Computer Aided Engineering -Part 1 ANSYS-1

電腦輔助工程分析 -Part 1 ANSYS-1

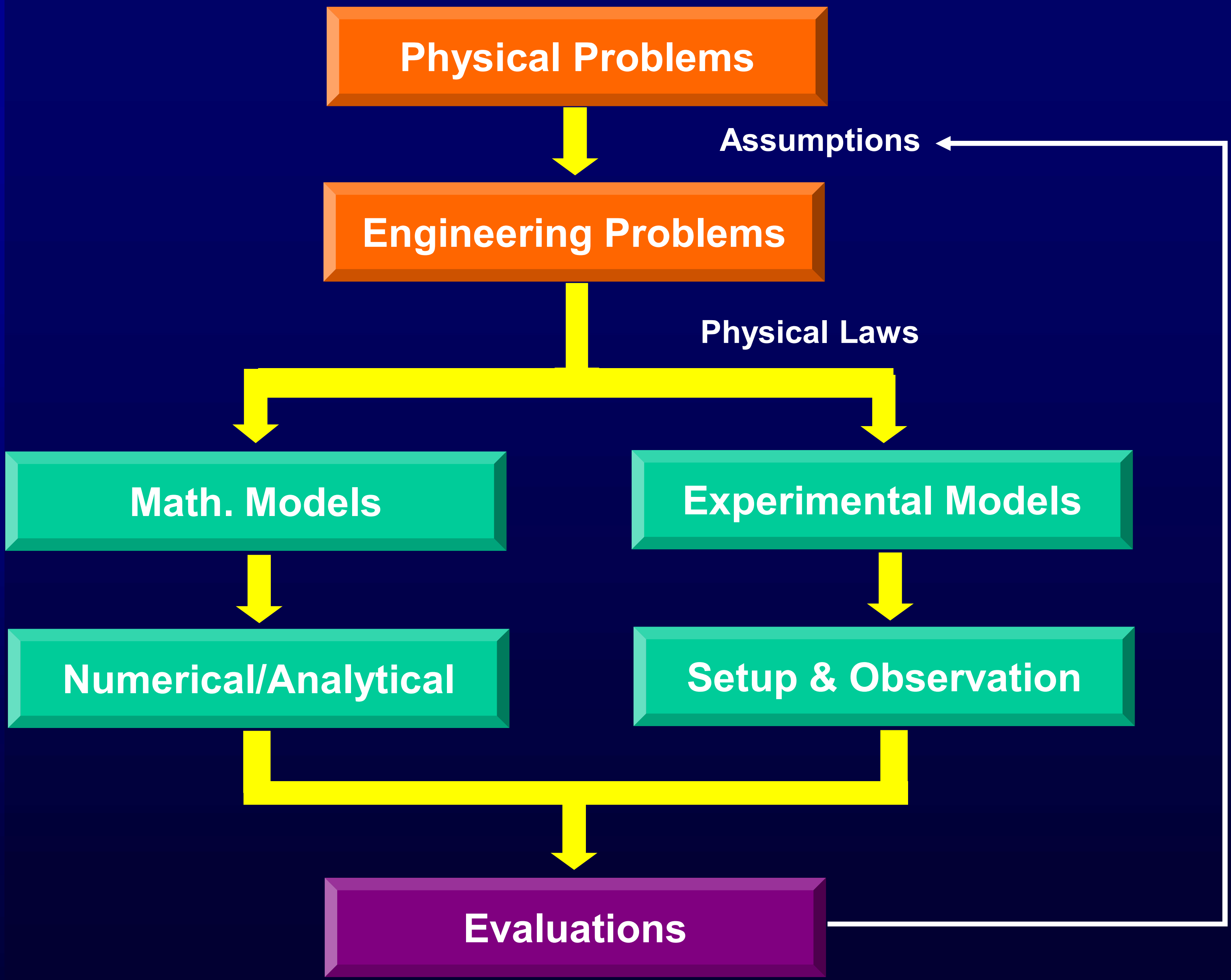
林峻立 特聘教授

Chun-Li Lin, Ph.D.

國立陽明交通大學 生物醫學工程系

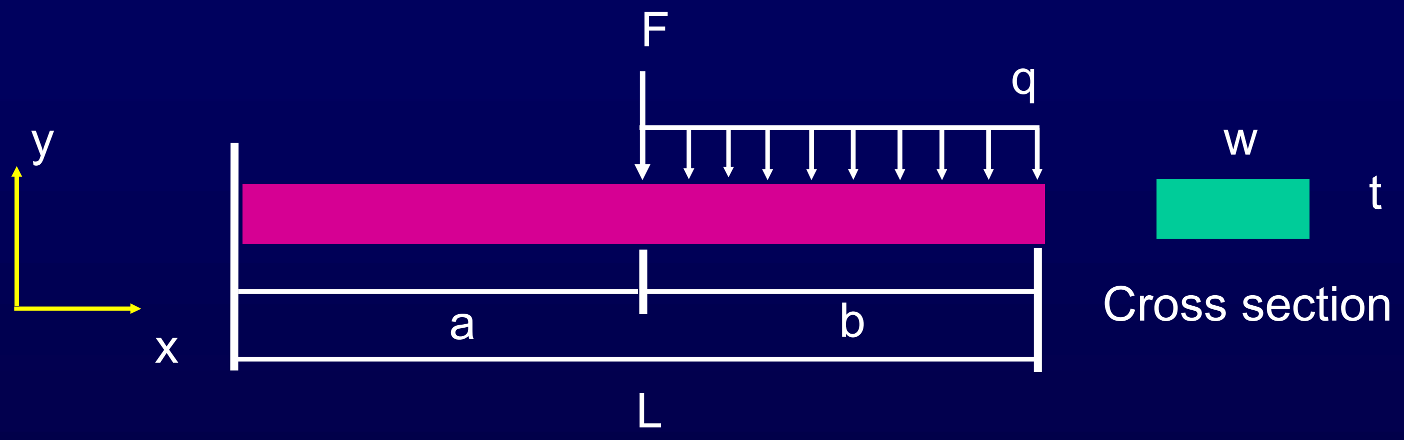
Department of Biomedical Engineering,
National Yang Ming Chiao Tung University.

2022/02


Physical Problems
Assumptions
Engineering Problems
Physical Laws
Math. Models
Experimental Models
Numerical/Analytical
Setup & Observation
Evaluations

Fundamental Concepts in FEM

- **Analytical Method**



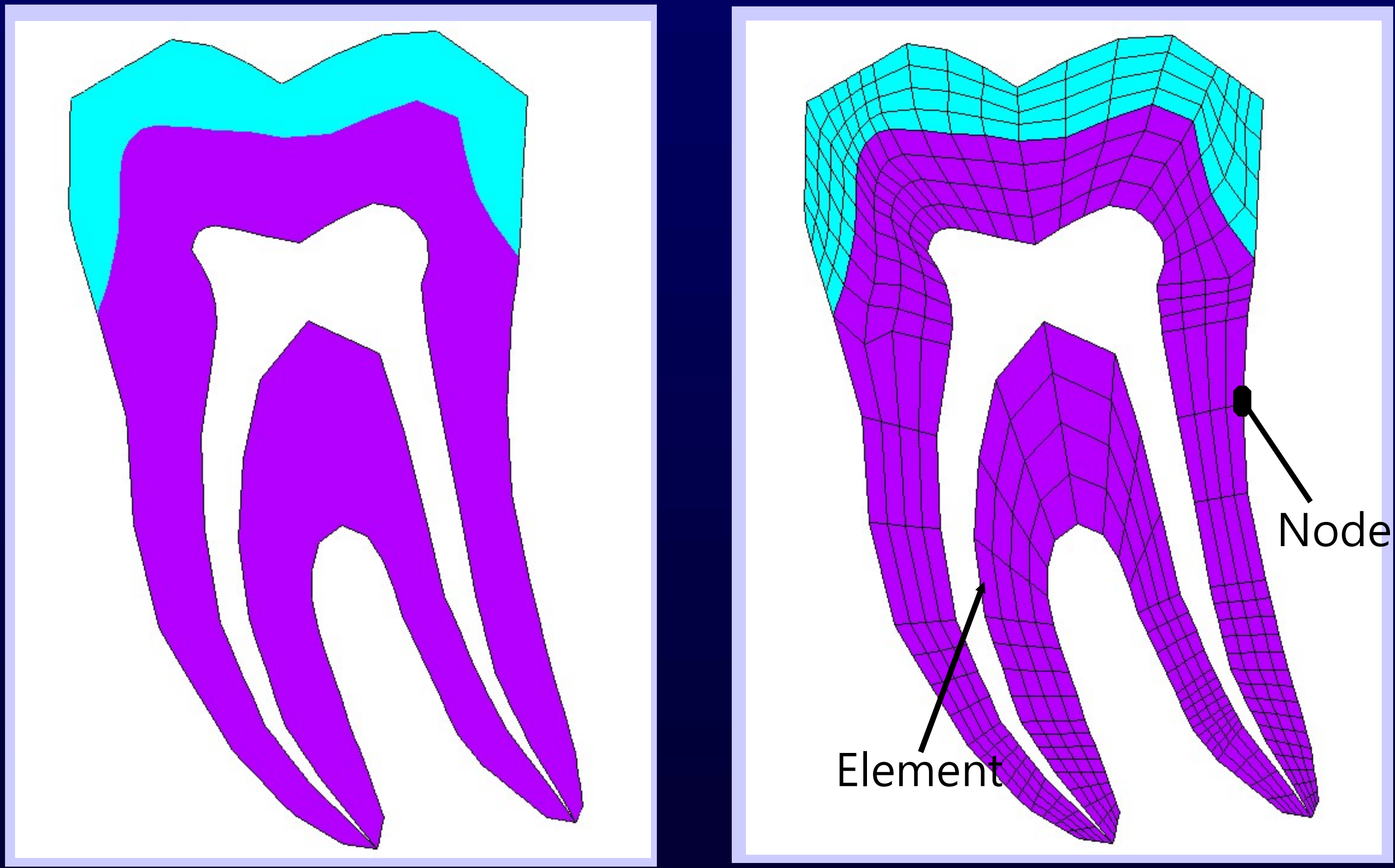
$$y=Fa^3(3L-a)/6EI+q(3L^4-4a^3L+a^4)/24EI$$

- **Numerical Method**
  - **FEM,BEM, FDM, etc.**

Fundamental Concepts in FEM

- 實際的物理問題很難利用單一的微分方程式描述，更無法順利求其解析(analytical solution)解
- 有限元素法的精神是將複雜的幾何外型的結構物體切割成許多簡單的幾何形狀稱之為元素(element)
- 元素與元素間以“節點”(node)相連
- 由於元素是簡單的幾何形狀，故可順利寫出元素的力平衡方程式並求得節點上之變位、應變及應力等
- 藉由內插法求得元素內任意點的變位、應變及應力等

Fundamental Concepts in FEM

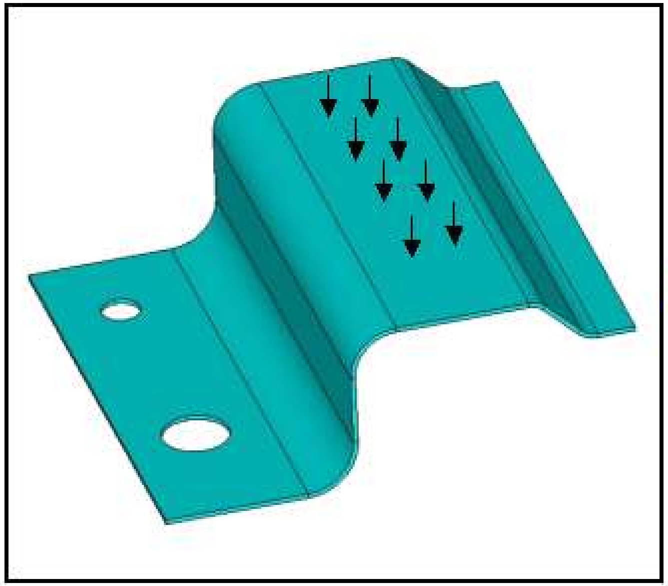


General Concept of CAE

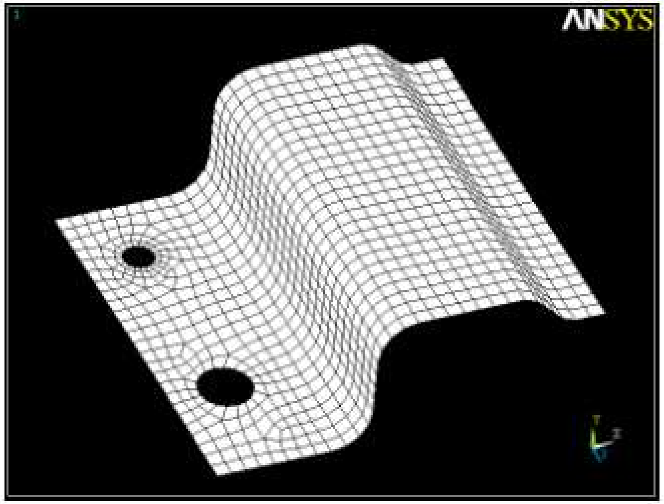
■所謂的CAE是指「computer-aided engineering」之縮寫，中文普遍稱為「電腦輔助工程」或「電腦輔助工程分析」。大略來說，只要是應用電腦來模擬分析實際物理問題，均可將其稱為CAE。

■CAE之分析類型很多，它包含了結構應力分析、振動分析、流體分析、熱傳分析、電磁場分析、塑膠射出成型流動分析、鑄造流動分析、機構運動與動力學分析等。

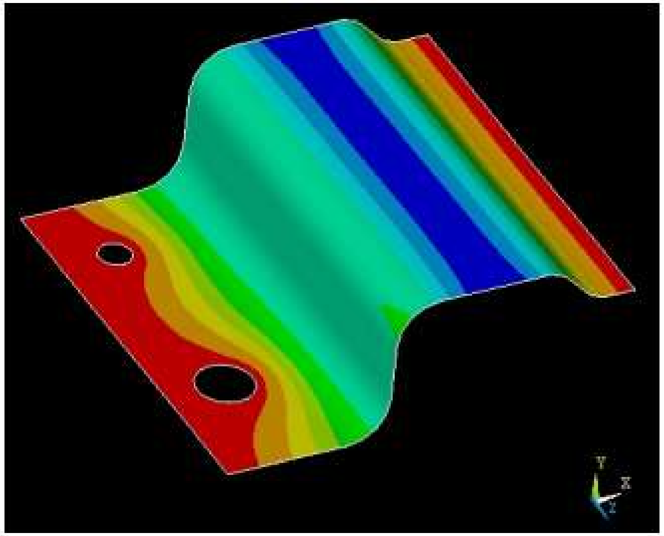
■以固體力學為例，其CAE之主流數值方法為有限元素法(finite element method, FEM)，亦可稱為有限元素分析(finite element analysis, FEA)，它的基本概念是把一個實際的連續性物體做離散化，分割成許多個元素(elements)與節點(nodes)，統稱為網格(mesh)，而每個元素均遵守力學基本理論模式。



(a)實際工程問題



(b)元素網格



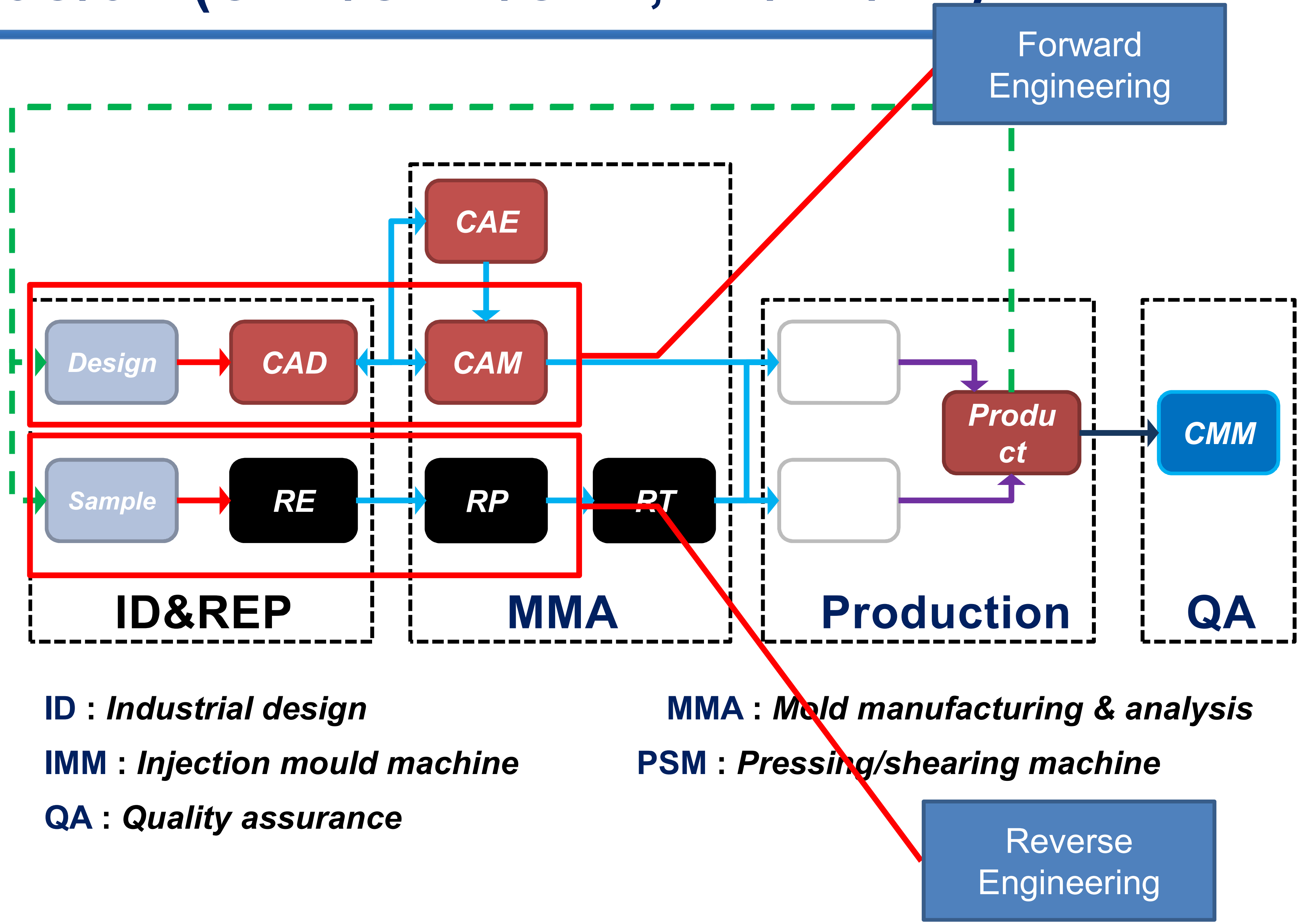
(c)模擬之變形

General Concept of CAE

■CAE和電腦輔助設計(computer-aided design,CAD)與電腦輔助製造(computer-aided manufacturing, CAM)同屬於電腦輔助之工具，近年來發展的CAD/CAM/CAE系統已成為工業界產品研發的利器，尤其成熟的CAD/CAM設計系統早已在許多台灣產業生根。

■近年來CAE也逐漸受到國內產業界的重視。面對市場上激烈的競爭，各公司提升研發能力已是刻不容緩的事，而CAE正可成為提升研發能力的一大利器。

3C/3R (CAD/CAM/CAE, RE/RP/RT)



ID : *Industrial design*

MMA : *Mold manufacturing & analysis*

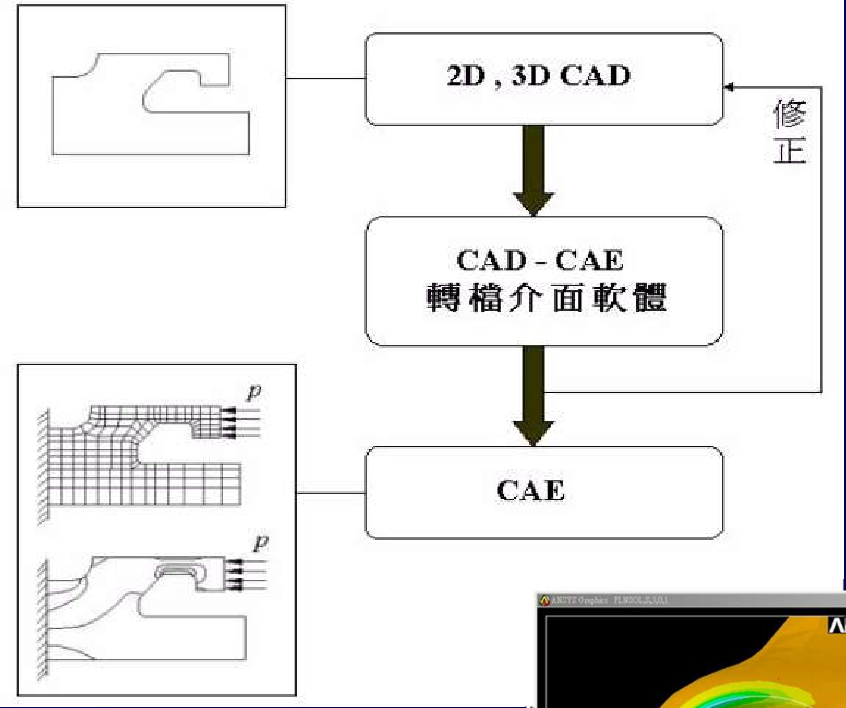
IMM : *Injection mould machine*

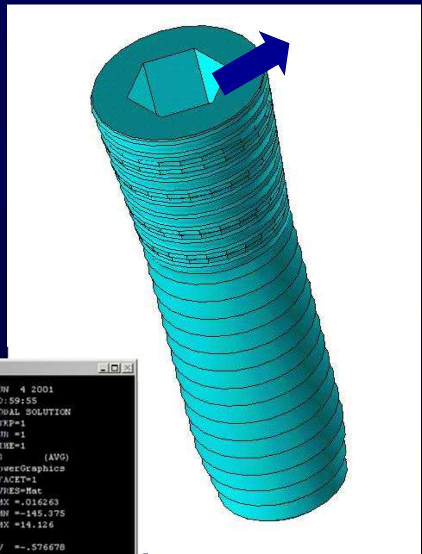
PSM : *Pressing/shearing machine*

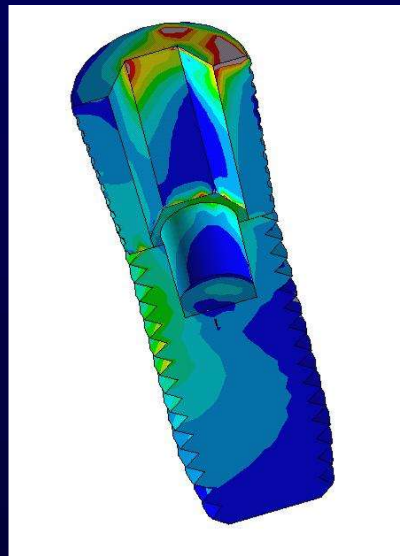
QA : *Quality assurance*

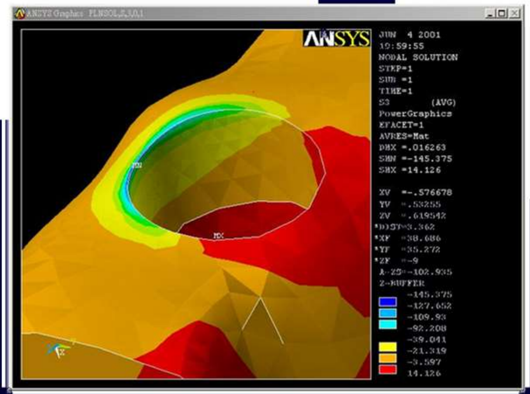
General Concept of CAE

- CAD\CAE\CAM









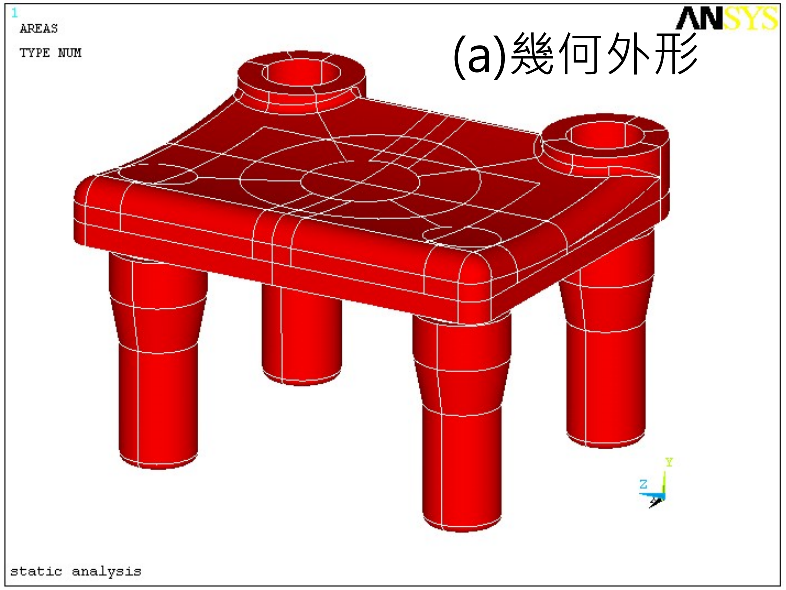
General Concept of CAE

- 有限元素法的歷史與軟體應用
  - 有限元素(finite element)」一詞最早出現於1960年，由學者Clough所提出。
  - 商業化有限元素軟體可分為泛用型(general-purpose)(廣泛分析型)與特用型(special-purpose)(特定分析)兩大主類(表)
  - 與CAD 整合軟體，通常CAE功能較差(Pro/Engineer)
  - 前後處理器，用來建模及顯示結果(Femap)
  - 有限元素分析可應用於結構、熱傳、流體、電磁、耦合場等物理領域，可應用之行業包括了機械工業、航空工業、土木業、電子電機業、電腦產業、半導體、微機電、生物科技等。

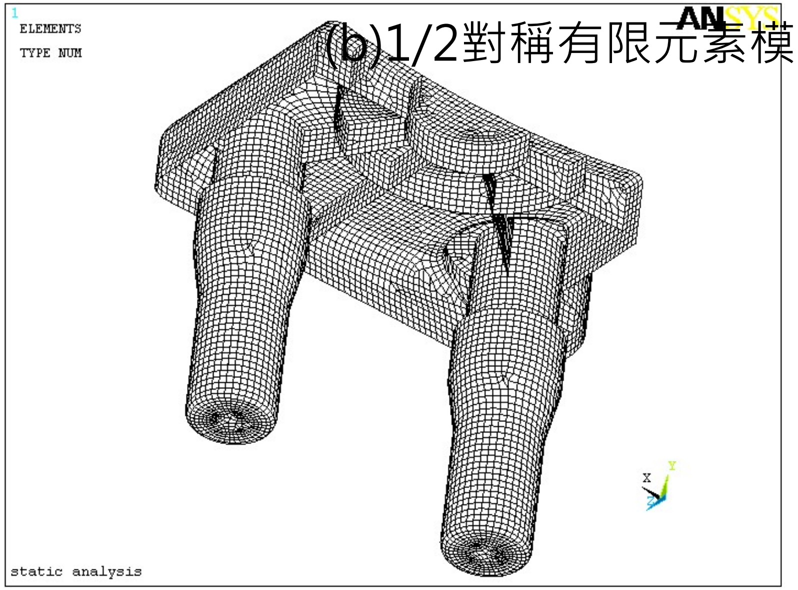
軟體名稱	類型	分析領域與功能	公司
ABAQUS	泛用型	結構、熱傳、耦合場等	ABAQUS, Inc.
ANSOFT Products (HFSS, Maxwell...)	泛用型	電磁場、電路等	Ansoft Corporation
ANSYS	泛用型	結構、熱傳、流體、電場、耦合場等	ANSYS, Inc.
COSMOS	泛用型	結構、熱傳、流體、電場、耦合場等	Structural Reseach & Analysis Corp.
DEFORM	特用型	金屬成型(鍛造、擠製、輥軋)、切削等	Scientific Forming Technologies Corporation
DYNAFORM	特用型	板金成型等	Engineering Technologies Associaates, Inc. (ETA)
Femap	前後處理 (亦可求解熱流)	前後處理、熱傳、流體等	UGS Corp.
Femlab	泛用型	結構、熱傳、流體、電磁、耦合場、化學工程等	COMSOL, Inc.
Hypermesh	前後處理	前後處理	Altair Engineering
I-Deas NX	泛用型 (與CAD整合)	結構、熱傳、流體等	UGS Corp.
LS-DYNA	泛用型	結構動態顯示法、結構靜態、熱傳、流體、耦合場等	Livermore Software Technology Corporation (LSTC)
MSC.Dytran	泛用型	結構動態顯示法、流固耦合場等	MSC. Software
MSC.Marc	泛用型	結構、熱傳、流體、電磁、耦合場等	MSC. Software
MSC.Nastran	泛用型	結構、熱傳、氣彈力學等	MSC. Software
MSC.Patran	前後處理	前後處理	MSC. Software
Pro/ENGINEER Simulation Software (Pro/MECHANICA)	泛用型 (與CAD整合)	結構、熱傳等	Parametric Technology Corporation
SAP2000	特用型	土木結構分析	Computers and Strucutres, Inc. (CSI)
STAAD Pro	特用型	土木結構分析	Research Engineers International

BME
NYMU

塑膠椅之力學分析

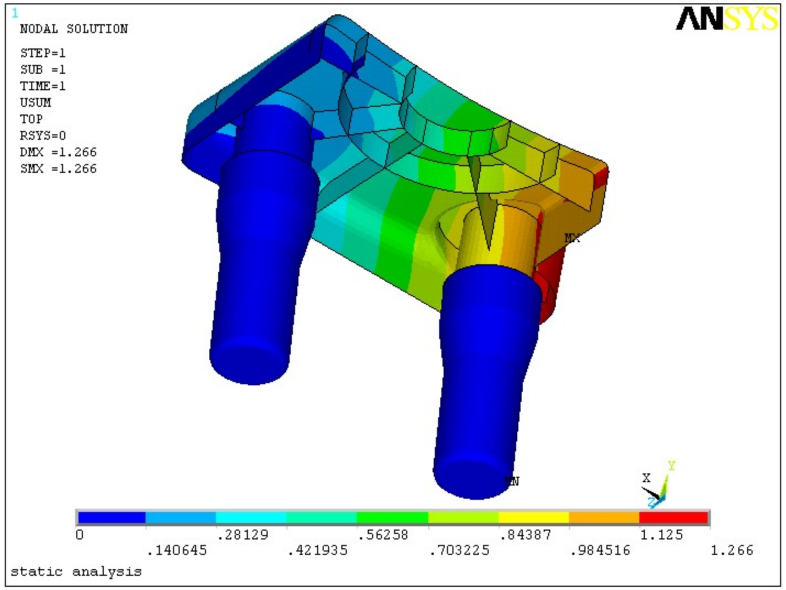


(a)幾何外形

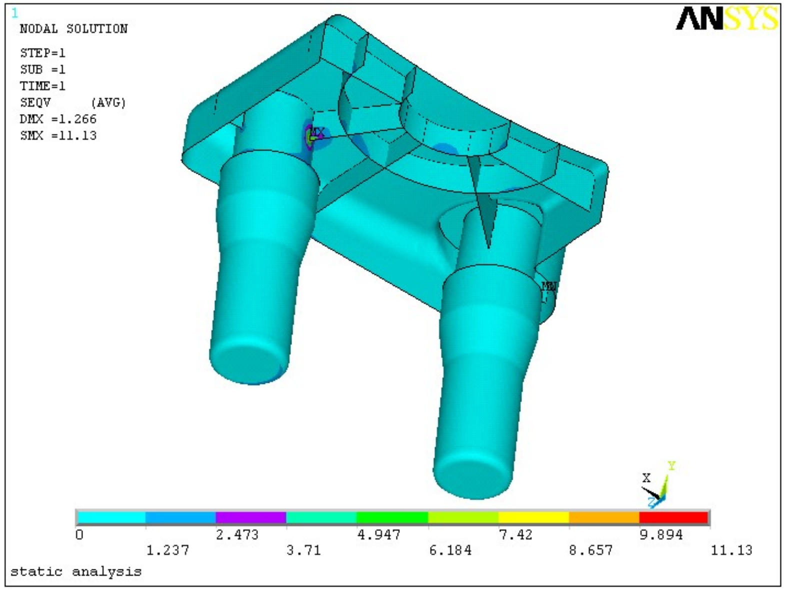


(b)1/2對稱有限元素模型

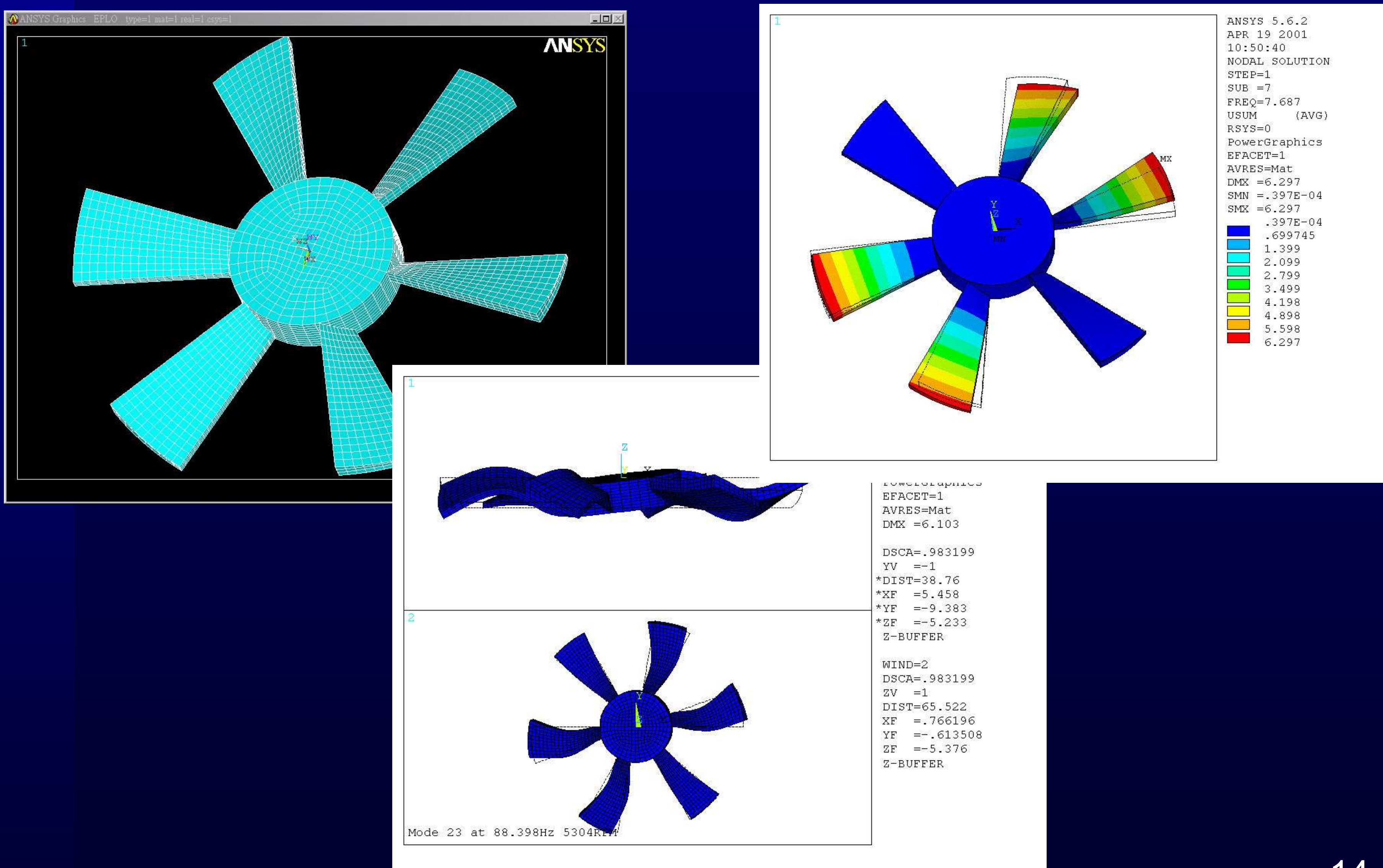
(c)變形



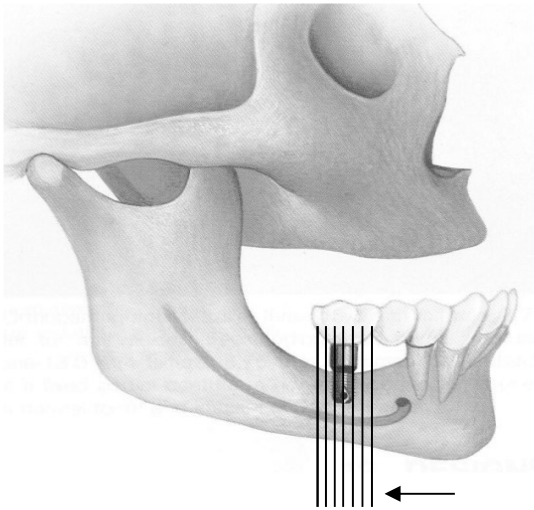
(d)應力場

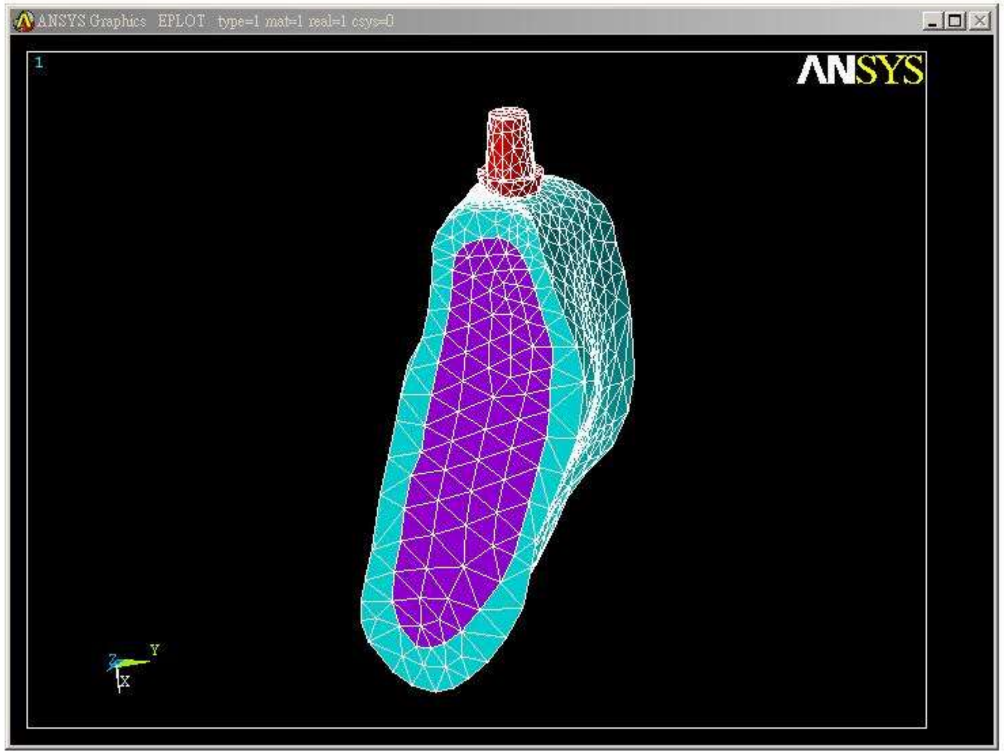


電腦散熱風扇葉片模態分析

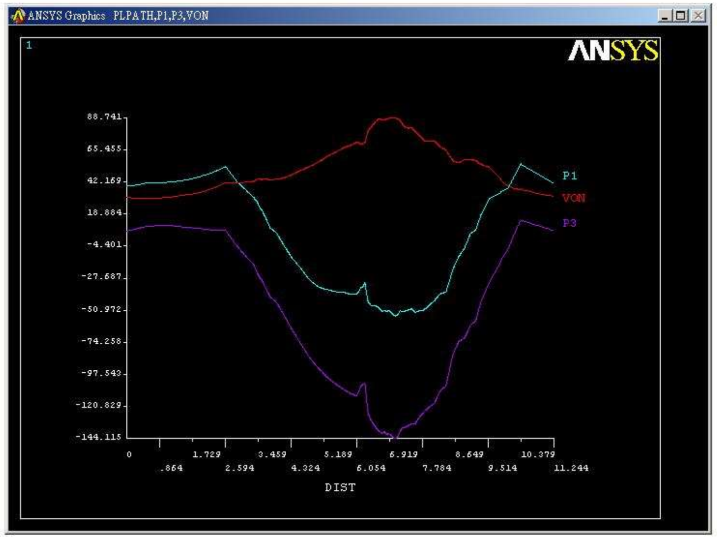


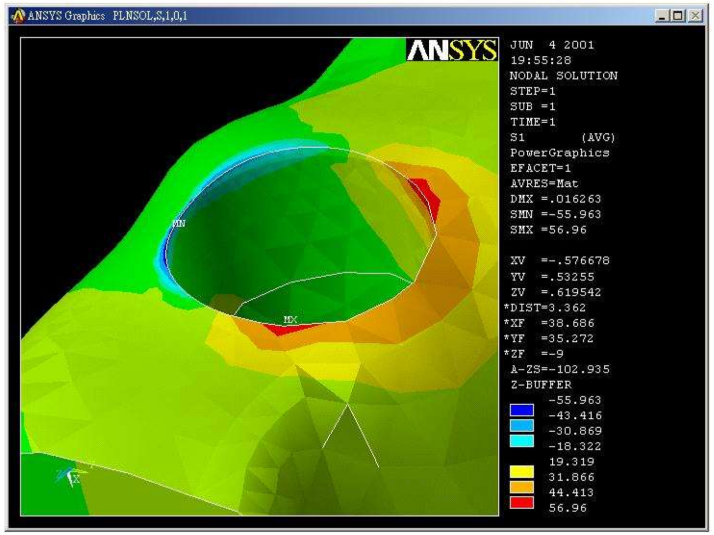
Biomechanical Analysis of Dental Implant

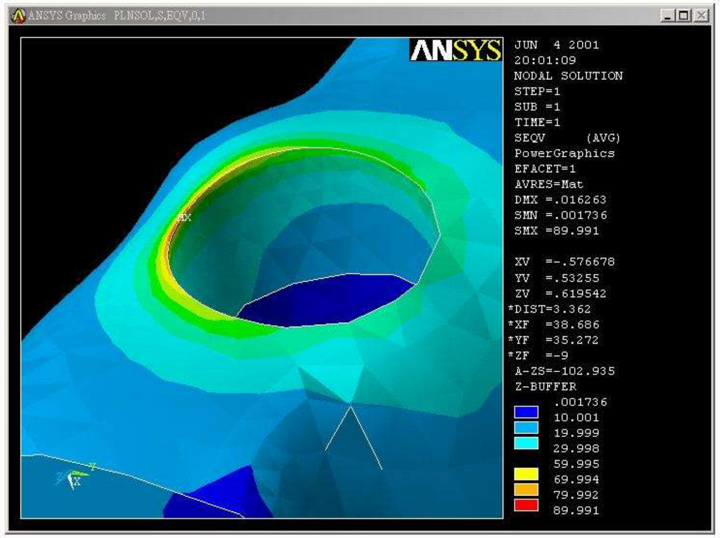


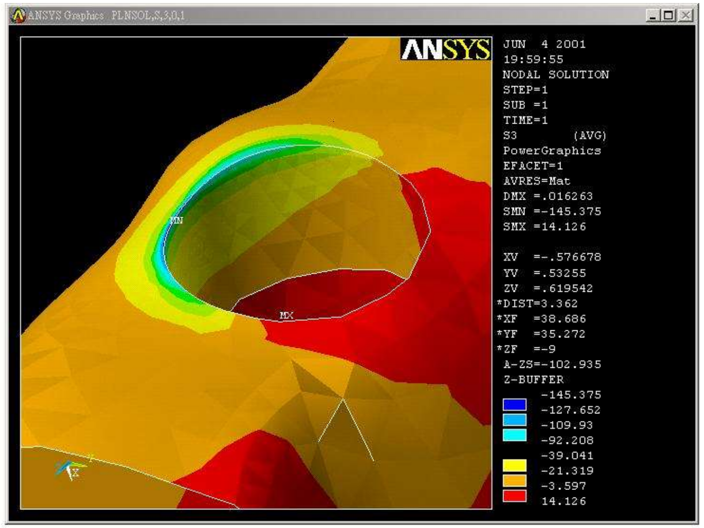


Biomechanical Analysis of Dental Implant







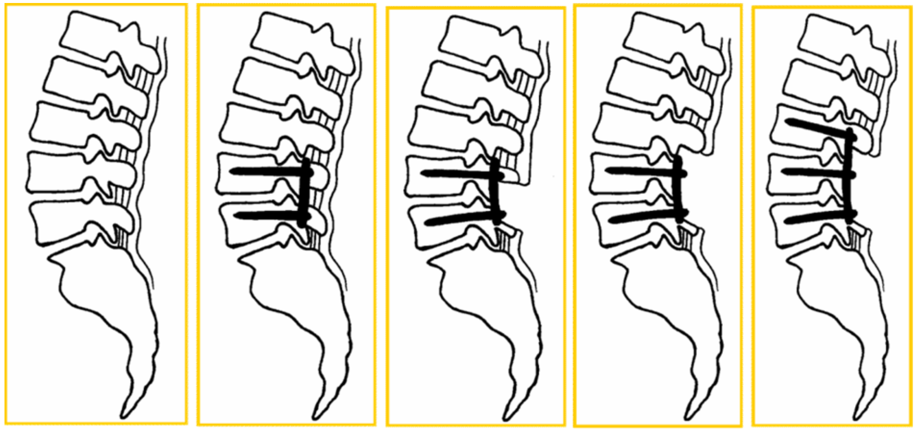


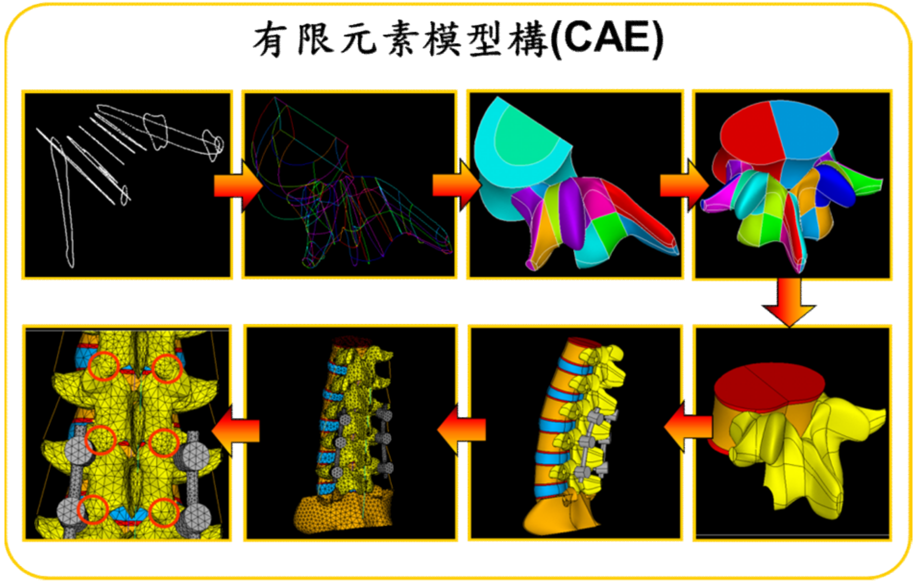
CAE Application in Spine Biomechanics

❖ 下顎骨雙邊矢向骨切開固定手術之生物力學分析

◆ 術後鄰近節不穩定因素

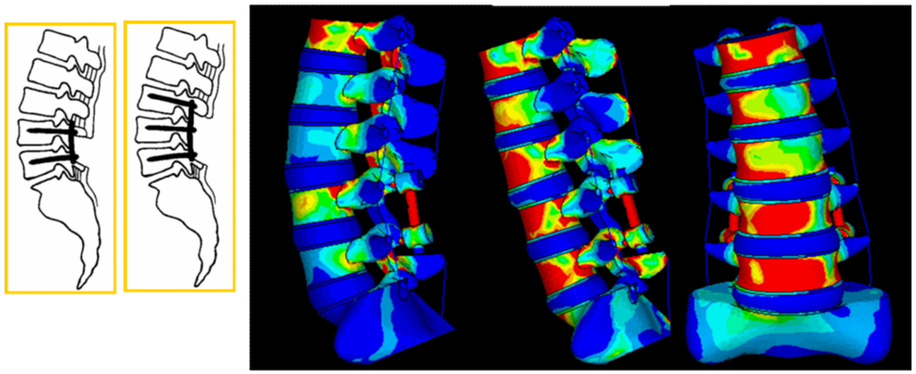
- 脊椎融合術範圍(D.E)
- 脊椎減壓術範圍(C.D)



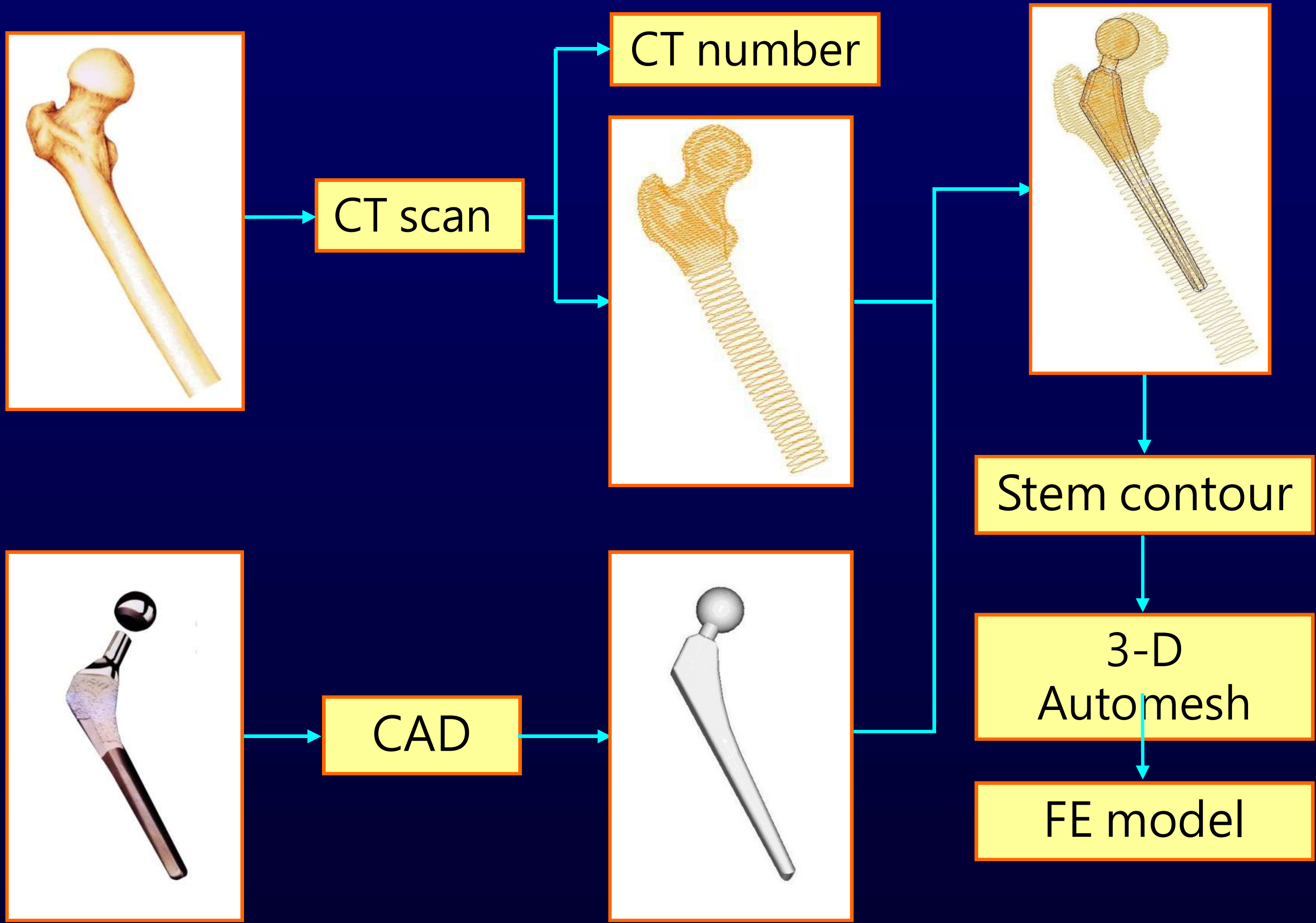


■ 在前彎與側彎負載下，全減壓術移除後側張力帶機制導致鄰近節不穩定，尤其以上鄰近節應力集中最嚴重，建議盡量實行半減壓術

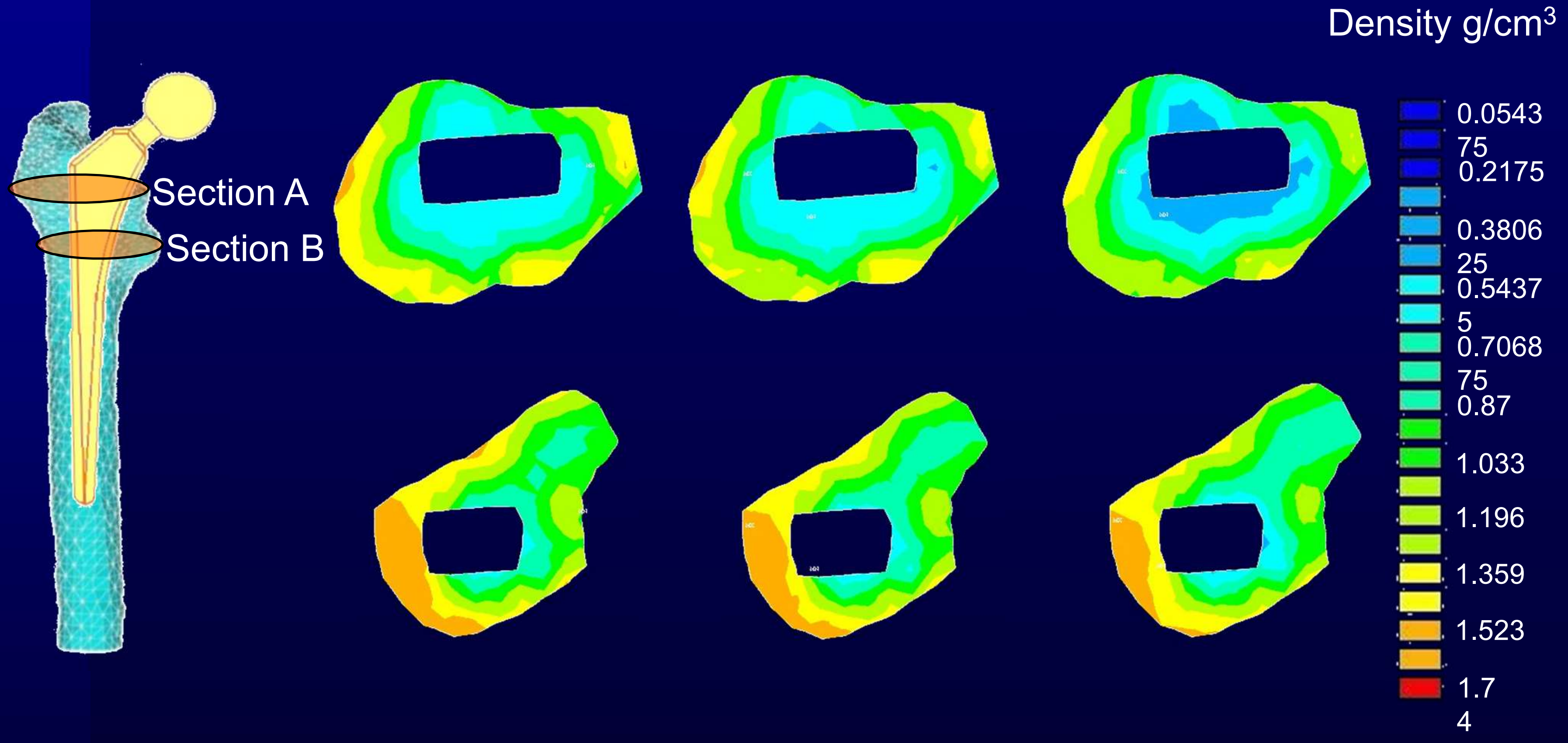
■ 若必需實行全減壓術時，則延長融合範圍的確能有效減低鄰近節不穩定發生之機會



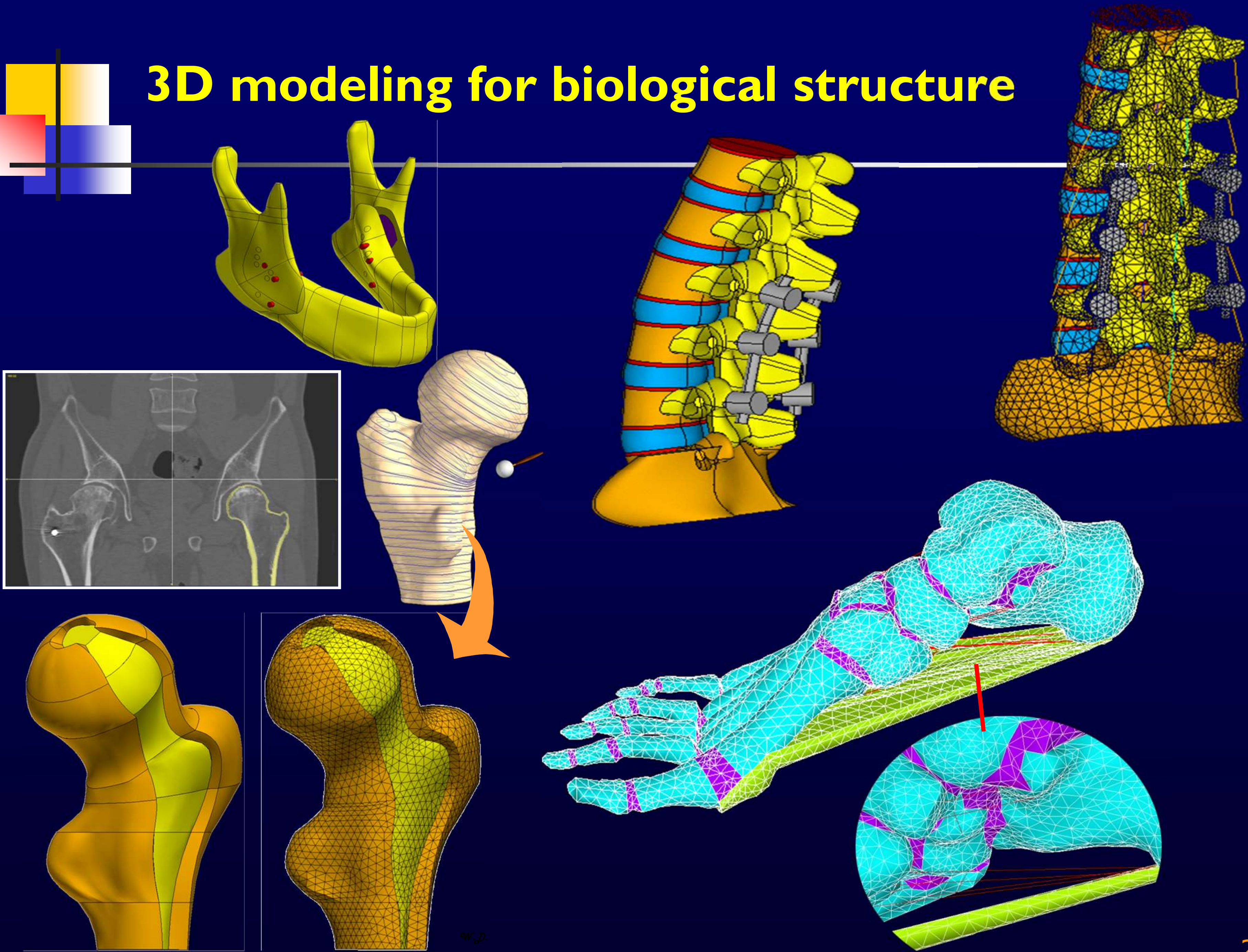
Computer Aided Analysis for Bone Remodeling



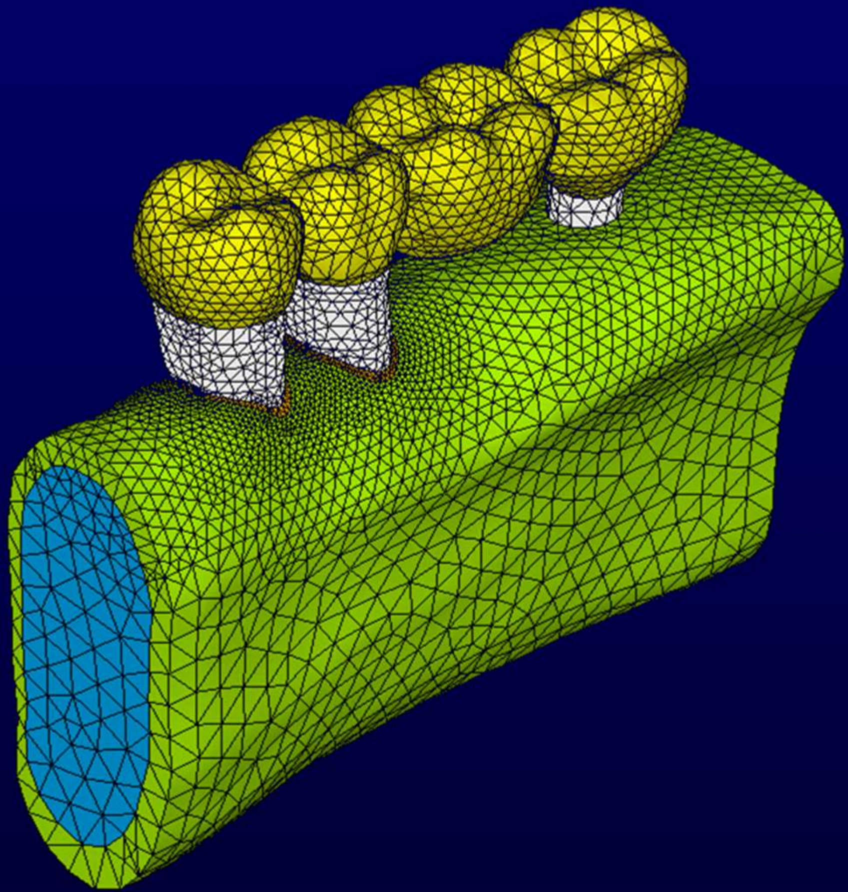
Computer Aided Analysis for Bone Remodeling



3D modeling for biological structure



3D modeling for biological structure



C.L. Lin, J.C. Wang*, S.T. Chen, “Evaluation of stress induced of implant type and number of splinted teeth in different periodontal supported tooth-implant supported FPDs: a nonlinear finite element analysis”, *Journal of Periodontology*, Vol. 81, pp.121-130, 2010.

FE Package - ANSYS

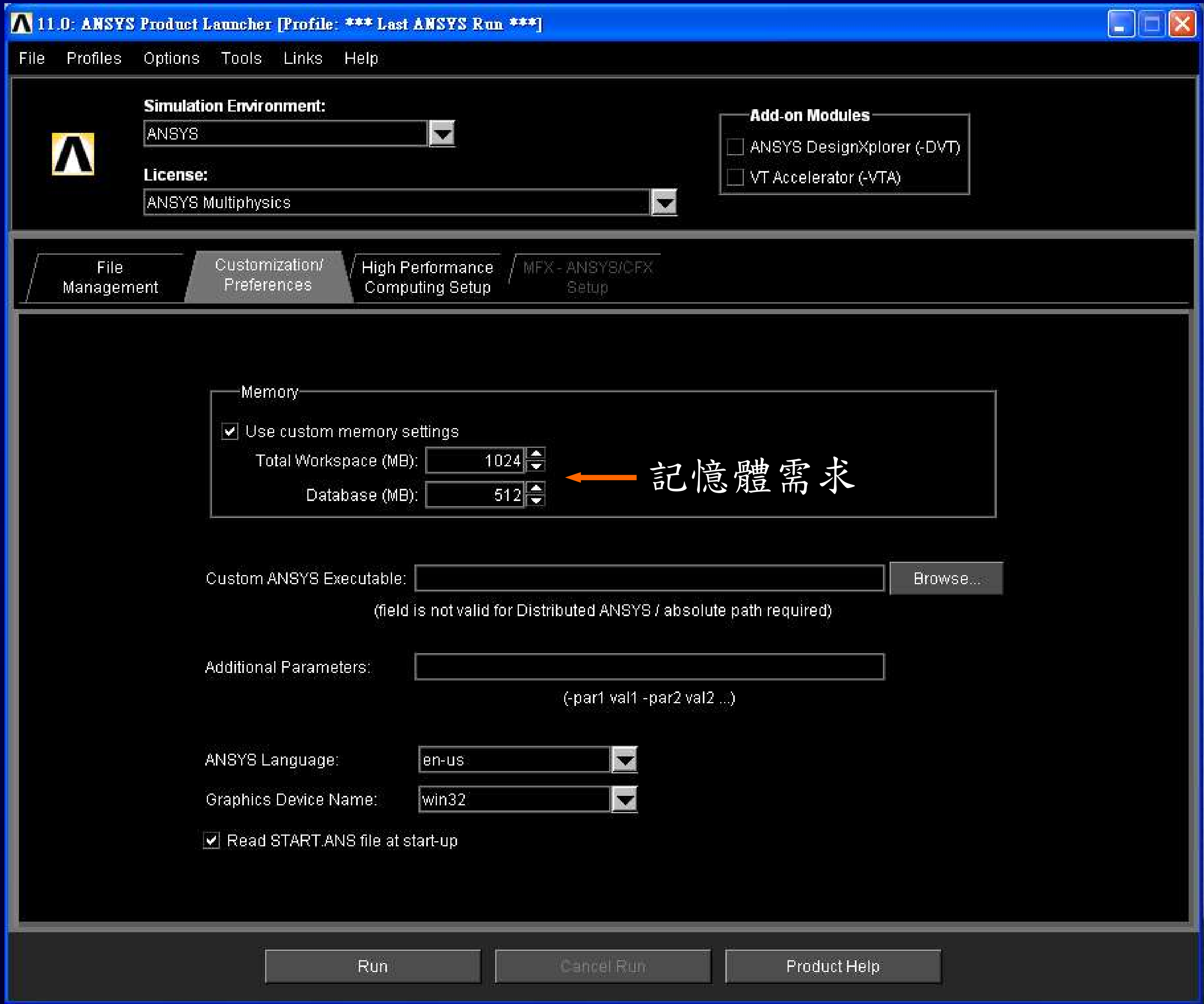
- 本課程所應用的有限元素軟體為
  - ANSYS-Classical
  - ANSYS WorkBench
- ANSYS為一套商業化之泛用型(general-purpose)有限元素分析軟體
- ANSYS是以有限元素法做為數值近似方法，分析功能包括固體力學、熱傳學、流體力學、電磁學以及跨領域的耦合場(coupled field)分析等。

FE Package - ANSYS

- ANSYS Classical 14 之主要模組分類：
  1. ANSYS Structural
  2. ANSYS Mechanical
  3. ANSYS Professional
  4. ANSYS FLOTRAN
  5. ANSYS Emag
  6. ANSYS Multiphysics
  7. ANSYS LS-DYNA
  8. ANSYS ED
  9. ANSYS Educational Products
  10. ANSYS Design Space
- 本課程所包括的內容是有限元素法與ANSYS在工程上的應用，範圍以固體力學(solid mechanics)基本模擬分析為主
- 高等電腦模擬分析
  - Non-linear structural analysis
  - Sub-modeling analysis
  - Element birth and dead
  - Hyperelasticity
  - Contact
  - APDL

ANSYS使用入門


11.0: ANSYS Product Launcher [Profile: *** Last ANSYS Run ***]
File Profiles Options Tools Links Help
Simulation Environment:
ANSYS
License:
ANSYS Multiphysics
Add-on Modules
ANSYS DesignXplorer (-DVT)
VT Accelerator (-VTA)
File Management
Customization/ Preferences
High Performance Computing Setup
MFX - ANSYS/CFX Setup
工作目錄
Working Directory:
C:\Documents and Settings\CGU
Browse...
Job Name:
file
Browse...
工作名稱
Run
Cancel Run
Product Help


11.0: ANSYS Product Launcher [Profile: *** Last ANSYS Run ***]
File Profiles Options Tools Links Help
Simulation Environment:
ANSYS
License:
ANSYS Multiphysics
Add-on Modules
ANSYS DesignXplorer (-DVT)
VT Accelerator (-VTA)
File Management
Customization/ Preferences
High Performance Computing Setup
MFX - ANSYS/CFX Setup
Memory
Use custom memory settings
Total Workspace (MB): 1024
Database (MB): 512
記憶體需求
Custom ANSYS Executable:
Browse...
(field is not valid for Distributed ANSYS / absolute path required)
Additional Parameters:
(-par1 val1 -par2 val2 ...)
ANSYS Language: en-us
Graphics Device Name: win32
Read START.ANS file at start-up
Run
Cancel Run
Product Help

1應用指令功能表列

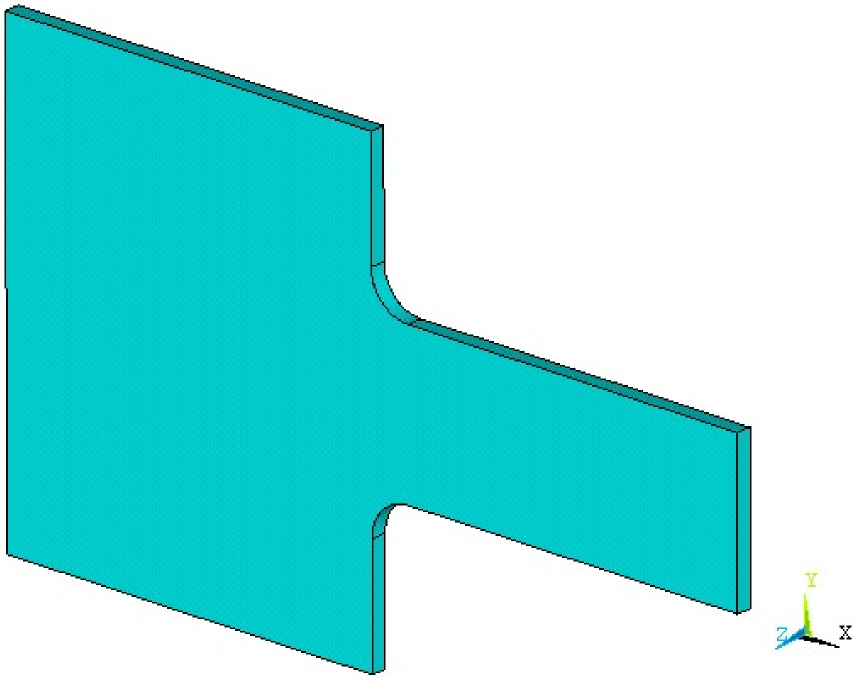

ANSYS Multiphysics Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
4指令輸入處
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
3工具列
ANSYS Main Menu
Preferences
Preprocessor
Solution
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
Design Opt
Prob Design
Radiation Opt
Run-Time Stats
Session Editor
Finish
2主功能表
1
NODES
ANSYS
SEP 16 2006
15:19:40
Y
X
5圖形視窗
ANSYS 8.0 Output Window
ACTIVATING THE GRAPHICAL USER INTERFACE (GUI). PLEASE WAIT
Creating the GUI panes
Creating the GUI menubar
Creating the GUI status
Creating the GUI Graphics Context
Creating the Application Context
Creating the GUI toolbar
Creating the GUI toolbar - "ANSYS Standard Toolbar"
Creating the GUI toolbar - "ANSYS Main Menu"
Creating the GUI toolbar - "ANSYS Toolbar"
Creating the GUI toolbar - "ANSYS Graphical View"
CUTTING PLANE SET TO THE WORKING PLANE
Loading ANSYS Help System
Welcome to ANSYS
PRODUCE NODAL PLOT IN DSYS= 0
TURN OFF WORKING PLANE DISPLAY
PRODUCE NODAL PLOT IN DSYS= 0
PRODUCE NODAL PLOT IN DSYS= 0
6輸出視窗
real=1
csys=0

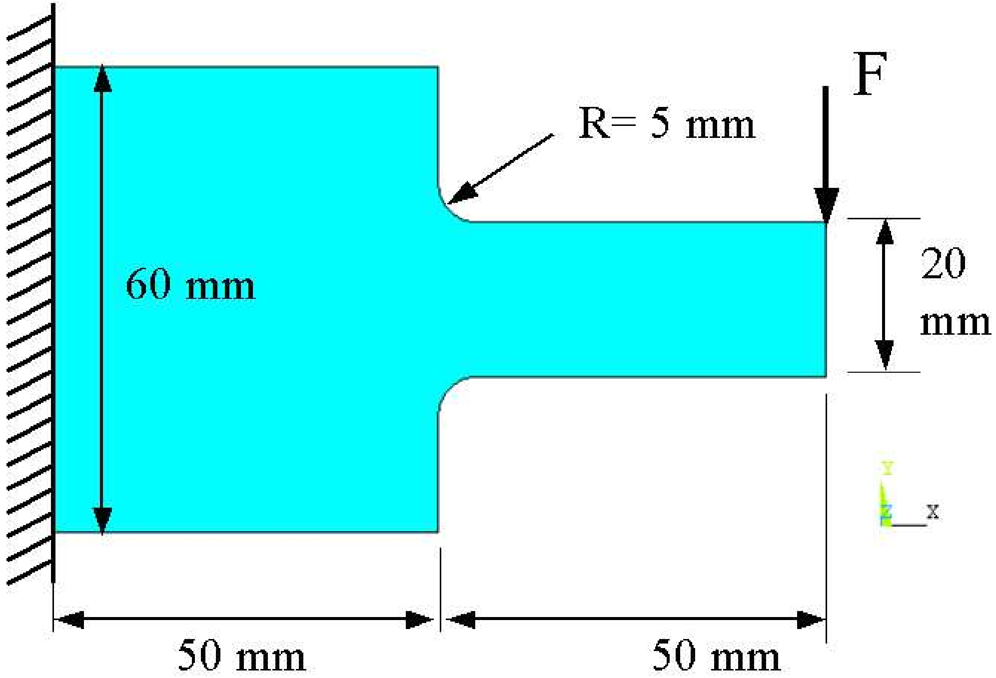
ANSYS使用入門

- 範例步驟，大致可分成以下階段：
  1. 分析類型選定
  2. 元素型式設定
  3. 材料性質設定
  4. 幾何外形建模
  5. 有限元素網格之建立
  6. 加入負荷與邊界條件
  7. 最後檢查
  8. 求解
  9. 後處理：觀察分析結果、輸出數據或圖形動畫
- 所有的有限元素分析軟體都可大略切割成三部分：前處理器(pre-processor)、求解器(solver)與後處理器(post-processor)

ANSYS 使用入門 - Ex1

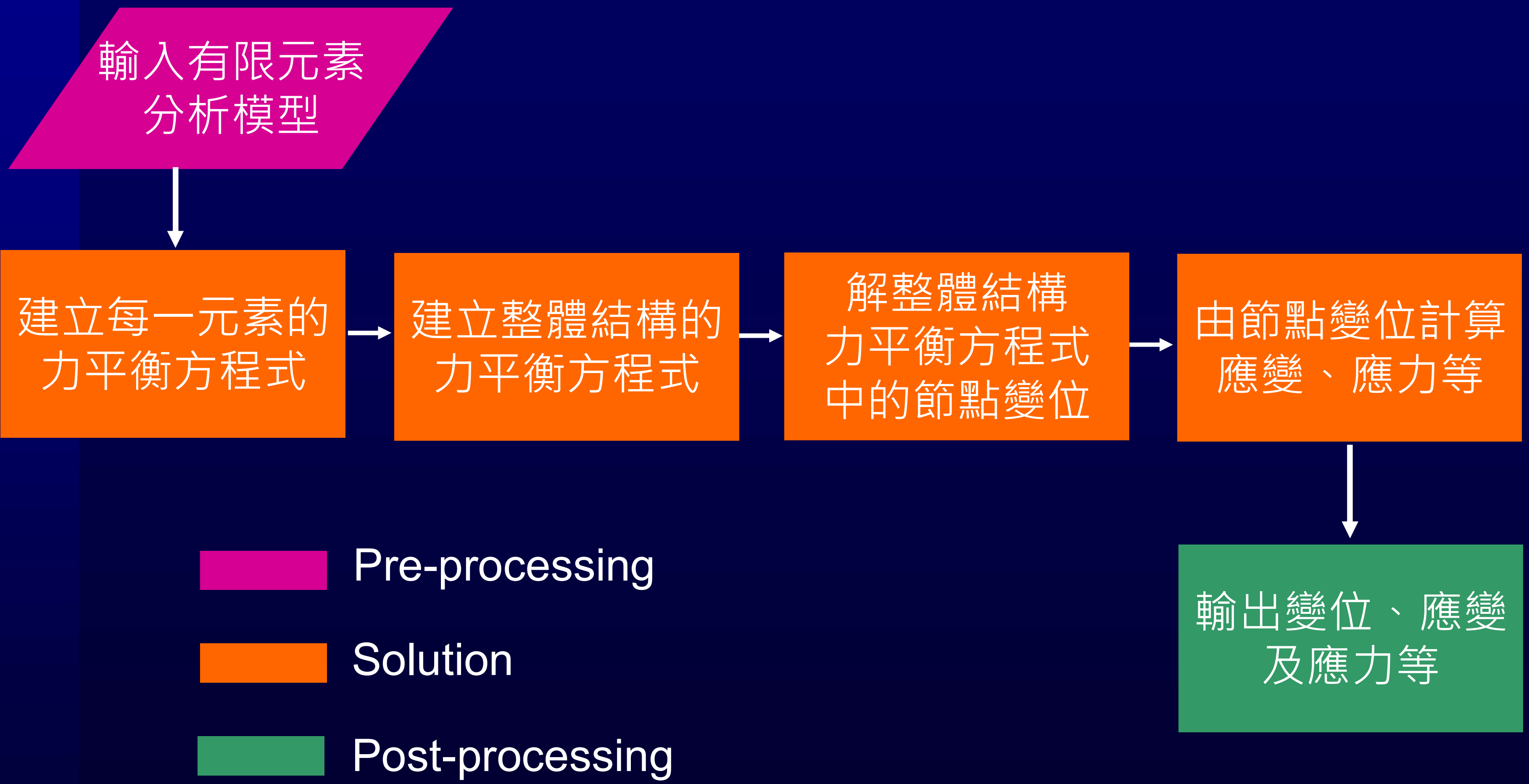
- 厚度2mm,左端固定,右端施力F=10N,求應力分佈,材料為鋼(E=210000MPa),Possion’ s ratio=0.3





Fundamental Concepts in FEM

- 有限元素分析程序摘要



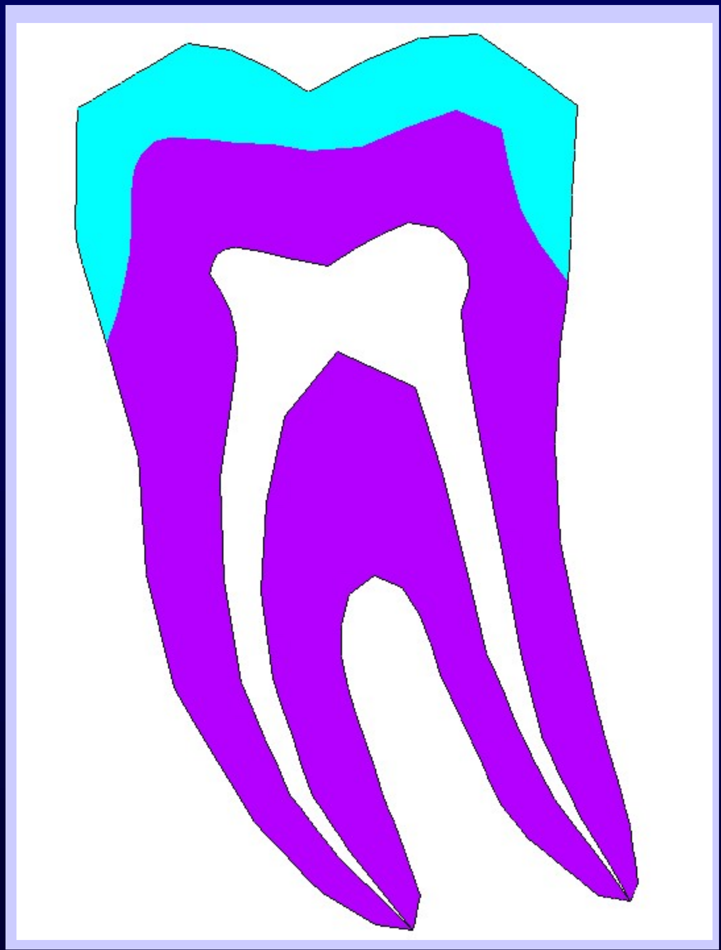
Fundamental Concepts in FEM

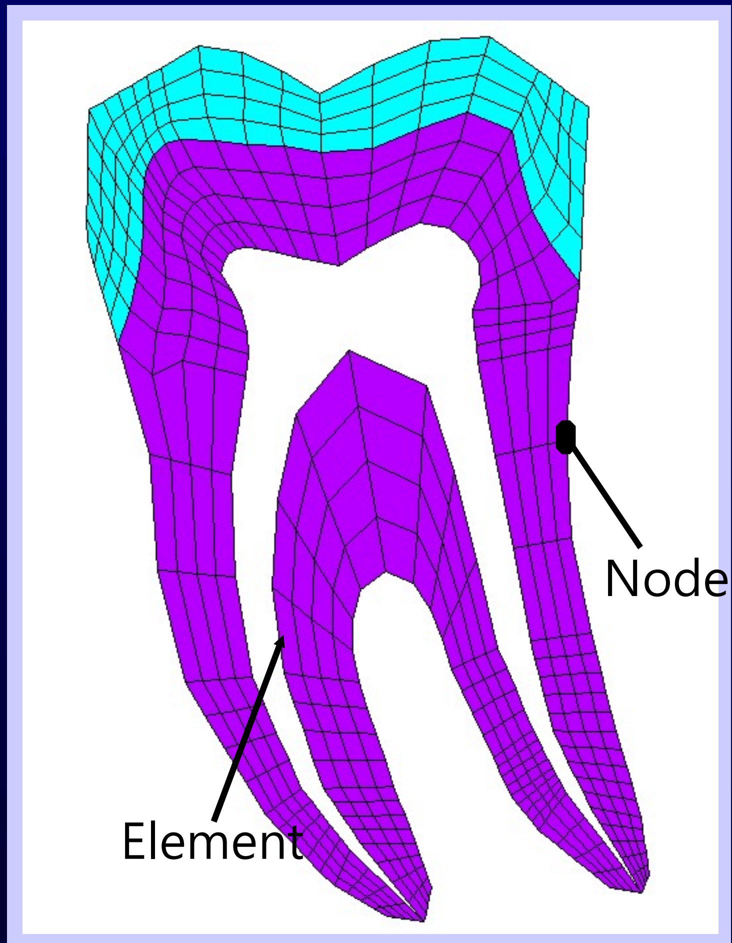
- Pre-processing
  - Geometry
  - Mesh (Element type)
  - Materials
  - Boundary and loading conditions
- Solution
- Post-processing
  - Present the results by graphs…

Fundamental Concepts in FEM

- 實際的物理問題很難利用單一的微分方程式描述，更無法順利求其解析(analytical solution)解
- 有限元素法的精神是將複雜的幾何外型的結構物體切割成許多簡單的幾何形狀稱之為元素(element)
- 元素與元素間以“節點”(node)相連
- 由於元素是簡單的幾何形狀，故可順利寫出元素的力平衡方程式並求得節點上之變位、應變及應力等
- 藉由內插法求得元素內任意點的變位、應變及應力等

Fundamental Concepts in FEM





Fundamental Concepts in FEM

- 元素力平衡方程式
  - [k]{d}={f}
  - {d}是該元素節點上所有自由度組合而成的向量
  - {f}稱為元素的力向量
  - [k]稱為元素的剛度矩陣
- 整體結構力平衡方程式
  - [K]{D}={F}
  - {D}是結構上所有自由度組合而成的向量
  - {F}稱為結構的力向量
  - [K]稱為結構的剛度矩陣

Fundamental Concepts in FEM

- 求出節點的變位後 [k]{d}={f}，透過下式可求得應變及應力

$$\varepsilon_x = \frac{\partial u_x}{\partial x} \qquad \varepsilon_x = \frac{\sigma_x}{E} - \nu\frac{\sigma_y}{E} - \nu\frac{\sigma_z}{E}$$

$$\varepsilon_y = \frac{\partial u_y}{\partial x} \qquad \varepsilon_y = \frac{\sigma_y}{E} - \nu\frac{\sigma_z}{E} - \nu\frac{\sigma_x}{E}$$

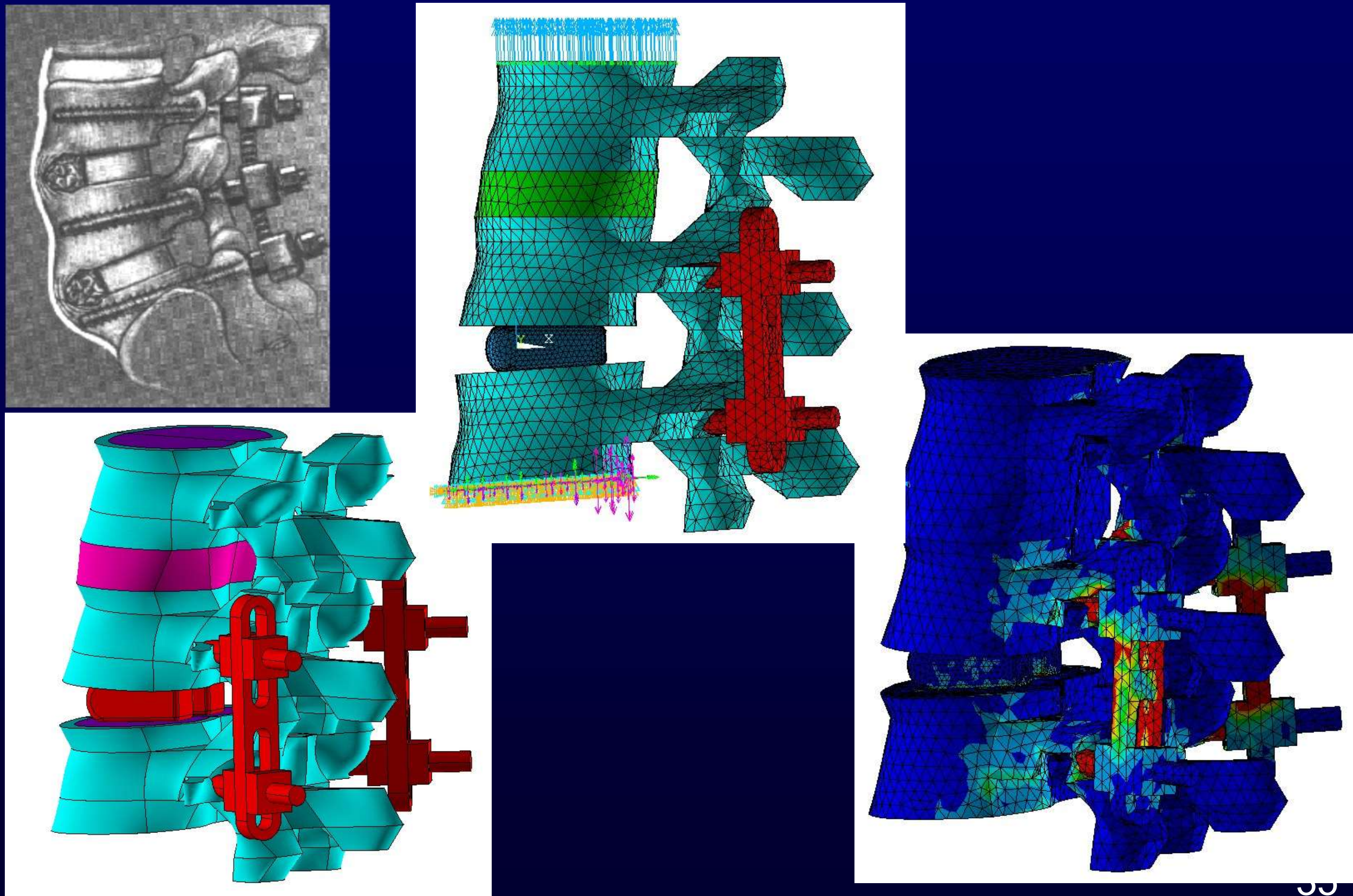
$$\varepsilon_z = \frac{\partial u_z}{\partial x} \qquad \varepsilon_z = \frac{\sigma_z}{E} - \nu\frac{\sigma_x}{E} - \nu\frac{\sigma_y}{E}$$

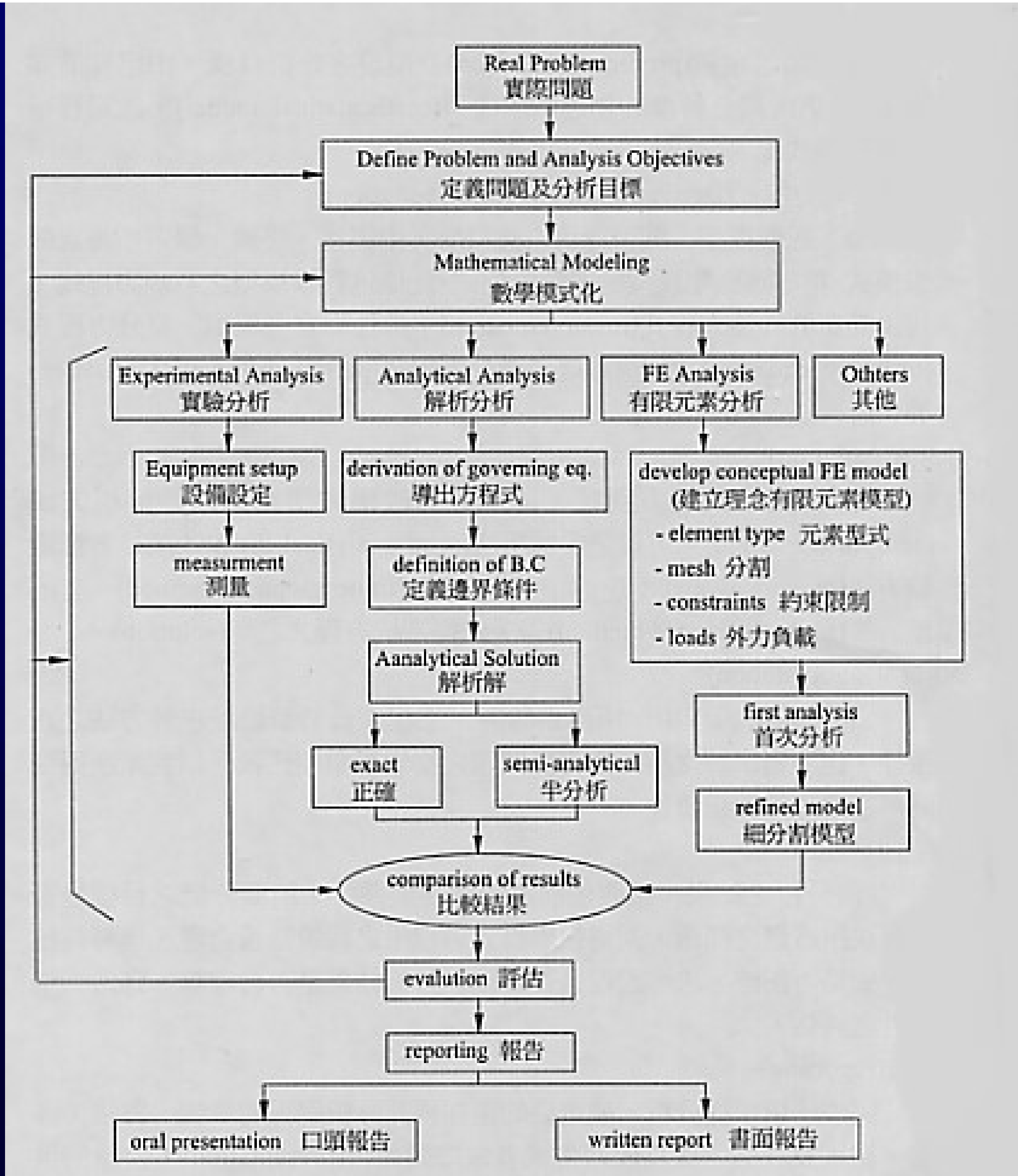
$$\gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} \qquad \gamma_{xy} = \frac{\tau_{xy}}{G}$$

$$\gamma_{yz} = \frac{\partial u_y}{\partial z} + \frac{\partial u_z}{\partial y} \qquad \gamma_{yz} = \frac{\tau_{yz}}{G}$$

$$\gamma_{zx} = \frac{\partial u_z}{\partial x} + \frac{\partial u_x}{\partial z} \qquad \gamma_{zx} = \frac{\tau_{zx}}{G}$$

Fundamental Concepts in FEM




Real Problem
實際問題
Define Problem and Analysis Objectives
定義問題及分析目標
Mathematical Modeling
數學模式化
Experimental Analysis
實驗分析
Analytical Analysis
解析分析
FE Analysis
有限元素分析
Othters
其他
Equipment setup
設備設定
measurment
測量
derivation of governing eq.
導出方程式
definition of B.C
定義邊界條件
Aanalytical Solution
解析解
exact
正確
semi-analytical
半分析
develop conceptual FE model
(建立理念有限元素模型)
- element type 元素型式
- mesh 分割
- constraints 約束限制
- loads 外力負載
first analysis
首次分析
refined model
細分割模型
comparison of results
比較結果
evalution 評估
reporting 報告
oral presentation 口頭報告
written report 書面報告

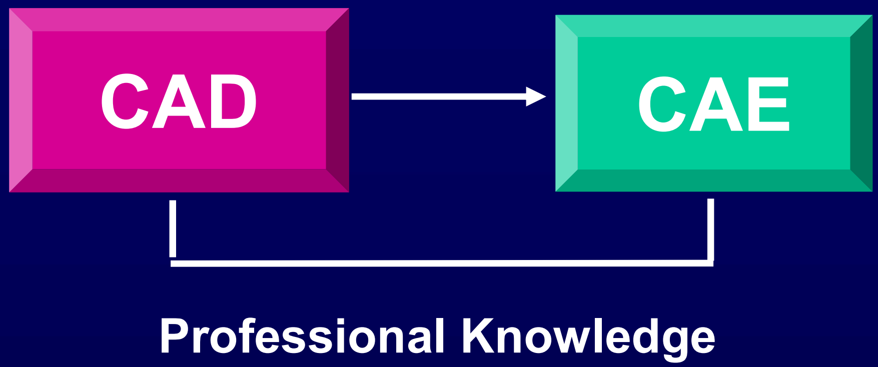
Fundamental Concepts in FEM

- FEM
  - A numerical method for solving P.D.E.
- Advantage
  - Can handle
  - Arbitrary geometry & material complexity
  - Provide more detailed *mechanical responses*
  - Becoming a powerful analytical tool
- Disadvantage
  - Require large amount of input data
  - Computation time

Fundamental Concepts in FEM

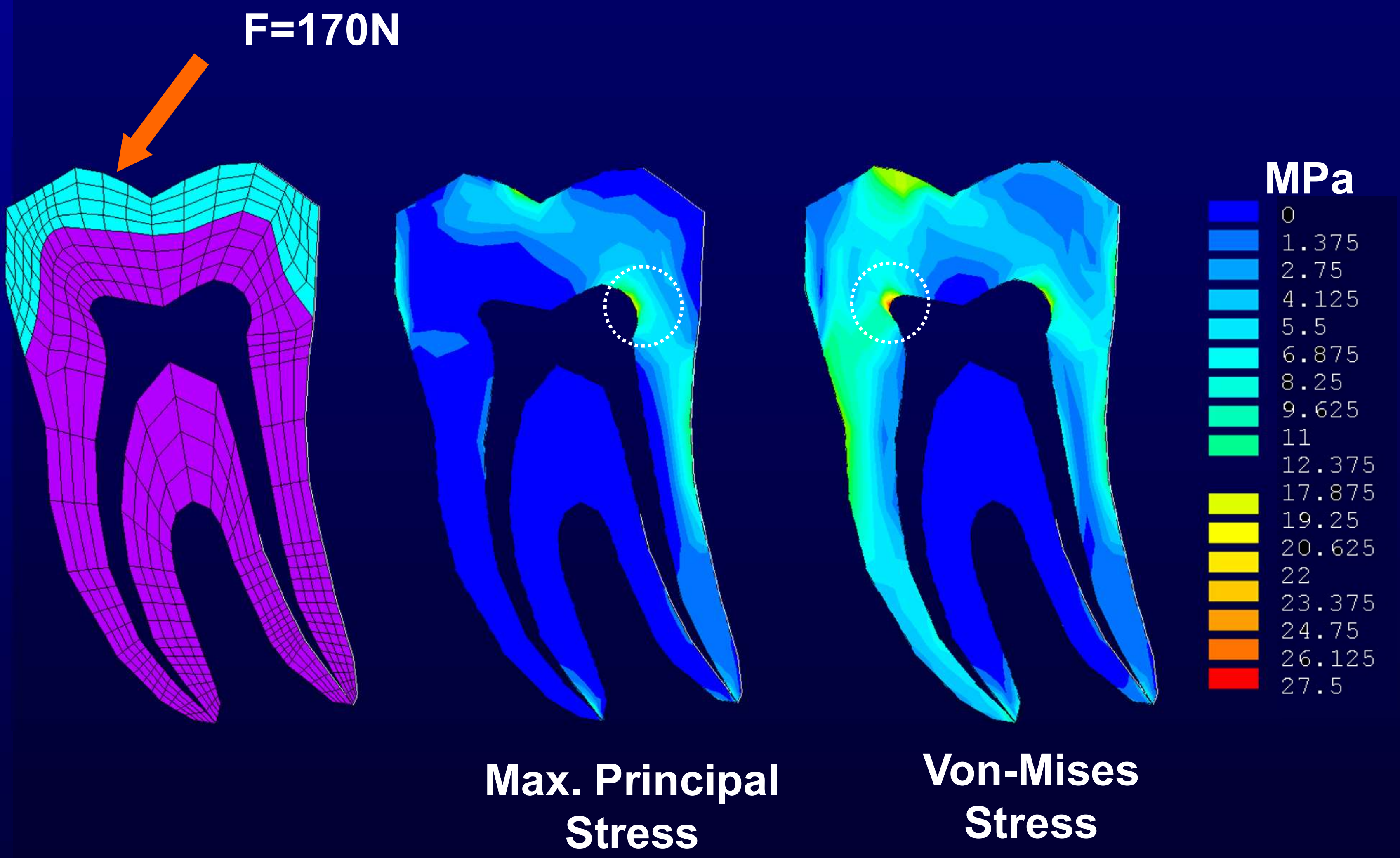
- The simulated analytical results could be plausible and incredulous by
  - inaccurately geometry approximation
  - material distribution
  - uncertainty loading and boundary condition
- Pre-processing technique of FEM
  - *meshing procedure* for bio-structures is still a big obstacle especially in 3-D applications

General Concept of CAE



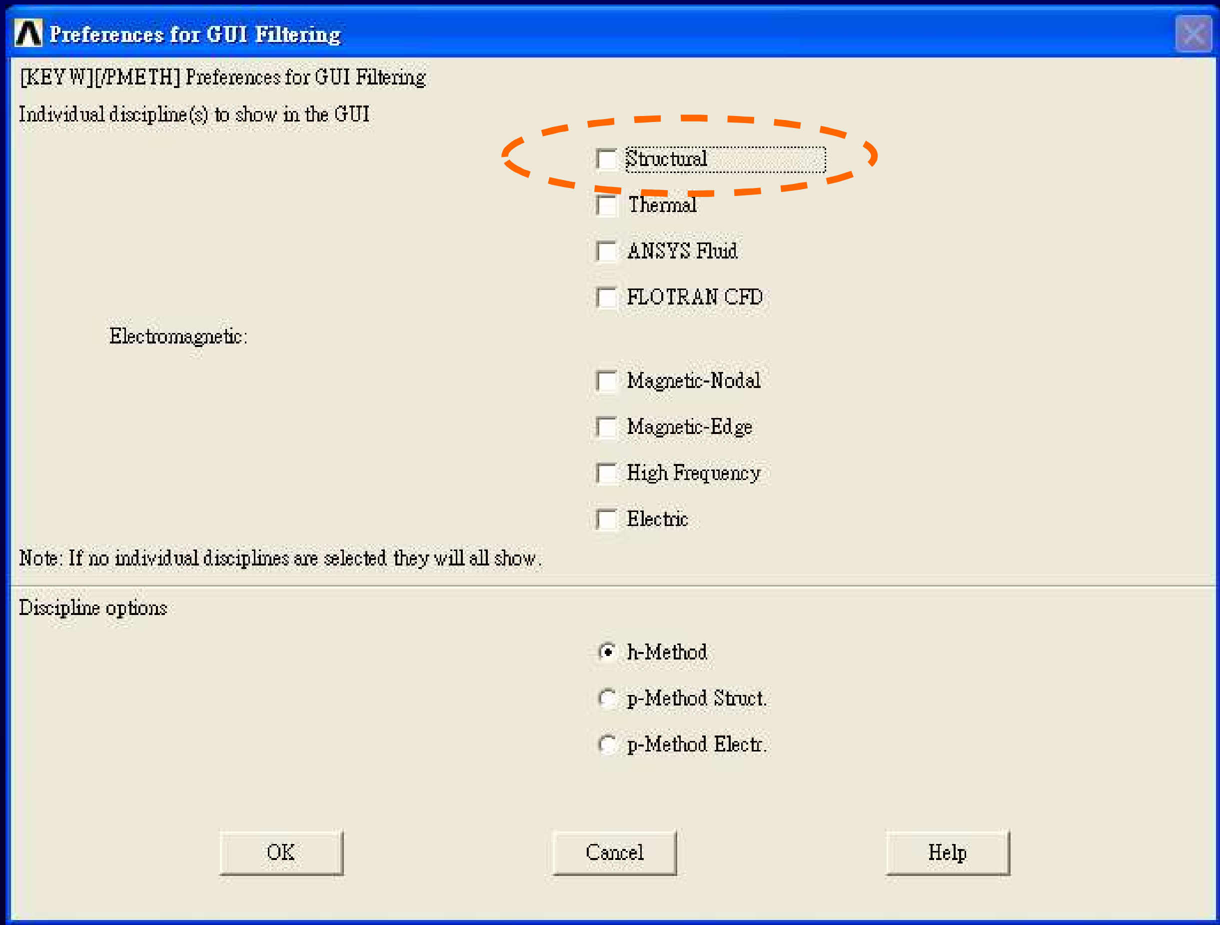
- Professional Knowledge (Physical Problem)
  - Structural Mechanics
  - Thermal (Heat Transform)
  - Fluid Flow
  - Electro-magnetic, etc.

General Concept of CAE

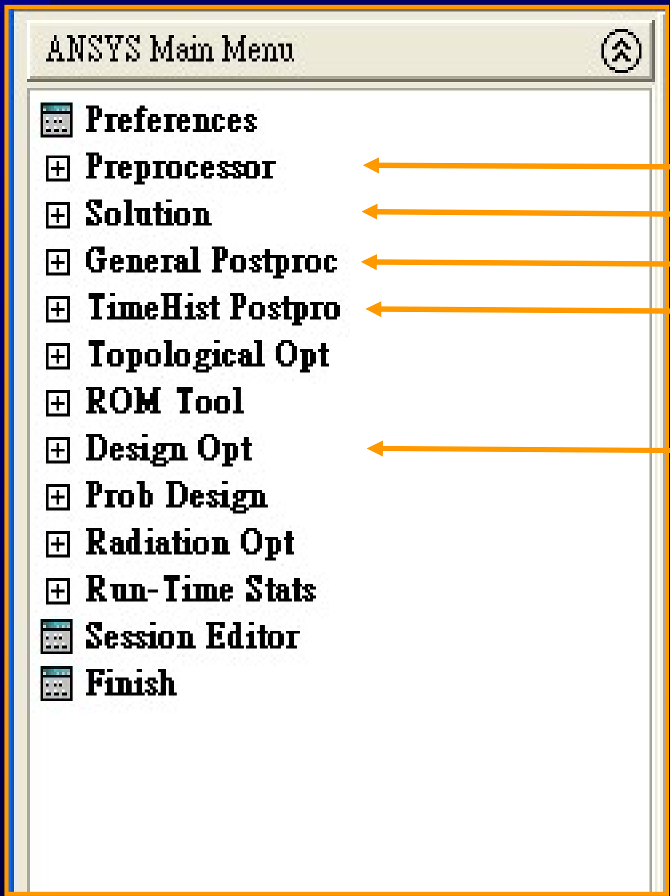


Introduction of ANSYS

- 選定分析類型

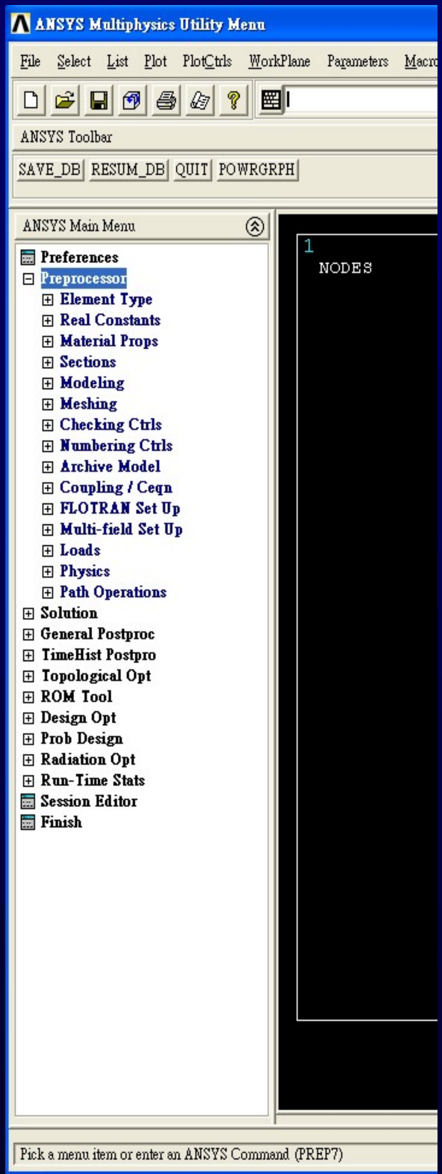


Introduction of ANSYS



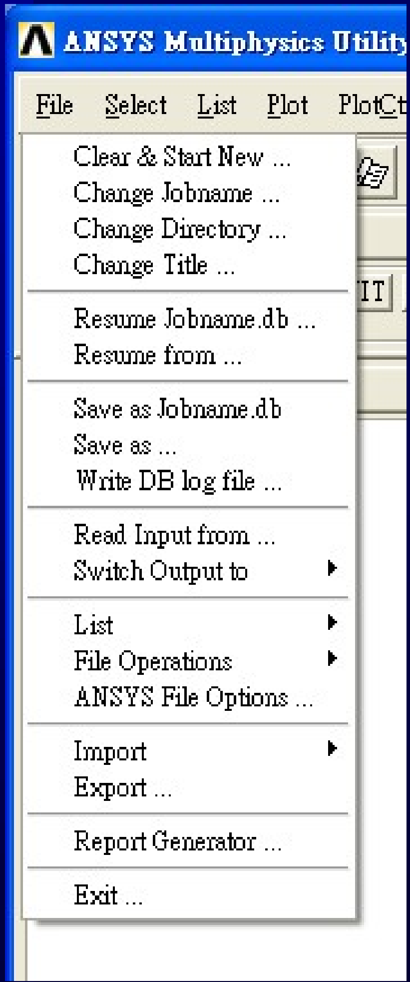
前處理
解題
後處理(Post1)
後處理(Post26)
最佳化

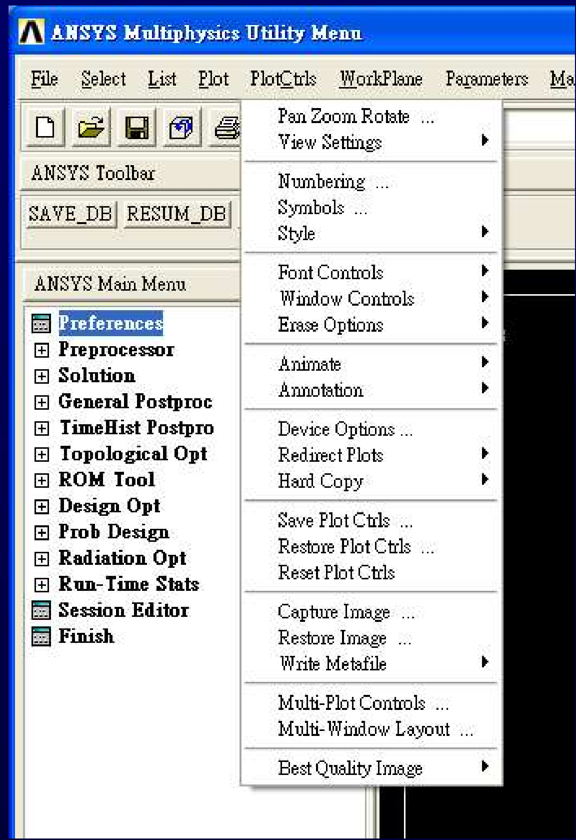
Pre-processing, solution and post-processing



Introduction of ANSYS

- 工具列 - 利用靜態分析各例題說明





BME

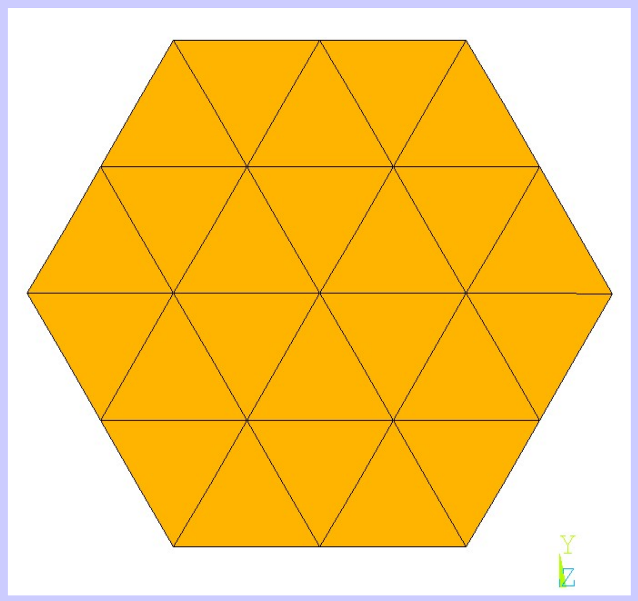
NYMU

Introduction of ANSYS

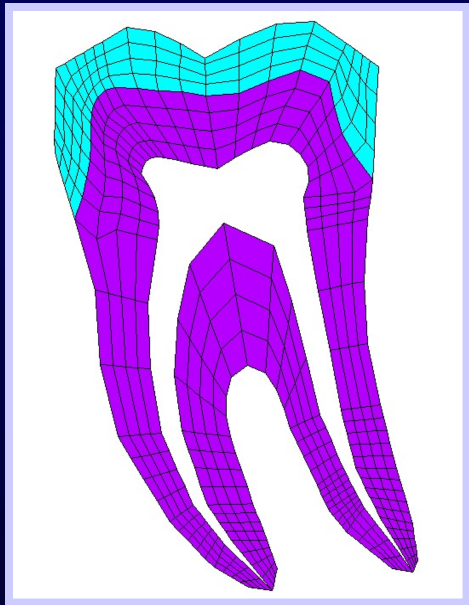
- Pre-processing
  - Element type
    - 2D, 3D…
  - Real Constants
  - Material Props
  - Modeling
    - Model generation techniques
  - Meshing
    - Direct generation
    - Solid modeling
  - Other techniques
    - Merge….

Introduction of ANSYS

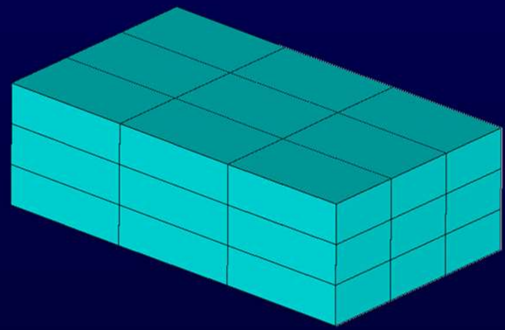
- **Element type definition**
  - **The ANSYS element library contains more than 200 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM188, PLANE183, SOLID186, etc.**



2D Triangular Structural Solid



2D Structural Solid



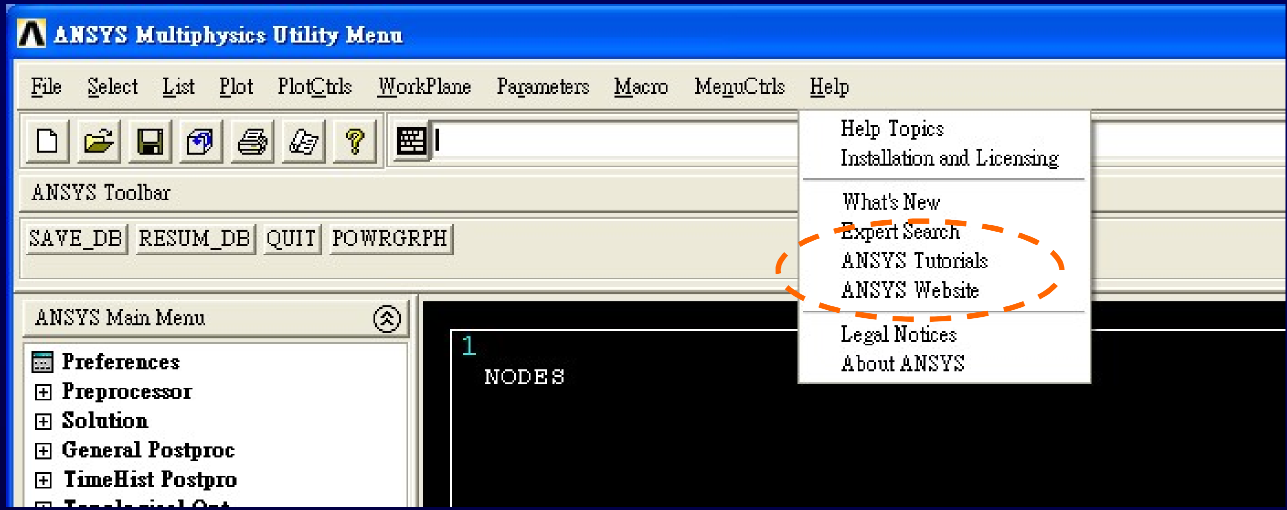
3D Structural Solid

Introduction of ANSYS

- The element type determines, among other things:
  - The degree-of-freedom set (which in turn implies the discipline--structural, thermal, magnetic, electric, quadrilateral, brick, etc.)
  - Whether the element lies in two-dimensional or three-dimensional space.
  - Main Menu>Preprocessor>Element Type>Add/Edit/Delect
    - 定義元素類型號碼
    - 元素特性 (平面應變,平面應力,軸對稱等)
    - 元素特性座標系統等

Introduction of ANSYS

- Utility Menu



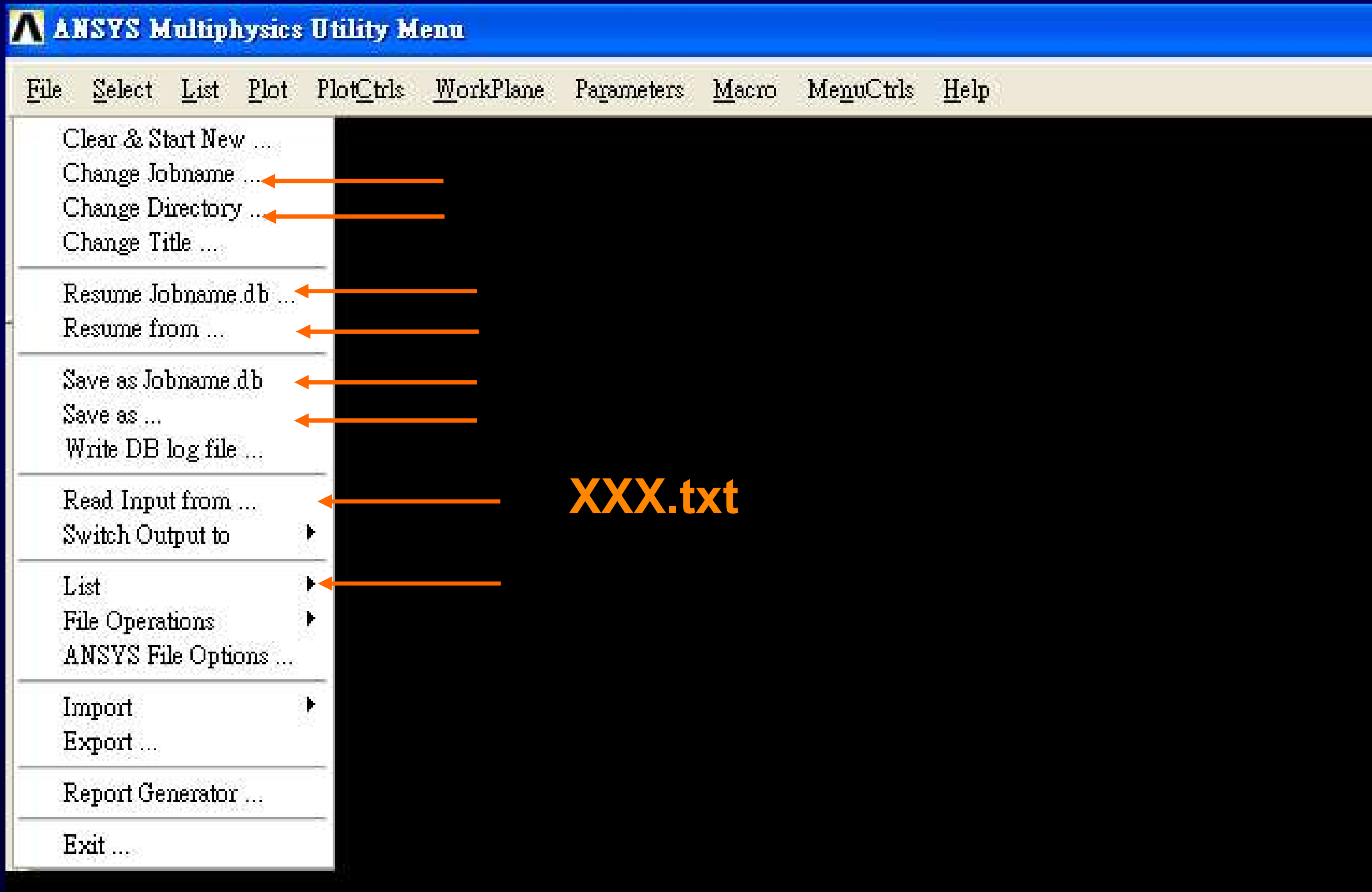
- Help
  - ANSYS Tutorials
  - ANSYS Web-site

Introduction of ANSYS

- ANSYS 重要檔案
  - *JOBNAME*.DB (Binary)
    - Node, element, model…..
    - Results…
  - *JOBNAME*.RST or .THR or .RMG (Binary)
    - Results
  - *JOBNAME*.ERR (ASCII)
    - Error messages
  - *JOBNAME*.LOG(ASCII)
    - Commands

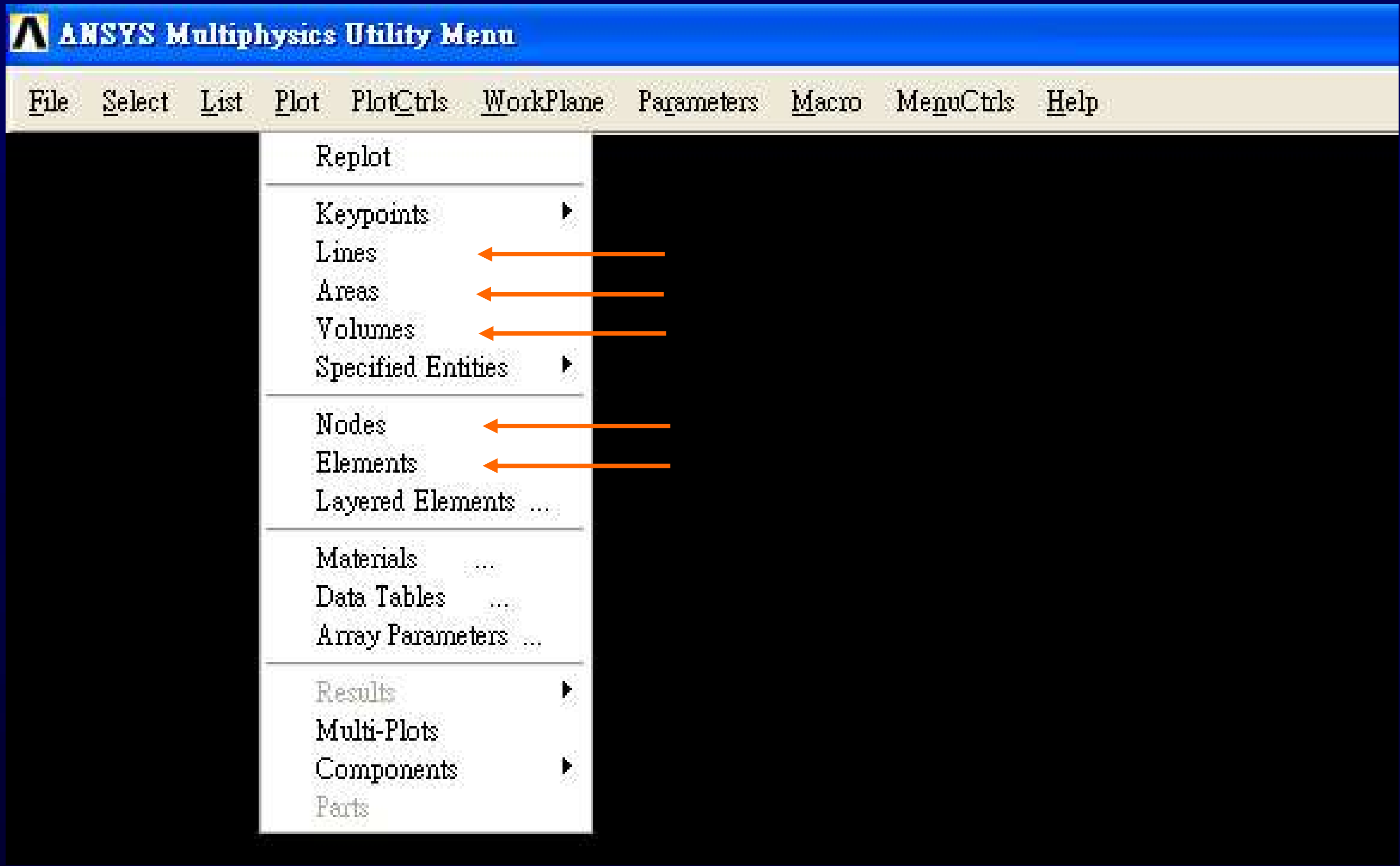
Introduction of ANSYS

- **File (demo)**



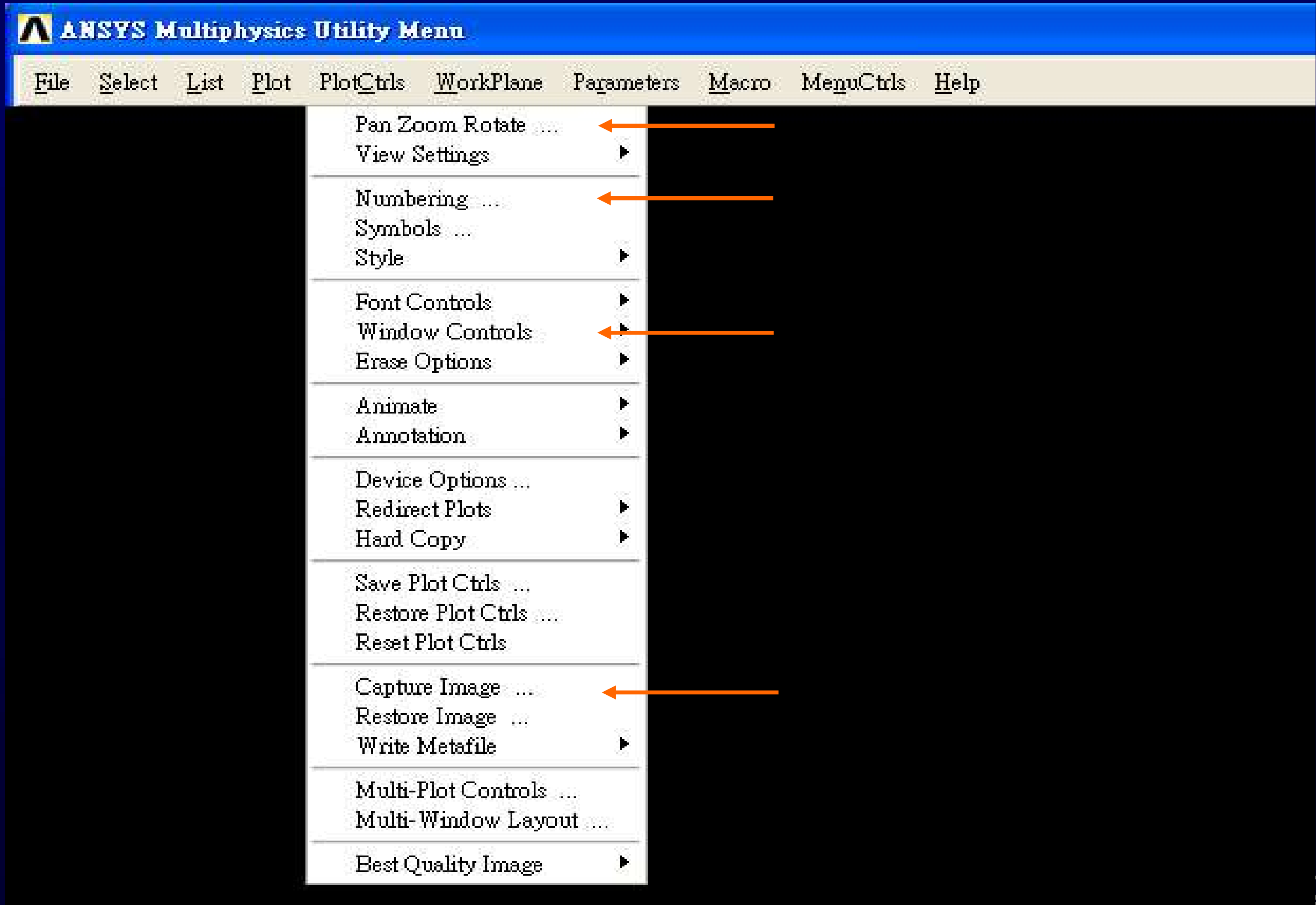
Introduction of ANSYS

- Plot, List (demo)



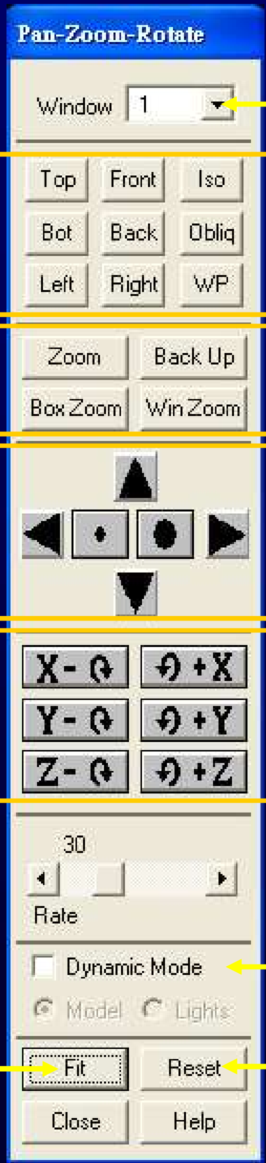
Introduction of ANSYS

- **PlotCtrls (demo)**



Introduction of ANSYS

- 圖形控制



視窗號碼

圖形不同方向示意圖

圖形區域放大

圖形移動

檢視圖形

對 X, Y, Z軸旋轉

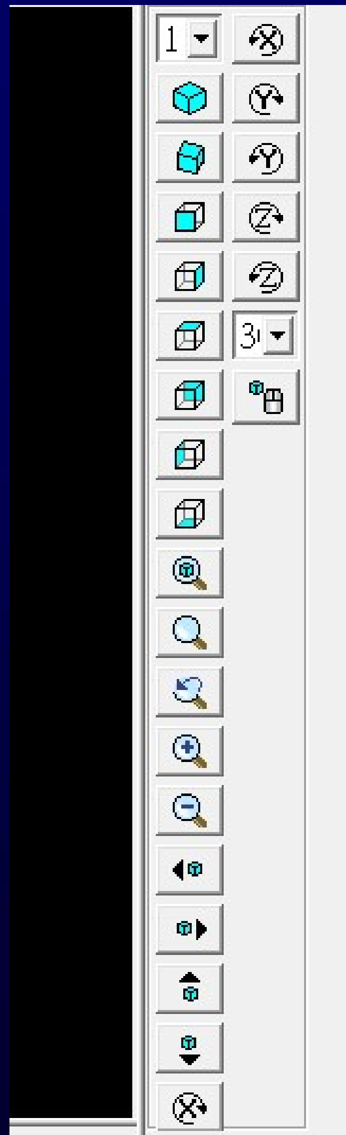
開啟動態模式

滑鼠右:旋轉

滑鼠左:平移

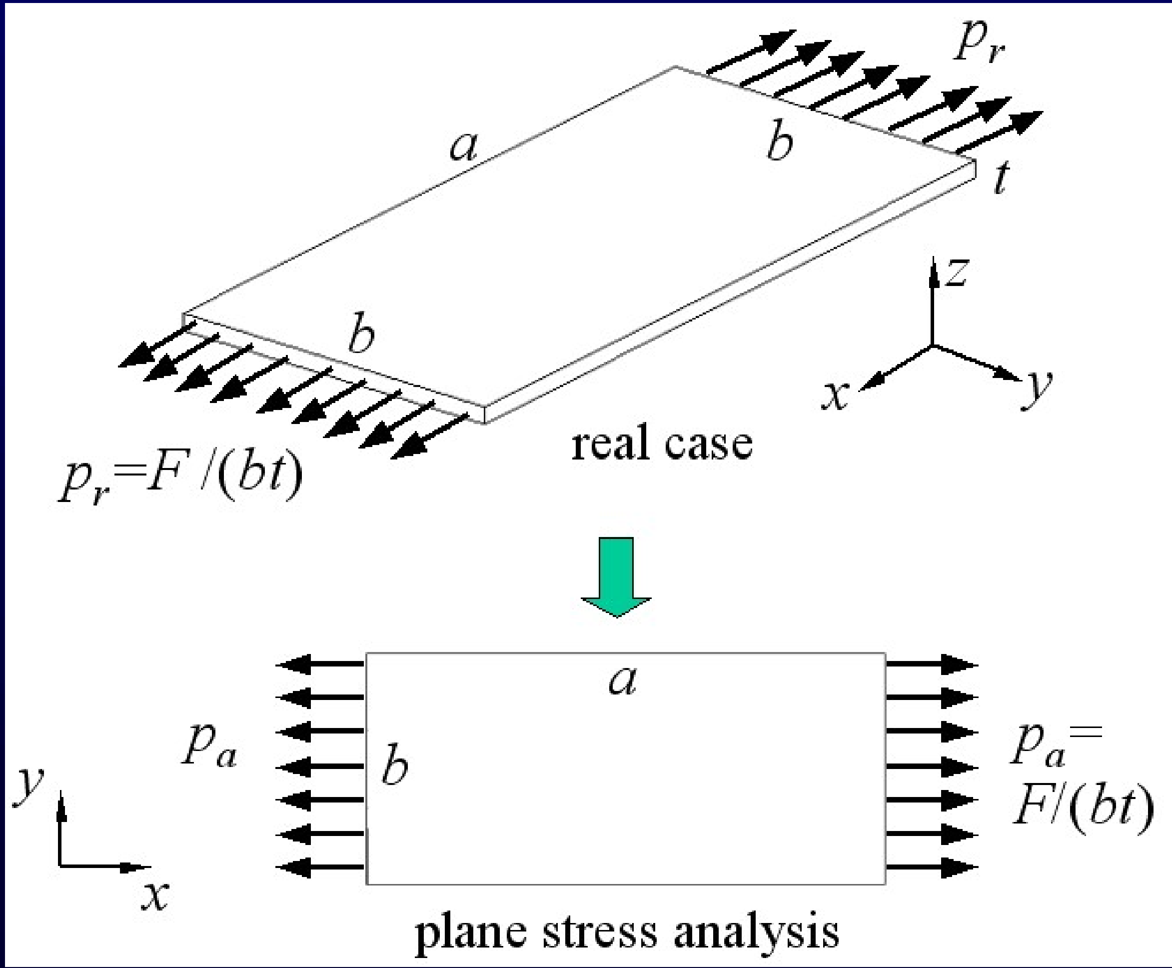
顯示圖形於整個視窗中

回復系統自訂方向視圖



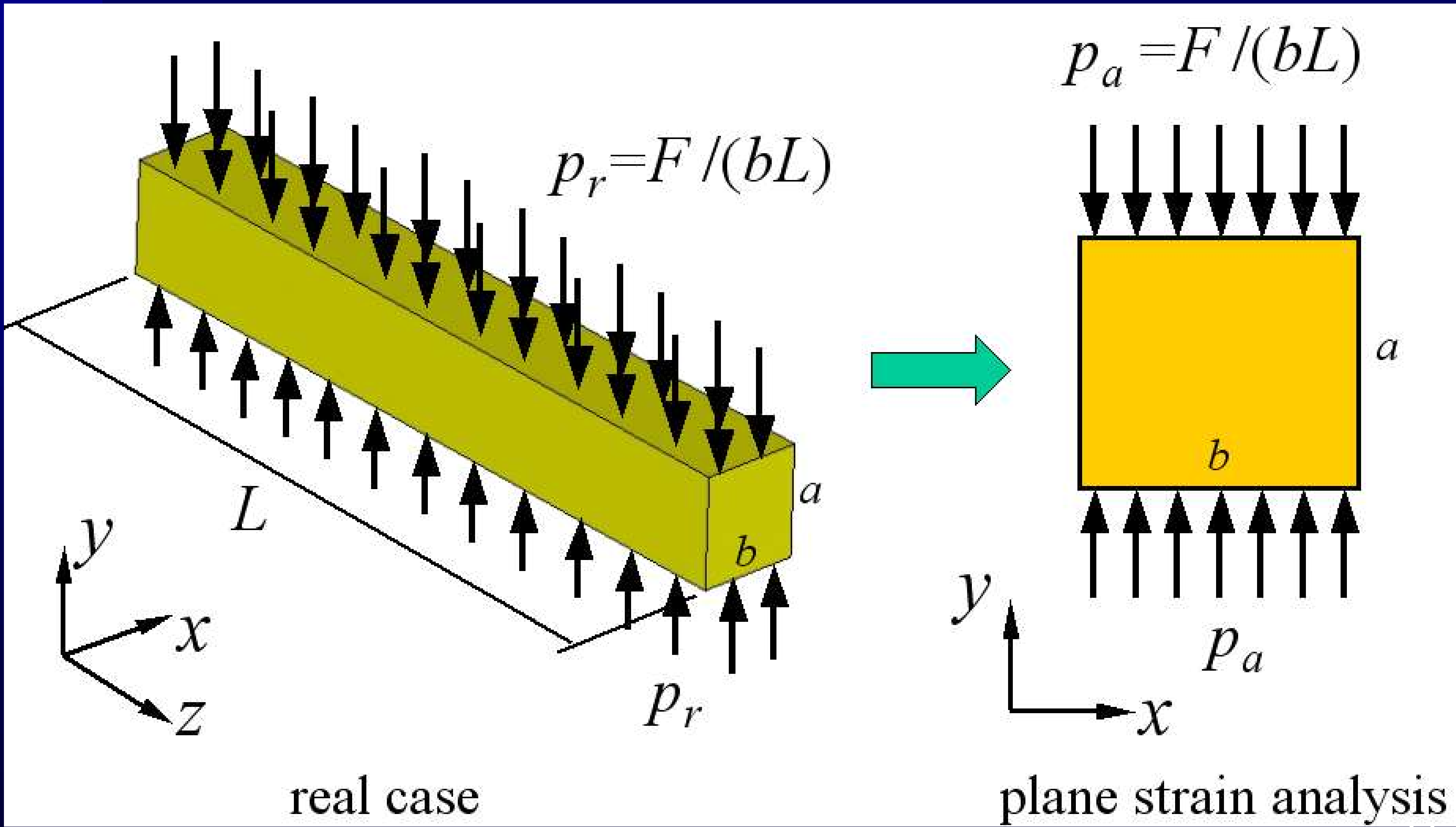
2D Plane problem

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



2D Plane problem

- Plane strain

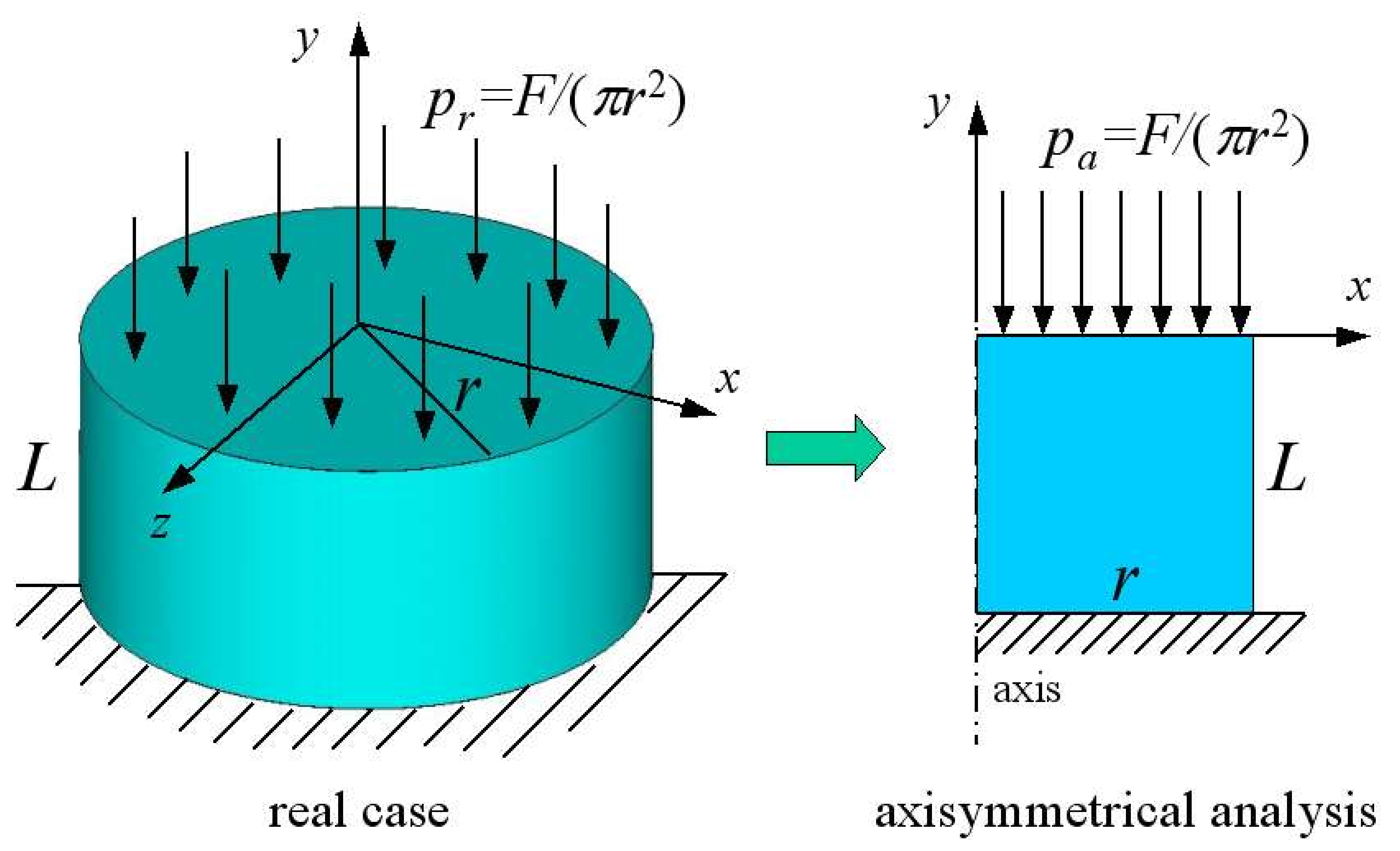


BME

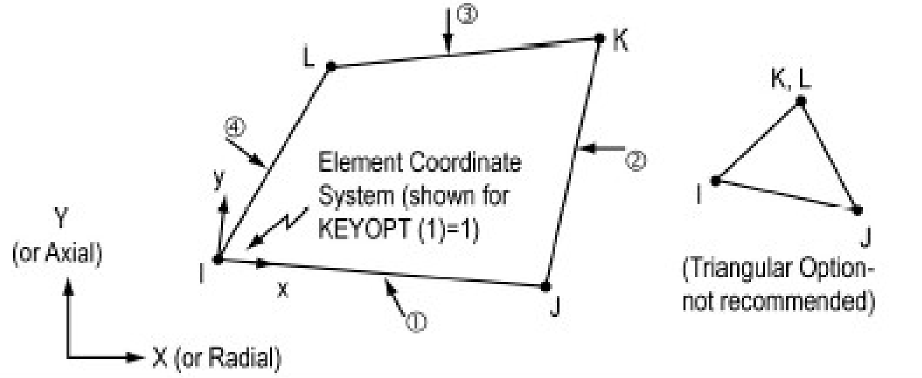
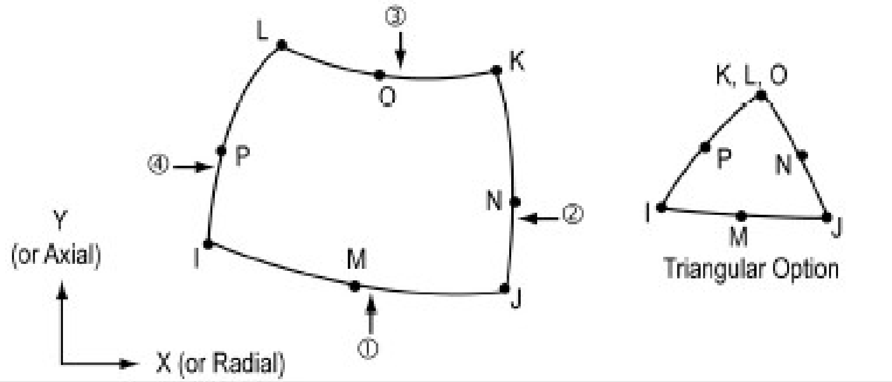
NYMU

2D Plane problem

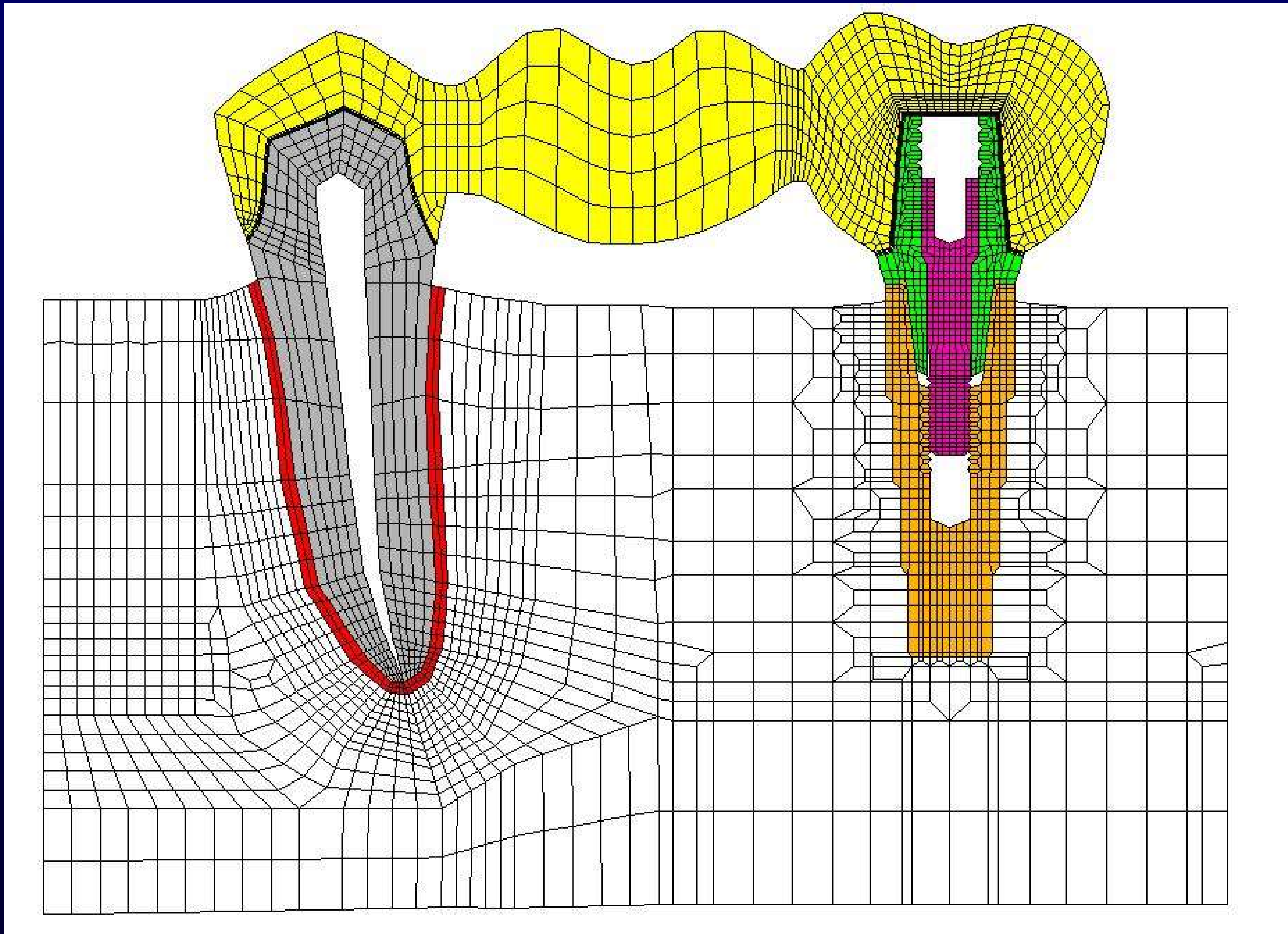
- **Anxisymmetric**



ANSYS 常用平面元素 (PLANE182/183)

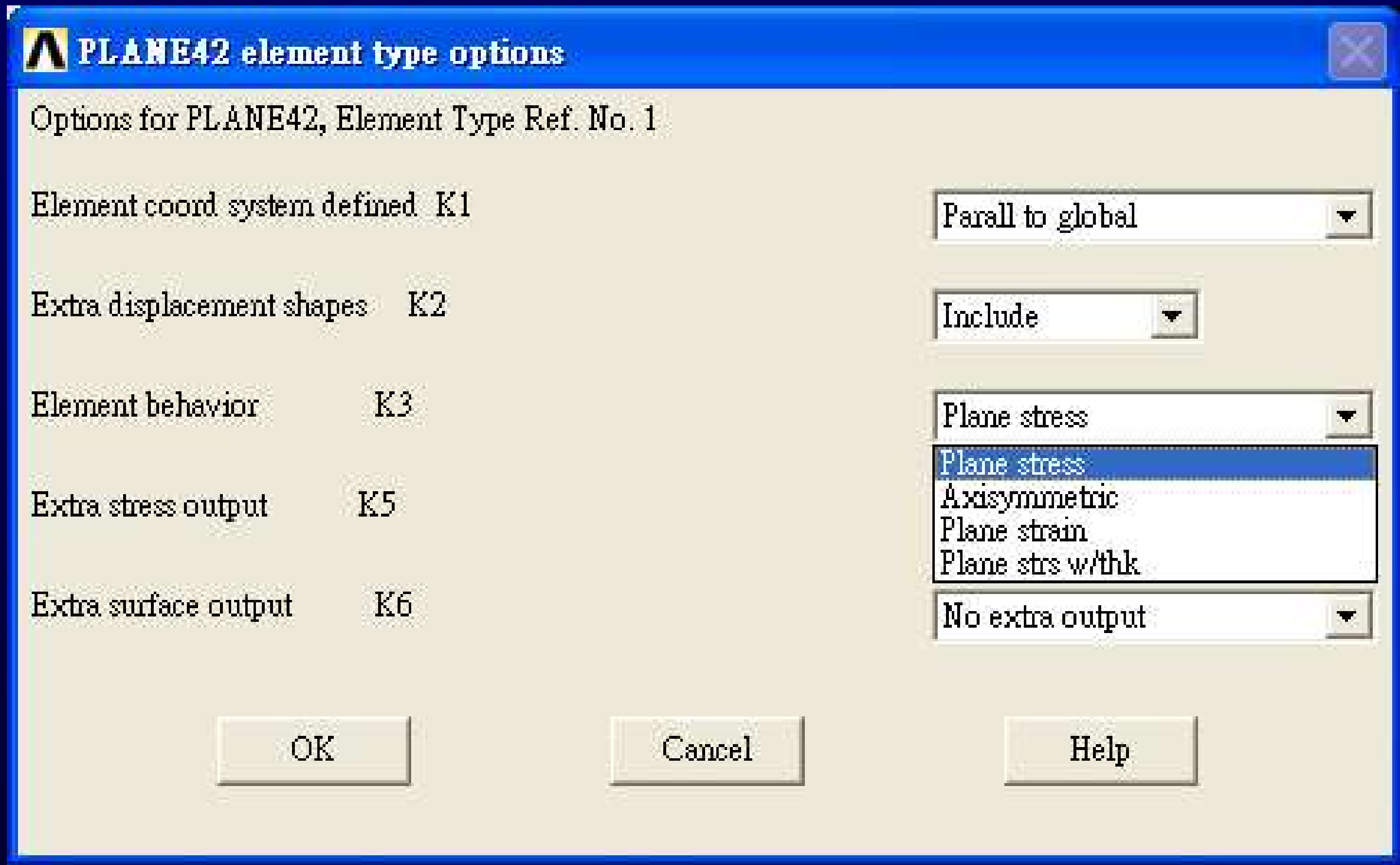
二維平面或軸對稱元素	**PLANE182**	2-D Structural Solid		節點數：4 節點自由度：UX,UY (*u*,*v*)
二維平面或軸對稱元素	**PLANE183**	2-D 8-Node Structural Solid		節點數：8 節點自由度：UX,UY (*u*,*v*)

Introduction of ANSYS –Plane182/183



Introduction of ANSYS

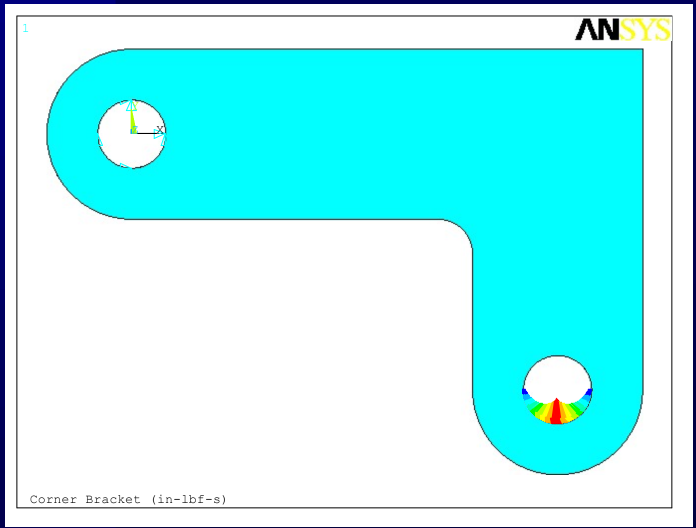
- Plane182/183
  - Plane stress, plane strain,
  - Axisymmetric, Plane strs w/thk

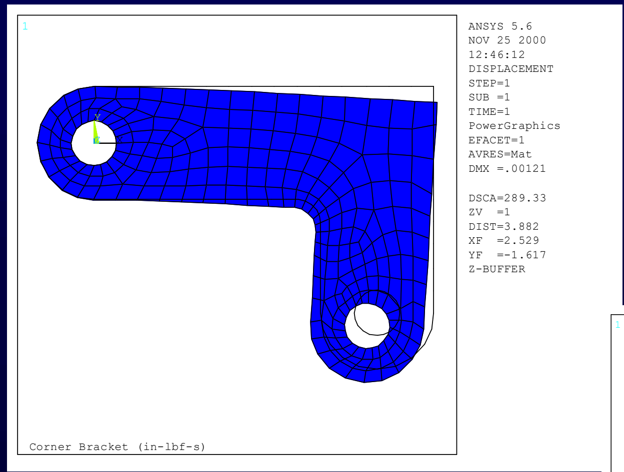


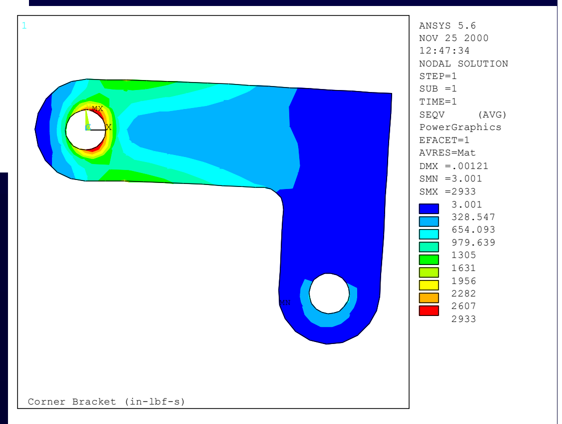
Introduction of ANSYS

- **The first case is called a "plane stress" case, since**

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







Introduction of ANSYS

- In the second case, if the cross-sections are along the radial directions then it is called an “axisymmetric” case

- In the third case, if the cross-sections are along the thickness then it is called a “plane strain” case, since

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

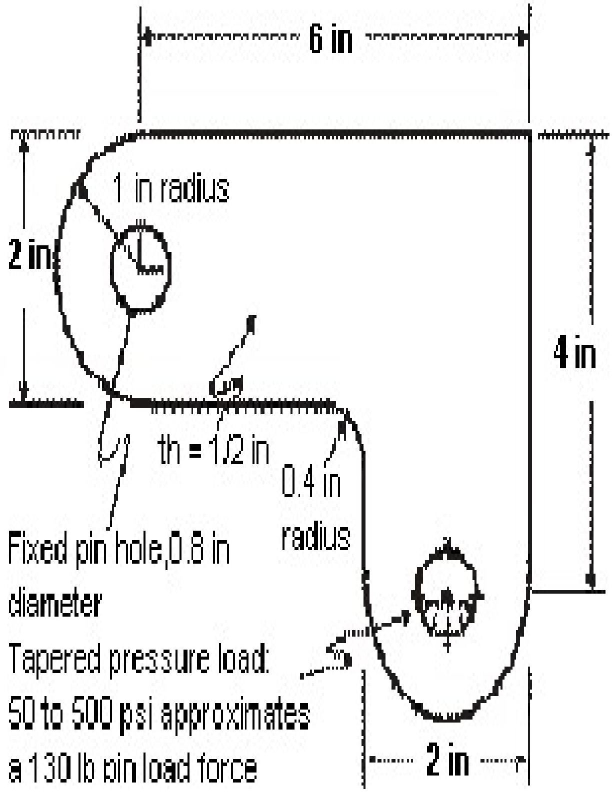
Plane stress example: (Ex 2) (PLANE183)

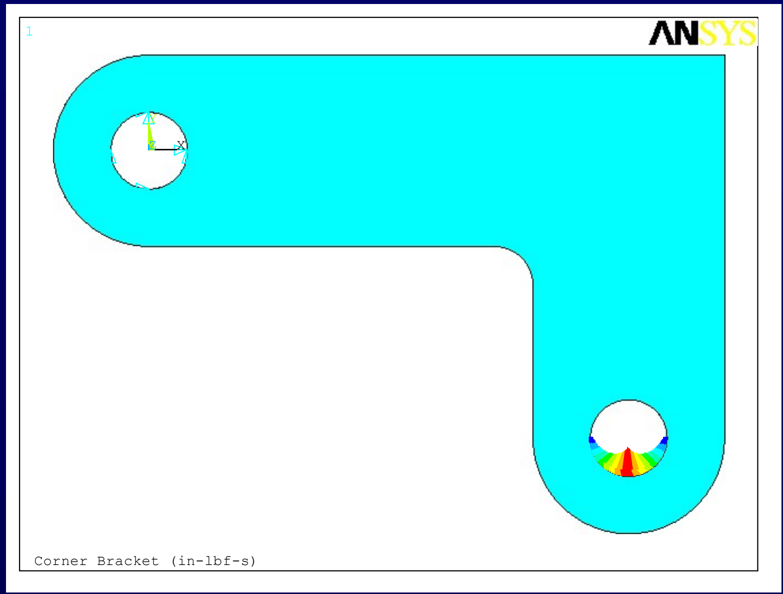
Static Analysis of a Corner Bracket

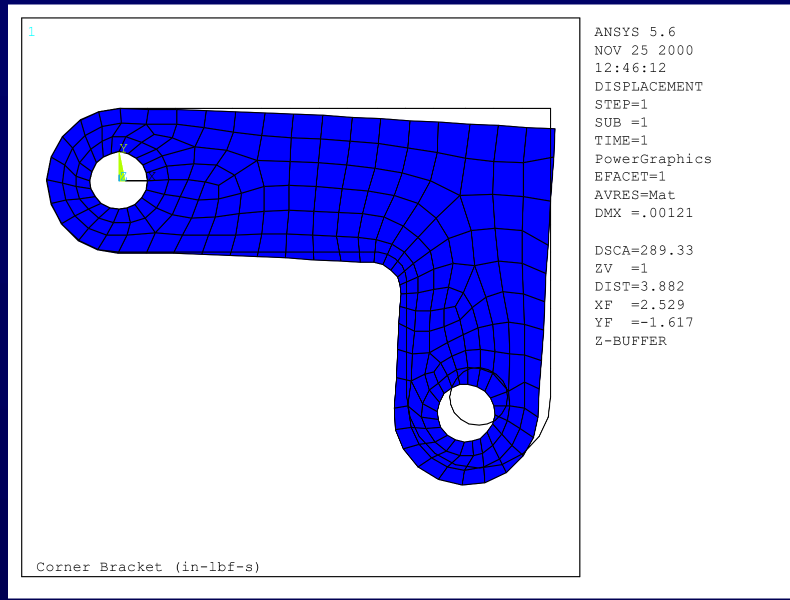
This is a simple, single load step, structural static analysis of the corner angle bracket shown below. The upper left-hand pin hole is constrained (welded) around its entire circumference, and a tapered pressure load is applied to the bottom of the lower right-hand pin hole. The objective of the problem is to demonstrate the typical ANSYS analysis procedure. The US Customary system of units is used.

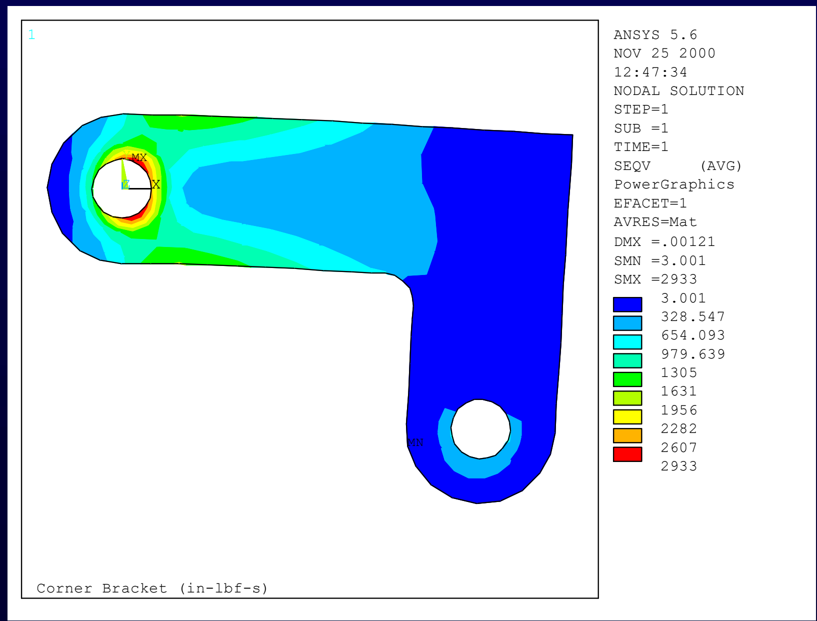
Young's modulus : 30e6 psi

Poisson's ratio : 0.27



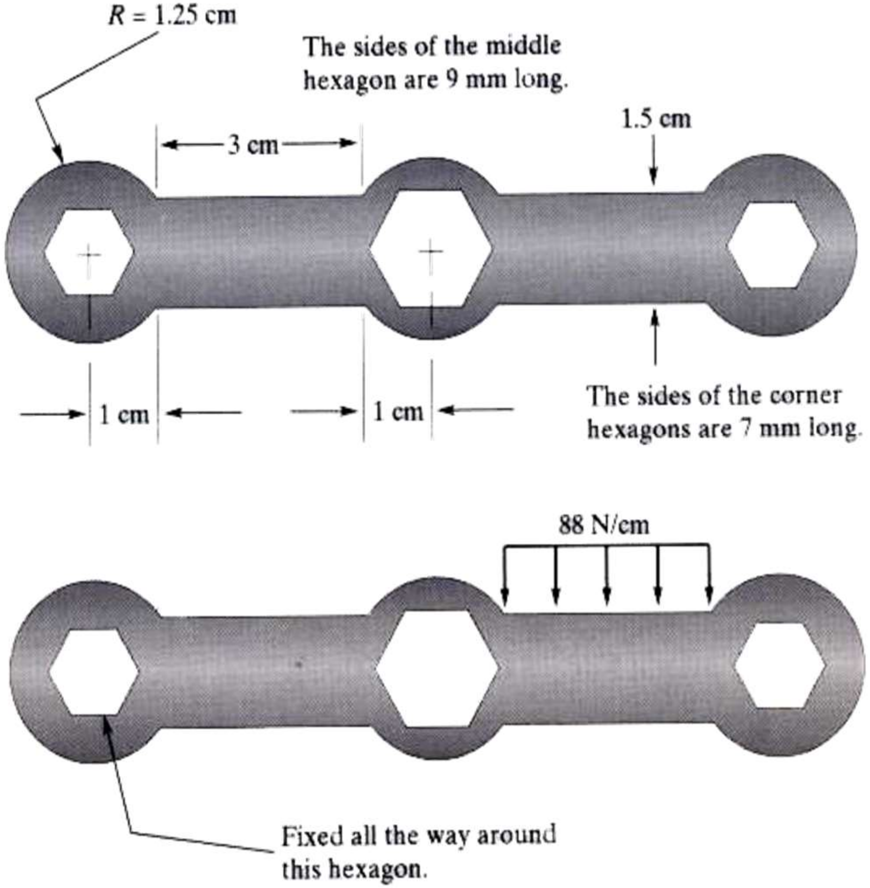

ANSYS
Corner Bracket (in-lbf-s)


ANSYS 5.6
NOV 25 2000
12:46:12
DISPLACEMENT
STEP=1
SUB =1
TIME=1
PowerGraphics
EFACET=1
AVRES=Mat
DMX =.00121
DSCA=289.33
ZV =1
DIST=3.882
XF =2.529
YF =-1.617
Z-BUFFER
Corner Bracket (in-lbf-s)


ANSYS 5.6
NOV 25 2000
12:47:34
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (AVG)
PowerGraphics
EFACET=1
AVRES=Mat
DMX =.00121
SMN =3.001
SMX =2933
3.001
328.547
654.093
979.639
1305
1631
1956
2282
2607
2933
Corner Bracket (in-lbf-s)

Other plane stress example: (Ex 3)

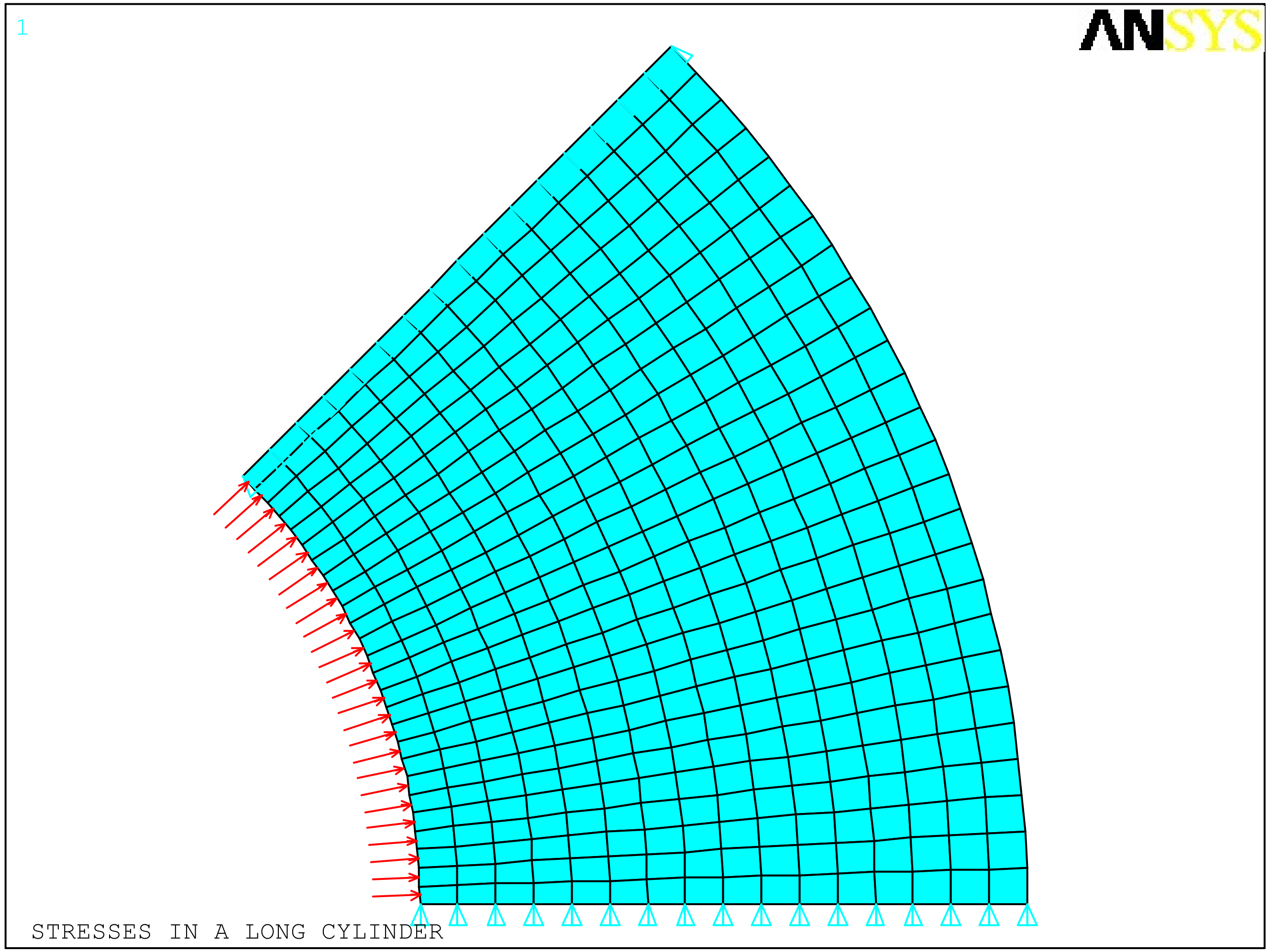
- **The bicycle wrench shown in figure is made of steel with a modulus of elasticity E = 200 GPa and Poisson's ratio v= 0.32. The wrench is 3 mm thick. Determine the von Mises stresses under the given distributed load and boundary conditions. The following steps demonstrate how to (1) create the geometry of the problem . (2) choose the appropriate element type (3) apply boundary condition.(4) *obtain nodal results*.**



Plane strain example: (Ex 4)(PLANE182)

- Stresses in a Long Cylinder
  - A long thick-walled cylinder (with inner radius = 4 in. and outer radius = 8 in.) 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. 30E6 v=0.3 Density=0.00073

Plane strain example: (Ex 4)

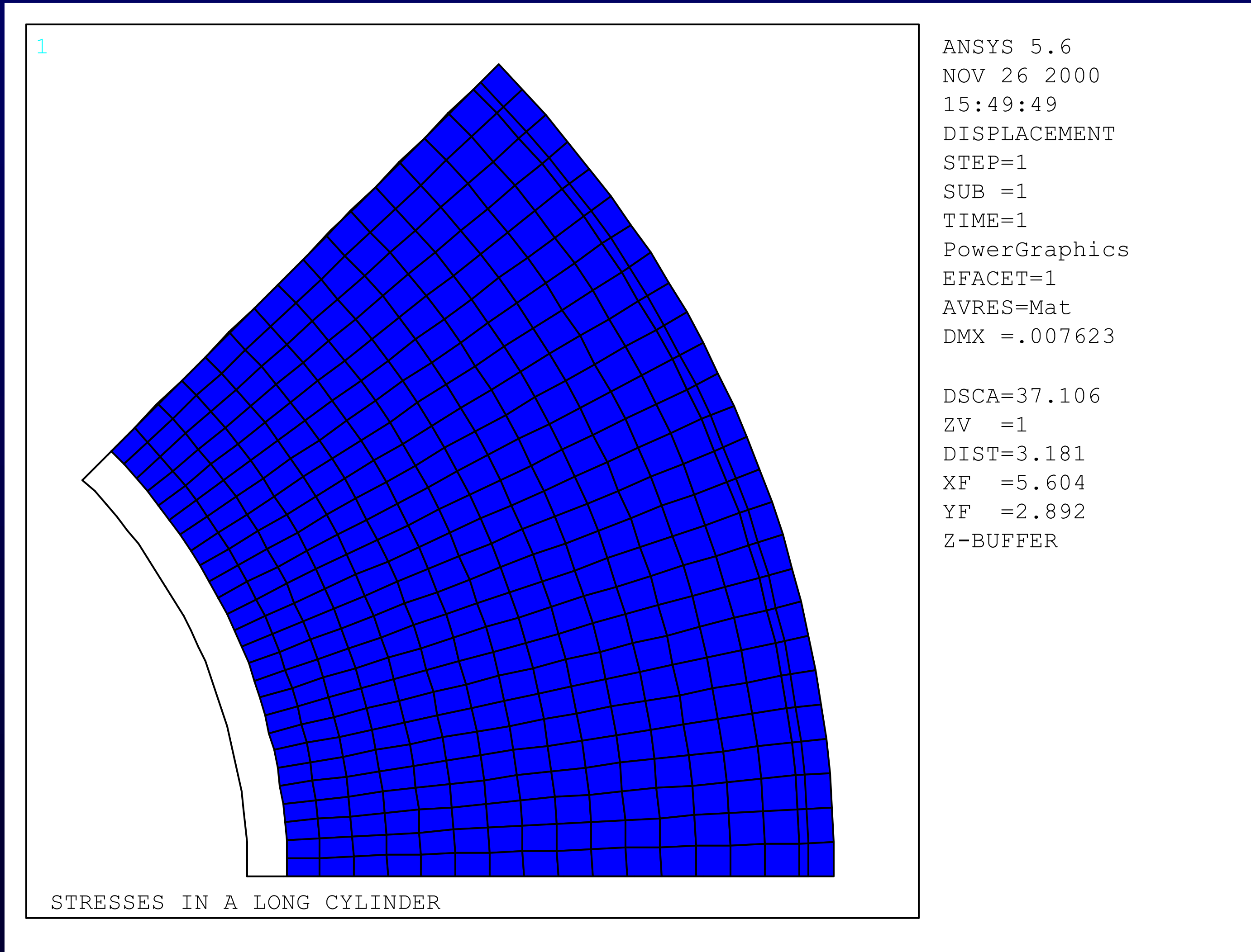


Plane strain example: (Ex 4)

- Plane Strain Approach -- commands

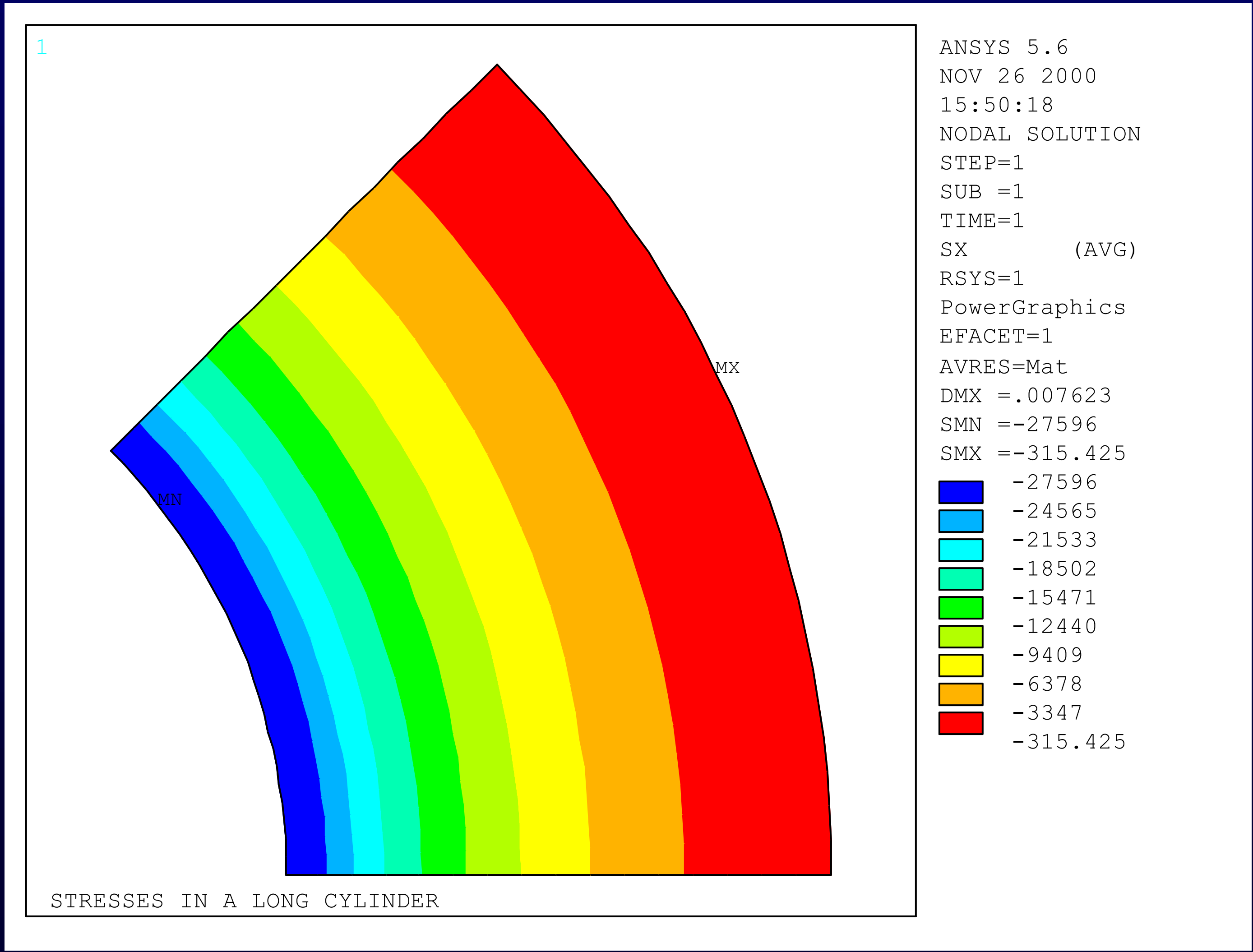
```
/CLEAR
/TITLE, STRESSES IN A LONG
   CYLINDER
/UNITS, BIN

/PREP7

  ET, 1, PLANE182,,, 2
  MP, EX, 1, 30E6
  MP, DENS, 1, .00073
  MP, NUXY, 1, 0.3

  CSYS, 1 (Cylindrical)
  K, 1, 4, 0 (r, degree)
  K, 2, 8, 0
  K, 3, 4, 45
  K, 4, 8, 45
  A, 3, 1, 2, 4

ESIZE, .25
MSHK, 1
MSHA, 0, 2D
AMESH, 1 (Mapping)
NROTAT, ALL
FINISH

/SOLU

S, LOC, Y, 0
NSEL, A, LOC, Y, 45
D, ALL, UY, 0

NSEL, S, LOC, X, 4
SF, ALL, PRES, 30000
 (Loading)
ALLSEL

/PBC, U,, ON
/PSF, PRES, NORM, 2
EPLOT
SOLVE

NSEL, S, LOC, X, 4
SF, ALL, PRES, 0
 (delect)
ALLSEL
OMEGA,,, 1000 (rotation)
SOLVE
FINISH

/POST1

SET, 1
PLDISP, 2
RSYS, 1 (Cylindrical)
PLNSOL, S, X
PLNSOL, S, Y
```

Plane strain example: (Ex 4)



Plane strain example: (Ex 4)

- Radial Stress



BME

NYMU

Plane strain example: (Ex 4)

- Hoop Stress