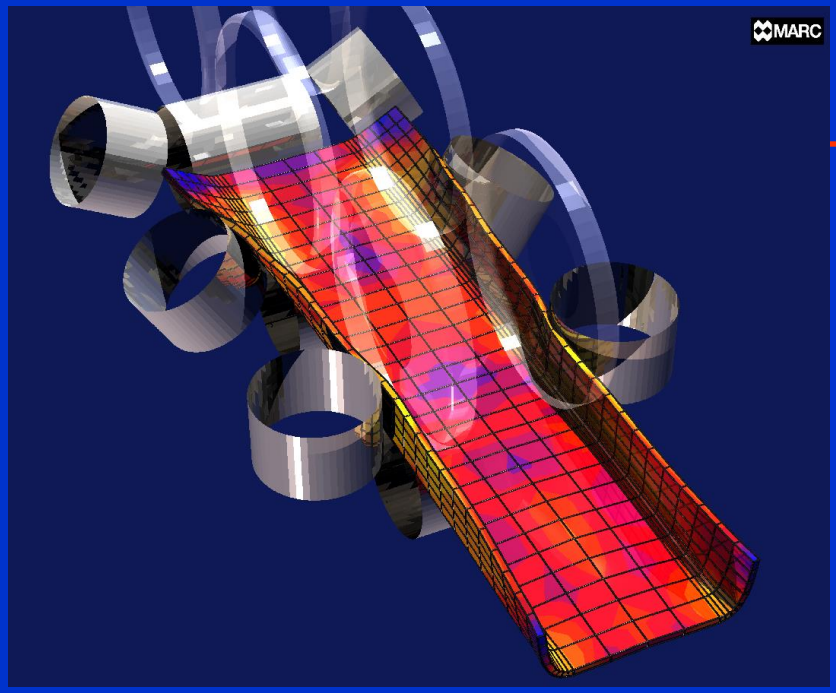
ANSYS®

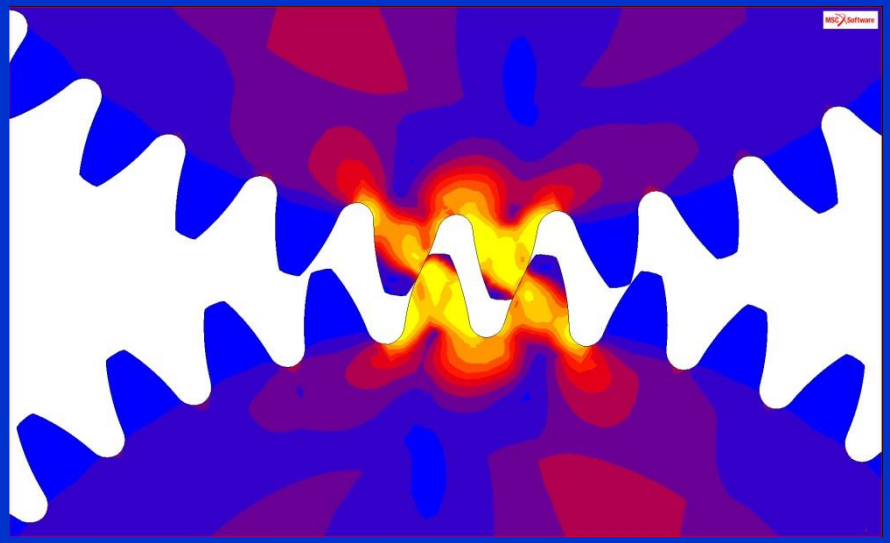
Module 9

Introduction to Nonlinear Analysis

Metal Forming



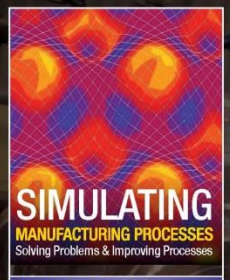
Gear Contact



simulating REALITY

MSC Software Magazine

Volume III | Supplemental Issue - Summer 2013



SIMULATING
MANUFACTURING PROCESSES
Solving Problems & Improving Processes

THE HADLEY GROUP

Simulation Helps Increase Sales by $4M

Nonlinear FEA Validates New Cold Roll Forming Process

COMTES/PILSEN STEEL

Getting to the Root Cause of Cracks

Heat Transfer Analysis Helps Solve Tough Forging Problem

INNOVATING
MOMENTUM IN GROUND VEHICLES
New Tire Development & Optimized Engines

BIG TYRE

Designing the Right Tire

Nonlinear FEA Accelerates Development of Non-Pneumatic, Non-Solid Tire for Mining Industry

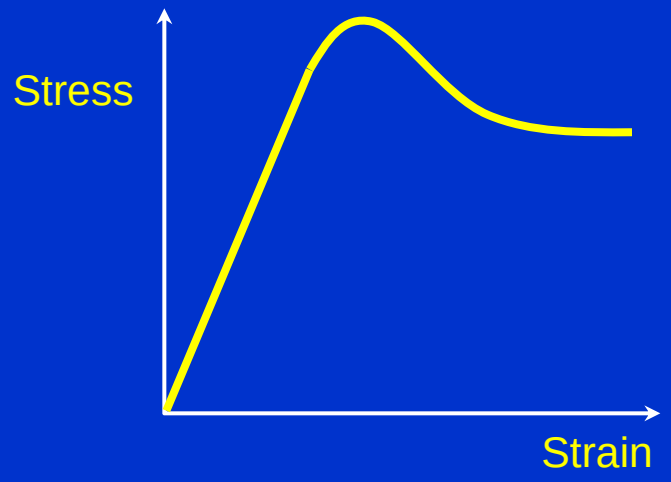
MANUFACTURING COMPANIES RELY ON MARC

The World is Nonlinear

9. Introduction to Nonlinear Analysis

ANSYS Training Manual

- A *nonlinear* analysis is needed if the loading on a structure causes significant changes in stiffness. Typical reasons for stiffness to change significantly are:
  - Strains beyond the elastic limit (plasticity)
  - Large deflections, such as with a loaded fishing rod
  - Contact between two bodies

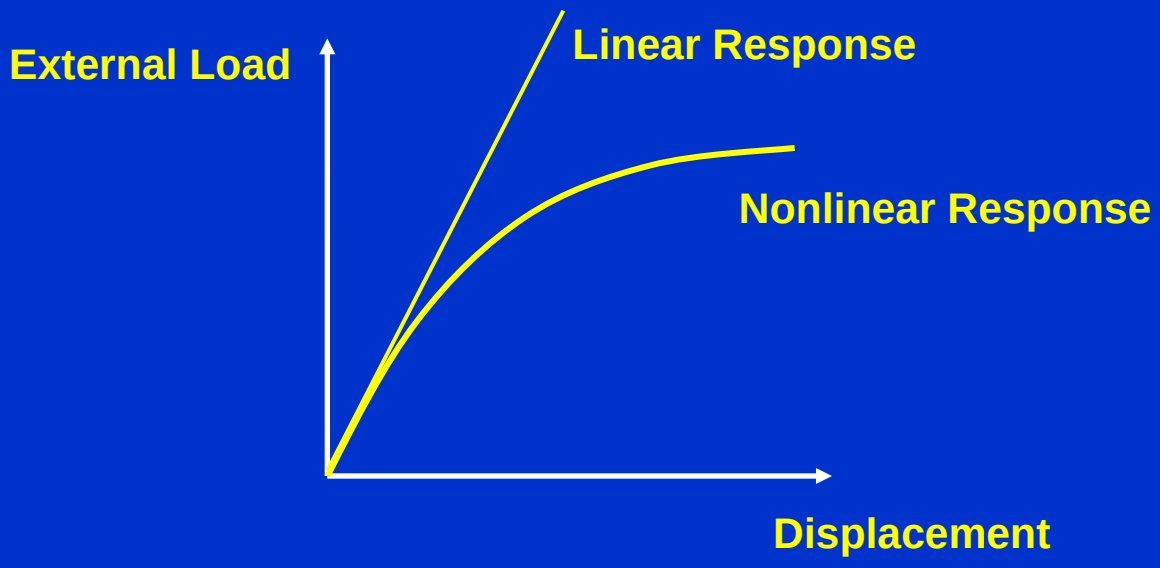


...Introduction to Nonlinear Analysis

- In this chapter, we will present a brief introduction to the basics of nonlinear solution via the following topics:
  - A. Basic Concepts
  - B. Typical Procedure
  - C. Workshop
- The purpose is to give you a "taste" of nonlinear analysis. There are many, many aspects of nonlinear analysis that are well beyond the scope of this training course and are covered in:
  - *ANSYS Structural Analysis Guide*
  - *Basic Structural Nonlinearities* Training Manual
  - *Advanced Contact and Bolt Pretension* Training Manual
  - *Advanced Structural Nonlinearities* Training Manual

A. Basic Concepts

- **When a load causes significant changes in stiffness, the load-deflection curve becomes nonlinear.**
- **The challenge is to calculate the nonlinear displacement response using a linear set of equations.**



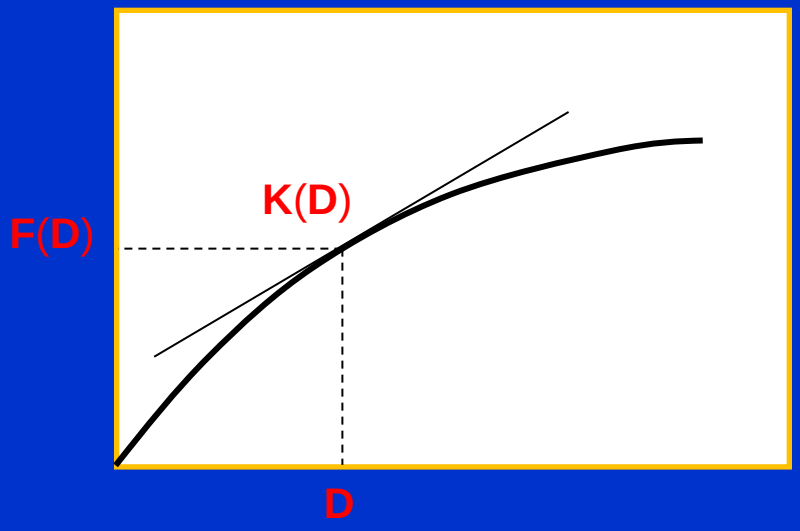
Equations for a Nonlinear Structure

元素的力平衡方程式 $k \times d = f$

整體結構的力平衡方程式 $K \times D = F$

非線性結構的力平衡方程式，其形式還是如上所描述，但是其中的**K**及 **F**不再是常數，而是依 **D**的改變而改變:

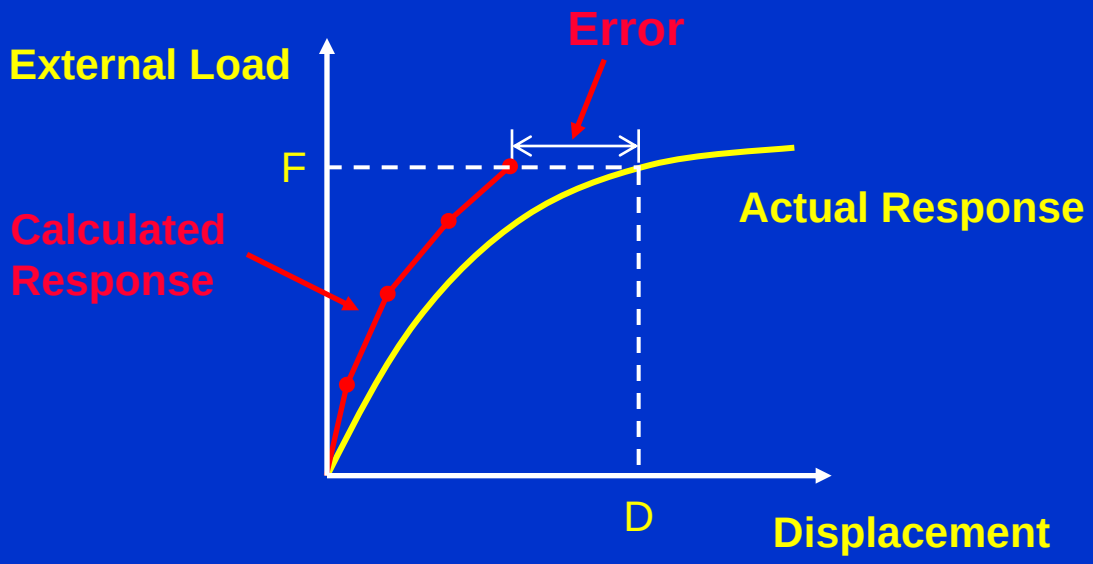
$$K(D) \times D = F(D)$$



求解施力**F**時之位移量**D**

...Basic Concepts

- **Incremental method :**
- **This approach is to apply the load gradually by dividing it into a series of increments and adjusting the stiffness matrix at the end of each increment.**
- **The problem with this approach is that errors accumulate with each load increment, causing the final results to be out of equilibrium.**



...Basic Concepts

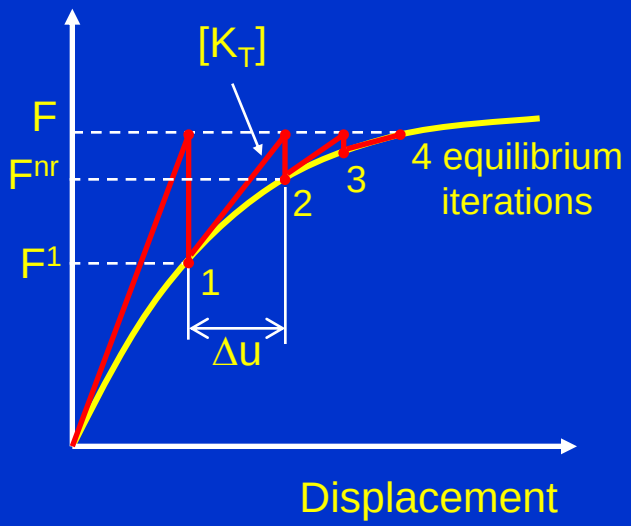
- **ANSYS uses the Newton-Raphson algorithm:**
  - **Applies the load gradually, in increments.**
  - **Also performs *equilibrium iterations* at each load increment to drive the incremental solution to equilibrium.**
  - **Solves the equation** $[K_T]\{\Delta u\} = \{F\} - \{F^{nr}\}$

    $[K_T]$ **= tangent stiffness matrix**

    $\{\Delta u\}$ **= displacement increment**

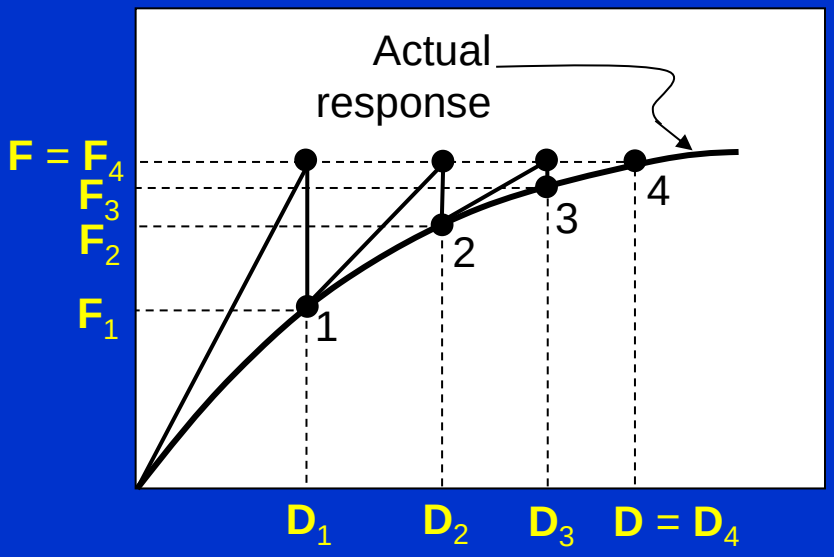
    $\{F\}$ **= external load vector**

    $\{F^{nr}\}$ **= internal force vector**



  - **Iterations continue until** $\{F\} - \{F^{nr}\}$ **(difference between external and internal loads) is within a tolerance.**
  - **Some nonlinear analyses have trouble converging. Advanced analysis techniques are available in such cases (covered in the *Advanced Structural Nonlinearities* training course).**

Convergence Criteria in ANSYS



兩次疊代之間的變位差ΔD

$$\|\Delta \mathbf{D}\| \le 0.05 \|\mathbf{D}_{\max}\|$$

兩次疊代之間的外力差ΔF

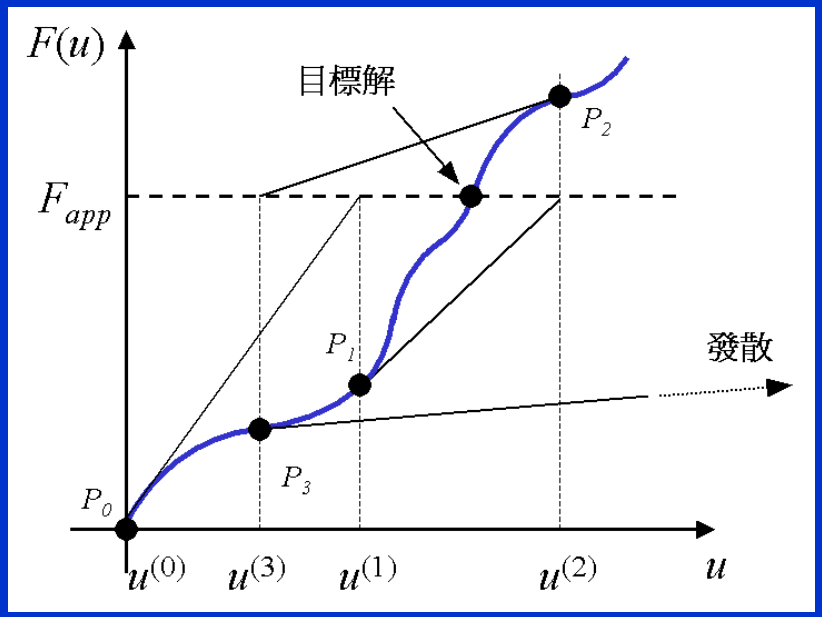
$$\|\Delta \mathbf{F}\| \le 0.005 \|\mathbf{F}_{\max}\|$$

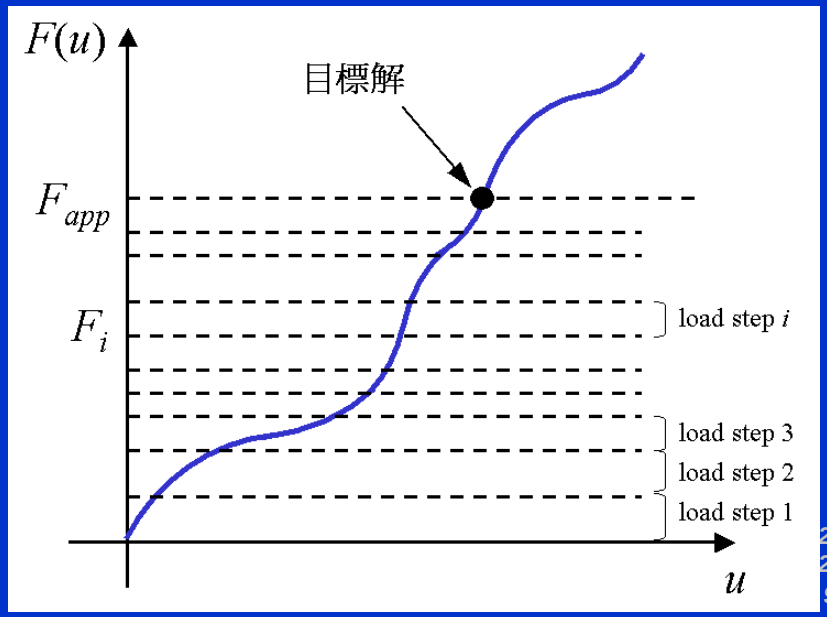
Newton-Raphson method之疊代發散

ANSYS Training Manual

若**F(u)**函數太複雜，或給定的起始點**P0**之值不對，則疊代分析有可能會發散，即無法收斂至目標解。

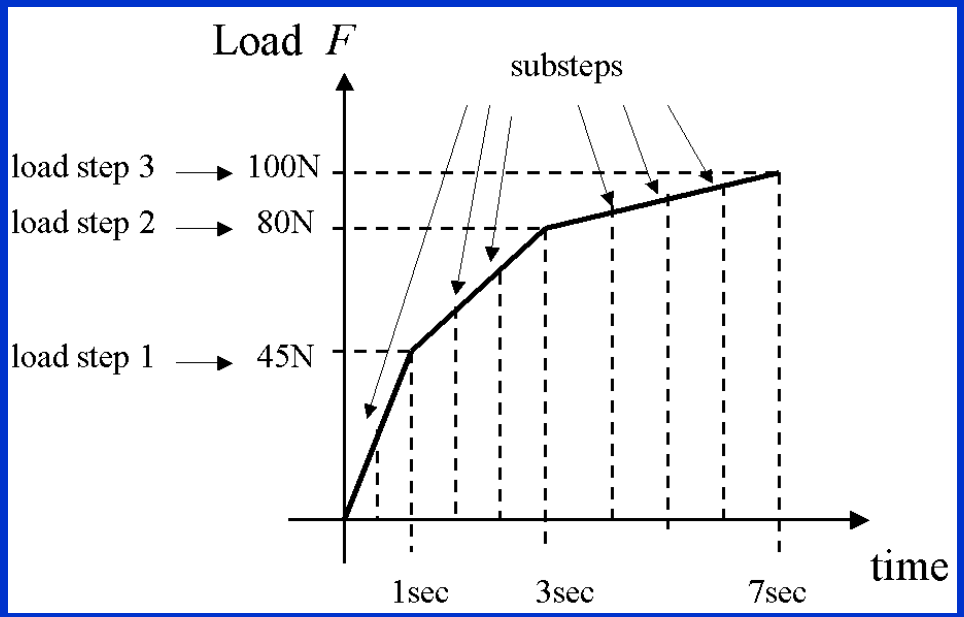
將外力**Fapp**分割成數個負荷增量**Fi (i =1,2,3,…)**來接連計算，**Fi** 一步一步施加於結構，在每個增量範圍內均分別做牛頓-拉福森法疊代計算，由於每個增量區間之函數較單純，所以較容易收斂。

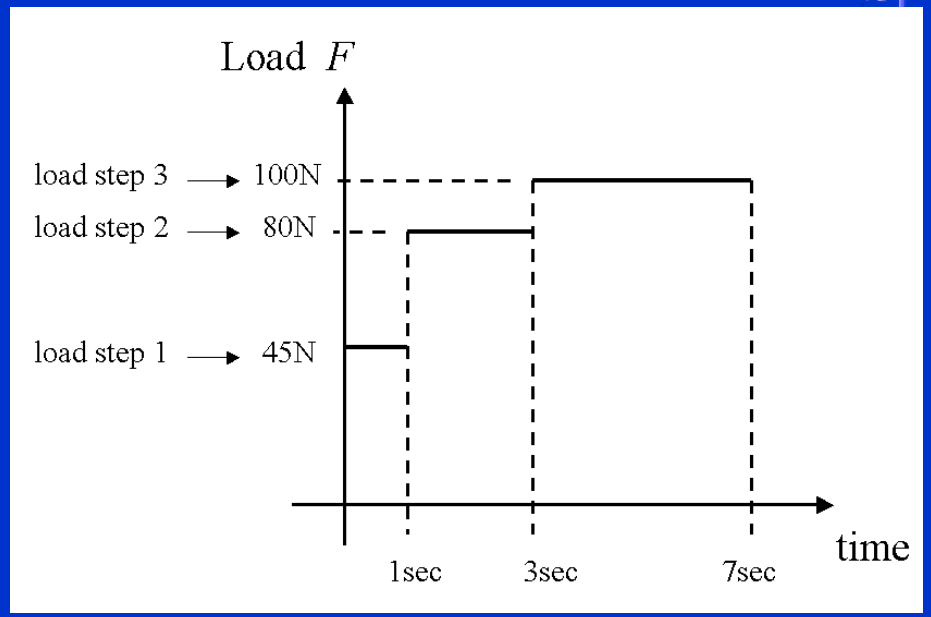




在靜態分析中無時間效應，使用者所定義之時間僅為負荷增量之參考點，並無物理意義，此「時間」並不參與計算。

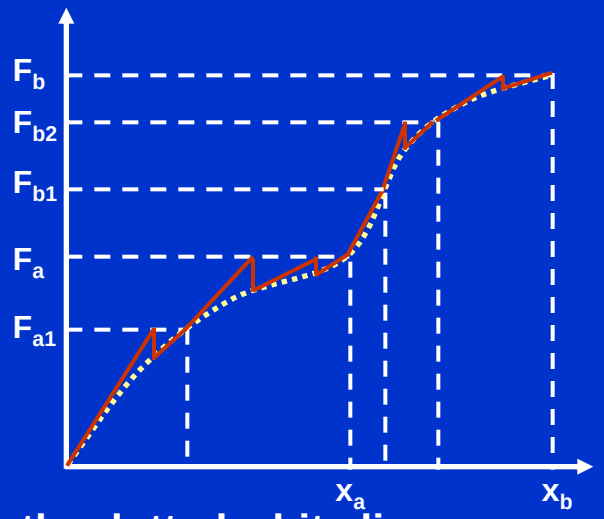
以ANSYS設定受力由0N增至100N為例，若將100N之力分為三個負荷增量(load steps)，再個別細分為2個、3個和4個副增量(substeps)，且在ANSYS設定中使用傾斜線(ramped)之加力方式，施力過程便如圖8.12所示，施力分別為「0N→45N(0～1秒)」，「45N→80N(1～3秒)」，「80N→100N(3～7秒)」。





… Nonlinear Solution

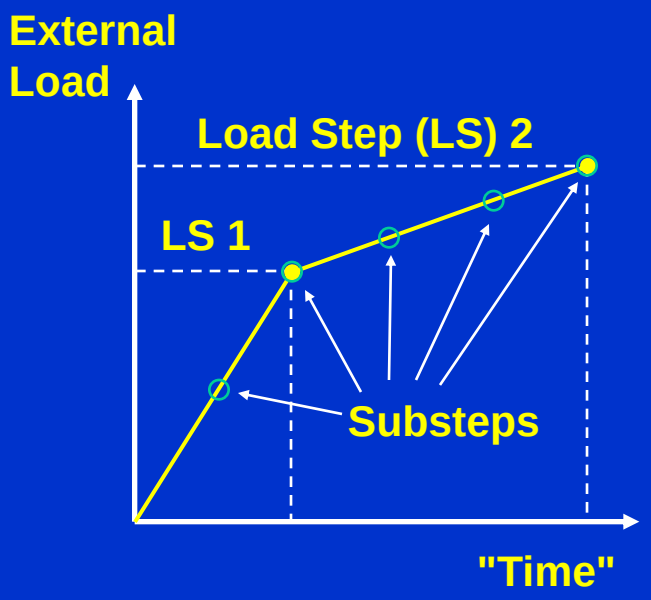
- **It is useful to understand how loads are managed**
  - ***Load steps*** **are changes in general loading.**
    - **Simulation usually solves all nonlinear models with one load step, but, in the case of Pretension Bolt Loads, this is done in two load steps. The bolt preload is applied first, then all other loads are applied next. These *load steps* can be thought of as F_a and F_b.**
  - ***Substeps*** **apply the loads in an incremental fashion**
    - **Because of the complex response, it may be necessary to apply the load incrementally. For example, F_{a1} may be near 50% of the F_a load. After the load for F_{a1} is converged, then the full F_a load is applied. F_a has 2 substeps while F_b has 3 substeps in this example**



  - ***Equilibrium iterations*** **are the corrective solutions to obtain a converged substep**
    - **In the example on right, the iterations between the dotted white lines indicate equilibrium iterations.**

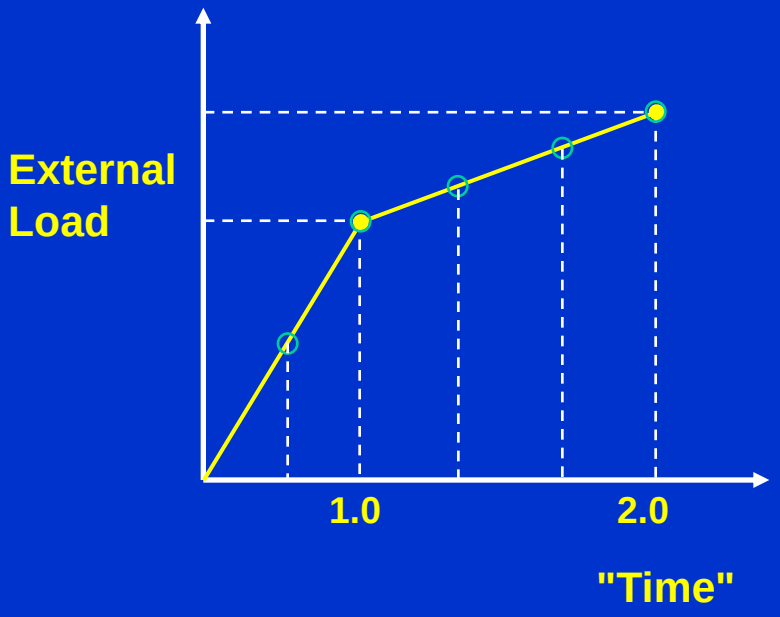
...Basic Concepts

- **This process is repeated for each load increment until the full external load has been applied.**
- **Thus a nonlinear solution typically involves the following:**
  - **一個 SOLVE 命令代表一個 load step**
  - **One or more *load steps* to apply the external loads and boundary conditions. (This is true of linear analyses too.)**
  - **Multiple *substeps* to apply the load gradually. Each substep represents one load increment. (A linear analysis needs just one substep per load step.)**
  - ***Equilibrium iterations* to obtain equilibrium (or *convergence*) at each substep. (Does not apply to linear analyses.)**



Time and Time Step

- **Each load step and substep is associated with a value of *time*.**
- ***Time* in most nonlinear static analyses is simply used as a counter and does not mean actual, chronological time.**



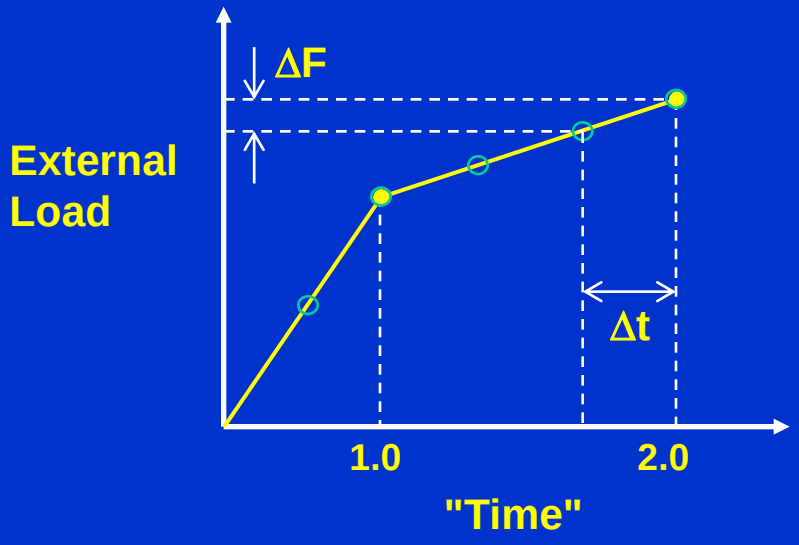
  - **By default, time = 1.0 at the end of load step 1, 2.0 at the end of load step 2, and so on.**
  - **For rate-independent analyses, you can set it to any desired value for convenience. For example, by setting time equal to the load magnitude, you can easily plot the load-deflection curve.**

Concepts of Time

- The ends of load steps or substeps can be identified by time.
- For dynamic problem, time is used as a real-world clock.
- For static problem, time is used as a counter.

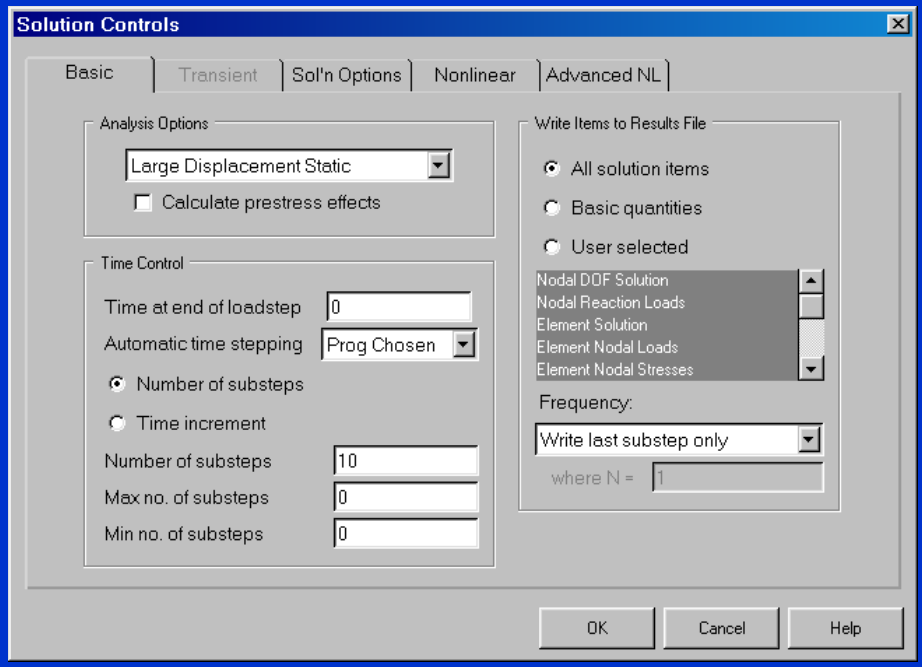
...Basic Concepts

- The "time increment" between each substep is the ***time step*** **Δt.**
- Time step Δt determines the load increment ΔF over a substep. The higher the value of Δt, the larger the ΔF, so Δt has a direct effect on the accuracy of the solution.
- ANSYS has an ***automatic time stepping*** algorithm that predicts and controls the time step size for all substeps in a load step.



B. Typical Procedure

- **Assuming that the geometry and meshing step have been completed, typical steps for a nonlinear solution are:**
  1. **Specify analysis type (usually static).**
  2. **Specify solution controls using Solution > Analysis Type > Sol'n Controls. Many controls are available, but the common ones are:**
     - **small or large deflection**
     - **time and ΔT or number of substeps**
     - **output controls**
  3. **Apply the loading.**
  4. **Save the database.**
  5. **Solve.**



Nonlinear Geometry (NLGEOM)

- NLGEOM turns ON/OFF the effects of geometric nonlinearity.

Number of Substeps (NSUBST, DELTIM)


```
NSUBST, NSBSTP, NSBMX, NSBMN, Carry
```

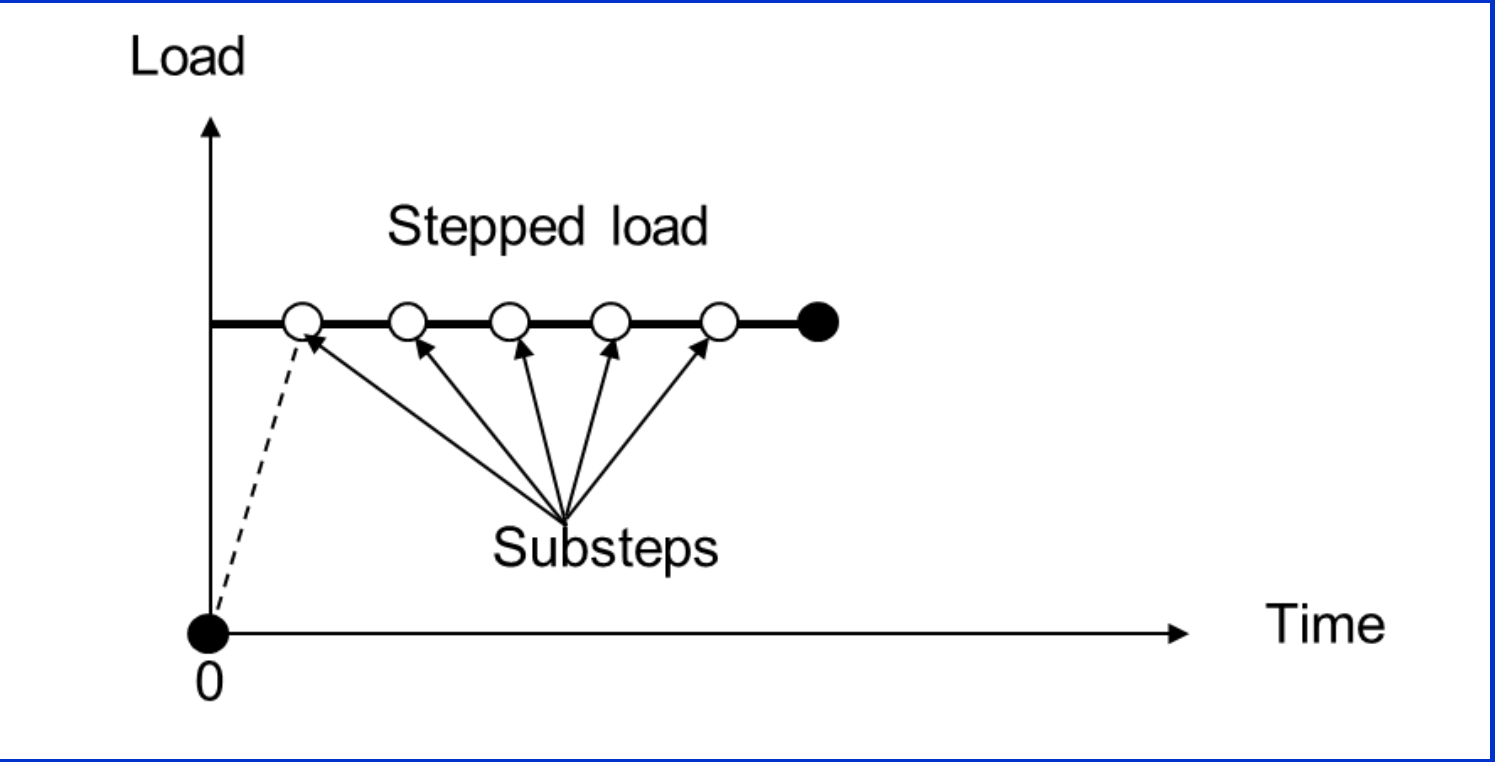
```
DELTIM, DTIME, DTMIN, DTMAX, Carry
```

- NSUBST specifies number of substeps for the following load step.
- DELTIM specifies time for each substep for the following load step.
- The two commands are equivalent, i.e., $TIME = DELTIM \times NSUBST$

Automatic Time Stepping (AUTOTS)

- For nonlinear problem, the user-input Δt is used as initial incremental time.
- ANSYS adjusts Δt automatically according to the convergence behavior of the solution.
- AUTOTS turns ON/OFF the functions of auto time stepping.

Ramped/Stepped Loading (KBC)



當load step是非常stepped時，必須取非常細的substeps

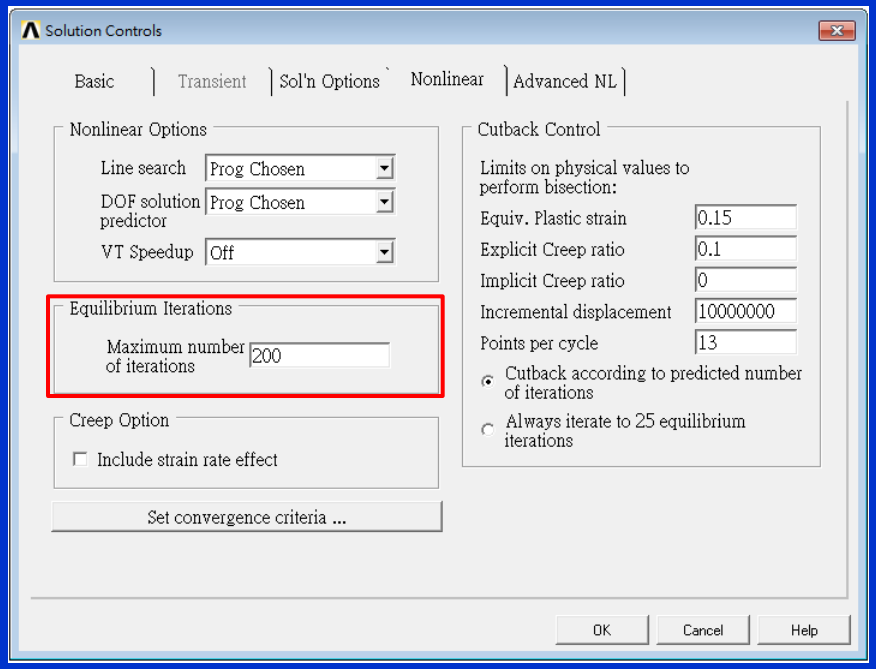
Time Step Prediction Based on Contact Status (SOLCONTROL)

SOLCONTROL, *Key1*, *Key2*

- SOLCONTROL command can be used to activate automatic nonlinear solution control algorithm.
- The default is ON.
- The second key of SOLCONTROL command is to ask ANSYS to adjust time steps according to contact status.

Number of Equilibrium Iterations (NEQIT)

- NEQIT specifies the number of iterations beyond which ANSYS would start another "attempt".
- The default is between 15 and 26.



C. Workshop

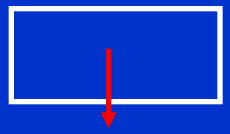
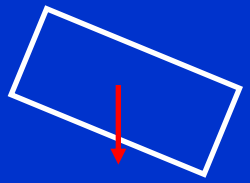
- **This workshop consists of the following problem:**

  W9. Arched Beam

  Please refer to your *Workshop Supplement* for instructions.

… Load Orientation

- It is important to note the orientation of loads and its effect on the structure in large-deflection analyses:

Load	Direction Before Deflection	Direction After Deflection
Acceleration (constant direction)		
Force, Moment, Bolt Load (constant direction)		
Pressure (always normal to surface)