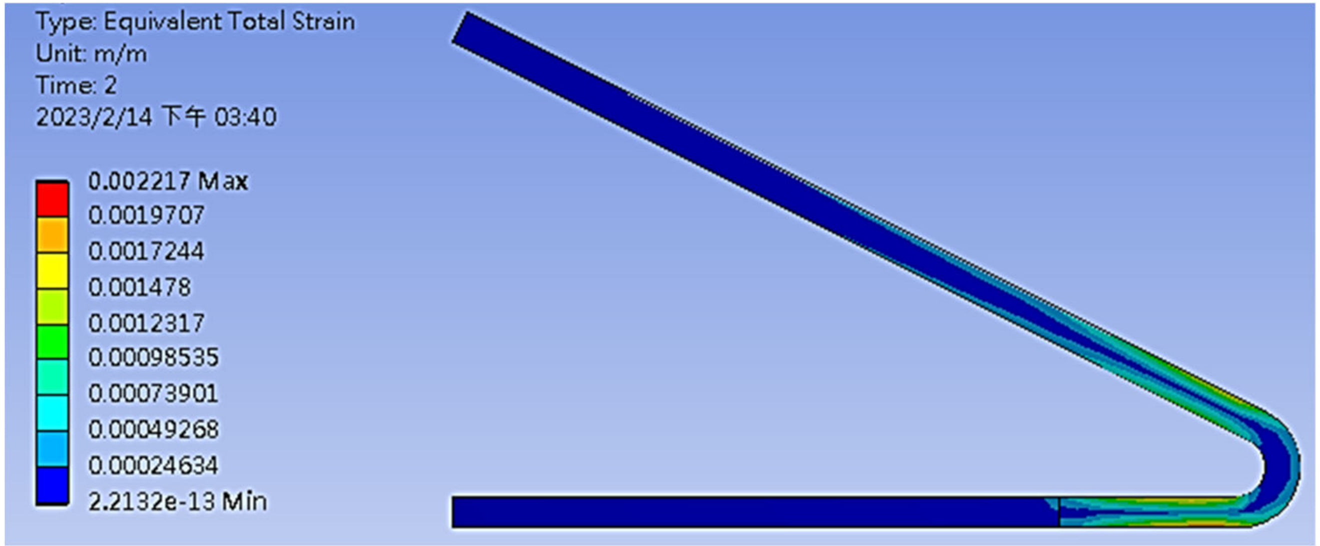
最大應力超過降伏
→發生塑性變形

等效應力
Equivalent Stress

Nonlinear – Ex.12-1

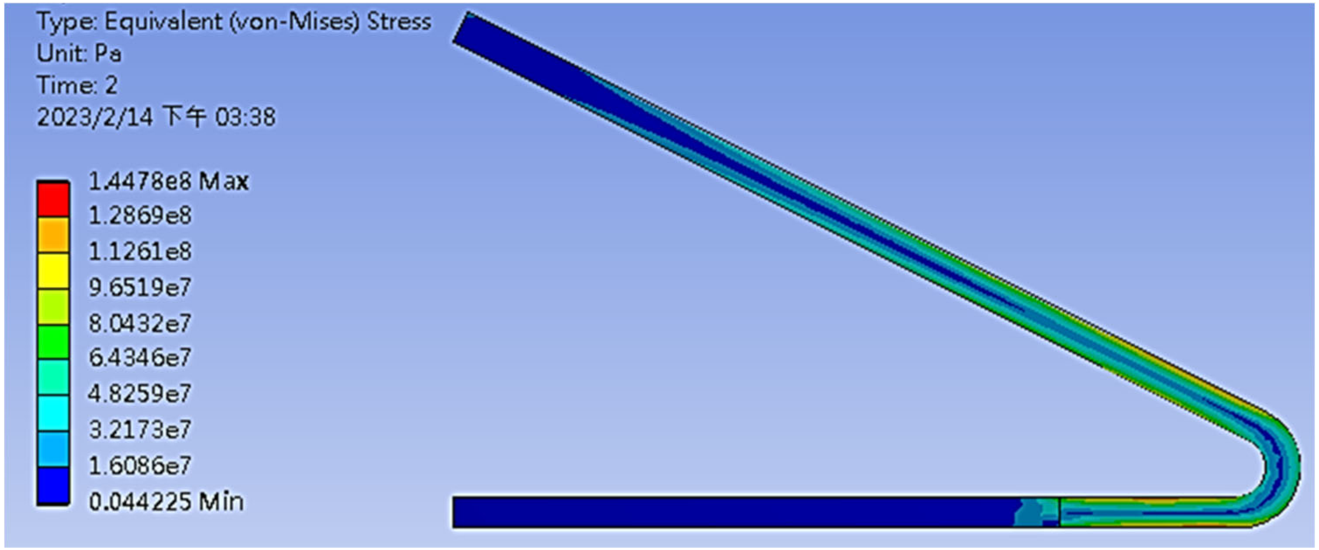
學習目標
- 材料性質設定
- Form new part
- Mesh-*Sweep*
- 大變形

Step. 2 bilinear material



殘留總應變(彈性+塑性)

等效總應變
Equivalent Total Strain



殘留應力

等效應力
Equivalent Stress

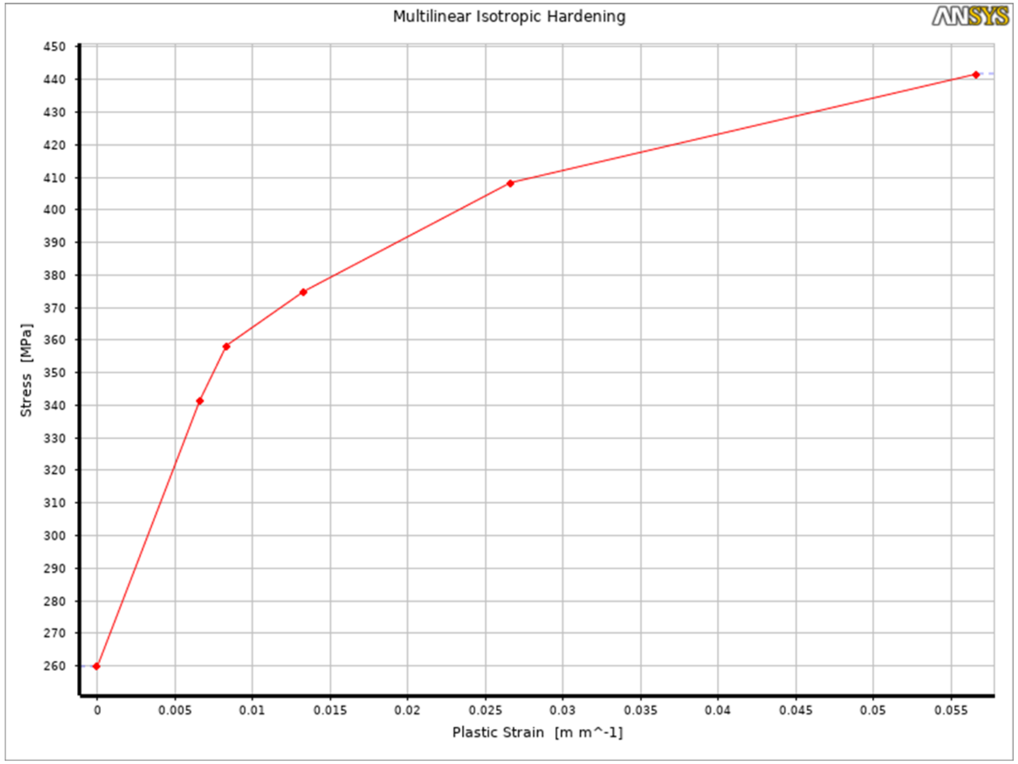
Nonlinear – Ex.12-2

殘留應力分析

如圖所示類似彈簧片的結構，其尺寸為l=0.02m，h=0.015m，a=0.02m，b=0.01m，圓角R=0.001m，厚度t=0.001m。邊界條件如圖，長度a之底面部分全部拘束固定，當給定之y方向位移d_y=-0.004m時，該結構若有塑性變形，求出移除負荷後的殘留變形。

(2)材料為**multilinear material**，數據採用自文獻，楊氏模數E=71200MPa，普松比v=0.31，降伏強度S_y=260MPa，真應力應變曲線原為一實際曲線，經MISO之多直線段簡化且輸入ANSYS後如圖，各點座標如下所示。

multilinear material

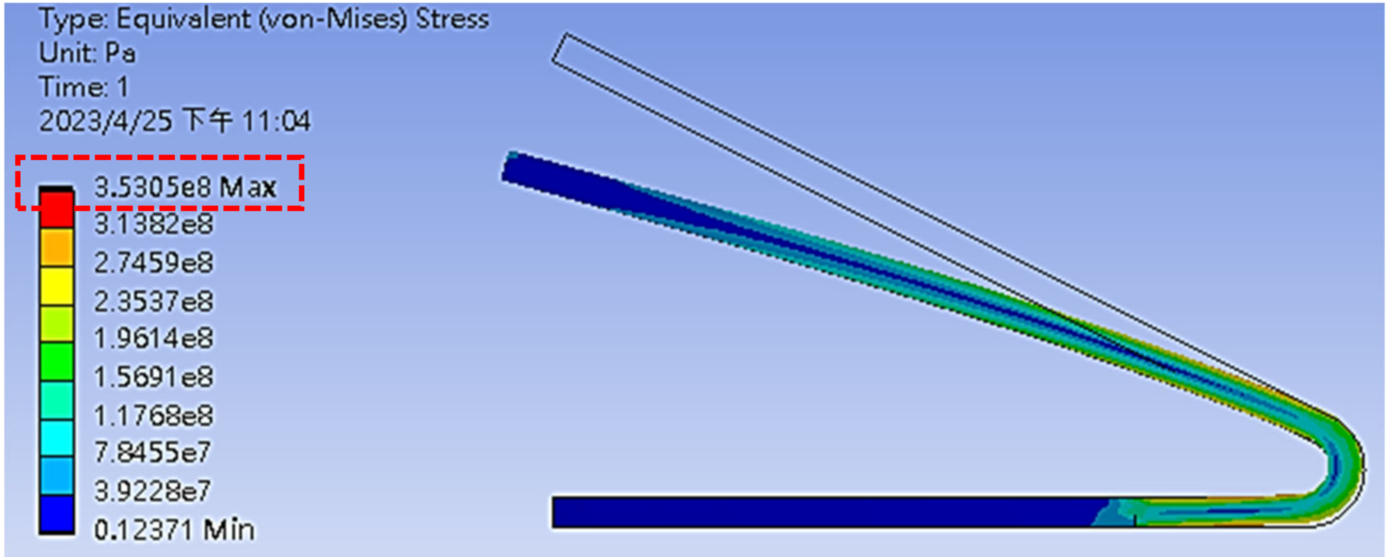


1 (0, 260)
2 (0.0066, 341.68)
3 (0.0083, 358.34)
4 (0.0133, 375.01)
5 (0.0266, 408.34)
6 (0.0566, 441.68)

Nonlinear – Ex.12-2

學習目標
- 材料性質設定
- Form new part
- Mesh-*Sweep*
- 大變形

Step. 1 multilinear material



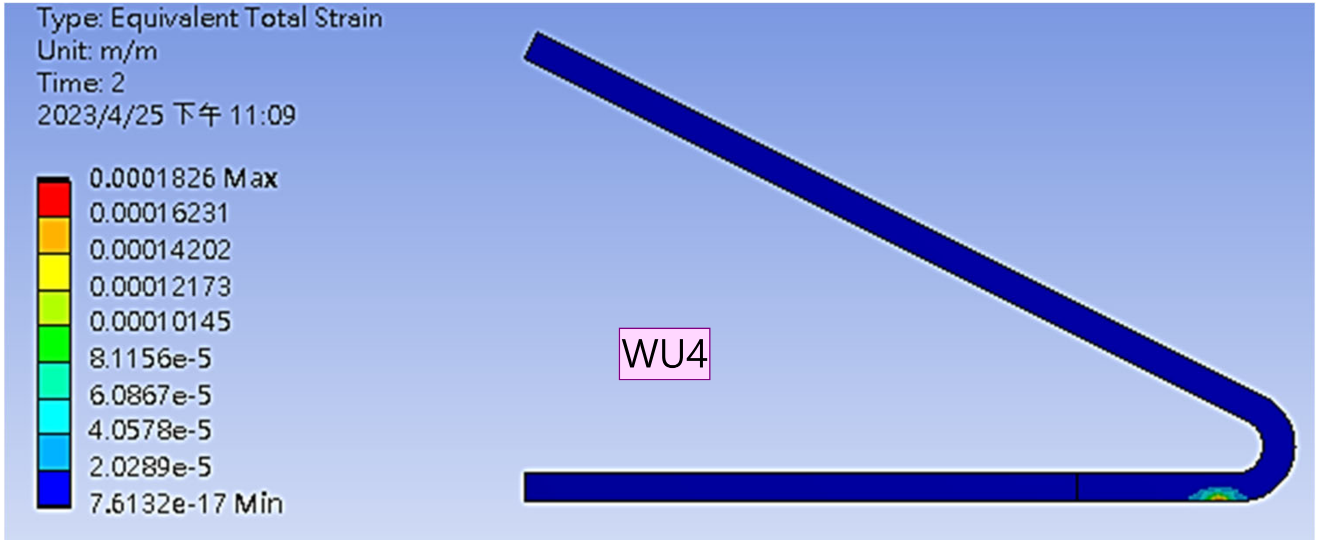
最大應力超過降伏
→發生塑性變形

等效應力
Equivalent Stress

Nonlinear – Ex.12-2

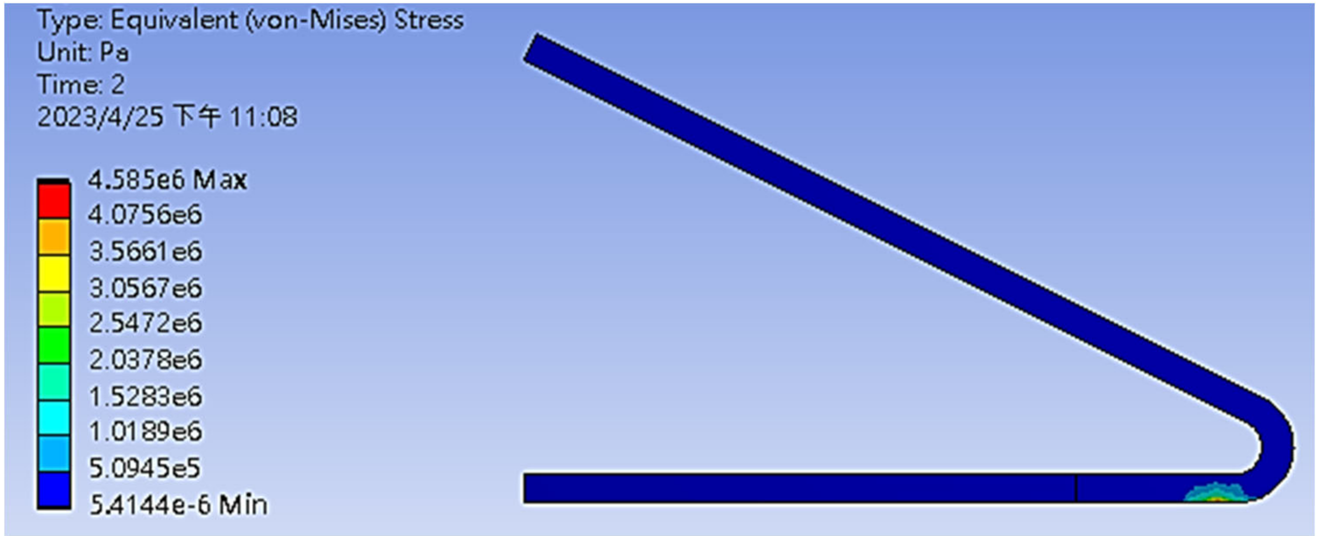
學習目標
- 材料性質設定
- Form new part
- Mesh-*Sweep*
- 大變形

Step. 2 multilinear material



殘留總應變(彈性+塑性)

等效總應變
Equivalent Total Strain



殘留應力

等效應力
Equivalent Stress

WU4

換圖

Windows User, 2023/4/24

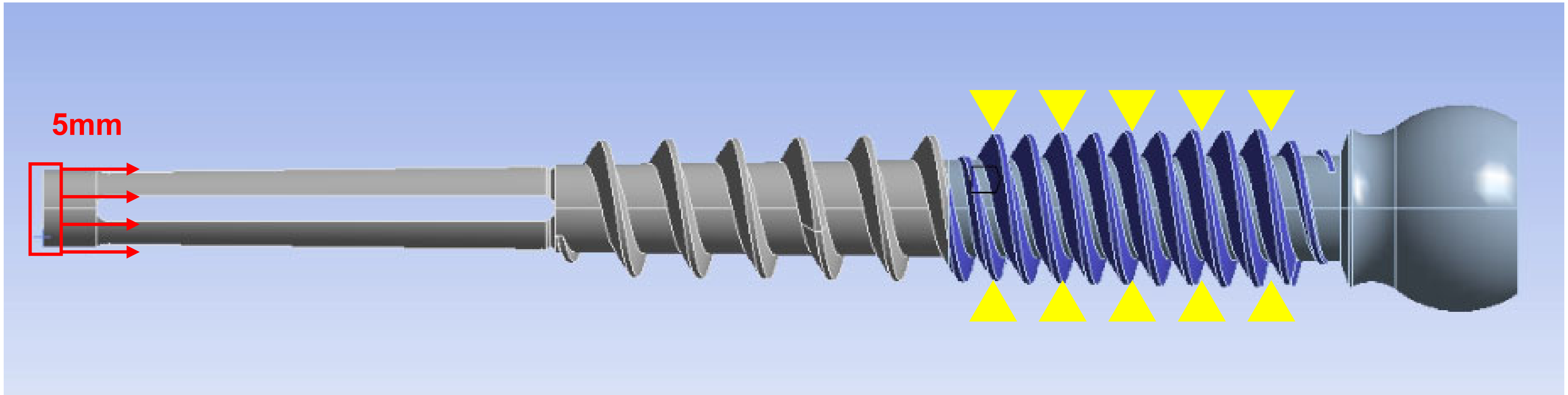
Nonlinear – Ex. 13

殘留應力分析

如圖所示雙螺紋骨釘結構，邊界條件如圖，細螺紋段全部拘束固定，底部給定軸向位移5mm(其他兩方向固定)，比較當材料為線性與非線性時，最大應力與變形量之差異，若該結構有產生塑性變形，求出移除負荷後的殘留變形。

(1)材料為鈦合金，設為**線性材料**，楊氏模數E=110000Mpa，普松比v=0.3。

(2)材料為鈦合金，設為**非線性材料**，楊氏模數E=110000Mpa，普松比v=0.3，Tangent Modulus E_T=1250MPa，降伏強度S_y=800MPa。



Nonlinear – Ex.13

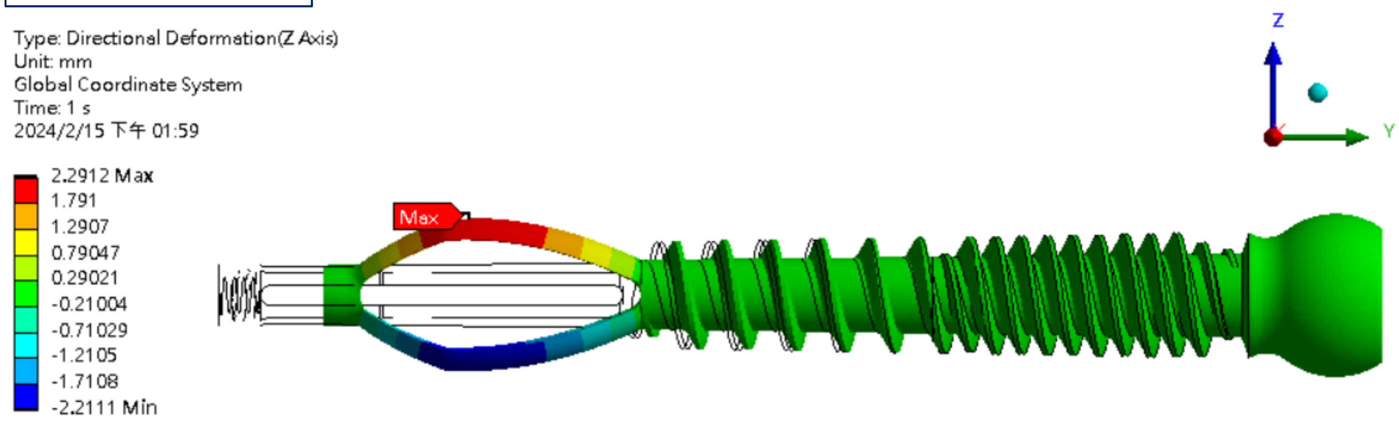
學習目標
- 材料性質設定
- 位移控制設定
- 單軸變形量顯示

殘留應力分析

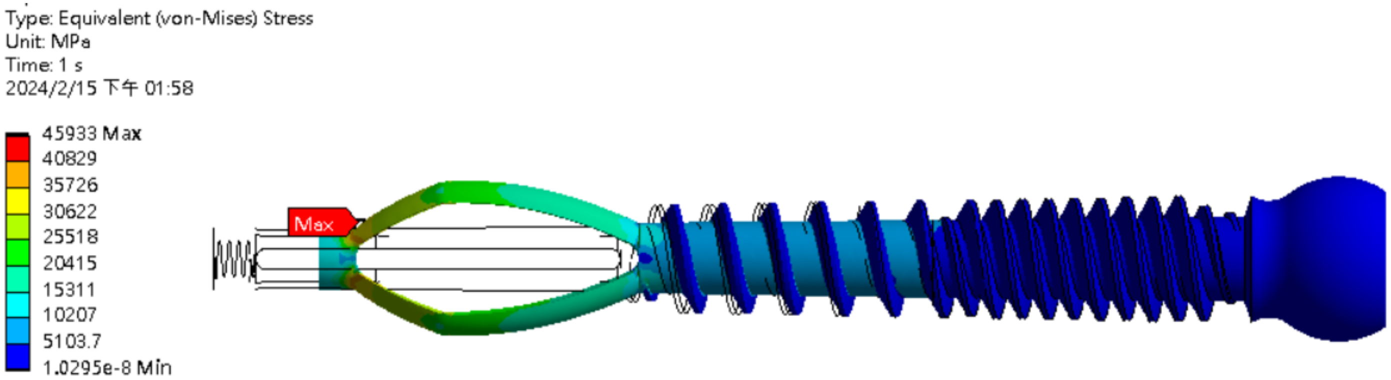
如圖所示雙螺紋骨釘結構，邊界條件如圖，細螺紋段全部拘束固定，底部給定軸向位移5mm(其他兩方向固定)，比較當材料為線性與非線性時，最大應力與變形量之差異，若該結構有產生塑性變形，求出移除負荷後的殘留變形。

(1)材料為鈦合金，設為**線性材料**，楊氏模數E=110000Mpa，普松比v=0.3。

Linear material



變形量
Z-Directional Deformation



等效應力
Equivalent Stress

Nonlinear – Ex. 13

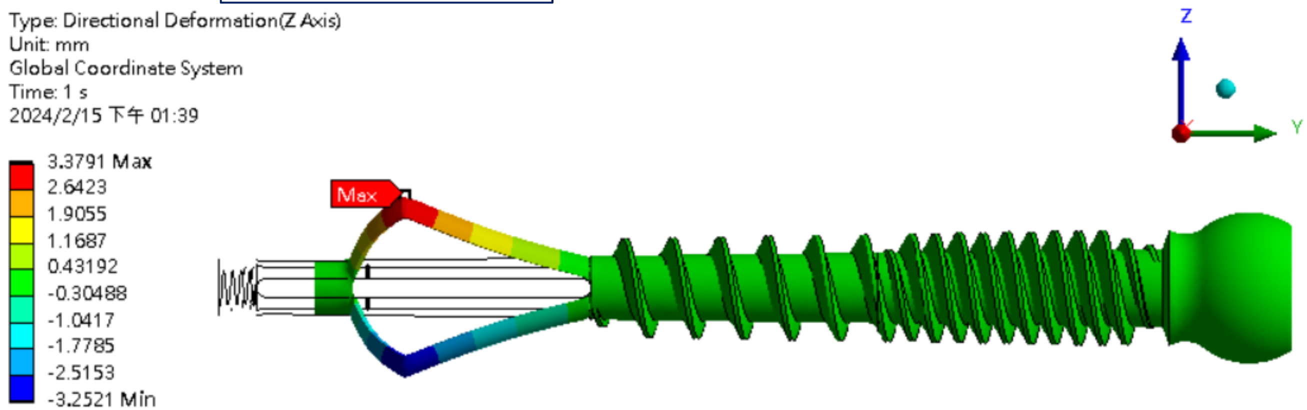
學習目標
- 材料性質設定
- 位移控制設定
- 單軸變形量顯示

殘留應力分析

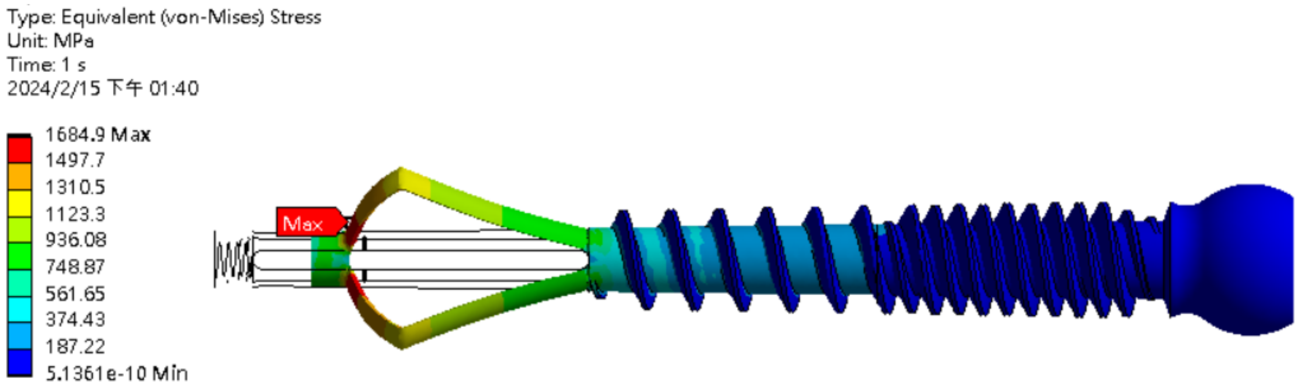
如圖所示雙螺紋骨釘結構，邊界條件如圖，細螺紋段全部拘束固定，底部給定軸向位移5mm(其他兩方向固定)，比較當材料為<u>線性</u>與<u>非線性</u>時，最大應力與變形量之差異，若該結構有產生塑性變形，求出移除負荷後的殘留變形。

(2)材料為鈦合金，設為**非線性材料**，楊氏模數E=110000Mpa，普松比v=0.3，Tangent Modulus E_T=1250MPa，降伏強度S_y=800MPa。

Step. 1 **Bilinear material**



變形量
Z-Directional Deformation



等效應力
Equivalent Stress

Nonlinear – Ex.13

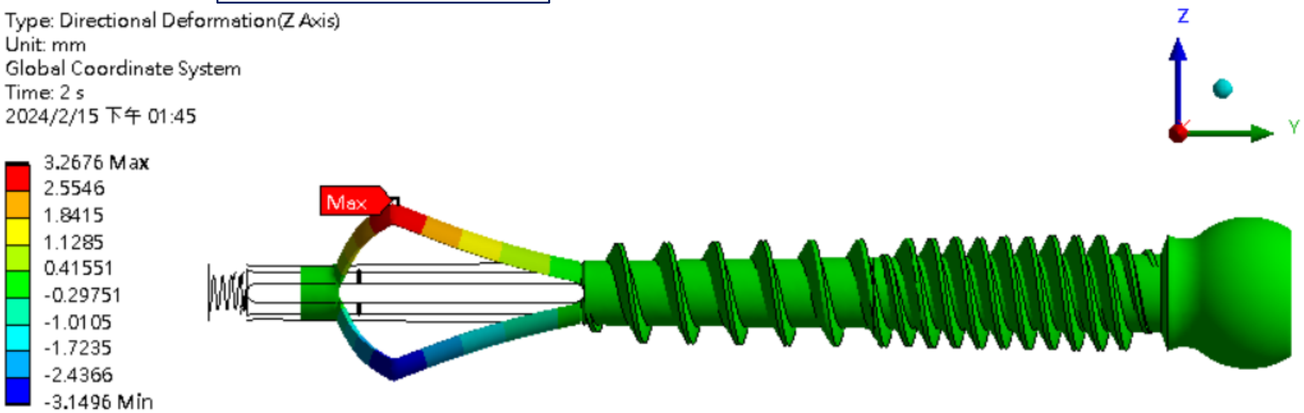
學習目標
- 材料性質設定
- 位移控制設定
- 單軸變形量顯示

殘留應力分析

如圖所示雙螺紋骨釘結構，邊界條件如圖，細螺紋段全部拘束固定，底部給定軸向位移5mm(其他兩方向固定)，比較當材料為線性與非線性時，最大應力與變形量之差異，若該結構有產生塑性變形，求出移除負荷後的殘留變形。

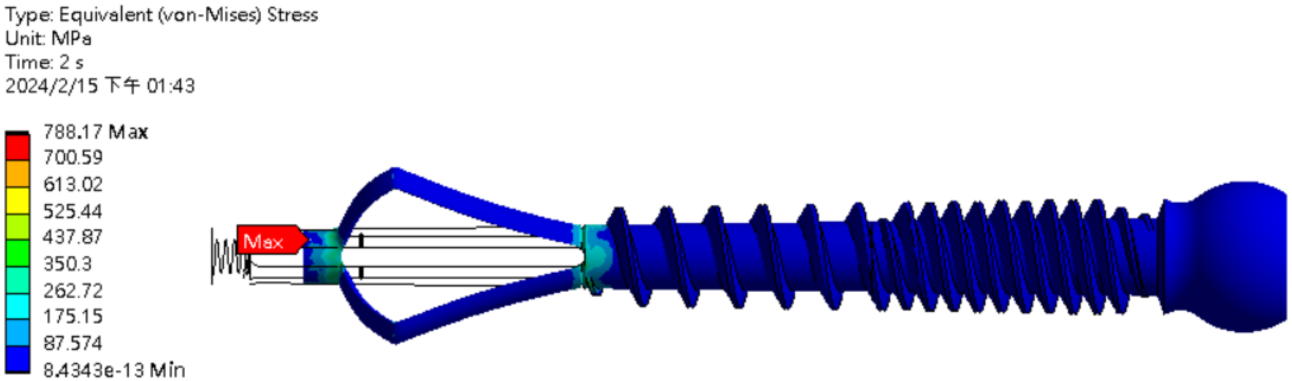
(2)材料為鈦合金，設為**非線性材料**，楊氏模數E=110000Mpa，普松比v=0.3，Tangent Modulus E_T=1250MPa，降伏強度S_y=800MPa。

Step. 2 **Bilinear material**



塑性變形

變形量
Z-Directional
Deformation

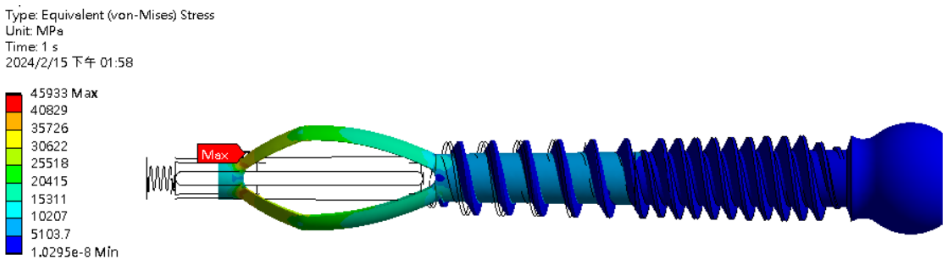


殘留應力

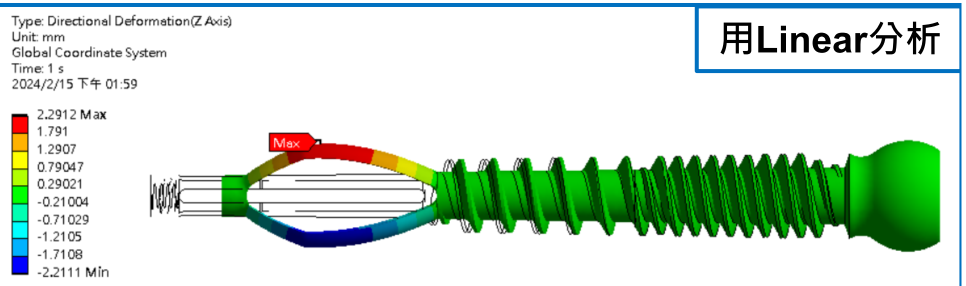
等效應力
Equivalent Stress

Nonlinear – Ex.13

用Linear分析

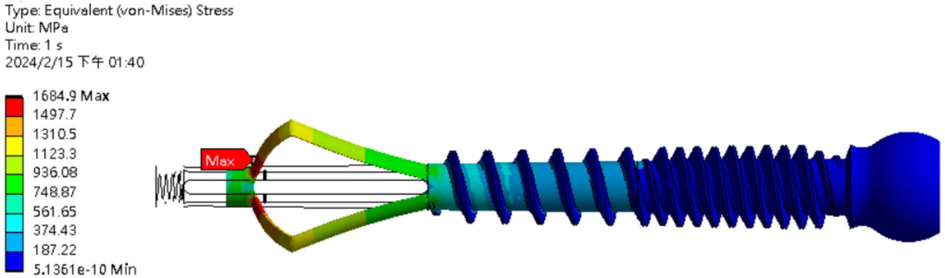


位移5mm時 等效應力45933MPa

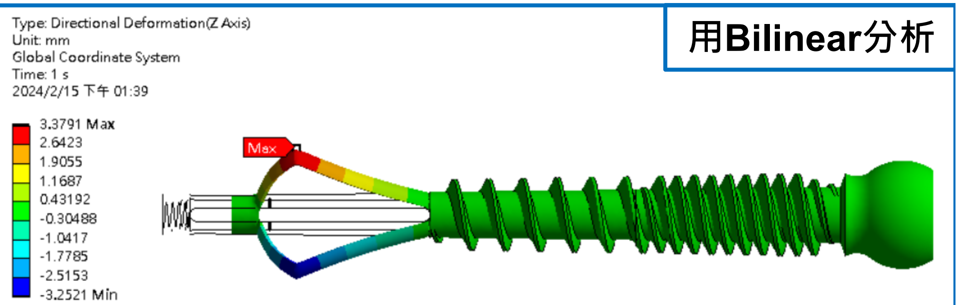


位移5mm時 橫向位移2.2912mm

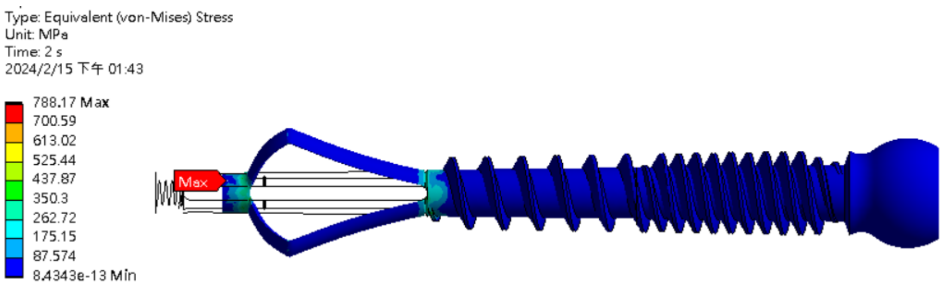
用Bilinear分析



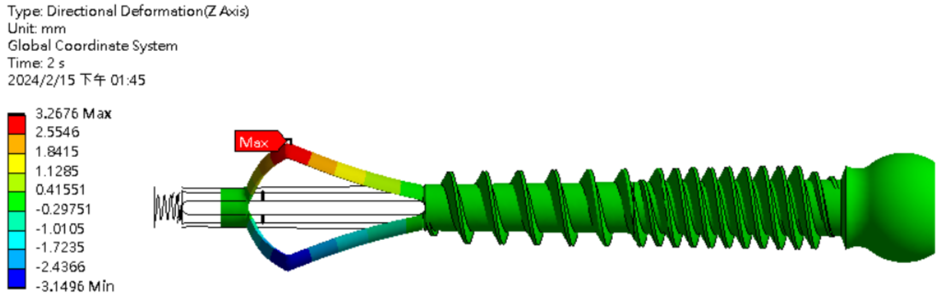
Step.1 位移5mm時 等效應力1270MPa



Step.1 位移5mm時 橫向位移3.38mm

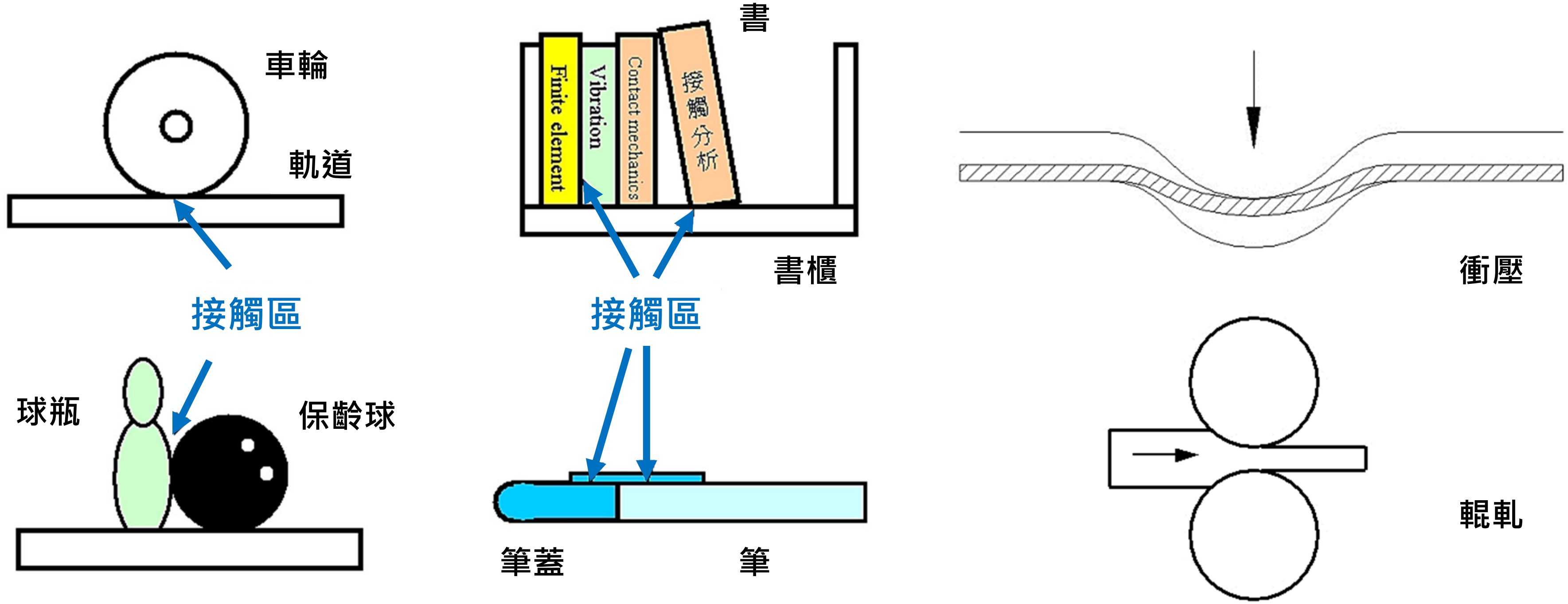


Step.2 移除位移後 等效應力788MPa(殘留應力)

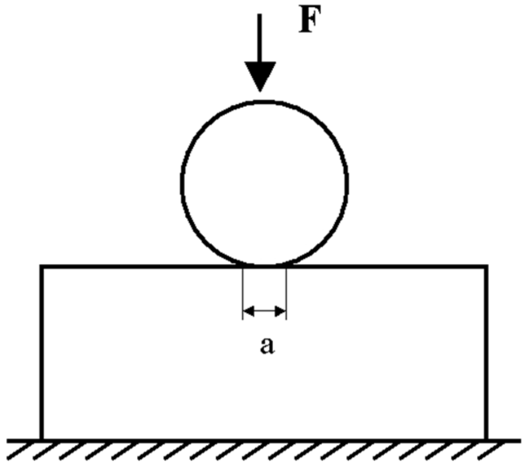


Step.2 移除位移後 橫向位移3.27mm(塑性變形)

Contact Analysis



機械加工製程之
接觸問題(塑性變形)



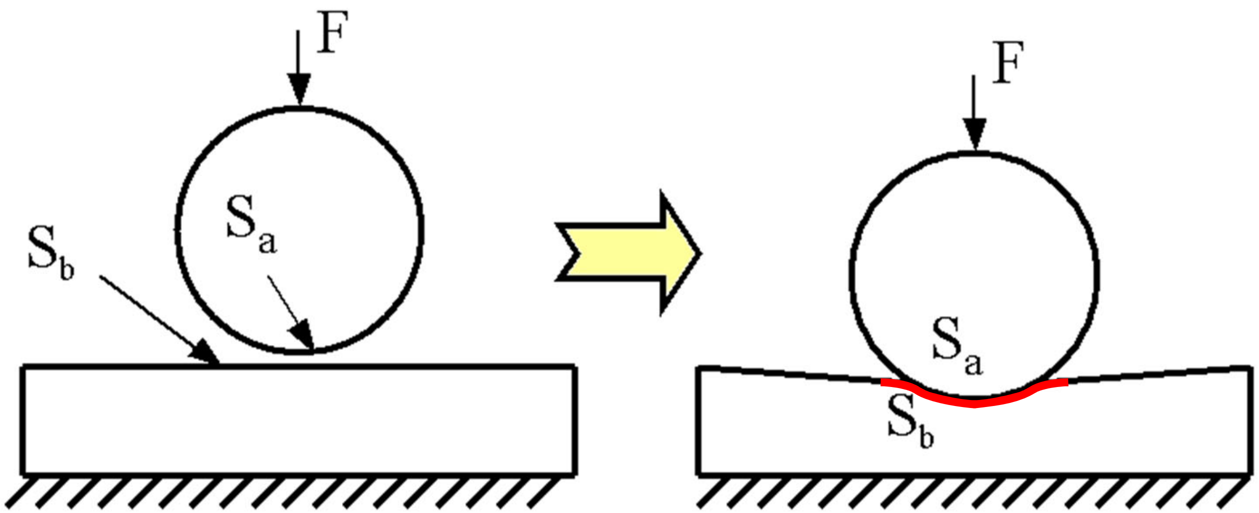
兩物體之
彈性接觸問題

Contact Analysis

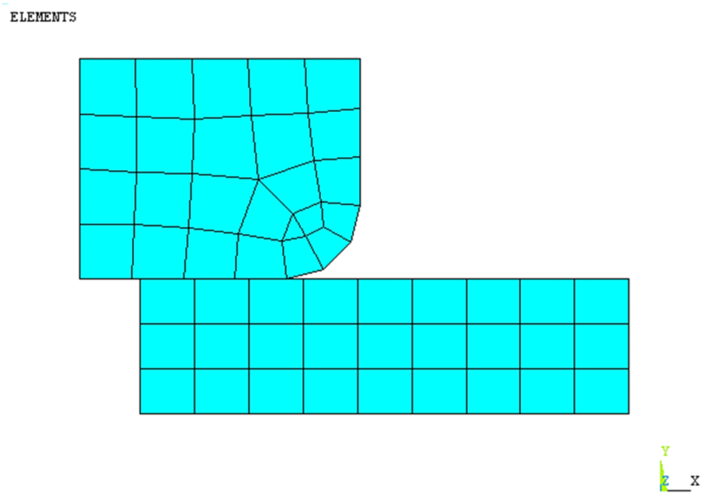
- 有限元素接觸分析是於物體之間的接觸面上加入非貫穿(non-penetration)條件

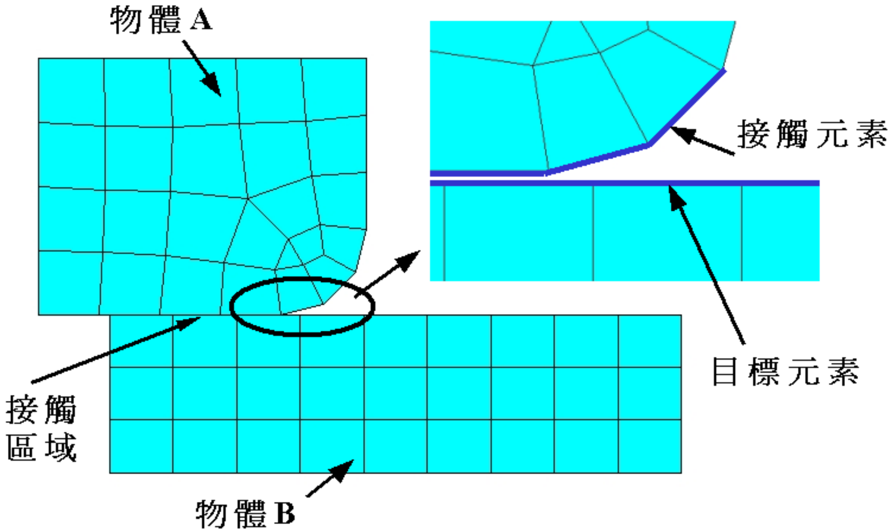
- 以下圖之兩物體接觸為例，首先須將Sa和Sb兩個面定義為接觸面，下令兩接觸面不可貫穿，如此一來，只要圓形物體受力變形，便可透過兩接觸面Sa和Sb將力量傳至矩形物體，使得矩形物體也跟著變形，即完成了接觸分析

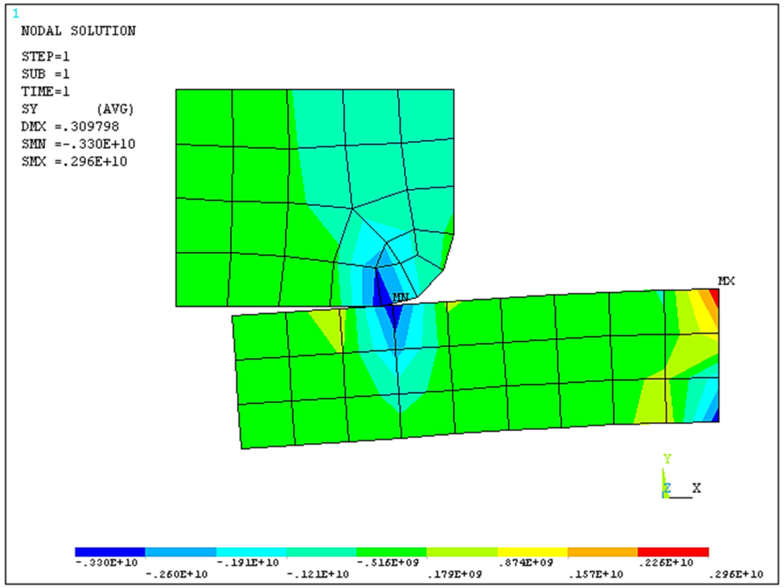
- ANSYS是利用接觸元素(contact elements)來模擬接觸面，只要接觸區域的接觸元素一被建立，計算時就會考慮到接觸條件



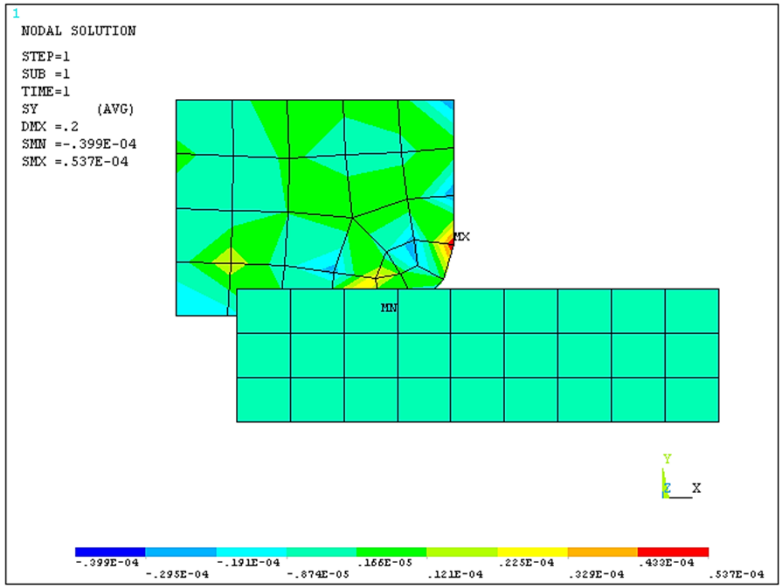
Contact Analysis







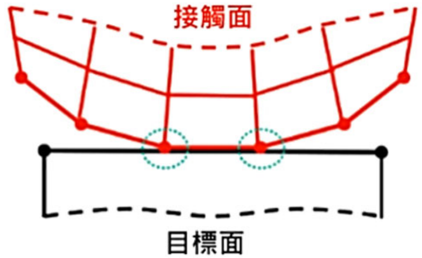
合理之接觸分析結果



不合理之接觸分析結果
(未建立接觸元素)

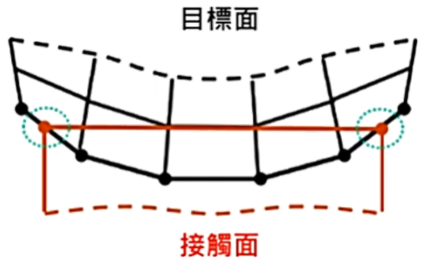
Contact Analysis

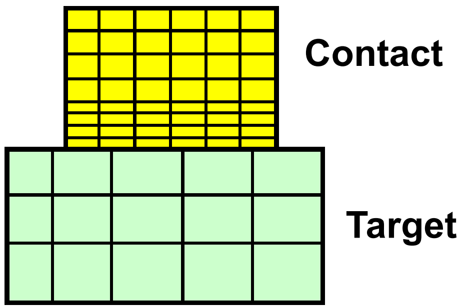
- 在ANSYS定義中，目標面(Target surface)的節點可以穿透接觸面(Contact surface)，接觸面的節點則不可穿透目標面

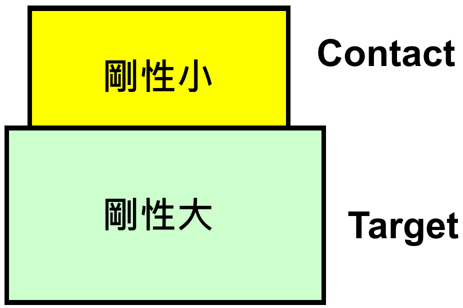


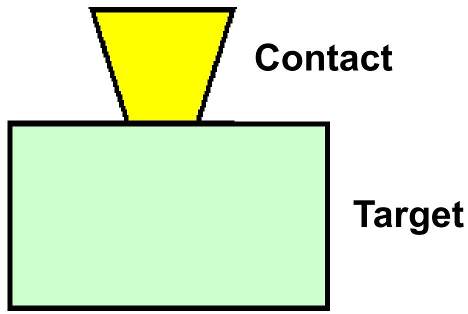
- 接觸對之接觸元素和目標元素建立原則

目標面(Target surface)	接觸面(Contact surface)
網格較粗	網格較細
剛性較大(硬)	剛性較小(軟)
面積顯著較大(平面、凹面)	面積顯著較小(尖、凸面)





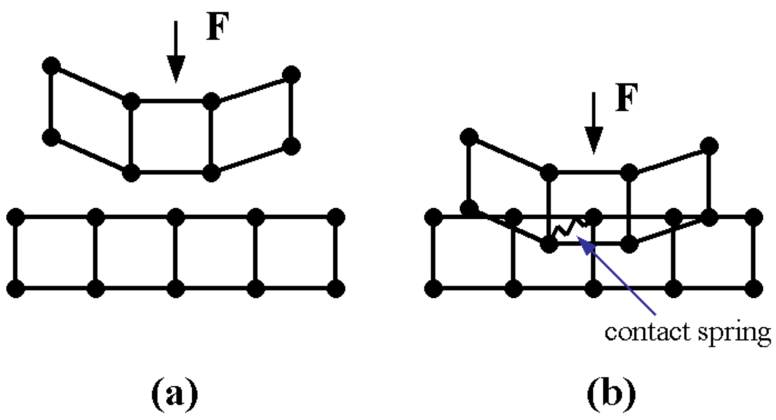




資料來源https://www.youtube.com/watch?v=kQgYAisb5V4&list=PLF1WR5I3KTKQ4IQkA56C1m-FU40mB1mzz&index=3

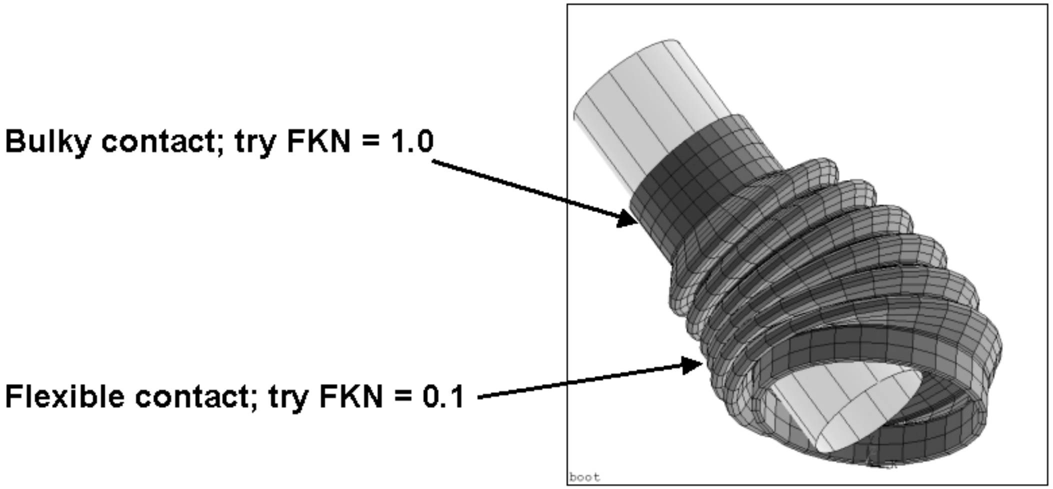
實常數與摩擦係數

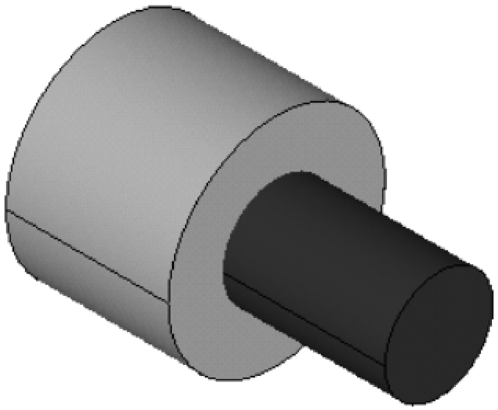
- 關於接觸對之非貫穿接觸條件，是由目標元素和接觸元素所共用的實常數(real constants)來決定。
- 而經常設定的實常數有四個，分別為FKN、FTOLN、FKT、TAUMAX，摩擦係數則於材料係數中給定，其符號為MU。
- ANSYS面對面接觸元素內定之計算法則為augmented Lagrangian method，所以必須設定接觸剛度(contact stiffness) KN和貫穿公差(penetration tolerance)兩種計算常數，其在面對面接觸元素之實常數中，接觸剛度和貫穿公差分別使用FKN和FTOLN兩個實常數來設定。
- FKN之意義如圖所示，可想像兩物體有限元素模型接觸對之接觸面上，有一接觸彈簧(contact spring)，其彈簧係數即為接觸剛度，而接觸剛度KN定義為FKN乘以接觸體之剛度



FKN 經驗值

- 一般經驗值為：(a)針對有大體積變形(bulk deformation)之接觸狀況，例如圖上之兩物體接觸，先設定FKN=1，再測試答案合理性；(b)若兩物體之接觸情形有彎曲(bending)狀況，例如圖下的彎曲接觸，先設定FKN=0.01～0.1，再測試答案合理性。



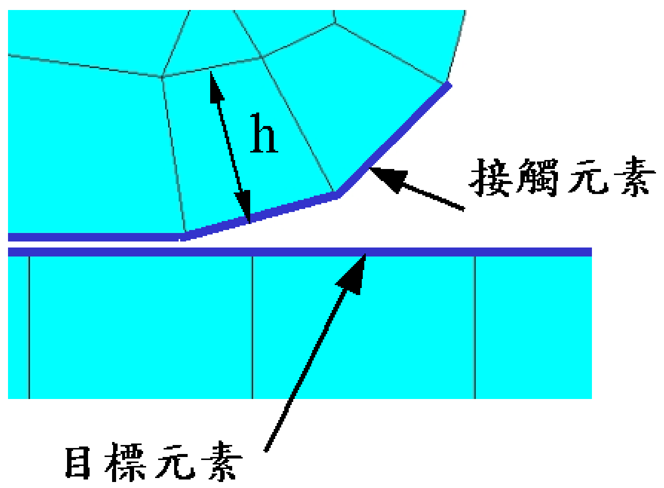


FKN	Max. SEQV
0.001	4,000
0.01	20,000
0.1	65,000
1	91,000
10	92,900
100	93,000

FTOLN

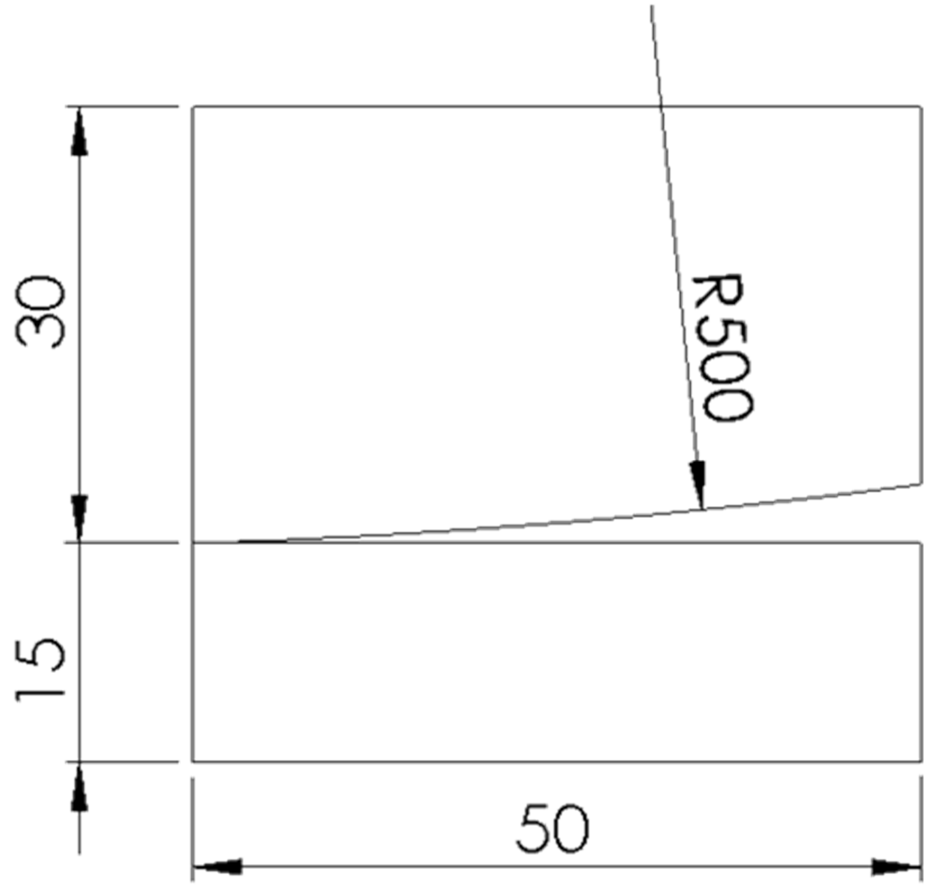
- 在接觸面的計算上，兩物體接觸貫穿量必須小於貫穿公差才算是接觸
- 而FTOLN代表了貫穿公差之計算參數，ANSYS定義之貫穿公差為FTOLN乘以接觸面底下元素深度h，如圖所示。
- ANSYS以庫倫摩擦模型(Coulomb friction model)來模擬接觸摩擦現象，公式為：
- 越小的貫穿公差(越小的FTOLN)越接近實際接觸情況，過小的貫穿公差會造成數值無法收斂，FTOLN內定值為0.1(常設定範圍0.01-0.05)

penetration tolerance = (FTOLN)*h

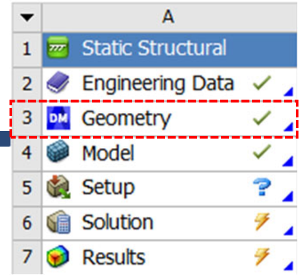


Nonlinear & Contact – Ex.14

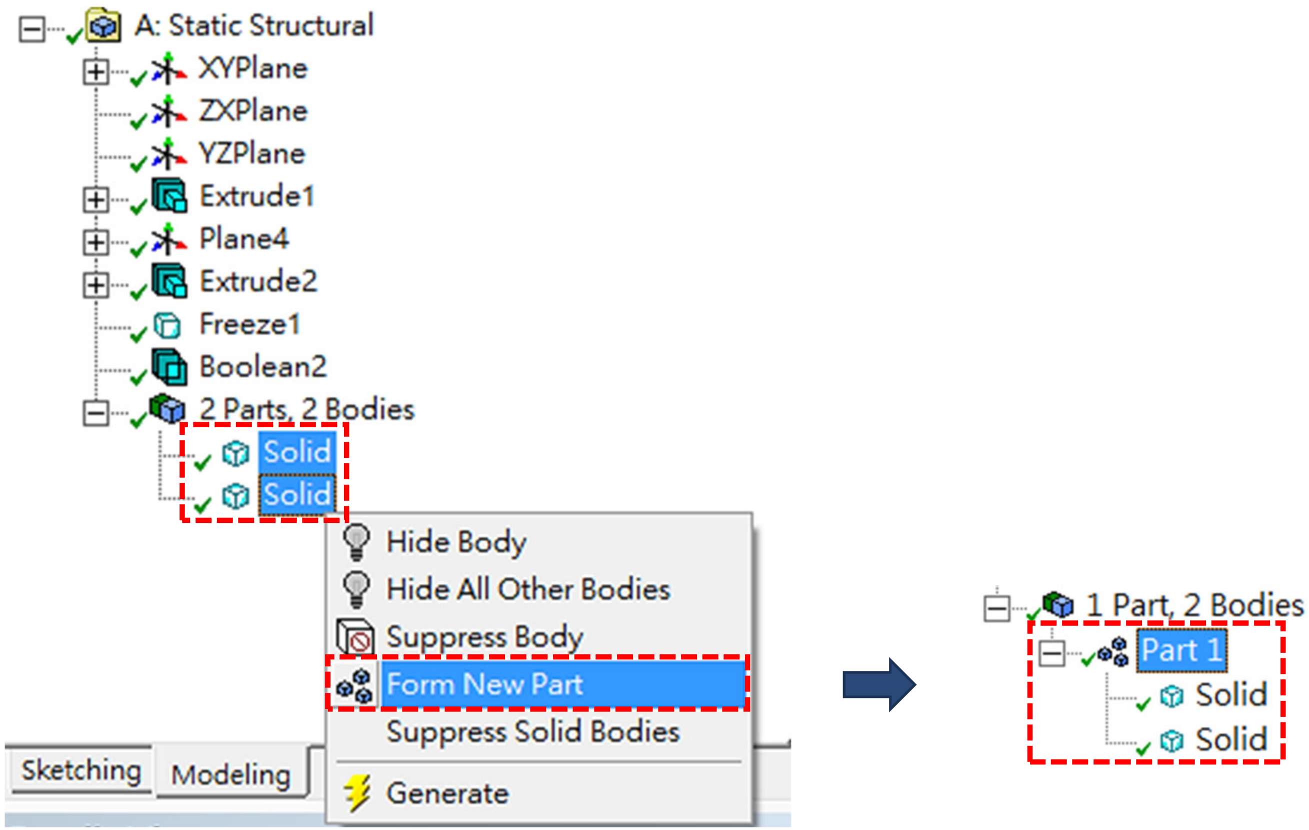
2D模型如圖所示，其為剛性接觸的兩物體且下端整面為固定並於上端邊線受一5MPa之壓力，請針對該模型進行接觸剛性分析。



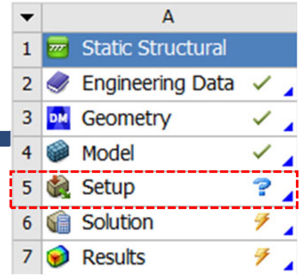
Introduction of ANSYS Workbench



- 介面條件設定
  - 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，模型匯入後軟體會自動判斷出非連續面之部份

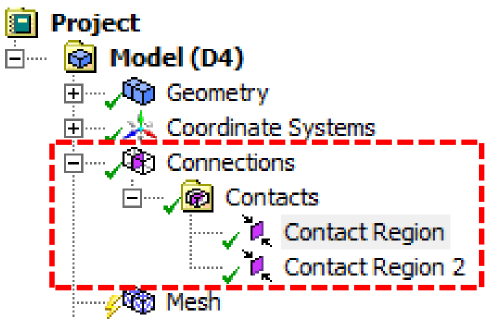


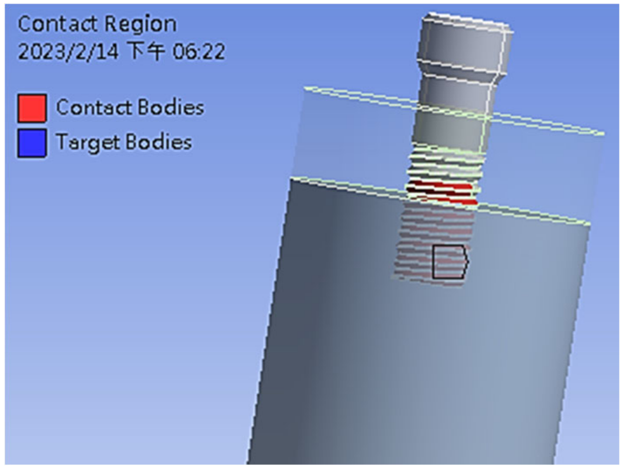
Introduction of ANSYS Workbench



■ 接觸(Contact)設定

- ➢ 軟體會自動偵測到不同part之界面，並於Connection中顯示所有之contact區域
- ➢ 接觸行為設定
  - ✓ **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**
    - 類似Bonded，僅適用於面(3D)或邊(2D)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**
    - 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





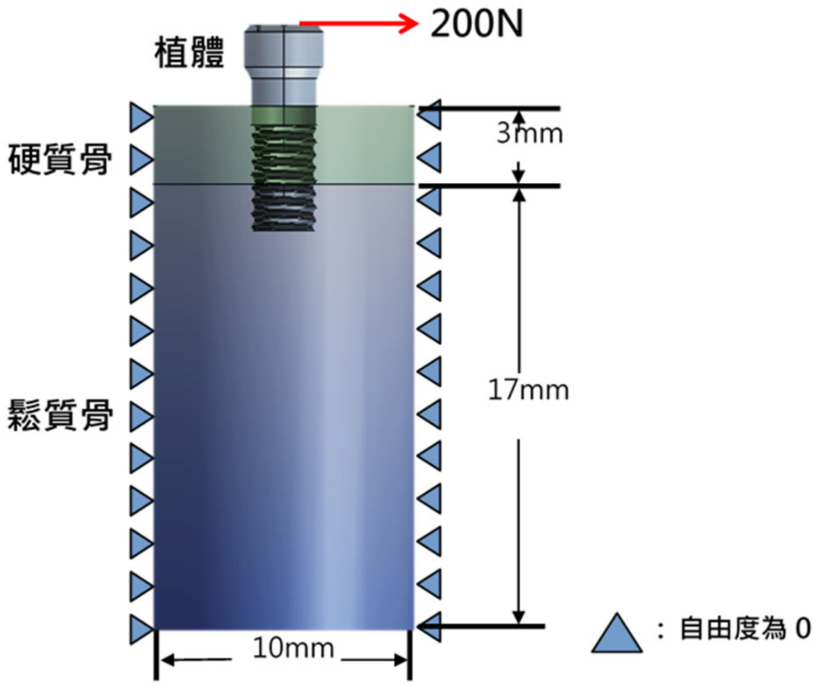
Contact – Ex.15

人工牙根分析

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.igs)匯入，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)。各材料特性：硬質骨(楊氏係數17000MPa；蒲松比0.3)、鬆質骨(楊氏係數200MPa；蒲松比0.2)及植體(鈦合金楊氏係數110000MPa；蒲松比0.33)，採用四面體網格，網格尺寸：植體0.5mm、硬質骨0.8mm、鬆質骨1.0mm、硬質骨內側螺紋面0.5mm。觀察硬質骨最大主應變(Maximum Principal strain)及植體最大等效應力(von-Mises stress)情形。

(1)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)

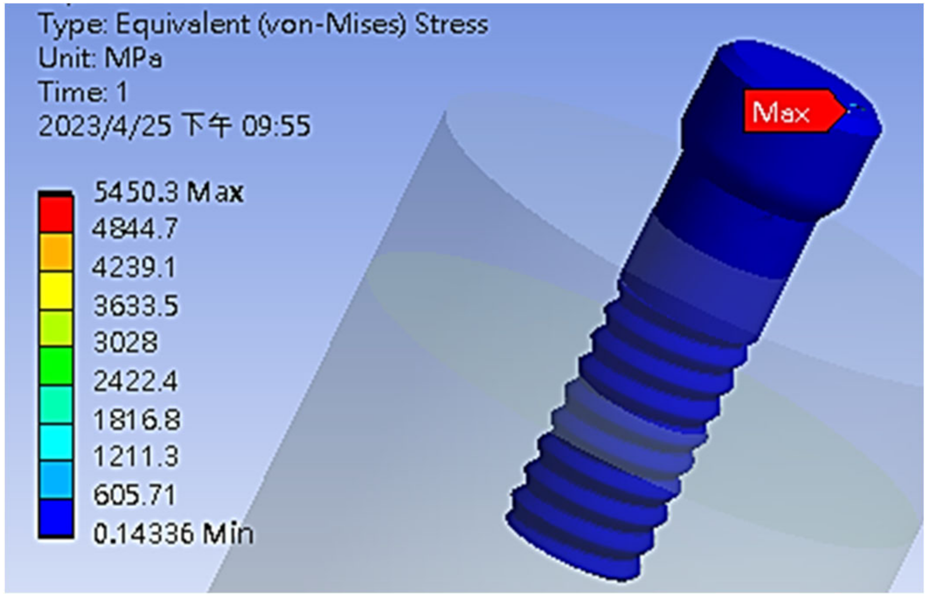
(2)植體與硬質骨/鬆質骨界面結合(bonded)狀態之設定(模擬植體與骨頭已骨整合)



Contact – Ex.15

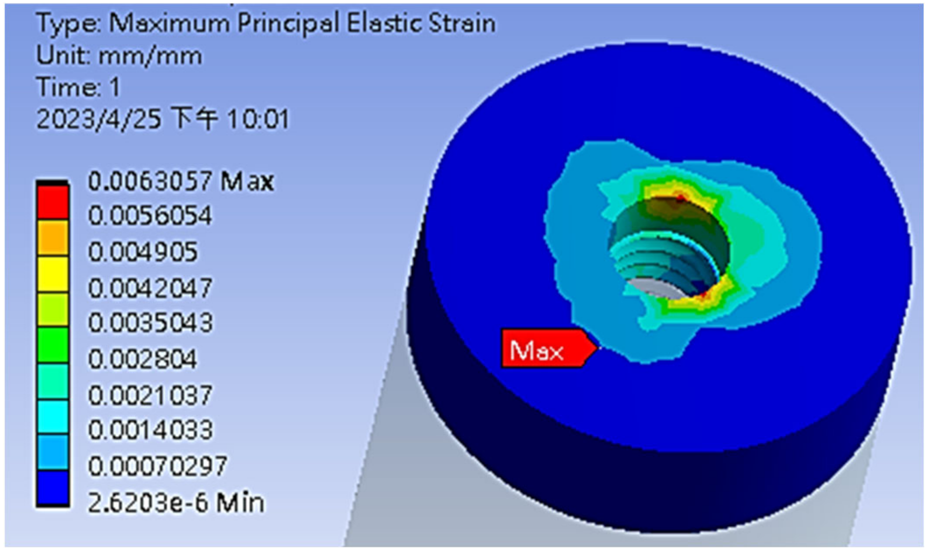
學習目標
- 外部模型導入
- Offset plane
- Move
- Contact設定

(1) Unbonded



Screw

等效應力
Equivalent Stress



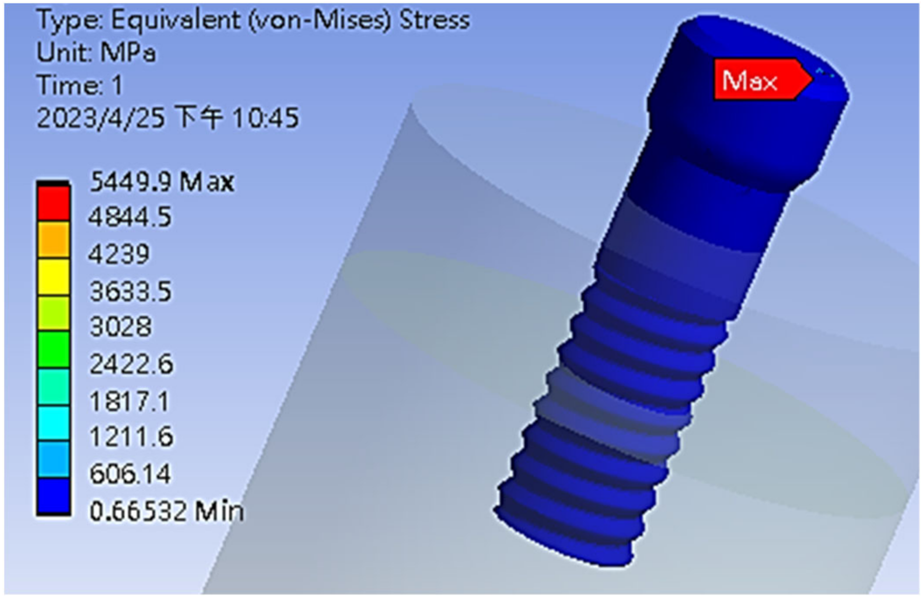
Cortical Bone

最大主應變
Maximum Principal Strain

Contact – Ex.15

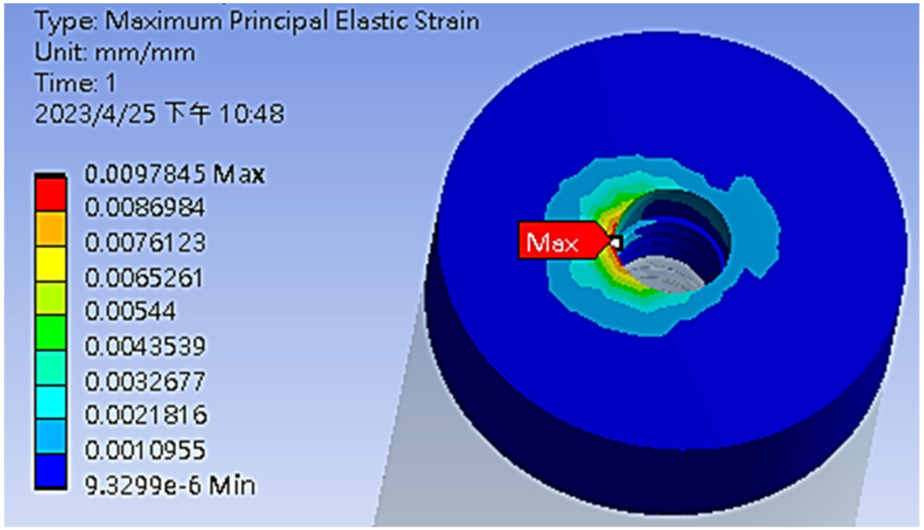
學習目標
- 外部模型導入
- Offset plane
- Move
- Contact設定

(2) Bonded



Screw

等效應力
Equivalent Stress



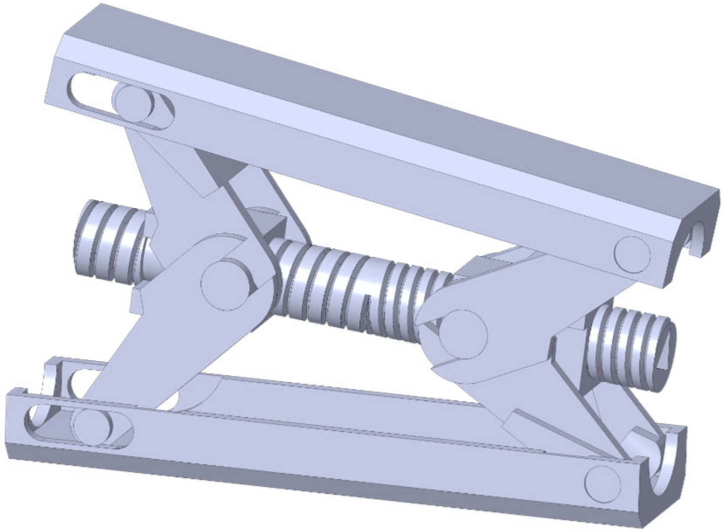
Cortical Bone

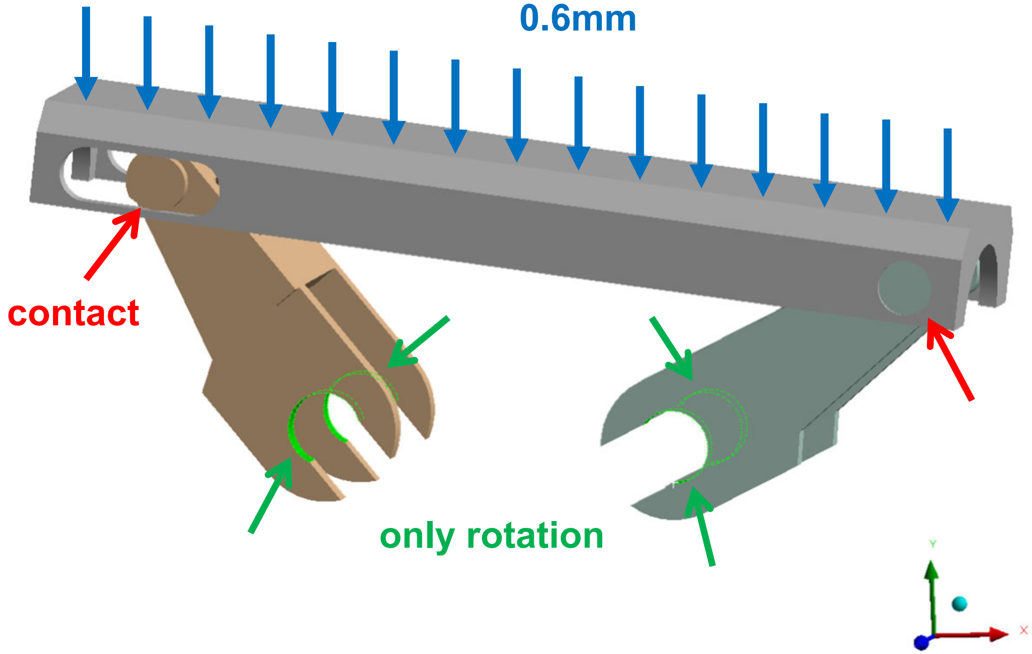
最大主應變
Maximum Principal Strain

Contact – Ex. 16

椎籠分析

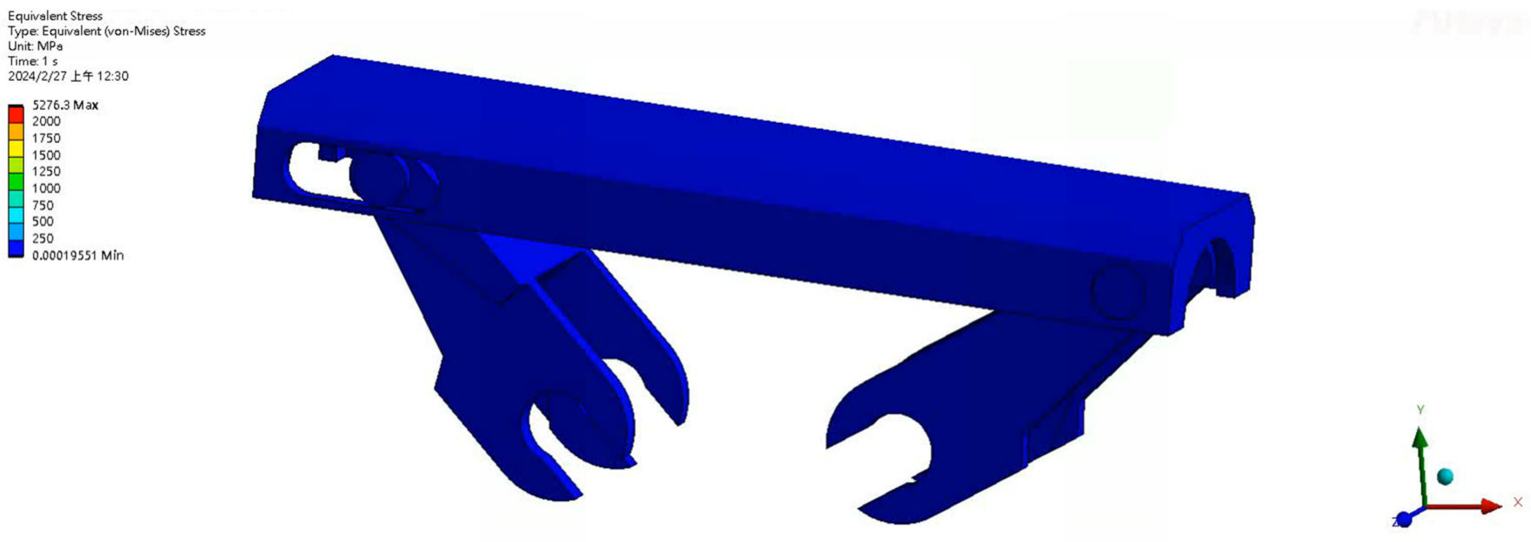
請匯入外部CAD軟體建構之椎籠上半部圖檔(cage.igs)，包括右圖之上方三零件(如右圖所示)，施加垂直向下位移0.6mm於椎籠頂部平面上，並限制連桿下方與圓軸接觸位置(如下圖綠色箭頭處)僅可做旋轉運動(X/Z方向自由度為0)。材料楊氏係數110000MPa、蒲松比0.3，採用四面體網格、網格尺寸0.5mm，上方兩圓軸與頂板之接觸設定(contact)為Frictional、摩擦係數0.3 。**觀察整體結構運動與破壞情形，並求出最大等效應力(von-Mises stress)。**

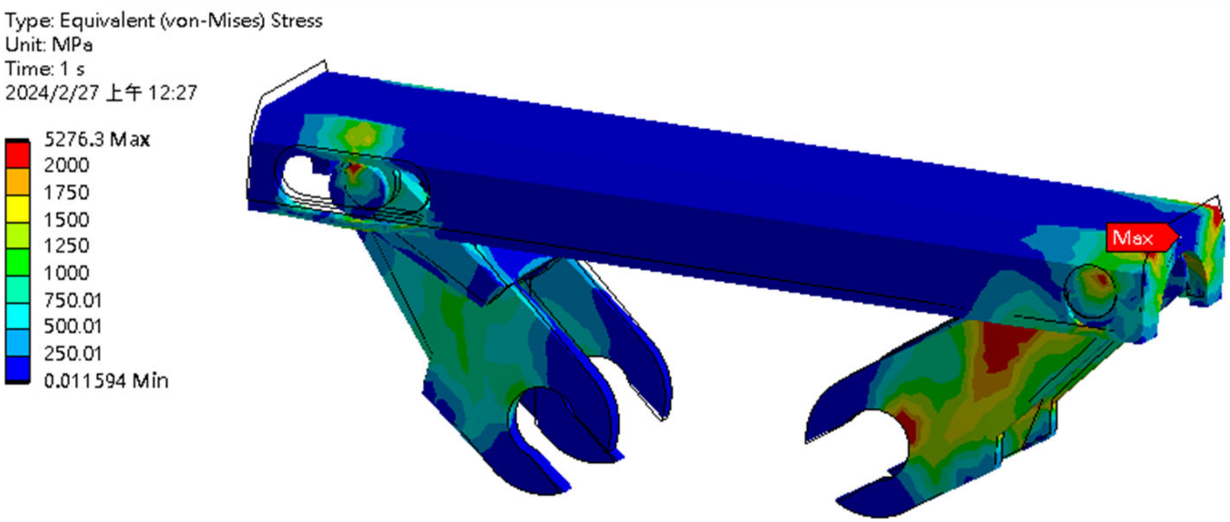




Contact – Ex.16

學習目標
- Suppress模型
- Contact設定
- 圓柱座標設定

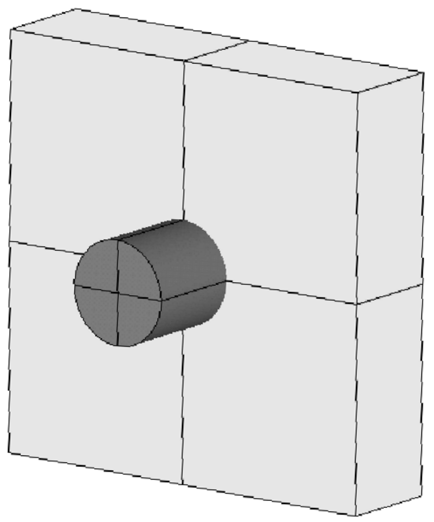




等效應力
Equivalent Stress

Ex 16-1:3D Pin-hole contact problem
Classical Contact Tutorial

- **Problem Description**
  - This is a 3-D analysis of a steel pin contacting a smooth pinhole in a block. You will define two different load steps. The objective of the first load step is to observe the interference fit stresses of the pin which is geometrically thicker than its pinhole. The objective of the second load step is to observe the stresses, contact pressures and reaction forces due to the motion of the pin being pulled out from the block



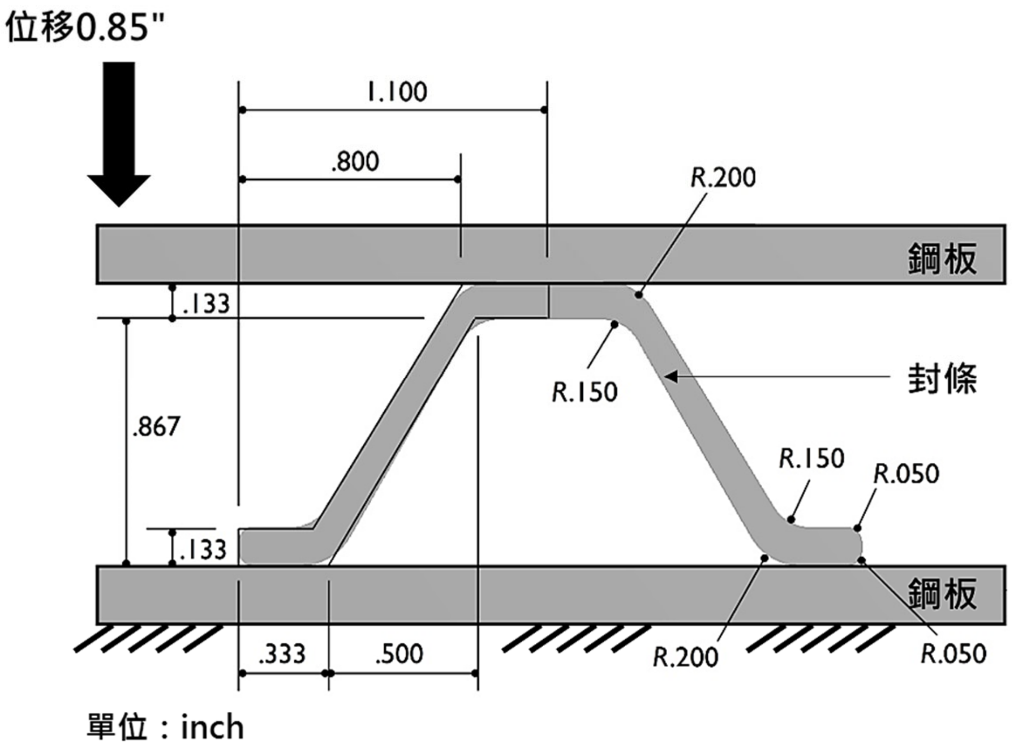
- **The dimensions of the model are as follows: PIN radius = 0.5 mm, length = 2.5 mm. BLOCK width = 4 mmm, length = 4 mm, depth = 1 mm. PINHOLE radius = 0.49 mm, depth = 1 mm. Both solids are made of structural steel. (u=0.2)**
- **Approach and Assumptions**
  - **You will use two load steps to set up the analysis: Load Step 1: Interference Fit -- solve the problem with no additional displacement constraints. The pin is constrained within the pinhole due to its geometry. Stresses are generated due to the general misfit between the target (pinhole) and the contact (pin) surfaces. Load Step 2: Pull-out -- move the pin by 3 mm out of the block using DOF displacement conditions on coupled nodes.**

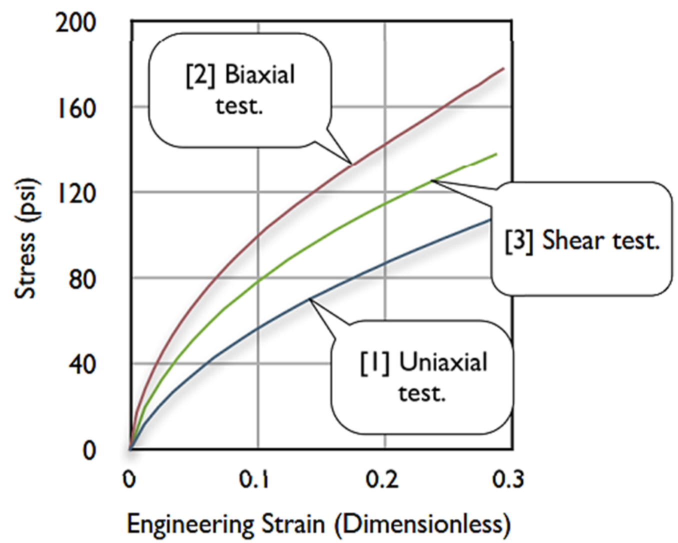
Nonlinear & Contact – Ex.17 (來源：成功大學李輝煌教授)

非線性材料

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。

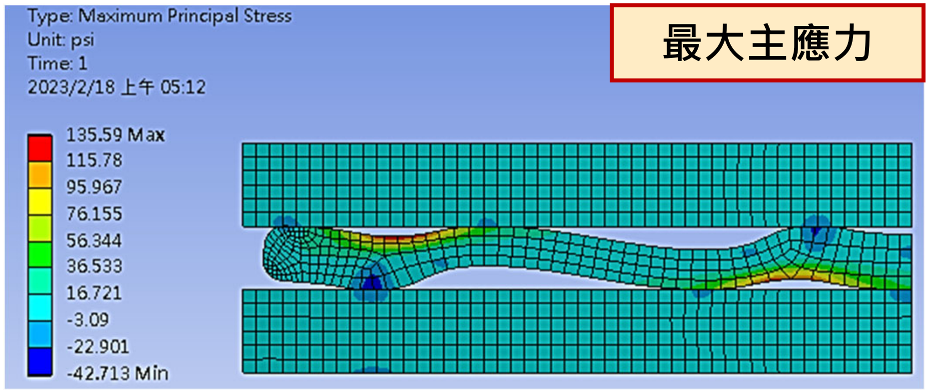
本習題將學習如何藉由實驗數據輸入得到超彈性材料特性，並模擬封條受兩鋼板擠夾之力學行為。此次將以2D進行模型建構，並以PLANE STRAIN進行模擬後觀察其最大主應力(變)/最小主應力(變)/剪應力(變)。

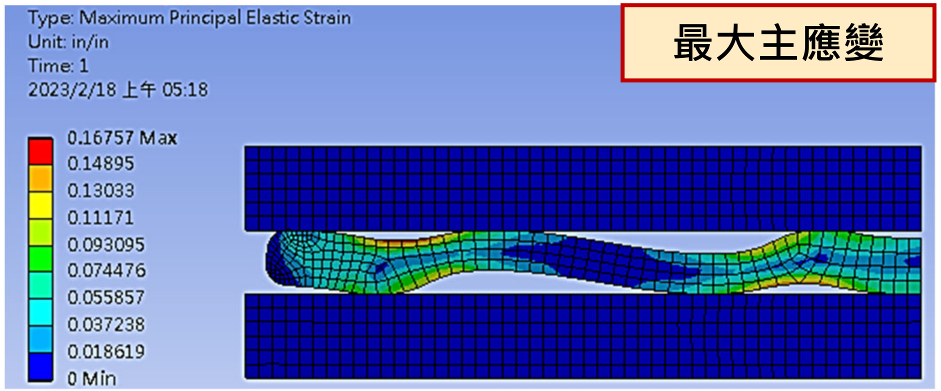


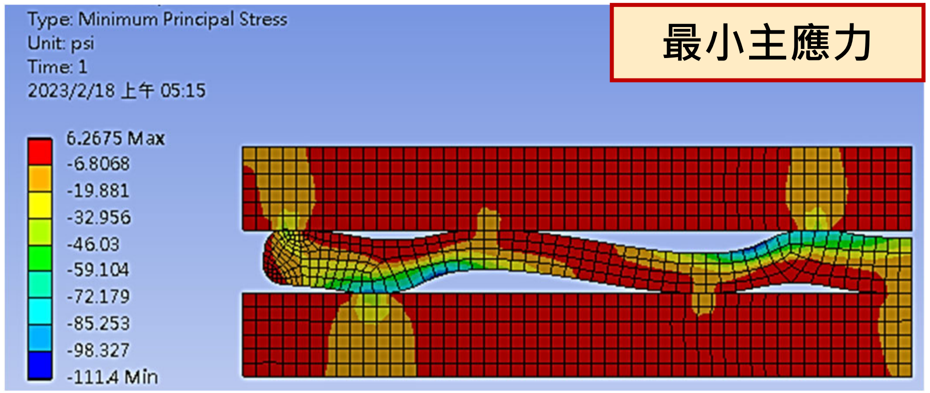


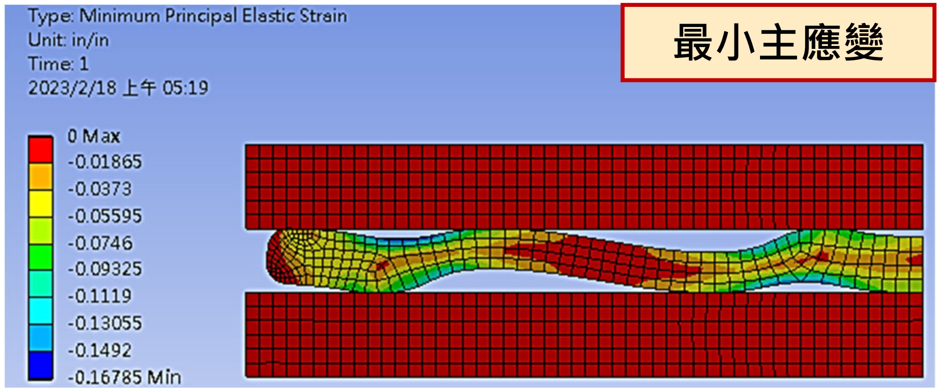
Nonlinear & Contact – Ex.17

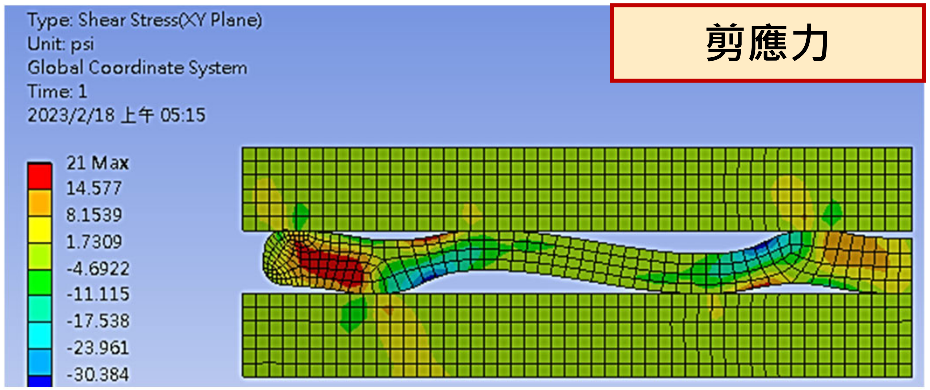
學習目標
- 材料data匯入
- Hyperelastic

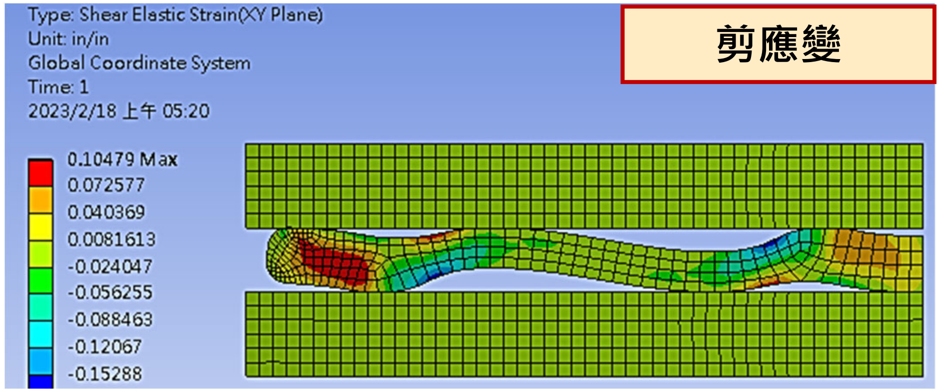








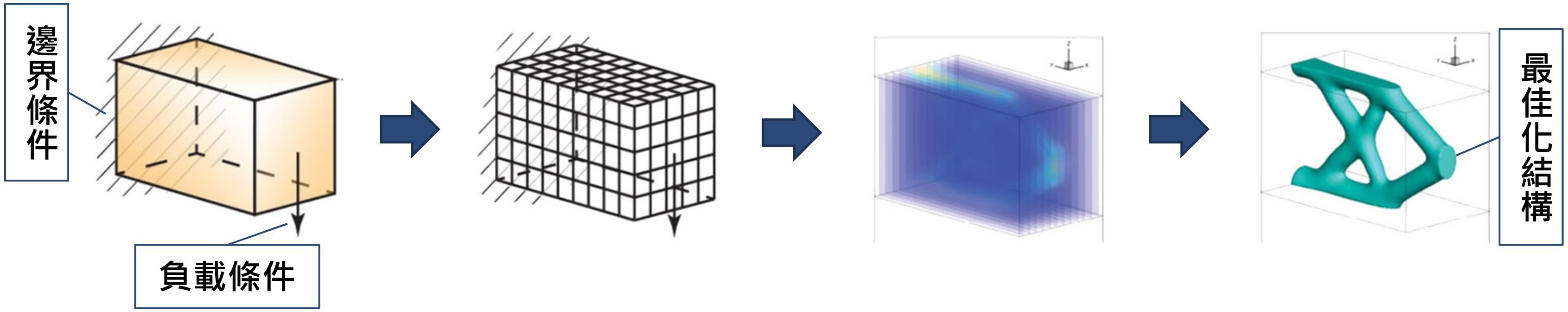




P82 對答案
POP, 2023/2/18

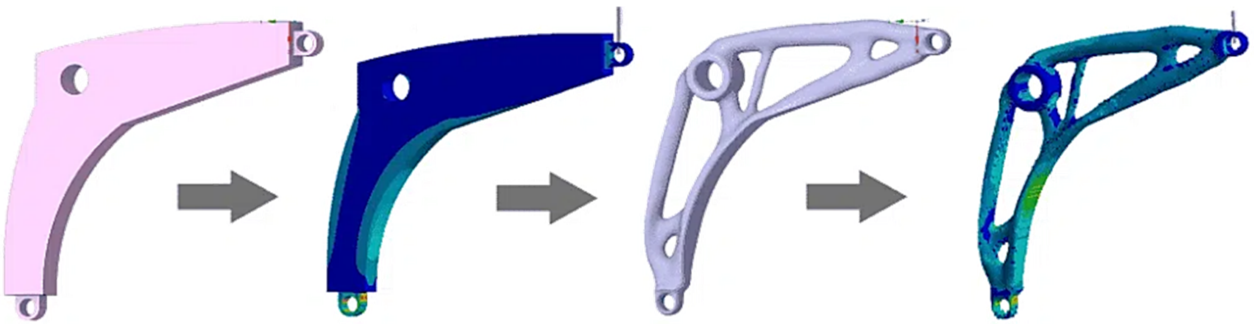
結構最佳化

- 拓樸最佳化(Topology Optimization)是一種結構優化技術，可自行定義設計範圍(Design Domain)，根據給定的限制條件與目標函數，解出符合給定條件之最佳結構



- 拓樸最佳化結合有限元素法及最佳化演算法，將最佳密度值分配給定義域中的每個元素，求得該結構之最佳材料分布情形

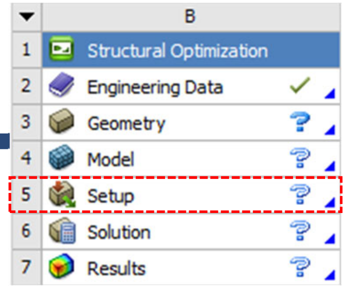
- 可實現在一定的結構強度要求下，將材料做出最有效之配置，以節省不必要的材料達到輕量化之目標



Sutradhar et. al., 2017

圖片來源: https://courses.ansys.com/index.php/courses/topology-optimization-using-ansys-mechanical/lessons/topology-optimization-of-a-bell-crank-lesson-1/

Introduction of ANSYS Workbench

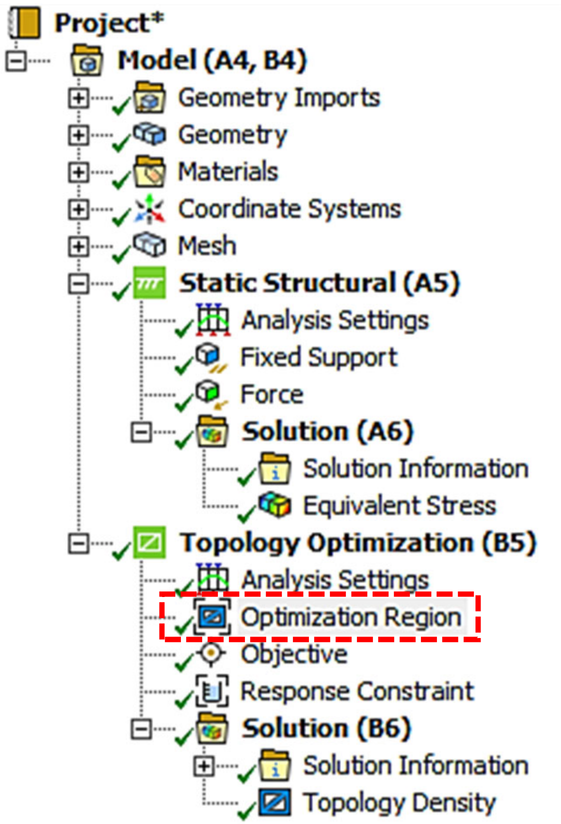


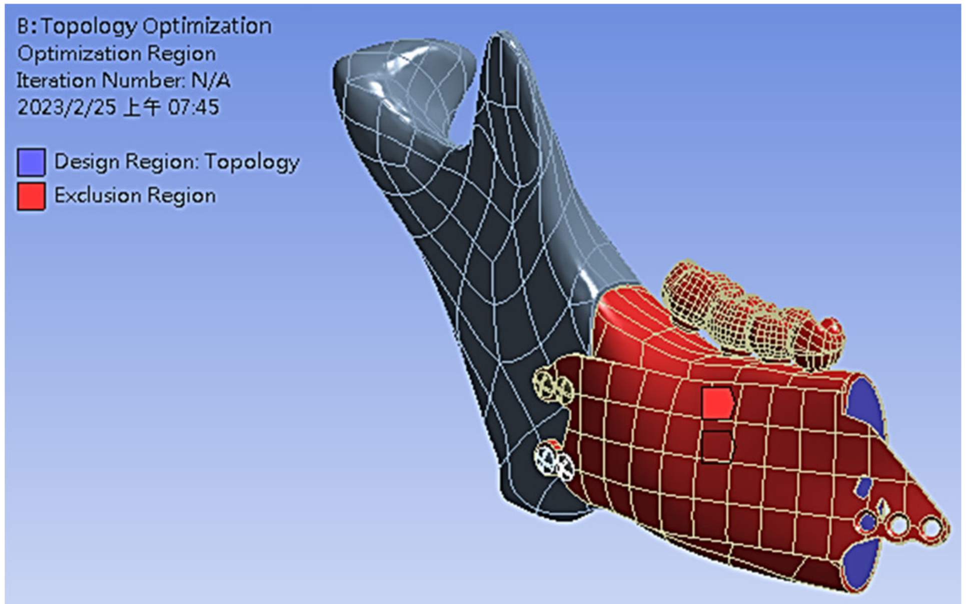
■ Topology Optimization (Structural Optimization)

- ➢ 設計範圍 - Optimization Region
  - ✓ Design Region
  - ✓ Exclusive Region

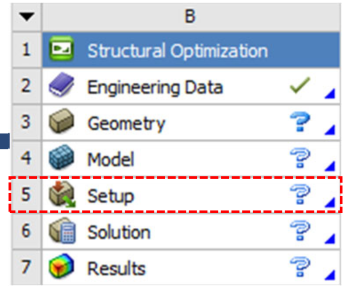
Details of "Optimization Region"

Design Region	
Scoping Method	Geometry Selection
Geometry	All Bodies
Exclusion Region	
Define By	Boundary Condition
Boundary Condition	All Boundary Conditions
Definition	
Suppressed	No
Optimization Option	
Optimization Type	Topology Optimization - Density Based

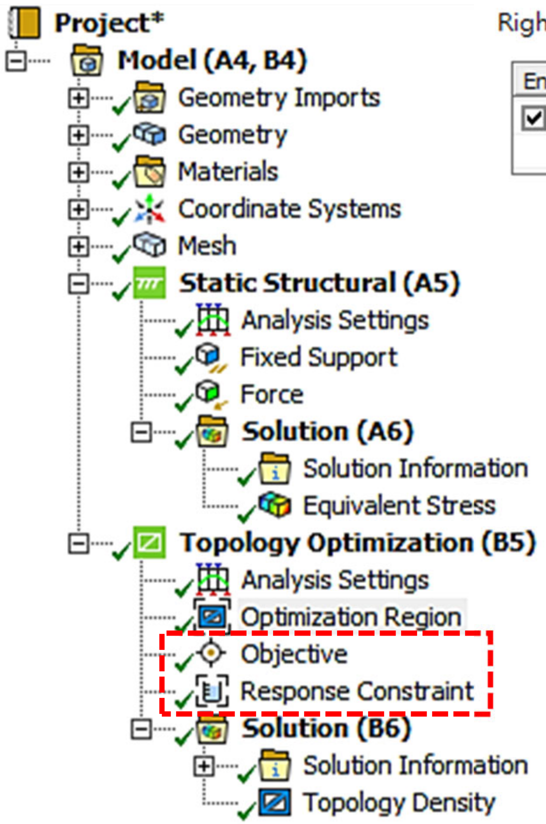


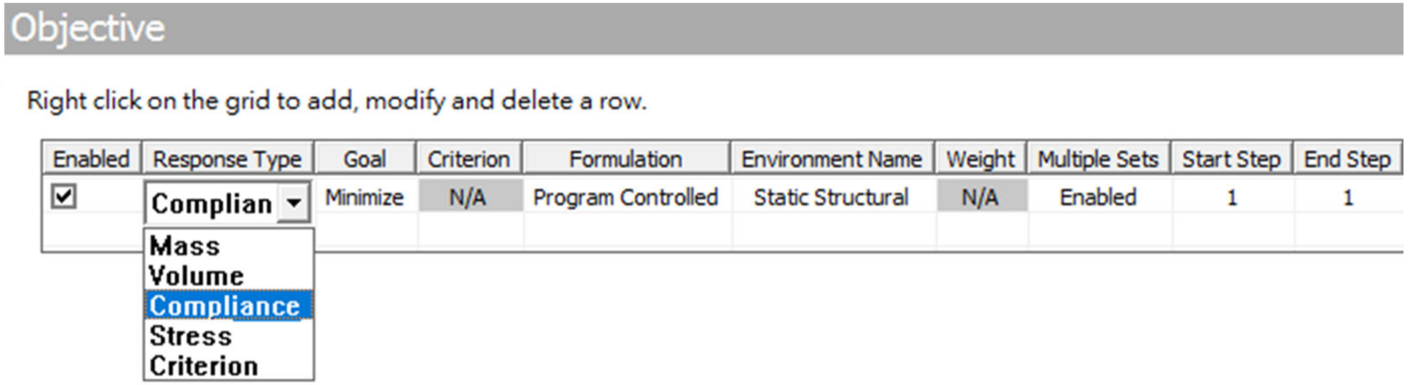


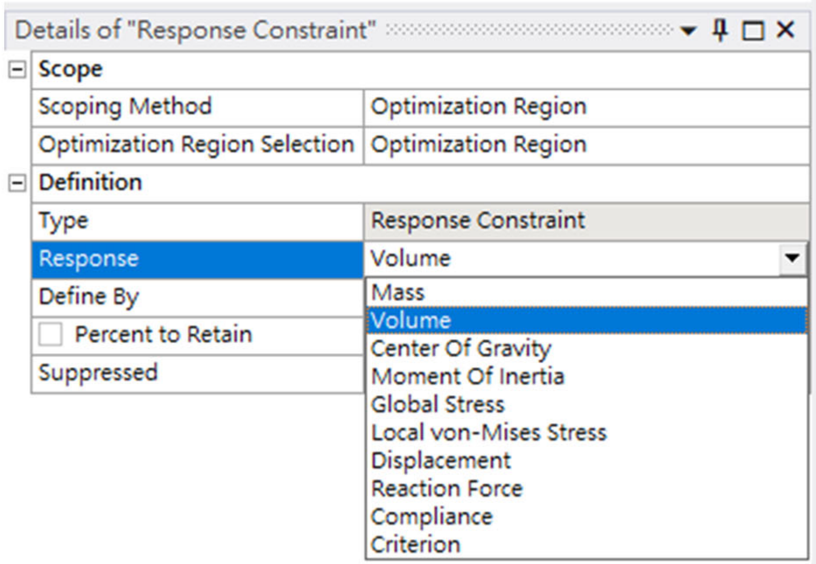
Introduction of ANSYS Workbench



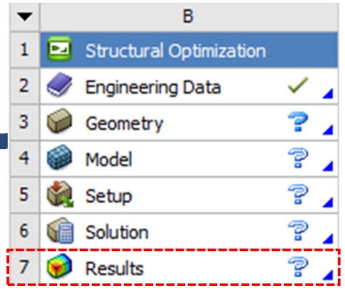
- **Topology Optimization (Structural Optimization)**
  - 目標函數 - Objective
  - 限制條件 - Response Constraint





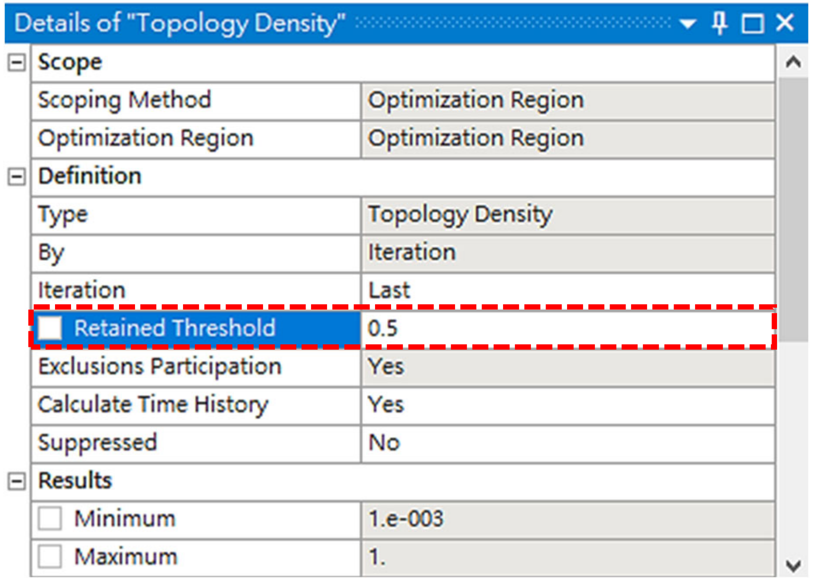


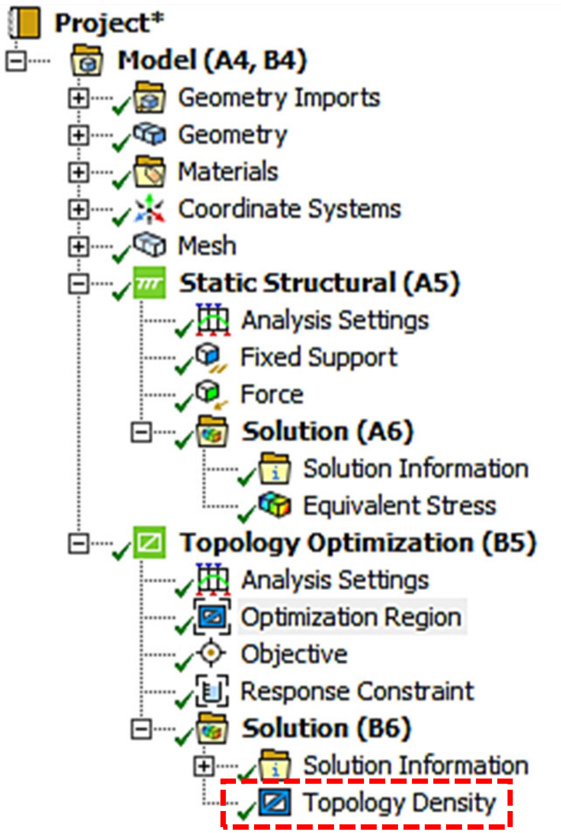
Introduction of ANSYS Workbench

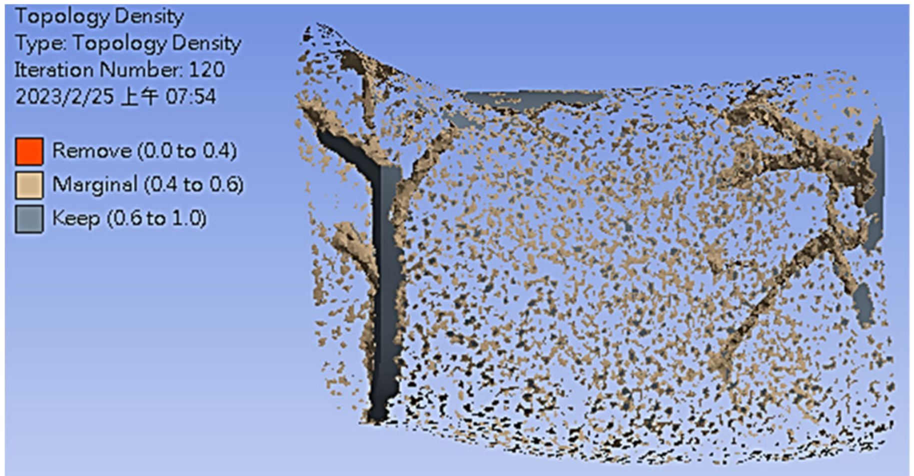


■ Structural Optimization (Topology Optimization)

➢ 最佳結構 - Topology Density(拓樸密度)

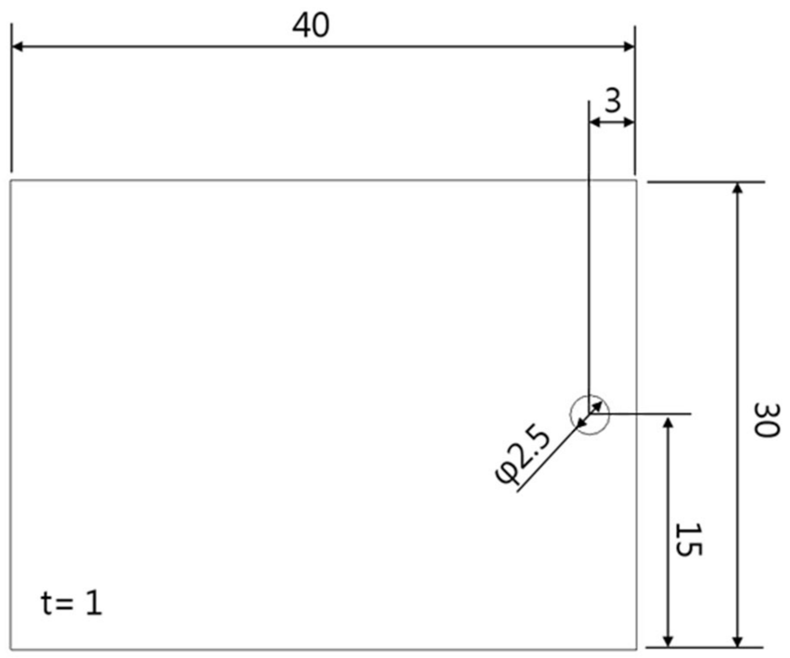




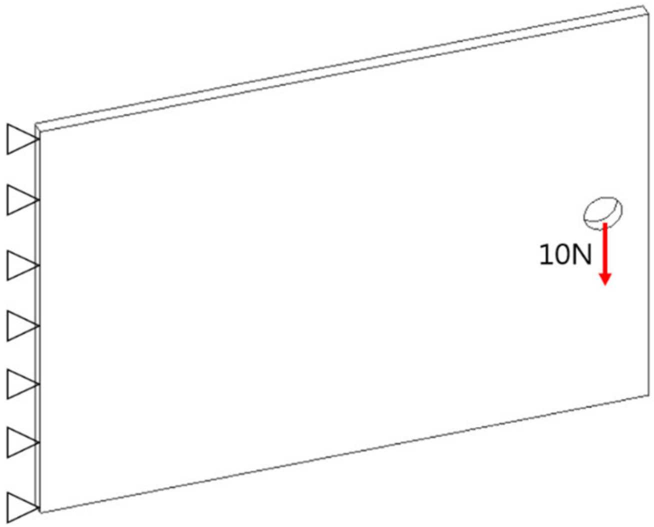


Structural Optimization – Ex.18

平板模型如圖所示，請針對該模型進行拓撲最佳化(**Topology Optimization**)，最佳化目標為剛度最大情形下，減少**40%**體積。材料使用**Structural steel**，網格形式採用四面體(**Tetrahedrons**)，網格尺寸為**1mm**，平板左側固定並於圓孔曲面上施以力量(**Y**方向，向下**10N**)。

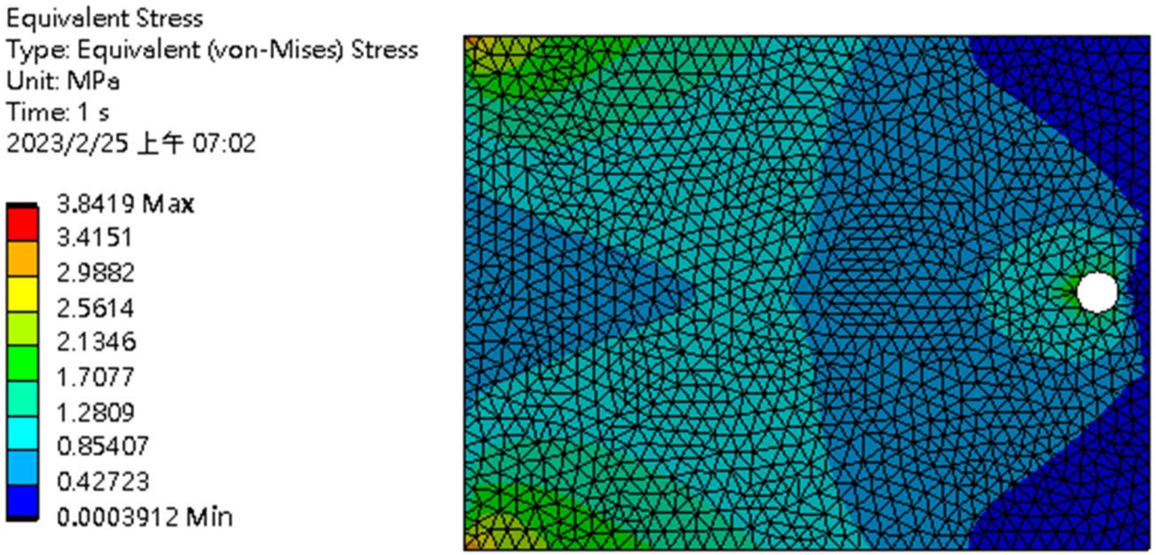


Unit : mm

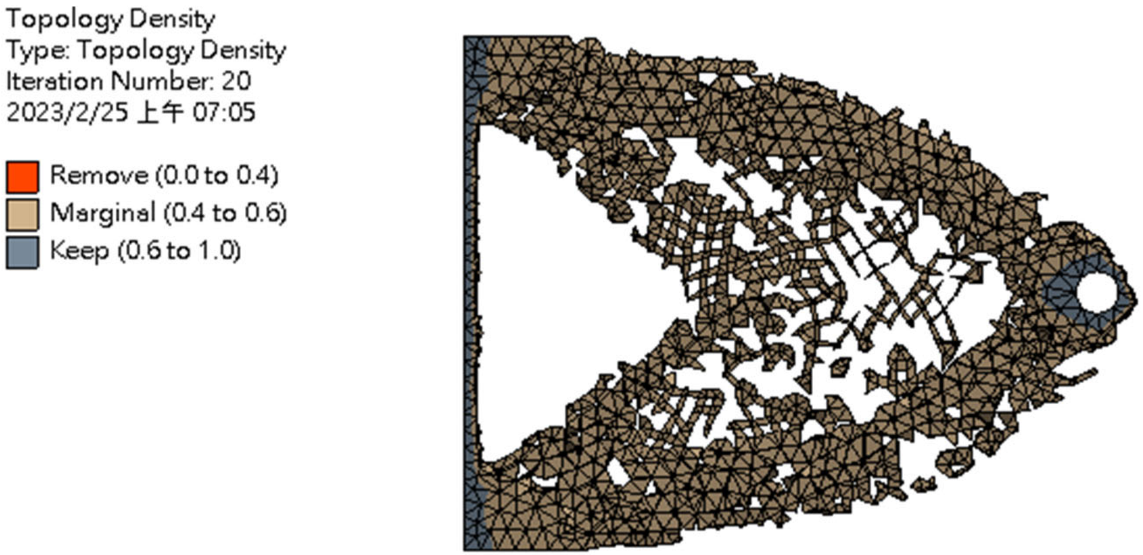


Structural Optimization – Ex.18

學習目標
- 拓樸最佳化設定



等效應力
Equivalent Stress

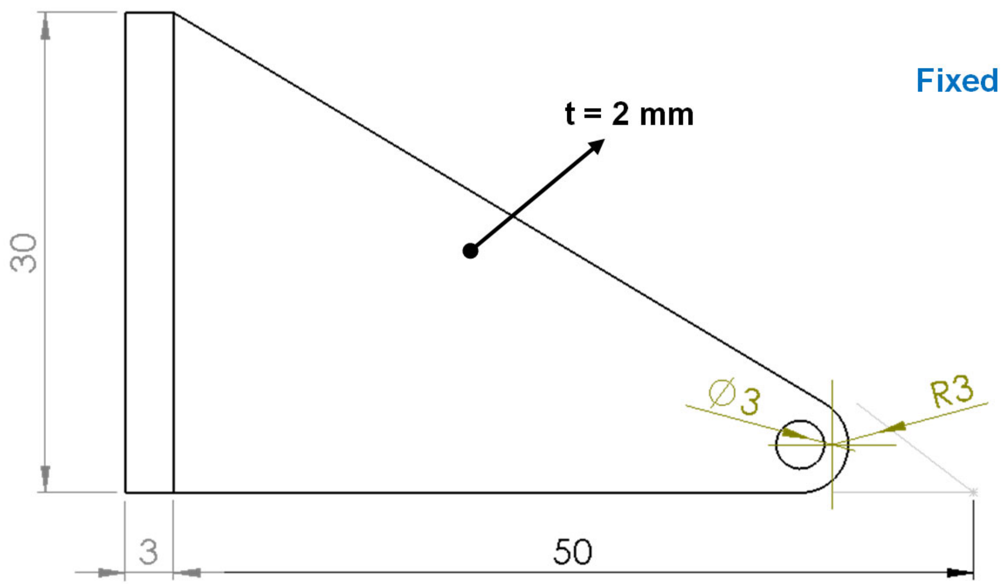


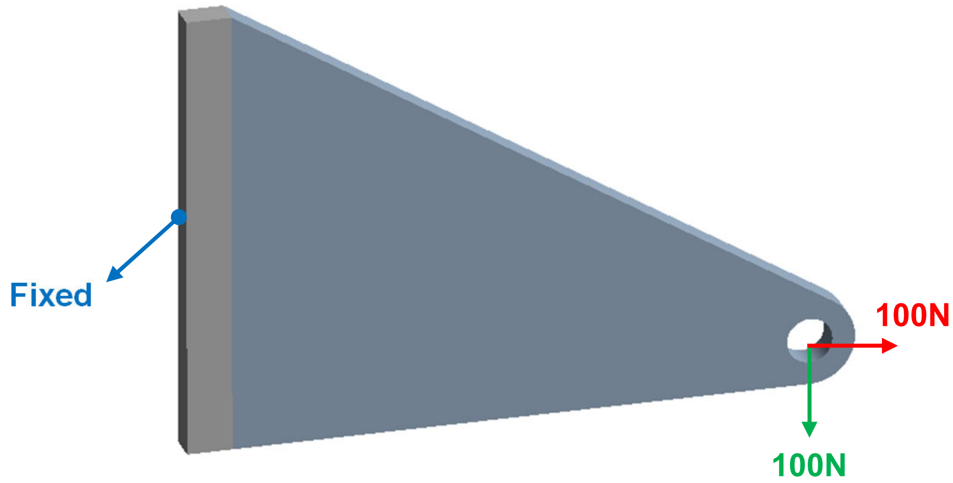
拓樸密度結構
Topology Density

Design Validation – Ex.19

三角bracket模型如圖所示，請針對該模型右側區塊進行拓撲最佳化(Topology Optimization)，最佳化目標為剛度最大情形下，減少50%體積。材料使用Structural steel，網格尺寸為1mm，左側固定，右側圓孔曲面上施以力量，並利用Design Validation將最佳化結構再次進行力學分析。試比較不同力量下，最佳化結果與結構應力分析結果差異。

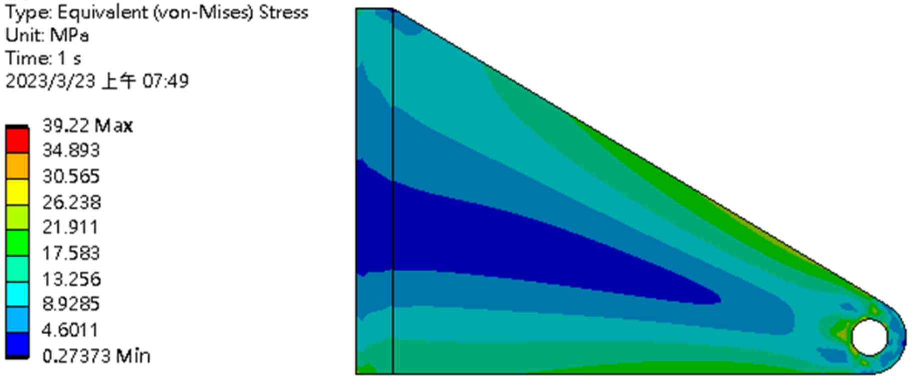
1) Y方向，向下100N
2) X方向，向右100N



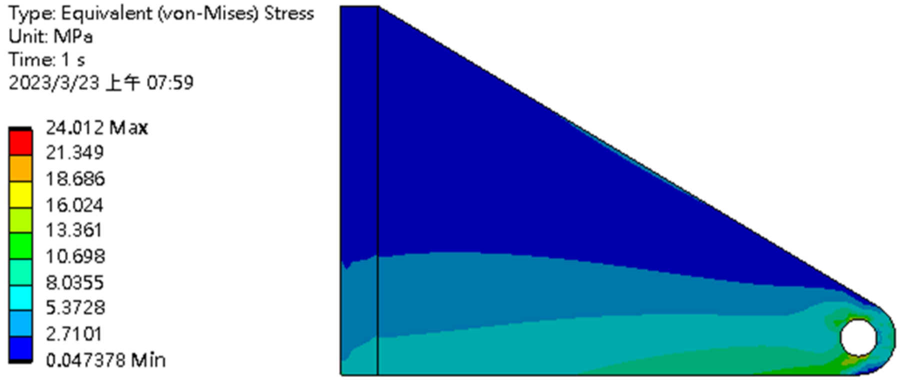


Design Validation – Ex.19

學習目標
- Design Validation
- SpaceClaim基礎修模



原始模型等效應力(向下)
Equivalent Stress



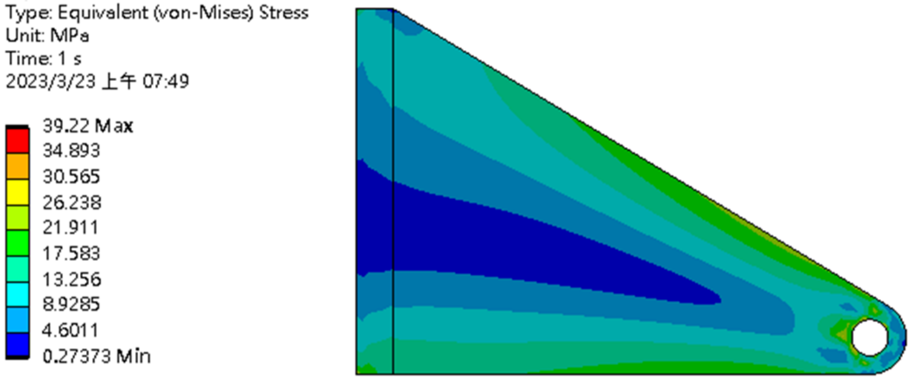
原始模型等效應力(向右)
Equivalent Stress

拓樸密度結構(向下)
Topology Density

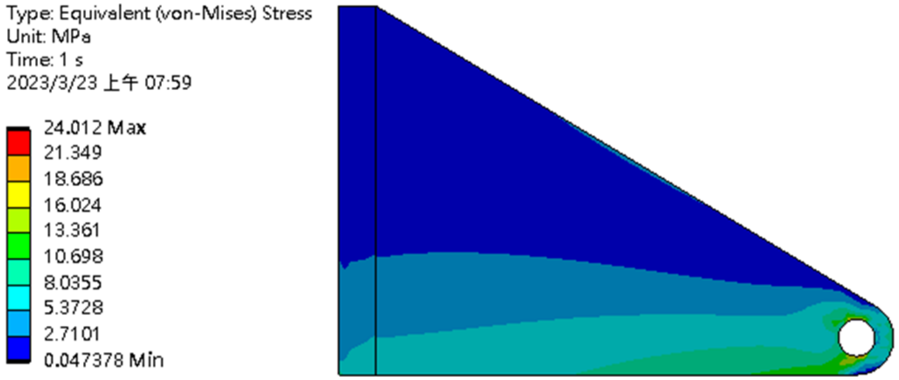
拓樸密度結構(向右)
Topology Density

Design Validation – Ex.19

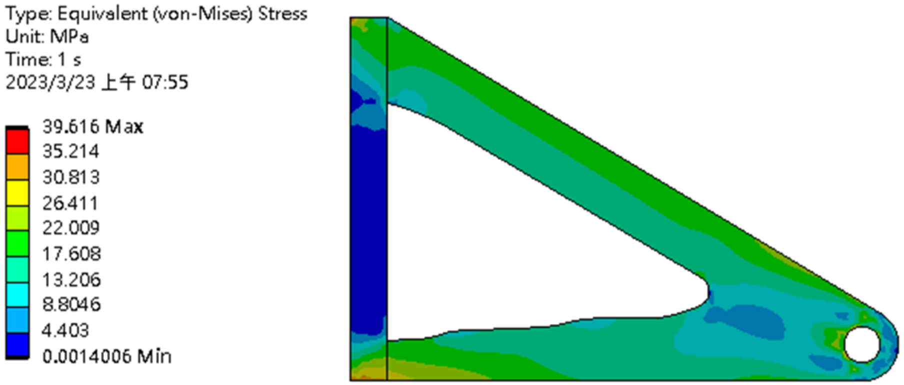
學習目標
- Design Validation
- SpaceClaim基礎修模



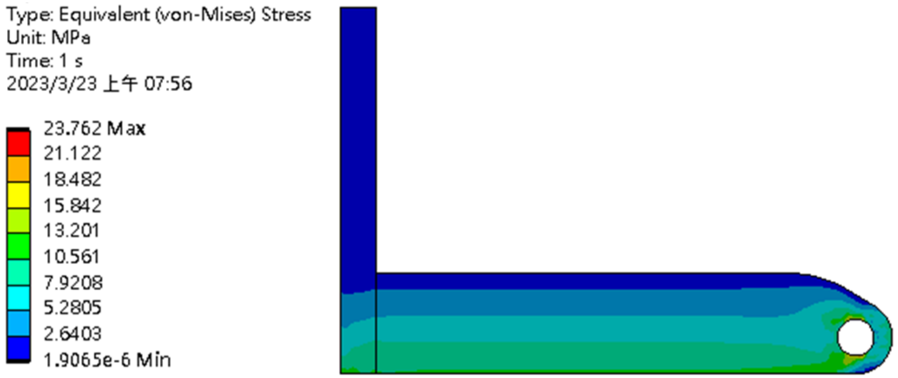
原始模型等效應力(向下)
Equivalent Stress



原始模型等效應力(向右)
Equivalent Stress



拓樸結構(向下)等效應力
Equivalent Stress



拓樸結構(向右)等效應力
Equivalent Stress

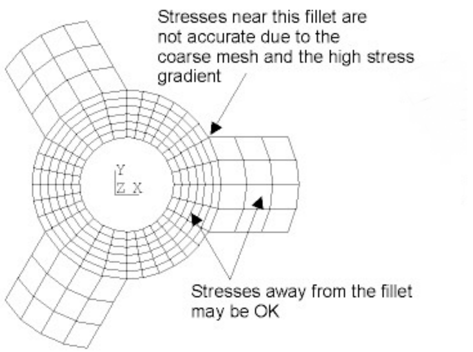
Submodeling

■ Understanding Submodeling

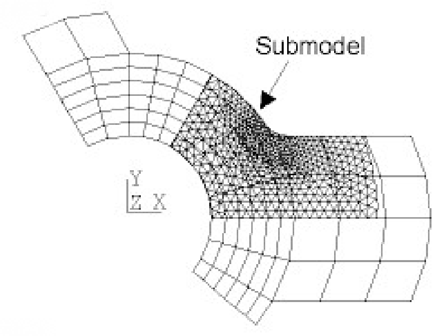
- In finite element analysis, the finite element mesh is sometimes too coarse to produce satisfactory results in a specific region of interest, such as a stress concentration region in a stress analysis as shown in Figure

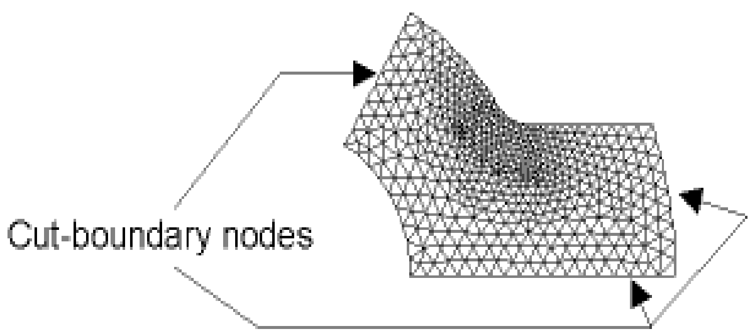
■ To obtain more accurate results in such a region, you have two options:

- (a) reanalyze the entire model with greater mesh refinement (time-consuming and costly)
- (b) generate an independent, more finely meshed model of only the region of interest and analyze it. Obviously, option (submodeling technique)



(a)

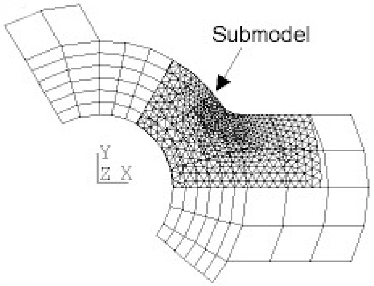


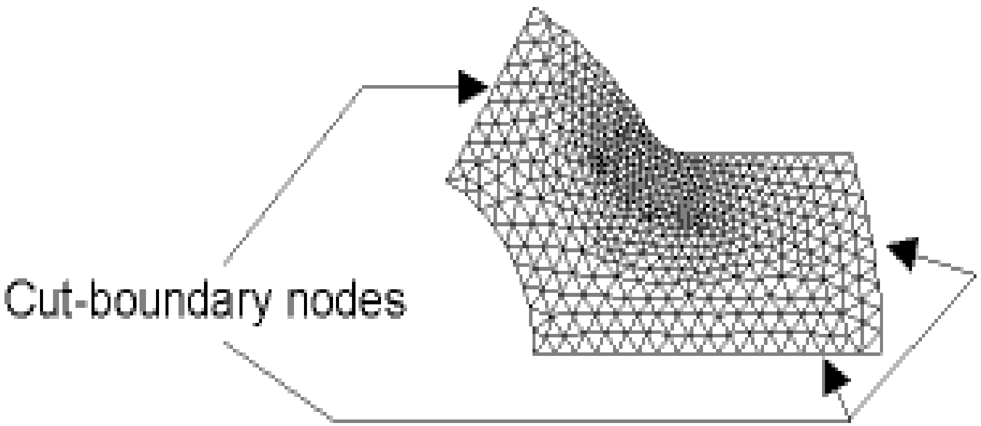


Submodeling

Submodeling

- the cut-boundary displacement method or the specified boundary displacement method. The cut boundary is the boundary of the submodel which represents a cut through the coarse model. Displacements calculated on the cut boundary of the coarse model are specified as boundary conditions for the submodel.



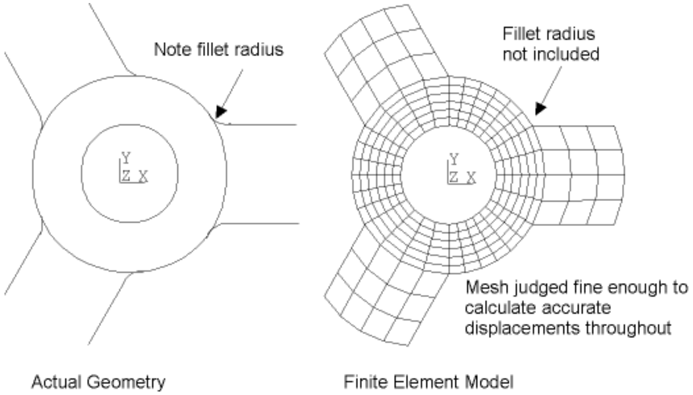


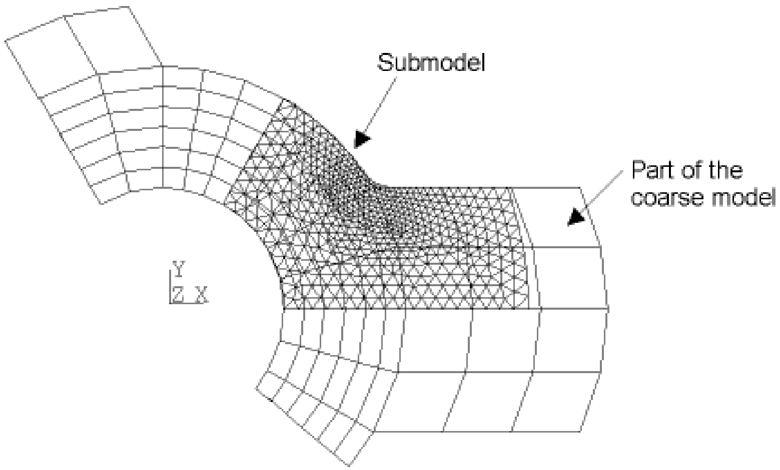
Submodeling is based on St. Venant's principle

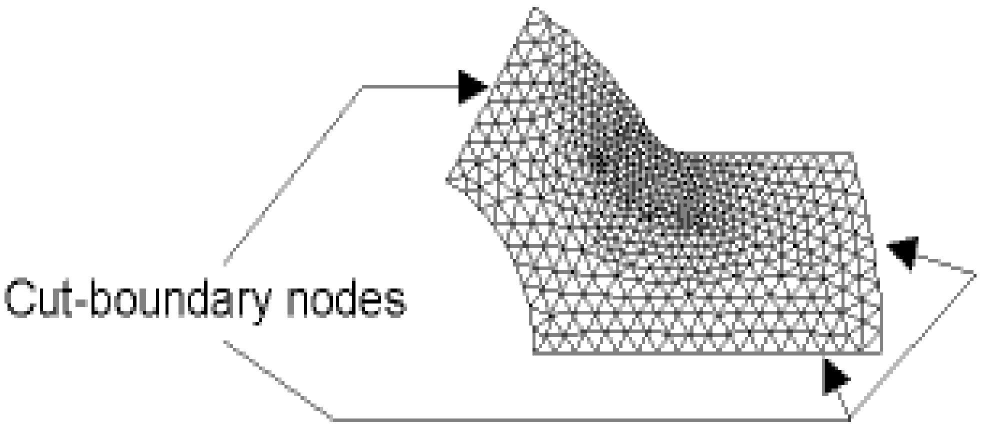
- If an actual distribution of forces is replaced by a statically equivalent system, the distribution of stress and strain is altered only near the regions of load application. The principle implies that stress concentration effects are localized around the concentration; therefore, if the boundaries of the submodel are far enough away from the stress concentration, reasonably accurate results can be calculated in the submodel.

Submodeling

- How to do the submodeling technique?
  1. Create and analyze the coarse model.
  2. Create the submodel.
  3. Perform cut boundary interpolation.
  4. Analyze the submodel.





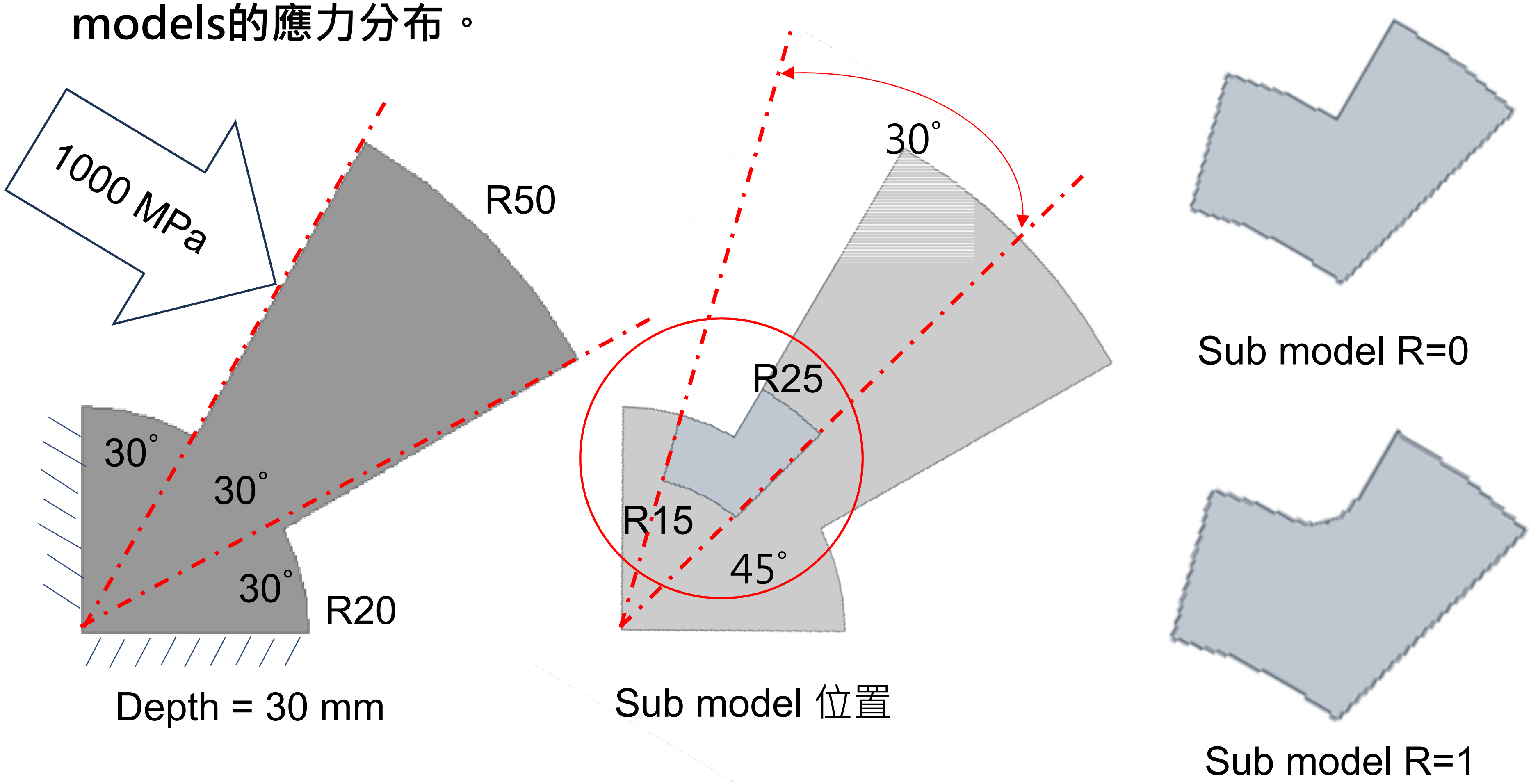


Submodeling

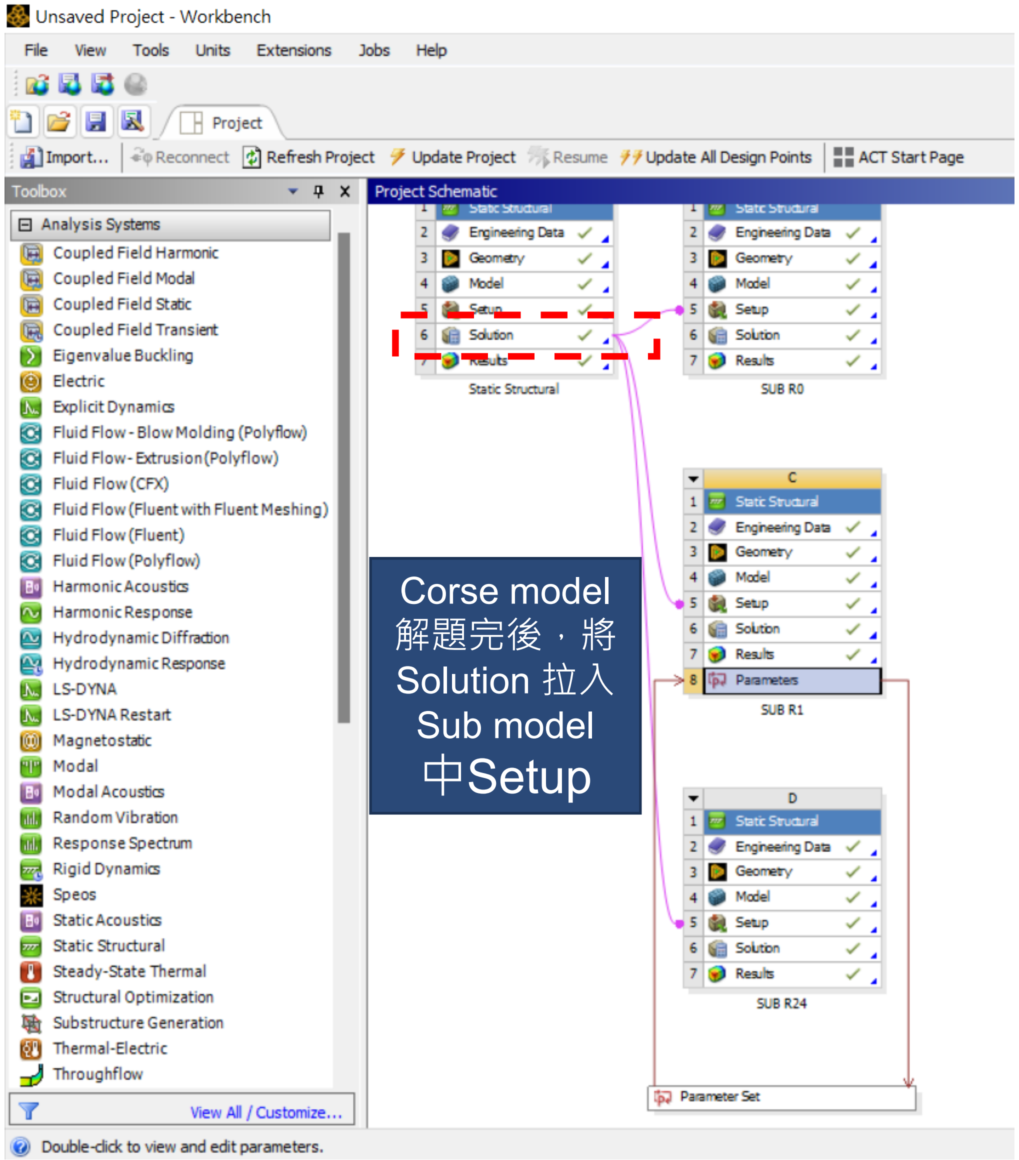
- The submodeling technique has other advantages:
  - It reduces, or even eliminates, the need for complicated transition regions in solid finite element models.
  - It enables you to experiment with different designs for the region of interest (different fillet radii, for example).
  - It helps you in demonstrating the adequacy of mesh refinements.

Example 20

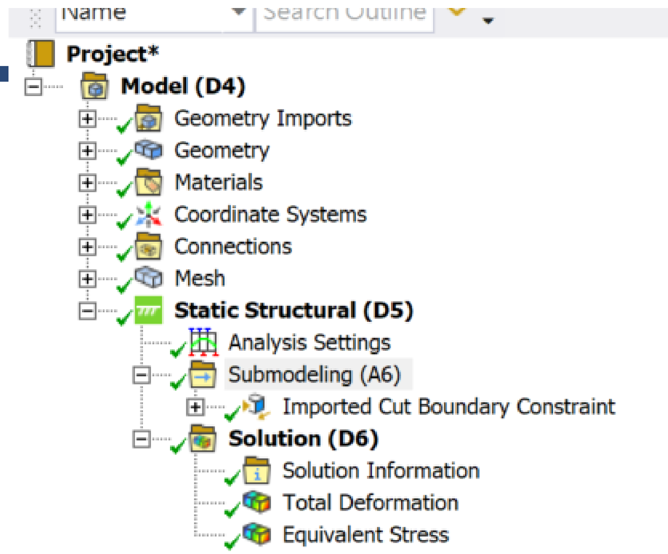
■ 如圖有風扇結構一部份，尺寸/負荷/邊界條件如圖，請建構出Corse 模型並分析，及建構如圖之Sub model (無倒角R0/倒角R1/倒角R2.XX)，其中Sub model R2.X 請利用參數化算出可能值後再建構，並比較Corse及各Sub models的應力分布。



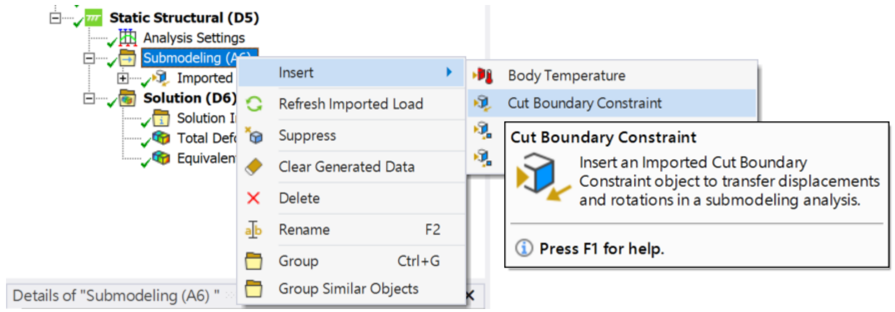
Example 20



Corse model 解題完後，將 Solution 拉入 Sub model 中Setup



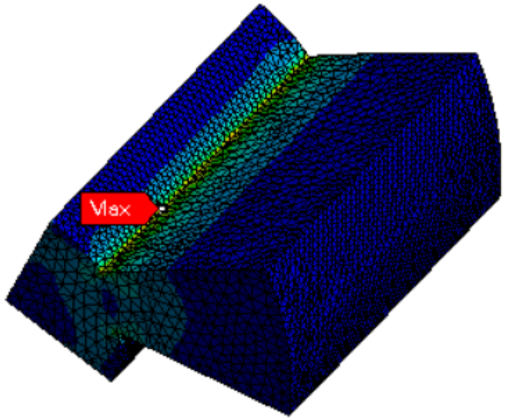
對Sub model mesh並拿掉Corse model 中fix support及force，即會在Static structure中出現Submodeling



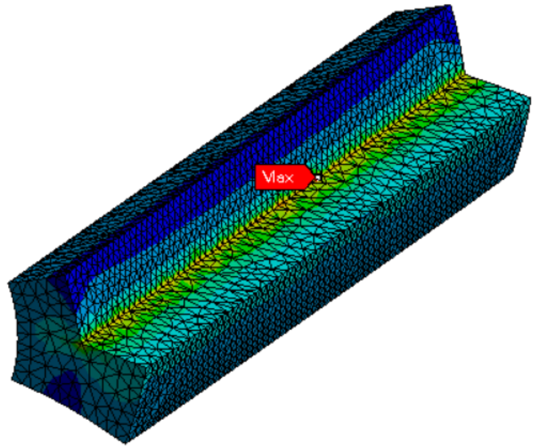
Submodeling 按右鍵選擇 Cut Boundary Constraint 選要輸入的面

Example 20

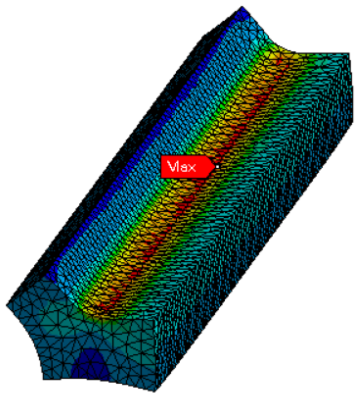
R2.4的決定，可以利用R角及最大應力值為參數(P)進行外部計算，待收斂後可再建構並計算一次Sub model R2.4



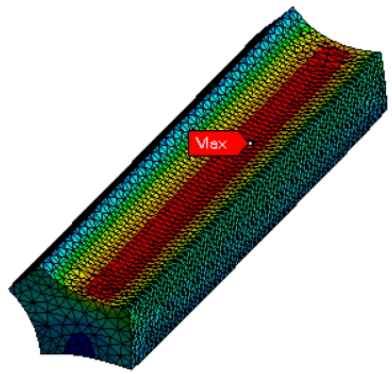
Corse model (40626)



Sub model R0 (49787)

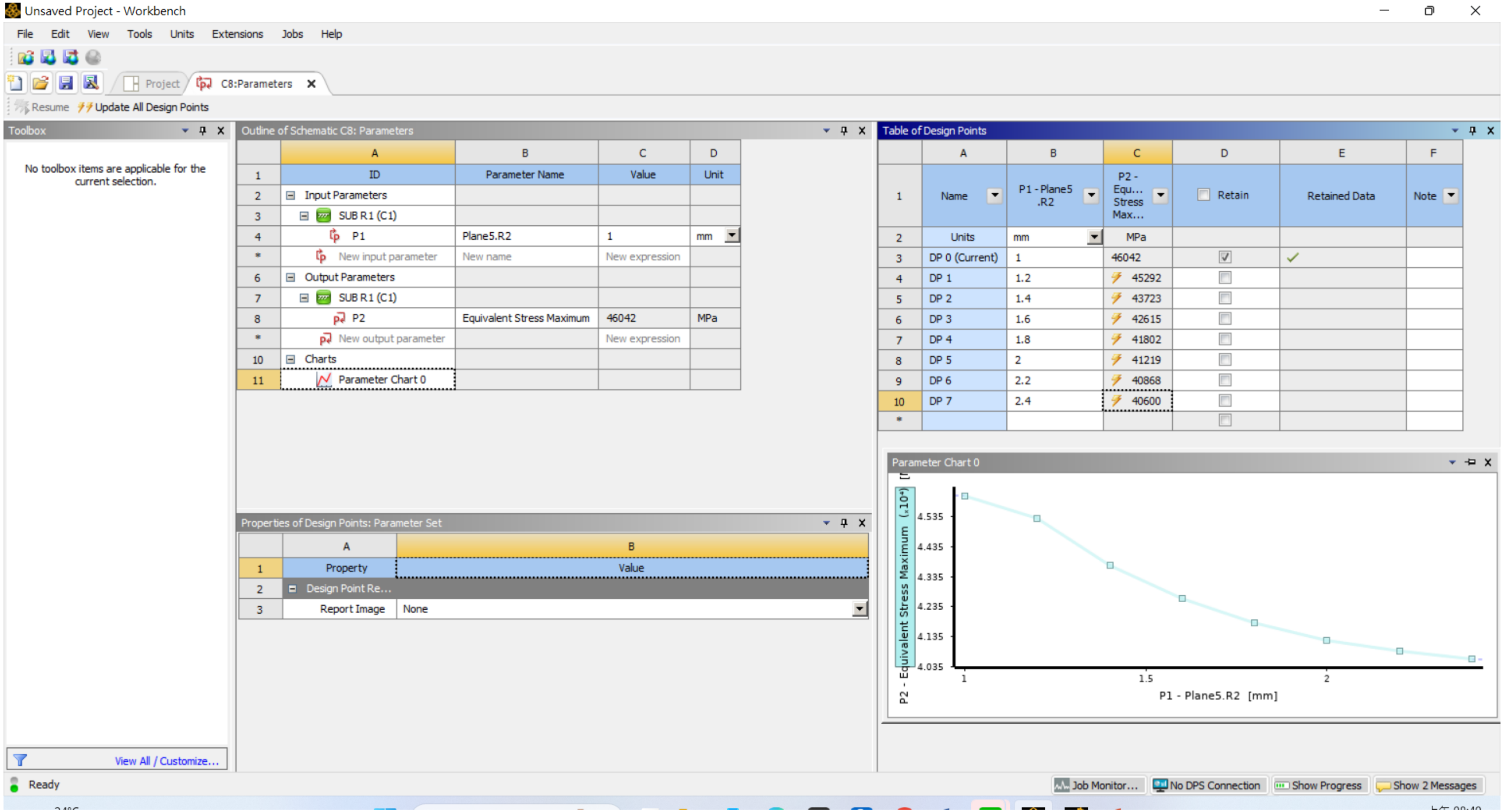


Sub model R1 (46415)



Sub model R2.4 (40478)

Example 20


Unsaved Project - Workbench
File Edit View Tools Units Extensions Jobs Help
Project
C8:Parameters
Resume
Update All Design Points
Toolbox
No toolbox items are applicable for the current selection.
View All / Customize...
Outline of Schematic C8: Parameters
A B C D
1 ID Parameter Name Value Unit
2 Input Parameters
3 SUB R1 (C1)
4 P1 Plane5.R2 1 mm
* New input parameter New name New expression
6 Output Parameters
7 SUB R1 (C1)
8 P2 Equivalent Stress Maximum 46042 MPa
* New output parameter New expression
10 Charts
11 Parameter Chart 0
Properties of Design Points: Parameter Set
A B
1 Property Value
2 Design Point Re...
3 Report Image None
Table of Design Points
A B C D E F
1 Name P1 - Plane5 .R2 P2 - Equ... Stress Max... Retain Retained Data Note
2 Units mm MPa
3 DP 0 (Current) 1 46042
4 DP 1 1.2 45292
5 DP 2 1.4 43723
6 DP 3 1.6 42615
7 DP 4 1.8 41802
8 DP 5 2 41219
9 DP 6 2.2 40868
10 DP 7 2.4 40600
Parameter Chart 0
P2 - Equivalent Stress Maximum (x10⁴) [MPa]
4.535
4.435
4.335
4.235
4.135
4.035
1
1.5
2
P1 - Plane5.R2 [mm]
Ready
Job Monitor...
No DPS Connection
Show Progress
Show 2 Messages