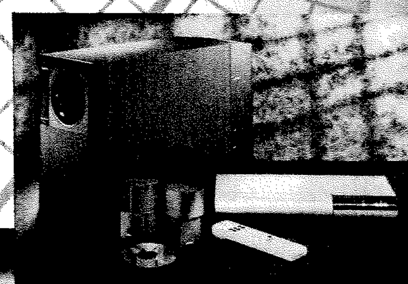
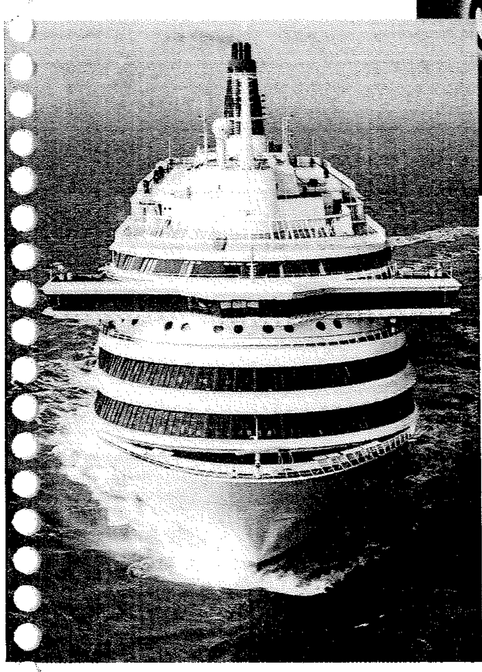
Structural Nonlinearities

Training Manual

for Release 5.5

001156 (Second Edition)
Volume I
April 30, 1999





ANSYS, Inc.
is a UL registered
ISO 9001 : 1994
Company

Structural Nonlinearities

For Revision 5.5

STRUCTURAL NONLINEARITIES
Revision History

001156	Upd0	ANSYS Release 5.5	September 30, 1998
001156	Upd1	ANSYS Release 5.5	April 30, 1999

Registered Trademarks

ANSYS® is a registered trademark of SAS IP Inc.
All other product names mentioned in this manual are trademarks or registered trademarks of their respective manufacturers.

Disclaimer Notice

This document has been reviewed and approved in accordance with ANSYS, Inc. Documentation Review and Approval Procedures.

"This ANSYS, Inc. software product (the Program) and program documentation (Documentation) are furnished by ANSYS, Inc. under an ANSYS Software License Agreement that contains provisions concerning non-disclosure, copying, length and nature of use, warranties, disclaimers and remedies, and other provisions. The Program and Documentation may be used or copied only in accordance with the terms of that License Agreement."

Table of Contents

Chapter 1

Seminar Overview

Session Objective

- At the end of this session you will be able to describe the following:
  - 1. Training Course Objectives
  - 2. Classes of Nonlinear Problems
  - 3. ANSYS Nonlinear Solution

Training Course Objectives

- Provide a high level theoretical understanding of nonlinear solutions.
  - This training course is not intended to provide detailed theoretical instruction in nonlinear mechanics.
  - The intent is to provide a conceptual understanding of the underlying physics for the various classes of nonlinear problems and solution techniques.

Training Course Objectives

- **Emphasize the most commonly used nonlinear features of ANSYS.**
  - Nonlinear Geometry
    - Large Strains, Large Deflections and Large Rotations
  - Structural Stability
    - Pre and Post Buckling Analysis
  - Plasticity
  - Hyperelasticity
  - Contact Nonlinearities

Training Course Objectives

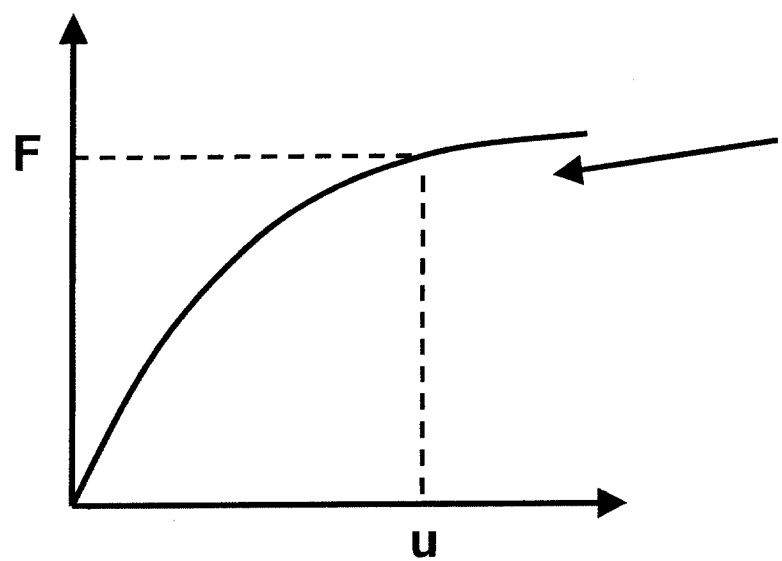
- Provide instruction on nonlinear solutions reinforced with hands-on workshops.
  - This training course is organized and presented by nonlinear topic.
  - The emphasis of the material presented is on the more commonly used program options. Where applicable the ANSYS documentation is referenced for program options not fully detailed in these notes.

Training Course Objectives

- Provide trouble shooting and guidance for issues of non-convergence.
  - Convergence difficulty is typically the biggest challenge of obtaining a nonlinear solution.
  - In all sections, recommended solution settings, trouble shooting, guidance, and modeling tips are presented.
  - In addition, a section on element selection is presented to offer guidance in selecting the appropriate element type for a nonlinear analysis.

Nonlinear Behavior

The fundamental characteristic of a nonlinear structure is a changing structural stiffness with changes in load. If you were to plot the load deflection curve for a nonlinear structure the relationship between force and deflection would be a nonlinear function.



The slope of the load deflection curve changes as the load is increased. In this case the structure "softens".

Nonlinear Behavior

Nonlinear structural behavior arises for a number of reasons, which can be grouped into three main categories:

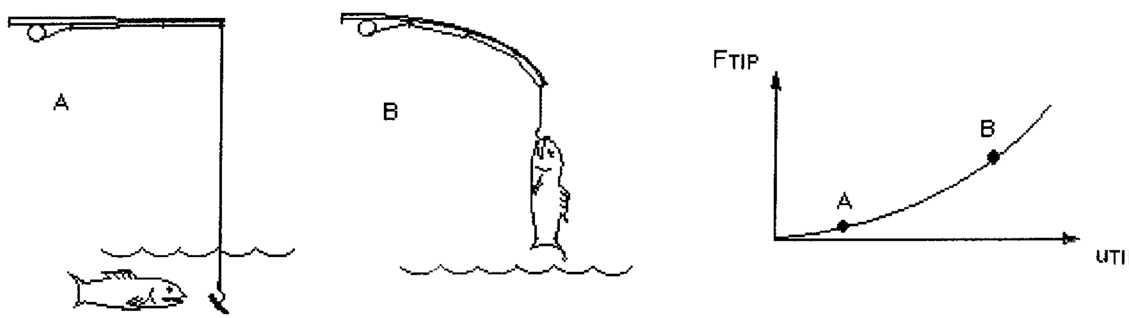
1. *Nonlinear Geometry*
   Large Strains, Large Deflections, Large Rotations

2. *Material Nonlinearities*
   Plasticity, Hyperelasticity, Viscoelasticity, Creep

3. *Changing Status Nonlinearities*
   Contact, Element Birth and Death

Nonlinear Geometry

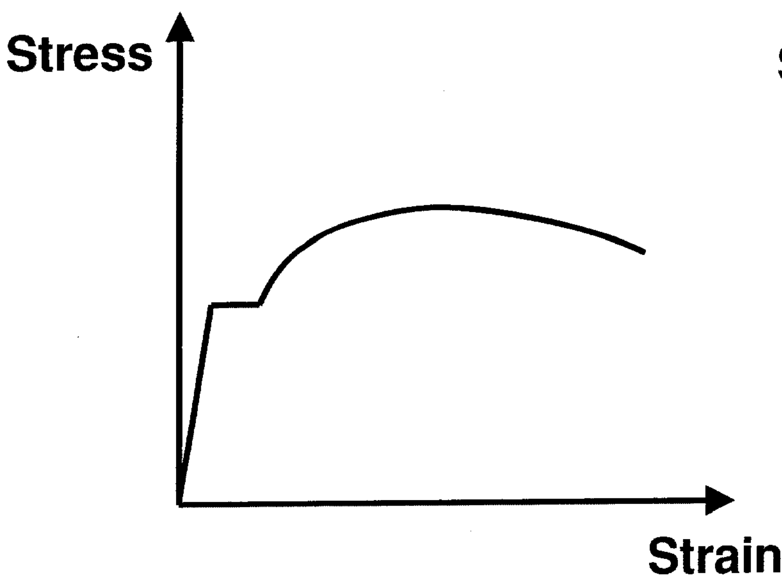
If a structure experiences large deformations, its changing geometric configuration can cause nonlinear behavior. Large deflections, large strains and large rotations are examples of nonlinear geometry.



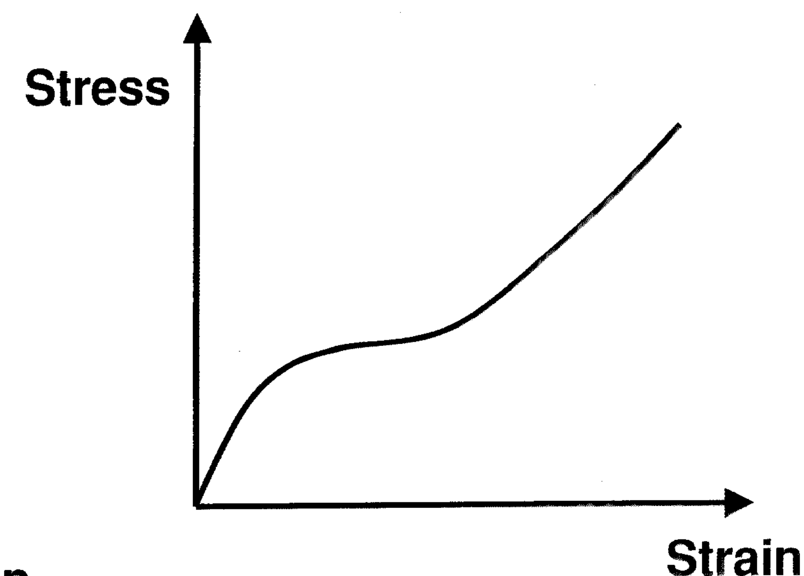
Under light lateral loads the rod tip is flexible, as the load increases the geometry of the rod changes (becomes curved) and the moment arm decreases (load moves), causing the rod to have a stiffening response.

Material Nonlinearities

A nonlinear stress-strain relationship is a common cause of nonlinear structural behavior.



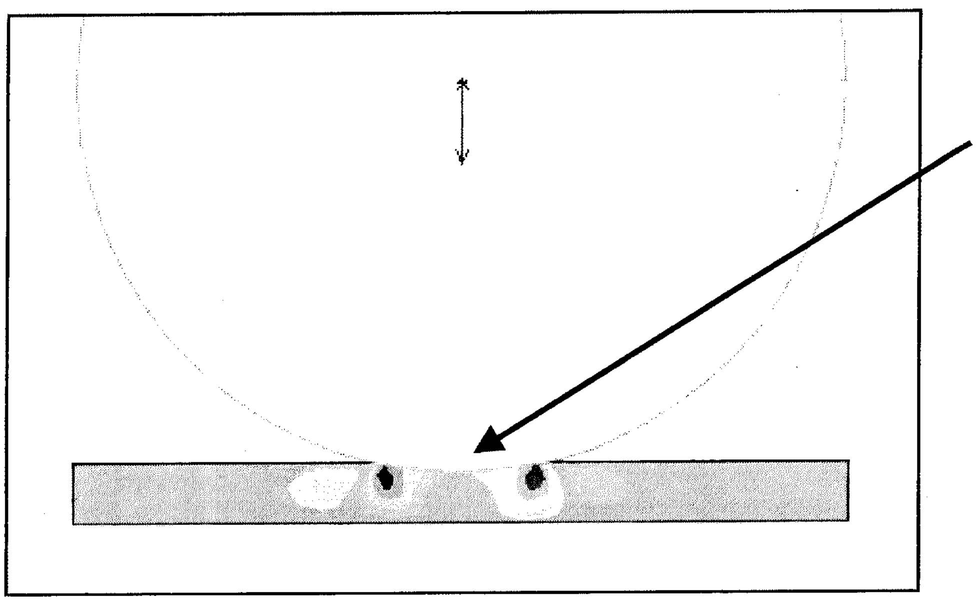
Steel



Rubber

Changing Status Nonlinearities

Many problems exhibit nonlinear behavior that is status-dependent. For example a cable can be either slack or taut. Two parts in an assembly may be in contact or out of contact.



In this Hertz contact example, the contact area is an unknown. It depends on the magnitude of the applied load.

Consequences of Nonlinear Analysis

- Potential path dependence of structural response, which means that the order in which the loads are applied may be significant.

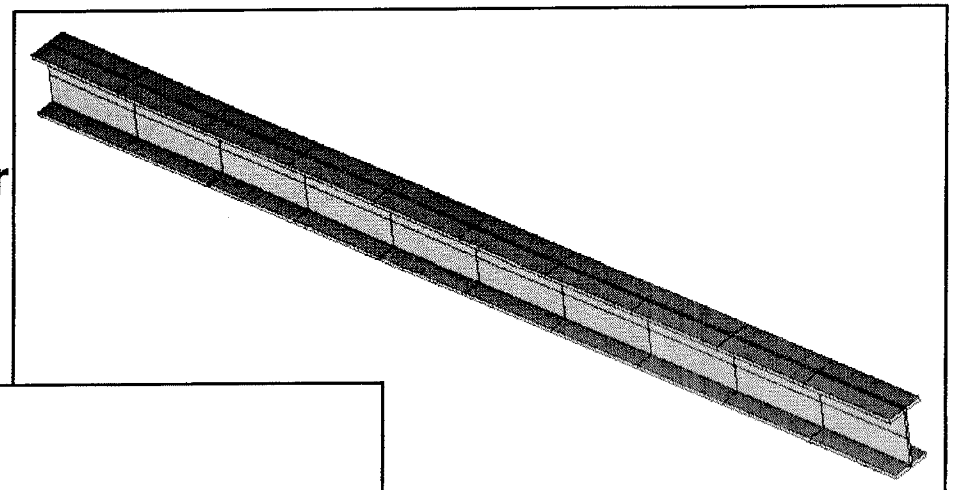
- The structural response is non-proportional to the applied loading. Therefore, the principle of superposition does not apply!

- Points of structural instability may be traversed along the load path.

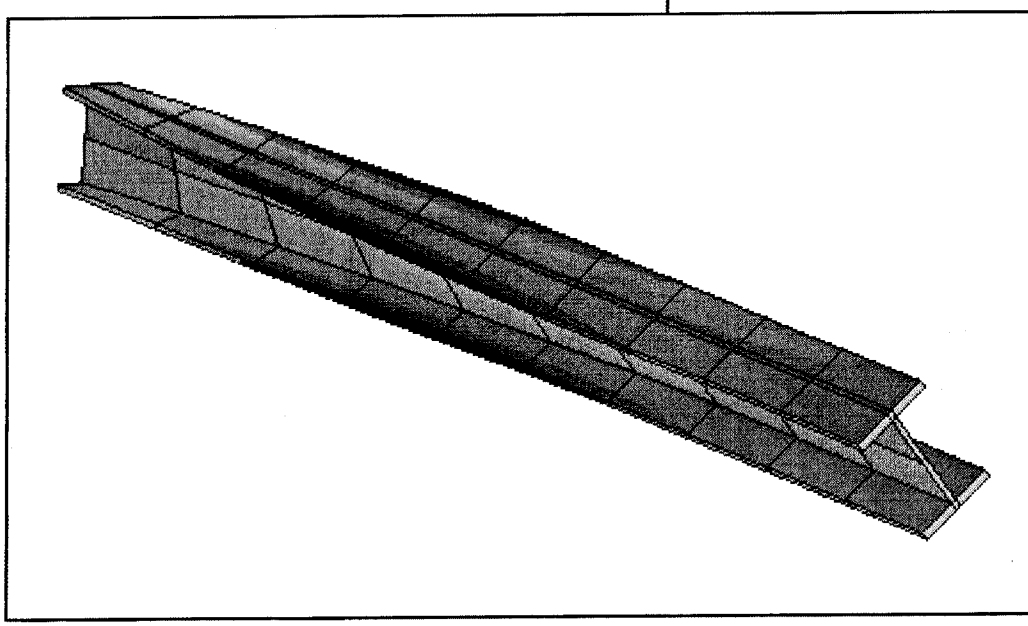
Applications of Nonlinear Analysis

- Some typical nonlinear applications include:
  - Nonlinear Buckling Analyses
  - Metal Forming Studies
  - Crash and Impact Analyses
  - Manufacturing Process Analyses (interference fits, assembly contact, etc.)
  - Analyses of Nonlinear Materials (elastomers, polymers)
  - Analyses of Systems Subjected to Ultimate Loads (plastic behavior and shakedown response)

Applications of Nonlinear Analysis



Lateral torsional buckling of a wide- flange cantilever beam.

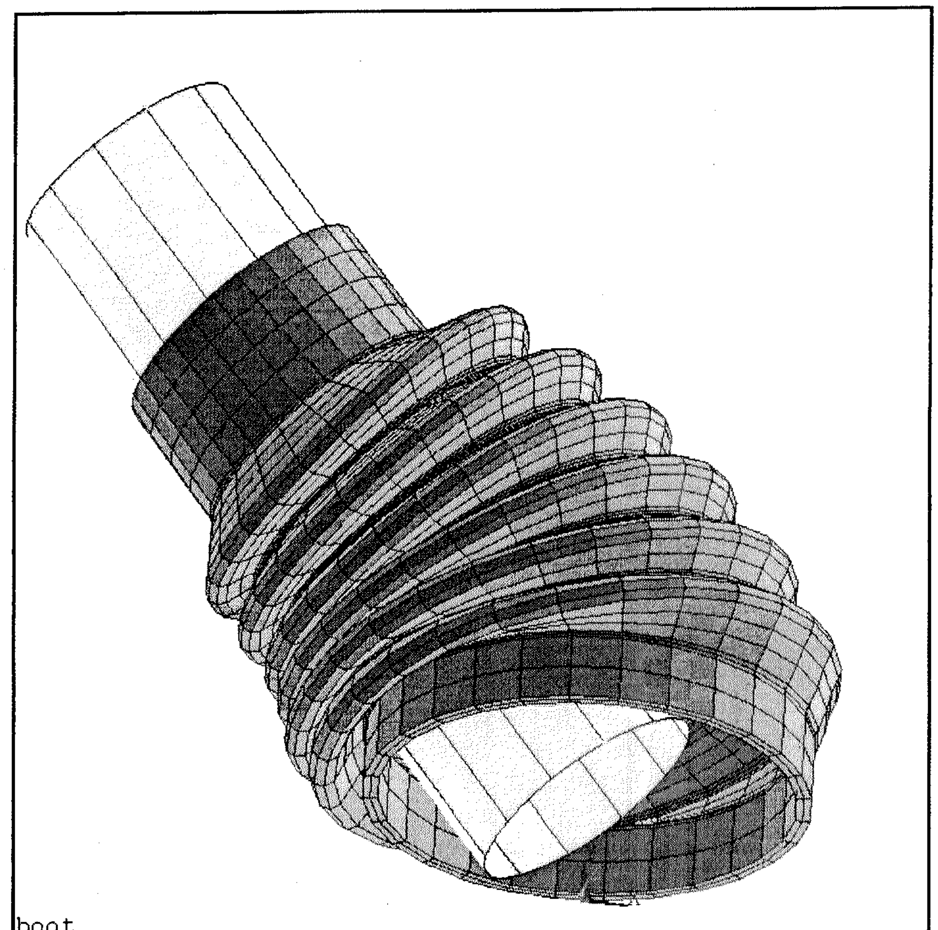


An example of a structural stability problem caused by *nonlinear geometry.*

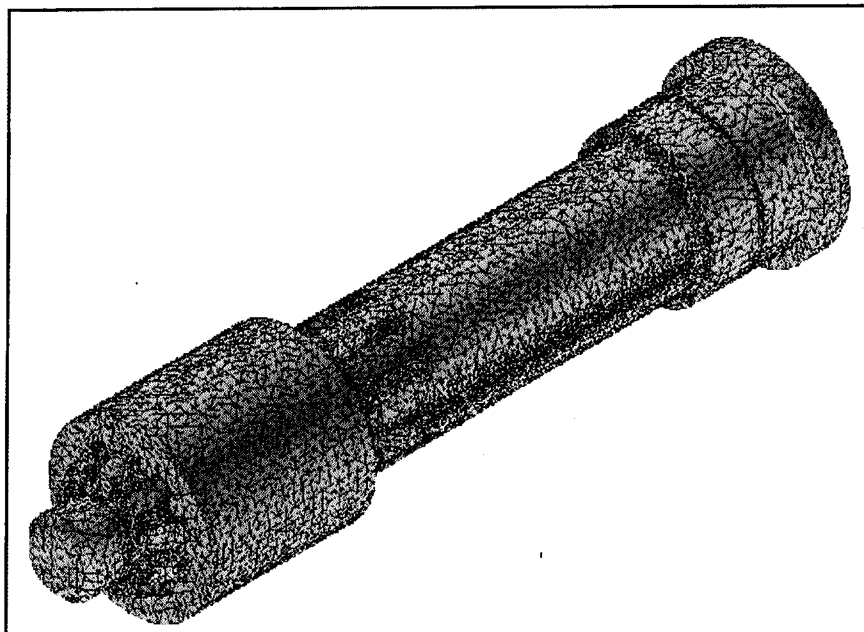
Applications of Nonlinear Analysis

Rubber Boot Seal

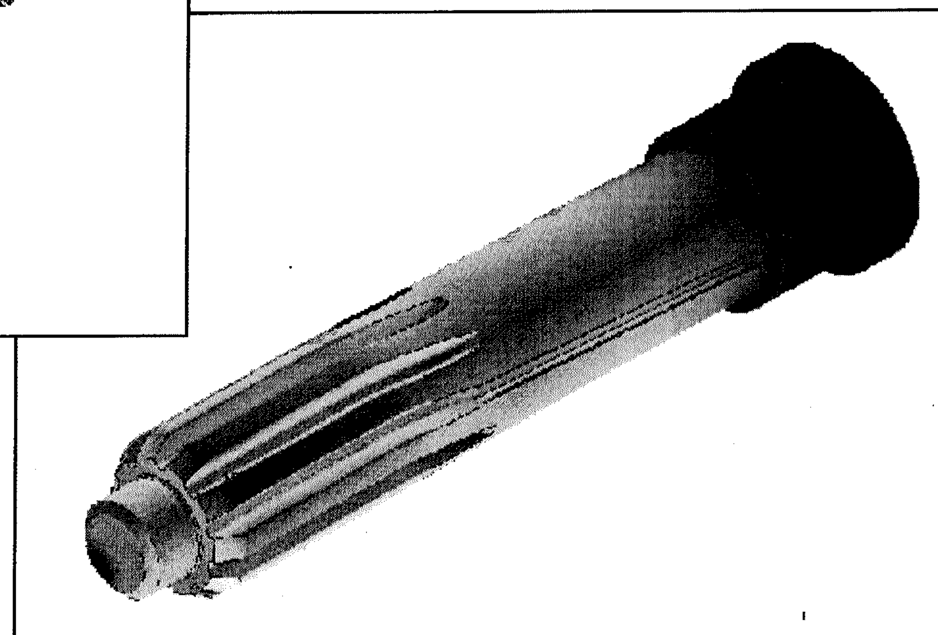
An example of *nonlinear geometry* (large strain and large deformation), *nonlinear material* (rubber), and *changing status nonlinearities* (contact).



Applications of Nonlinear Analysis



Splined shaft interference fit, example of a *contact or changing status* nonlinearity.



ANSYS Nonlinear Solution

- **The nonlinear features in the ANSYS program have been implemented to be:**

1. Easy to Use

– Many ease of use features have been implemented such as; automatic nonlinear solution control, and the Contact Wizard.

2. Robust

– Able to provide solutions to real world engineering problems for a broad variety of nonlinear applications.

3. Accurate

– Features such as contact with higher order elements, a variety of material models, and an extensive library of element formulations combine accurate results with a robust solution.

Chapter 2

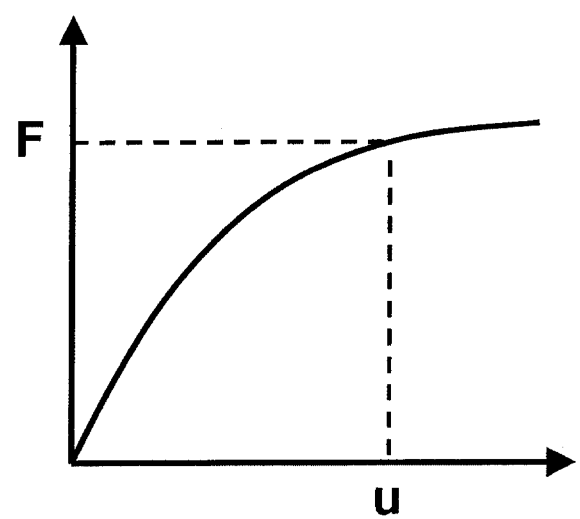
Nonlinear Solution

Session Objective

- At the end of this session you will be able to describe and demonstrate the following:
  1. Newton-Raphson Method
  2. Convergence
  3. Load Steps, Substeps, Equilibrium Iterations
  4. Automatic Time Stepping
  5. Output File Information
  6. Automatic Nonlinear Solution Control
  7. Nonlinear Solution Procedure
  8. Advanced Solution Controls
  9. Restarting an Analysis

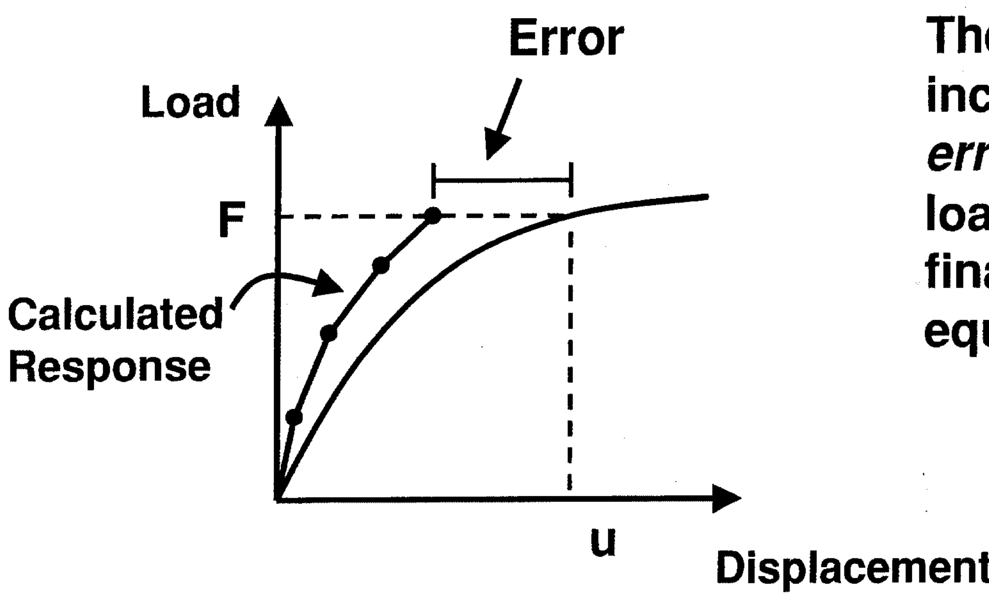
Nonlinear Solution

- Given a nonlinear load-displacement relationship, the objective is to find the response. In a nonlinear analysis the response can not be predicted directly with a set of linear equations. The solution is obtained by dividing the load into a number of *increments*, and determining the *equilibrium condition* for each of the increments.



Incremental Loading

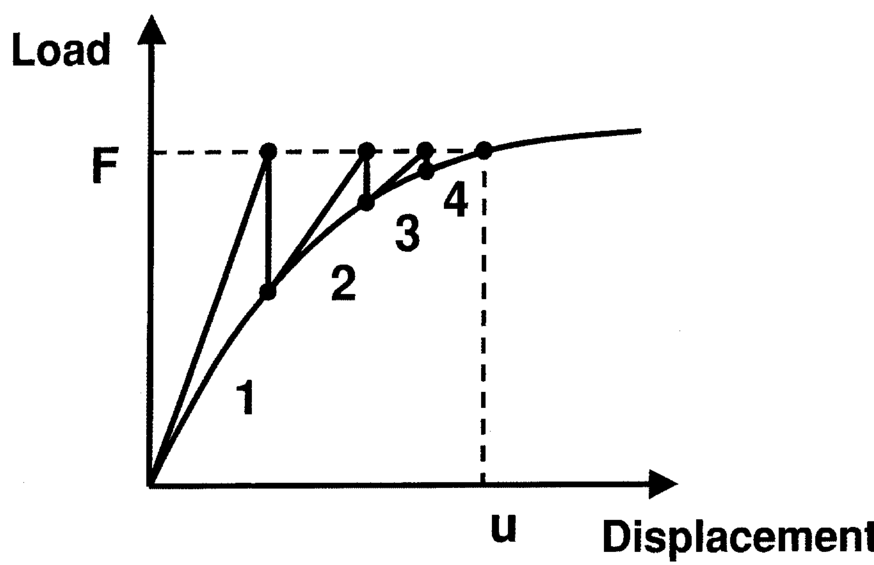
- One approach to nonlinear solutions is to divide the load into a series of *increments*. At the completion of each incremental solution, the stiffness matrix is adjusted to reflect the nonlinear response.



The problem with a purely incremental approach is that *errors accumulate* with each load increment, causing the final results to be out of equilibrium.

Newton-Raphson Method

- ANSYS overcomes the problem of an incremental solution by using *Newton-Raphson equilibrium iterations*. An equilibrium iteration attempts to drive the solution to equilibrium at the end of each load increment.



A full Newton-Raphson iterative solution for one increment of load. (Four iterations are shown.)

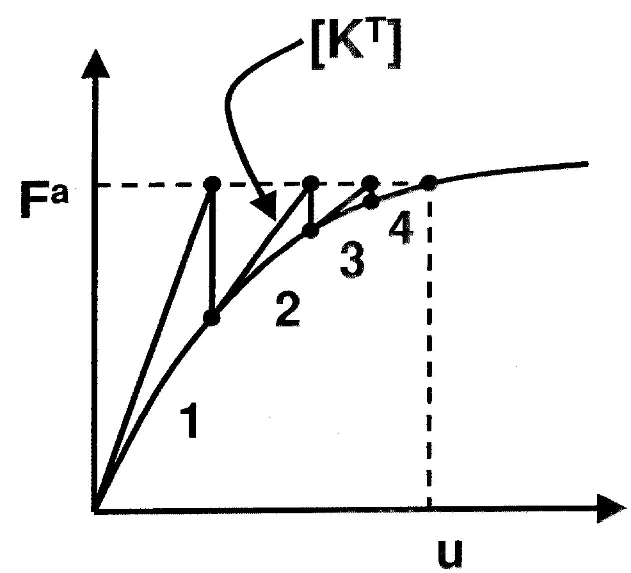
Newton-Raphson

The Newton-Raphson method iterates to a solution using the equation:

$$[K^T]\{\Delta u\} = \{F^a\} - \{F^{nr}\}$$

where:

- $[K^T]$ = Tangent Stiffness Matrix
- $\{\Delta u\}$ = Displacement Increment
- $\{F^a\}$ = Applied Load Vector
- $\{F^{nr}\}$ = Internal Force Vector



The goal is to iterate to *convergence* (defined later).

Newton-Raphson Method

- The Newton-Raphson method is a *numerical method* used by ANSYS to solve a nonlinear system of equations. The Newton-Raphson method is based on applying the load incrementally and iterating to enforce equilibrium at each load increment.

- The advantage of the Newton-Raphson method is that for a consistent tangent stiffness (more discussion on this topic later) the Newton-Raphson has a *quadratic rate of convergence*. This means that with each iteration the error in the solution is proportional to the square of the previous error.

What is Convergence?

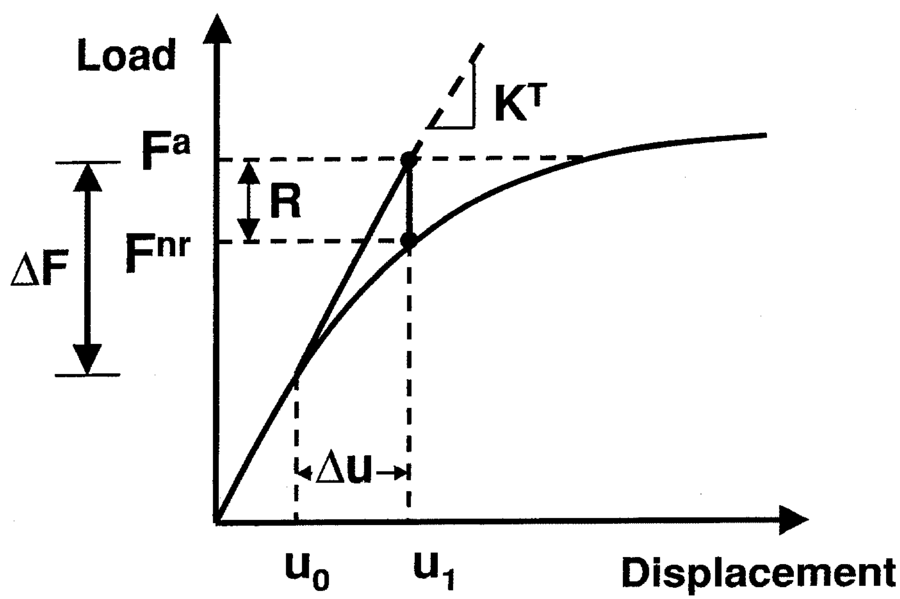
- The Newton-Raphson method requires a measure of *convergence* to decide when to stop iterating. Given the external loads on a body (F^a), and the internal forces (F^{nr}), (generated by the stresses in the elements acting at a node), the external forces must balance the internal forces for a body to be in equilibrium,

$$F^a - F^{nr} = 0$$

Convergence is a measure of equilibrium.

Convergence

- Consider the Newton-Raphson iteration shown below. Based on the configuration of the structure at u_0, the tangent stiffness is computed K^T, the displacement increment Δu is computed based on ΔF, and the configuration of the structure is updated to u_1.



The internal forces (element forces) are then computed in the updated configuration. The Newton-Raphson residual for the iteration is then:

$$R = F^a - F^{nr}$$

Convergence

- In practice the Newton-Raphson residual (F^a - F^{nr}) will never be exactly equal to zero. When the residual becomes small within a tolerance, the Newton-Raphson iterations are terminated and an equilibrium solution is obtained.

- Mathematically, convergence is attained when the norm of the residual loads defined as, $||\{F^a\} - \{F^{nr}\}||$, is less than some specified tolerance times a reference force value.

Convergence Criteria

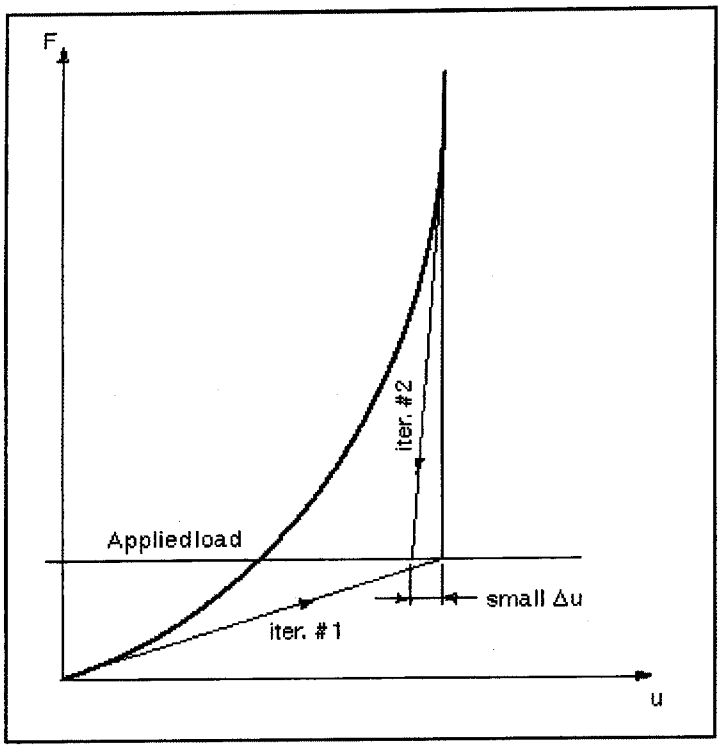
- ANSYS by default uses a force/moment and displacement/rotation convergence criteria for determining equilibrium convergence.

- A default tolerance of 0.5% for force/moment residuals and 5% for displacement/rotation increments is used.

- Experience suggests that these tolerances are accurate for most problems. The default settings are intended to be neither too “tight” nor too “loose” for a wide range of engineering problems.

Convergence

- **Force based convergence provides an absolute measure of convergence, as it is a measure of equilibrium between the internal and external forces.**

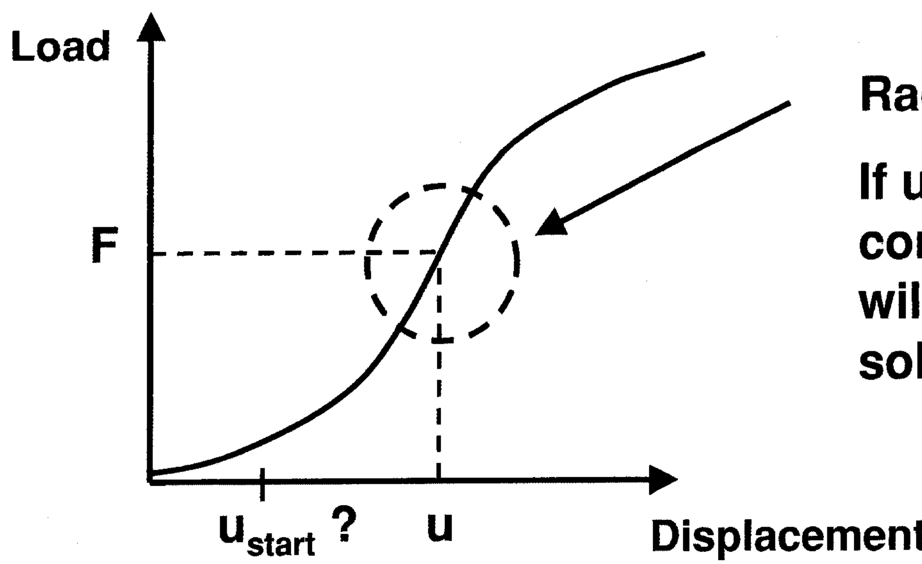


Displacement based checking should only be used as a *supplement* to force based convergence.

Relying on displacement convergence alone can in some cases lead to erroneous results.

Radius of Convergence

- Although the Newton-Raphson has a quadratic rate of convergence with a consistent tangent stiffness, it is not guaranteed to converge! The Newton-Raphson procedure is only guaranteed to converge if the starting configuration is inside the *radius of convergence*.

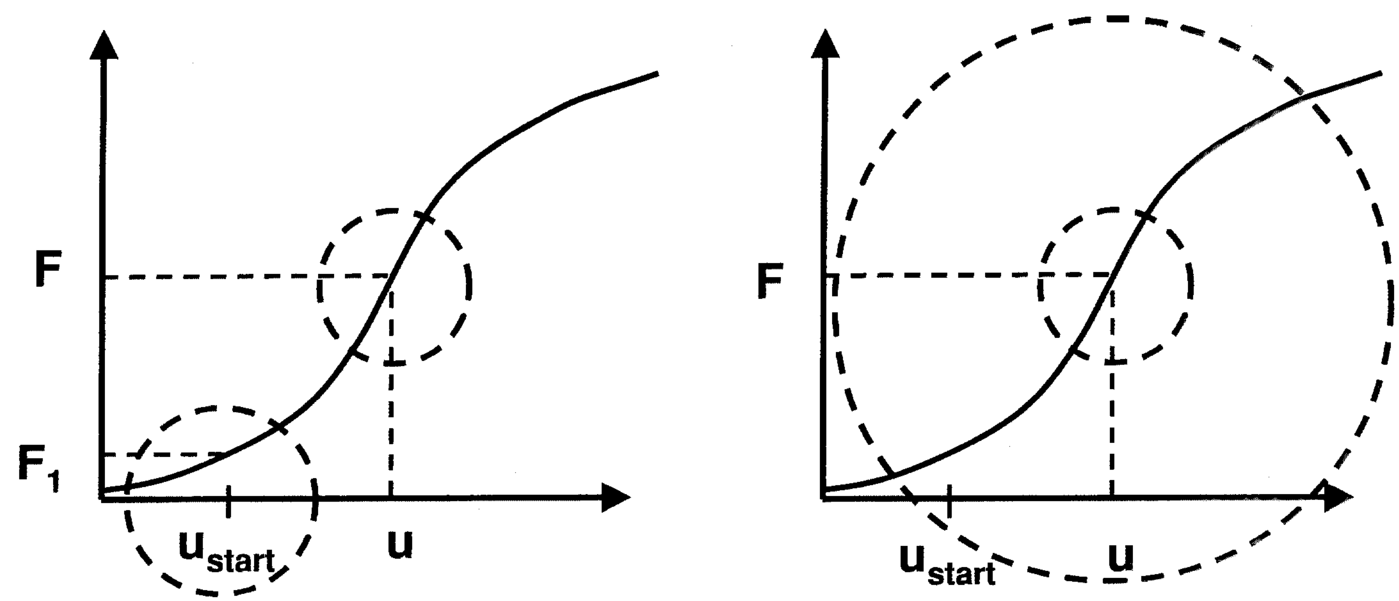


Radius of Convergence

If u_{start} is inside the radius of convergence the solution will converge, if outside the solution diverges.

Radius of Convergence

- There are a number of solution tools (which we will discuss) implemented in ANSYS to *both* apply load incrementally (to begin the solution inside the radius of convergence), and to enlarge the radius of convergence.



Apply Load Incrementally

Enlarge Radius of Convergence

Tangent Stiffness

- In order to achieve a quadratic rate of convergence the tangent stiffness matrix needs to be fully consistent. There are four components to the tangent stiffness matrix $[K^T]$:

$$[K^T] = [K^{inc}] + [K^u] + [K^\sigma] - [K^a]$$

where

$[K^{inc}]$ = Main Tangent Stiffness Matrix

$[K^u]$ = Initial Displacement Matrix

$[K^\sigma]$ = Initial Stress Matrix

$[K^a]$ = Initial Load Matrix

Tangent Stiffness

- The tangent stiffness matrix represents the slope of the load deflection curve in multi-dimensional space. (We will discuss the tangent stiffness matrix in more detail in the Geometric Nonlinearities section.)

$[K^{inc}]$ is the main tangent stiffness matrix.

$[K^{u}]$ accounts for the stiffness associated with the changes in element shape and position.

$[K^{\sigma}]$ accounts for the stiffness associated with the element stress state; it incorporates the effects of stress stiffening.

$[K^{a}]$ accounts for the stiffness associated with the change in orientation of the pressure loads due to deformation.

Load Steps and Substeps

- In ANSYS, the applied loading on a structure is described by defining a set of *load steps*.
- The load in a given *load step* is applied incrementally. Each increment of load is referred to as a *substep*.

Load Steps, Substeps, and Equilibrium Iterations

- A nonlinear solution can be organized into three levels:

Load Step

- A *load step* is the top level defined over "time", where solution options, loads and boundary conditions are applied.

Substeps

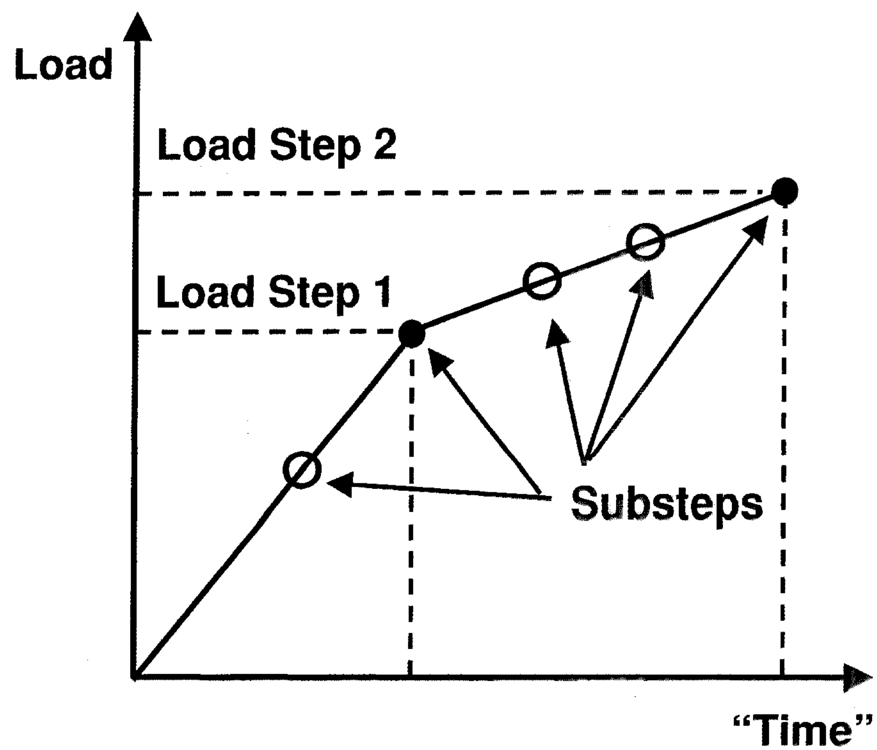
- A *substep* is an increment of load within a load step. Substeps are used to apply the load gradually.

Equilibrium Iterations

- An *equilibrium iteration* is an attempt by ANSYS to obtain a converged solution for the given substep (load increment).

Load Steps, Substeps, and Equilibrium Iterations

- Equilibrium iterations are performed for each increment of the load.
- Load step one has two substeps, and load step two has three substeps.
- *Each load step and substep is associated with a value of "time".*



Two Load Step Solution

"Time" in Nonlinear Solutions

- Every load step and substep is associated with a value of "time". *Substeps* can also be referred to as *time steps.*

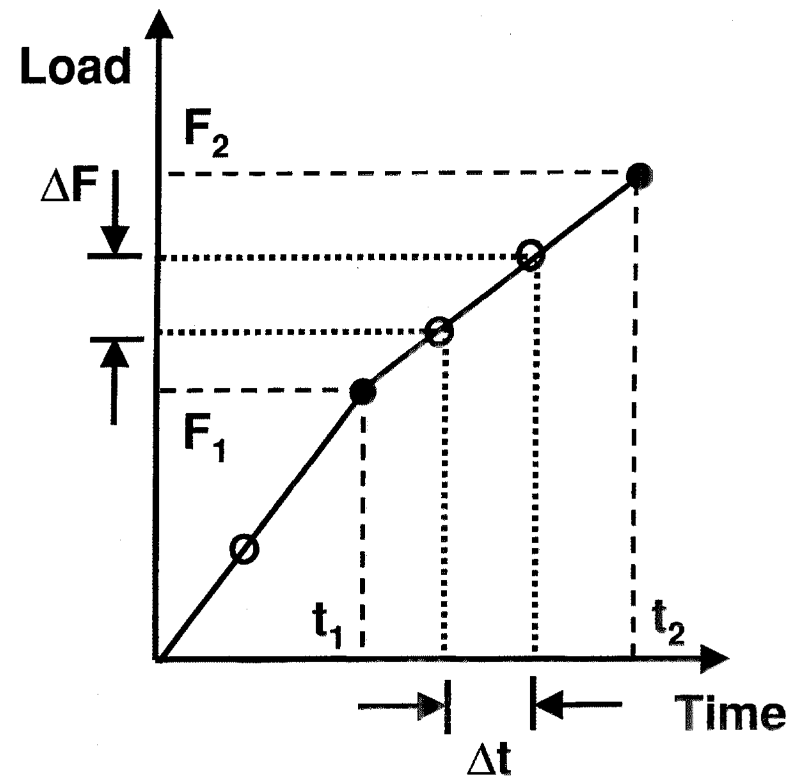
- "Time" represents the actual *temporal* scale in a rate-dependent analysis (creep, viscoplasticity) and in a transient analysis.

- For rate-independent, static analysis "time" is used as a counter. In a static analysis "time" can be set to any convenient value.

Modeling Tip: "Time" in a static analysis can be set equal to the magnitude of the applied load. This will make it easier to plot the load deflection curve.

Automatic Time Stepping

- The size of load increment (ΔF) over a substep is determined by the size of time step, Δt.
- The time step size can be specified by the user *or* automatically predicted and controlled by ANSYS.
- The *automatic time stepping* algorithm predicts and controls the time step size (load increment) for all substeps within a load step.



Automatic Time Stepping

- The automatic time stepping algorithm is one of the several algorithms imbedded in the Nonlinear Solution Controls. (More discussion on the Nonlinear Solution Controls later.)

- Based on the previous solution history and the physics of the problem, the automatic time stepping algorithm either increases (opens up) or decreases (bisects, cuts back) the time step size for a substep.

Output File Information

- During a nonlinear solution the output window contains the convergence information as the problem is being solved. The output window contains:
  - Force/Moment Residuals {R}

    `FORCE CONVERGENCE VALUE`
  - Maximum Degree of Freedom Increment $\{\Delta u\}$

    `MAX DOF INC`
  - Force Convergence Criteria

    `CRITERION`
  - Load Step and Substep Number

    `LOAD STEP      1    SUBSTEP     14`

Output File Information

- **The output window contains (continued):**
  - **Iteration Number for the Current Substep**

    ```
    EQUIL ITER 4 COMPLETED
    ```

  - **Cumulative Iterations**

    ```
    CUM ITER = 27
    ```

  - **Time Value and Time Step Size**

    ```
    TIME =    59.1250      TIME INC =    5.00000
    ```

  - **Automatic Time Stepping Information**

    ```
    AUTO STEP TIME:  NEXT TIME INC =  5.0000 UNCHANGED
    ```

Output File Information

```
*** LOAD STEP     1   SUBSTEP    14  COMPLETED.    CUM ITER =      27
*** TIME =     54.1250          TIME INC =   5.00000
*** MAX PLASTIC STRAIN STEP = 0.1512       CRITERION = 0.2500
*** AUTO STEP TIME:  NEXT TIME INC =  5.0000      UNCHANGED

    FORCE CONVERGENCE VALUE  =  349.2      CRITERION=  2.598
    DISP CONVERGENCE VALUE   = 0.1320      CRITERION= 0.9406     <<< CONVERGED
   EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.1645E-01
    FORCE CONVERGENCE VALUE  =  10.35      CRITERION=  2.095
    DISP CONVERGENCE VALUE   = 0.2409E-01  CRITERION= 0.9406     <<< CONVERGED
   EQUIL ITER   2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.1127E-01
    FORCE CONVERGENCE VALUE  =  4.687      CRITERION=  2.113
    DISP CONVERGENCE VALUE   = 0.1024E-01  CRITERION= 0.9406     <<< CONVERGED
   EQUIL ITER   3 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.3165E-02
    FORCE CONVERGENCE VALUE  =  2.179      CRITERION=  2.107
    DISP CONVERGENCE VALUE   = 0.5611E-02  CRITERION= 0.9406     <<< CONVERGED
   EQUIL ITER   4 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.1385E-02
    FORCE CONVERGENCE VALUE  = 0.9063      CRITERION=  2.108     <<< CONVERGED
   >>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION   4
*** LOAD STEP     1   SUBSTEP    15  COMPLETED.    CUM ITER =      31
*** TIME =     59.1250          TIME INC =   5.00000
*** MAX PLASTIC STRAIN STEP = 0.2136       CRITERION = 0.2500
*** AUTO STEP TIME:  NEXT TIME INC =  5.0000      UNCHANGED
```

Output File Information

- The information in the output file can be used for debugging the solution. The following list represents some typical questions regarding the solution progress:
  - Force/Moment Residuals - How is the solution converging? Are the residuals increasing, decreasing, oscillating?
  - Degree of Freedom Increment - Is the DOF increment becoming smaller, larger, oscillating?
  - Force Convergence Criteria - Is this value too large or too small within the context of your problem? How does it compare to the *Force Convergence Value?*

Output File Information

- <u>Load Step and Substep Number</u> - Where in the load history is the solution? How many substeps have been used for the current load step?

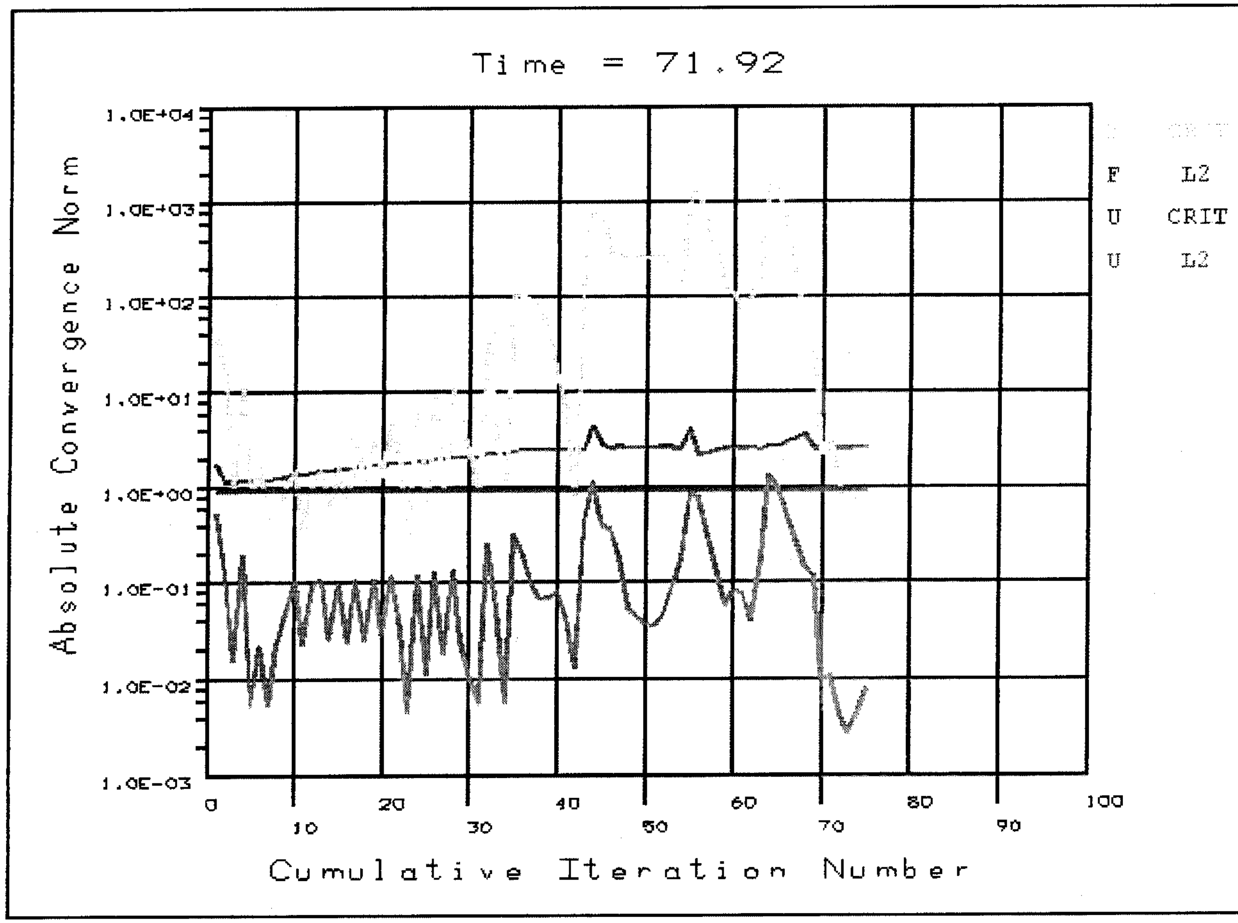
- <u>Iteration Number</u> - How many iterations per substep are being used? Is the load increment too large, too small?

- <u>Time Parameter</u> - At what point in the load history is the solution?

- <u>Automatic Time Stepping Information</u> - Is the time step being cut back or enlarged for the next substep? Is the convergence history smooth or difficult?

Graphical Convergence Information

Displayed in the graphics window is a graphical display of the convergence history. Time, iteration number, and residual information is displayed. The display is updated as the solution progresses.



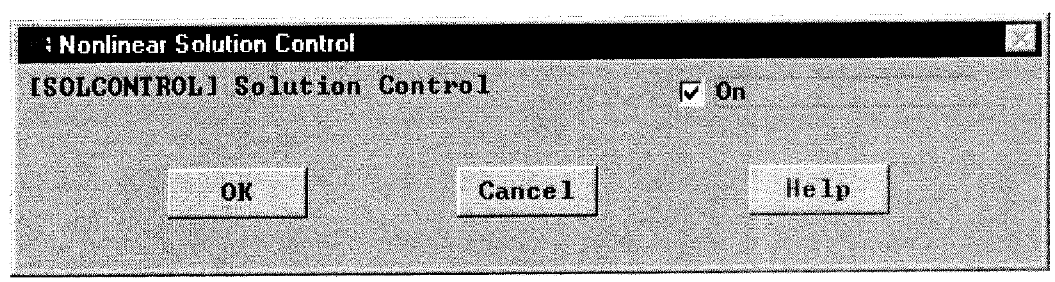
Automatic Nonlinear Solution Control

- For nonlinear analyses *automatic nonlinear solution control* is active by *default*. Solution control provides an optimal set of defaults and intelligent internal heuristics to provide:

  1. A robust and accurate nonlinear solution procedure with *minimal* user intervention.

  2. A solution procedure that will result in a savings in overall computing time (secondary goal to robustness).

Solution > Solution Ctrl ...



Automatic Nonlinear Solution Control

- The default settings activated by Solution Control are dependent on the physics of the problem. Solution Control is applicable to both static and dynamic nonlinear structural and nonlinear heat transfer analyses.

- The remainder of this section will focus on the nonlinear solution controls and the procedure for performing a nonlinear solution. The *advanced controls* are used to override the Solution Control settings.

- *The defaults outlined in the remainder of this section assume that the Solution Control is turned on!*

Nonlinear Solution Procedure

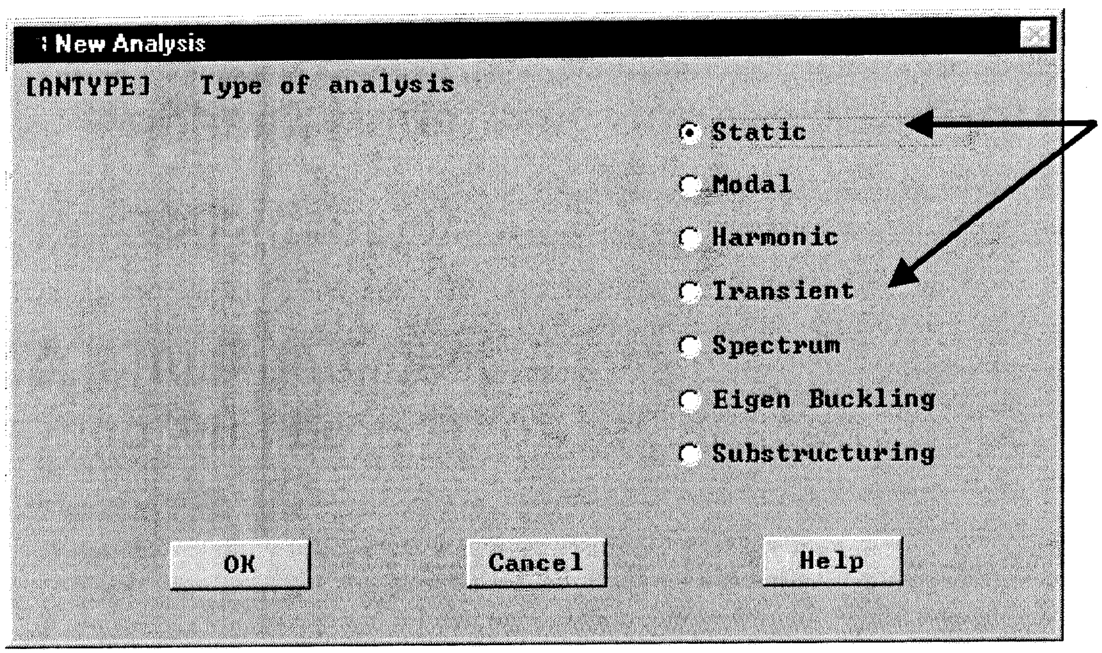
The following procedure outlines the typical steps necessary to perform a nonlinear analysis:

1. Specify the Analysis Type
2. Specify Nonlinear Geometry On or Off
3. Specify "Time" for the Load Step
4. Set the Number of Substeps by NSUBST *or* DELTIM
5. Apply Loads and Boundary Conditions
6. Specify Output Controls and Monitor Variables
7. Save the Database
8. Solve the Load Step

Analysis Type

The analysis type defines whether the analysis is static or dynamic. *Note that the analysis type can not be changed after the first load step.*

Solution > New Analysis ...



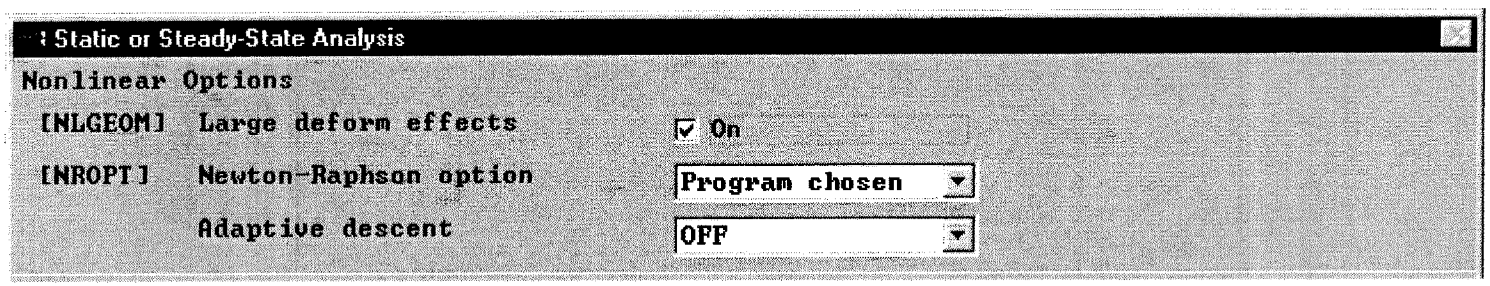
The only two valid choices for a nonlinear analysis are *Static* or *Transient*.

The default is Static.

Nonlinear Geometry

Activating nonlinear geometry will include nonlinear geometric effects in the analysis; large strains, large deflections, and large rotations. Geometric nonlinearities will be covered in the next section. The default is off.

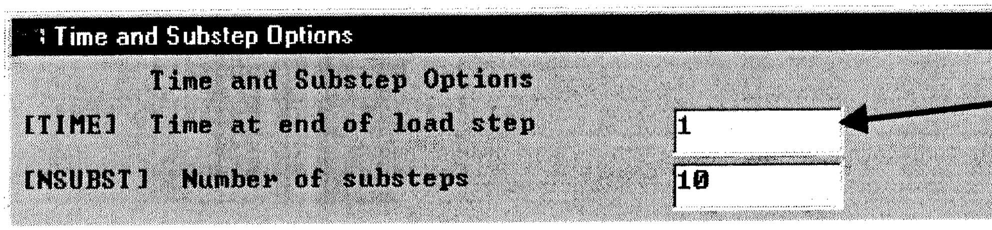
Solution > Analysis Options ...



If you are not sure if nonlinear geometric effects are important, activate NLGEOM to be safe.

Load Step Time

Solution > Time/Freq > Time and Substps ...



Time

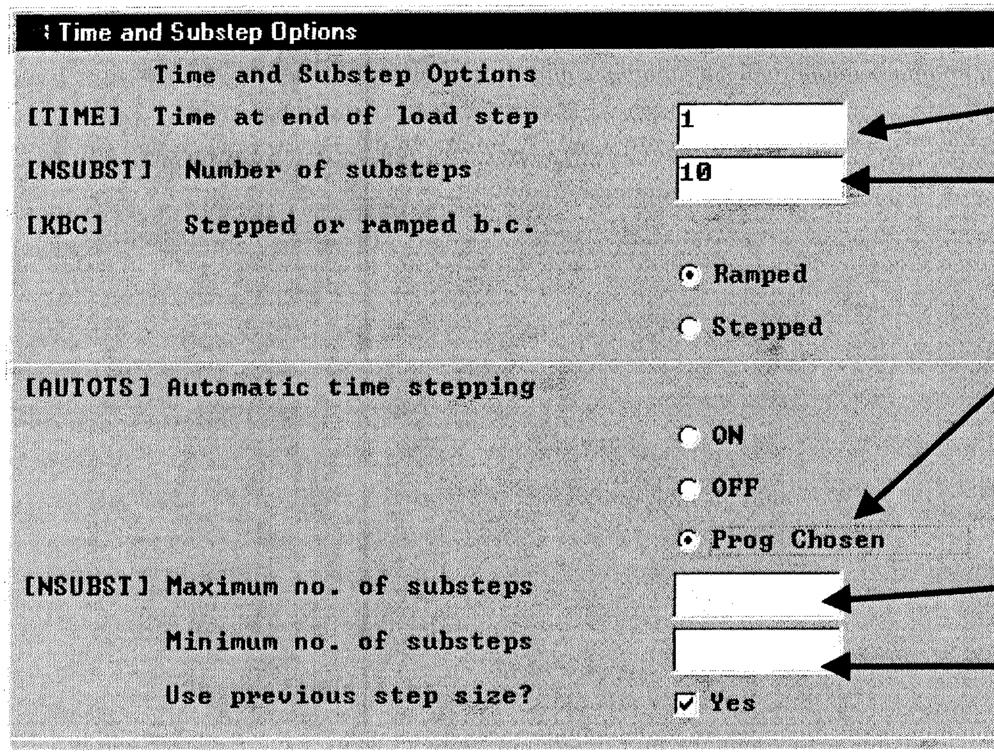
- **If "time" is not specified it will default to TIME + 1.0. Where TIME is the value at end of previous load step.**

- **The default "time" for the first load step is 1.0.**

- ***For a static, rate-independent analysis "time" can be any convenient value. For proportional loading it can be set equal to the value of the load at the end of load step.***

Time and Substeps

Solution > Time/Freq > Time and Substps ...



Time

Initial Number of Substeps

Automatically turned on by Solution Control.

Max. Number of Substeps

Min. Number of Substeps

Number of Substeps

- The number of substeps (N) specifies the incremental load ($\Delta F_{initial}$) over the first substep of a load step through the initial time step size ($\Delta t_{initial}$) as

$$\Delta F_{initial} = (F_{end} - F_{begin}) / N$$

- It is *strongly recommended* that the user specify the number of substeps for the load step.

- If number of substeps is not specified, ANSYS will select a default for the number of substeps, and warn the user.

Number of Substeps

- The *maximum* number of substeps (N_{max}) specifies the minimum load increment over a substep through the mimimum time step size as

$$\Delta F_{min} = (F_{end} - F_{begin})/N_{max}$$

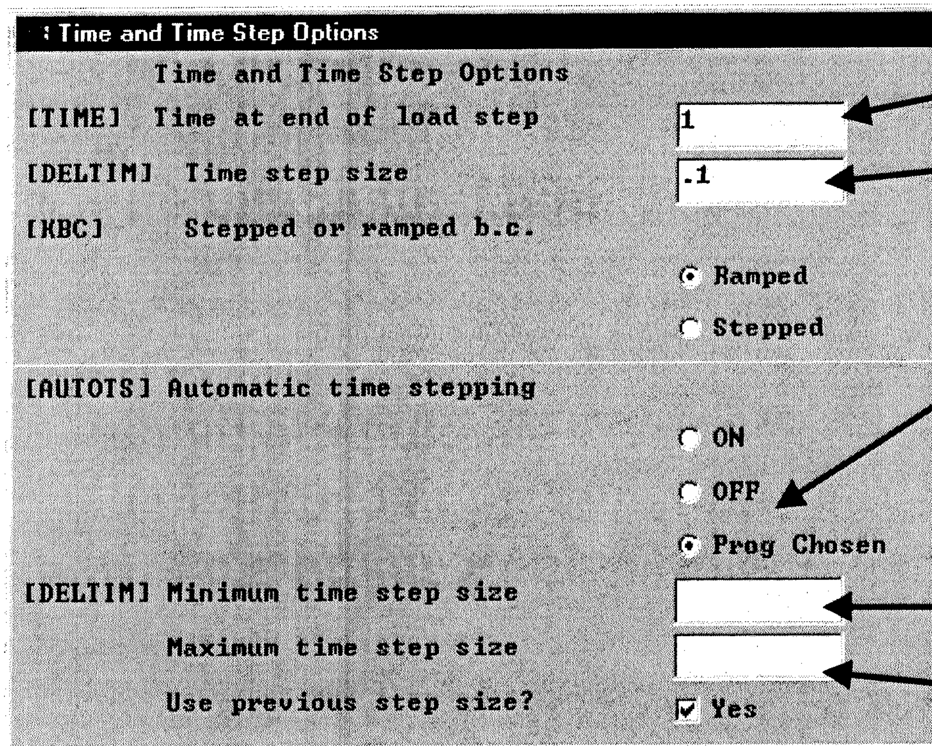
- The *minimum* number of substeps (N_{min}) specifies the maximum load increment over a substep through the maximum time step size as

$$\Delta F_{max} = (F_{end} - F_{begin})/N_{min}$$

- ***The minimum and maximum time step sizes defined through N_{max} and N_{min} influence the time step size increases and decreases in the automatic time stepping algorithm.***

Or Time and Time Steps

Solution > Time/Freq > Time - TimeStps ...



Time

Initial Time Step

Automatically turned on by Solution Control.

Minimum Time Step

Maximum Time Step

Or Time Step Size

- Setting the *time step sizes* is analogous to specifying the *number of substeps*. The initial time increment ($\Delta t_{initial}$) specifies the incremental load ($\Delta F_{initial}$) over the first substep of the load step as

$$\Delta F_{initial} = (F_{end} - F_{begin})*\Delta t_{initial} / (T_{end} - T_{begin})$$

- The *minimum* time step (Δt_{min}) specifies ΔF_{min} as

$$\Delta F_{min} = (F_{end} - F_{begin})*\Delta t_{min} / (T_{end} - T_{begin})$$

- The *maximum* time step (Δt_{max}) specifies ΔF_{max} as

$$\Delta F_{max} = (F_{end} - F_{begin})*\Delta t_{max} / (T_{end} - T_{begin})$$

Number of Substeps *or* Time Step Sizes

- Specifying the number of substeps *or* specifying the time step sizes is a matter of user preference. Both procedures specify the initial, maximum and minimum time step size (load increment) for the load step.

$$\Delta t_{initial} = (T_{end} - T_{begin})/N$$

$$\Delta t_{min} = (T_{end} - T_{begin})/N_{max}$$

$$\Delta t_{max} = (T_{end} - T_{begin})/N_{min}$$

- The number of substeps *or* the initial time step size is an important parameter relating to solution robustness and efficiency. *It is strongly recommended to specify this parameter.*

- Although ANSYS has built in defaults for the number of substeps *or* the time step sizes, the default settings are *arbitrary*.

Number of Substeps *or* Time Step Sizes

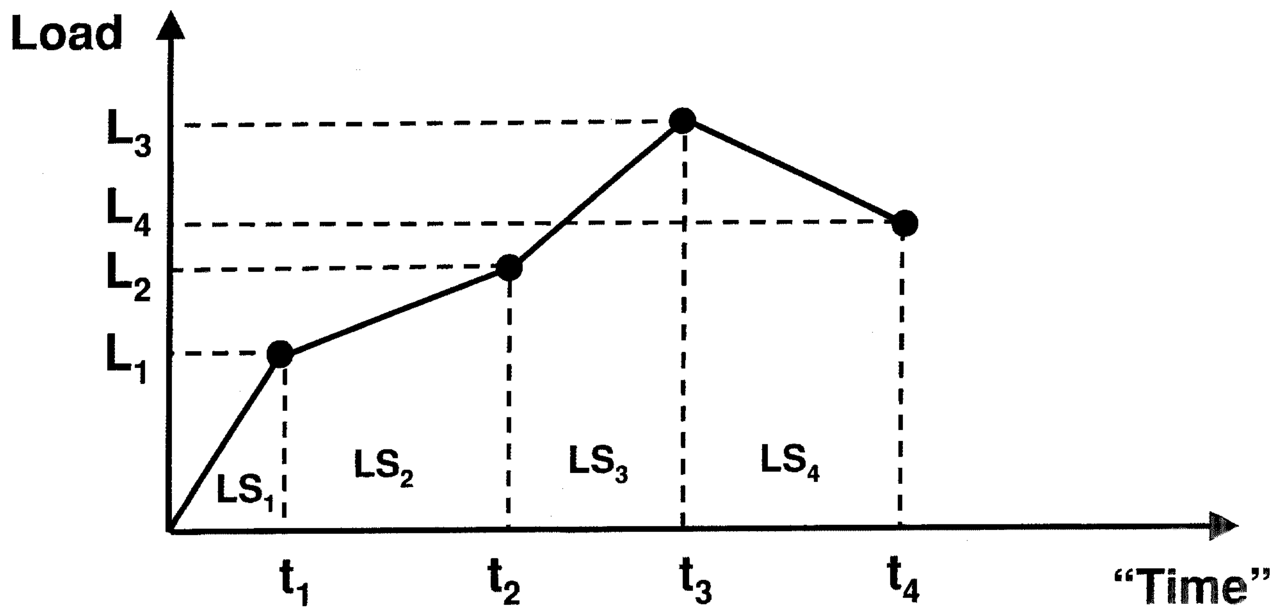
- If the convergence behavior is smooth, the automatic time stepping will increase the load increment. If convergence is difficult, the automatic time stepping will *bisect or cutback* the load increment.

- The goal is to specify an optimal number of maximum and minimum substeps, *or* optimal small and large time step sizes to allow the automatic time stepping algorithm to increase and/or decrease the load increment as required, based on the solution history.

Time Step Bisection

- Time step *bisection* occurs when the program decides that convergence can not be achieved within the current substep. Time step *bisection* provides a method of automatically recovering from convergence failure.
- Upon bisection the current substep is discarded, the time step size is cut in half, and the program automatically restarts the solution. If required, the solution can bisect repeatedly within a given substep until convergence is achieved.
- Repeated bisection leads to a smaller and smaller time step size. If the time step size is smaller than the minimum time step the solution stops. (This may be an indication of an instability or other phenomenon.)

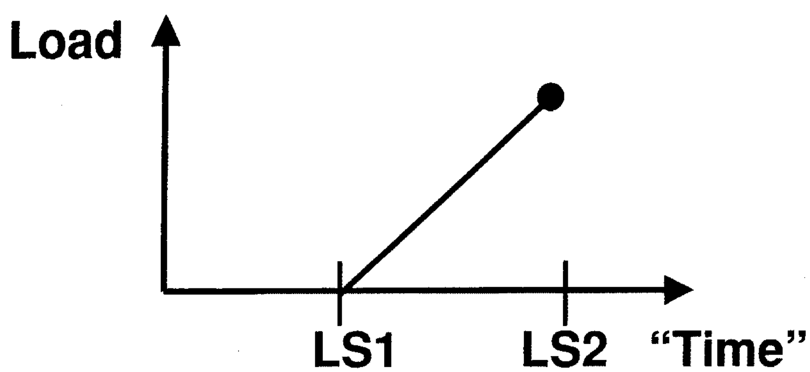
Defining Loads



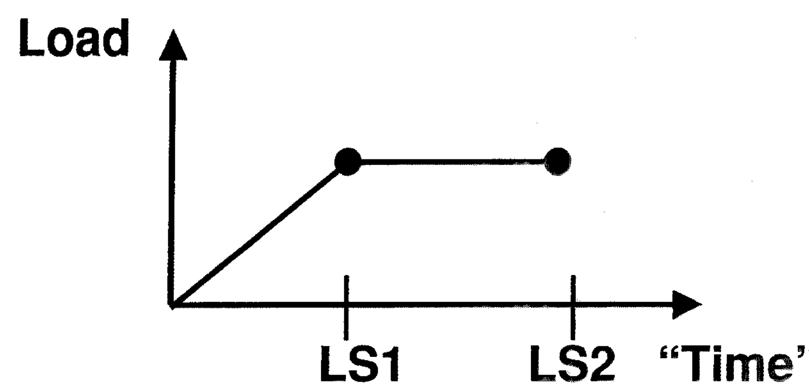
ANSYS linearly interpolates loads for all substeps within a load step. For simple time varying loads you can use multiple load steps to define the load history.

Defining Loads

If you are running an analysis with multiple load steps, there are a few considerations when applying and changing loads from one load step to the next.

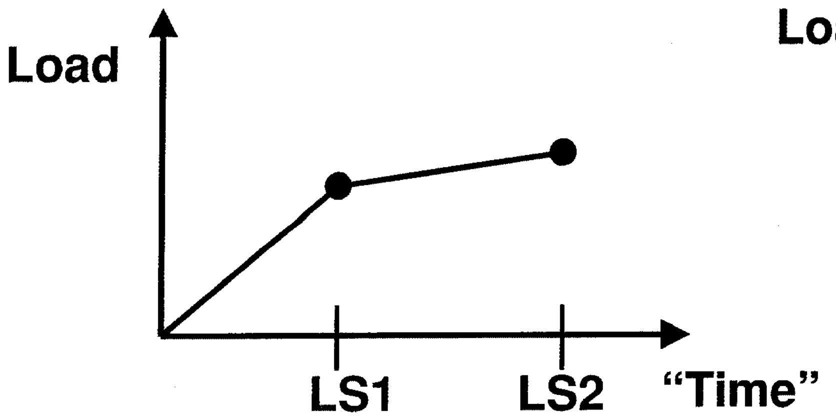


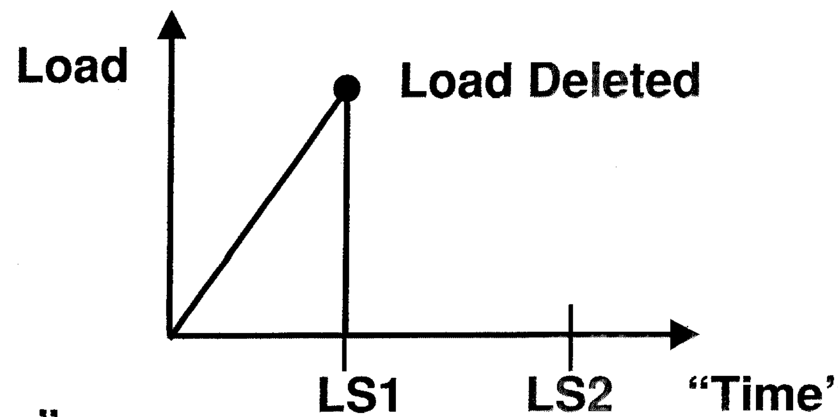
Newly applied loads are ramped from zero at the start of the load step to their full value at the end of the load step.



Loads which are unchanged will retain their values for the next load step.

Defining Loads





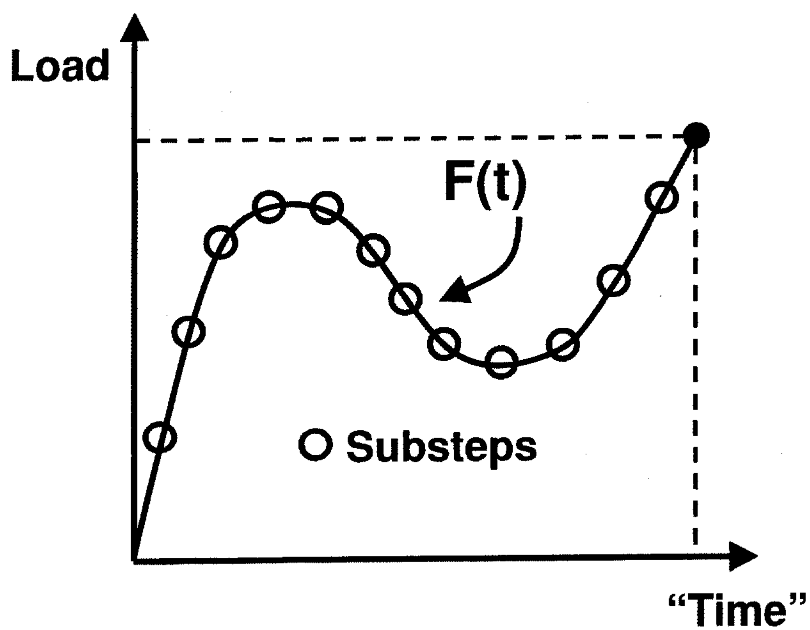
When a load is redefined, its value is ramped from the value it had at the end of the previous load step.

When a load is deleted the effect is a step change to zero. *This is generally not advised; better modeling practice is to ramp the load to zero over a small time increment.*

Defining Loads

- For complex time varying loads a load curve can be defined using an APDL table array parameter.

- Refer to the *ANSYS Basic Analysis Procedures Guide* for more detail on using load curves.



The problem is solved as one load step with several substeps. The load is linearly interpolated from substep to substep. The load increment sizes can be defined directly or specified by the automatic time stepping algorithm.

ANSYS

Load Definition Issues

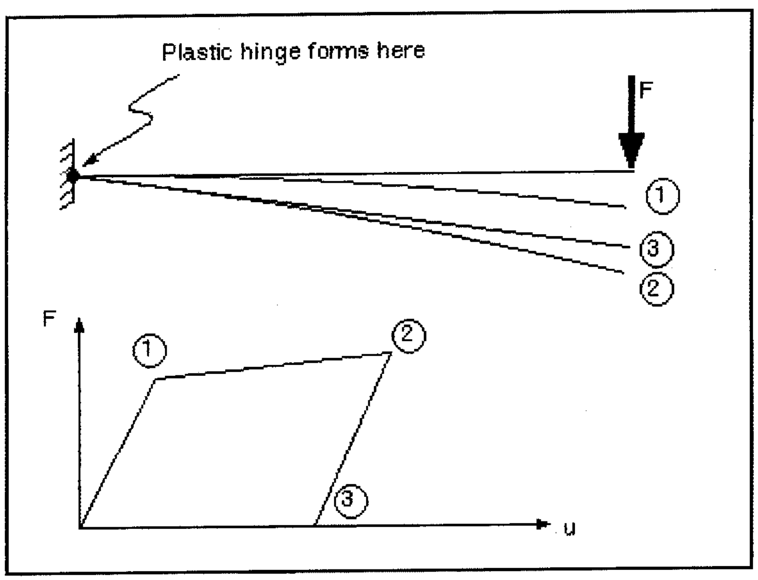
- If all energy put into a system is recovered when the loads are removed, the system is *conservative*. If energy is dissipated by the system (such as plastic deformation or sliding friction), the system is *nonconservative.*

- An analysis of a conservative system is path independent; the loads can be applied in any order.

- An analysis of a nonconservative system is path dependent; *the actual load history must be followed.* Path dependent problems also require that the *load be applied slowly* (using many substeps). *Superposition does not hold* for a path dependent problem.

Load Definition Issues

Shown below is a beam with a plastic hinge, an example of a nonconservative or path dependent system.

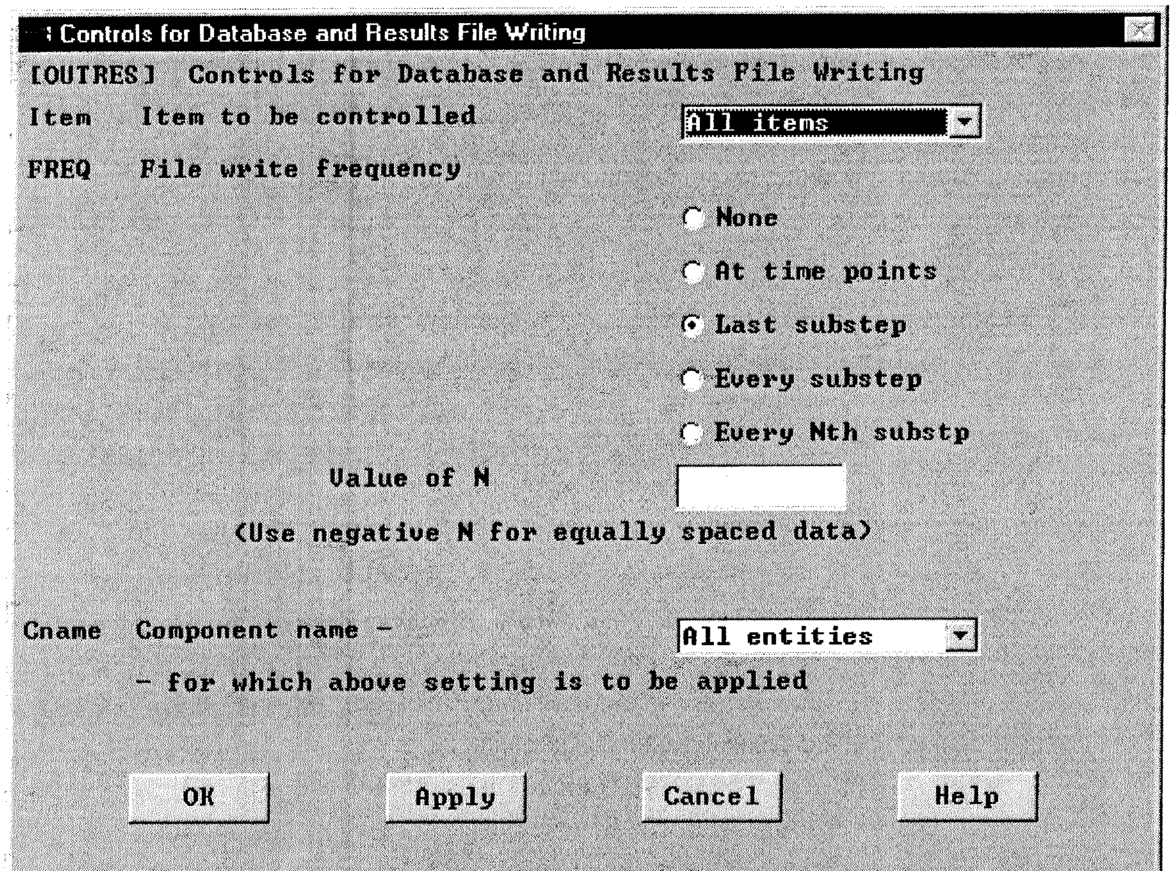


1→ 2→ 3 is *not* the same as 1→ 3.

When solving a path dependent problem you should verify that an adequate number of *load increments* (many substeps) is used to solve the problem. Additionally, an accurate load history *must* be used.

Output Controls

Solution > Output Ctrls > DB/Results File ...



This option controls what is written to the results file (jobname.rst) and how often the data is written.

Note that by default, only the last substep of the load step is written to the results file.

Monitor File

The monitor file (jobname.mntr) is written during a nonlinear solution to provide a compact summary to review the solution convergence history.

LOAD STEP	SUB-STEP	NO. ATTMP	NO. ITER	TOTL ITER	INCREMENT TIME/LFACT	TOTAL TIME/LFACT	VARIAB 1 MONITOR FY	VARIAB 2 MONITOR MxDs	VARIAB 3 MONITOR MxPl
1	1	1	3	3	1.0000	1.0000	-222.25	-.900E-01	0.44557E-02
1	2	1	2	5	1.0000	2.0000	-225.39	-.18000	0.12632E-01
1	3	1	2	7	1.5000	3.5000	-229.30	-.31500	0.18422E-01
1	4	1	1	8	2.2500	5.7500	-234.55	-.51750	0.27567E-01
1	5	1	1	9	3.3750	9.1250	-242.38	-.82125	0.41447E-01
1	6	1	2	11	5.0000	14.125	-255.10	-1.2712	0.61834E-01
1	7	1	1	12	5.0000	19.125	-266.66	-1.7212	0.65103E-01
1	8	1	2	14	5.0000	24.125	-281.02	-2.1713	0.67789E-01
1	9	1	2	16	5.0000	29.125	-295.08	-2.6212	0.72824E-01
1	10	1	2	18	5.0000	34.125	-310.10	-3.0713	0.78528E-01
1	11	1	2	20	5.0000	39.125	-326.12	-3.5212	0.86029E-01
1	12	1	3	23	5.0000	44.125	-340.46	-3.9712	0.10108
1	13	1	2	25	5.0000	49.125	-356.76	-4.4213	0.12270

Monitor File

- The monitor file contains information for each substep of a load step which includes; the number of attempts (bisections) which occurred, the number of iterations used, the load increment, CPU time, maximum displacement, and maximum equivalent plastic strain.

- The information contained in the monitor file can be useful for debugging an analysis.

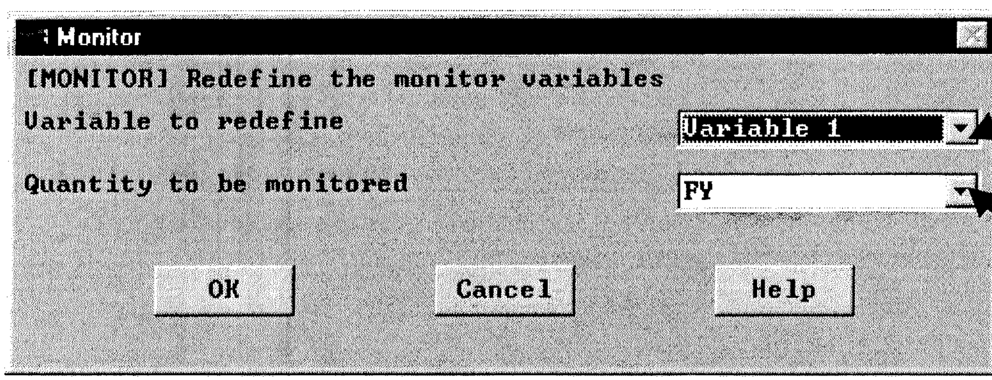
Monitor File - Guidelines

- If multiple attempts are made to solve the first substep of a load step, this implies that the initial time step size is too large. Specify a larger number of substeps *or* a smaller initial time step size.
- If a large number of attempts are made for any substep, you may want to increase the maximum number of substeps *or* decrease the minimum time step size for the load step.
- You may also want to abort the solution based on the monitor variables reaching a particular value. (For example, displacement at a node exceeds a certain value.)

Monitor File

The last three columns of the monitor file, variables 1, 2, and 3, default to CPU time, maximum displacement, and maximum plastic strain. You can redefine variables 1, 2, and 3 to monitor nodal displacements or reaction forces.

Solution > Nonlinear > Monitor ...



Variables to be redefined (1, 2, and/or 3).

Quantity to be monitored.

Save and Solve

- Before executing a nonlinear solution, it is good modeling practice to save the database with the load information. If you need to restart the solution you *will* need a copy of the database with the loads defined for the load step you wish to restart. (More information on solution restarts later.

- Solve the current load step.

Advanced Solution Controls

- The following *advanced solution controls* represent the set of controls which may be used to redefine the defaults activated by Solution Control:
  - Equation Solver
  - Ramped or Stepped Loading
  - Time Step Prediction
  - Deactivate Automatic Time Stepping
  - Number of Equilibrium Iterations
  - Convergence Criteria

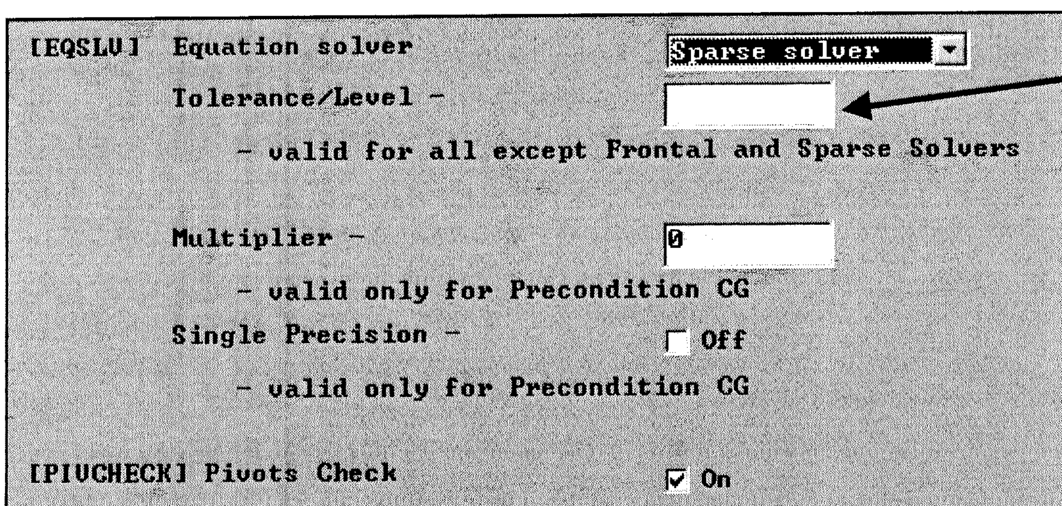
Advanced Solution Controls

- The following set of *advanced solution controls* represent the set of controls which are less frequently used to reset the defaults activated by Solution Control:
  - Newton-Raphson Options
  - Line Search
  - Predictor
  - Adaptive Descent
  - Deactivate Stress Stiffening
  - Cutback Control
  - Time Integration Effects
  - Solution Stopping Controls

Equation Solver

- For a nonlinear analysis, the three most common choices for an equation solver are:
  - Sparse (direct solver, default with Solution Control)
  - Frontal (direct solver)
  - Preconditioned Conjugate Gradient (iterative)

Solution > Analysis Options ...



Tolerance for PCG solver, default is 1e-8. This value is not applicable for the frontal or sparse solver.

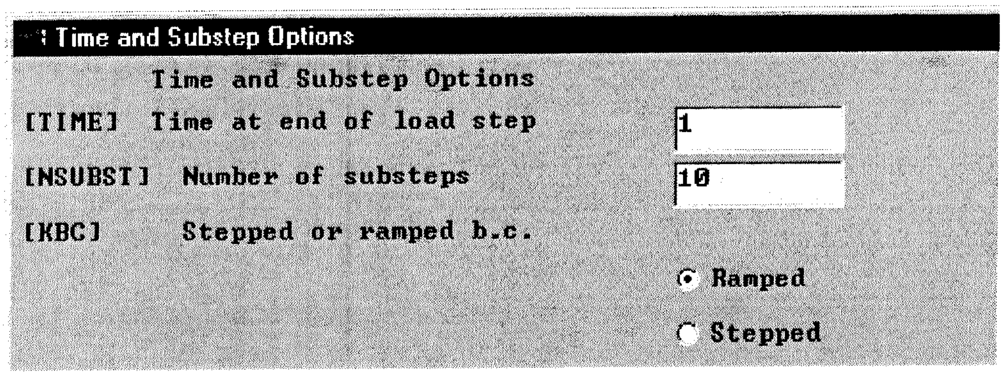
Equation Solver Guidelines

- If the model is a beam/shell model, or a beam/shell/solid model use the *sparse solver*.
- If the model is a 3D solid model (Solid92 or Solid45), and the number of degrees of freedom is relatively large (>100,000) use the *PCG*.
- If the problem is ill-conditioned, or has a wide range of element stiffnesses (contained in output file), use the *sparse solver*.
- If unsymmetric matrices are present use the *sparse solver*.

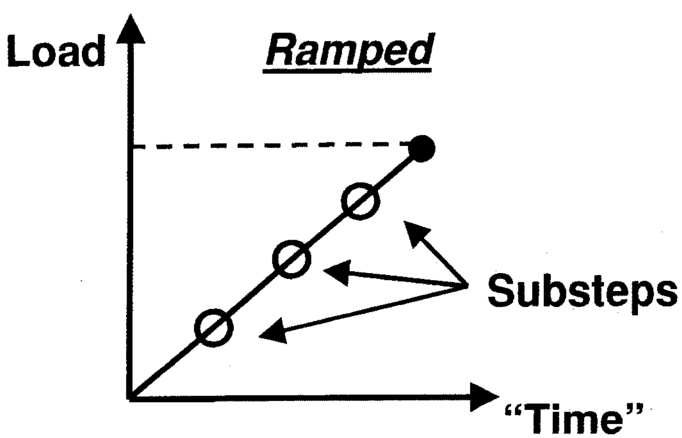
Note: If parallel processing is available, the frontal solver may be faster than the sparse solver, since the frontal solver is optimized for parallel processing.

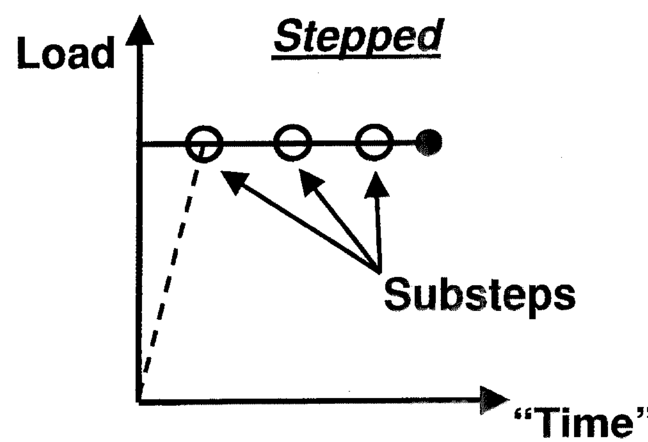
Ramped or Stepped Loading

Solution > Time/Freq > Time and Substps ...



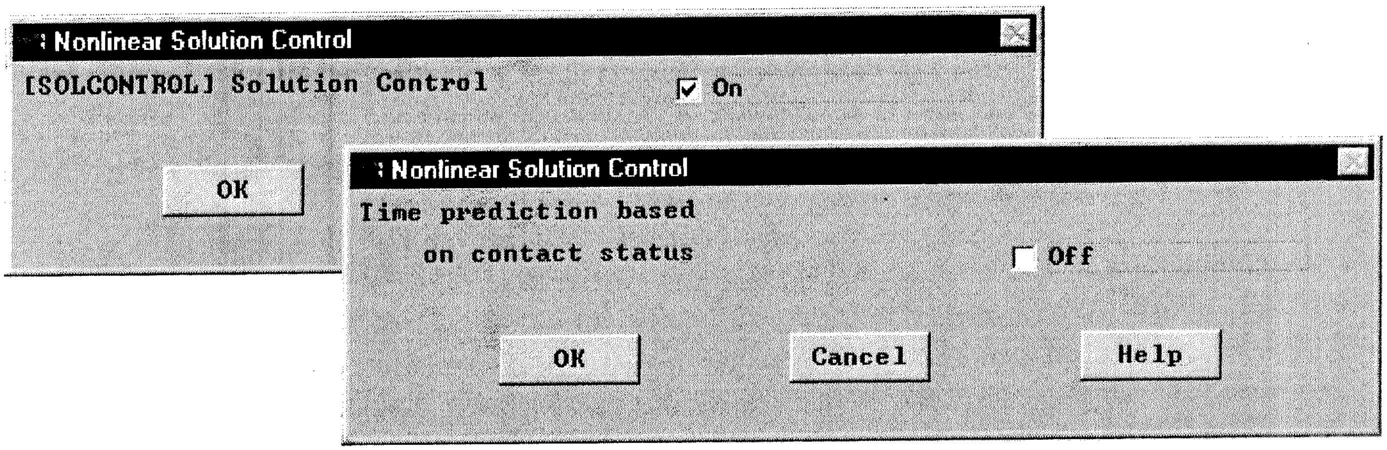
This option controls whether the loads are ramped (default for static) or stepped within a load step. *In a static analysis the loads should not be step applied.*





Time Step Prediction

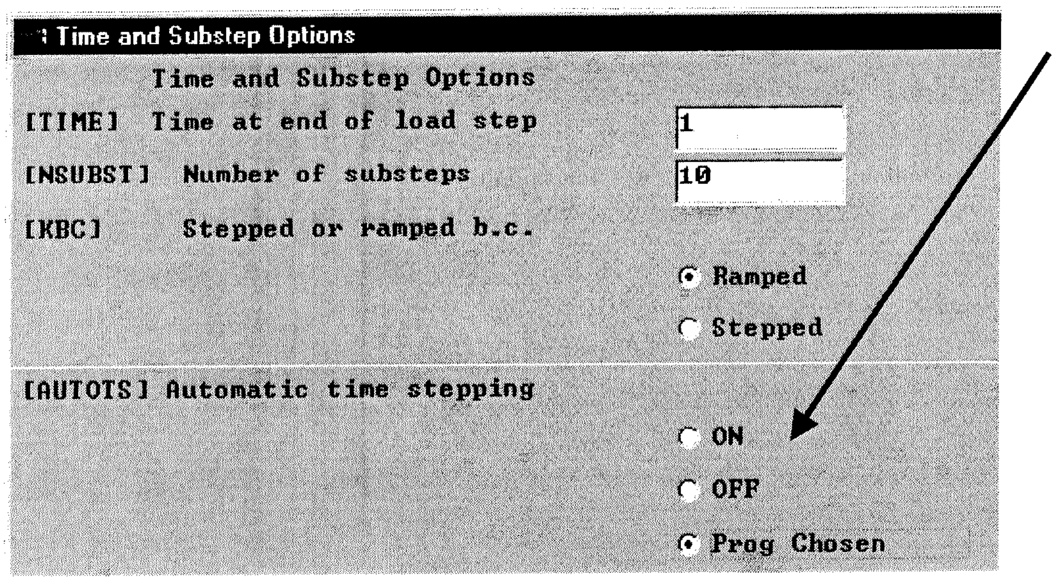
Solution > Solution Ctrl ...



This option controls whether or not the automatic time stepping algorithm is influenced by changes in the status of the contact elements in the model. The default value is off. (We will discuss this option in more detail in the Contact Nonlinearities section.)

Deactivate Automatic Time Stepping

Solution > Time/Freq > Time and Substps ...

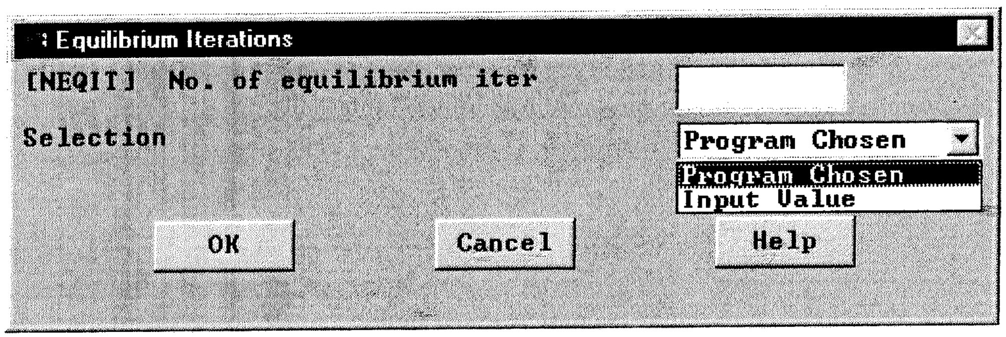


By default automatic time stepping is on, and this is generally the most efficient option.

The automatic time stepping can be deactivated and a uniform number of substeps specified.

Number of Equilibrium Iterations

Solution > Nonlinear > Equil Iter ...



Solution Control uses a default between 15 and 26 iterations depending on the physics of the problem. (The idea is to employ a small load increment with fewer quadratically converging iterations.) For contact problems you may want to increase the number of iterations (discussed in the Contact Nonlinearities section).

Convergence Criteria

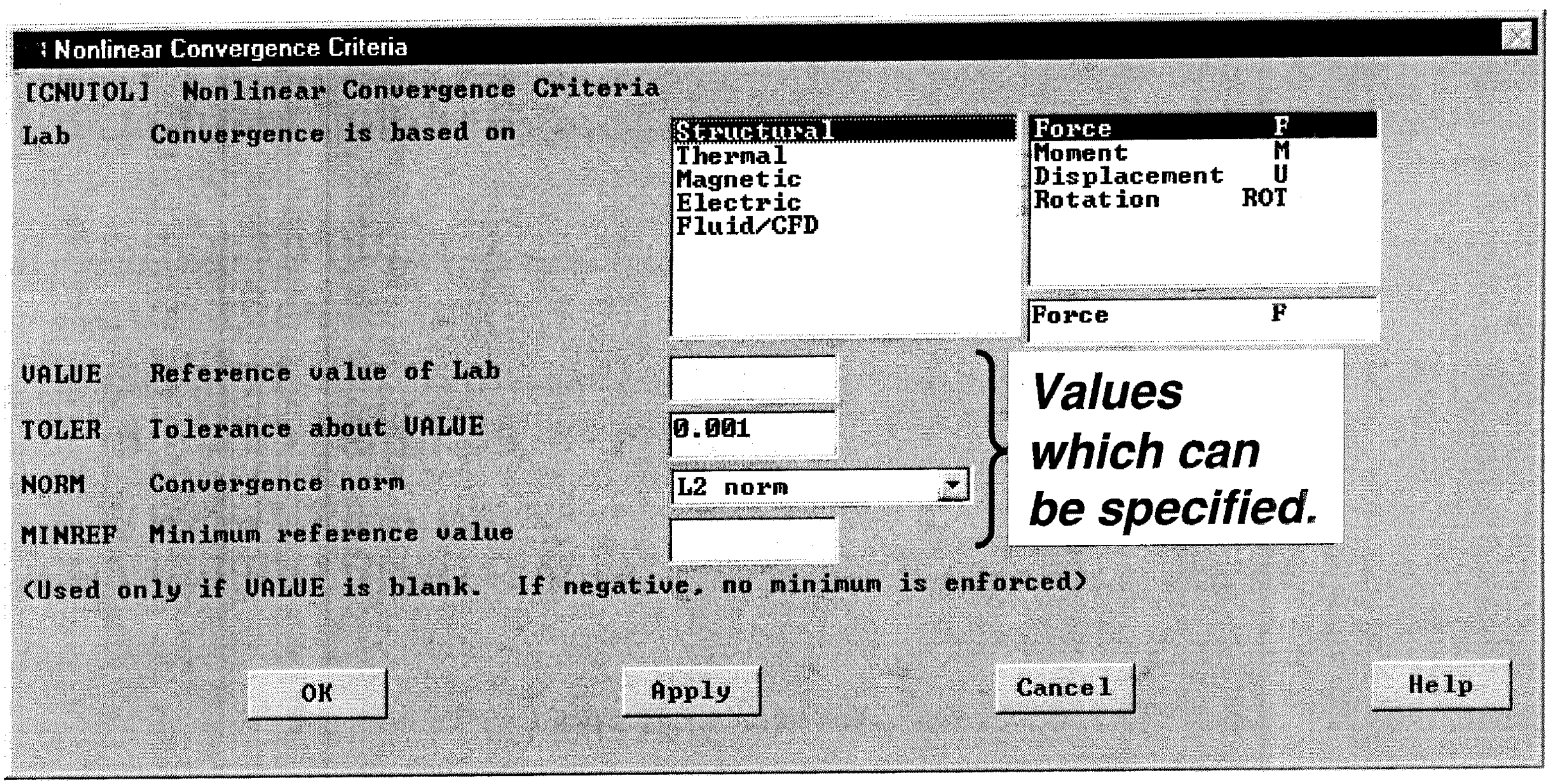
- By default ANSYS will check convergence based on a square root sum of the squares (SRSS) of the force and moment residuals equal to 0.5% of a SRSS of the applied loads (this is called an L2 norm). As a double check on convergence ANSYS also checks an L2 norm on displacement of 5%.

- *If you explicitly redefine the convergence criteria, the default criteria will be overwritten. If you redefine the force criteria, you will have to add displacement checking.*

- Remember that you should *always* define force convergence checking, as it is the measure of equilibrium.

Convergence Criteria

Solution > Nonlinear > Convergence Crit ...



Convergence Criteria

VALUE, TOLER and MINREF Parameters

- The force residual is checked against VALUE * TOLER. The default for VALUE is the SRSS of the applied loads. In general it is better practice to adjust TOLER to change the convergence criteria and let VALUE default.
- MINREF represents the minimum possible value of the VALUE parameter. The MINREF (defaults to 0.001) should represent numeric zero for your analysis. If MINREF is set equal to -1 no minimum is enforced.

Convergence Criteria

Convergence Norms L1, L2, Infinite

- The *L1 norm* uses the absolute sum of the force residuals to compare against the convergence criteria.
- The *L2 norm* (default) uses a SRSS of the force residuals.
- The *infinite norm* checks the maximum residual at all degrees of freedom. (This option has the effect of checking each degree of freedom in the model for convergence independently.)

Convergence Criteria Guidelines

- Using a "loose" convergence criteria is not a solution to convergence difficulty! (It can let the program "converge" to an incorrect solution!)

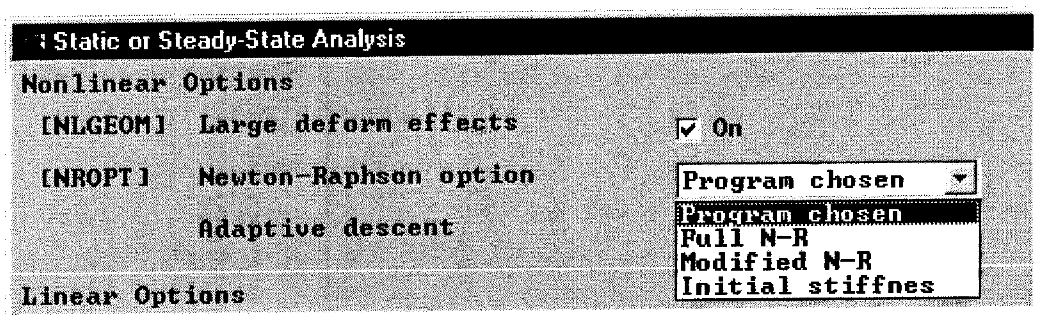
- You may want to tighten the convergence criteria to improve the accuracy of your results, but it will require more equilibrium iterations. If you want to tighten the criteria, change TOLER by one or two orders of magnitude.

- *You should also be certain that MINREF makes sense for the context of your analysis. Do not ignore any MINREF warning messages during solution!*

Newton-Raphson Options

Solution Control uses the full Newton-Raphson by default to achieve a quadratic rate of convergence. You can specify the Newton-Raphson option using:

Solution > Analysis Options ...



This setting will control how often the tangent stiffness matrix is updated in the solution.

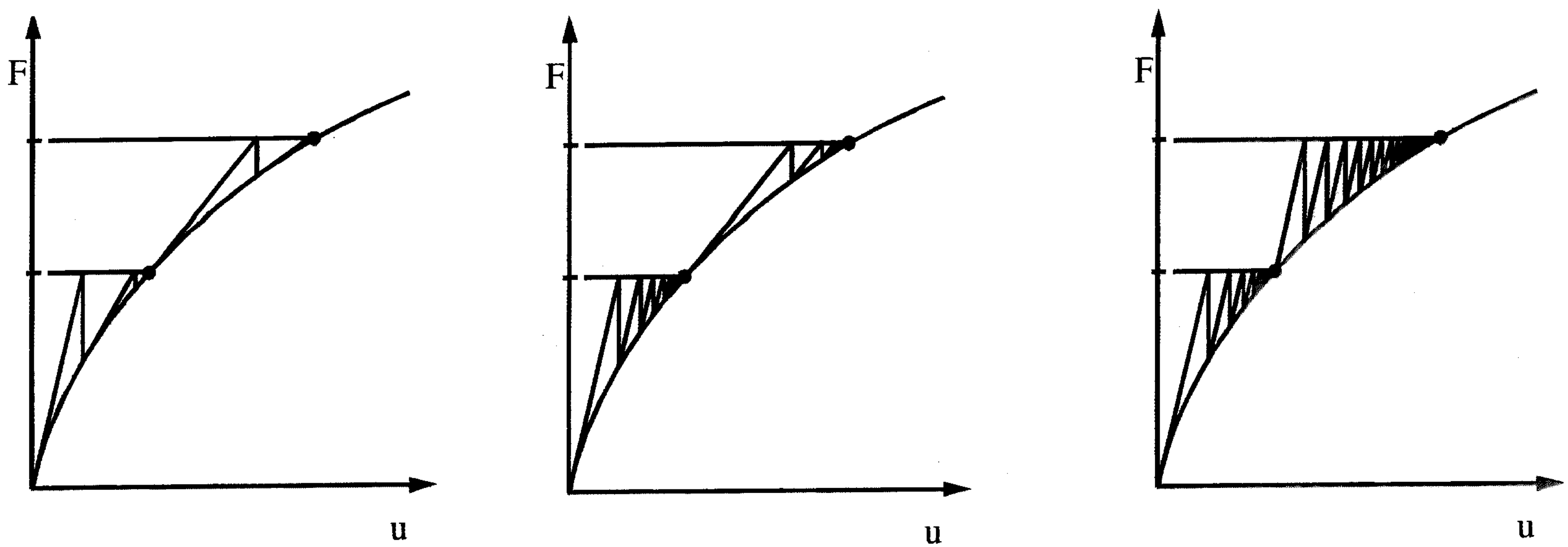
Newton-Raphson Options

- The full Newton-Raphson option updates the tangent stiffness matrix every iteration. A quadratic rate of convergence can be obtained with the full Newton-Raphson option (default).

- The modified Newton-Raphson option updates the tangent stiffness matrix only at the start of each substep. This option may save some CPU time if the problem is mildly nonlinear.

- The initial stiffness Newton-Raphson option reuses the initial elastic stiffness for every iteration. *Convergence is very slow. This option is rarely used.*

Newton-Raphson Options



Full Newton-Raphson

[K^T] is reformulated for every iteration.

Modified

[K^T] is reformulated once for each substep.

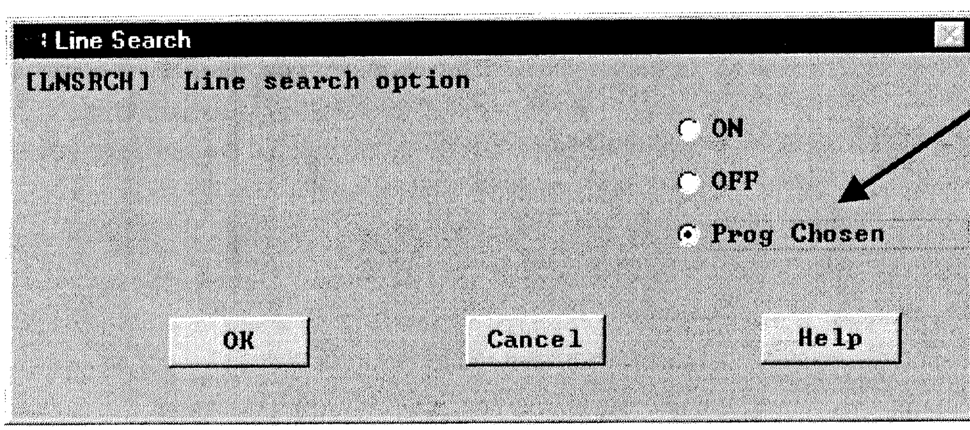
Initial Stiffness

The initial elastic stiffness is reused for each iteration.

Line Search

The line search procedure is a convergence enhancement tool. Solution Control will toggle line search on and off as required. When activated the displacement increment is multiplied by a program calculated scale factor between 0 and 1, when a stiffening response is detected.

Solution > Nonlinear > Line Search



Default is program chosen.

Line Search

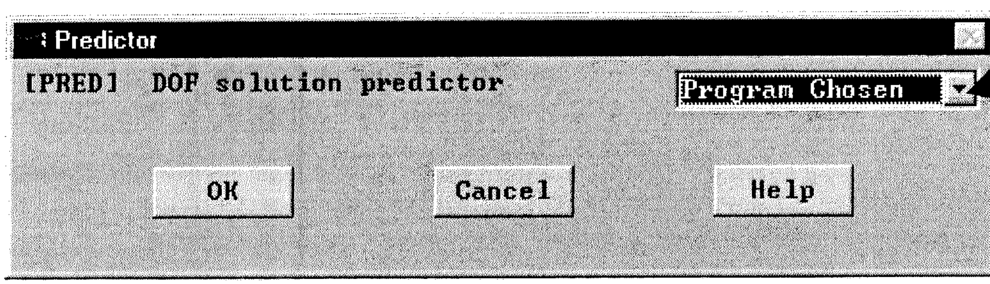
- The line search procedure is a *very powerful convergence enhancement tool*. It will not destabilize a solution if activated, and in many cases it will enhance a slowly converging solution. It does however require some additional CPU time to calculate the line search parameter.

- Line search is very effective in enhancing the convergence of an oscillating solution. If you see the `MAX DOF INC` in the output window oscillating between a positive and a negative value, activate the line search!

Predictor

- The predictor is a tool which can be used to accelerate a solution if the problem has a *smooth nonlinear response.* The default with solution control turns off the predictor if there are rotational degrees of freedom in the model, or if the current time step is reduced by the automatic time stepping algorithm.

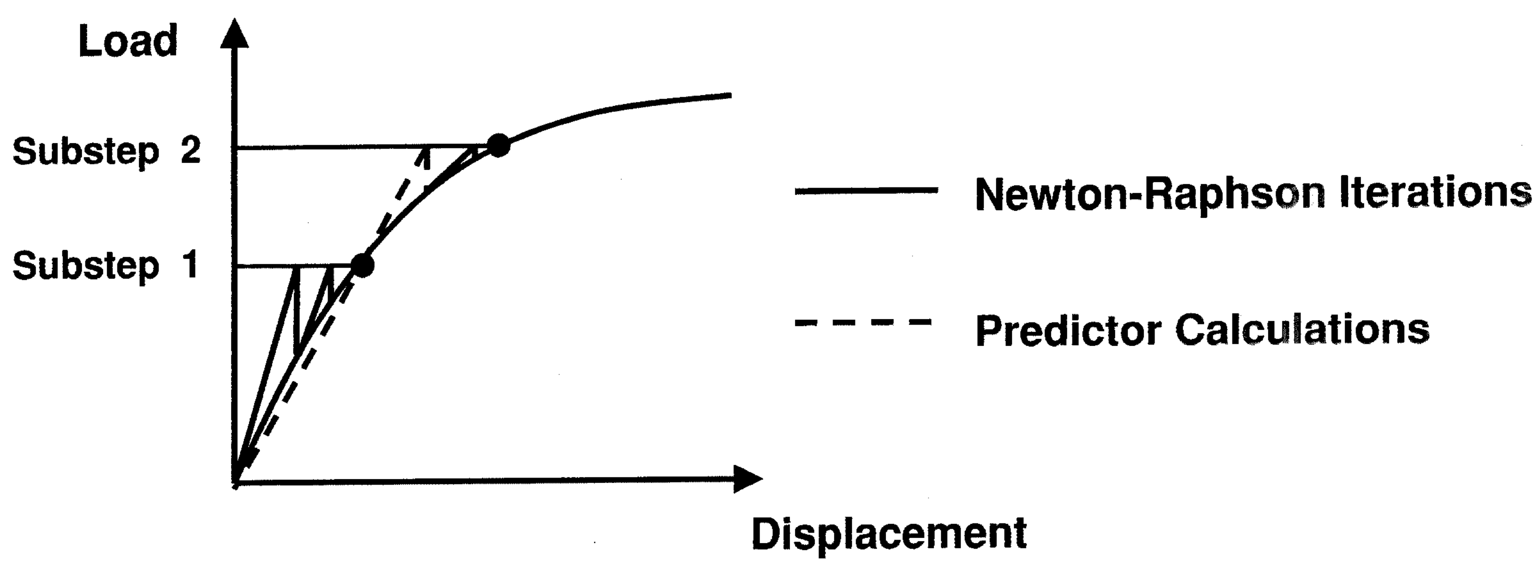
Solution > Nonlinear > Predictor



Default is program chosen.

Predictor

The predictor attempts to accelerate convergence by predicting the degree of freedom solution for the first equilibrium iteration of every substep. The predictor will extrapolate the results of the last substep to obtain a starting point for the next solution.



Predictor

- If the nonlinear response is smooth (and the time step sizes are reasonably small) the predictor can accelerate convergence.

- *If the nonlinear response is not smooth, or large rotations are incorporated in the analysis the predictor can cause divergence!*

- *Do not use the predictor for a large rotation analysis.*

Adaptive Descent

The adaptive descent allows the full Newton-Raphson to use a weighted combination of the secant and tangent stiffness matrices:

$$[K] = \xi[K^S] + (1 - \xi)[K^T]$$

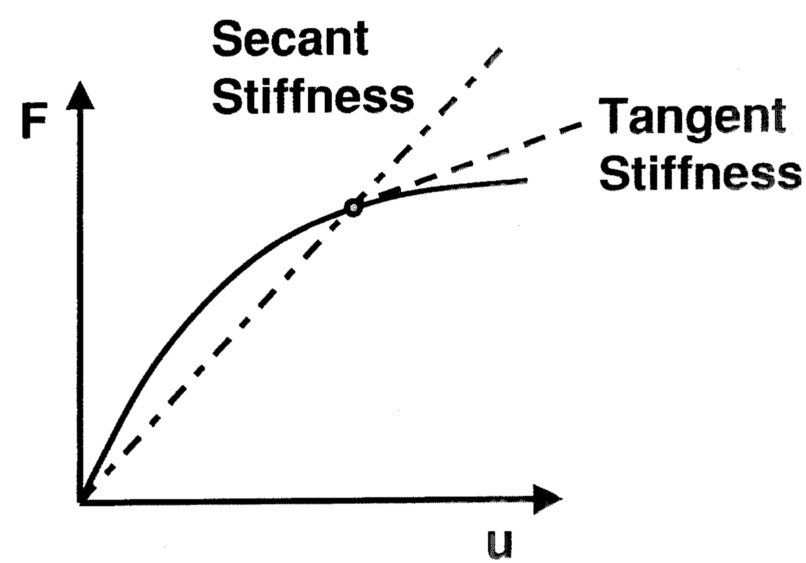
where:

ξ = Descent Parameter

$(0 \leq \xi \leq 1)$

$[K^T]$ = Tangent Stiffness

$[K^S]$ = Secant Stiffness

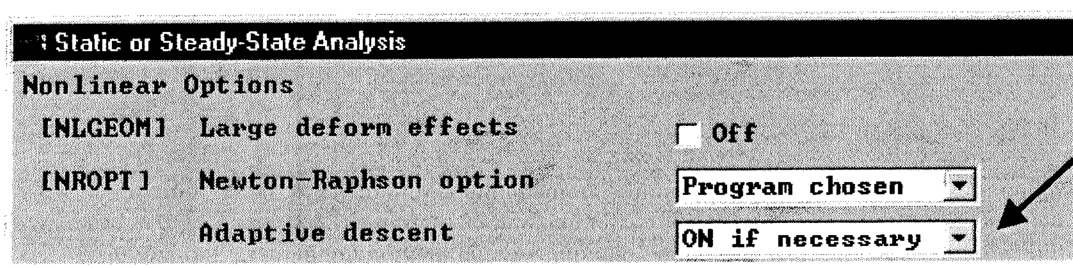


Adaptive Descent

- The adaptive descent option is *rarely* invoked by solution control. Solution control will only invoke the adaptive descent for the older node-to-node and node-to-surface contact elements (12, 48, 49 and 52) with friction.

- The adaptive descent should not be used in conjunction with the line search option! The two options are mutually exclusive.

Solution > Analysis Options ...

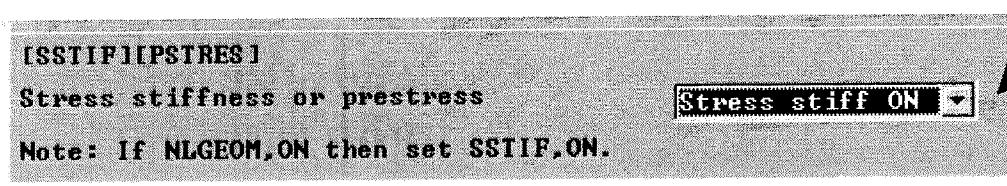


Default is off, and recommended in most cases.

Deactivate Stress Stiffening

- With solution control active, turning on nonlinear geometry (NLGEOM) will by default include the stress stiffening terms in the nonlinear stiffness matrix (includes [K^σ] in the nonlinear stiffness matrix [K^T]). As an option you can select not to include [K^σ] in the formulation of the nonlinear stiffness matrix for a subset of older elements.

Solution > Analysis Options ...



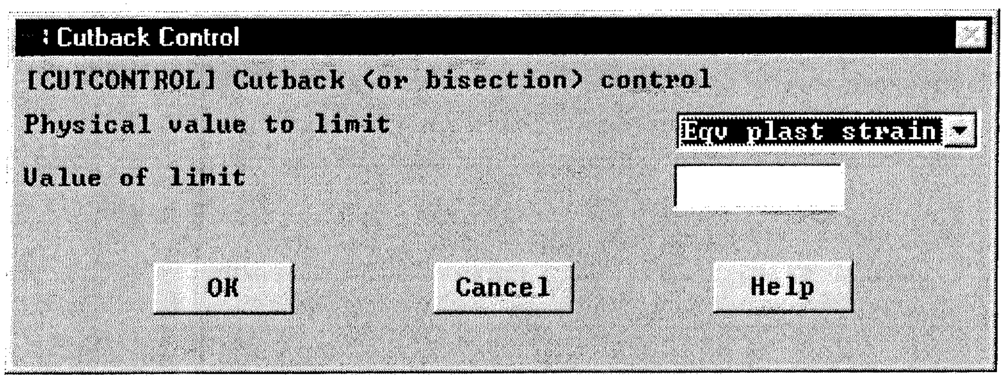
Rarely deactivated as including stress stiffening leads to a fully consistent [K^T].

This command has no effect for elements 106, 107, 108, 169, 170, 171, 172, 173, 174, 181, 182, 185, 188, and 189!

Cutback Control

Cutback control allows you to adjust the controls over the cutbacks used by the automatic time stepping algorithm.

Solution > Nonlinear > Cutback Control ...

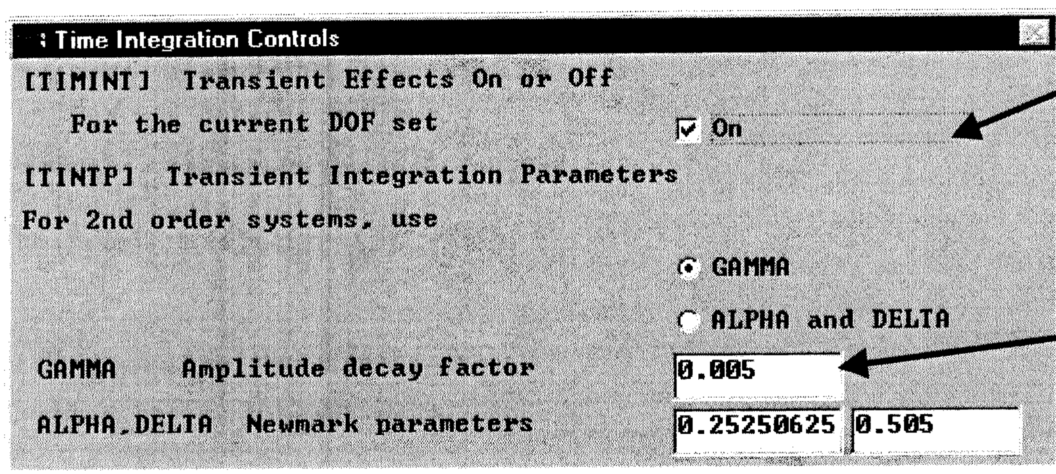


This function allows you to specify the maximum equivalent plastic strain increment (defaults to 15%), maximum creep ratio (defaults to 10%), number of points per cycle (defaults to 13 for a dynamic problem), and number of predicted iterations used per substep.

Time Integration Effects

- This function allows you to deactivate time integration effects in a transient analysis. Deactivating time integration is typically used for the first load step(s) of a transient analysis to establish the initial conditions for the transient.

Solution > Time/Frequency > Time Integration ...



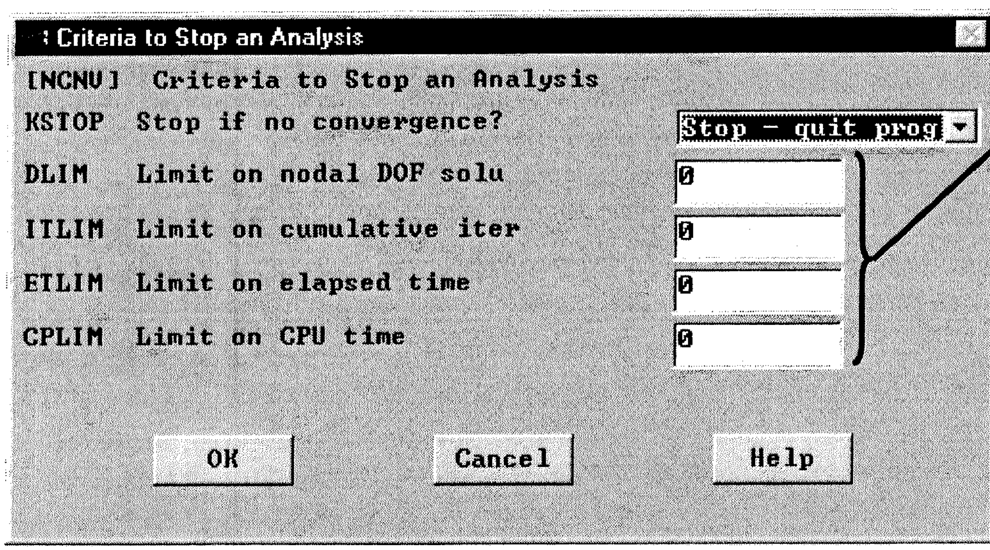
You can turn off time integration in a transient analysis.

(GAMMA allows you to introduce numerical damping in a transient solution.)

Stopping Control

- **This function allows you to control the termination of a nonlinear analysis. (Usually the default settings are appropriate.)**

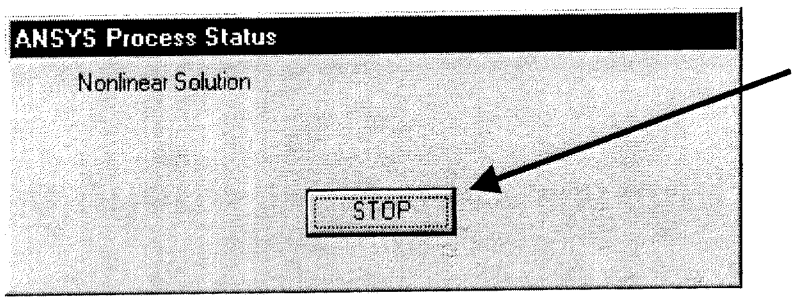
Solution > Nonlinear > Criteria to Stop ...



You can set limits on the displacement, cumulative iterations, elapsed and CPU time.

Terminating an Analysis

- Terminating analysis can be accomplished by pushing the "STOP" button for an interactive run or by using the abort file (jobname.abt) for a batch process.



Pushing the "STOP" button cleanly terminates the analysis (can be restarted).

- Creating the abort file (jobname.abt) containing the word *nonlinear* on the first line in the working directory will cleanly stop a batch process.
- Note that terminating the solution using STOP or the ABORT file causes ANSYS to automatically save the database.

Restarting an Analysis

- An analysis which has stopped from *normal program termination* (STOP button pushed, abort file, solution completed) can be restarted.
- An analysis which has terminated from the *system level* (a system break, system crash, or power failure) can not be restarted.
- You may want to restart an analysis to continue the load history or to recover from a convergence failure.
- A restart can only be performed from the last converged load step and substep.

Restarting an Analysis

The model must meet the following conditions to restart:

- Static or transient analysis
- The initial run must have terminated cleanly
- The following files must also be available:

  Jobname.db - The database file saved with the *appropriate loads and boundary conditions for the load step to be restarted.*

  Jobname.emat - Element matrices.

  Jobname.esav or .osav - Element saved data or old element saved data (see table).

  Jobname.rst - Not required, but if available appended to.

Restarting an Analysis

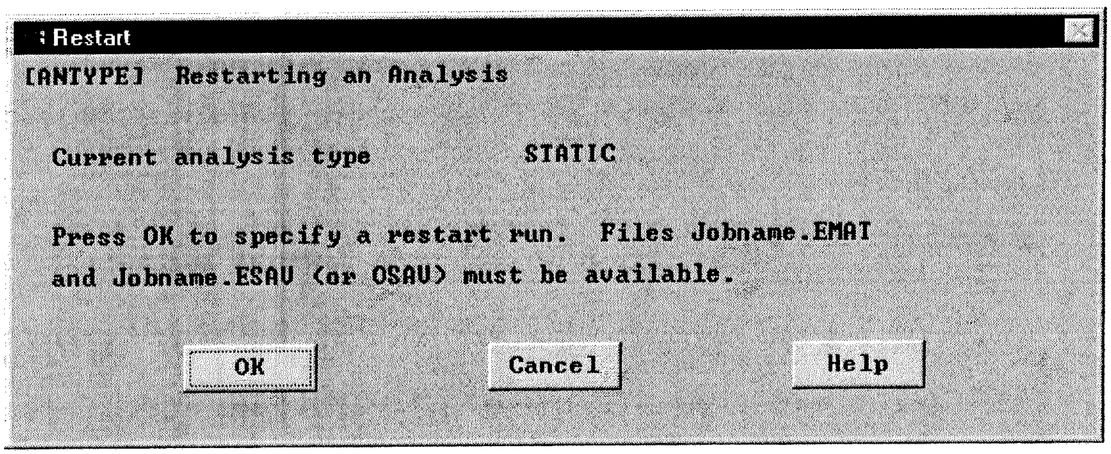
Presented below are the element saved data file requirements for a restart. Note if *Jobname*.osav is required, it must be copied to *Jobname*.esav before restarting the analysis.

Termination	Element Saved Data File	Corrective Action
Normal	*Jobname*.esav	Add more load steps.
Non-Convergence	*Jobname*.osav	Define smaller time step, adjust solution controls.
Insufficient Iterations	*Jobname*.esav	Add more equilibrium iterations.
Cumulative Iteration Limit Exceeded	*Jobname*.esav	Increase iteration limit.
Time Limit Exceeded	*Jobname*.esav	None (restart analysis).
Displacement Limit Exceeded	*Jobname*.osav	Same as non-convergence.
Negative Pivot	*Jobname*.osav	Same as non-convergence.
Abort File – Converging	*Jobname*.esav	None (restart analysis).
Abort File – Diverging	*Jobname*.osav	Same as non-convergence.
“Full” Results File	*Jobname*.esav	Increase the results file limits, and reduce the amount results of output.
System Crash	Not Applicable	No restart is possible.

Restart Procedure

1. Resume the database and enter solution.
2. Indicate that this is a restart analysis.

   Solution > Restart ...



Make sure that you are using the Jobname.esav or Jobname.osav as required!

3. Define a new load step, or adjust the solution controls as necessary.
4. Save the database and initiate a new solution.

Chapter 3

GEOMETRIC NONLINEARITIES

Large Strain, Large Displacement and Large Rotation Behavior

Session Objective

- At the end of this section you will be able to describe and demonstrate the following:

  1. Geometrically Nonlinear Behavior

     - Large Strain, Large Deflection, Large Rotation

  2. Nonlinear Stress and Strain Measures

  3. Consistent Nonlinear Stiffness Matrix

  4. Obtaining the Solution

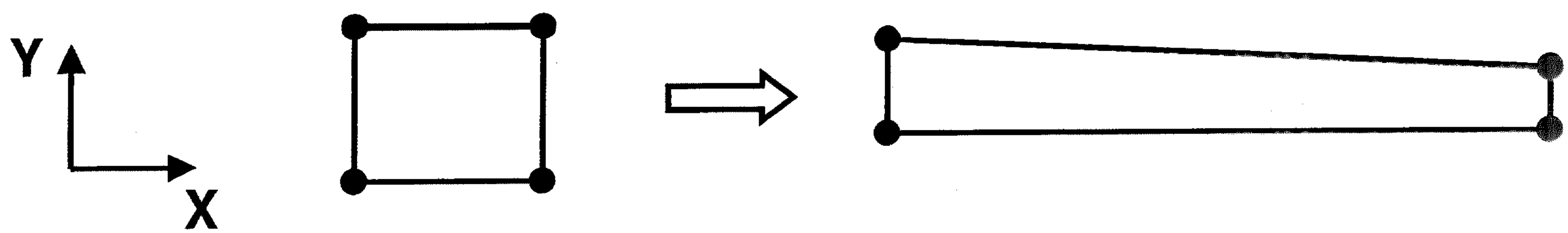
  5. Trouble Shooting

What is Geometrically Nonlinear Behavior?

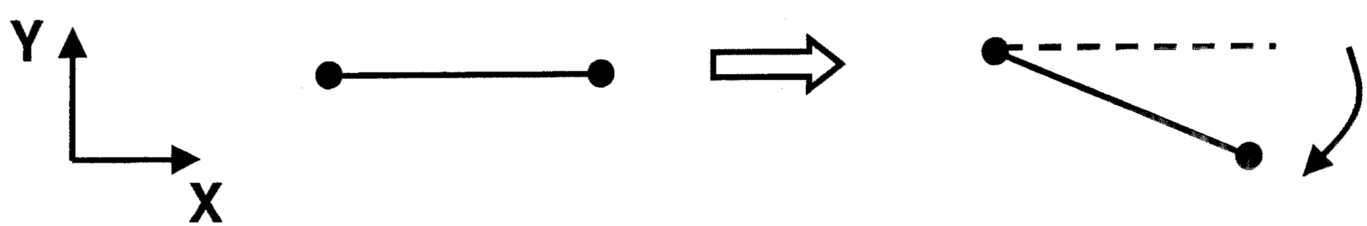
- Behavior in which changes in geometry, induced in a deforming body, have a significant effect on the load-deflection (i.e. stiffness) characteristics of the body.
- Geometrically nonlinear behavior does not only refer to large deflection behavior but refers to any change in structural response induced by changes in geometry. This includes large strain, large deflection (large rotation) and stress stiffening.

Geometrically Nonlinear Behavior

- If an element's shape changes (area, thickness, etc.), its individual element stiffness will change.

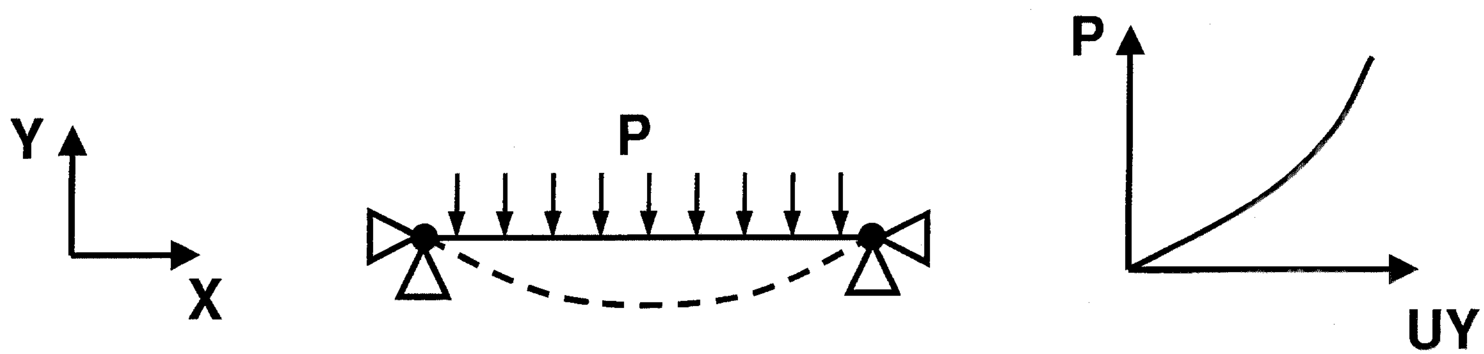


- If an element's orientation changes, the transformation of its local stiffness into global components will change.



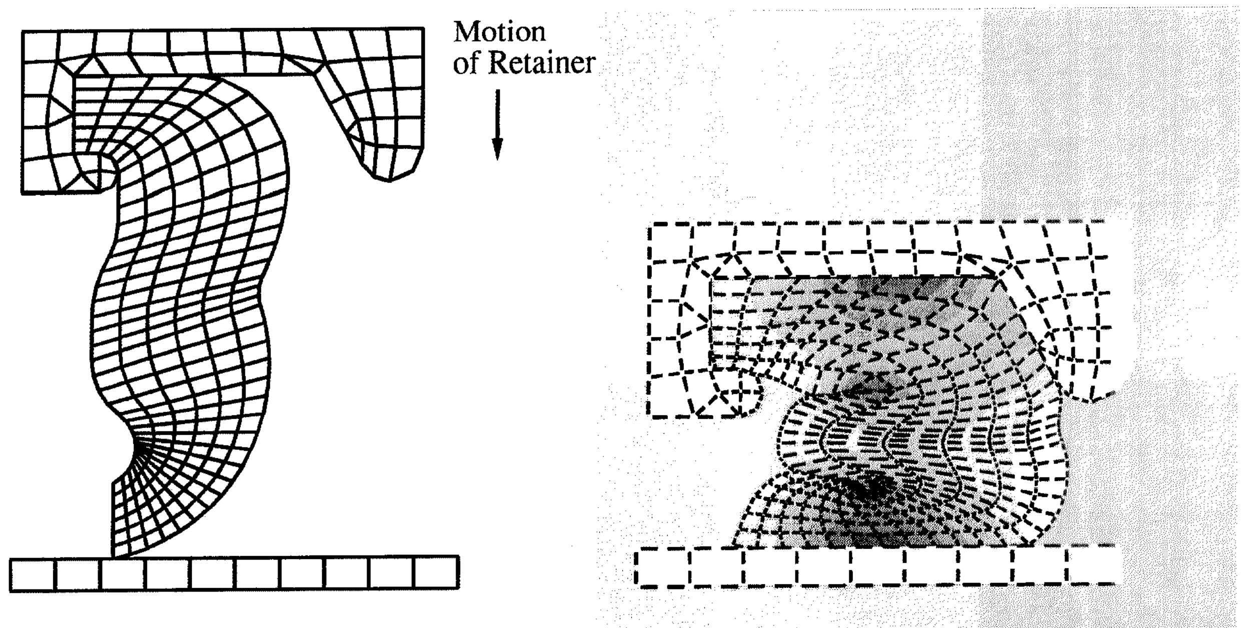
Geometrically Nonlinear Behavior

- If an element's strains produce a significant in-plane stress state (membrane stresses), the out-of-plane stiffness can be significantly affected.



- As the vertical deflection increases (Y), significant membrane stresses (SX) lead to a stiffening response.

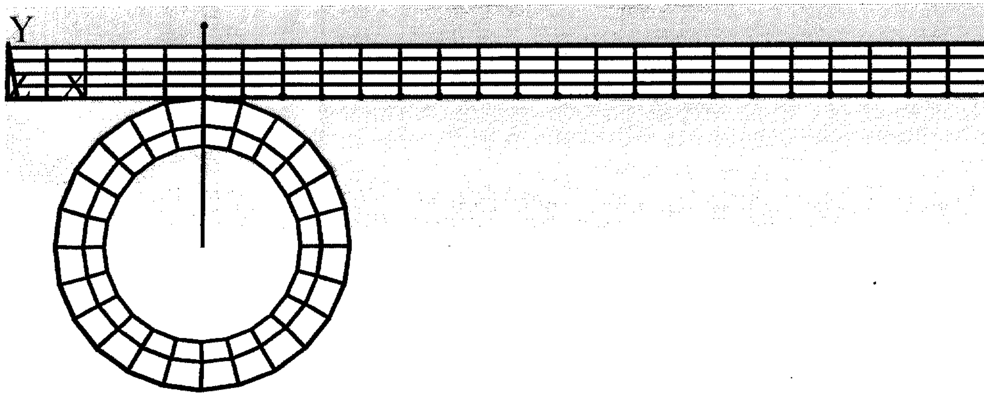
Examples of Geometrically Nonlinear Behavior

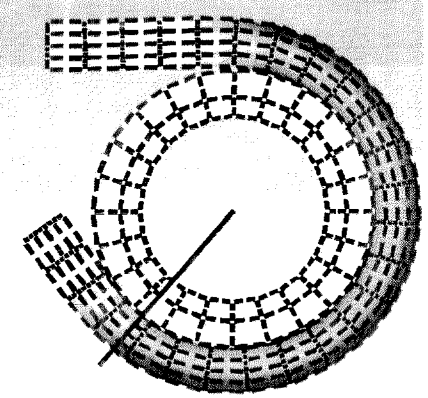


(a) Initial geometry (b) Deformed shape with hydrostatic stresses

- This is an example of large strain analysis of compression of an axisymmetric rubber seal. This analysis includes contact, including self contact when the seal folds over on itself.

Examples of Geometrically Nonlinear Behavior





Initial Geometry

Deformed Shape

- This example illustrates the wrapping of a steel bar around a mandrel. The bending of metal into a different shape is a common operation in manufacturing. In this example, the strains are on the order of 25% and the rotation of the tip is approaching 270 degrees!

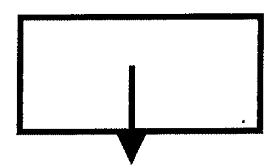
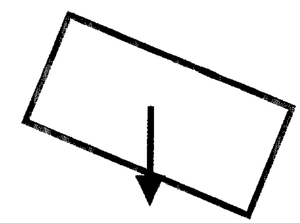
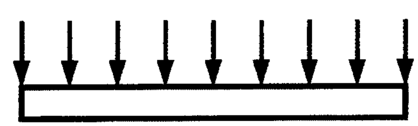
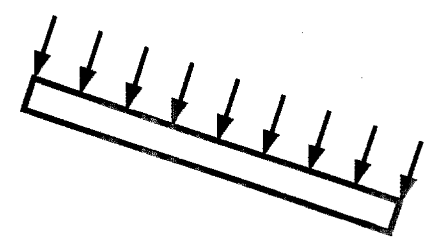
Complexities Introduced by Geometric Nonlinearites

- Follower Forces
- Nonlinear Stress and Strain Measures
- Consistent Nonlinear Stiffness Matrix
- Material Nonlinearities
- Incompressibility
- Mesh Distortion

Follower Forces and Load Directions

- Consider what happens to your loads when your structure experiences large deflections and rotations.
- In many instances, loads will maintain constant direction during deformation.
  - In other cases, forces will change direction "following" the elements as they undergo large rotations.
- ANSYS can model both situations, depending on the type of load applied.

Load Directions

Load	Direction Before Deflection	Direction After Deflection
Acceleration		
Nodal Force		
Element Pressure		

Load Directions

Accelerations and Concentrated Forces:

- Assumed to maintain their original direction, regardless of element orientation.

Surface Pressure Loads:

- Assumed to rotate with the element and thereby always act normal to the deformed element surface (i.e. these are true follower forces).

- Pressure surfaces are updated to account for large strain effects. Thus, for a constant applied pressure, the total pressure load will change as the surface area changes.

What is Large Strain?

- Large strain analyses assume that strain is no longer infinitesimal, but rather it is finite or large.
- Strains can be considered large when they exceed more than a few percent, and when the changing geometry can no longer be neglected.
- Large strain theory accounts for shape changes (such as thickness, area, etc.) and any large rotations.

Requirements for a Definition of Strain

- Strains are the means by which we characterize the *deformation* of a body. Although the mathematical definition of strain we implement can be arbitrary to some extent, there are some requirements which it must satisfy.

Strain Definition Requirements

- The strain measure should be zero when there is no deformation and non-zero when there is deformation.
- The strain measure should relate to stress through a material stress-strain relationship.
- If rotations in the material are large, the strain measure should be zero under a rigid body rotation.

Correspondence of Stress and Strain Definitions

- Corresponding to the *strains* which measure the deformation of a body, are the stresses which measure force per unit area in a body.

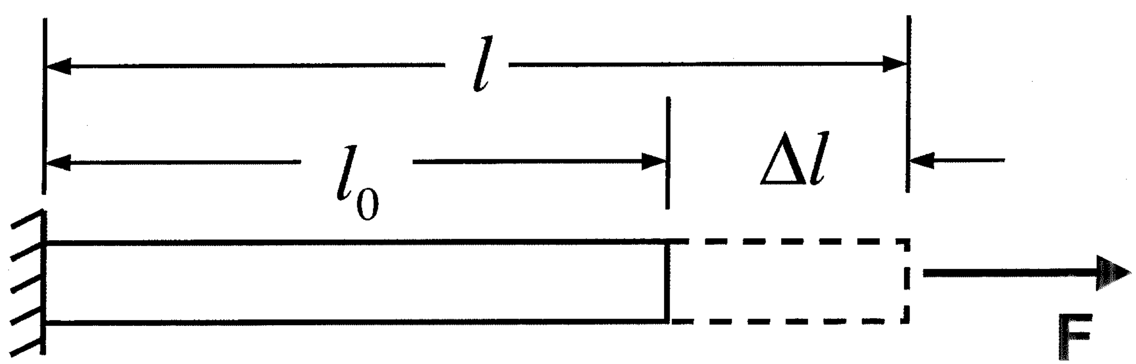
- Although the strain measure we employ can be arbitrary, we find that the stress measure we use depends on the strain measure we have defined.

Conjugacy of Stress and Strain Definitions

- In a nonlinear large strain analysis the stress measures employed *must be conjugate* to the strain measures.
- *Conjugacy* means that when the stresses are multiplied by the strains, we get a scalar quantity (the strain energy) which is independent of the stress and strain measures selected.

Stress and Strain Definitions for the 1-D Case

- We will examine the various stress and strain definitions through a simple one-dimensional example.



Engineering Strain

- The *engineering strain* ε is a *small* (infinitesimal) strain measure which is computed using the original geometry

$$\varepsilon = \frac{\Delta l}{l_0}$$

- The engineering strain measure is a linear measure since it depends on the original geometry, i.e. length l_o which is known as priori.

- It is limited (in theory) to small rotations of the material because a moderate rigid body rotation will result in non-zero strains. (However, ANSYS implementation overcomes this limitation in practice.)

Engineering Stress

- The conjugate stress measure to engineering strain ε is engineering stress σ , which uses the current force F and the original area A_0 in its computation.

$$\sigma = \frac{F}{A_0}$$

Logarithmic Strain

In 1-D, the *logarithmic strain* measure ε_l is computed as

$$\varepsilon_l = \int_{l_o}^{l} \frac{dl}{l} = Ln\left(\frac{l}{l_0}\right)$$

This measure is a *nonlinear* strain measure since it is a nonlinear function of the unknown final length l. It is also referred to as the *log* strain. The 3-D equivalent of the log strain ε_l is the *Hencky* strain.

The logarithmic strain does not automatically accommodate arbitrarily large rotations in large strain problems.

True Stress or Cauchy Stress

The conjugate 1-D stress measure to the log strain ε_l is the *true stress* τ , which is computed by dividing the current force F by the current (or deformed) area A :

$$\tau = \frac{F}{A}$$

This measure is also commonly referred to as the *Cauchy stress.*

ANSYS

Green-Lagrange Strain

The Green-Lagrange strain ε_G is computed in 1D using

$$\varepsilon_G = \frac{1}{2}\left(\frac{l^2 - l_0^2}{l_0^2}\right)$$

This measure is nonlinear because it depends on the square of the updated length l which is an unknown.

An advantage of this strain measure, over the log or Hencky strain ε_l, is that it automatically accommodates arbitrarily large rotations in large strain problems.

Second Piola-Kirchhoff Stress

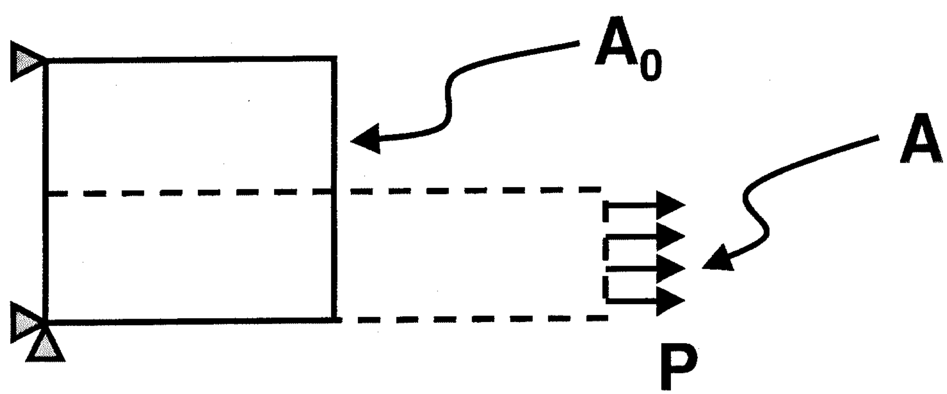
The conjugate stress measure for the Green-Lagrange strain ε_G, is the Second Piola-Kirchhoff stress S. It is computed in 1-D by

$$S = \frac{l_0}{l}\frac{F}{A_0}$$

It should be noted that this stress has little physical meaning. For output purposes, ANSYS always reports stresses for options that use this measure as Cauchy or True stresses τ.

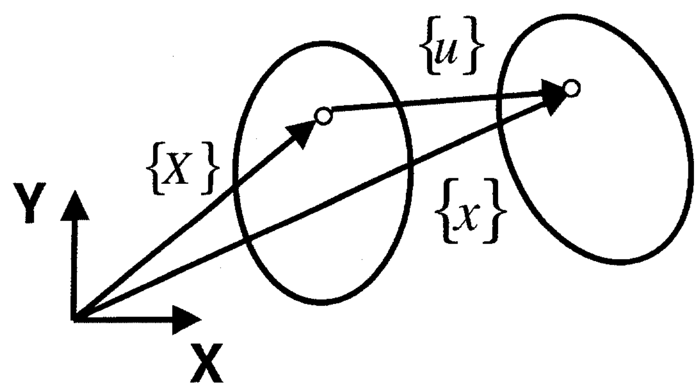
Extending the Nonlinear Strain Definitions to the General 3-D Case

- In 2-D and 3-D, not only does the basis for length change when a component is undergoing large strain deformations, but thickness, area and volume also change.



Motion and Deformations

- When a body is under the influence of some external loads, it will move and deform.



- If we watch the motion of a point attached to the body, with its initial position $\{X\}$ and its final position $\{x\}$, it will be seen to displace an amount $\{u\}$ where

$$\{u\}=\{x\}-\{X\}$$

Deformation Gradient

The deformation gradient is a measure of how much the body has deformed and is defined as

$$[F] = \frac{\partial\{x\}}{\partial\{X\}}$$

The deformation gradient $[F]$ contains information on the:

- Volume Change
- Rotation
- Shape Change due to Straining

Deformation Gradient

- Note that by definition, the *deformation gradient* $[F]$ *eliminates translations*; a necessary requirement for a strain definition.

- When defining our strains, we also want to *exclude the rotation* part (since it does not contribute to straining) and isolate the shape change part. We can accomplish this by implementing the *Polar Decomposition Theorem.*

Polar Decomposition

- The deformation gradient $[F]$ can be separated into a rotation and a strain part using the polar decomposition theorem:

$$[F]=[R][U]$$

$[R]$ = rotation matrix containing information about the amount and direction that the material point rotated as a rigid body

$[U]$ = stretch matrix containing the information about the straining of the body at the material point

Implementing $[U]$ in the Definitions of Strain

- By knowing the stretch matrix $[U]$, the 3-D generalizations for the 1-D logarithmic strain and the 1-D Green-Lagrange strain ε_G can be ε_l constructed.

The Hencky Strain

The log (Hencky) strain is computed by:

$$[\varepsilon_H] = \ln([U])$$

where $[\varepsilon_H]$ is the strain tensor expressed in matrix form.

$[\varepsilon_H]$ is, in this case, the 3-D equivalent to the 1-D logarithmic, or true strain ε_l .

Green-Lagrange Strain

In 3-D, the Green-Lagrange strain can be computed directly from the stretch matrix $[U]$ as shown below:

$$[\varepsilon_G] = \frac{1}{2}\left[[U]^T[U] - [I]\right]$$

This measure directly neglects the rotation matrix $[R]$ from the strain field evaluation. $[\varepsilon_G]$ can be re-written in terms of the gradient of the displacement field as shown below:

$$[\varepsilon_G] = \left[\left[\frac{\partial\{u\}}{\partial\lfloor X\rfloor}\right] + \left[\frac{\partial\{u\}}{\partial\lfloor X\rfloor}\right]^T + \left[\left[\frac{\partial\{u\}}{\partial\lfloor X\rfloor}\right]^T\left[\frac{\partial\{u\}}{\partial\lfloor X\rfloor}\right]\right]\right]$$

The first two terms are the linear small strain terms and the last term is the nonlinear contribution to the strain measure.

Which Nonlinear Strain Measure Does ANSYS Use?

- In the ANSYS program, the nonlinear strain measure which is used (i.e. Hencky or Green-Lagrange) is dependent primarily on the material law:
  - For large strain plasticity analyses, ANSYS uses the log (Hencky) strain measure; the stress-strain data being given in true stress-log strain format.
  - For large strain hyperelastic and visco-elastic analyses, ANSYS uses either the Hencky or the Green-Lagrange strain depending on the element being implemented.

Extending the Nonlinear Stress Definitions to the General 3-D Case

- As in the 1-D case, there are conjugate stress measures which can be defined for each of the nonlinear strain definitions in 2-D and 3-D.

The Cauchy Stress

- The Cauchy or true stress tensor $[\tau]$ (written here in matrix form) gives the current force in the deformed configuration per unit deformed area. If we let

  $\{dA\}$ = vector defining components of an *elemental area* in the *deformed body*

  $\{dP\}$ = the corresponding elemental force acting on $\{dA\}$

- then, in 3-D, the Cauchy stress tensor $[\tau]$ relates $\{dP\}$ to $\{dA\}$ as

$$\{dP\}=[\tau]\{dA\}$$

- The Cauchy stress is a readily interpretable physical quantity.

The 2nd Piola-Kirchhoff Stress

- Let $\{d\hat{P}\}$ represent a force component derived from the transformation $\{d\hat{P}\} = [F]^{-1}\{dP\}$
- Also let $\{dA_0\}$ = vector defining an *elemental area* in the *un-deformed* body where $\{dA_0\} \underset{deformation}{\rightarrow} \{dA\}$
- The 2nd Piola-Kirchhoff stress tensor $[S]$ relates $\{d\hat{P}\}$ and $\{dA_0\}$ as $\{d\hat{P}\} = [S]\{dA_0\}$
- $[S]$ is a symmetric stress tensor which is frequently used in finite-strain elasticity formulations and is the conjugate stress tensor to the Green-Lagrange strain $[\varepsilon_G]$. $[S]$ is a non-physical stress tensor (a *pseudo stress* tensor). $[S]$ does not lend itself to direct physical interpretation.

The Relationship Between $[\tau]$ and $[S]$

- The physical Cauchy $[\tau]$ stress can be related directly to the non-physical 2nd Piola-Kirchhoff pseudo stress $[S]$ via the expression

$$[\tau] = \left(\frac{1}{\det[F]}\right)[F][S][F]^T$$

Which Stress Measure Does ANSYS Use in a Nonlinear Analysis?

- In order to ensure that the stress and strain components are conjugate, the stress measure (i.e. Cauchy or 2nd Piola-Kirchhoff) which ANSYS uses during a nonlinear analysis is dependent on the strain measure.
  - The Cauchy stress is used whenever the Hencky (log) strain measure is implemented.
  - The 2nd Piola-Kirchhoff stress is used whenever the Green-Lagrange strain is used.

Which Stress-Strain Measure Does ANSYS Require as Input?

- For large strain plasticity analyses (NLGEOM,ON) ANSYS expects a true stress-strain curve, whereas for small strain analyses (NLGEOM,OFF) ANSYS expects engineering stress-strain data.

- However, for small strain response, the engineering strain and logarithmic (true) strain are nearly identical. True stress and logarithmic strain data can be used for general plasticity applications.

- For hyperelastic analyses ANSYS expects engineering stress-strain data for the calculation of the Mooney-Rivlin constants.

Conversion Between Stress-Strain Measures

- For uniaxial stress-strain data, engineering stress versus engineering strain can be converted to true stress versus log strain by

$$\varepsilon_l = \ln(1+\varepsilon)$$

$$\tau = \sigma(1+\varepsilon)$$

- Note that the stress conversion above assumes that the material undergoing the large strains is incompressible or nearly incompressible. This assumption is valid for large plastic strains or hyperelastic materials.

Which Stress Measure Does ANSYS Output?

- Regardless of which stress measure (Cauchy or 2nd Piola-Kirchhoff) is used during numerical computation, ANSYS always outputs all stress results in terms of the physically interpretable Cauchy stress.

Which Elements Support Large Strain?

- BEAM 23
- BEAM188
- BEAM189
- HYPER56
- HYPER58
- HYPER74
- HYPER84
- HYPER86
- PIPE20
- PLANE2
- PLANE13
- PLANE42
- PLANE82
- PLANE182
- SHELL43
- SHELL91
- SHELL93
- SHELL181
- SOLID45
- SOLID92
- SOLID95
- SOLID185
- VISCO106
- VISCO107
- VISCO108

The Fully Consistent Nonlinear Tangent Stiffness Matrix

- It is a well known fact that in most nonlinear structural problems, the use of a consistent nonlinear tangent stiffness matrix can rapidly increase the rate of convergence of a Newton-Raphson based solution process.

- A consistent, or full tangent stiffness matrix normally leads to a quadratic convergence rate in the iterative solution process.

What is a Consistent Nonlinear Stiffness Matrix?

- The consistent nonlinear stiffness matrix $\left[K_e^{nl}\right]$ is obtained by taking the derivative of the discretized finite element equations and is a function of both the element internal force vector $\left\{F_e^{int}\right\}$ and the element applied load vector $\left\{F_e^{a}\right\}$.

The Discretized Nonlinear Static Finite Element Equations

The discretized nonlinear static finite element equations which are solved can be characterized at the element level by:

$$\sum_{e=1}^{N_e}\left\{[T_n]^T\{F_e^{int}\}-\{F_e^{a}\}\right\}=\{0\}$$

where

N_e = number of total elements

$\{F_e^{int}\}$ = element internal force vector in the element coordinate system

$[T_n]$ = transformation matrix transferring $\{F_e^{int}\}$ into the global coordinate system

$\{F_e^{a}\}$ = applied load vector at the element level in the global coordinate system

The Element Internal Force Vector

The element internal force vector $\{F_e^{int}\}$ is given by

$$\{F_e^{int}\} = \int_{V_e} [B_v]^T \{\sigma\} dV_e$$

where

$[B_v]$ = element ***strain - nodal displacement*** matrix

$\{\sigma\}$ = element stress vector

V_e = element volume

In terms of the internal force definition given above, the discretized nonlinear finite element equations (force balance) can be re-written as

$$\sum_{e=1}^{N_e} \left\{ [T_n]^T \left\{ \int_{V_e} [B_v]\{\sigma\} dV_e \right\} - \{F_e^a\} \right\} = \{0\}$$

Developing the Incremental Nonlinear Stiffness Matrix

The consistent nonlinear stiffness matrix $\left[K_e^{nl}\right]$ is derived by taking the derivative of the discretized finite element equations as shown below where

$$\left[K_e^{nl}\right] = \frac{\partial}{\partial\lfloor u \rfloor}\left\{\left[T_n\right]^T\left\{F_e^{int}\right\} - \left\{F_e^{a}\right\}\right\} = \left[K_e^{inc}\right] + \left[K_e^{\sigma}\right] + \left[K_e^{u}\right] - \left[K_e^{a}\right]$$

and where

$$\left[K_e^{inc}\right] \rightarrow K_e^{inc}(i,j) = \sum_{k=1}^{N_u} T_n(i,k) \sum_{l=1V_e}^{N_\sigma} \int B_v(k,l) \frac{\partial \sigma(l)}{\partial u(j)} dV_e$$

$$\left[K_e^{a}\right] = \frac{\partial\left\{F_e^{a}\right\}}{\partial\lfloor u \rfloor}$$

$$\left[K_e^{\sigma}\right] \rightarrow K_e^{\sigma}(i,j) = \sum_{k=1}^{N_u} T_n(i,k) \sum_{l=1V_e}^{N_\sigma} \int \frac{\partial B_v(k,l)}{\partial u(j)} \sigma(l) dV_e$$

$$\left[K_e^{u}\right] \rightarrow K_e^{u}(i,j) = \sum_{k=1}^{N_u} \frac{\partial T_n(i,k)}{\partial u(j)} \sum_{l=1V_e}^{N_\sigma} \int B_v(k,l) \sigma(l) dV_e$$

$N_u = Number\ of\ Element\ Degrees\ of\ Freedom$

$N_u = Number\ of\ Stress\ Components$

The Consistent Nonlinear Stiffness Matrix

$$\left[K_e^{nl}\right] = \left[K_e^{inc}\right] + \left[K_e^{\sigma}\right] + \left[K_e^{u}\right] - \left[K_e^{a}\right]$$

$\left[K_e^{inc}\right]$ = **the main tangent matrix**

$\left[K_e^{\sigma}\right]$ = **the initial stress matrix which incorporates the effects of *stress-stiffening***

$\left[K_e^{u}\right]$ = **the initial displacement-rotation matrix which includes the effect of changing geometry in the stiffness relation**

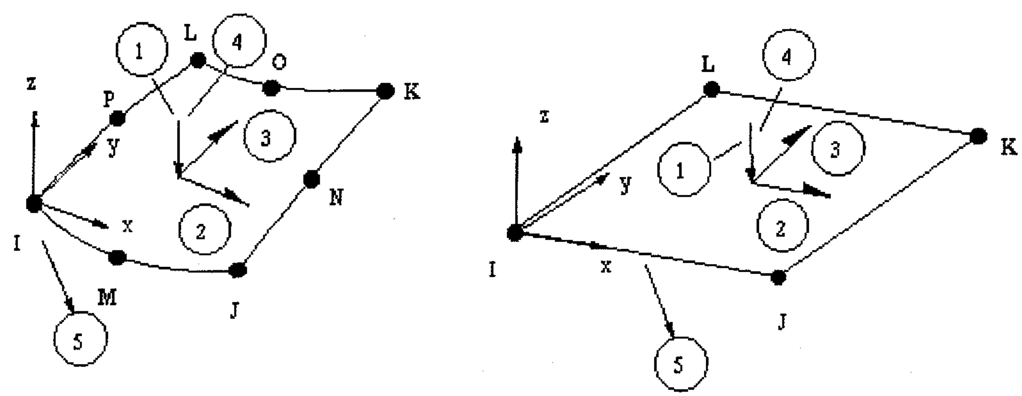
$\left[K_e^{a}\right]$ = **the initial load matrix which includes the effect of changing load orientation (*follower forces*) in the stiffness relation**

Consistent Nonlinear Stiffness

- The following beam and shell elements have a fully consistent tangent stiffness including the initial load matrix (pressure load stiffness) $\left[K_e^a\right]$:
  - <u>Beam Elements</u>: Finite strain beams; Beam188, Beam189
  - <u>Shell Elements</u>: Finite strain shell; Shell181

Pressure Load Stiffness

- To include the pressure load stiffness calculations *for other 3D solid or shell elements* use SURF154*.



Surface effect element; SURF154

An unsymmetric stiffness option is available for the pressure load stiffness. Use only if the rate of convergence is slow.

- *For 2D analyses use Beam188 or Beam189 on model edges for pressure load stiffness calculations.

Incompressibility in Nonlinear Materials

- A consequence of analyzing structural behavior in large strain ranges is the need to deal with nonlinear material behaviors such as nonlinear elasticity (hyperelasticity) and plasticity.
- Rubber-like hyperelastic materials and materials that flow such as plastic solids frequently exhibit a physical phenomenon known as incompressibility (i.e. straining with no volume change).

Origins of Incompressibility in Nonlinear Material Behavior

- In the case of rubbers and rubber-like materials, this incompressibility arises from a Poisson's ratio ν of the natural material approaching 0.5.

- In the case of solids deforming plastically, flow rules often allow little or no volume change. Thus, if plastic strains become extremely large, the material response becomes nearly incompressible in the large strain ranges.

Incompressibility and Mesh Locking

- Whatever the cause of the incompressibility condition, the presence of this effect means that standard, fully integrated, finite elements, when used in large strain analyses, may encounter numerical difficulties associated with mesh locking.

Eliminating Mesh Locking

- In order to cope with the mesh locking associated with the incompressibility effects arising in a large strain analysis, there are various element formulations available:
  - Incompatible Modes
  - Selective Reduced Integration
  - Uniform Reduced Integration
  - Mixed U-P Formulation
- We will discuss mesh locking and the various element technologies in the section on Element Selection.

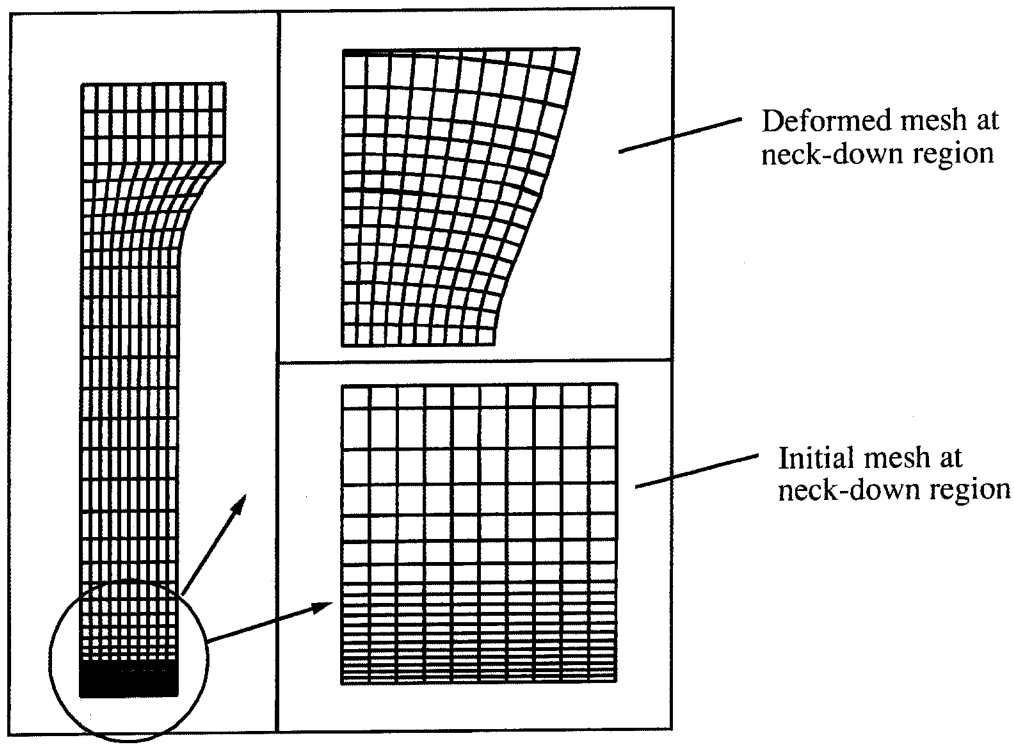
Mesh Distortion

- Each step of a large strain-large rotation analysis updates the coordinates of the nodes using the computed displacements. This “new” mesh is used as the starting point for the next step.
- If the mesh is severely distorted, the solution accuracy of that step is questionable. It is similar to performing a linear analysis with the distorted mesh.
- A distorted mesh can quickly degrade the accuracy of the analysis. There are cases where an element can “turn inside out”.

Mitigating the Effects of Mesh Distortion

- Grade the mesh to account for subsequent distortion.
- Avoid over constraining the deformation at the boundaries.
- Avoid the use of midside node elements which are more susceptible to mesh distortion problems.

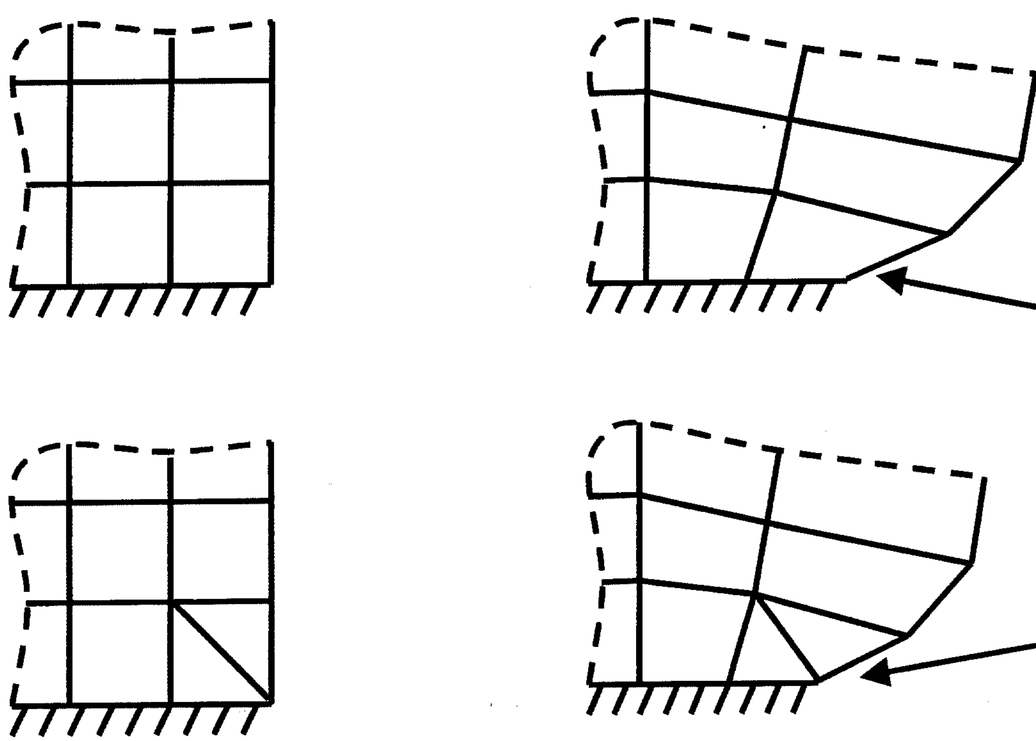
Anticipating Mesh Distortion in a Large Strain Analysis



- **The initial mesh is graded in anticipation of subsequent distortion in the neck-down region of the tensile specimen.**

Anticipating Mesh Distortion in a Large Strain Analysis

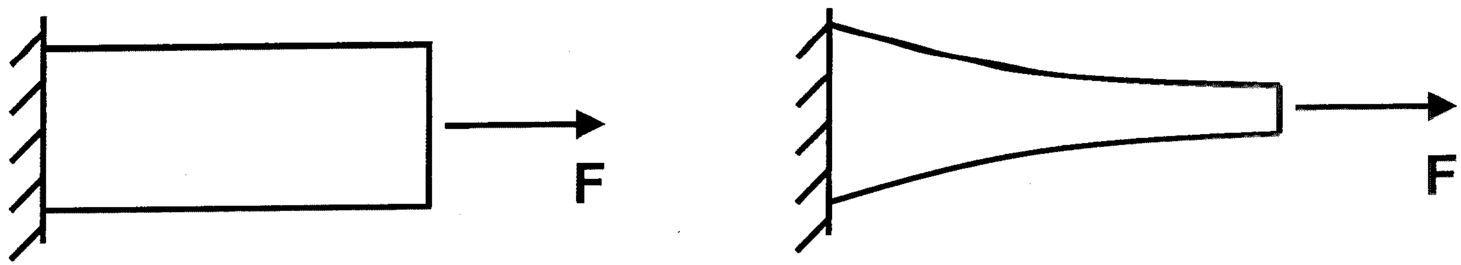
Un-deformed Mesh | **Deformed Mesh**



Large interior angle develops.

Corner elements maintain better shapes as triangles.

Avoid Over Constraining the Deformation at the Boundaries



- By constraining all degrees of freedom at the boundary, very large strains develop due to the Poisson effect.

Avoid the use of Midside Node Elements

- Avoid the use of midside node elements when modeling large strain, large displacement problems.

- Midside node (i.e. higher order) elements are more likely to have geometry updates such that the element's midside node moves completely through the element causing the generation of a negative pivot.

Large Deflection

- As a subset of the full large strain formulation, ANSYS supports large deflection (large rotation) small strain behavior for a subset of elements which do not support large strain behavior.

Large Deflection Theory

- In ANSYS, large deflection theory (as opposed to the full large strain theory) assumes that the deflections and rotations are large, but the mechanical strains (those that cause stresses) are small. Each element is not assumed to change shape except for rigid body motions.
- Large deflection theory is a subset of large strain theory which is available in many of ANSYS' older beam, shell and other nonlinear elements.

Large Deflection Implementation in ANSYS

- The procedure for large deflection theory is based on a corotational approach; i.e. a coordinate system, initially parallel to the element coordinate system, is "pasted" onto the element. This coordinate system will rotate as the element rotates.

- In this approach, material (or element) rotation is accounted for exactly. Strains are computed in the rotated system using the usual small strain (engineering strain) assumption.

Large Deflection Displacement, Stress, and Strain Directions

- Calculated displacements are output in the original directions because nodal coordinate system orientations are not updated in a large deflection analysis.

- Stress and strains in a large deflection (small strain) analysis are engineering values in the rotated coordinate system; i.e. the element coordinate system follows the rotation of the element.

When to Activate Large Deflection Effects

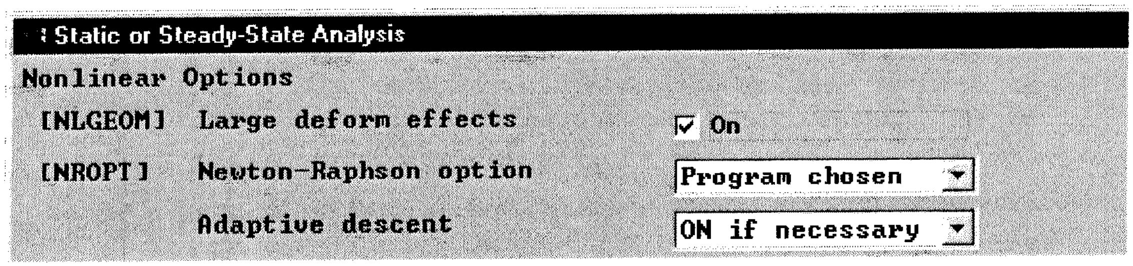
- Large deflection effects should be activated whenever element rotations become "large" enough to significantly affect the solution accuracy.

- Unfortunately, there are no absolute guidelines as to how large is "large". The difference between "small" and "large" deflections turns out to be highly problem dependent.

Obtaining the Solution

- **To activate a large strain solution and/or a large deflection for the elements that support this feature,**

  Solution > Analysis Option ...

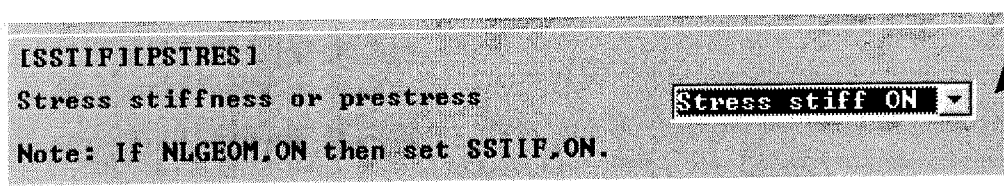


- **NLGEOM,ON activates the large strain option in those elements which support it. If you are using an element which only supports large deflection, NLGEOM will activate a large deflection solution. Refer to the *ANSYS Elements Manual.***

Obtaining the Solution

- With solution control active, turning on nonlinear geometry (NLGEOM) will by default include the stress stiffening terms in the nonlinear stiffness matrix (includes $[K^{\sigma}]$ in the nonlinear stiffness matrix $[K^{nl}]$). As an option you can select not to include $[K^{\sigma}]$ in the formulation of the nonlinear stiffness matrix for a subset of older elements.

Solution > Analysis Options ...



Rarely deactivated as including stress stiffening leads to a fully consistent $[K^{nl}]$.

This command has no effect for elements 106, 107, 108, 181, 182, 185, 188, and 189!

Obtaining the Solution

- The use of solution control is recommended (default).
- The full Newton-Raphson option without adaptive descent is the recommended Newton-Raphson option (default with solution control).
- The use of automatic time stepping is recommended (default with solution control). Be sure to set a small enough minimum time step for automatic time stepping.
- The line search option (LNSRCH) can also be helpful for oscillating convergence behavior.

Trouble Shooting

- In a large strain analysis anticipate mesh distortion, and grade the mesh appropriately. Refer to the guidelines on anticipating mesh distortion.
- Use the appropriate element type and integration rule to alleviate problems with mesh locking. (More information on this topic in the Element Selection section.)
- Include the pressure load stiffness (SURF154) if using elements which do not support this directly. If the rate of convergence is slow, activate the unsymmetric option KEYOPT(5)=1.

Trouble Shooting

- Time step sizes should be such that a maximum rotation per substep is less than five or ten degrees.
- Do not use the predictor for large rotation analyses.
- Use a sufficient mesh density with beam and shell elements; no one element should experience more than 30 degrees of flexure.
- If the automatic time stepping is bisecting repeatedly, this may be a sign of a physical instability. Plot the load deflection response.

NOTES

Chapter 4

STRUCTURAL STABILITY

Session Objective

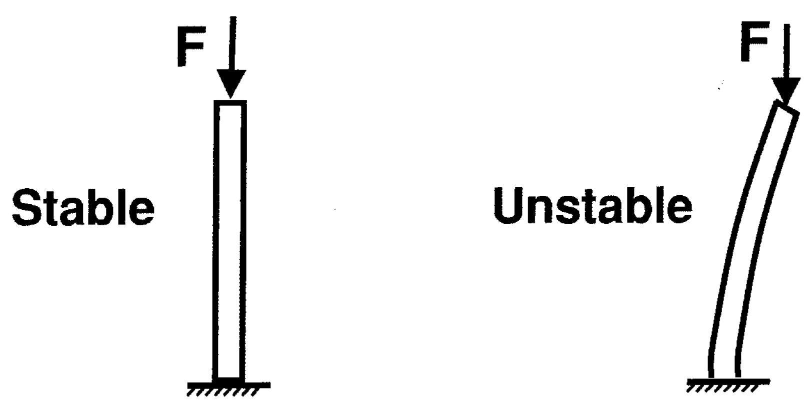
- At the end of this session you will be able to describe and demonstrate the following:

  1. Structural Stability
  2. Load Control, Displacement Control, Arc-Length
  3. Analysis Techniques for Pre-Buckling Analysis
  4. Analysis Techniques for Post-Buckling Analysis
  5. Trouble Shooting

Structural Stability

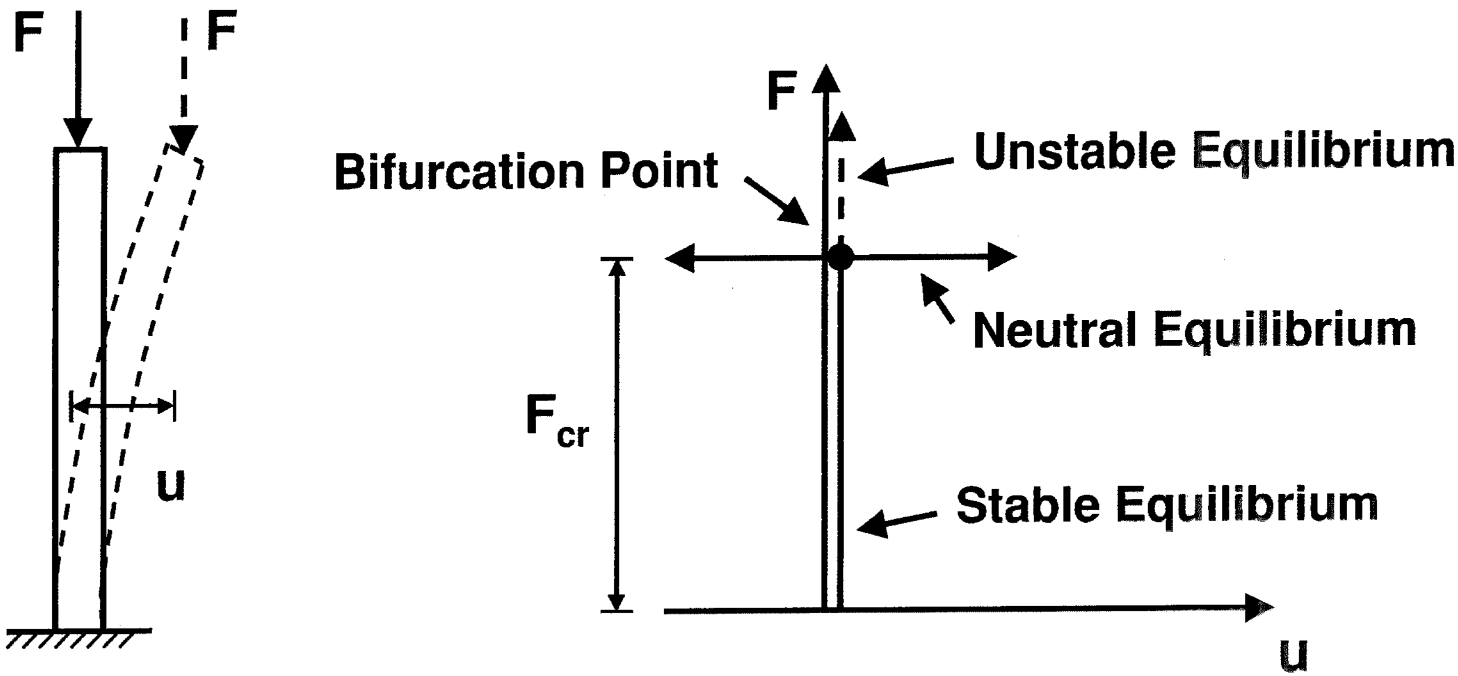
- Many structures require an evaluation of their structural stability. Thin columns, compression members, and vacuum tanks are all examples of structures where stability considerations are important.

- At the onset of instability (buckling) a structure will have a very large change in displacement $\{\Delta u\}$ under essentially no change in the load (beyond a small load perturbation).



Structural Stability

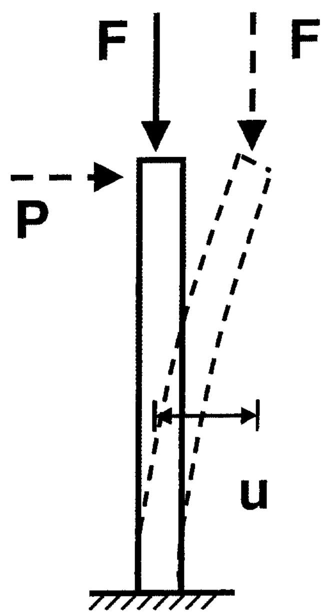
- An idealized fixed end column will exhibit the following behavior under increasing axial loads (F).



Structural Stability

Bifurcation Point

- A bifurcation point is a point in load history where two branches of the solution are possible.

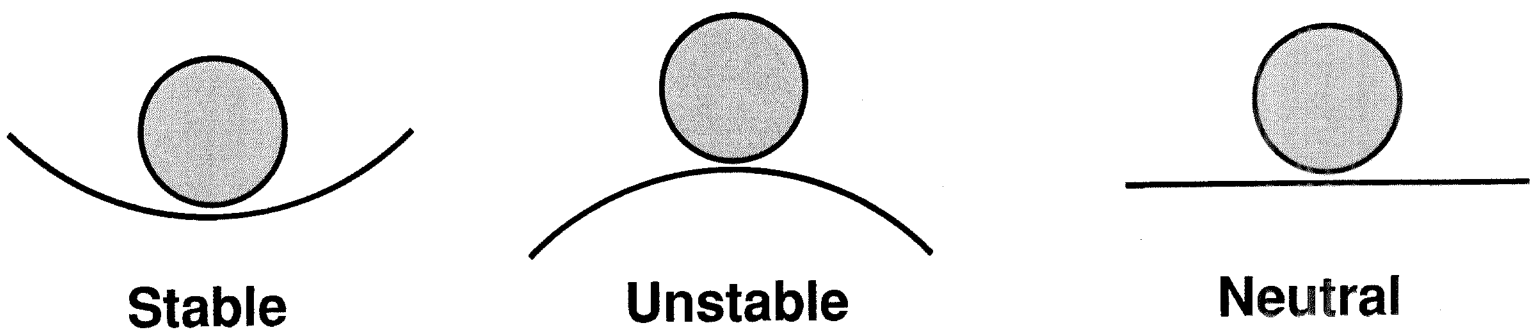


- In the case of the idealized fixed end column, at the critical load (F_{cr}), the column can buckle to the left or to the right. Thus two load paths are possible. In the case of real structures the existence of geometric imperfections or force perturbations ($P \neq 0$) will determine the direction of the load path.

Structural Stability

Stable, Unstable, and Neutral Equilibrium

- Consider the equilibrium of the ball shown below. If the surface is concave upward the equilibrium is stable, the ball will return to its original position if perturbed. If the surface is concave downward the equilibrium is unstable, if perturbed the ball will roll away. If the surface is flat the ball is in neutral equilibrium, if perturbed the ball will remain in its new position.



Structural Stability

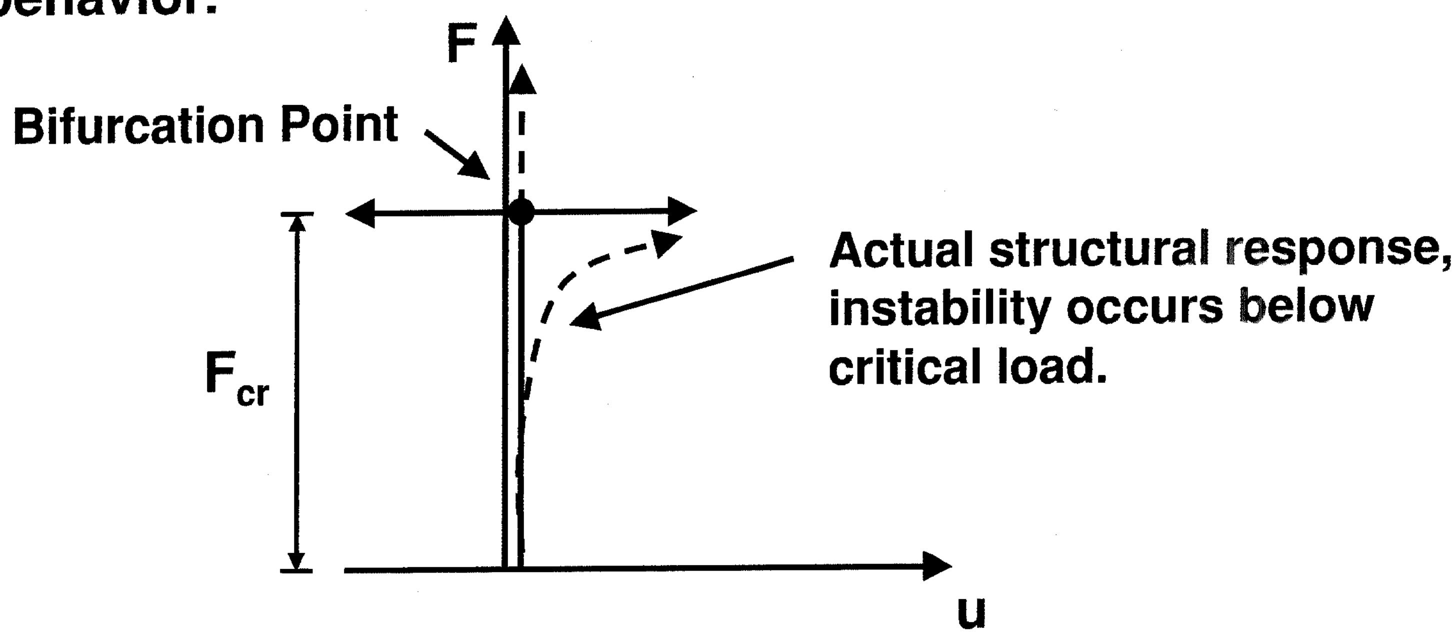
Critical Load

- At a value of $F < F_{cr}$ the column is in stable equilibrium. If a small perturbing force ($P \neq 0$) is introduced and then removed, the column will return to its original position. At values $F > F_{cr}$ the column is in unstable equilibrium, any perturbing force will cause collapse. At $F = F_{cr}$ the column is in neutral equilibrium, and this is defined as the critical load.

Structural Stability

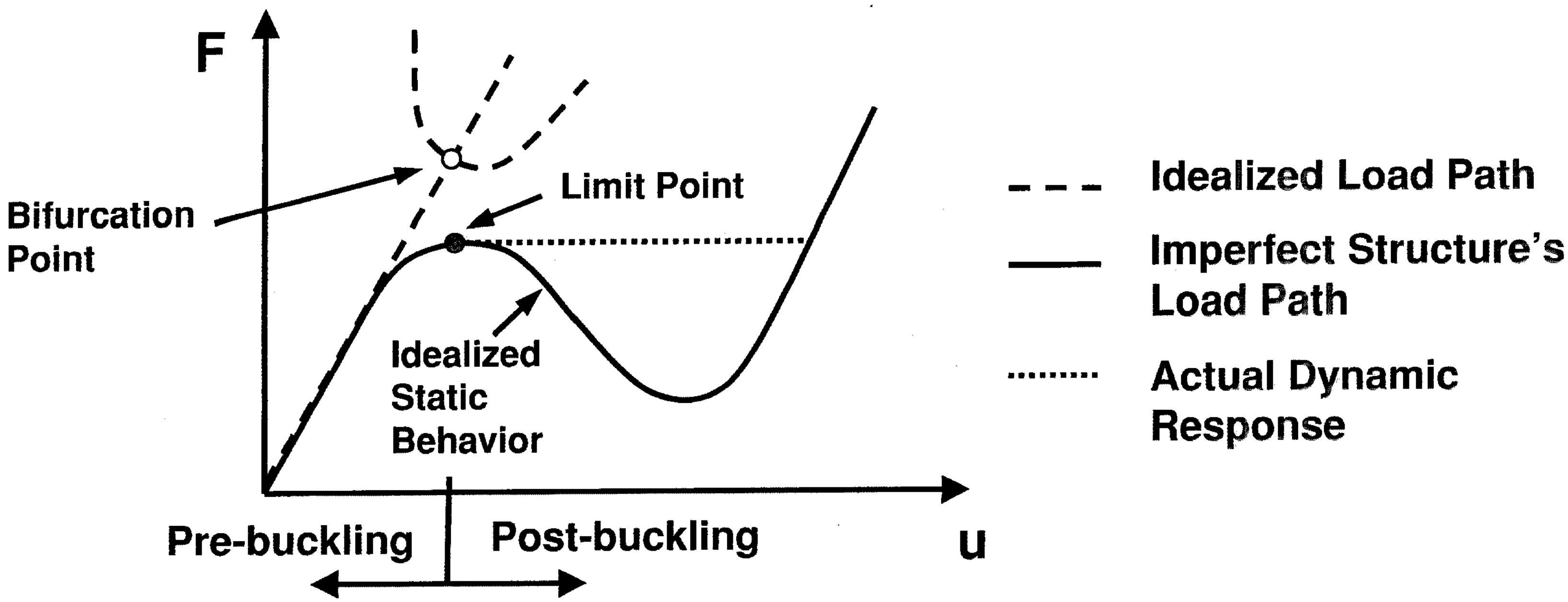
Limit Load

- In real structures the critical load can rarely be achieved. A structure generally will become unstable at a load lower than the critical load because of imperfections and nonlinear behavior.



Structural Stability

- Shown below is a generalized nonlinear load deflection curve. This figure illustrates the idealized load path, an imperfect structure's load path, and the actual dynamic response of the structure.

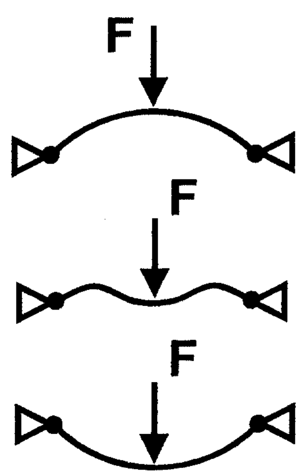


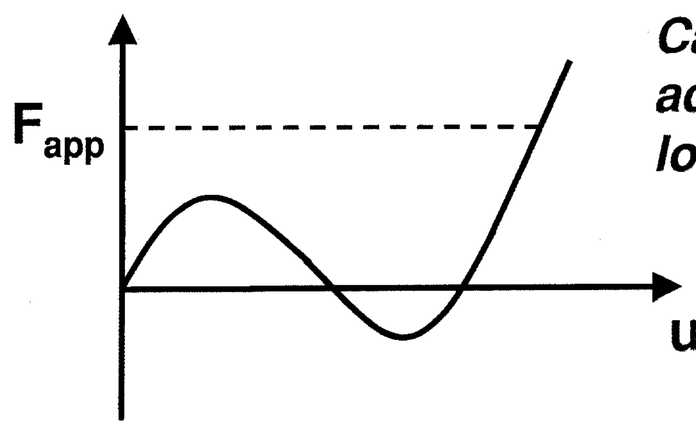
Load Control, Displacement Control, and Arc-Length Method

- There are various analysis techniques available for calculating the static force deflection response of a structure. These techniques include:
  - Load Control
  - Displacement Control
  - Arc-Length Method

Load Control

- Consider the snap through analysis of the shallow arch shown below. When the solution to a problem is performed with incrementally applied forces (F) the solution is performed using *load control*.

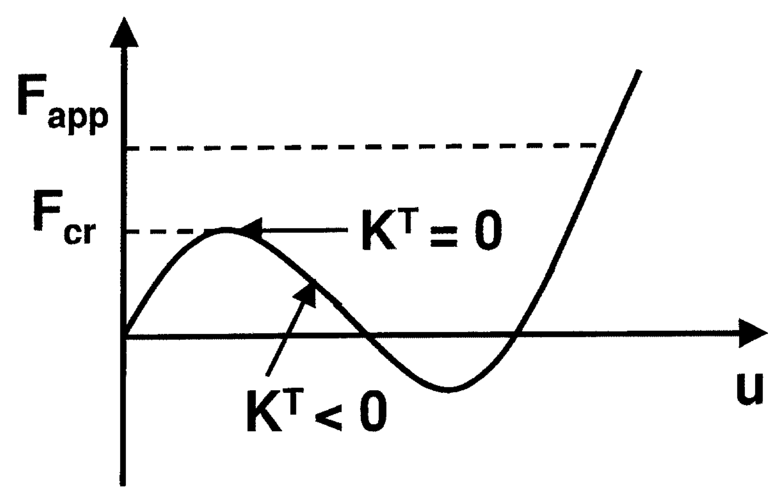




Can F_{app} be achieved with load control?

Load Control

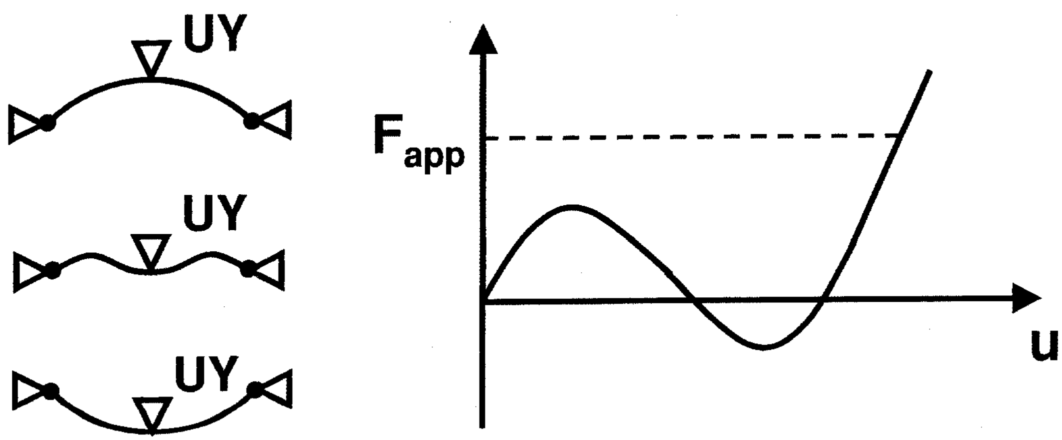
- The difficulty of using load control with the Newton-Raphson is that the solution can not progress past a point of instability. At the point of instability (F_{cr}) the tangent stiffness matrix K^T is singular. Using load control, the Newton-Raphson method will not converge. However, this type of analysis can be useful to characterize the pre-buckling behavior of a structure.



Only F_{cr} can be achieved using load control.

Displacement Control

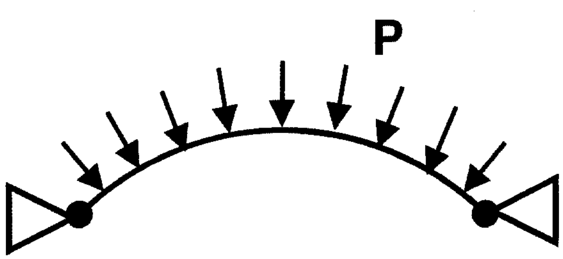
- When the arch is loaded with an incrementally applied displacement, as opposed to a force, the solution is performed using *displacement control*. The advantage of displacement control is that it produces a stable solution beyond F_{cr}. (The imposed displacement provides an additional constraint at the point of instability.)



F_{app} can be achieved with displacement control. (F_{app} is now the reaction force at the imposed displacement UY.)

Displacement Control

- **The disadvantage of displacement control is that it only works when you know what displacements to impose! If the arch is loaded with a pressure load as opposed to a concentrated force, displacement control is not possible.**

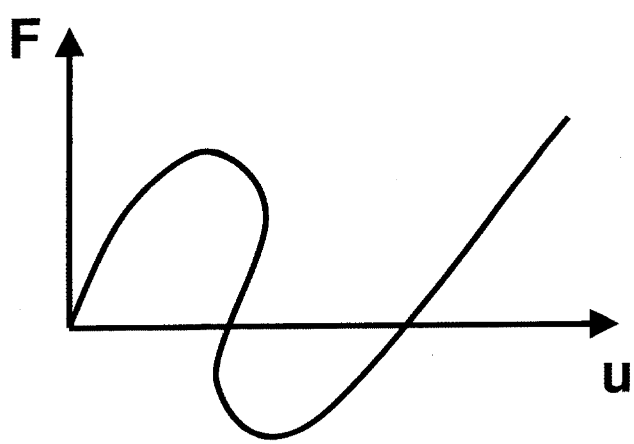


With a more complicated loading it is generally not clear which displacements to impose.

Arc-Length Method

- The *arc-length* is a solution method used to obtain numerically stable solutions for problems with instabilities ($K^T \to 0$), or negative tangent stiffnesses ($K^T < 0$).

- The arc-length method can be used for *static* problems with *proportional loading*.



- Although the arc-length method can solve problems with a complicated force-deflection response, it is best suited to solve problems where the response is smooth without sudden bifurcation points.

Arc-Length Method

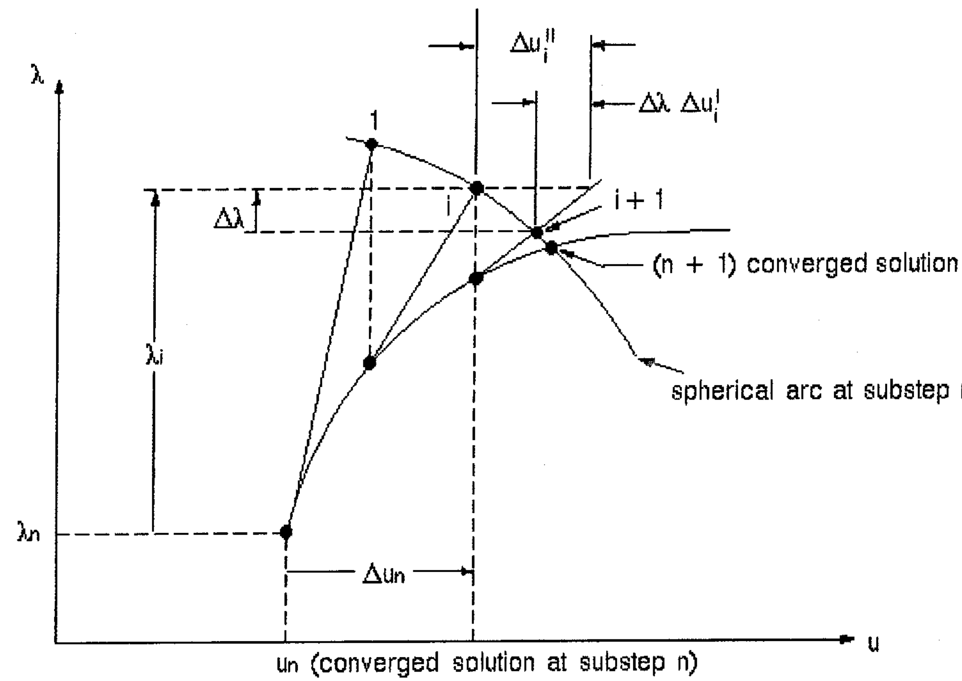
- The arc-length method solves for load and displacement simultaneously in a Newton-Raphson solution by introducing an additional unknown in the solution, the load factor λ ($-1 < \lambda < 1$). The Newton-Raphson equation can then be rewritten as,

$$[K^T]\{\Delta u\} = \lambda \{F^a\} - \{F^{nr}\}$$

- In order to accommodate the additional unknown a constraint equation must be introduced, the arc-length ℓ. The arc-length relates the load factor λ and displacement increments $\{\Delta u\}$ in the Newton-Raphson solution.

Arc-Length Method

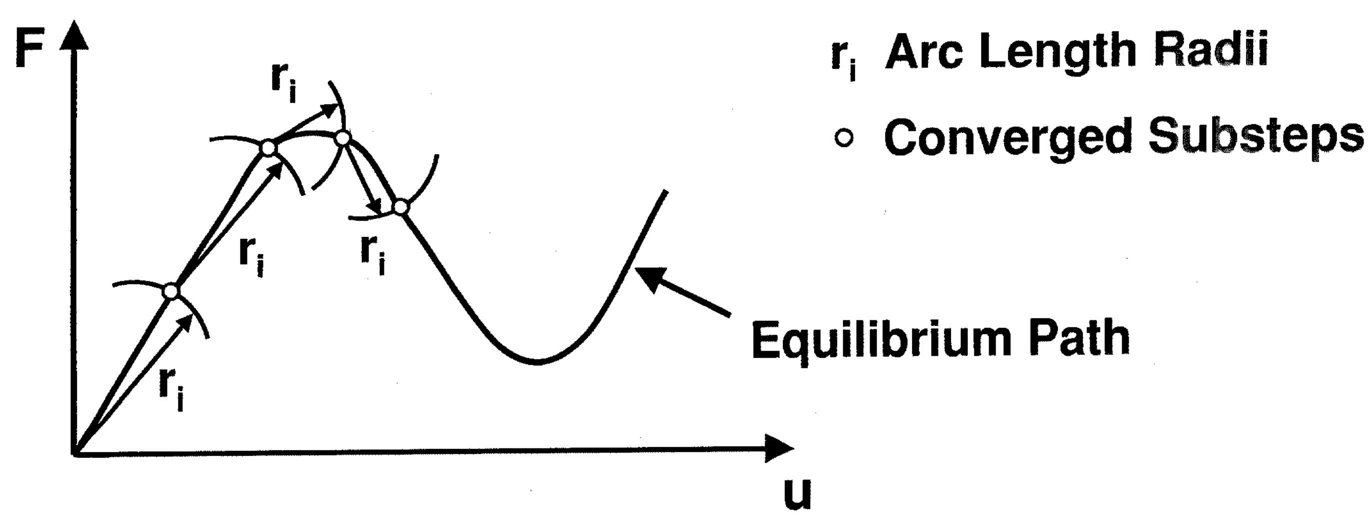
The arc-length relates the incremental load factor $\Delta\lambda$ to the incremental displacement Δu via a spherical arc. Shown here are the incremental load factor $\Delta\lambda$ and the incremental displacement Δu for the arc-length method with the full Newton-Raphson.



$$Arc\ Length\ Radius = \sqrt{\Delta u_n^2 + \lambda^2}$$

Arc-Length Method

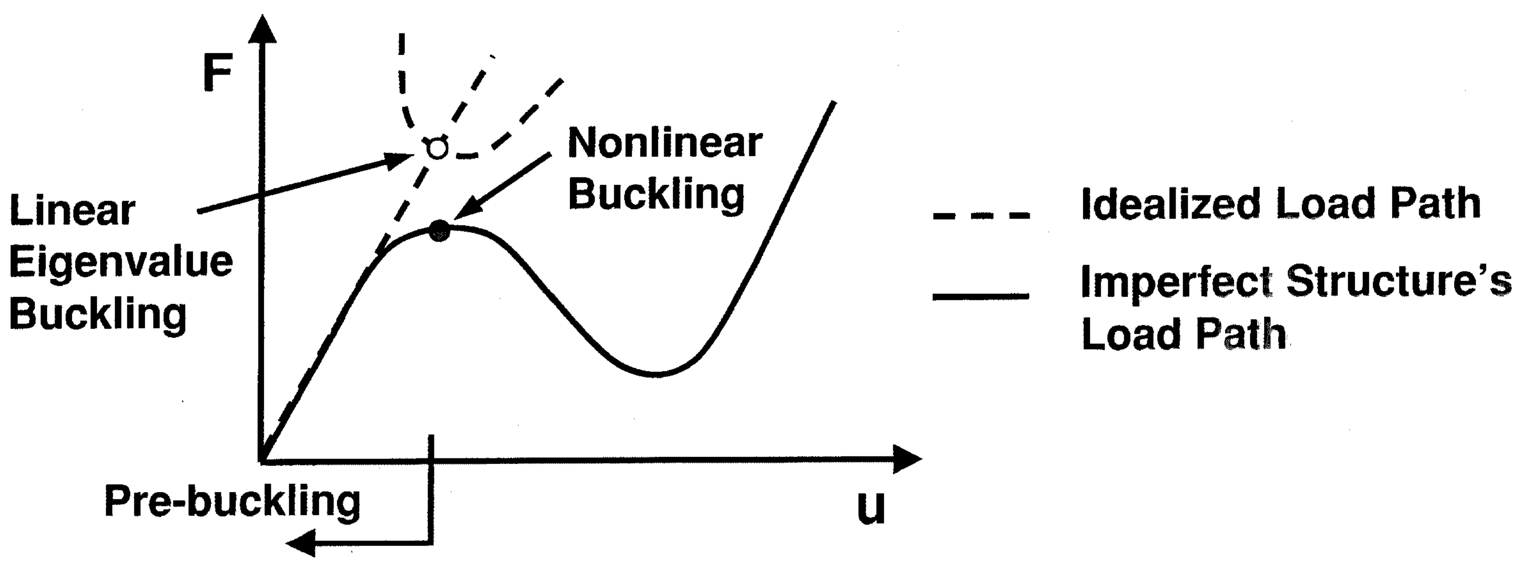
- By forcing the Newton-Raphson iterations to converge along a spherical arc which intersects the equilibrium path, solutions can be obtained for structures undergoing zero or negative stiffness behaviors.



Pre-Buckling Analysis

Analysis techniques for pre-buckling and collapse load analysis include:

- **Linear Eigenvalue Buckling**
- **Nonlinear Buckling Analysis**



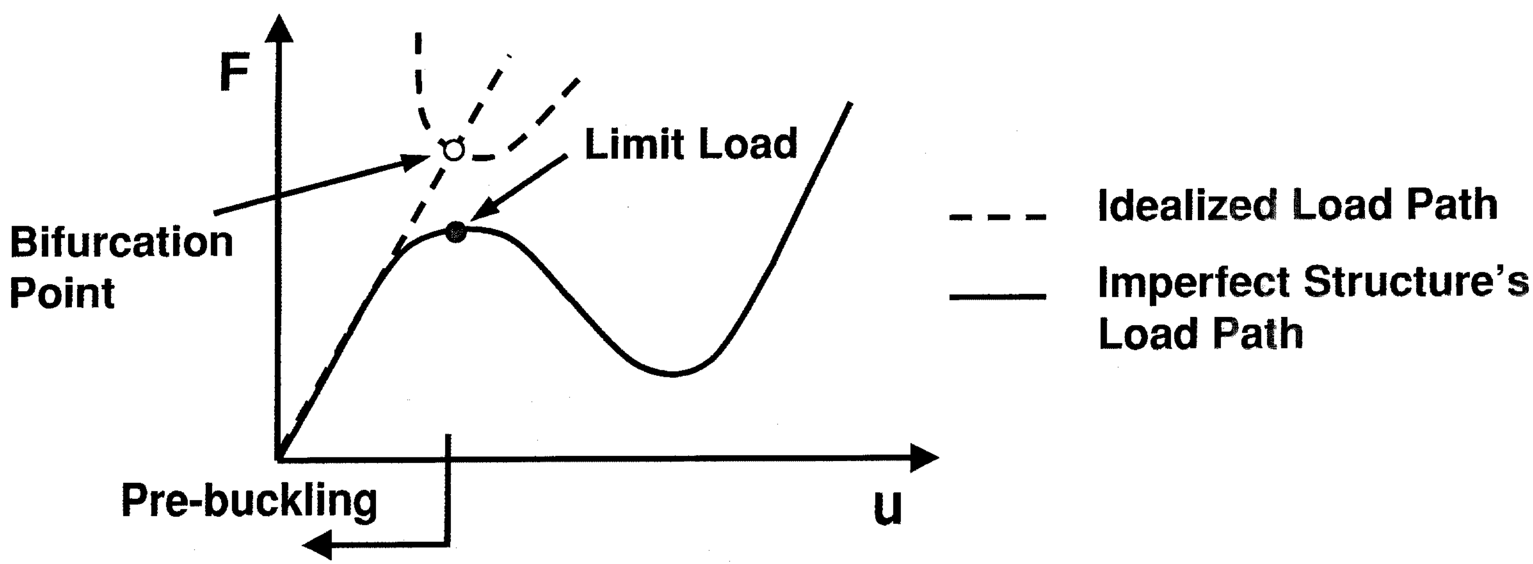
Eigenvalue Buckling

- *Eigenvalue buckling analysis* predicts the theoretical buckling strength (the bifurcation point) of an *ideal linear elastic* structure.

- The eigenvalue formulation determines the bifurcation points of a structure. This method corresponds to the textbook approach of linear elastic buckling analysis. The eigenvalue buckling solution of a Euler column will match the classical Euler solution.

Eigenvalue Buckling

- **However, imperfections and nonlinear behavior prevent most real world structures from achieving their theoretical elastic buckling strength. Eigenvalue buckling generally yields *unconservative* results, and should be used with caution.**



Eigenvalue Buckling

- Although eigenvalue buckling typically yields unconservative results there are two advantages to performing a linear buckling analysis:
  - Relatively inexpensive (fast) analysis.
  - The buckled mode shapes can be used as an initial geometric imperfection for a nonlinear buckling analysis.

The Basis of Eigenvalue Buckling

A linear buckling analysis is based on a classic eigenvalue problem. To develop the eigenvalue problem, first solve the load-displacement relationship for a linear elastic pre-buckling load state $\{P_0\}$; i.e. given $\{P_0\}$ solve for

$$\{P_0\} = [K_e]\{u_0\}$$

to obtain

$\{u_0\}$ = the displacements resulting from the applied load $\{P_0\}$, and

$\{\sigma\}$ = the stresses resulting from $\{u_0\}$

The Basis of Eigenvalue Buckling

Assuming the *pre-buckling displacements are small*, the incremental equilibrium equations at an *arbitrary* state ({P}, {u}, {σ}) are given by

$$\{\Delta P\} = \left[[K_e] + [K_\sigma(\sigma)]\right]\{\Delta u\}$$

where

$[K_e]$ = elastic stiffness matrix

$[K_\sigma(\sigma)]$ = *initial stress matrix* evaluated at the stress state $\{\sigma\}$

The Basis of Eigenvalue Buckling

Assuming pre-buckling behavior is a linear function of the applied load $\{P_0\}$,

$$\{P\} = \lambda\{P_0\} \qquad \{u\} = \lambda\{u_0\} \qquad \{\sigma\} = \lambda\{\sigma_0\}$$

then we can show that

$$[K_\sigma(\sigma)] = \lambda[K_\sigma(\sigma_0)]$$

Thus, the incremental equilibrium equations expressed for the *entire pre-buckling range* become

$$\{\Delta P\} = [[K_e] + \lambda[K_\sigma(\sigma_0)]]\{\Delta u\}$$

The Basis of Eigenvalue Buckling

At the *onset of instability* (the buckling load $\{P_{cr}\}$), the structure can exhibit a change in deformation $\{\Delta u\}$ in the case of

$$\{\Delta P\} \approx 0$$

By substituting the above expression into the previous incremental equilibrium equations for the *pre-buckling range* we have

$$[[K_e] + \lambda[K_\sigma(\sigma_0)]]\{\Delta u\} = \{0\}$$

The above relation represents a *classic eigenvalue problem.*

The Basis of Eigenvalue Buckling

In order to satisfy the previous relationship, we must have

$$\det[[K_e] + \lambda[K_\sigma(\sigma_0)]] = 0$$

In a finite element model with n degrees of freedom, the above equation yields an n^{th} order polynomial in λ (the eigenvalues). The eigenvectors $\{\Delta u\}_n$ in this case represent the deformation superimposed on the system during buckling. The elastic critical load $\{P_{cr}\}$ is given by the lowest value of λ calculated.

Eigenvalue Buckling Procedure

- An eigenvalue buckling analysis includes the following four main steps:
  1. Build the Model
  2. Obtain the Static Solution with Prestress
  3. Obtain the Eigenvalue Buckling Solution
  4. Review the Results

Eigenvalue Buckling Procedure

Build the Model

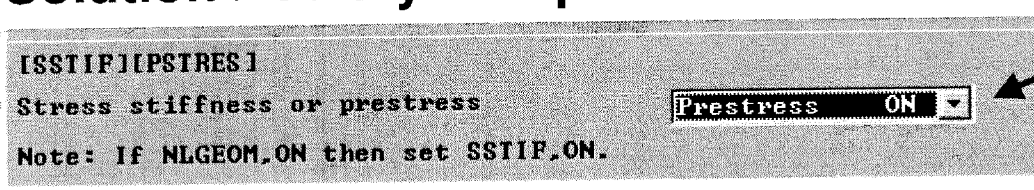
- This task is similar to most other analyses with the following two additional points:
  - Only linear behavior is valid. Nonlinear elements are treated as linear. Their stiffnesses are based on their initial status and are not changed.
  - Young's Modulus must be defined. Material properties may be linear, isotropic or orthotropic, nonlinear properties are ignored.

Eigenvalue Buckling Procedure

Obtain the Static Solution with Prestress

When obtaining the static solution the prestress flag *must* be set to perform a subsequent eigenvalue buckling analysis.

Solution > Analysis Options ...



Set *PSTRES,ON.* This causes the stress stiffness matrix to be stored for the eigenvalue buckling solution.

Eigenvalue Buckling Procedure

Obtain the Static Solution with Prestress

- Unit loads are usually sufficient. The eigenvalues calculated represent the buckling load factors on the applied load.
- Note that the eigenvalues represent scale factors for all loads. If certain loads are constant while other loads are variable you will need to ensure that the stress stiffness matrix from the constant loads is not factored (discussed later).

Eigenvalue Buckling Procedure

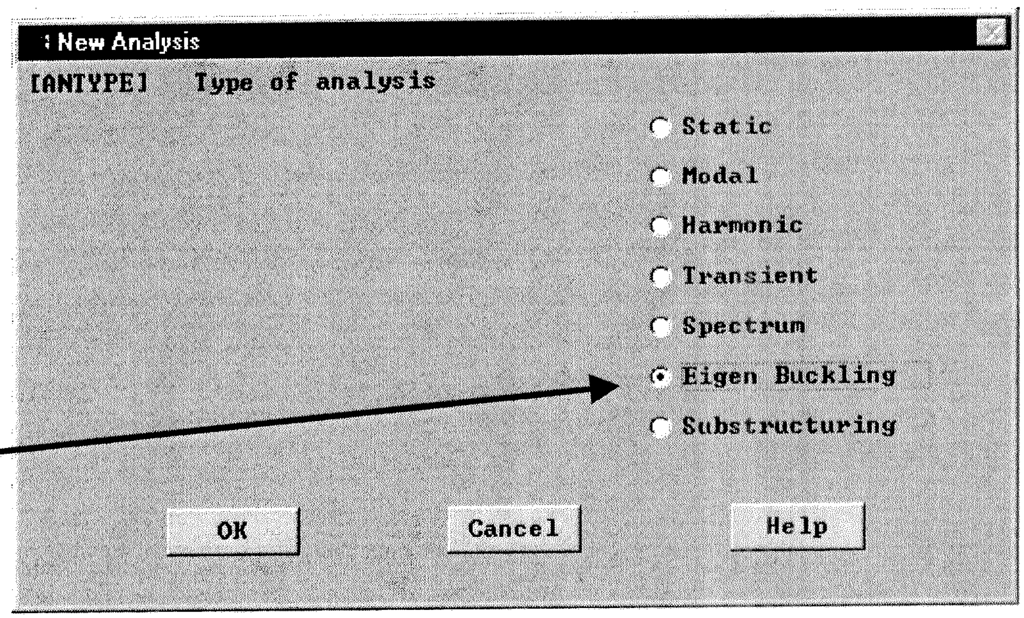
Obtain the Eigenvalue Buckling Solution

After completing the static solution, exit and re-enter solution, and specify eigenvalue buckling as the analysis type:

Solution >

New Analysis ...

Eigenvalue Buckling



Eigenvalue Buckling Procedure

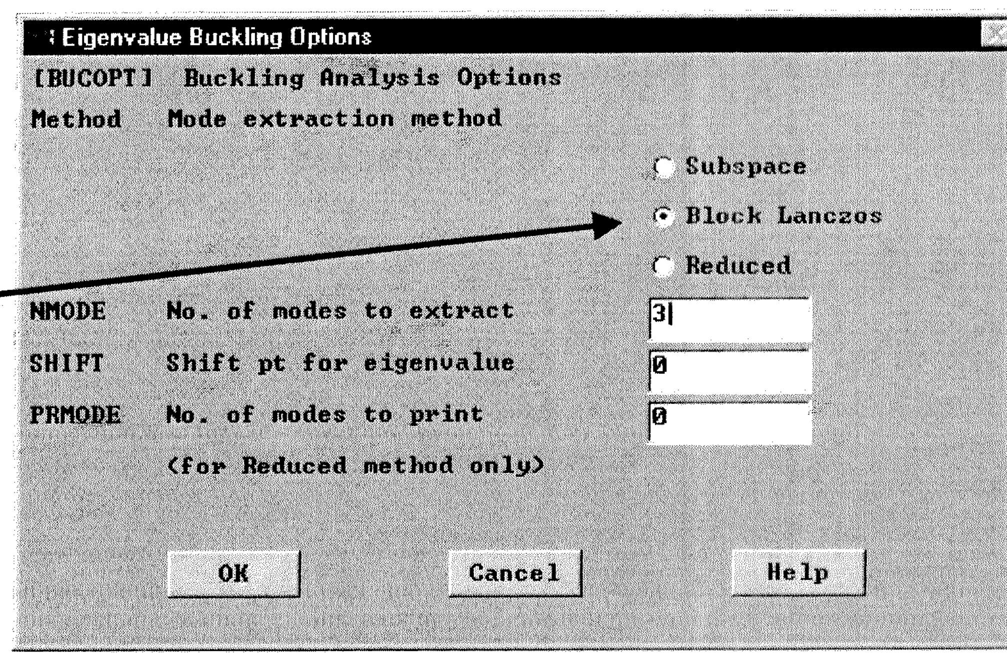
Obtain the Eigenvalue Buckling Solution

Specify the method of eigenvalue extraction and number of buckled modes to extract:

Solution >

Analysis Options ...

Block Lanczos is the recommended option.



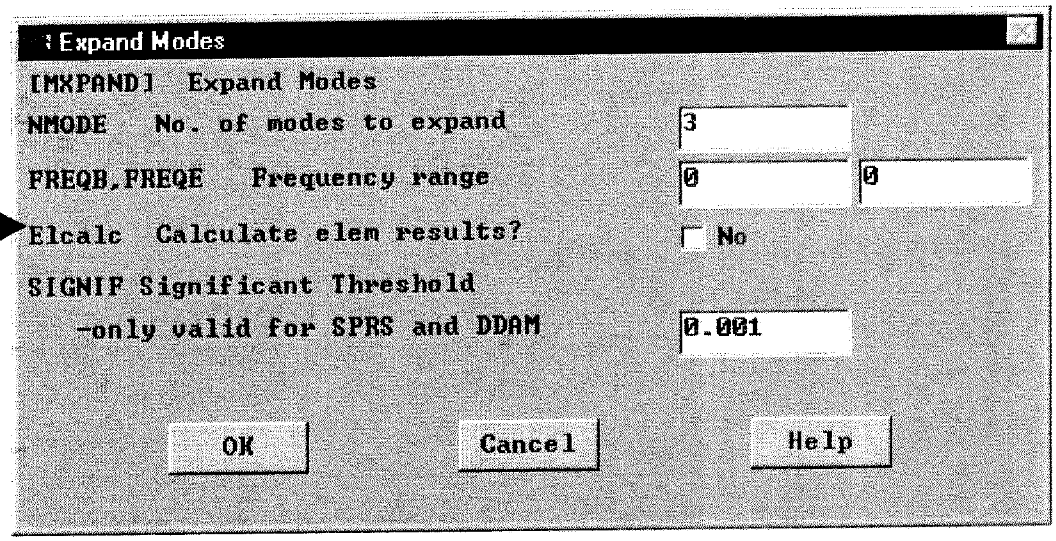
Eigenvalue Buckling Procedure

Obtain the Eigenvalue Buckling Solution

Specify the number of modes to write to the results file.

Solution > -Load Step Opts - Expansion Pass > Expand Modes ...

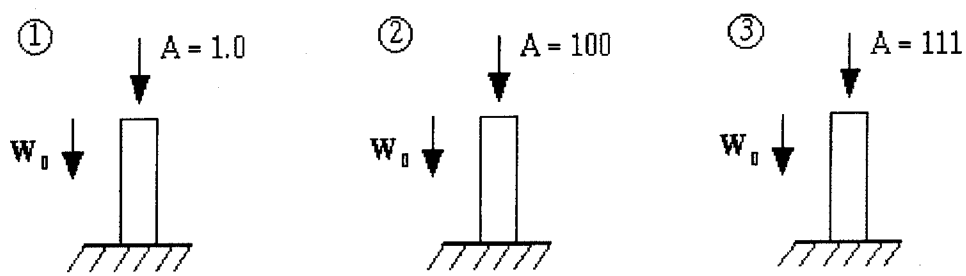
The relative stress distribution can also be calculated.



Eigenvalue Buckling Procedure

Note on Constant and Variable Loads

You can iterate on the eigensolution, adjusting the variable loads until the eigenvalue becomes 1.0 or nearly 1.0. Consider the example of a pole with self weight W_O and an externally applied load A. You can iterate, adjusting the value of A until $\lambda = 1.0$.



$\lambda = 100$:	$\lambda = 1.1$:	$\lambda = 0.99$:
$F = 100 + 100\ W_0$	$F = 110 + 1.1\ W_0$	$F = 110 + .99\ W_0$

Eigenvalue Buckling Procedure

Review the Results

- Results from an eigenvalue buckling analysis can be reviewed in the General Postprocessor. The results consist of load factors, buckled mode shapes, and the relative stress distributions.

- The maximum displacement of a buckled mode shape is normalized to 1.0. Therefore, the displacements do not represent actual deformations and the stresses are relative to the buckled mode shape.

Eigenvalue Buckling Procedure

Review the Results

- It is usually beneficial to review the first few buckling mode shapes. In a subsequent nonlinear buckling analysis the higher buckling modes of the structure may be important.

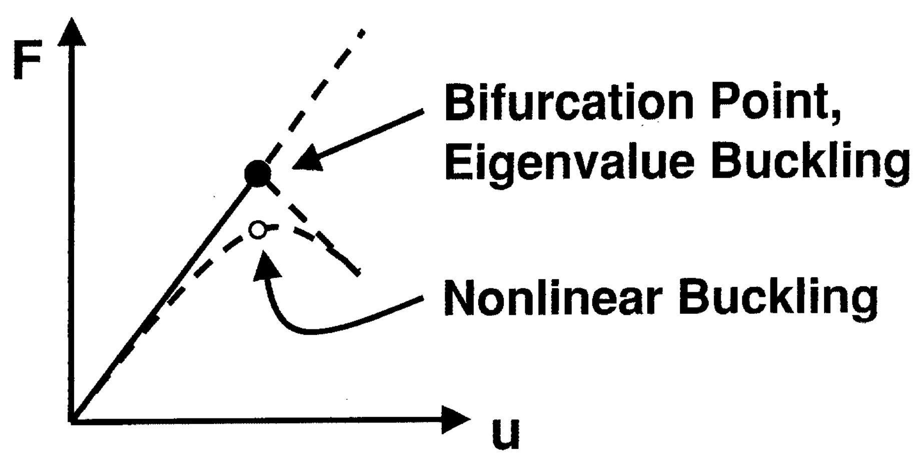
- If there are closely spaced eigenvalues, this indicates that the structure is imperfection sensitive. A nonlinear buckling analysis should be executed with the appropriate imperfection or perturbation.

Trouble Shooting

- In some cases negative eigenvalues are computed in an eigenvalue buckling analysis. This occurs when the eigenvalue extraction procedure encounters numerical difficulties. In this case, the shift point for eigenvalue extraction can be specified (BUCOPT). Eigenvalue extraction is most accurate near the shift point. This will require that you have some knowledge of the value of the critical load.

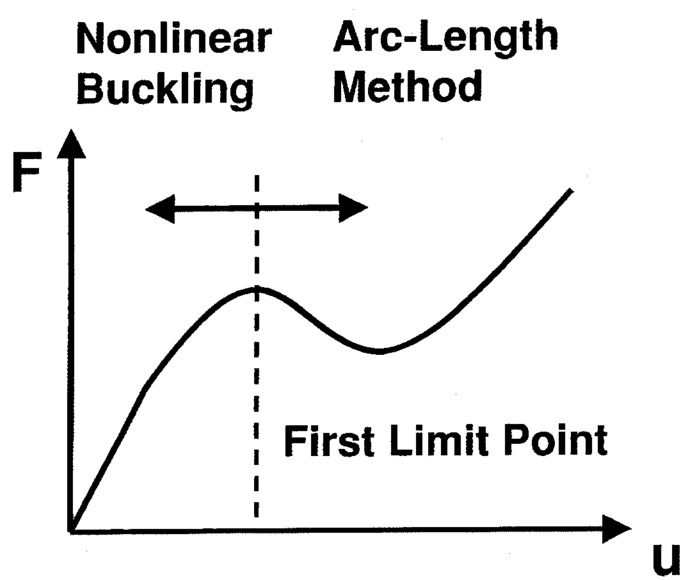
Nonlinear Buckling

- A *nonlinear buckling* analysis employs a nonlinear static analysis with gradually increasing loads to seek the load level at which a structure becomes unstable.
- Using a nonlinear buckling analysis, you can include features such as initial imperfections, plastic behavior, contact, large-deformation response, and other nonlinear behavior.



Nonlinear Buckling

- In a nonlinear buckling analysis, the goal is to find the first limit point (the largest value of the load before the solution becomes unstable). The arc-length method can be used to follow the post-buckling behavior.



- Nonlinear buckling is more accurate than eigenvalue buckling, and is therefore recommended for the design or evaluation of structures.

Nonlinear Buckling Procedure

- A nonlinear buckling analysis includes the following three main steps:
  1. Build the Model

     (Including an initial imperfection or perturbation)
  2. Obtain the Solution
  3. Review the Results

Nonlinear Buckling Procedure

Build the Model

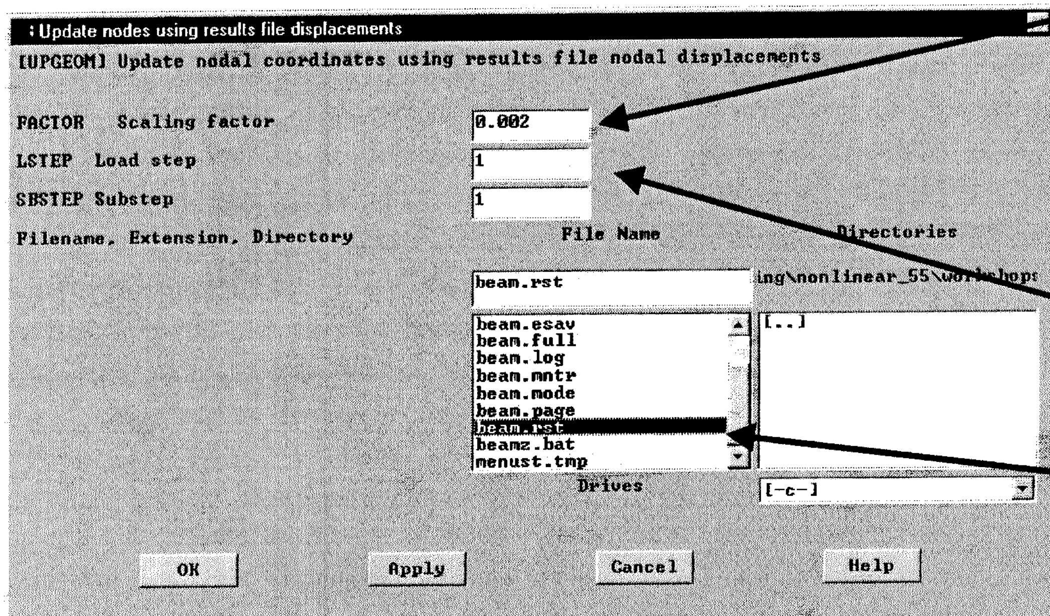
- This task is similar to most other analyses with the following additional points:
  - A small perturbation (such as a small force) or geometric imperfection is required to initiate buckling.
  - The buckled mode shape from an eigenvalue buckling analysis can be used to generate an initial imperfection.
  - The value of the applied load should be set to a value slightly higher (10 to 20%) than critical load predicted by the eigenvalue buckling analysis.

Nonlinear Buckling Procedure

Build the Model - Initial Imperfection

Creating an initial imperfection from the buckled mode shape.

Preprocessor > Update Geometry ...



Multiplier for the displacements to be added to the original geometry.

Mode Number

Results file from the eigenvalue buckling analysis.

Nonlinear Buckling Procedure

Build the Model - Initial Imperfection

- The magnitude of the initial imperfection will influence the results of the nonlinear buckling analysis. The initial imperfection will remove the sharp discontinuity in the load-deflection response.

- The value of the imperfection should be small relative to the overall dimensions of the structure. The value should match the size of the imperfection (real or postulated) in the real structure. Manufacturing tolerances can be used to estimate the magnitude of the imperfection.

Nonlinear Buckling Procedure

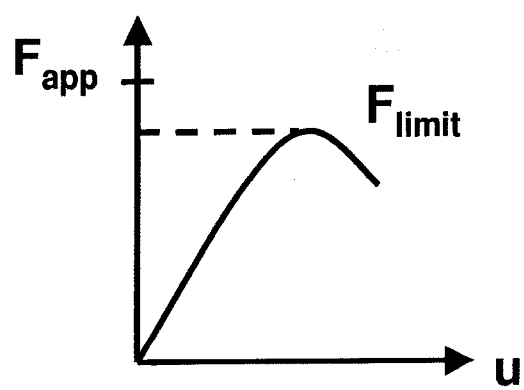
Obtaining the Solution

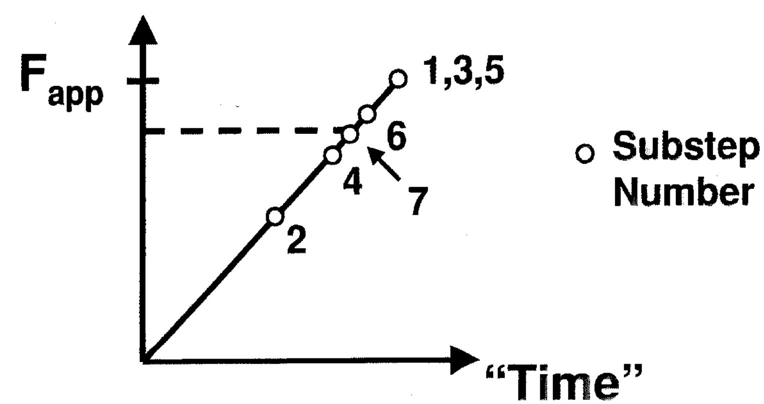
- A nonlinear buckling analysis is a static analysis with nonlinear geometric effects extended to the point where the structure reaches its limit load. Be sure that nonlinear geometry is activated (NLGEOM,ON).
- The use of solution control is recommended (default).
- Use the full Newton-Raphson option *without* adaptive descent (default with solution control). (Adaptive descent, which uses the secant matrix to mathematically stabilize the solution, can sometimes cause the solution to proceed beyond the limit point - yielding an "incorrect" solution!)

Nonlinear Buckling Procedure

Obtain the Solution

- Invoke automatic time stepping (default with solution control). With automatic time stepping on, the program *automatically* seeks out the buckling load. If the solution does not converge at a given load, the program bisects and attempts a new solution at a smaller load. As such the *minimum time step* will affect the precision of your results.

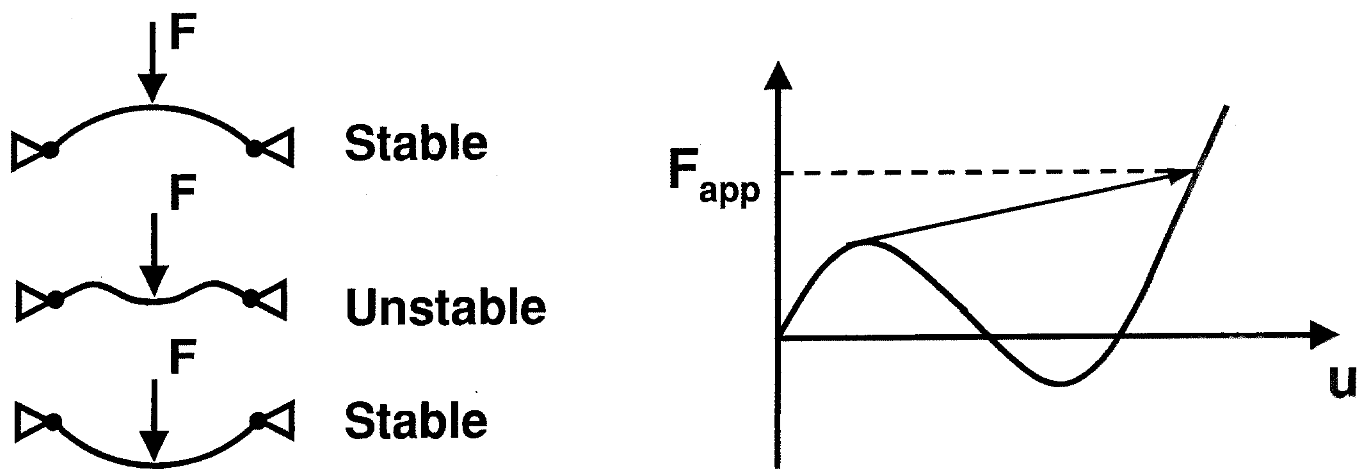




Nonlinear Buckling Procedure

Obtaining the Solution

- It is possible (although unlikely) that if the time steps are large you might "jump over" the instability and obtain a solution for a "snapped through" configuration. Be sure to plot the load deflection curve in the time history postprocessor.



Nonlinear Buckling Procedure

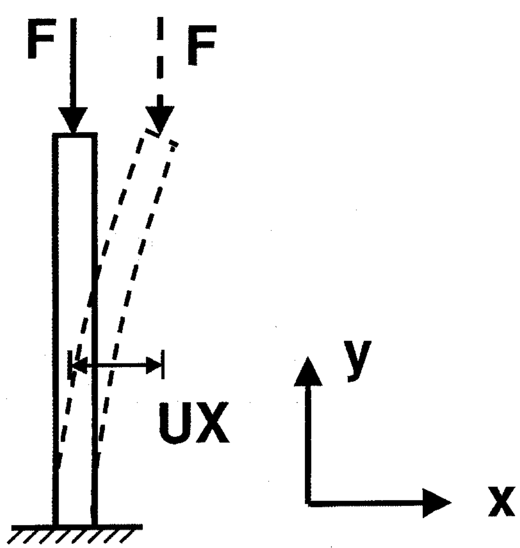
Obtain the Solution

- Be sure to set a small *minimum time step* to allow bisection.
- A value 10 to 20 percent higher than the eigenvalue buckling load will often be a good choice for the value of the applied load. Recall that you can set "time" equal to the value of the applied load for easier postprocessing.
- Be sure to write out results for a sufficient number of substeps (OUTRES), so that you can examine the load deflection curve in the general postprocessor.

Nonlinear Buckling Procedure

Review the Results

- Review the load deflection curve in the time history postprocessor. In the case of an in-plane loading you will need to plot the out-of-plane (lateral) deflection versus the load.



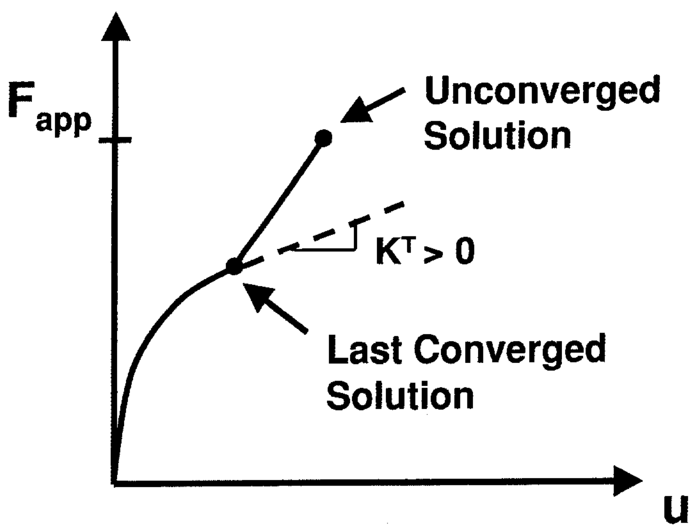
Plot force (F) versus displacement in the x-direction (UX).

Trouble Shooting

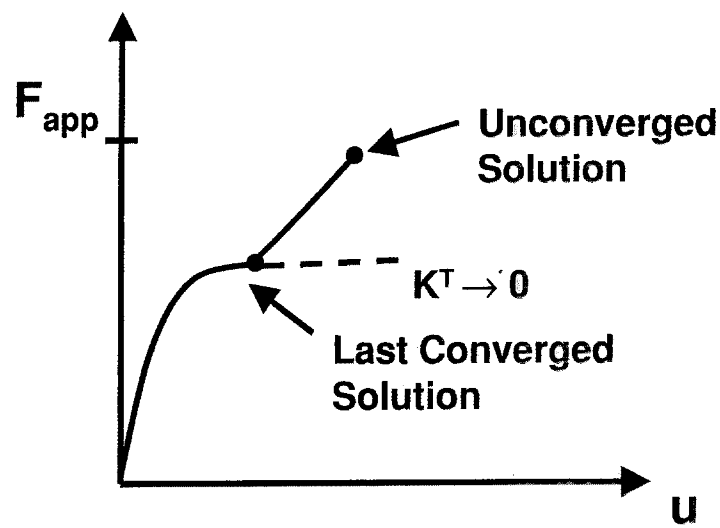
- The load deflection curve can help determine whether or not a physical instability or a numerical instability is the cause of the solution divergence. Recognize that an *unconverged solution does not necessarily mean that the structure has reached its maximum load!*

Trouble Shooting

- The tangent stiffness will approach zero as a structure nears its buckling load. A numerical or physical instability can be determined from the slope of the load-deflection curve



Numerical Instability



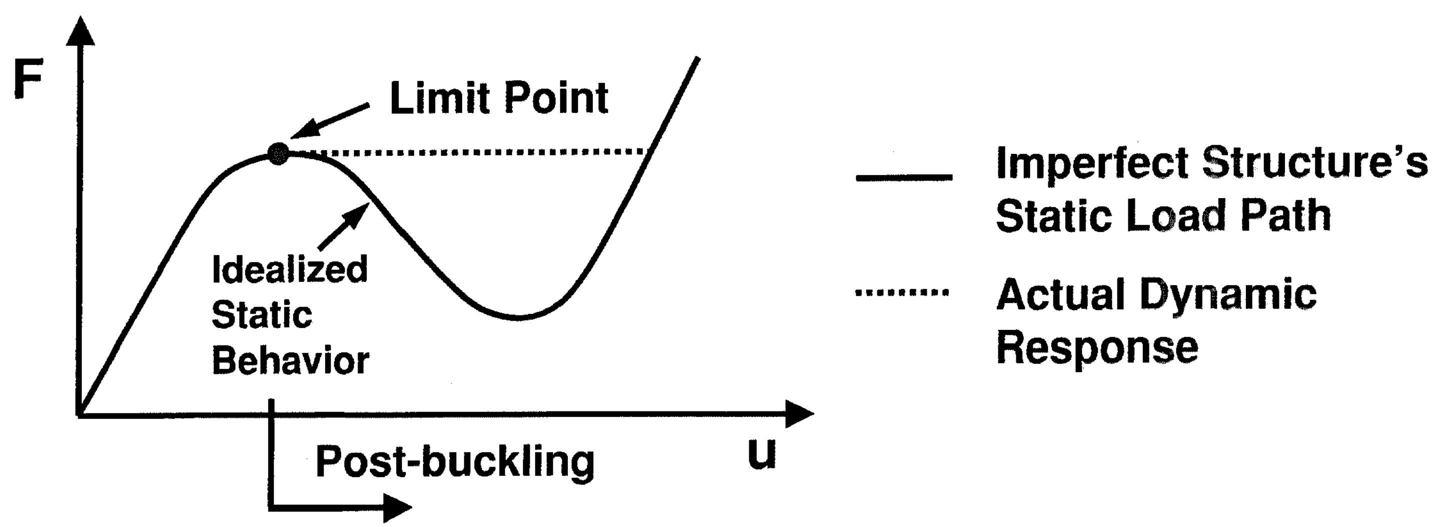
Physical Instability (buckling)

NOTES

Post-Buckling Analysis

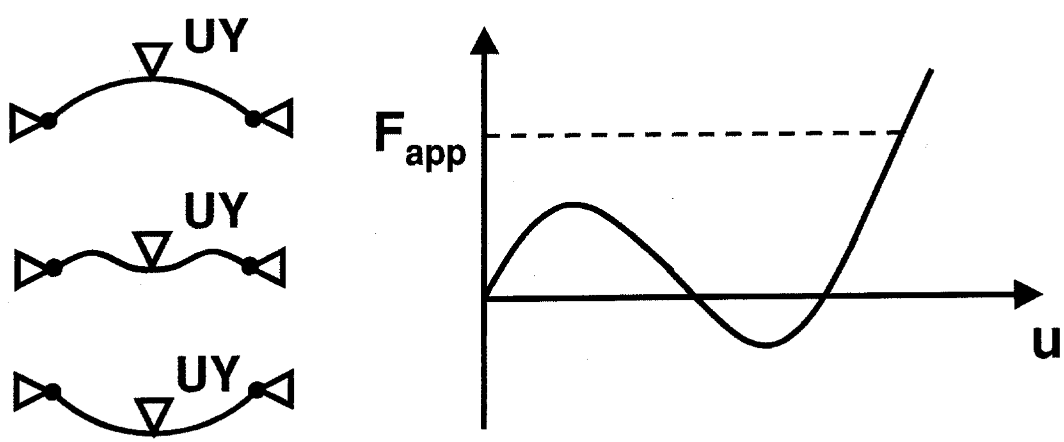
Analysis techniques for post-buckling include:

- Displacement Control
- Arc-Length Method



Displacement Control

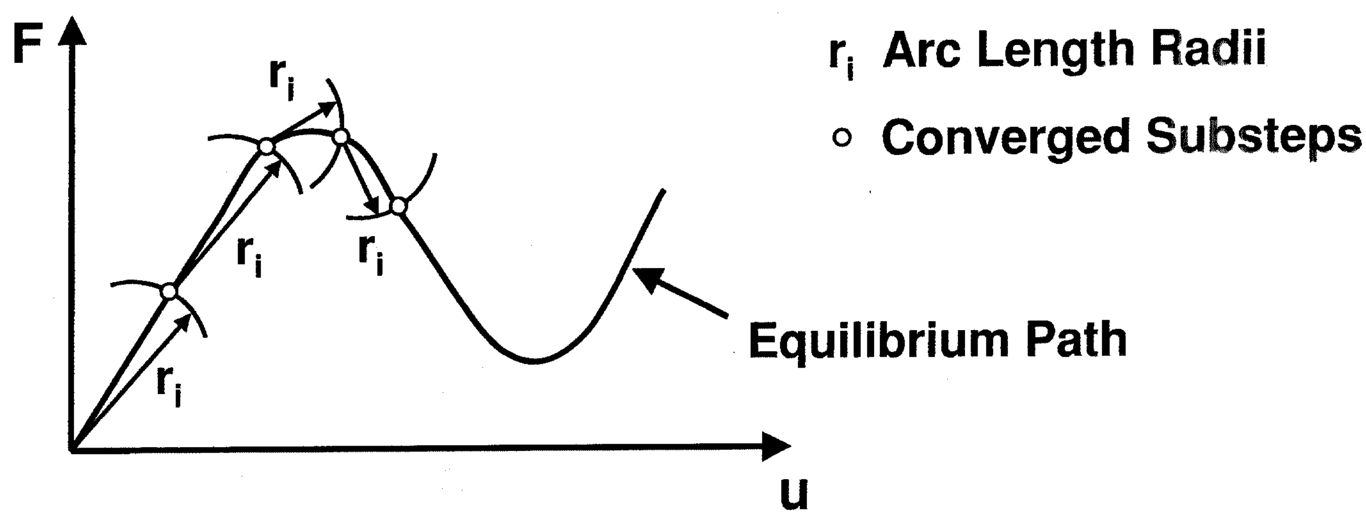
- As discussed earlier, displacement control can be used to predict a post-buckled response for simple loading conditions. The main disadvantage to displacement control is that for complicated loading it is generally not clear what displacements to impose.



F_{app} can be achieved with displacement control.

Arc-Length

- The *arc-length* is a solution method used to obtain numerically stable solutions to static problems with *proportional* loading experiencing instabilities ($K^T \rightarrow 0$), or negative tangent stiffnesses ($K^T < 0$).



Arc-Length Procedure

As with any other ANSYS analysis there are three main steps to perform an analysis using the arc-length:

1. Build the Model
2. Obtain the Solution
3. Review the Results

Arc-Length Procedure

Build the Model

- This task is similar to most other analyses with the following additional considerations:
  - The arc-length is only valid for a *force controlled proportional loading* analysis. No tabular loads are permitted.
  - The load factor is applied to all loads. Therefore, including a geometric imperfection to initiate an instability may be preferable to using a perturbation force.
  - The value of the applied load should be set to a value slightly higher (10 to 20%) than critical load predicted by the eigenvalue buckling analysis.

Arc-Length Procedure

Obtaining the Solution

- The arc-length is only valid for a static analysis (ANTYPE,STATIC). Be sure that nonlinear geometry is activated (NLGEOM,ON).

- The use of solution control is recommended (default).

- Do *not* use the line search (LNSRCH), the predictor (PRED), adaptive descent, automatic time stepping (AUTOTS, DELTIM), time (TIME), or time integration (TIMINT) with the arc-length method!

Arc-Length Procedure

Obtain the Solution

- A value 10 to 20 percent higher than the eigenvalue buckling load will often be a good choice for the value of the applied load. Do *not* set a value for "time", when using the arc-length. "Time" in an arc-length analysis is related to the load factor.

- Be sure to write out results for a sufficient number of substeps (OUTRES), so that you can examine the load deflection curve in the general postprocessor.

Arc-Length Procedure

Obtaining the Solution

- The reference arc-length is calculated from the value of the applied load and the number of substeps using the following equation:

Reference Arc-Length Radius = Total Load/NSBSTP

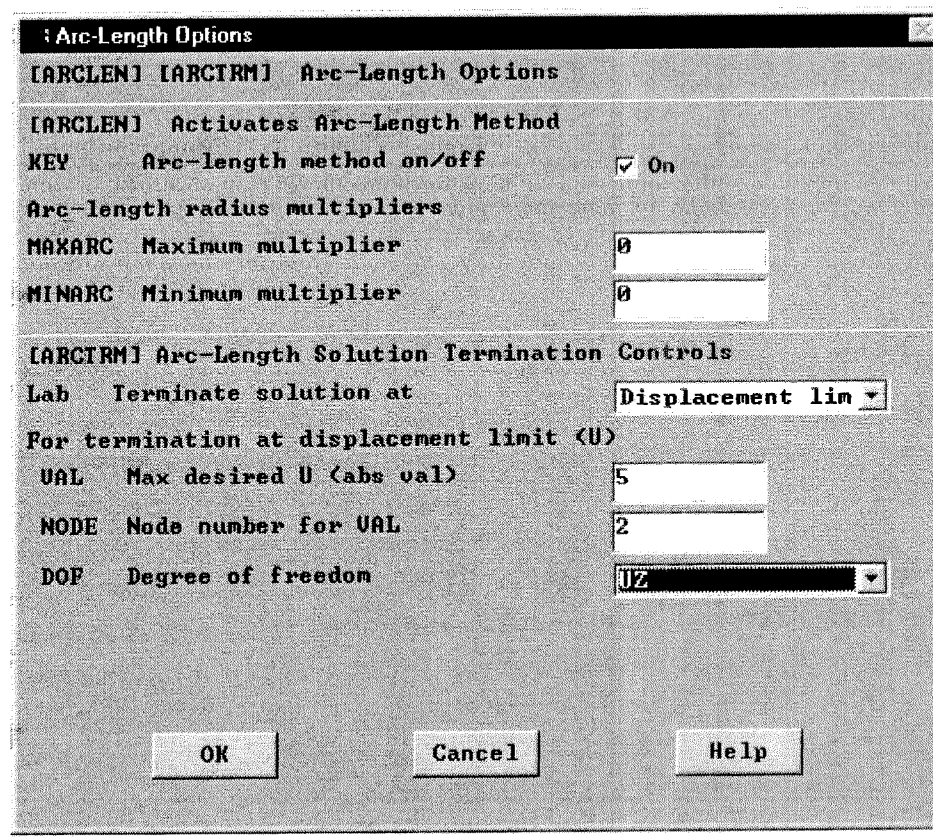
- where NSBSTP is the number of substeps specified. Using more substeps will allow the program to more closely follow the load deflection response but will also result in a longer solution time.

Arc-Length Procedure

Obtaining the Solution

To activate the arc-length use:

Solution > Nonlinear > Arc Length Opts ...



Arc-Length Procedure

Obtaining the Solution - Arc-Length Parameters

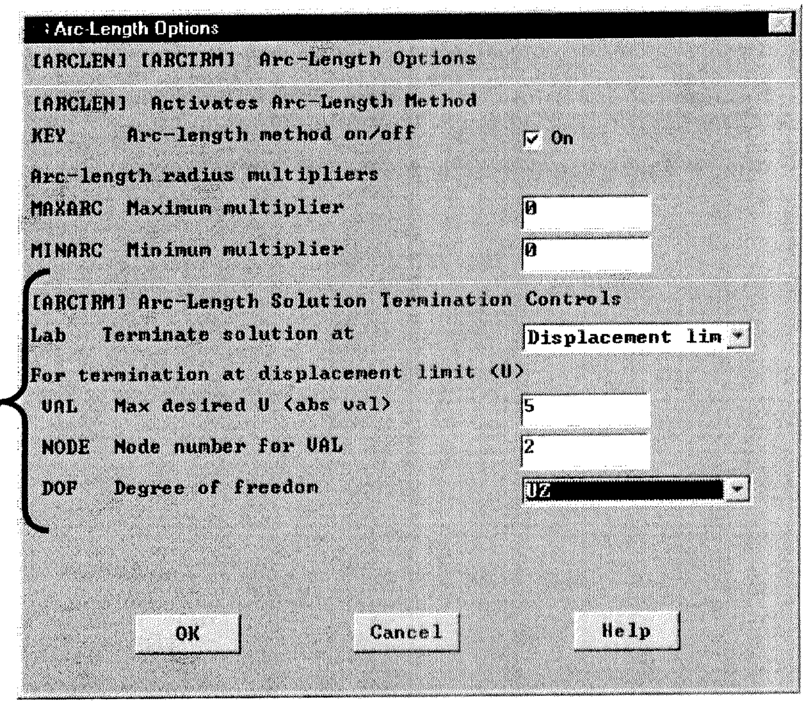
- The factors *MAXARC* and *MINARC* are arc-length radius multipliers which are used to define the limits of the arc-length radius. (Similar to setting the number of substeps for automatic time stepping.)

MAXARC defaults to 10
MINARC defaults to 0.001

Arc-Length Procedure

Obtaining the Solution - Arc-Length Termination Criteria

- The arc-length solution can be terminated by either achieving a load factor (λ) of 1.0 or by setting limits on the solution.
- The arc-length solution can be terminated by reaching the first limit point or achieving a maximum displacement criteria at a specific node.



Arc-Length Procedure

Review the Results

- Time in an arc-length analysis is related to the load factor. Therefore do *not* reference results by a "time" value when postprocessing an arc-length analysis. You should always reference the results by load step and substep number.

- "Time" in an arc-length analysis is not always increasing, and in some analyses it can be negative. In the time history postprocessor the total arc-length load factor (ALLF) and the arc-length load factor increment (ALDL) can be stored by using a solution summary (SOLU) item.

Arc-Length Procedure

Review the Results

- Be sure to review the load deflection curve in the time history postprocessor as in the case of a nonlinear buckling analysis.
- Note that the load factor (TIME) can increase or decrease, and can even become negative.

Trouble Shooting

- If the arc-length fails to converge, reducing the initial arc-length radius (NSUBST) can enhance convergence. Decreasing the lower arc-length radius multiplier (MINARC) can also improve convergence.

- "Drifting back" in which the analysis retraces its steps back along the load deflection curve is one difficulty that is caused by using too large or too small of an arc-length radius. You can use the number of substeps (NSUBST) and arc-length radius multipliers (MAXARC and MINARC) to adjust the arc-length radius.

Trouble Shooting

- Some trial and error may be required to determine the optimal settings for the arc-length radius. If the arc-length radius is too small the solution is very inefficient. If it is too large the solution may miss the buckling point or the "snap through" regime. The arc-length radius multipliers need to be set carefully.

- Be sure to plot the load-deflection curve. Establishing when the structure becomes unstable in the load history can be very useful when debugging an analysis.

Trouble Shooting

- In some cases, the arc-length procedure requires an initial geometric imperfection to trigger the nonlinear buckling mode. In such cases, use an eigenvalue buckling analysis to determine the mode, and add a geometric imperfection. For example, lateral torsional buckling of a cantilever beam would require a geometric imperfection, whereas a snap through analysis of a shallow arch would not require a geometric imperfection.

NOTES

Chapter 5

PLASTICITY

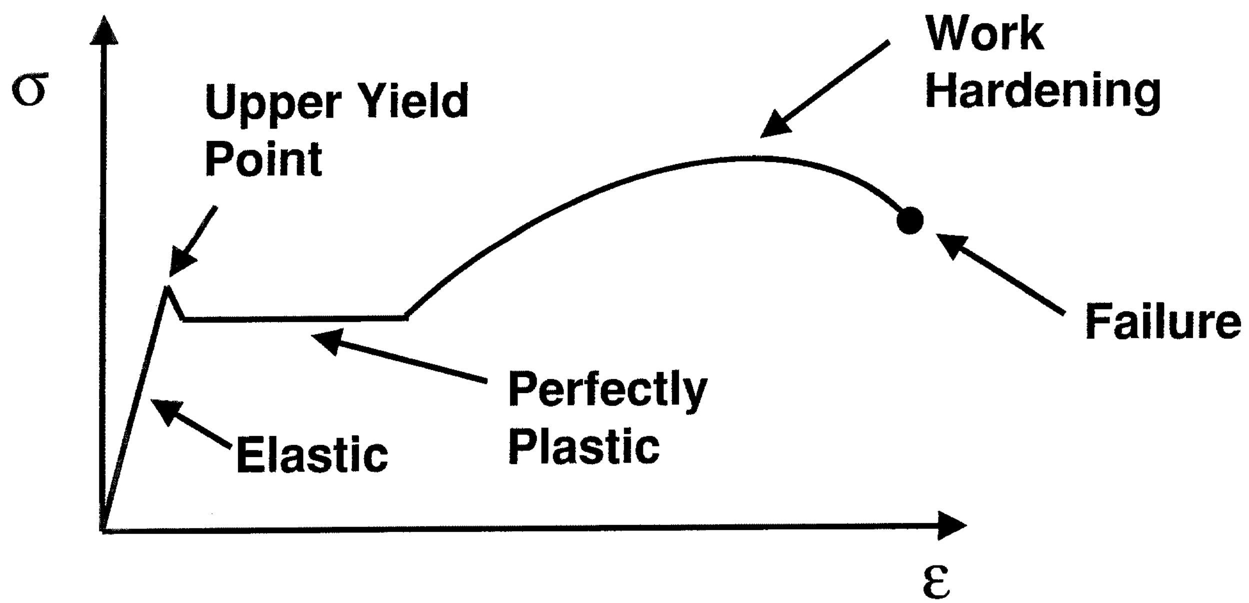
Session Objective

- **At the end of this session you will be able to describe and demonstrate the following:**

1. Plasticity - Preliminaries
2. Incremental Plasticity Theory
3. Hardening Rules - Isotropic and Kinematic Hardening
4. Plasticity Material Options
5. Element Recommendations
6. Obtaining the Solution
7. Output Variables
8. Trouble Shooting

Plasticity

- Plasticity is a material behavior in which the material deforms permanently (irreversible plastic strains develop) under the application of the applied loads.



Stress-Strain Curve for a Mild Steel (Exaggerated)

Path Dependence

- Plastic strains that develop in a material are irreversible. The solutions to plasticity problems are *nonconservative*, as energy is dissipated by the plastic straining. The solution to a *nonconservative* problem depends on the history of the load. Plasticity is a *path-dependent*, or *nonconservative* phenomenon.

- When analyzing a structure undergoing plastic straining, *the actual load history must be followed to ensure the correct solution.* Path dependent problems also require that the *load be applied slowly* (using many substeps). And, *superposition does not hold* for plasticity.

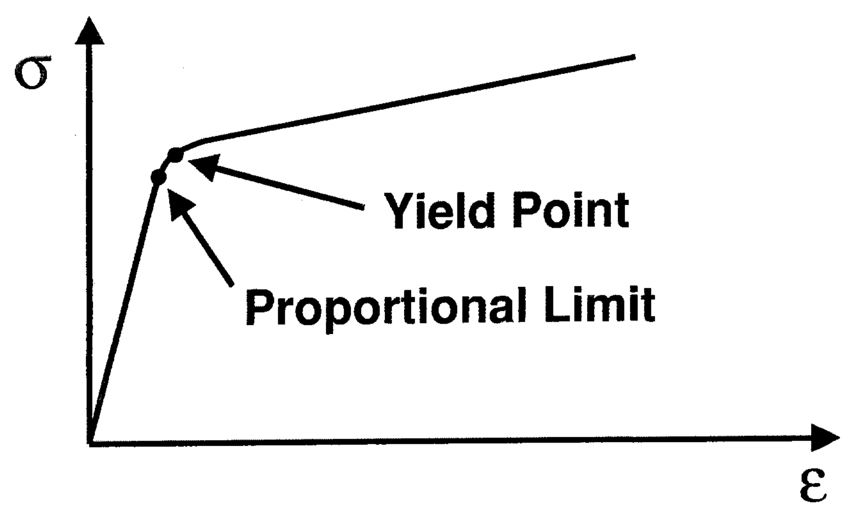
Plasticity - Preliminaries

The plastic response of a structure (typically due to a multiaxial stress state) is based on the results of a uniaxial test specimen. Based on the results of a uniaxial stress-strain test the following information can be derived:

- Proportional Limit
- Yield Point
- Work Hardening
- Bauschinger Effect

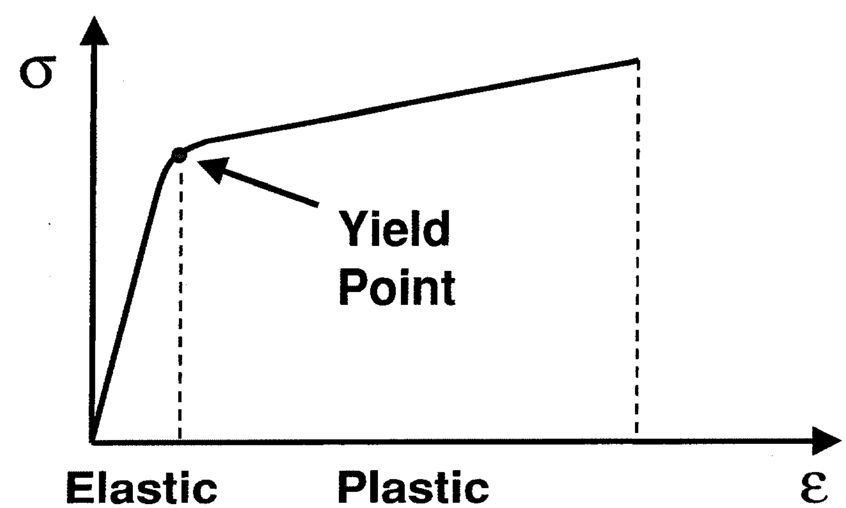
Yield Point

Most ductile metals behave linearly below a stress level, called the *proportional limit*. Below the *proportional limit*, the stress is linearly related to the strain. Additionally, below a stress level called the *yield point*, the stress-strain response is elastic. Below the *yield point*, any straining which occurs is completely recoverable upon removal of the load.



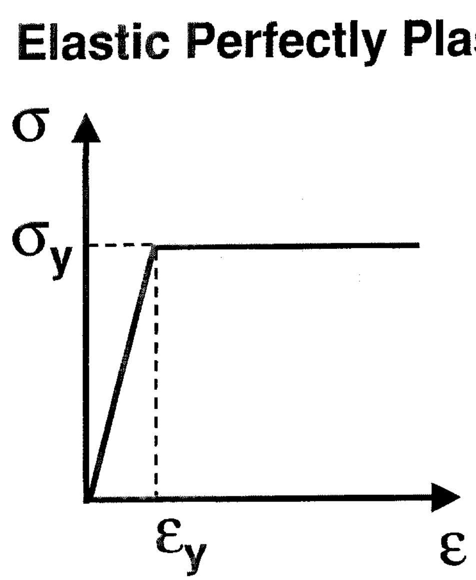
Yield Point

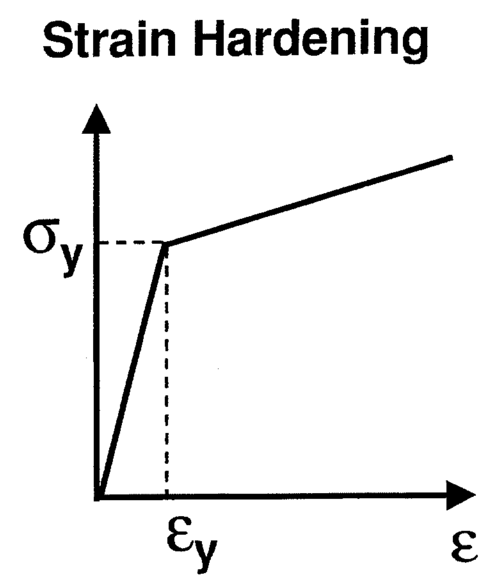
There is usually little difference between the yield point and the proportional limit, and the program always assumes them to be the same. The portion of the stress-strain curve below the yield point is called the elastic portion, and the portion above the yield point is the plastic or strain hardening portion.



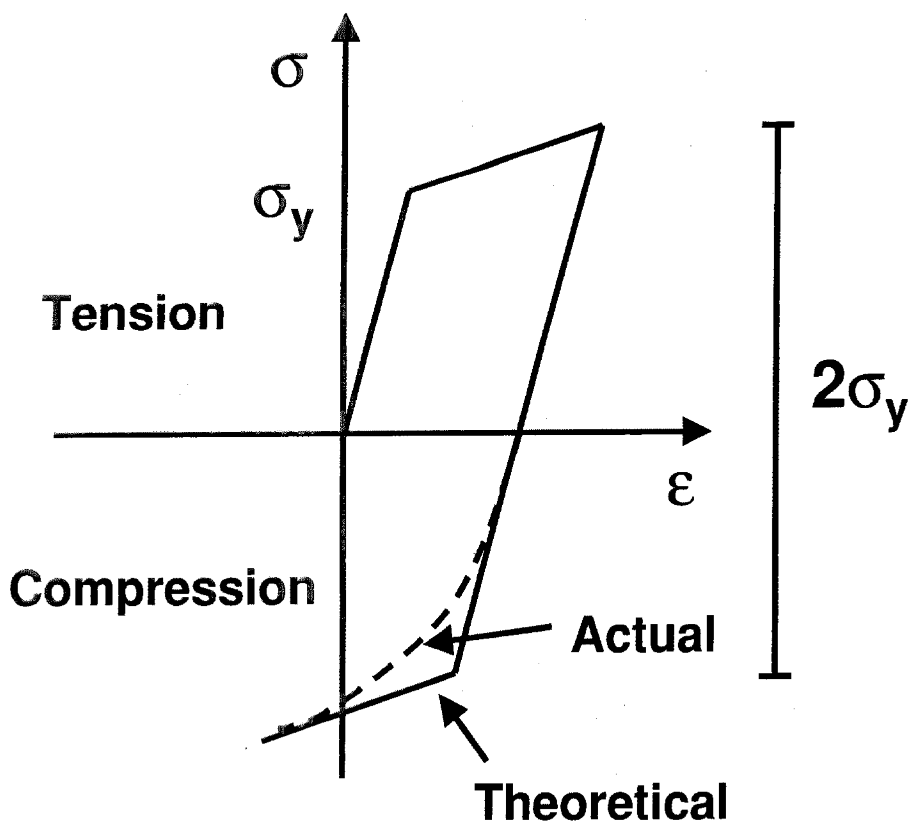
Strain Hardening

Strain hardening is typically represented by either *elastic perfectly plastic* material behavior or as *strain hardening* behavior. For the uniaxial case these relationships which represent plastic flow (deformation of the material when stressed over yield) are illustrated below:





Bauschinger Effect



Most metals exhibit a behavior called the Bauschinger effect for small strain cyclic loading. The Bauschinger effect is a subsequent decrease in the yield stress in compression after yielding in tension, such that an approximate $2\sigma_y$ difference exists between the respective yields in tension and compression.

Deviatoric Stress

Based on P.W. Bridgeman's classic experiments, hydrostatic stresses have practically no influence on the yielding of metals. Shear stress is the primary contributor to yielding.

For a general stress state $\{\sigma\}$, the stresses can be decomposed into a hydrostatic stress and deviatoric stresses. The deviatoric stresses represent a pure shear state as the hydrostatic stress is removed.

$$\{S\} = \text{Deviatoric Stress Vector}$$

$$\{S\} = \{\sigma\} - \sigma_m \, [1\ 1\ 1\ 0\ 0\ 0]^T$$

where: $\sigma_m = \text{Hydrostatic Stress} = 1/3(\sigma_x + \sigma_y + \sigma_z)$

Equivalent Stress

Yielding occurs due to the deviatoric stresses. Since the yield point is defined as a scalar value from the uniaxial stress-strain curve, the deviatoric stresses need to be represented by a scalar value to define a yield criterion.

The equivalent stress σ_e is derived from the deviatoric stresses and is a measure of the shearing energy. The equivalent stress is used to determine if yielding is occurring from general state of stress

$$\sigma_e = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]}$$

Note on Rate Dependency

- The magnitude of the plastic strains that develop may be a function of how fast (or slow) the loading is applied. If the plastic strains develop without regard to the time frame, the plasticity is said to be *rate-independent*. Conversely, plasticity that depends on the strain rate is called *rate-dependent*.

- The majority of this presentation will focus on *rate-independent* plasticity. The plastic straining is assumed to be time independent.

- ANSYS has a rate-dependent model (Anand's model) which is applicable for metal forming processes.

Incremental Plasticity Theory

Incremental plasticity theory provides a mathematical relationship that characterizes the increments of stress and strain ($\Delta\sigma$ and $\Delta\varepsilon$) to represent the material behavior in the plastic range. There are three basic components to incremental plasticity theory:

- The Yield Criterion
- The Flow Rule
- The Hardening Rule

Incremental plasticity theory recognizes that the final value of stress and strain in a plasticity problem is a path dependent phenomenon.

Yield Criterion

For the case of a uniaxial tensile specimen, yielding can be determined by comparing the axial stress to the material yield stress. However, for a multiaxial state of stress, it is necessary to define a yield criterion.

The *yield criterion* is a single-valued (scalar) measure of the stress state that may be readily compared to the yield stress from the uniaxial test. Therefore, knowing the stress state and the yield criterion, the program can determine if plastic straining will develop.

Yield Criterion

A common yield criterion is the von Mises yield criterion. Yielding begins whenever the internal energy of distortion (equivalent stress) exceeds a certain value. The von Mises equivalent stress is defined as:

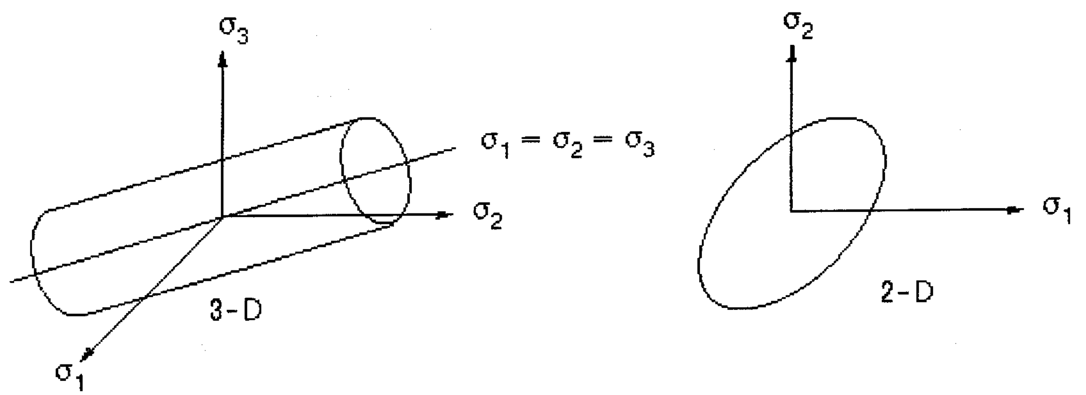
$$\sigma_e = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]}$$

where σ_1 σ_2 and σ_3 are the principal stresses. Yielding occurs when the equivalent stress exceeds the yield stress of the material:

$$\sigma_e > \sigma_y$$

von Mises Yield Criteria

The von Mises yield criterion can be plotted in principal stress space as:



The *yield surface* is a cylindrical surface aligned along the axis σ_1=σ_2=σ_3 in 3-D. In 2-D, the yield criterion plots as an ellipse. Any stress state inside this *yield surface* is elastic and any outside causes yielding.

Flow Rule

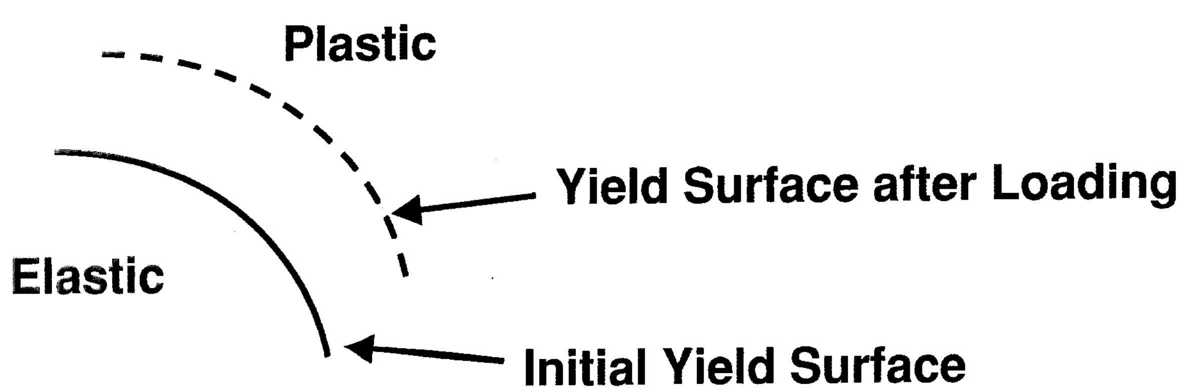
The total strain increment can be split into elastic and plastic components. The plastic flow defines the relationship between the stress and plastic strain increments ($\Delta\varepsilon^{pl}$).

The flow rule also prescribes the direction of the plastic straining when yielding occurs. Flow equations are derived from the yield criterion which implies that the plastic strains develop in a direction normal to the yield surface. Such a flow rule is termed associative. If some other flow rule is used (derived from a different function), it is called non-associative.

Hardening Rule

The *hardening rule* prescribes the material strain hardening by relating to the uniaxial case. The *hardening rule* describes how the yield surface is modified during the plastic flow.

The hardening rule determines when the material will yield again if the loading is continued or if the loading is reversed.



Hardening Rule

There are two hardening rules used by ANSYS to prescribe the modification of the yield surface:

- Isotropic Hardening

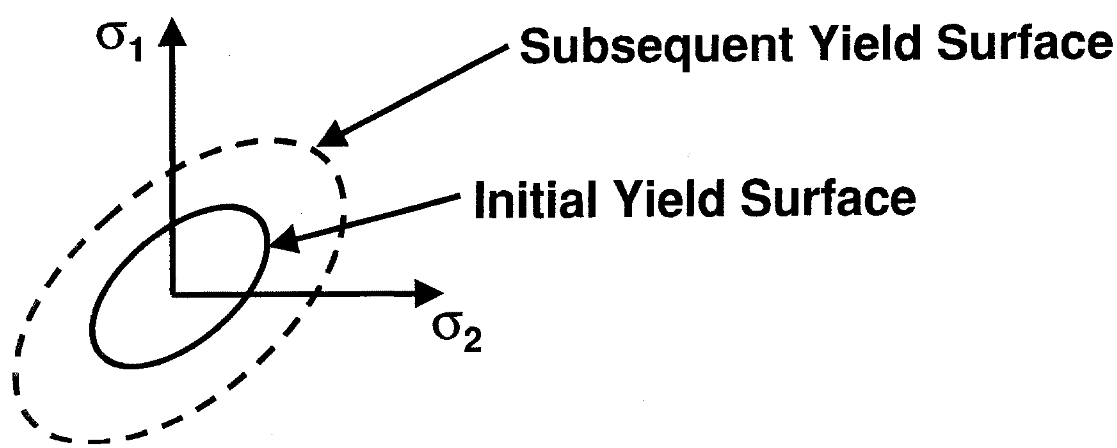
  The yield surface expands in size with plastic flow.

- Kinematic Hardening

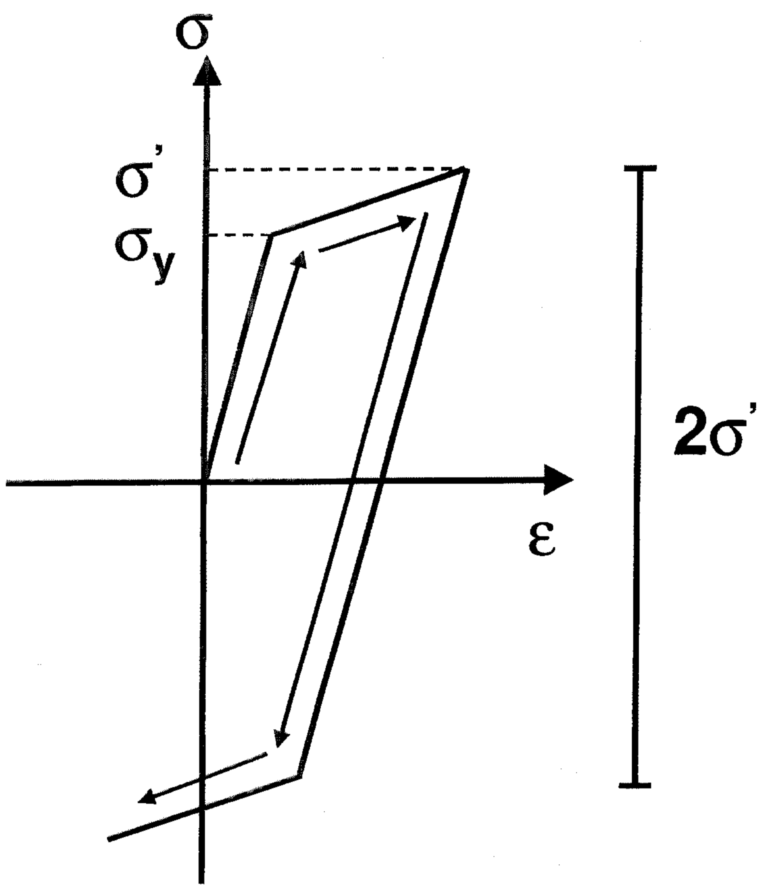
  The yield surface shifts in stress space.

Isotropic Hardening

Isotropic hardening predicts a uniform expansion of the initial yield surface during plastic flow. This hardening model assumes the plastic deformation is an isotropic process, as the Bauschinger effect is ignored. As such this model breaks down for cyclic loading.



Isotropic Hardening

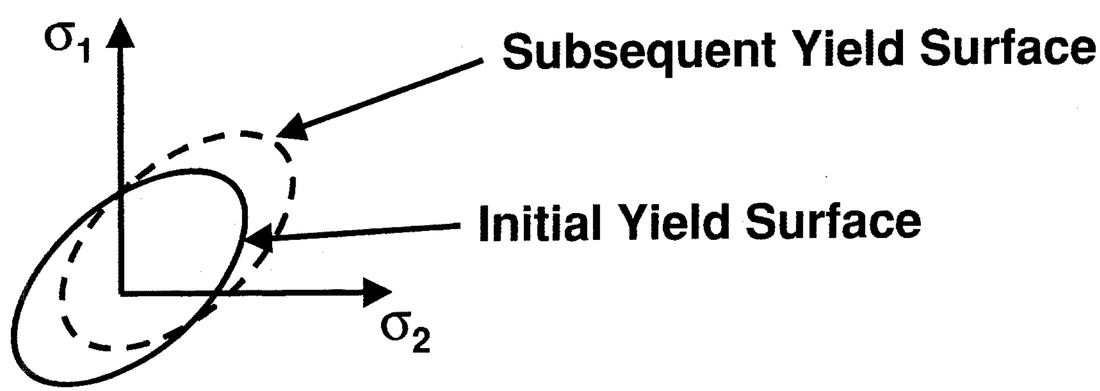


The stress-strain behavior for a uniaxial test specimen with isotropic hardening is shown. Note that the subsequent yield in compression is equal to the highest stress attained during the tensile phase.

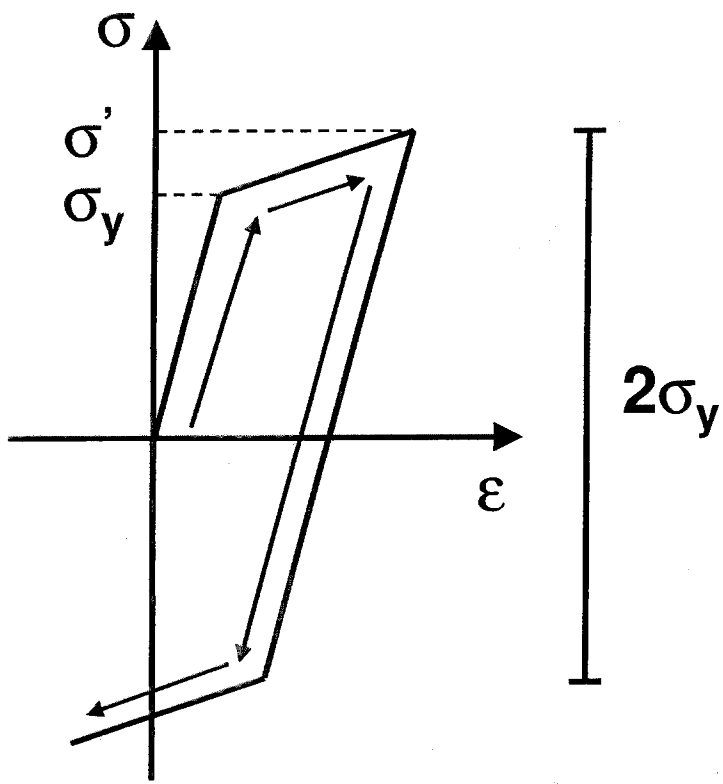
Isotropic hardening is often used for *large strain* or *proportional loading* simulations.

Kinematic Hardening

Kinematic hardening assumes that the initial yield surface translates with respect to the origin as a rigid body during plastic flow. An initially isotropic material is no longer isotropic after it yields, as the Bauschinger effect is included. Kinematic hardening is generally used for small strain and cyclic loading applications.



Kinematic Hardening



The stress-strain behavior for a uniaxial test specimen with kinematic hardening is shown. Note that the subsequent yield in compression is decreased by the amount that the yield stress in tension increased, so that a $2\sigma_y$ difference between the yields is always maintained.

For *large strain* simulations, kinematic hardening is *not* appropriate.

Plasticity Options

The ANSYS program has eight plasticity material options:

Bilinear Kinematic Hardening	BKIN
Bilinear Isotropic Hardening	BISO
Multilinear Kinematic Hardening	MKIN
Multilinear Kinematic Hardening	KINH
Multilinear Isotropic Hardening	MISO
Anisotropic	ANISO
Drucker-Prager	DP
Anand's Model	ANAND

The following table summarizes the plasticity options, including the yield criterion, flow rule, and the hardening rule.

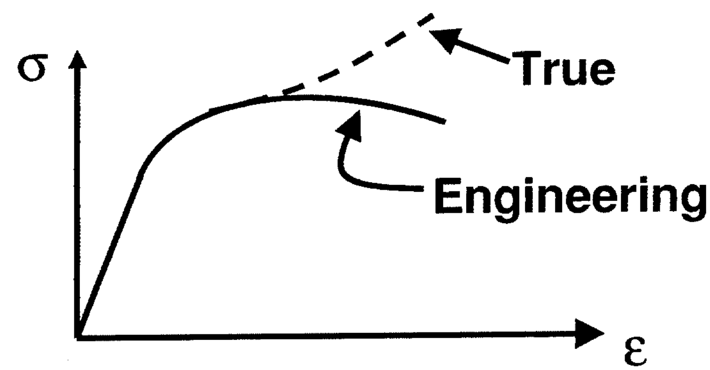
Plasticity Options

Name	Label	Yield Criterion	Flow Rule	Hardening Rule	Material Response
Bilinear Kinematic Hardening	BKIN	von Mises	Associative	Kinematic hardening	Bilinear
Bilinear Isotropic Hardening	BISO	von Mises	Associative	Isotropic hardening	Bilinear
Multilinear Kinematic Hardening	MKIN	von Mises	Associative	Kinematic hardening	Multilinear
Multilinear Kinematic Hardening	KINH	von Mises	Associative	Kinematic hardening	Multilinear
Multilinear Isotropic Hardening	MISO	von Mises	Associative	Isotropic hardening	Multilinear
Anisotropic	ANISO	Modified von Mises	Associative	Isotropic hardening	Bilinear
Drucker-Prager	DP	von Mises with dependence on hydrostatic stress	Associative or nonassociative	None	Elastic-perfectly plastic
Anand's Model	ANAND	von Mises*	Associative	Isotropic hardening	Nonlinear

***Anand's model does not use a yield criterion, but the effective stress computed is von Mises.**

Note on Engineering versus True Strain

Plasticity material data is usually supplied in the form of a tensile stress-strain curve. The data is in the form of engineering stress (P/A_0) versus engineering strain ($\Delta l/l_0$), or as true stress (P/A) versus true, or log, strain ($\ln(l/l_0)$).



Large strain plasticity analyses (NLGEOM,ON) expect the material constants from a *true stress-strain* curve, while *small strain* analyses (NLGEOM,OFF) can use *engineering stress-strain* data.

Note on Engineering versus True Strain

Since for small strain response, the engineering strain and logarithmic (true) strain are nearly identical, *true stress and logarithmic strain data can be used for general applications.*

To convert from engineering to true, use:

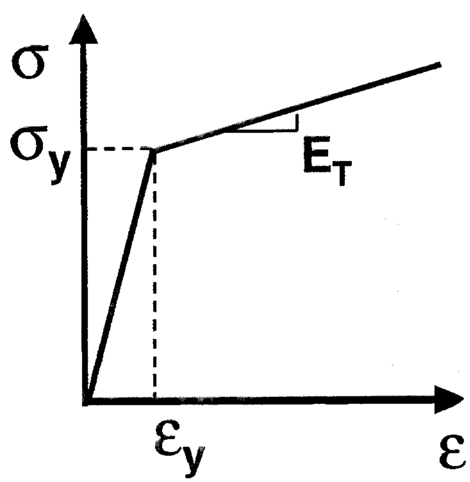
$$\varepsilon_{ln} = \ln(1 + \varepsilon_{eng})$$

$$\sigma_{true} = \sigma_{eng}(1 + \varepsilon_{eng})$$

Note that the stress conversion is valid only for incompressible plasticity stress-strain data.

Bilinear Kinematic Hardening

Bilinear Kinematic Hardening (BKIN) uses a bilinear representation of the stress-strain curve, which includes the elastic slope and the tangent modulus. The von Mises yield criterion with kinematic hardening is used, so that the Bauschinger effect is included. This option may be used for small-strain and cyclic loading applications.

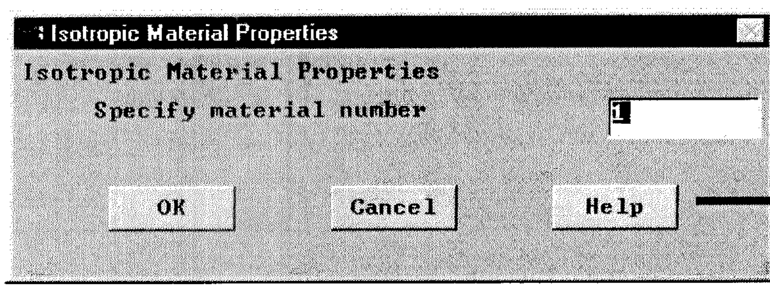


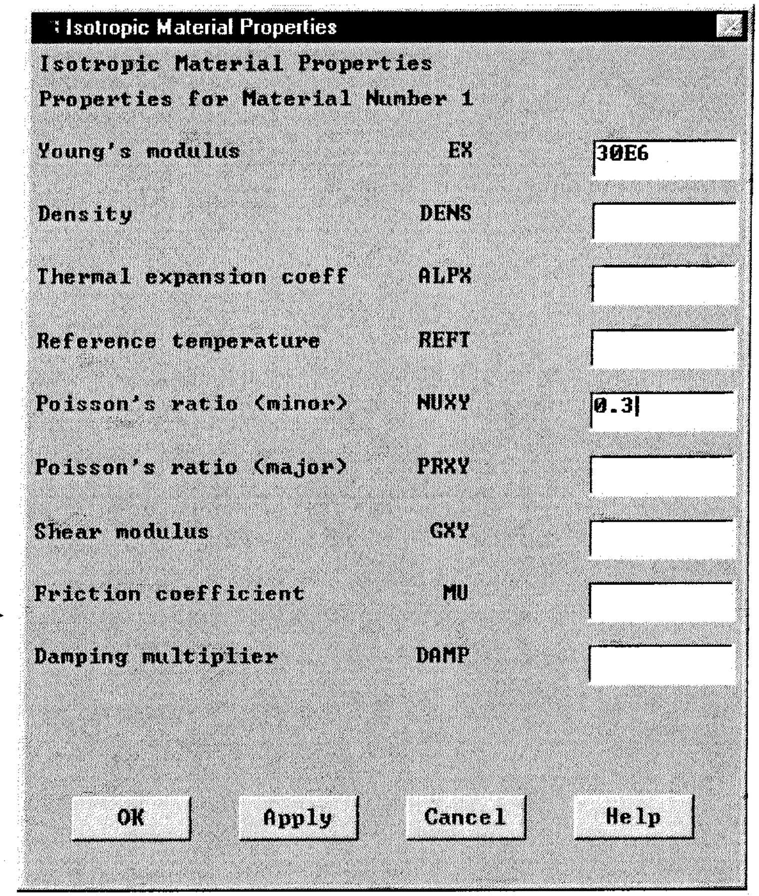
The required input values for bilinear kinematic hardening are the elastic modulus E, the yield stress σ_y, and the tangent modulus E_T.

Bilinear Kinematic Hardening

To define the elastic modulus (E) for the Bilinear Kinematic Hardening model use:

Preprocessor > Material Properties > Isotropic ...

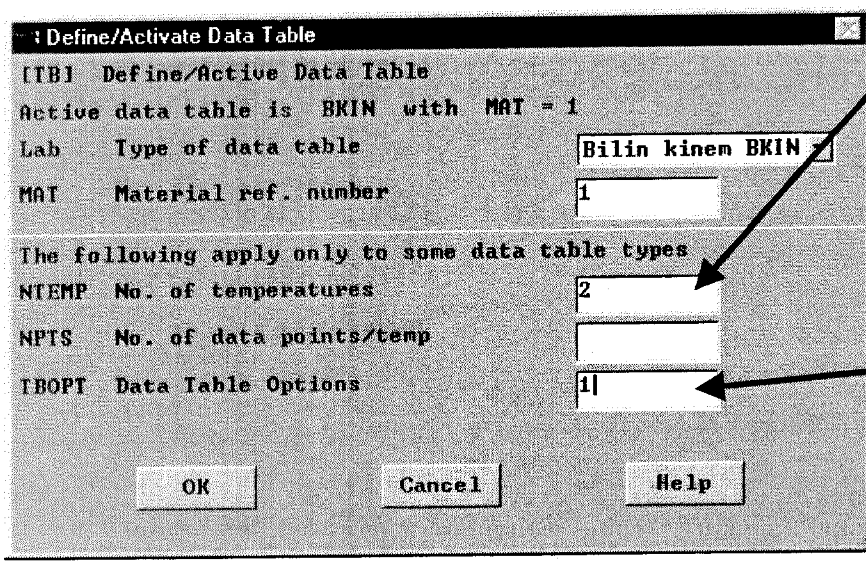




Bilinear Kinematic Hardening

To activate a nonlinear data table for the Bilinear Kinematic Hardening model use:

Preprocessor > Material Properties > Data Tables > Define Activate ...



Up to six temperature dependent curves can be defined.

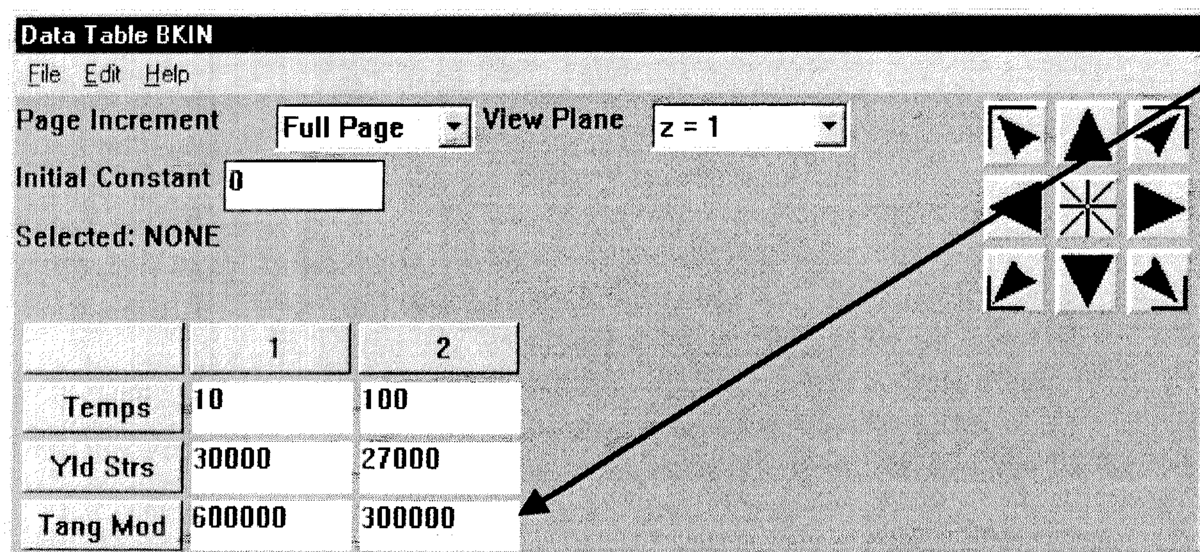
TBOPT=1 includes stress relaxation with temperature increases (Rice's model, default).

TBOPT=0 does *not* include stress relaxation with an increase in temperature (not recommended).

Bilinear Kinematic Hardening

To enter the yield stress and the tangent modulus for the Bilinear Kinematic Hardening model use:

Preprocessor > Material Properties > Data Tables > Edit Active ...

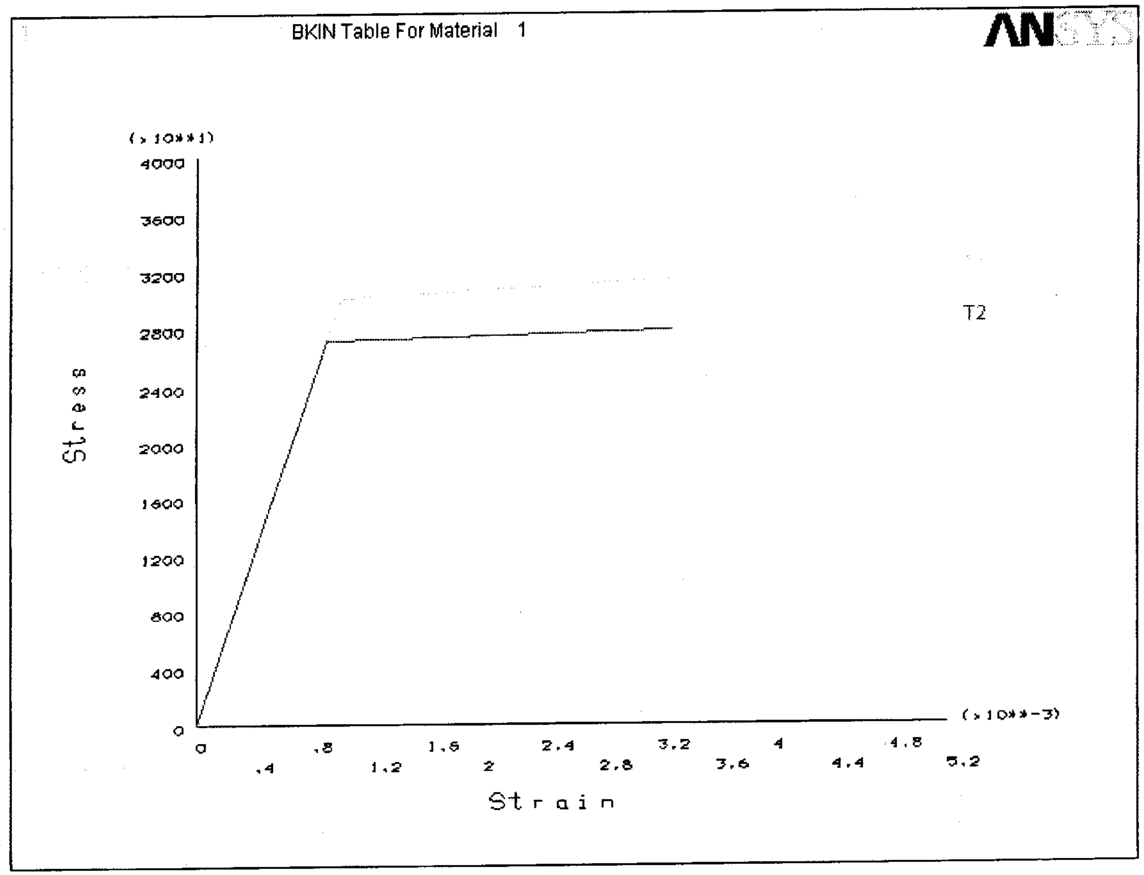


Note that the tangent modulus can not be negative or greater than the elastic modulus.

Bilinear Kinematic Hardening

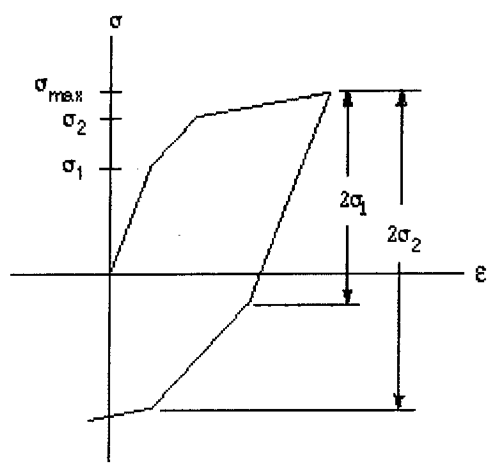
To graph the stress-strain curve for the Bilinear Kinematic Hardening model use:

Preprocessor > Material Properties > Data Tables > Graph ...



Multilinear Kinematic Hardening

There are two options for Multilinear Kinematic Hardening MKIN and KINH. Both material models use a multilinear representation of the stress-strain curve to simulate the effects of kinematic hardening. These options use a von Mises yield criterion and are useful for small strain plasticity analyses of metals.



Both MKIN and KINH are defined by entering the elastic modulus and the stress-strain data points. The input procedure for the *elastic modulus* (E) is the same as for the BKIN model.

Multilinear Kinematic Hardening MKIN Option

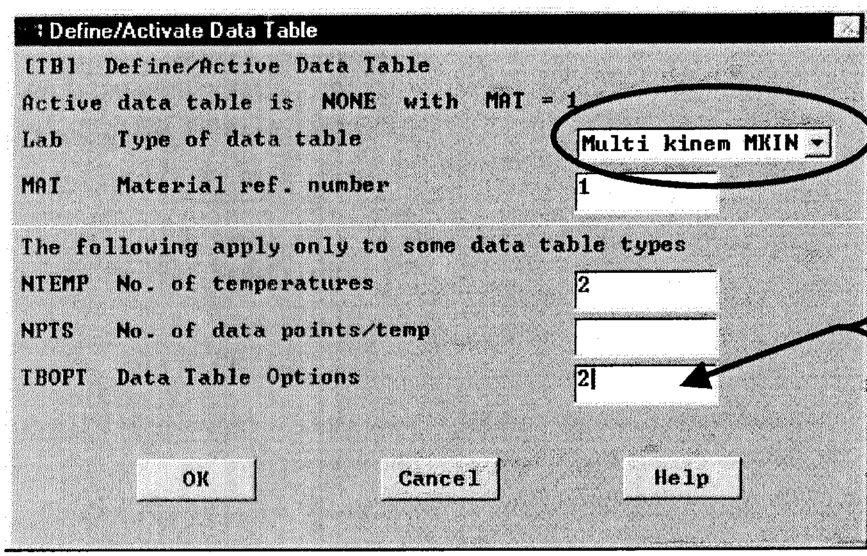
The MKIN option uses the Besseling or sublayer model (see *ANSYS Theory Manual).* The MKIN option allows a maximum of five stress-strain data points with up to five temperature dependent curves. The MKIN model has the following additional restrictions:

- Each stress-strain curve *must* have the same set of strain values.
- The first point of the curve *must* correspond to the elastic modulus.
- No segment can have a slope larger than the elastic modulus (negative slopes are allowed).
- For strain values past the end of the input curve perfectly plastic material behavior is assumed.

Multilinear Kinematic Hardening
MKIN Option

To activate the Multilinear Kinematic Hardening model use:

Preprocessor > Material Properties > Data Tables > Define Activate ...



TBOPT = 0 No stress relaxation with temperature increases (default).

TBOPT = 1 Recalculate total plastic strain using new weight factors.

TBOPT = 2 Scale plastic strains to keep total plastic strain constant; agrees with Rice's model (recommended).

Multilinear Kinematic Hardening KINH Option

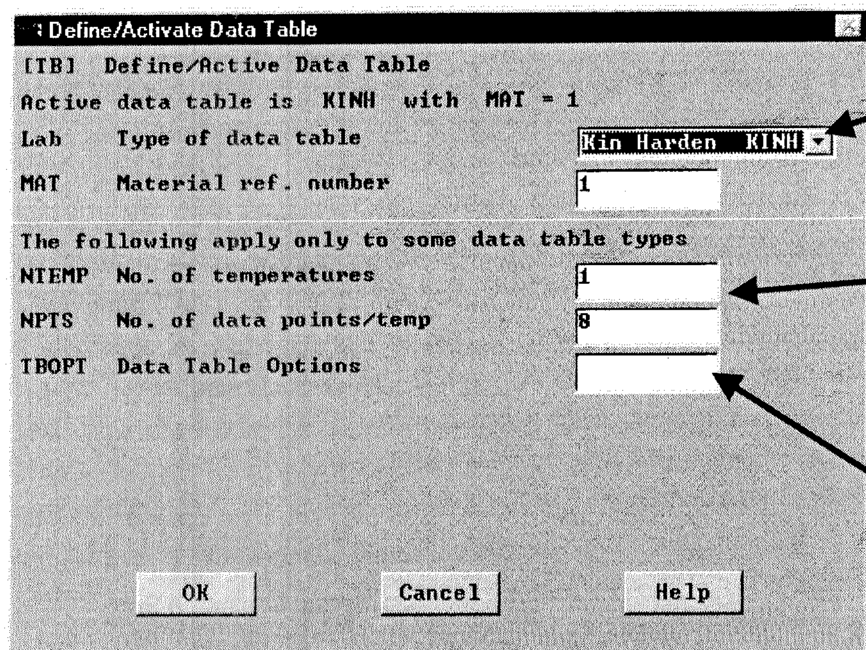
The KINH option removes some of the restrictions imposed by the MKIN model. (KINH has the same mechanical behavior as MKIN with TBOPT=2, Rice's model.) Up to 40 temperature dependent stress-strain curves can be defined, with up to 20 points per curve. Curves at different temperatures *must* have the same number of points, however the strain values can vary from one curve to the next.

The assumption is that the corresponding points on the different stress-strain curves represent the temperature dependent yield behavior of a particular sublayer.

Multilinear Kinematic Hardening
KINH Option

To activate the Multilinear Kinematic Hardening model use:

Preprocessor > Material Properties > Data Tables > Define Activate ...



Specify KINH.

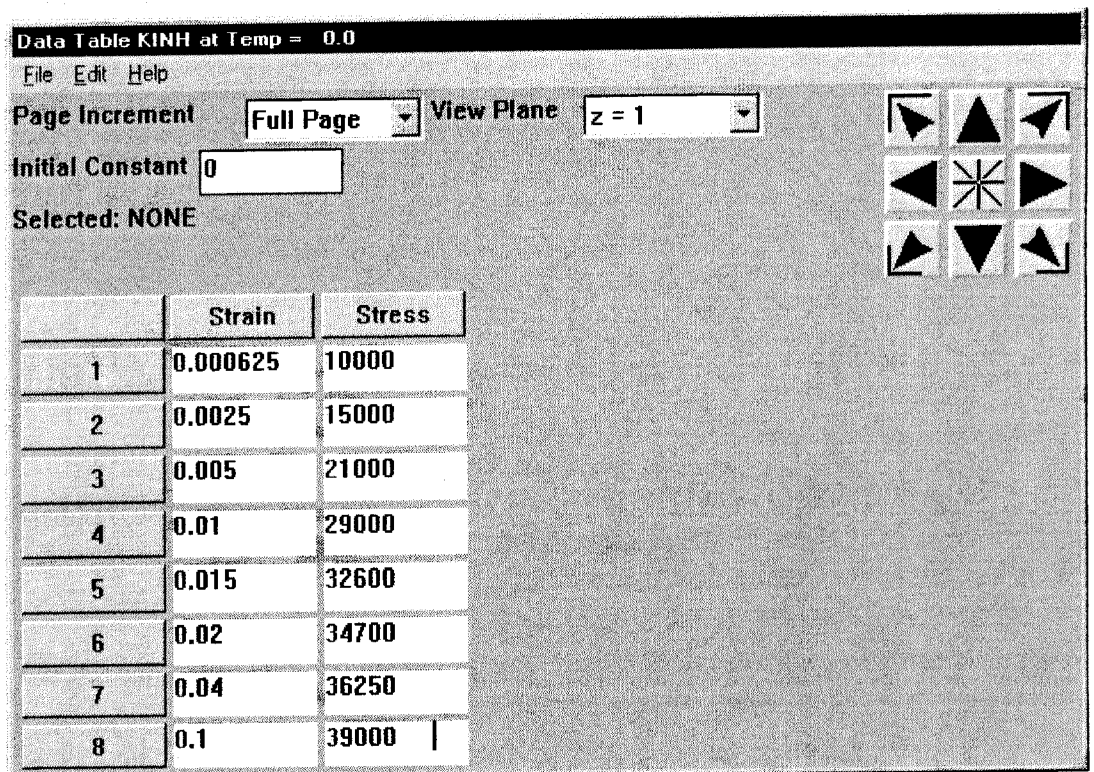
Number of temperatures and data points.

TBOPT has no effect for the KINH model.

Multilinear Kinematic Hardening
KINH Option

Enter the stress-strain data points.

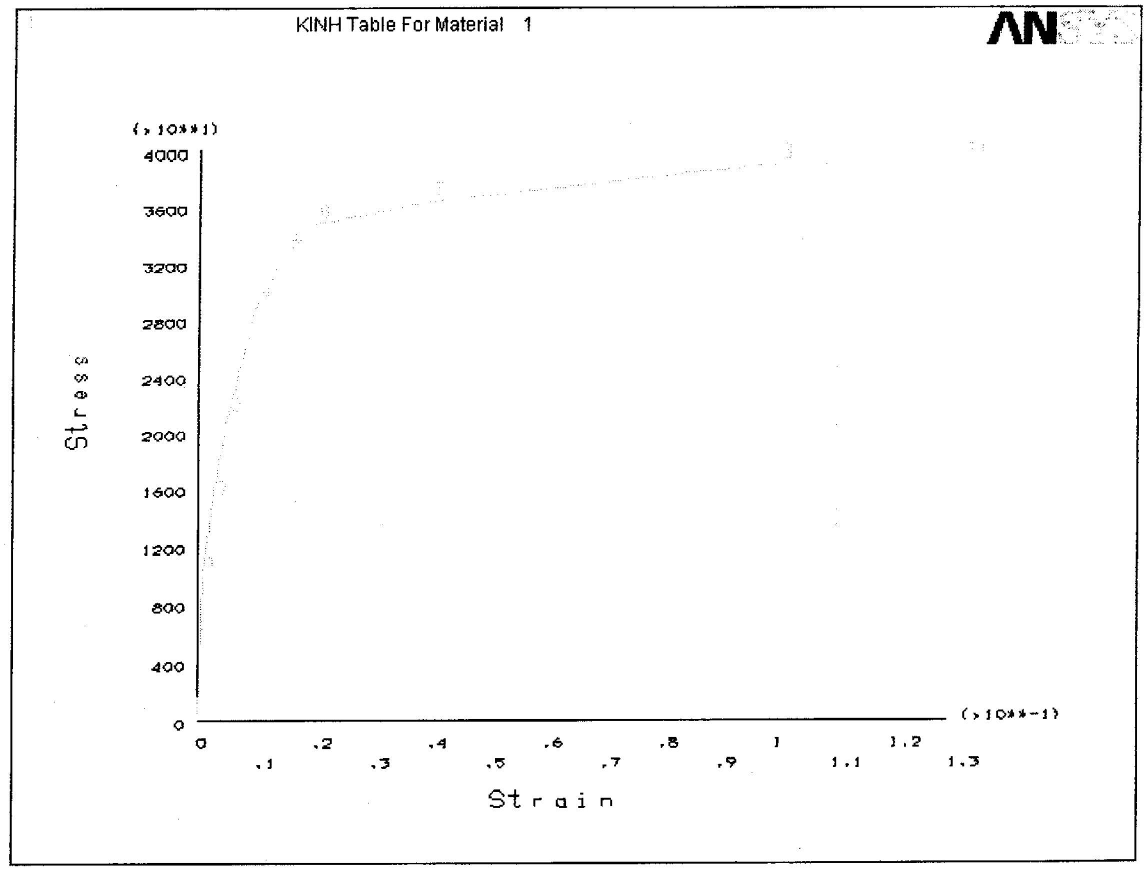
Preprocessor > Material Properties > Data Tables > Edit Active ...



Multilinear Kinematic Hardening KINH Option

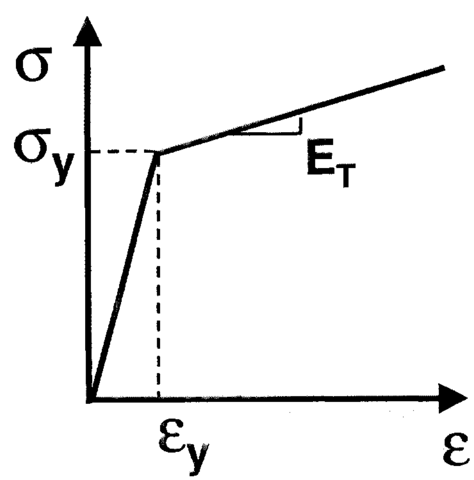
To graph the stress-strain curve for the Multilinear Kinematic Hardening model use:

Preprocessor > Material Properties > Data Tables > Graph ...



Bilinear Isotropic Hardening

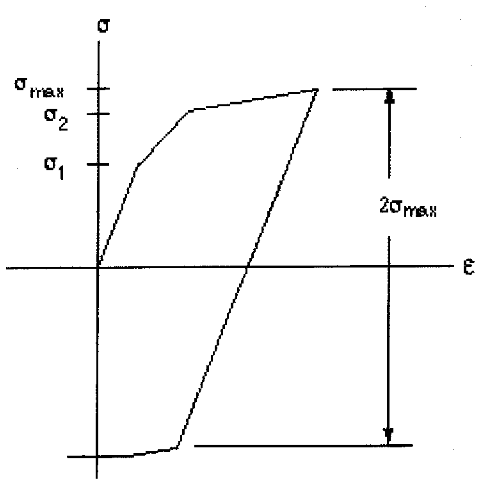
Bilinear Isotropic Hardening (BISO) also uses a bilinear representation of the stress-strain curve. The von Mises yield criterion with isotropic hardening is used. This option is generally used for large-strain applications of metal plasticity. Bilinear isotropic hardening is not recommended for cyclic loading.



The required input values for bilinear isotropic hardening are the elastic modulus E, the yield stress σ_y, and the tangent modulus E_T. The input procedure is the same as for bilinear kinematic hardening model.

Multilinear Isotropic Hardening

Multilinear Isotropic Hardening (MISO) also uses a multilinear representation of the stress-strain curve. The von Mises yield criterion with isotropic hardening is used. This option is generally used for proportional loading and large-strain applications of metal plasticity.



The Multilinear Isotropic Hardening model is defined by entering the elastic modulus and the stress-strain data points. The input procedure is the same as for the KINH model.

Multilinear Isotropic Hardening

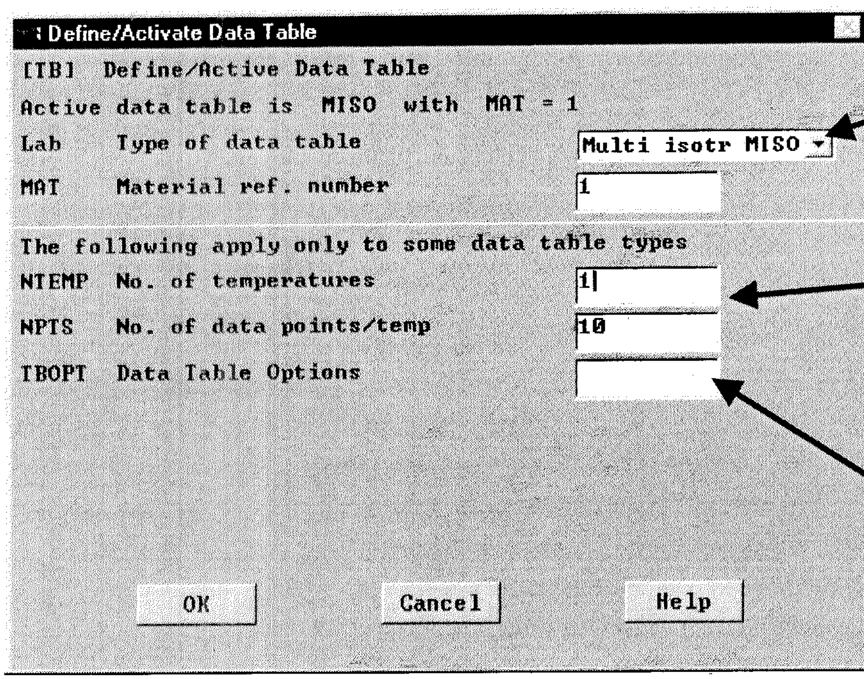
The MISO option allows a maximum of 100 stress-strain data points with up to twenty temperature dependent curves. The MISO model has the following additional restrictions:

- The first point of the curve *must* correspond to the elastic modulus.
- No segment can have a slope larger than the elastic modulus or less than zero.
- For strain values past the end of the input curve perfectly plastic material behavior is assumed.

Multilinear Isotropic Hardening

To activate the Multilinear Isotropic Hardening model use:

Preprocessor > Material Properties > Data Tables > Define Activate ...



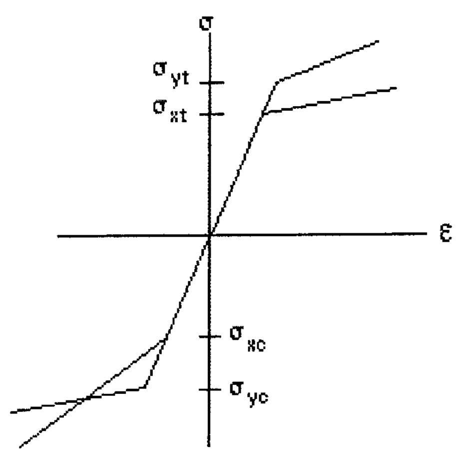
Specify MISO.

Number of temperatures and data points.

TBOPT has no effect for the MISO model.

Anisotropic Plasticity

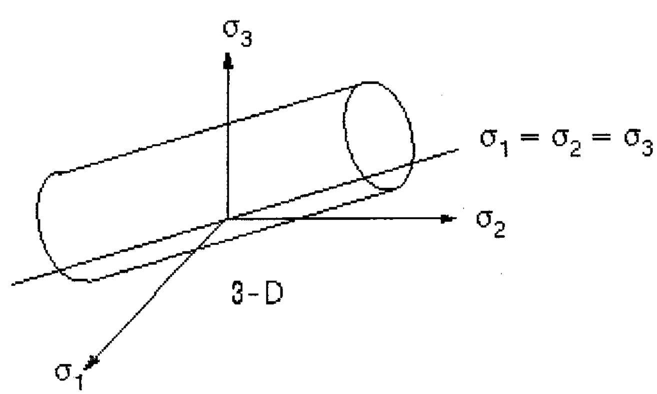
Anisotropic (ANISO) allows for different stress-strain behavior in the material x, y, and z directions as well as different behavior in tension and compression. A modified von Mises yield criterion is used with an isotropic hardening assumption.

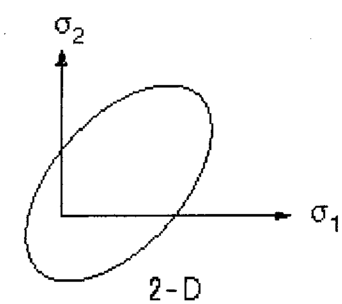


A bilinear representation of the stress-strain curves in each of the orthogonal directions (as well as the shear stress-shear strain curves) is used. No temperature-dependence is allowed with this option.

Anisotropic Plasticity

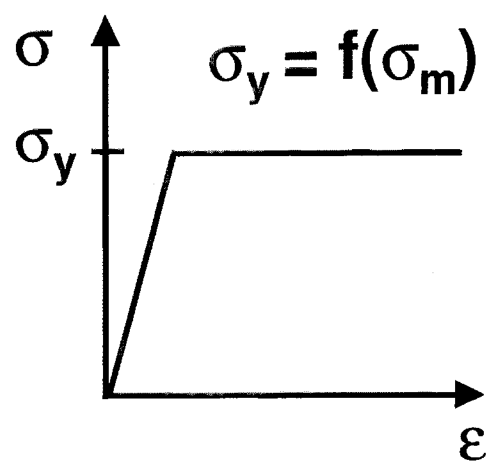
The initial yield surface is shifted in stress-space (if the yields in tension and compression are different) and is an elongated ellipse (if the yields in the different directions are different). ANISO is *not* recommended for major anisotropy such as composites. ANISO is only applicable for small-strain, proportional loading applications.





Drucker-Prager

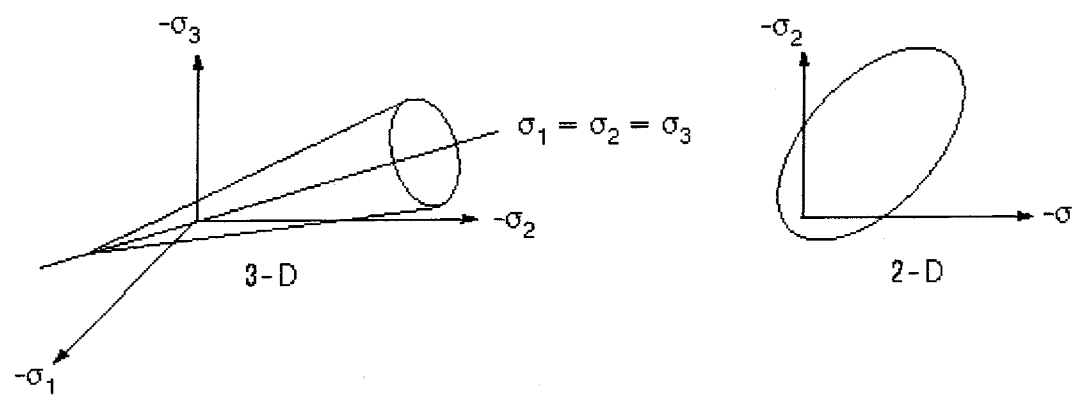
The Drucker-Prager (DP) model is applicable to granular materials such as soils, rock and concrete. A pressure-dependent von Mises yield criterion is used so that an increase in confinement pressure (hydrostatic stress, σ_m) produces a corresponding increase in the yield strength. Elastic-perfectly plastic material behavior is assumed.



The input consists of three constants; the cohesion value c, the angle (in degrees) of internal friction ϕ, and the dilatancy angle ϕ_f. The dilatancy angle ϕ_f controls the amount of volumetric expansion.

Drucker-Prager

The yield surface is a cone for the Drucker-Prager model. The yield stress in compression is greater than the yield stress in tension.



Note that the required input constants (c, ϕ, and ϕ_f) can be obtained from uniaxial data. Refer to the *ANSYS Theory Manual* for more details.

Anand's Model

Anand's model (ANAND) characterizes the large strain response of metals in a hot-working state. It is a rate-dependent model that allows nonlinear strain hardening as well as strain softening. The details of this model are described in the *ANSYS Theory Manual* and the references cited therein. Note that for Anand's model:

- The material temperature is assumed to be greater than one half of the melting temperature.
- Only isotropic elastic (and plastic) behavior is allowed.
- Only the elements Visco106, Visoc107, and Visco108 support this material model.

Element Formulations

For plasticity applications the following element formulations are available:

- Incompatible Modes (Extra Shapes)
  Solid45 default option, bending deformations
- Selective Reduced Integration (B-Bar)
  Nearly incompressible materials, bulk deformations
- Uniform Reduced Integration (URI)
  Nearly incompressible materials, bending deformations
- Mixed U-P Formulation
  Incompressible and nearly incompressible materials

Solid Element Recommendations

- For rate-independent plasticity the following elements are recommended:
- Bulk structural deformations with negligible bending use *Plane182, Solid185* Selective Reduced Integration (B-Bar).
- For small strain applications use incompatible mode elements *Plane42, Solid45.*
- For large strain applications use *Plane182* and *Solid185* with URI (preferred for large models) or *Solid95* with URI. Visco106, Visco107, and Visco108 can also be used (even for rate independent plasticity).

Beam and Shell Element Recommendations

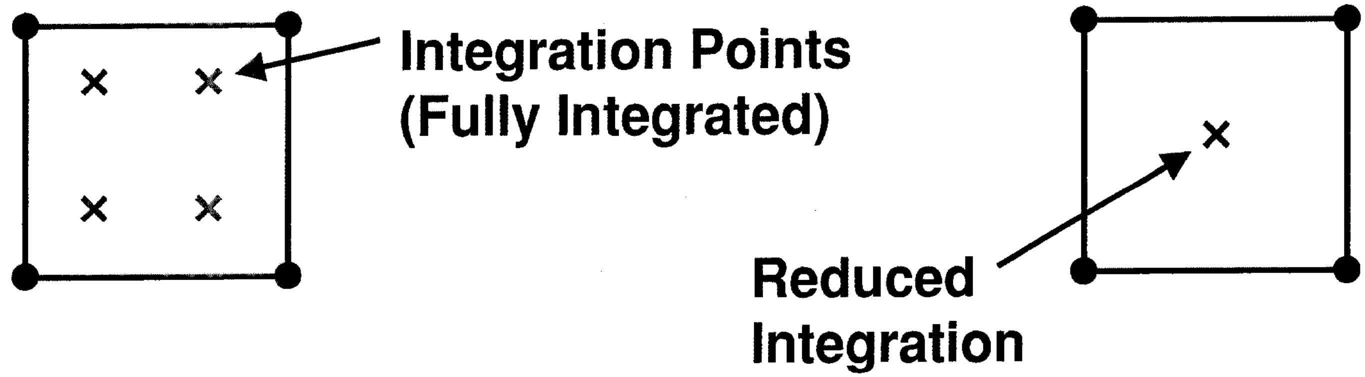
- For rate-independent plasticity the following beam and shell elements are recommended:

- For Isotropic Hardening plasticity (BISO or MISO) use the finite strain beam and shell elements *Beam188, Beam189, and Shell181.*

- For Kinematic Hardening plasticity (BKIN, MKIN or KINH) use the plastic beam and shell elements *Beam23, Beam24, Shell43, and Shell143.*

Meshing Considerations

Plasticity calculations take place at the finite element integration points. Therefore, it is important to consider the *integration point density* when meshing your model. Reduced integration elements (single integration point) will require a more refined mesh.

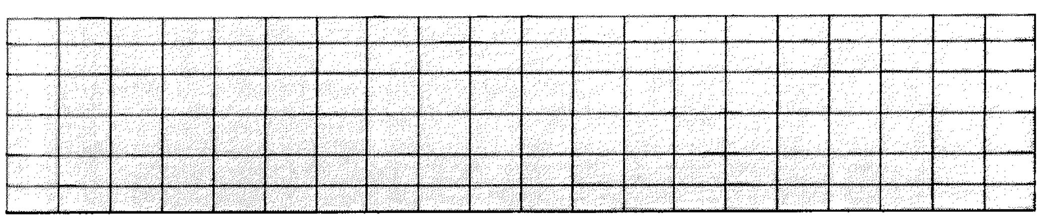


Meshing Considerations

For bending applications you need to have adequate refinement through the thickness and may want to grade the mesh towards the surface.

Plastic hinge regions also must be sufficiently discretized to capture the localized effect. If the problem is a large strain solution the mesh should be structured to ensure good element shapes throughout the element distortion.

Sample Mesh Density for Bending



Obtaining the Solution

Due to the path dependent nature of plasticity solutions the use of solution control is recommended (default).

The full Newton-Raphson option without adaptive descent is the recommended Newton-Raphson option (default with solution control).

Since plasticity is a path dependent phenomenon the load must be applied gradually. The use of automatic time stepping is recommended (default with solution control).

Obtaining the Solution

Plasticity requires that the load increments are small, especially in the case of a large strain analysis. Be sure to set a small enough initial and minimum time step for automatic time stepping.

The line search option (LNSRCH) can also be helpful as a convergence tool for plasticity analyses, especially for a large strain solution.

The predictor (PRED) can reduce the total number of iterations if the stress-strain curve is smooth. Do NOT use the predictor option if the problem has large rotations.

Obtaining the Solution

When solving a plasticity problem, *the actual load history must be followed to ensure the correct solution.* Path dependent problems also require that the *load be applied slowly* (using many substeps). The automatic time stepping will cutback the load increment if too large an equivalent plastic strain increment (> 15%) is taken within a substep. The maximum plastic strain increment can be changed with the use of CUTCONTOL.

Output Quantities

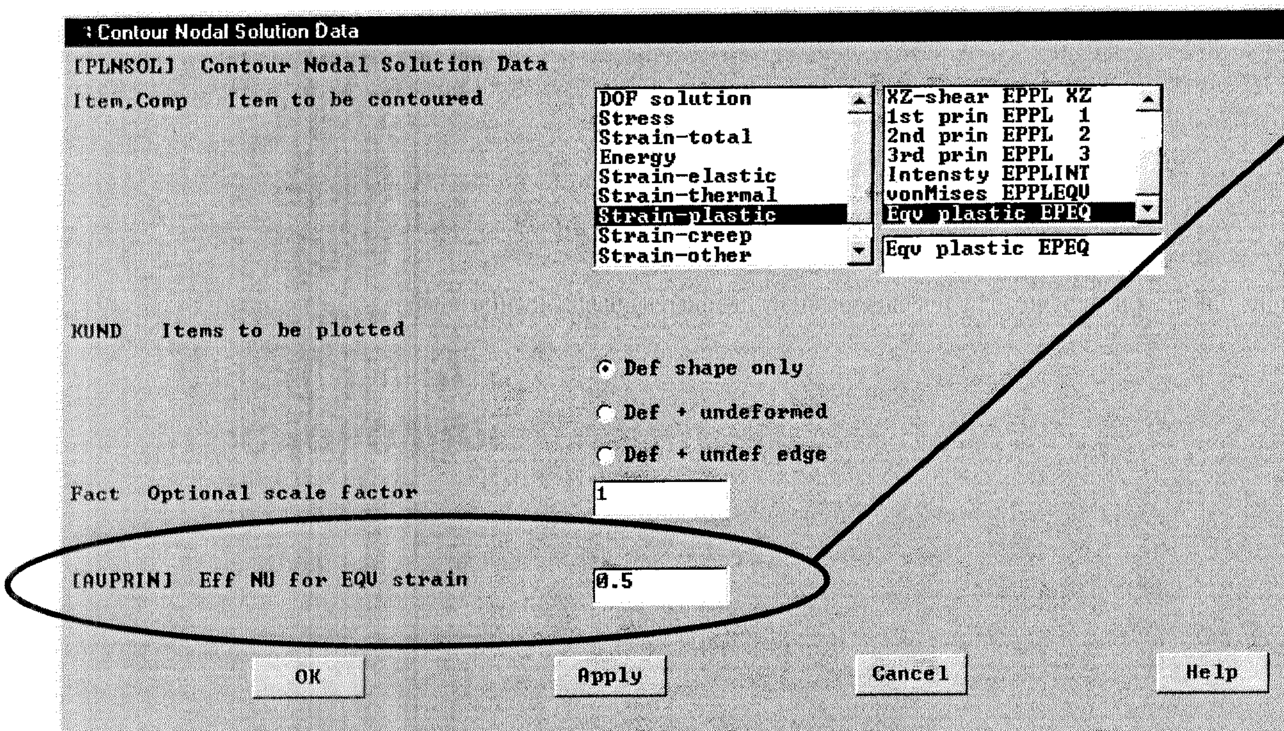
The following quantities are reported for each node in a plasticity analysis:

EPEL	Elastic Component Strains
EPPL	Plastic Component Strains
EPTO	Total Strain
EPEQ	*Accumulated Equivalent Plastic Strain*
HPRES	Hydrostatic Pressure Stress
SRAT	Stress Ratio
PLWK	Accumulated Plastic Work
PSV	Plastic State Variable

Output Quantities

To display the plasticity output quantities:

General Postproc > Plot Results > Nodal Solu ...



The effective Poisson's ratio (ν') for *equivalent* elastic, *equivalent* plastic, and *equivalent* total strain calculations (default is ν'=0)

Output Quantities

Elastic Component Strains (EPEL)

The *elastic component strains* are the current elastic strains in the model.

$$\varepsilon_x^{el} \quad \varepsilon_y^{el} \quad \varepsilon_z^{el} \quad \gamma_{xy}^{el} \quad \gamma_{yz}^{el} \quad \gamma_{xz}^{el}$$

Plastic Component Strains (EPPL)

The *plastic component strains* are the current plastic strains in the structure. These strains represent the summation of the plastic strain increments, $\Delta\varepsilon^{pl}$.

$$\varepsilon_x^{pl} \quad \varepsilon_y^{pl} \quad \varepsilon_z^{pl} \quad \gamma_{xy}^{pl} \quad \gamma_{yz}^{pl} \quad \gamma_{xz}^{pl}$$

Output Quantities

Total Component Strains (EPTO)

The *total component strains* are the total mechanical strains in the structure. They are the summation of the current component elastic strains (EPEL) and the current component plastic strains (EPPL).

$$\varepsilon_x^{tot} \quad \varepsilon_y^{tot} \quad \varepsilon_z^{tot} \quad \gamma_{xy}^{tot} \quad \gamma_{yz}^{tot} \quad \gamma_{xz}^{tot}$$

Output Quantities

Accumulated Equivalent Plastic Strain (EPEQ)

The *accumulated equivalent plastic* strain is the summation of the equivalent plastic strain increments. In other words, the accumulated equivalent plastic strain identifies where you are on the uniaxial stress-strain curve.

$$\varepsilon_{eqa}^{pl} = \sum \Delta \varepsilon_{eqv}^{pl}$$

Output Quantities

Interpretation of Equivalent Strains

The equivalent strain for elastic strain, plastic strain, and total strain can be computed from a general von Mises equation:

$$\varepsilon_{eqv} = \frac{1}{\sqrt{2}(1+\nu')}\left[(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2 + \frac{3}{2}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{xz}^2\right)\right]^{\frac{1}{2}}$$

where, ε_x, ε_y, etc. are the appropriate component strains, and ν' is the effective Poisson's ratio.

Output Quantities

Equivalent Elastic Strain

The *equivalent elastic strain* is related to the equivalent stress when $\nu' = \nu$ by the following:

$$\sigma_{eqv} = E\,\varepsilon_{eqv}^{el}$$

Equivalent Plastic Strain

The *equivalent plastic* strain is calculated based on the *current plastic strain components*. The *equivalent plastic strain* can only be related to the *accumulated equivalent plastic strain* under proportional loading and only when ν' is set to 0.5.

Output Quantities

Equivalent Total Strain

The *equivalent total strain* is calculated from the *total component strains* . For the *equivalent total strain* the effective Poisson's ratio needs to be set appropriately. If $\varepsilon^{pl} >> \varepsilon^{el}$ then use $\nu' = 0.5$. For other values the effective Poisson's ratio can be estimated by:

$$\nu' = \frac{1}{2} - \left(\frac{1}{2} - \nu\right)\frac{\varepsilon_{eqv}^{el}}{\varepsilon_{eqv}^{tot}}$$

Output Quantities

Hydrostatic Pressure Stress (HPRES)

The hydrostatic pressure stress is defined as:

$$\sigma_m = \frac{1}{3}\left(\sigma_x + \sigma_y + \sigma_z\right)$$

Stress Ratio (SRAT)

If the *stress ratio* is less than one the node is elastic. If the stress ratio is one or greater the node is currently undergoing active plastic straining.

Output Quantities

Plastic Work (PLWK)

The *accumulated plastic work* is output by Shell181, Plane182, Solid185, Beam188, Beam189, Visco106, Visco107, and Visco108.

Plastic State Variable (PSV)

The *plastic state variable* is output *only* by Visco106, Visco107, and Visco108. The plastic state variable is the deformation resistance for Anand's model, and the accumulated equivalent plastic strain for other options.

Output Quantities

Note on Total Strain in the Time History Postprocessor

The effective Poisson's ratio (ν') is only available in the general postprocessor. To display the stress-strain response at a point (for proportional loading) in the time history postprocessor, use the following:

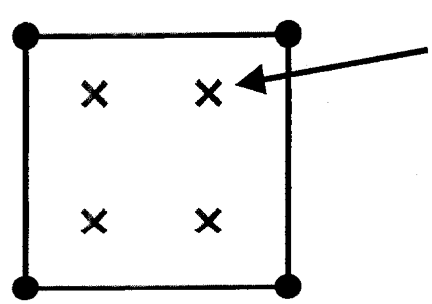
$$\varepsilon_{tot} = \frac{SEQV}{E} + EPEQ$$

where, SEQV is the von Mises stress

E is the elastic modulus, and
EPEQ is the accumulated equivalent plastic strain.

Output Quantities

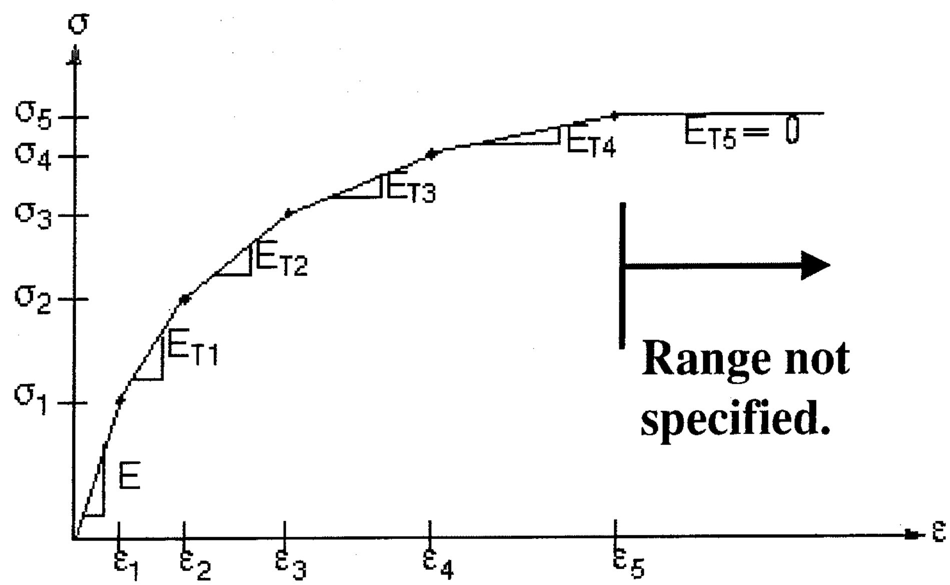
The nodal plasticity output quantities are the values of the integration point nearest to that node. If all integration points of an element are elastic, then the nodal elastic strains and stresses for that element are extrapolated to the node from their values at the integration points. If any integration points are experiencing plastic straining, then the reported nodal elastic strains and stresses for all nodes of that element are, in fact, the integration point values.



Integration Points values are *not* extrapolated for points with active plastic straining.

Trouble Shooting

Is the plastic strain state beyond the data supplied for the material? Perfect plasticity is assumed by ANSYS for the unspecified range. You may have a plastic hinge which is causing a physical instability.



Trouble Shooting

If incompatible mode elements are being used, is volumetric locking a problem? Review the hydrostatic pressure for a "checkerboard" pattern (alternating pressure values across elements). Refine the mesh and/or switch element types.

If reduced integration elements are being used, are hourglass modes a problem? Refine the mesh (preferred) or increase the hourglass stiffness factor.

Is the mesh density adequate? Be sure to adequately discretize the model to capture bending response. Plastic hinge regions also must be sufficiently discretized to capture the localized effect.

Trouble Shooting

Stress singularities should be avoided unless the elements in these areas are large. Examples of modeling features that create singularities are:

- Single point loads or single point constraints
- Re-entrant corners
- Single node connections between model parts
- Single node coupling or contact conditions

Stress singularities can cause local element distortion which can lead to divergence, or if using reduced integration, stress singularities can cause hourglass behavior.

NOTES

Chapter 6

HYPERELASTICITY

Session Objective

- At the end of this session you will be able to describe and demonstrate the following:

  1. Definition of Hyperelastic Material Behavior
  2. Hyperelastic Theory
  3. Hyperelastic Material Options
  4. Procedures for Using Hyperelastic Materials
  5. Determining Mooney-Rivlin Coefficients
  6. Obtaining the Solution
  7. Trouble Shooting

Hyperelasticity

- Hyperelastic materials are materials which can undergo very large recoverable (elastic) deformations. Strains on the order of a few hundred percent are typical. Since they are purely elastic in nature, hyperelastic materials are conservative and do not depend on the load path.

- Hyperelastic materials are typically used to model rubber and many other polymer materials.

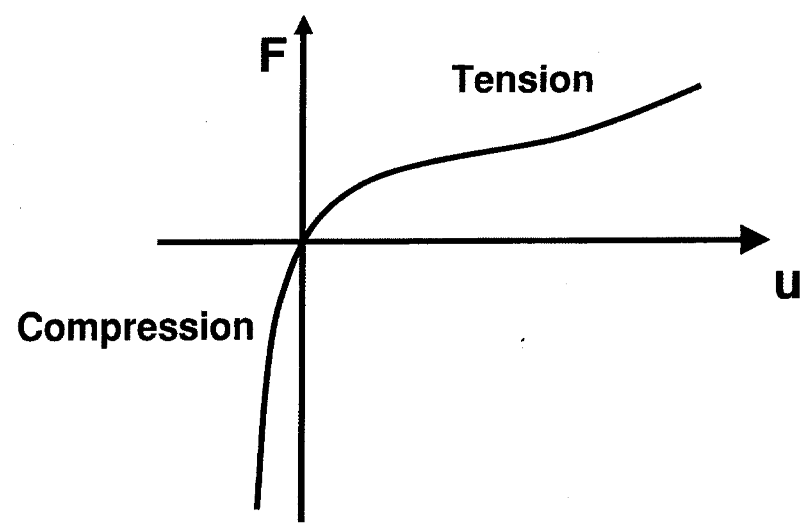
- The stresses for a hyperelastic material are usually derived from strain energy density functions. ANSYS provides two options to characterize this behavior. The Mooney-Rivlin option is suitable for incompressible materials. The Blatz-Ko option is applicable for compressible foam materials.

Hyperelasticity

- Rubber is a nearly incompressible material. Incompressible materials can undergo large deformations and strains without an appreciable change in volume. Nearly incompressible materials generally have Poisson's ratio values between .48 and .5.

A typical force deflection curve for a rubber bar.

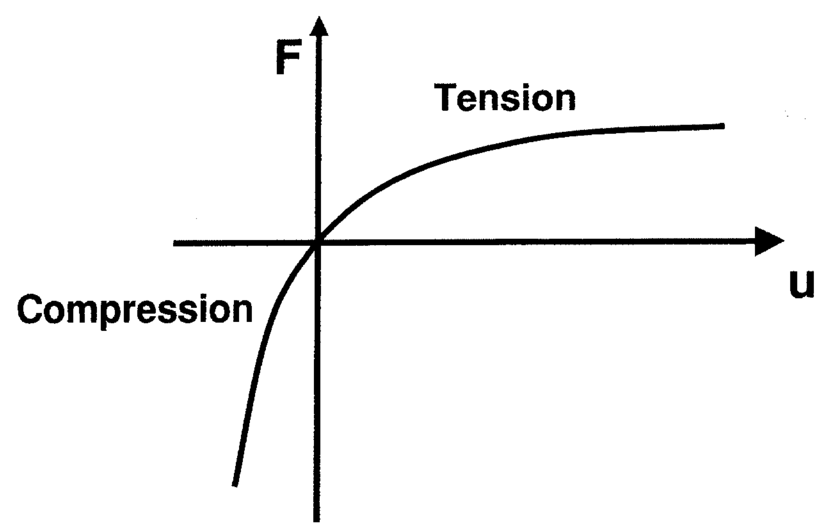
Note that in tension the response softens then stiffens again. In compression the material stiffens dramatically.



Hyperelasticity

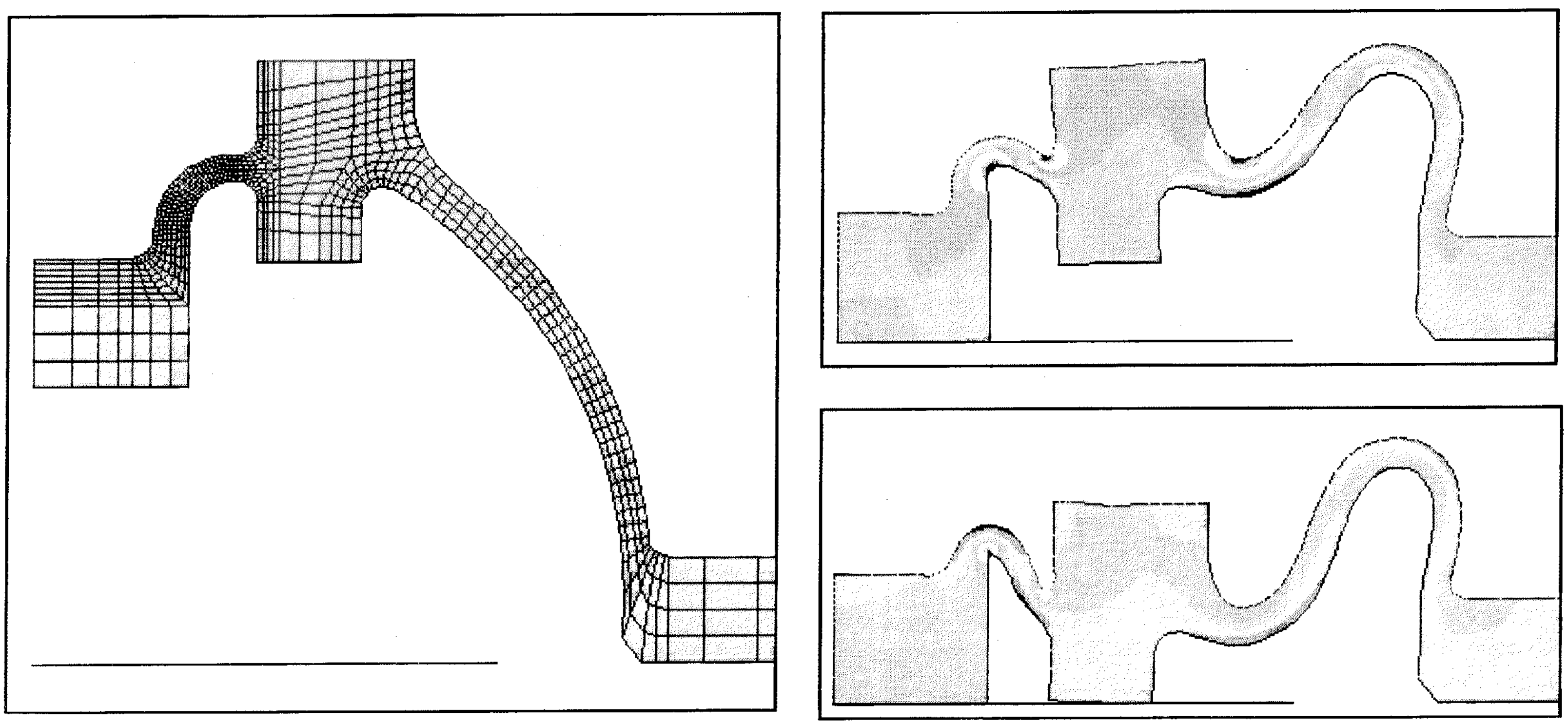
Foam is a hyperelastic material which is compressible. Shown below is a typical force deflection curve for a resilient foam material, as opposed to a rigid (crushable) foam.

Note that in tension the material is very weak. In compression the material stiffens as the voids in the foam close.



Hyperelasticity

The following example exhibits the large recoverable deformations which hyperelastic materials can undergo.



Snap-Through of a Hyperelastic Keyboard Dome

Hyperelastic Theory

- We will briefly touch on the following aspects of hyperelastic theory:
  - The Strain Energy Density Function
  - Incompressibility
  - The Hyperelastic Elements

Hyperelasticity Theory

Strain Energy Density Function

For a given strain value, the stresses in a hyperelastic material can be determined from the derivative of the strain energy density function (W) with respect to the strain components.

$$[S] = \frac{\partial W}{\partial [E]}$$

where [E] are the known Green-Lagrange strains and [S] are the computed 2nd Piola-Kirchoff stresses. W is in terms of the energy per unit original volume.

Hyperelasticity Theory

- In the ANSYS program, we assume that the hyperelastic materials are isotropic, having identical material properties in every direction. In this case, we can write the strain energy density function in terms of the strain invariants I1, I2, and I3.

- Strain invariants are measures of strain which are independent of the coordinate system used to measure the strains. Their use implies that isotropic material behavior has been assumed. Both Mooney-Rivlin and Blatz-Ko strain energy density functions are written in terms of the strain invariants.

Hyperelasticity Theory

An example of a strain energy density equation written in terms of the strain invariants is:

$$W = a_{10} (I_1 - 3) + a_{01} (I_2 - 3) + 1/2\ \kappa (I_3 - 1)^2$$

where a_{10} and a_{01} are material constants and κ is the bulk modulus. (This is the two-parameter Mooney-Rivlin strain energy density function for incompressible materials.)

The particular strain energy density options available will be discussed later in this chapter.

Hyperelasticity Theory

Incompressibility

- Many hyperelastic materials are nearly incompressible (for example rubber.) The incompressibility condition must be accounted for in the element formulation.

- The incompressible hyperelastic elements (u-p formulation) include pressure as an independent degree of freedom in addition to displacements.

- These pressure DOFs allow the incompressibility constraint to be satisfied without degrading the overall solution. The pressure DOFs are internal DOFs and are condensed out within the element (like extra-shape terms).

Hyperelasticity Theory

ANSYS has two families of hyperelastic elements. One for incompressible material behavior (rubber), and one for compressible material applications (foam).

Element Description	Incompressible	Compressible
4-Node 2D Solid (Plane Strain or Axisymmetric)	HYPER56	HYPER84 KEYOPT(1)=0 KEYOPT(2)=1
8-Node 2D Solid (Plane Strain or Axisymmetric	HYPER74	HYPER84 KEYOPT(1)=1 KEYOPT(2)=1
8-Node 3D Brick	HYPER58	HYPER86
10-Node 3D Tetrahedron	HYPER158	N/A
4-Node 3D Shell	SHELL181	N/A

Hyperelasticity Options

- Three material options are available in the program:
  - Mooney-Rivlin is a material option applicable to incompressible rubber or rubber-like materials. Two, five, or nine material constants are used to describe the material behavior.
  - Blatz-Ko is a material option applicable to compressible foam-type materials such as polyurethane rubbers. The only material constant required is the initial shear modulus G.
  - User Programmable Hyperelasticity

Mooney-Rivlin

- If the Mooney-Rivlin constants for a material to be analyzed are known, the hyperelastic constants can be input into ANSYS directly.

- If the Mooney-Rivlin constants are unknown for a material, ANSYS can derive the constants from laboratory test data. (We will discuss the procedure for entering the constants and determining the constants from test data later.)

- ANSYS supports a two, five or nine parameter Mooney-Rivlin model. The form of the strain energy density function is shown on the following pages.

Mooney-Rivlin

The strain energy density function (W) is given in polynomial form by:

$$W = \sum_{k+l=1}^{N} a_{kl}(\mathrm{I}_1 - 3)^k (\mathrm{I}_2 - 3)^l + 1/2\kappa(\mathrm{I}_3 - 1)^2$$

where,

a_{kl} **are the constants of the nine parameter cubic Mooney-Rivlin relationship**

$$\kappa = \frac{2(a_{10} + a_{01})}{(1 - 2\nu)}$$ **(Bulk Modulus)**

I_1, I_2, I_3 **are the Strain Invariants**

NOTE: $\kappa = \dfrac{E}{3(1 - 2\nu)}$ **, giving** $E \approx 6(a_{10} + a_{01})$

Mooney-Rivlin

The strain energy density function (W) is shown for the two, five, and nine parameter Mooney-Rivlin models:

Two Parameter Mooney-Rivlin Model

$$W = a_{10}\,(I_1 - 3) + a_{01}\,(I_2 - 3) + 1/2\,\kappa\,(I_3 - 1)^2$$

Five Parameter Mooney-Rivlin Model

$$W = a_{10}(I_1-3) + a_{01}(I_2-3) + a_{20}(I_1-3)^2 + a_{11}(I_1-3)(I_2-3) + a_{02}(I_2-3)^2 + 1/2\,\kappa(I_3-I)^2$$

Nine Parameter Mooney-Rivlin Model

$$W = a_{10}(I_1-3) + a_{01}(I_2-3) + a_{20}(I_1-3)^2 + a_{11}(I_1-3)(I_2-3) + a_{02}(I_2-3)^2 + a_{30}\,(I_1-3)^3 + a_{21}(I_1-3)^2\,(I_2-3) + a_{12}(I_1-3)\,(I_2-3)^2 + a_{03}(I_2-3)^3 + 1/2\,\kappa(I_3-I)^2$$

Mooney-Rivlin

- The Mooney-Rivlin parameters must satisfy certain constraints in order to produce valid material behavior. These constraints are imposed in order to maintain:
  - Positive definiteness of the strain energy density function
  - Stress is always positive (or negative) at the extremes of uniaxial deformation
  - Stress is a continuous function of deformation
- If any constraint is not satisfied, the program will issue a warning message.

Mooney-Rivlin

Positive definiteness simply means that when the material is deformed, the strain energy must increase.

For the two parameter model there is one constraint:

1. $a_{10} + a_{01} > 0$

For the five parameter model the constraints are:

1. $a_{10} + a_{01} > 0$
2. $a_{20} > 0$
3. $a_{02} < 0$
4. $a_{20} + a_{11} + a_{02} > 0$

For the nine parameter model the constraints are:

1. $a_{10} + a_{01} > 0$
2. $a_{30} > 0$
3. $a_{03} < 0$
4. $a_{30} + a_{21} + a_{12} + a_{03} > 0$

Mooney-Rivlin

- These constraints are usually violated when insufficient experimental data is used to derive the constants. As long as your analysis is within the range of the test data and undergoes the same mode of deformation, then the constraints may be violated without affecting the solution accuracy or its convergence properties.

- If the constraints are violated and your model experiences modes of deformation which you do not have experimental data for, then you may encounter convergence difficulties and you should carefully validate your results. It is not recommended that you perform analysis outside the range of test data.

Blatz-Ko

- The only material constant required for this material option is the initial shear modulus G. The shear modulus is derived by the program from the input of the elastic modulus and Poisson's ratio, G= E/(2(1 + ν)).

- This option is available only with the compressible elements. See the hyperelastic element descriptions.

- The strain energy density function for a Blatz-Ko material has the form:

$$W = \frac{G}{2}\left(\frac{I_2}{I_3} + 2\sqrt{I_3} - 5\right)$$

Hyperelasticity Procedure

- As with any other analysis, the procedure for using hyperelasticity consists of three main steps:
  - Build the model
  - Obtain the solution
  - Review the results

Select Appropriate Element

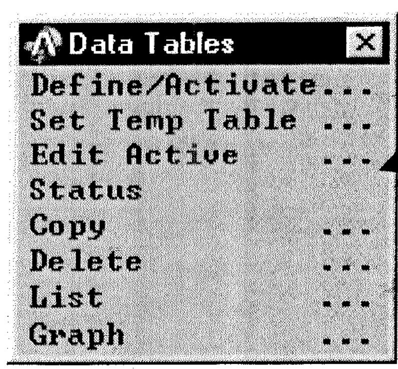
Depending on whether you want to model compressible or incompressible material and the model dimensionality, select the appropriate element type.

Element Description	Incompressible	Compressible
4-Node 2D Solid (Plane Strain or Axisymmetric)	HYPER56	HYPER84 KEYOPT(1)=0 KEYOPT(2)=1
8-Node 2D Solid (Plane Strain or Axisymmetric	HYPER74	HYPER84 KEYOPT(1)=1 KEYOPT(2)=1
8-Node 3D Brick	HYPER58	HYPER86
10-Node 3D Tetrahedron	HYPER158	N/A
4-Node 3D Shell	SHELL181	N/A

Known Material Constants

If the material constants are known for a hyperelastic material, they can be input from the following menu path.

Preprocessor > Material Properties > Data Tables >



The Mooney-Rivlin data table can then be activated and edited from the Data Tables menu.

Procedure For Determining Mooney-Rivlin Constants

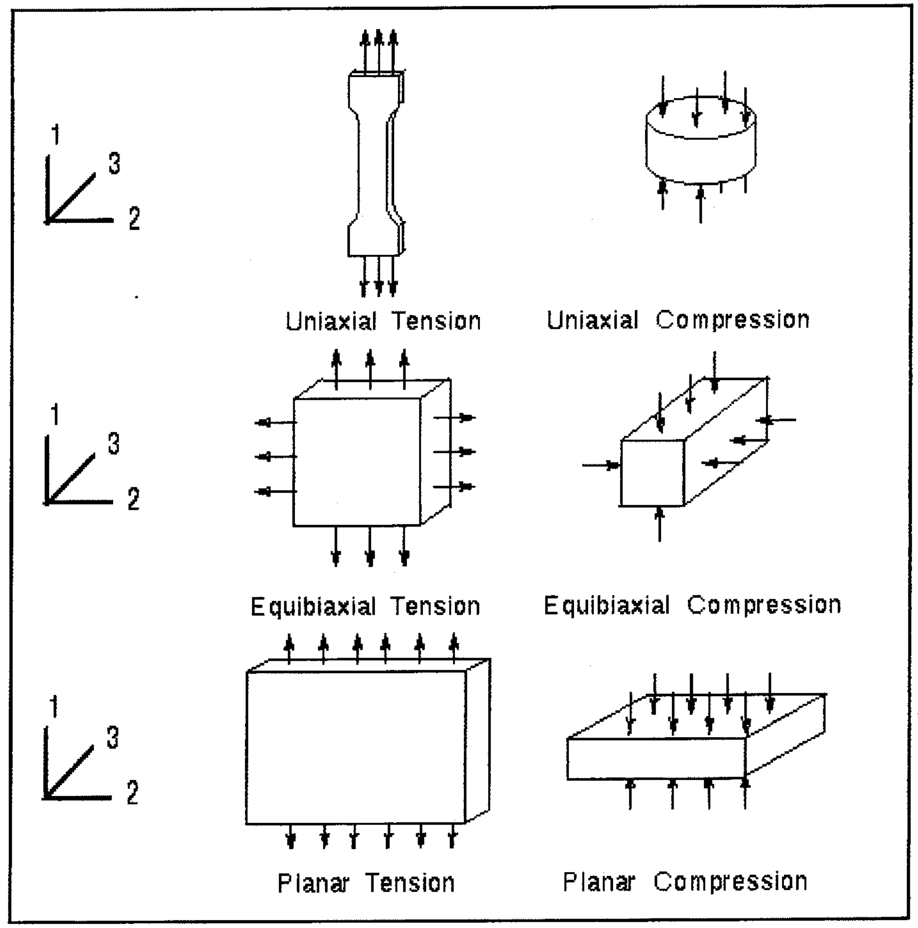
- The Mooney-Rivlin constants for hyperelastic materials are not generally available in the open literature. Therefore, most sets of Mooney-Rivlin constants need to be derived from experimental stress-strain data.

- For hyperelastic materials, simple deformation test data can be used to accurately characterize the Mooney-Rivlin constants. ANSYS can determine the Mooney-Rivlin constants from experimental data.

Mooney-Rivlin Constants

Test data may come from one or more of the following six tests:

- Uniaxial Tension
- Uniaxial Compression
- Equibiaxial Tension
- Equibiaxial Compression
- Planar Tension (Shear)
- Planar Compression (Shear)



Mooney-Rivlin Constants

- Although ANSYS accepts up to six different deformation states, it can be shown that apparently different loading conditions have identical deformations. This is due to the incompressibility of hyperelastic materials which allows the superposition of hydrostatic stresses without altering a material's deformation.

- Upon addition of hydrostatic stresses, the following modes of deformation can be shown to be identical:

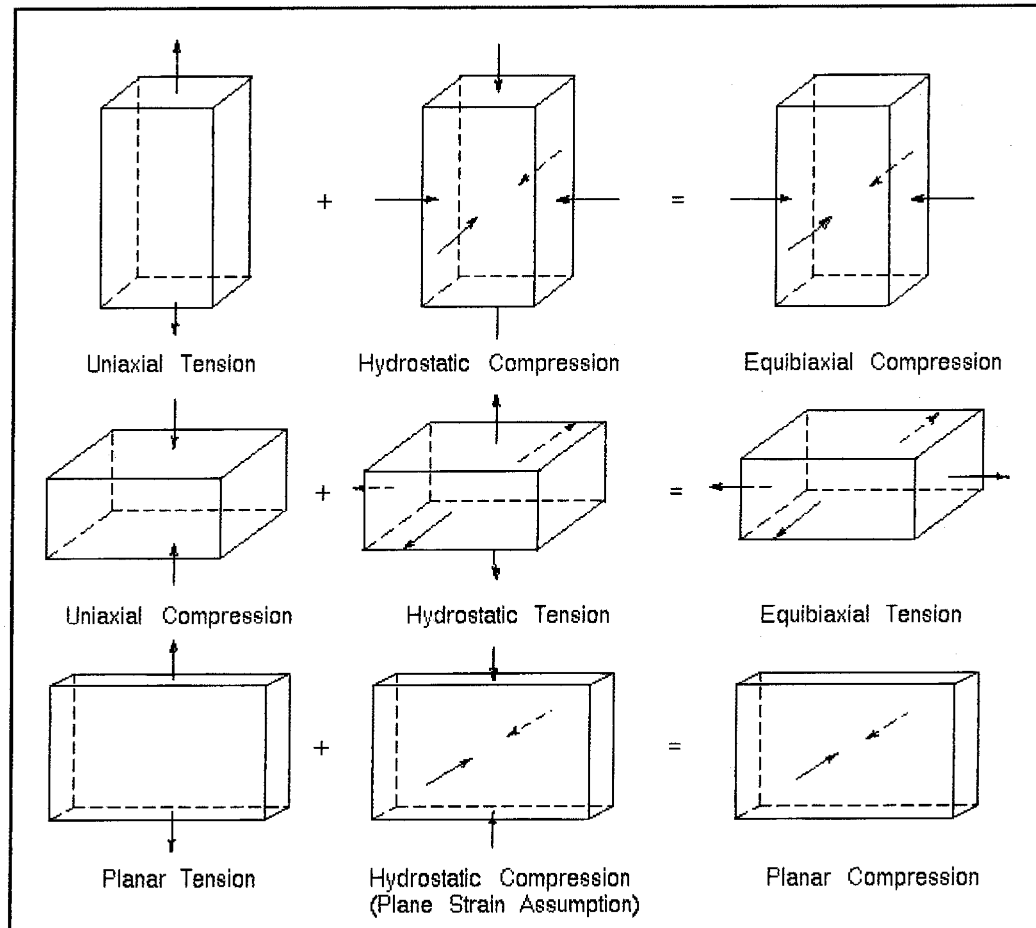
    1. Uniaxial Tension and Equibiaxial Compression

    2. Uniaxial Compression and Equibiaxial Tension

    3. Planar Tension and Planar Compression

Mooney-Rivlin Constants

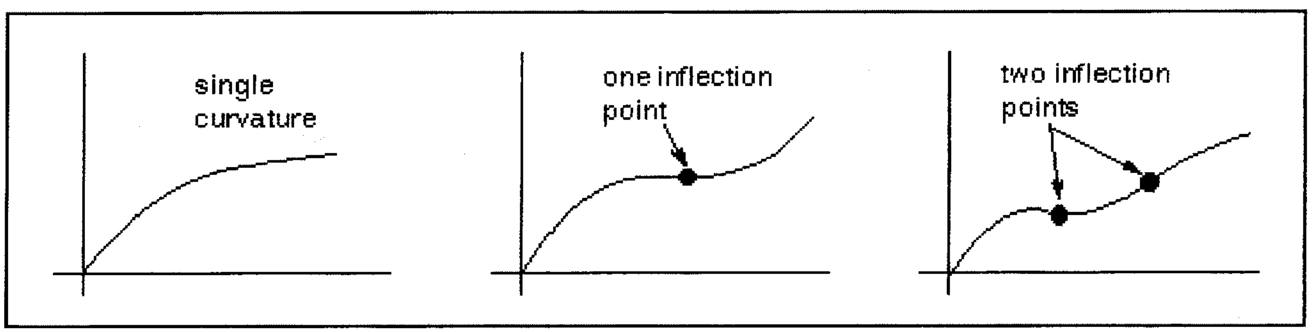
There are only three independent types of stress-strain data for hyperelastic material analysis. The equivalent modes of deformation upon addition of hydrostatic stress are shown to the right.



Mooney-Rivlin Constants

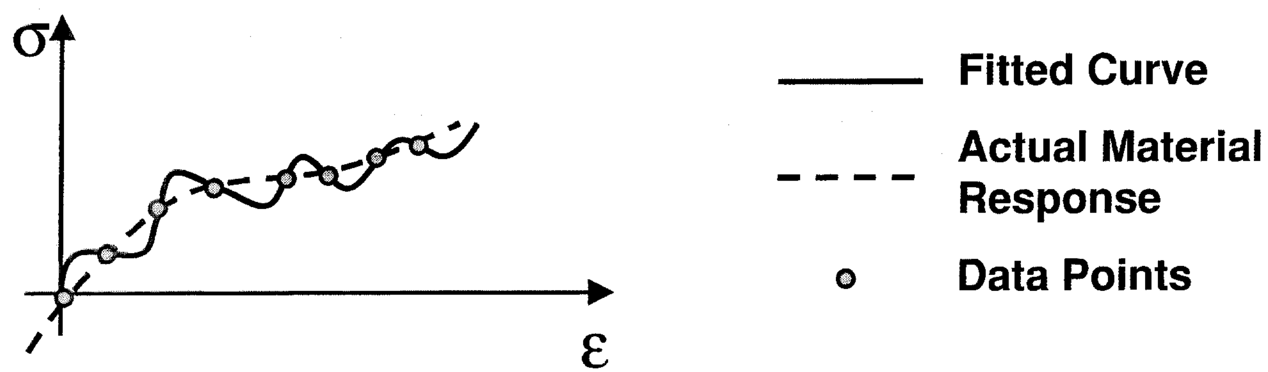
How many Mooney-Rivlin constants should be used?

If the stress-stain curve contains ...	Try this Mooney-Rivlin function ...
No inflection points (single curvature)	Two-term
One inflection point (double curvature)	Five-term
Two inflection points	Nine-term



Mooney-Rivlin Constants

- You should use at least twice as many data points as the desired number of constants to be calculated. Using more terms usually improves the statistical quality of the curve fit (more tightly fitted through the data points), but the overall shape of the curve may be worse.



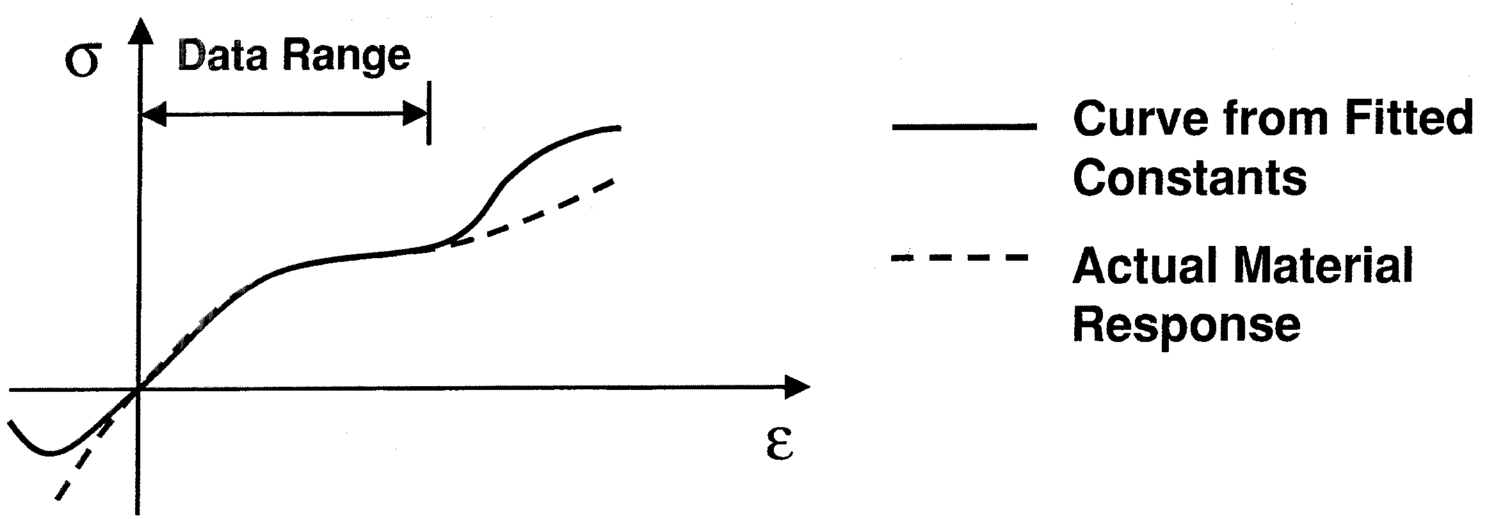
- As a practical matter, you should probably select the two, five, or nine constant function which results in the best combination of tight data fit and satisfactory curve shape.

Mooney-Rivlin Constants

- Mooney-Rivlin constants should be derived using test data which represents all modes of deformation and strain ranges that the model will experience:
  - If you have only uniaxial tension data, do not create a model which experiences significant shear deformations.
  - If your test data extends between 0% and 100% strain, do not make a model that experiences 150% strain.
- Using strains beyond the range of the test data can produce erroneous results.

Mooney-Rivlin Constants

- Note that where the supplied data points are available, the derived constants match the actual data well. Outside the range the constants may produce a response that does not match the physical response. This can cause convergence difficulties.



Determining Mooney-Rivlin Constants

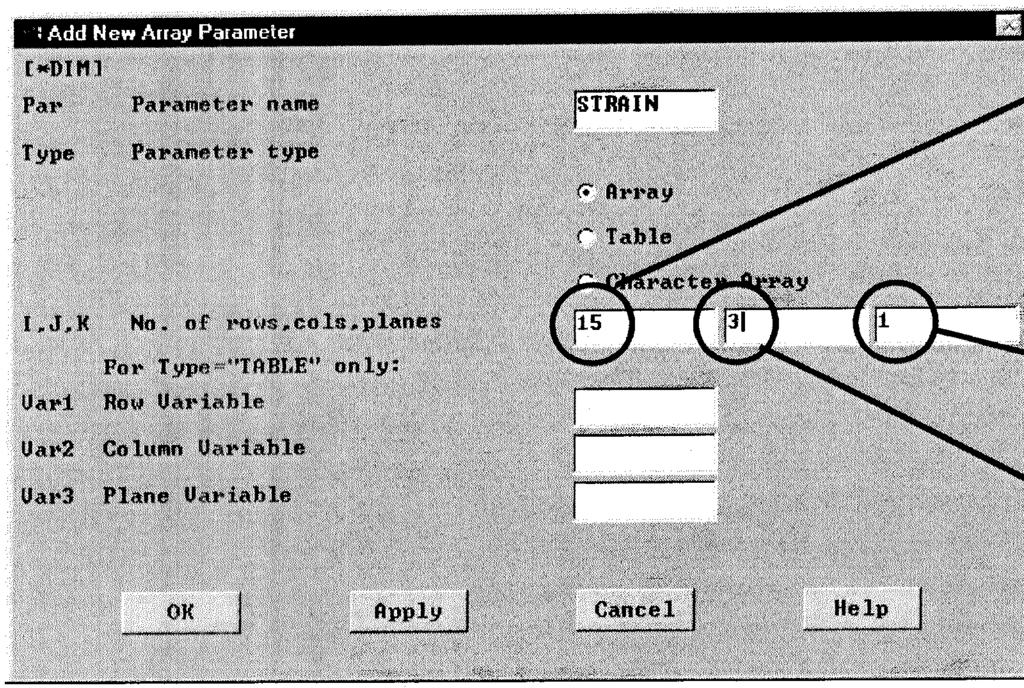
The GUI procedure for determining and applying the Mooney-Rivlin constants consists of four main steps:

1. Dimension the Stress and Strain Data Arrays
2. Fill in the Stress and Strain Data Arrays
3. Calculate the Mooney-Rivlin Constants
4. Evaluate the Quality of the Mooney-Rivlin Constants

Mooney-Rivlin Constants

Dimension the Stress and Strain Data Arrays

Parameters > Array Parameters > Define/Edit > Add



The maximum number of data points from any *one* test.

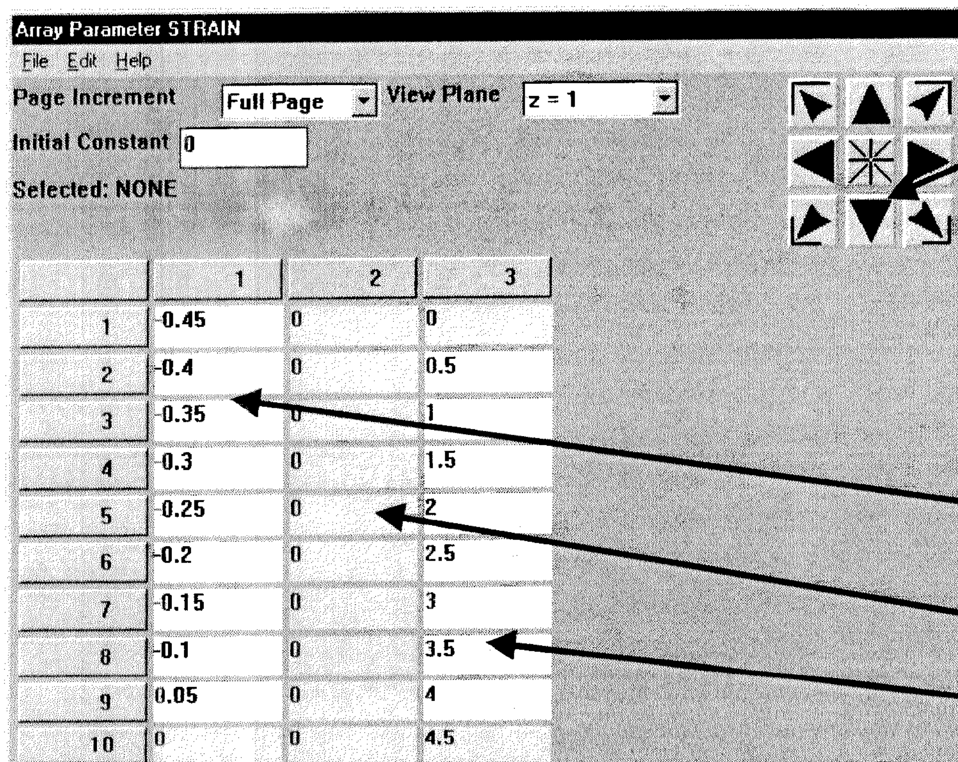
Always **1**

Always **3 (for 3 test types)**

Mooney-Rivlin Constants

Fill in the Stress and Strain Data Arrays

Parameters > Array Parameters > Define/Edit >



	1	2	3
1	-0.45	0	0
2	-0.4	0	0.5
3	-0.35	0	1
4	-0.3	0	1.5
5	-0.25	0	2
6	-0.2	0	2.5
7	-0.15	0	3
8	-0.1	0	3.5
9	0.05	0	4
10	0	0	4.5

Note, you can hit the down arrow to scroll down to fill in the rest of the array data.

Uniaxial (+/-)

Biaxial (+/-)

Planar (+/-)

Mooney-Rivlin Constants

Fill in the Stress and Strain Data Arrays

The stress and strain information needs to be input as engineering stress and engineering strain.

Each column of the array refers to a *specific* test type as shown in the table below:

Array Column	**Test Type**
First Column	Uniaxial Tension and/or Uniaxial Compression
Second Column	Equibiaxial Tension and/or Equibiaxial Compression
Third Column	Shear (Planar Tension and/or Compression)

Mooney-Rivlin Constants

Fill in the Stress and Strain Data Arrays

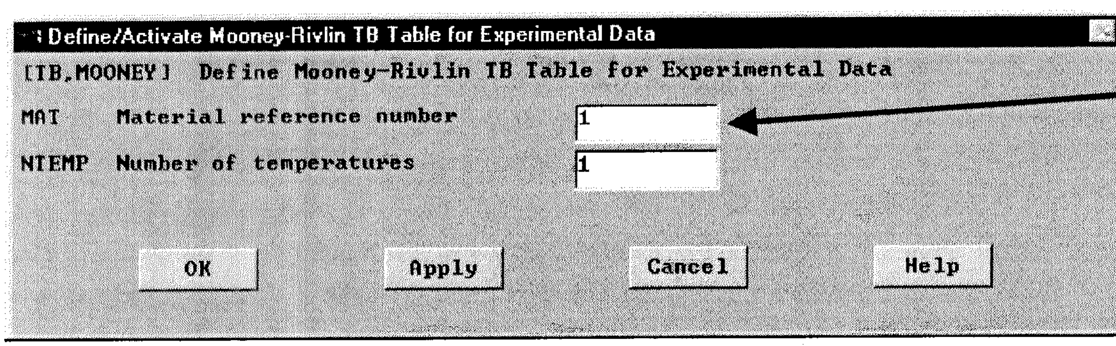
Note that the columns of the STRESS and STRAIN arrays do *NOT* have a 1:1 correspondence with the modes of deformation. Follow the table to determine which test corresponds to each column.

If you do not have data for all three columns, you *must* leave the missing columns blank.

Mooney-Rivlin Constants

Calculate the Mooney-Rivlin Constants

Preprocessor > Material Props > Mooney-Rivlin > Define Table >



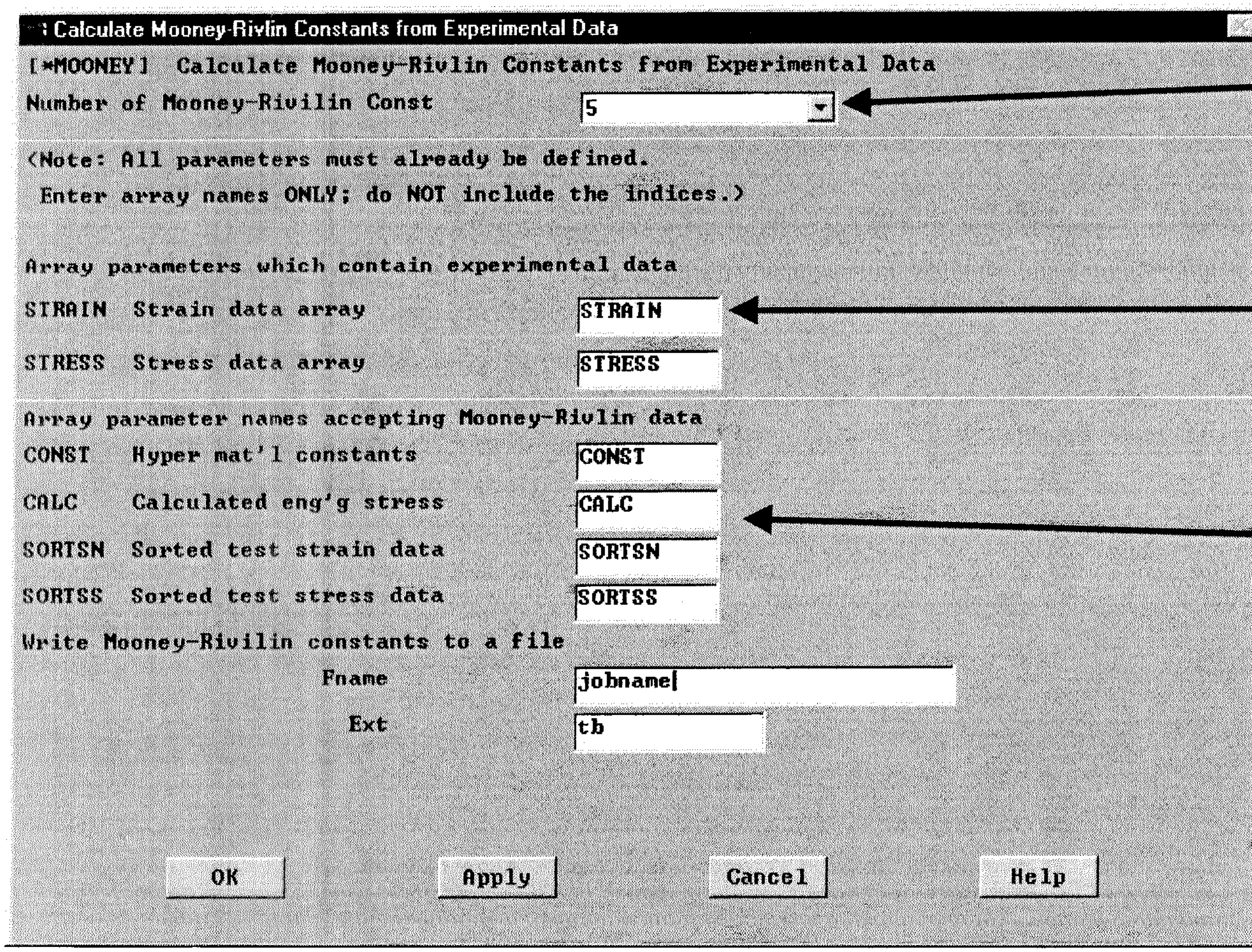
Enter material number and number of temperatures.

Mooney-Rivlin Constants

Calculate the Mooney-Rivlin Constants

- Preprocessor > Material Props > Mooney-Rivlin > Calculate Constants
- This will execute the GUI function to calculate the Mooney-Rivlin constants (shown on the next slide).
- The *MOONEY function will create the Mooney-Rivlin constants and store them in three places:
  - the database
  - the array CONST
  - the file specified (defaults to jobname.tb.)
- For a given material, after *one* use of *MOONEY, you can (in subsequent analyses) simply issue */INPUT, jobname,tb* to read in these constants.

Mooney-Rivlin Constants



Number of Constants

Names of Data Arrays

Arrays created automatically from the GUI.

Mooney-Rivlin Constants

Evaluate the Quality of the Mooney-Rivlin Constants

The Output Window will contain the following:

```
FOLLOWING TEST DATA TYPES HAVE BEEN SPECIFIED FOR COMPUTING 5 TERM MOONEY-
  RIVLIN SERIES

        TYPE OF TEST DATA      # OF DATA POINTS
        -----------------      ----------------
         UNIAXIAL                     9
         EQUIBIAXIAL                  0
         SHEAR                       15

 ROOT MEAN SQUARE (RMS) ERROR AND COEFFICIENT OF DETERMINATION (COD) FOR 2, 5
  AND 9 TERM SERIES

 # OF TERMS    COEFFICIENTS COMPUTED     RMS ERROR (%)    COD
 ----------    ---------------------     -------------   ------
     2         C10,C01                    0.1496E+02      1.0000
     5         C10,C01,C20,C11,C02        0.9439E+01      1.0000
     9         C10,C01,C20,C11,C02,       0.7240E+01      1.0000
               C30,C21,C12,C03
```

Mooney-Rivlin Constants

Evaluate the Quality of the Mooney-Rivlin Constants

The Output Window will contain the following (continued):

```
The user has chosen 5 term series for which the following constants have been
computed

    C10 =   -0.547267E+00
    C01 =    0.699294E+00
    C20 =    0.165070E+01
    C11 =   -0.249608E+01
    C02 =    0.845377E+00


 *** WARNING *** C02 IS NON-NEGATIVE.
 *** CHECK RESULTS CAREFULLY ***
```

Mooney-Rivlin Constants

Evaluate the Quality of the Mooney-Rivlin Constants

- The root mean square (RMS error) and the coefficient of determination (COD) are statistical measures of the quality of the curve fit. The RMS error expressed as a percentage, should be "close" to zero. The coefficient of determination will be less than 1.0, but should be "close" to 1.0 (typically 0.99 or better).
- The output file will also contain any warning messages if any of the constraints were not satisfied for the calculated Mooney-Rivlin constants.

Mooney-Rivlin Constants

Evaluate the Quality of the Mooney-Rivlin Constants

In addition to checking the RMS error and the COD, you should also plot the calculated stress-strain data versus the experimental stress-strain data.

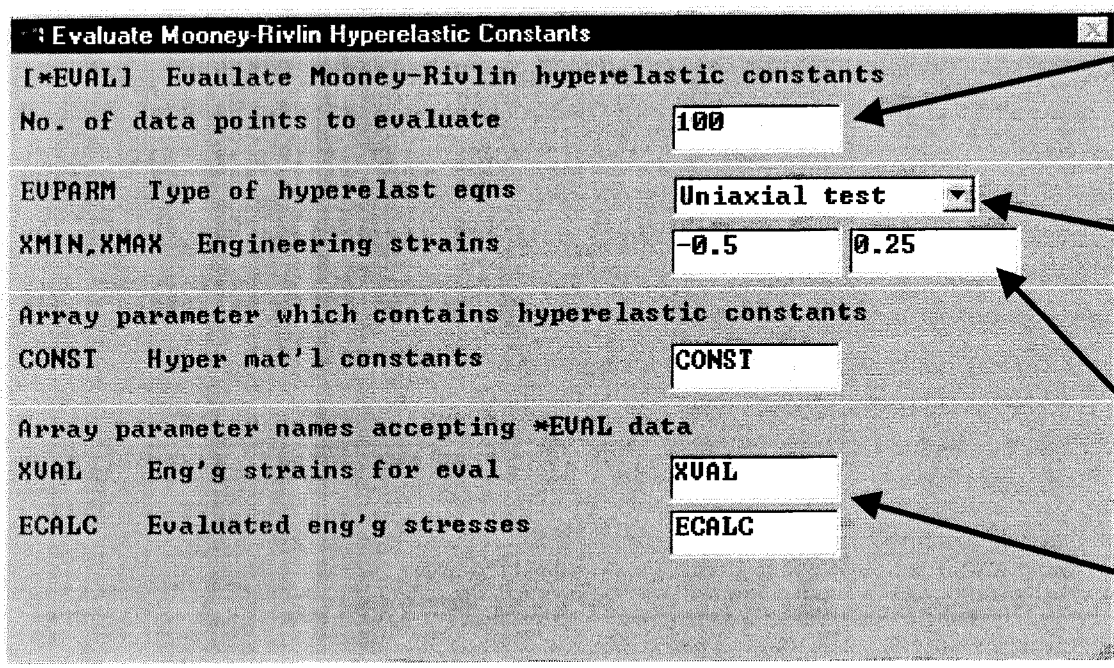
There are two steps to this process -

- Calculate Stress and Strain Values
- Graph the Calculated versus the Experimental Values

Mooney-Rivlin Constants

Calculated Stress-Strain Data

Material Props > Mooney-Rivlin > EvaluateConst



Number of calculated stress-strain points

Type of test for calculated data

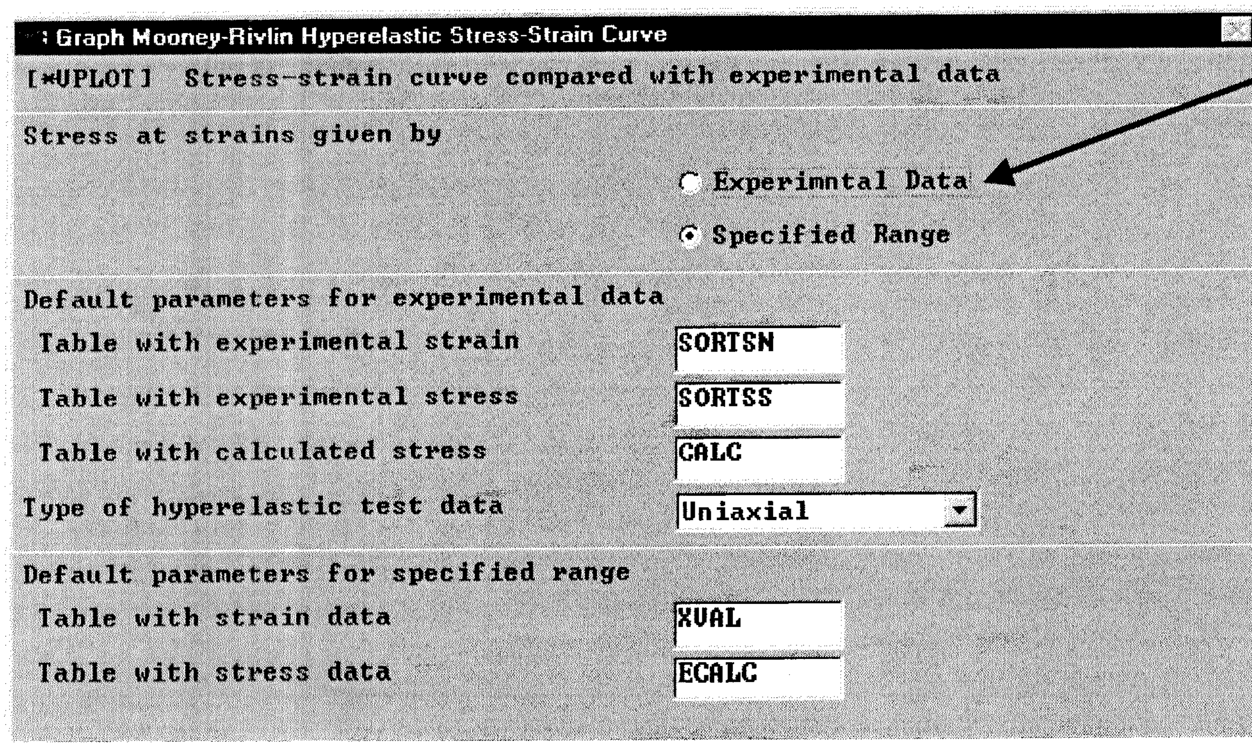
Strain Range

Calculated Stress and Strain Arrays

Mooney-Rivlin Constants

Graph Calculated Values versus Experimental Data

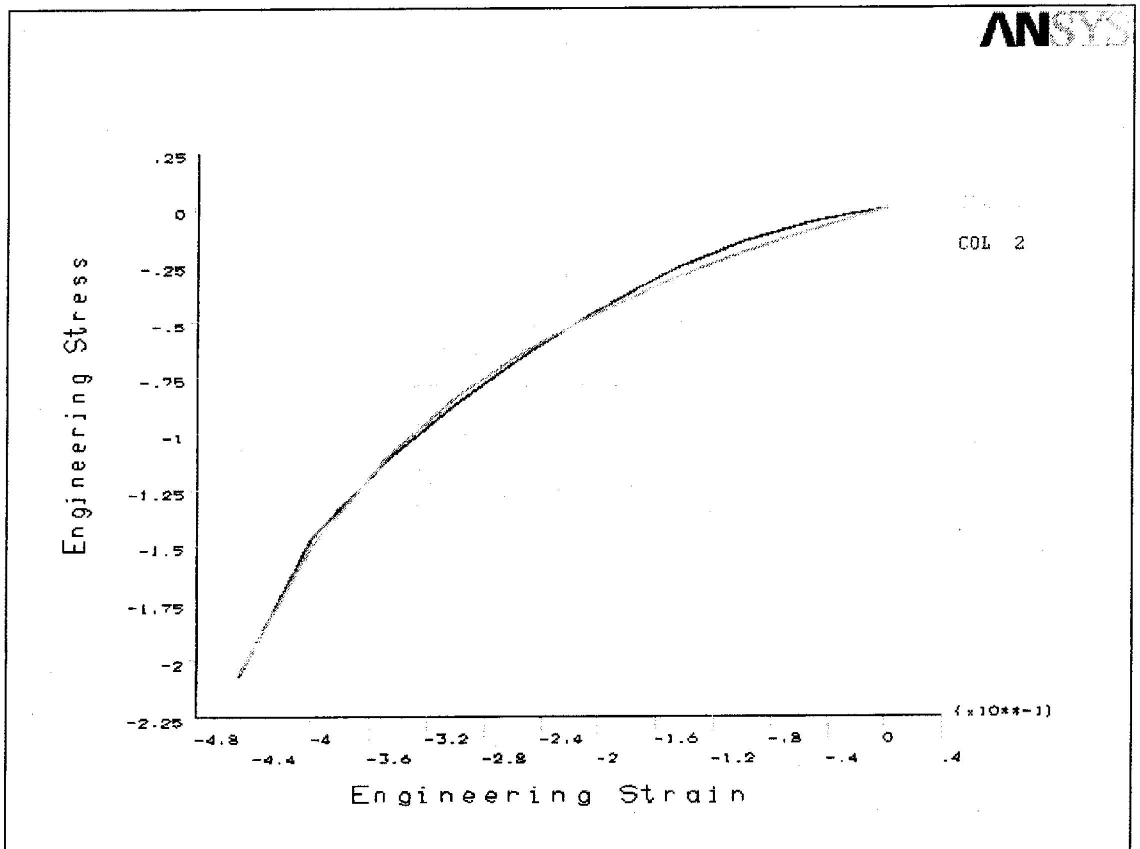
Material Props > Mooney-Rivlin > Graph ...



Controls the strain range of the graph, useful for showing how the Mooney-Rivlin model performs outside of the experimental range.

Mooney-Rivlin Constants

Graph Calculated Values versus Experimental Data



The graph shows the calculated values versus the experimental values for the range of the experimental data.

Mooney-Rivlin Constants

Note on Extrapolated Data

Evaluating the Mooney-Rivlin constants can be used to evaluate the Mooney-Rivlin model outside of the range of the experimental data. However, use extrapolated data with caution! If you do not have enough test data to cover the full range of stains for your model, *get more test data!*

Poisson's Ratio

- The only other material property which needs to be specified for a Mooney-Rivlin model is Poisson's ratio. For an incompressible material, Poisson's ratio is 0.5.
- For HYPER56, 58, 74, and 158, Poisson's ratio should not exceed 0.499999 (five nines).
- For SHELL181 in plane stress applications, you may use 0.5 exactly.

Obtaining the Solution

- Instabilities can occur during a hyperelastic analysis. These generally show up as a negative pivot error. The instability can be physical or numerical.

- Physical instabilities can occur if the component buckles or wrinkles. The instability can be global (the entire component buckles) or can be localized, such as surface wrinkling in areas of high compressive stress.

- Numerical instabilities can occur if the material constants produce a non-positive definite strain energy density function, especially if the deformation is outside the range of the experimental data. They can also occur during the enforcement of the incompressibility constraint.

Obtaining the Solution

- Hyperelasticity requires a large strain solution. Be sure that a large strain solution is activated (NLGEOM,ON).
- Due to the highly nonlinear nature of hyperelastic solutions, use of solution control is recommended (default).
- The full Newton-Raphson option without adaptive descent is the recommended Newton-Raphson option (default with solution control).

Obtaining the Solution

- Hyperelasticity requires that the load increments are small, especially in the case of an incompressible material. Be sure to set a small enough minimum time step for automatic time stepping.
- The line search option (LNSRCH) can also be helpful as a convergence tool for hyperelastic analyses.
- If you suspect you are experiencing a numerical instability use slightly different values for the Mooney-Rivlin constants, or recalculate the constants using more data points.

Review the Results

Stresses are output as true stress in the original element coordinate system.

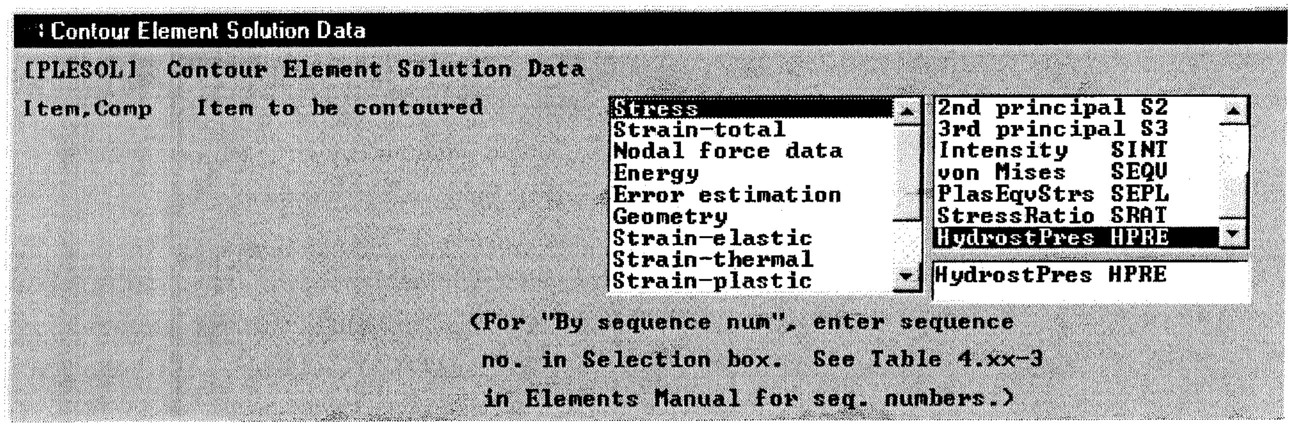
Strains are also in the original element coordinate system and depend on the element type:

- The incompressible elements HYPER56, 58, and 74 output true (log) strains and true stress.

- The compressible elements HYPER84 and 86 output Green-Lagrange strains and true stress.

Review the Results

Additionally, HYPER56, 58, and 74 output the pressure DOF value as HPRES. This value may be accessed in the General Postprocessor or The Time History Postprocessor as a "stress".



HYPER84 and 86 output the unit extension in each direction (UNEXTN) and the shear rotation angles (ROTANG).

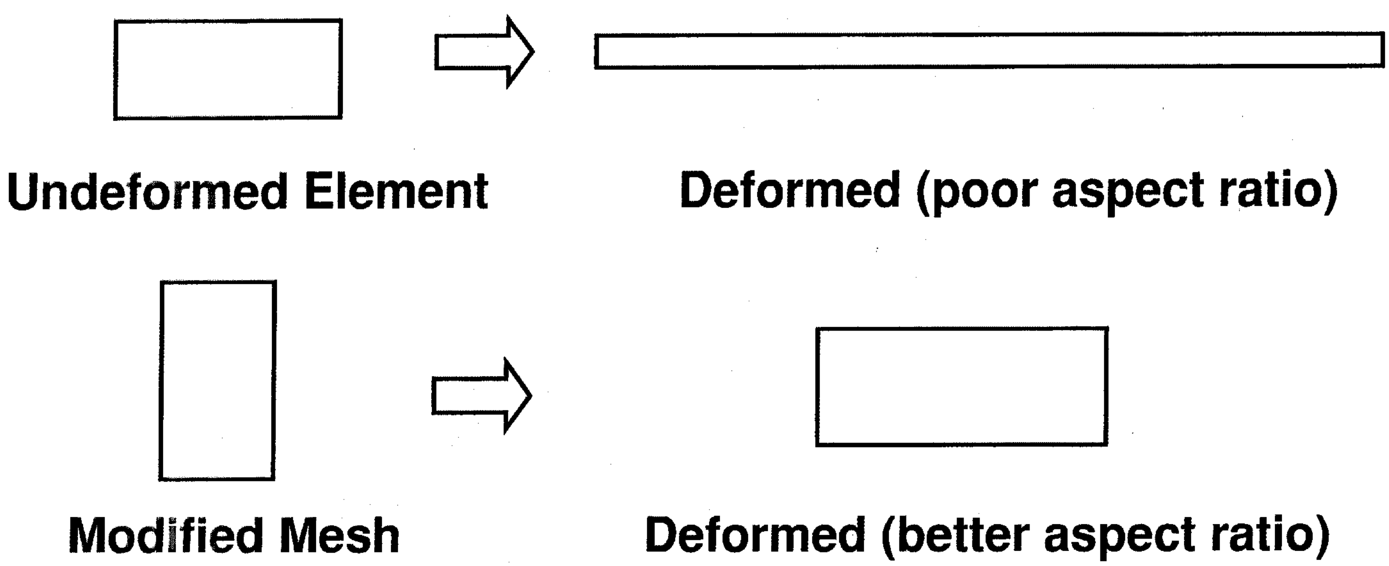
Trouble Shooting

- Some values of Mooney-Rivlin constants result in stable stiffness matrices whereas others do not. Choose constants with caution, and experiment with slightly different values if the analysis is unsuccessful.

- Review the hydrostatic pressure (HPRES) in a PLESOL plot. The pressure distribution should be smooth. "Checkerboarding" (alternating pressure values across elements) indicates that the mesh was too coarse or midside-noded elements are needed, because the incompressibility constraint was not satisfied.

Trouble Shooting

- Check the mesh distortion. Poorly shaped elements at any point in the solution can lead to convergence difficulties. Check for poor aspect ratios and angles between element edges approaching 0 or 180 degrees.



Trouble Shooting

- For highly constrained analyses (i.e., hyperelastic material "trapped" in a metal structure) using the incompressible elements (HYPER56, 58, 74, and 158), the mesh may have to be relatively fine in order to obtain a solution. These elements must have sufficient pressure DOFs to compute the solution.
- As a guideline, you should have at least twice as many unconstrained displacement DOFs as pressure DOFs in 2-D, and three times as many in 3-D. Failure to use a fine-enough mesh can lead to volumetric locking (evidenced by checkerboard HPRES plots).
- If you are experiencing an instability, plot the force deflection curve to determine if the instability is physical or numerical.

ANSYS

Chapter 7

ELEMENT SELECTION
For Nonlinear Analysis

Session Objective

- At the end of this session you will be able to describe and demonstrate the following:

  1. Continuum Element Formulations
  2. Shear Locking, Volumetric Locking, Hourglass Modes
  3. Solid Element Recommendations
  4. Shell Element Recommendations
  5. Beam Element Recommendations

Element Formulations

- Conventional Displacement Method
  - Solid45 KEYOPT(1)=1 rarely used due to shear locking
- Incompatible Modes (Extra Shapes)
  - Solid45 default option, bending deformations
- Selective Reduced Integration (B-Bar)
  - Nearly incompressible materials, bulk deformations
- Uniform Reduced Integration (URI)
  - Nearly incompressible materials, bending deformations
- Mixed U-P Formulation
  - Incompressible materials, hyperelasticity

Element Formulations

Why so many different element formulations?

- General nonlinear solution is expensive. Different element technologies are available to handle various types of nonlinear problems more efficiently.
- Different material behaviors (elastic, plastic, hyperelastic) and different structural behaviors (bulk deformations, bending) lead to different choices in element formulations.
- The ANSYS element library provides a "tool kit" of appropriate element technologies for different types of nonlinear problems.

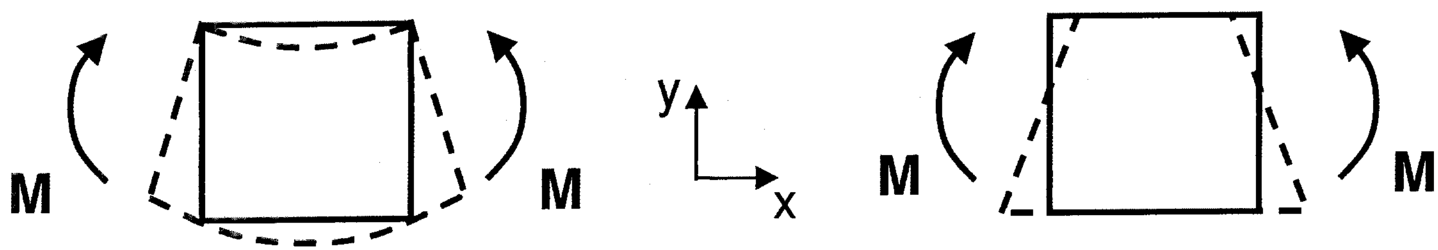
Conventional Displacement Formulation

- Fully integrated lower order elements *without extra shape functions* (Solid45 KEYOPT(1)=1) and fully integrated higher order elements (Plane2) are examples of conventional displacement formulations.
- For the conventional displacement formulation, the numerical integration rule is numerically exact for all components of strain energy. Nodal displacements (UX, UY, UZ) are the only primary variables.

  NOTE: lower order elements with this formulation are rarely used due to shear locking and volumetric locking.

Shear Locking

- Fully integrated lower order elements exhibit "overstiffness" in bending problems. This formulation includes shear strains in bending which do not physically exist, called parasitic shear. (From beam theory in pure bending the shear strain, $\gamma_{xy} = 0$.)



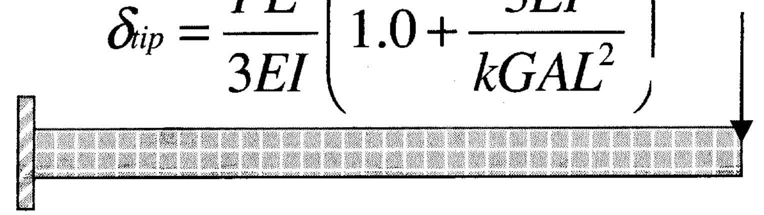
Pure bending deformation for a differential volume, plane sections remain plane, top and bottom edges become arcs, $\gamma_{xy} = 0$.

Fully integrated lower order element deformation, top and bottom edges remain straight, right angles are not preserved, γ_{xy} is non zero.

Shear Locking Example

- As the aspect ratio increases, fully integrated lower order elements will lock in bending.
- Second order elements such as Plane82 do not have this problem with shear locking (quadratic shape function allows the edges to curve).
- Mesh refinement generally does not help shear locking!

$$\delta_{tip} = \frac{PL^3}{3EI}\left(1.0 + \frac{3EI}{kGAL^2}\right)$$



Number of Elements		*Normalized tip deflection (Plane42)*
Depth	**Length**	**Exclude Extra shapes**
1	10	0.02
4	10	0.02
10	10	0.02
10	20	0.07

Element Formulations Not to be Used in Bending

Element Type	*when Keyopt*	*is set to Keyopt Value*
42	2	1
45	1	1
*43/143**	3	1
181	3	1
182	1	0
185	2	0

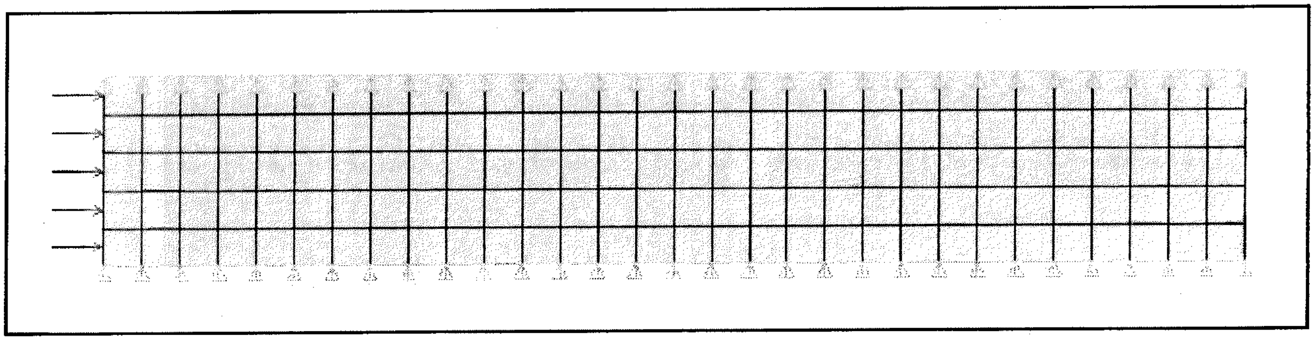
***Does not use extra shapes in geometrically nonlinear analysis.**

Volumetric Locking

- Volumetric locking occurs in fully integrated elements when the material behavior is nearly incompressible (Poisson's ratio approaches 0.5). The incompressibility can occur from a hyperelastic material or plastic flow. Spurious pressure stresses develop in the element, which cause the element to have an "over-stiffness" for deformations that should not cause any volume change.
- Volumetric locking can occur for various stress states, including plane strain, axisymmetric, and 3-D stress. For plane stress volumetric locking does not occur.

Volumetric Locking Example

Radial Displacement of a Thick Axisymmetric Cylinder



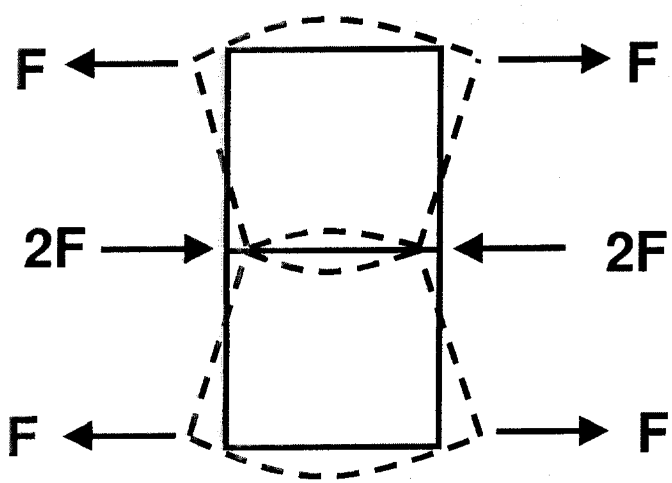
Poisson's ratio	Analytical	Plane42 w/ extra shape	Plane42 w/o extra shape
0	3.75	3.7745	3.7236
0.25	4.4531	4.4837	4.4037
0.3	4.5825	4.6143	4.5227
0.49	5.0399	5.0763	4.1971
0.499	5.0602	5.0968	1.7
0.4999	5.0623	5.0989	0.2441

Volumetric Locking

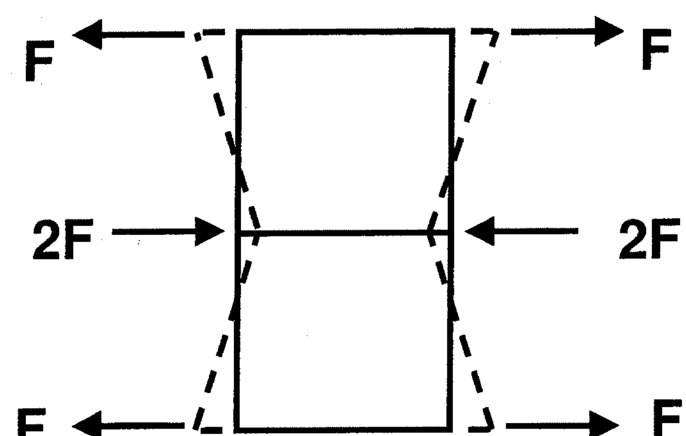
- Volumetric locking can be detected by a "checkerboard" pattern to the pressure stresses (changing significantly from one element to the next). In a hyperelastic model the hydrostatic pressure (HPRES) can be contoured with an element contour plot (PLESOL) to verify this behavior.
- If volumetric locking is suspected try refining the mesh in the areas of high hydrostatic stress or switch the element type.

Incompatible Modes Formulation

- The shape functions of lower order fully integrated elements can be augmented by modes which account for a state of constant curvature. These additional modes are added as internal degrees of freedom. They are called incompatible modes because they lead to gaps and overlaps in the mesh.



Incompatible Modes



No Incompatible Modes

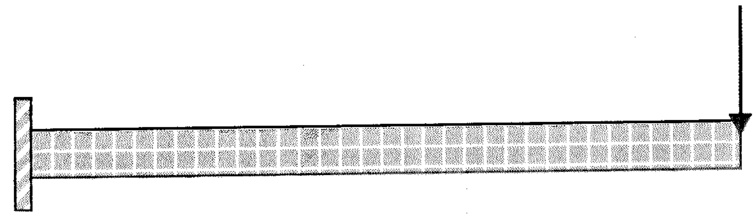
Incompatible Modes Formulation

- Incompatible modes generally provide accurate results. Interelement gaps and overlaps tend to soften the structure and prevent the "overstiffness" associated with a conventional displacement formulation.
- Note, that elements with incompatible modes lose their extra shapes when created in a degenerate form. For example Plane42 shaped as a quadrilateral has incompatible modes, but when condensed to a three node triangle Plane42 becomes a constant strain triangle.

```
*** WARNING ***
The mesh of area 2 contains PLANE42 triangles, which are much too stiff
in bending.  Use quadratic (6 or 8-noded) elements if possible.
```

Incompatible Modes Formulation

- Incompatible modes increase accuracy with an additional formulation cost:
  - 4 DOF/Element in 2D
  - 9 DOF/Element in 3D
  - Increased .esav File Size
- Element formulations based on incompatible modes: 5, 41, 42, 45, 43, 63, 143, 181.
- Designed for bending!



Deflection of a Cantilever Beam (depth/length = 1/100)

Number of Elements		*Normalized tip deflection (Plane42)*	
Depth	Length	Include Extra shapes	Exclude Extra shapes
1	10	**0.9972**	0.02
4	10	**0.9973**	0.02
10	10	**0.9974**	0.02
10	20	**0.9993**	0.07

Selective Reduced Integration (B-Bar)

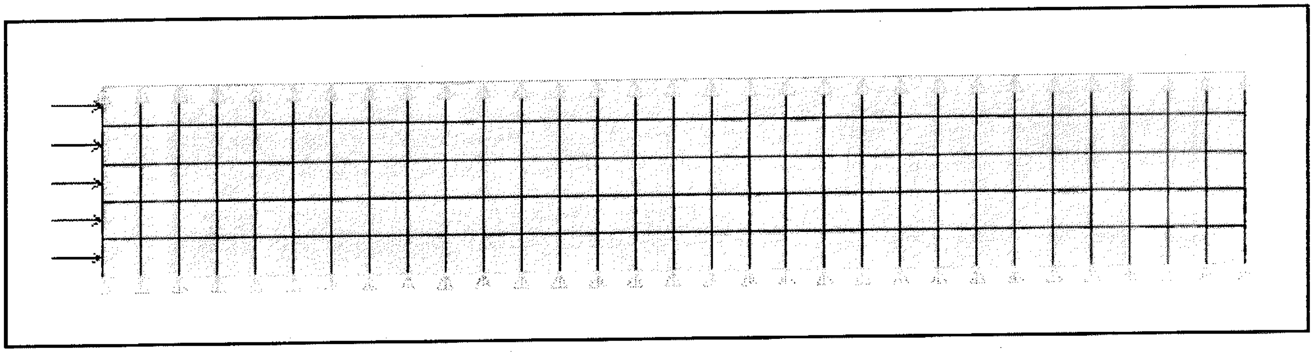
- Selective reduced integration (B-Bar) eliminates volumetric locking in nearly incompressible materials by using an average volumetric strain for the element instead of the integration point strain. B-Bar splits the B matrix into two parts, the volumetric (hydrostatic) and deviatoric strains.

$$\varepsilon = \overline{B}u$$

- The advantage of B-Bar is that no additional degrees of freedom are added to the solution.
- The B-Bar formulation is recommended for bulk deformation plasticity applications, as shear locking (bending) is still a problem for B-Bar.

B-Bar Example

Radial Displacement of a Thick Axisymmetric Cylinder



Poisson's ratio	Analytical	Plane182	Plane42 w/ extra shape	Plane42 w/o extra shape
0	3.75	3.75	3.7745	3.7236
0.25	4.4531	4.4531	4.4837	4.4037
0.3	4.5825	4.5825	4.6143	4.5227
0.49	5.0399	5.0399	5.0763	4.1971
0.499	5.0602	5.0602	5.0968	1.7
0.4999	5.0623	5.0623	5.0989	0.2441

Uniform Reduced Integration

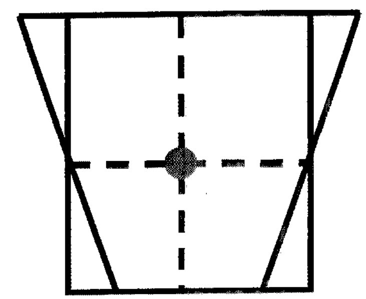
- Uniform reduced integration (URI) uses a lower order integration rule than is required for numerically exact integration. This leads to a more flexible element which helps to eliminate shear and volumetric locking.

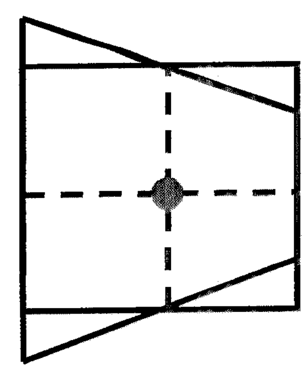
- No additional degrees of freedom are required with URI. File sizes are reduced, and less CPU time is required for element calculations (especially with material nonlinearities). However, URI leads to modes of deformation which have zero strain energy, called zero energy or *hourglass modes*.

Hourglass Modes

- Hourglass modes are modes of deformation which result in zero strain energy due to the deformation.

- In the lower order element with one integration point shown to the right, two modes of deformation are illustrated where the single integration point does not capture any strain energy in the element. This can lead to physically unrealistic behavior.





Lower Order Elements & URI

- URI results in hourglass modes. If left unchecked hourglass modes can lead to an uncontrollable distortion of the mesh. To control hourglass modes ANSYS uses a small hourglass stiffness to control the zero energy modes of deformation.
- Point loads and point constraints should be avoided when using URI and lower order elements. A refined mesh is also required for an accurate stress prediction.
- Although hourglass behavior presents another item to check when verifying a solution, these elements are very computationally efficient in a nonlinear analysis.

Hourglass Control

- ANSYS lower order elements which include a URI formulation are; Plane182, Solid185, Solid45, and Shell181. These elements with a URI formulation are compatible with the elements of ANSYS/LS-Dyna.
- ANSYS provides default values for the hourglass stiffness. In most cases the default values should be adequate, but they can be overridden by a real constant scale factor.
- In all cases the "artificial energy" created by the hourglass modes should be monitored. You can use the element table item AENE to store the "artificial energy".

Hourglass Control

- If hourglass modes occur in your model, the recommended course of action, in order of preference is to:
  - Remove any point loads or constraints
  - Refine the mesh
  - Use an alternate element type
  - Increase hourglass stiffness scaling factor
- In all cases monitor the "artificial energy". A good rule of thumb is that the ratio of "artificial energy" to total energy (AENE/SENE) should be less than 5%.

Second Order Elements & URI

- *Second order* elements using uniform reduced integration do not have the same difficulties with hourglass modes as *lower order* elements.

- *Plane82* uses a 2 x 2 Gauss Integration rule. There is only one zero energy mode that does not propagate as long as there is more than one element in model!

- *Solid95* uses a 2 x 2 x 2 Gauss Integration rule (KEYOPT(11) = 1). Zero energy modes do not propagate as long as the model contains more than one element in each direction!

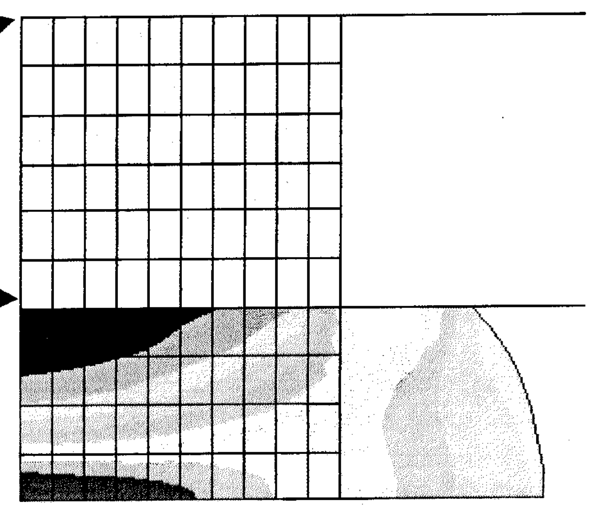
- ***These elements are recommended for most applications as there are generally no difficulties with hourglass modes.***

URI Example

Upset Forging Example

Initial Geometry

Deformed Geometry



Target
Reaction = 800

Note, the increase in solution speed using B-Bar or URI versus the incompatible modes formulation.

	Fully Integrated with Incompatible Modes	B-Bar	URI
CPU Time	433	101	67
Reaction	812	809	808
ESAV Size	311	147	82

U-P Mixed Formulation

- A U-P mixed formulation includes hydrostatic pressure as an independent degree of freedom in addition to displacements. Separate interpolation functions are used for the displacement and pressure degrees of freedom.

- A U-P mixed formulation is ideally suited for incompressible material models, such as rubber and rubber-like materials.

- The ANSYS elements that use this formulation include the hyperelastic elements Hyper56, Hyper58, Hyper74, and Hyper158; as well as the viscoplastic solids Visco106, Visco107, and Visco108.

Solid Element Recommendations

Linear Analysis

- For linear analysis use any first order element that has incompatible modes (*Plane42, Solid45* in non degenerate form.) Second order elements are preferred for high stress gradients, and stress concentration studies.
- Use *Solid92* for high performance solutions.
- *Solid95* with reduced integration is adequate in most problems, required for Poisson's ratio approaching 0.5.

Solid Element Recommendations

Nearly Incompressible Materials (Plasticity)

- Bulk structural deformations with negligible bending use *Plane182, Solid185* Selective Reduced Integration (B-Bar).
- For small strain applications use incompatible mode elements *Plane42, Solid45.*
- For large strain applications use *Plane182* and *Solid185* with URI (preferred for large models) or *Solid95* with URI. Visco106, Visco107, and Visco108 can also be used (even for rate independent plasticity).

Solid Element Recommendations

Incompressible Material (Hyperelasticity)

- The incompressibility constraint of rubber-like materials require the U-P mixed formulation, use *Hyper56, Hyper58, Hyper158, or Hyper74.*

Shell Elements - Overview

- Shell elements can be used when the overall thickness of a structure is small relative to the characteristic length. A ratio greater than ten of thickness to length is used to determine the applicability of shell elements.

- Various shell theories exist in the open literature, this arises from the approximations used to characterize the displacements of a shell.

- The shell elements in ANSYS use different formulations depending on the type of problem to be solved. The three basic shell formulations include; membrane theory, "thin" shell theory, and "thick" shell theory.

Shell Elements - Overview

Membrane Theory

- *Shell41* uses membrane theory. Shell41 neglects bending and transverse shear, only membrane effects are included.

Classical Love-Kirchhoff Theory

- *Shell63* is a "thin" shell. Shell63 includes bending and membrane effects but neglects transverse shear deformation.

Reissner/Mindlin Theory

- *Shell43, 143, 181, 91, 93 and 99* are "thick" shells. They include bending, membrane, and transverse shear effects. Transverse shear is accounted for as a constant shear strain through the thickness. This first order approximation is applicable only to "moderately thick" shells.

Shell Elements In Plane Behavior

- The in plane response of shells is assumed to be in the state of plane stress. Volumetric locking for shell elements is therefore not an issue. (Shell181 supports hyperelasticity as perfectly incompressible, Poisson's ratio = 0.5.)
- The in plane formulation of shell elements is similar to the formulation of planar solid elements (incompatible modes) for the membrane behavior.
- *Shell41, 43, 63, and 181* support incompatible modes for in plane behavior.
- *Shell181* also supports uniform reduced integration with hourglass control (default option).

Shell Element Recommendations

Linear Analysis

- If shell thickness is very small use *Shell63.* No transverse shear effects are included with Shell63.
- If transverse shear deformation is important use *Shell43, Shell93, or Shell143* for homogeneous materials, use *Shell91 or Shell99* for composites.
- Note that *Shell181* with uniform reduced integration (default) may be faster for large models, but will require a more refined mesh.

Shell Element Recommendations

Nonlinear Analysis

- ***Isotropic Hardening Plasticity and Hyperelasticity***
  - Use *Shell181.* The benefits include; a smaller .esav file size, less CPU time, pressure load stiffness effects, ability to import initial stresses, and thickness changes in a contact analysis.
- ***Kinematic Hardening Plasticity, Creep***
  - Use *Shell143, Shell43, or Shell93.* Shell43 and Shell143 are applicable for small strain plasticity. Shell93 is a curved (higher order) shell.

Beam Elements - Overview

- Beams can be used to analyze structures subjected to lateral or transverse loads. Typical beam applications include; machine shafts, building frames, bridges, etc. The two beam element formulations available in ANSYS are:

<u>Euler/Bernoulli Beams</u>

- *Beam3 and Beam4* include bending, axial, and torsional deformations. Transverse shear deformation is not included in the element formulation (but can be applied as a flexibility factor).

<u>Timoshenko Beams</u>

  - *Beam188 and Beam189* include bending, axial, torsional, and transverse shear deformations in the element formulation.

Beam Element Recommendations

Linear Analysis

- For linear models use *Beam3, Beam4, Beam188 or Beam189.* Beam3 and Beam4 use Hermitian polynomials for shape functions and have a cubic response in bending. Beam188 uses a linear polynomial for a shape function, and Beam189 uses a quadratic polynomial (can act as a curved beam).

- Note that a more refined mesh is required with the finite strain beams. However, *Beam188 and Beam189* have many advanced pre and post processing features which make their use attractive for linear models.

Beam Element Recommendations

Nonlinear Analysis

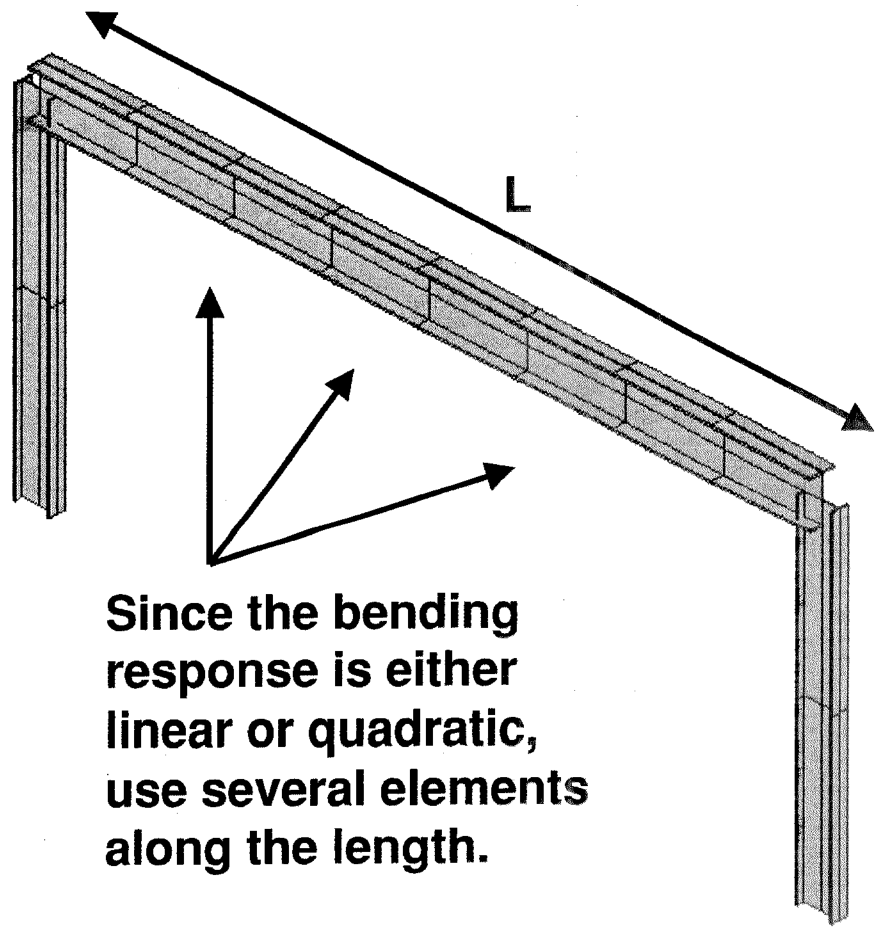
- Use *Beam188 or Beam189* to model isotropic hardening plasticity, large strain, buckling (both eigenvalue and nonlinear collapse), and/or large rotation problems.

- For use as shell stiffener elements, Beam188 is fully compatible with Shell181, and Beam189 is fully compatible with Shell93.

- *Note that all beam elements assume unrestrained section warping. Use shell elements to model beam structures when torsional warping restraint is important.*

Beam188 and Beam189

Since Beam188 and Beam189 use a first order approximation for shear deformation (constant shear stress through the depth), they should only be used for reasonably slender beams. As an approximate guideline, $(GAL^2)/(EI) > 30$, where L is overall length of the member (not the length of individual elements).



Since the bending response is either linear or quadratic, use several elements along the length.

Chapter 8

CONTACT NONLINEARITIES

Session Objective

- At the completion of this session you will be able to describe and demonstrate the following:

  1. Enforcing Contact Compatibility
  2. Rigid-to-Flexible and Flexible-to-Flexible Contact
  3. Node-to-Node, Node-to-Surface, and Surface-to-Surface Elements
  4. Contact Stiffness
  5. Contact Element Procedures
  6. Contact Wizard

Contact Problems

- Contact problems are highly nonlinear and are typically the most challenging class of nonlinear problems to solve.
- Contact problems present two significant challenges:
  - In most contact problems the region of contact is an unknown. Surfaces can come into and go out of contact in an abrupt manner, which results in an abrupt change of the stiffness of the system.
  - Most contact problems include friction. Friction is a path dependent phenomenon which requires an accurate load history. Frictional response also can be chaotic, making solution converge difficult .

Note on Couples and Constraint Equations

- If you have frictionless contact, and your contact region is always bonded, and the analysis is a small deflection, small rotation problem you may be able to use couples or constraint equations instead of contact. For more information refer to the ANSYS Modeling and Meshing Guide.
- The advantage of using couples or constraint equations is that the analysis remains linear.

Contact Classification

Contact problems fall into two general classes of problems: rigid-to-flexible and flexible-to-flexible.

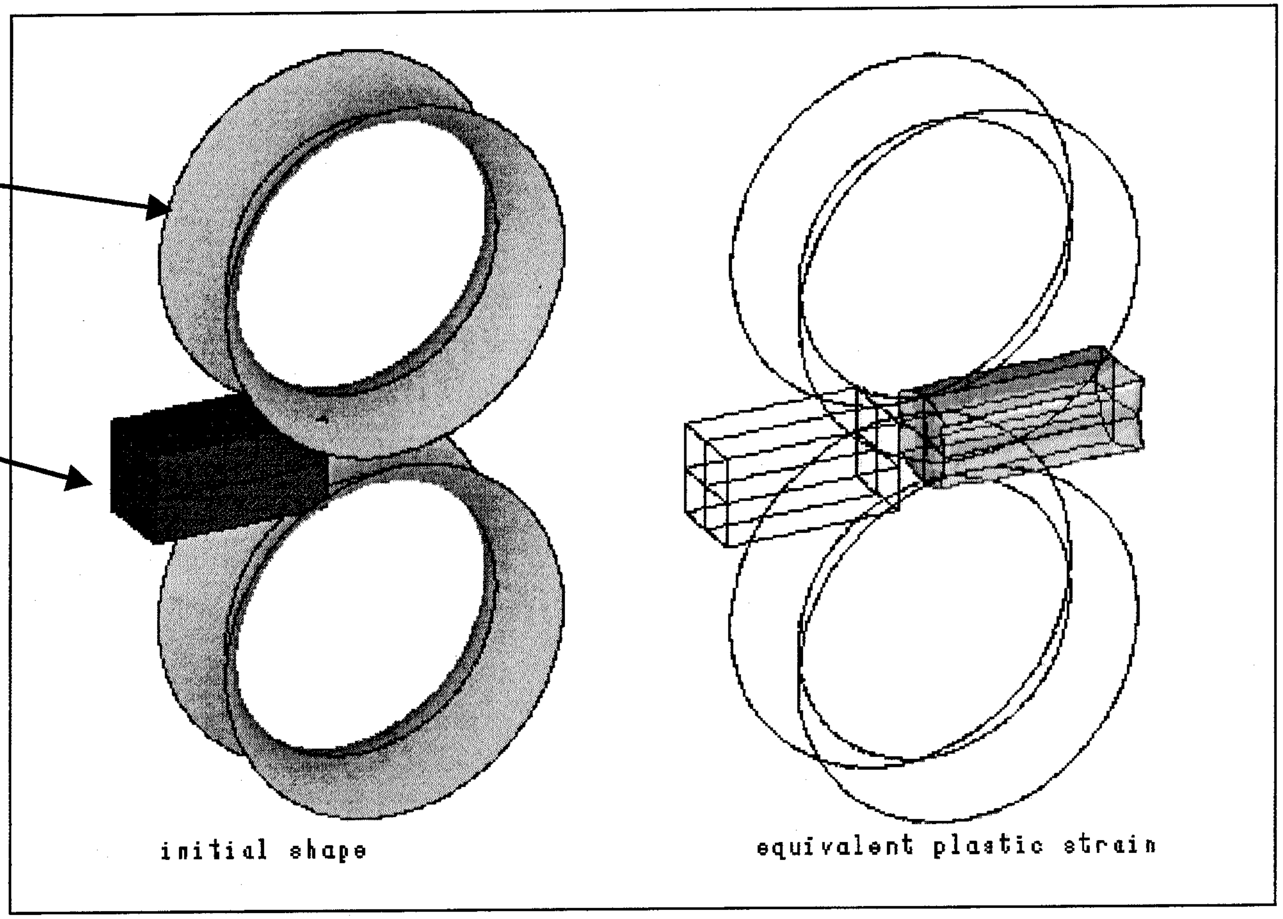
Rigid-to-Flexible: One or more contacting surfaces are treated as rigid. (One surface has a significantly higher stiffness than the other.) Many metal forming problems fall into this category.

Flexible-to-Flexible: Both or all contacting bodies are deformable. (All surfaces have similar stiffnesses.) A bolted flange connection would be an example of flexible-to-flexible contact.

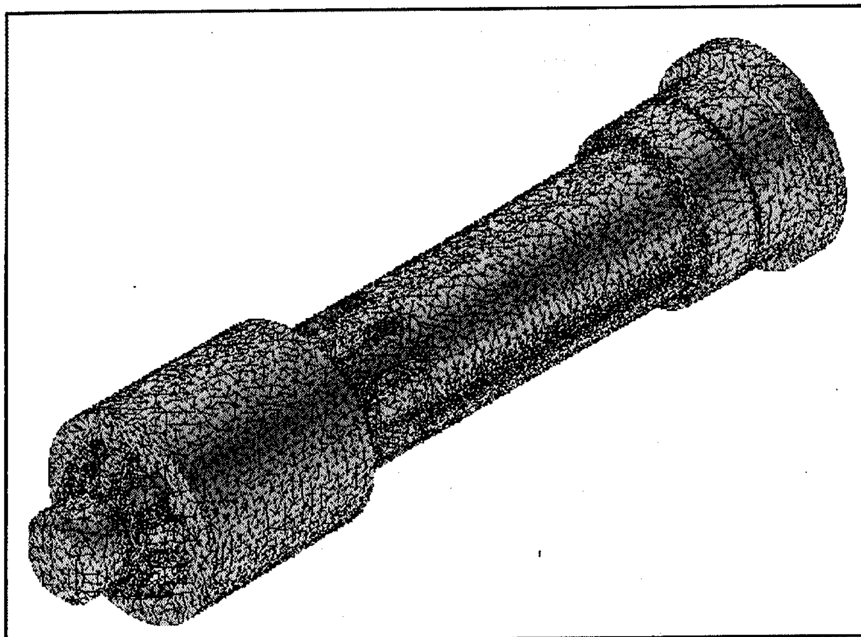
Rigid-to-Flexible Contact

Rigid Surface

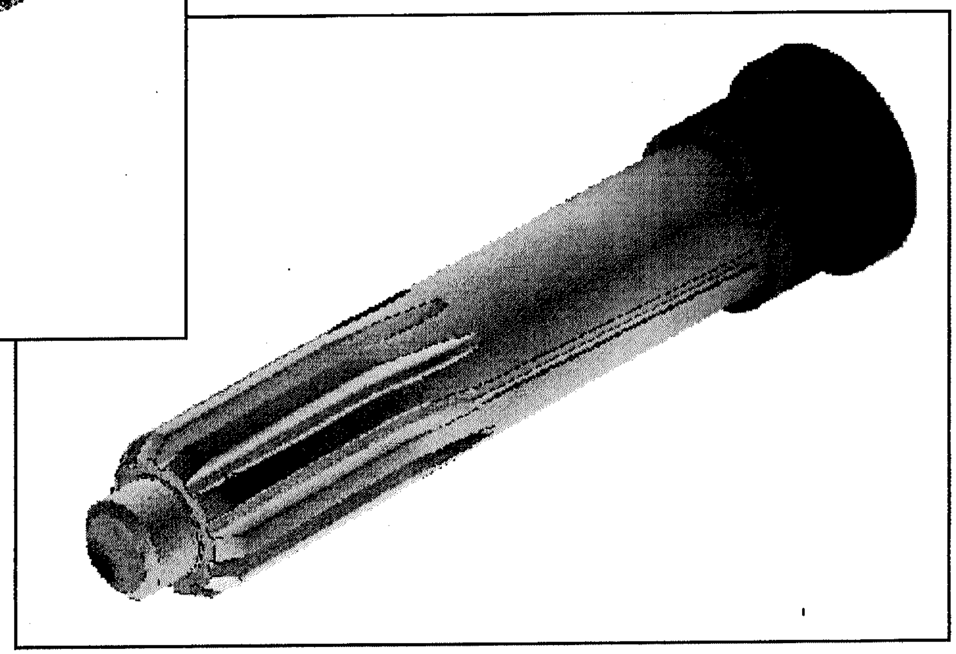
Deformable Body



Flexible-to-Flexible Contact

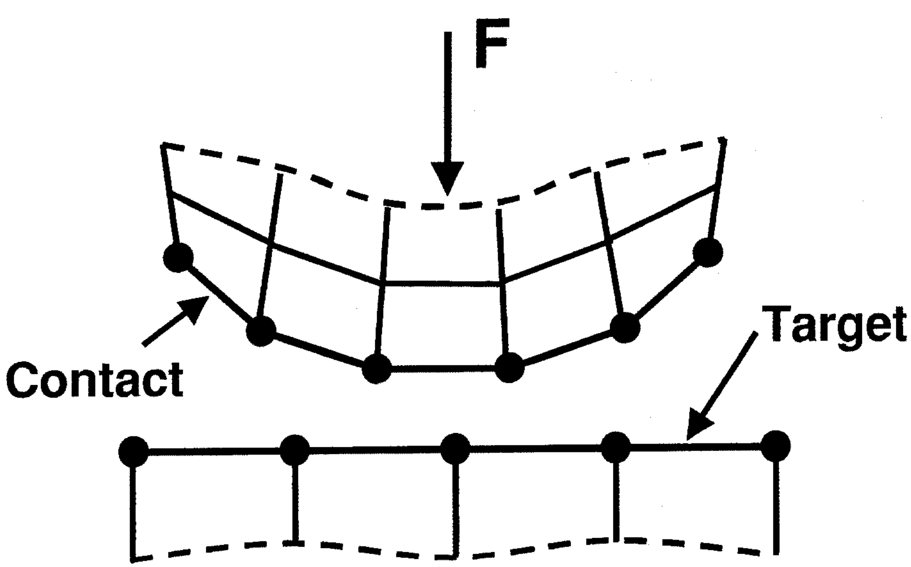


Splined shaft interference fit, both parts are flexible.

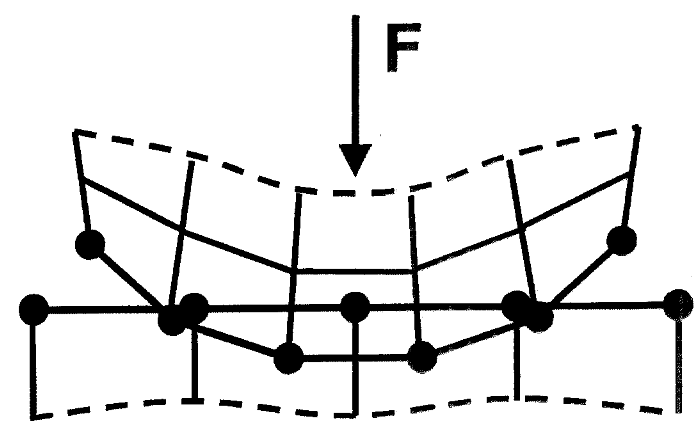


Contact Compatibility

- In order to prevent the contacting surfaces from passing through each other, a relationship must be established between the two surfaces. Otherwise the two surfaces will pass through one another.

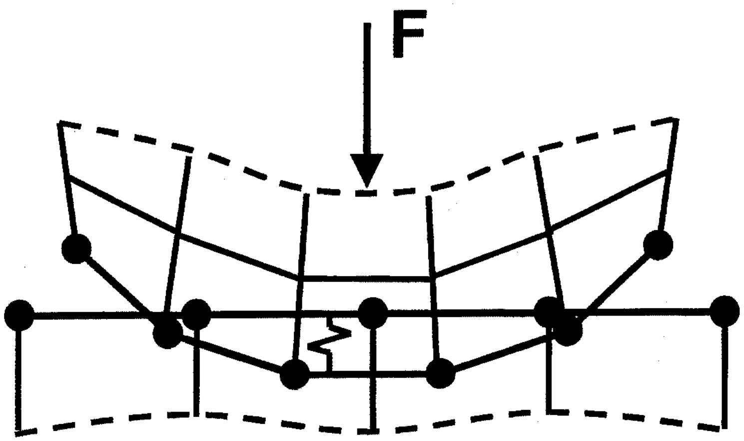


Penetration occurs without enforcing contact compatibility.



Contact Compatibility

- Using a spring to enforce contact compatibility is called the penalty method. The spring stiffness or the contact stiffness is called the penalty parameter.



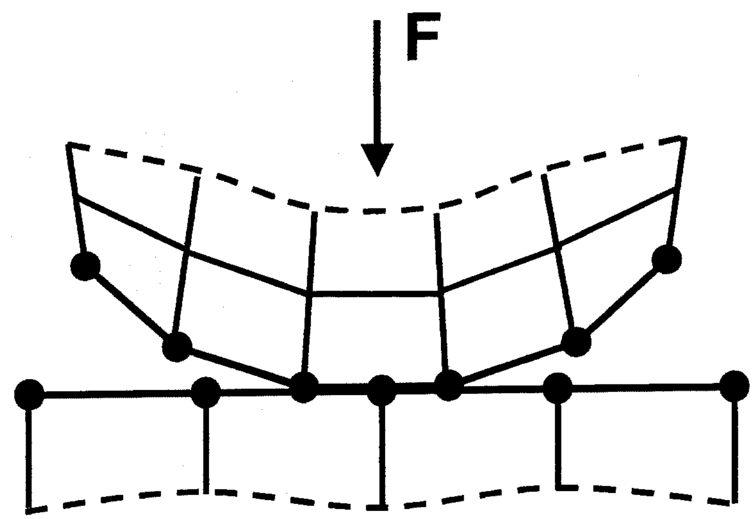
The spring will deflect an amount Δ such that equilibrium is satisfied,

$$F = k\,\Delta$$

The higher the contact stiffness (k) the less penetration occurs at the contact surface. However, too high of a value can lead to convergence difficulties.

Contact Compatibility

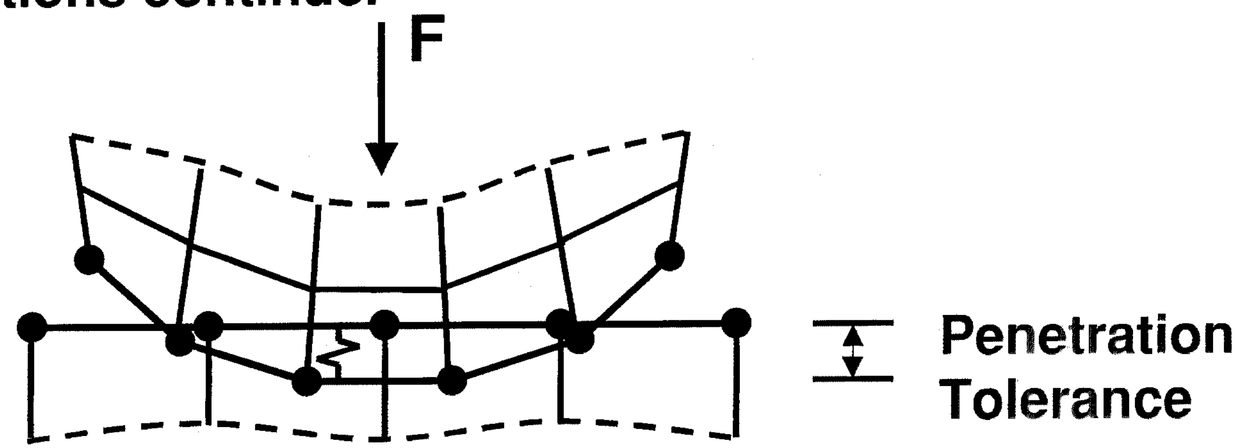
- An alternative method, the *Lagrange Multiplier* method, adds an extra degree of freedom (contact pressure), to satisfy the impenetrability condition.



Contact Compatibility

- Combining both the penalty method and the Lagrange multiplier to enforce contact compatibility is called the *augmented Lagrangian.*

- In the first series of iterations, contact compatibility is determined based on the penalty stiffness. Once equilibrium is achieved, the penetration tolerance is checked. At this point, if necessary, the contact pressure is augmented and the iterations continue.



Augmented Lagrangian

```
CONTAC49/173/174:    44 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  = 0.3994E+05  CRITERION=  647.1
EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.1434
LINE SEARCH PARAMETER =   1.000      SCALED MAX DOF INC = -0.1434
CONTAC49/173/174:    18 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  =  3972.      CRITERION=  197.2
EQUIL ITER   2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.1749
LINE SEARCH PARAMETER =  0.6128      SCALED MAX DOF INC = -0.1072
CONTAC49/173/174:    12 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  =  2372.      CRITERION=  160.4
EQUIL ITER   3 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.1974E-01
LINE SEARCH PARAMETER =   1.000      SCALED MAX DOF INC =  0.1974E-01
CONTAC49/173/174:    12 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  =  215.9      CRITERION=  128.8
EQUIL ITER   4 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.1392E-01
LINE SEARCH PARAMETER =  0.6505      SCALED MAX DOF INC =  0.9053E-02
CONTAC49/173/174:    12 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  =  95.60      CRITERION=  128.0     <<< CONVERGED
EQUIL ITER   5 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.4200E-02
LINE SEARCH PARAMETER =   1.000      SCALED MAX DOF INC = -0.4200E-02
CONTAC49/173/174:    12 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  =  2226.      CRITERION=  128.2
EQUIL ITER   6 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.3365E-01
LINE SEARCH PARAMETER =  0.9708      SCALED MAX DOF INC =  0.3267E-01
CONTAC49/173/174:     4 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  =  84.35      CRITERION=  128.5     <<< CONVERGED
EQUIL ITER   7 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.8724E-02
LINE SEARCH PARAMETER =  0.9712      SCALED MAX DOF INC =  0.8473E-02
CONTAC49/173/174:     4 CONTACT POINTS HAVE TOO MUCH PENETRATION
FORCE CONVERGENCE VALUE  =  447.1      CRITERION=  128.6
EQUIL ITER   8 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.1173E-01
LINE SEARCH PARAMETER =  0.8080      SCALED MAX DOF INC =  0.9478E-02
FORCE CONVERGENCE VALUE  =  94.89      CRITERION=  128.6     <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION   8
*** LOAD STEP    1   SUBSTEP    5  COMPLETED.    CUM ITER =     30
*** TIME =   1.00000        TIME INC =  0.200000
```

Penetration correction due to equilibrium.

Augmenting contact stresses to reduce penetration.

Oscillation occurs during correction stage.

Contact Elements

There are three types of contact elements available in ANSYS:

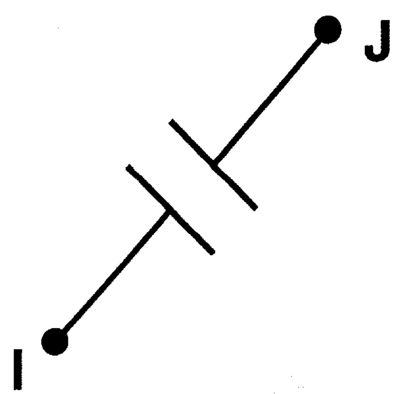
Node-to-Node - This implies that the final location of contact is *known* beforehand.

Node-to-Surface - Contact areas are unknown and large amounts of sliding are permitted.

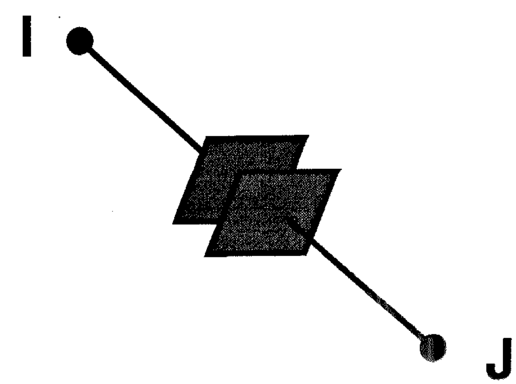
Surface-to-Surface - Contact areas are unknown and large amounts of sliding are permitted. (Several advantages over node-to-surface formulation.)

Node-to-Node Elements

The two most commonly used node-to-node elements:



Contact12 - 2D Gap



Contact52 - 3D Gap

Both Contact12 and Contact52 use the penalty method to enforce contact compatibility. This requires the input of a penalty stiffness. (We will discuss the calculation of the penalty stiffness in more detail in a later section.)

Node-to-Node Elements

- Node-to-node contact elements can be used to model point-to-point contact. A pipe whip model where the contact point is always known is an example of point-to-point contact.

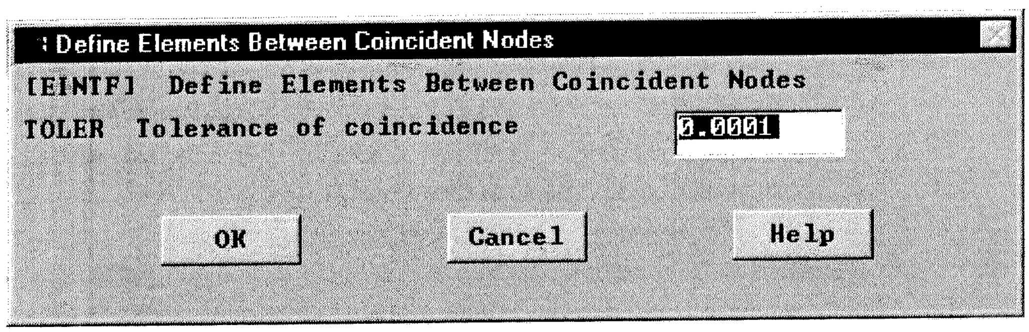
- These elements can also be used to model surface-to-surface contact problems *if the nodes on the two surfaces line up, the amount of relative sliding deformation is negligible, and the deflections (rotations) of the two surfaces remain small.* An interference fit problem is an example of a surface-to-surface contact problem where the use of gap elements may be sufficient, if the above conditions are met.

- *Note that node-to-node contact may be used only with lower order elements.*

Node-to-Node Procedure

Contact12 and Contact52 can be created by either using direct generation or by creating elements at coincident nodes.

Preprocessor -> Create -> Elements -> At Coincid Nd



Contact12 should be created between coincident nodes. However, Contact52 requires a separation of 1E-6 to orient the element.

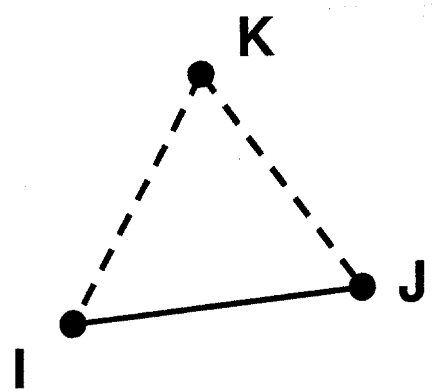
Node-to-Node Procedure

Additional Information:

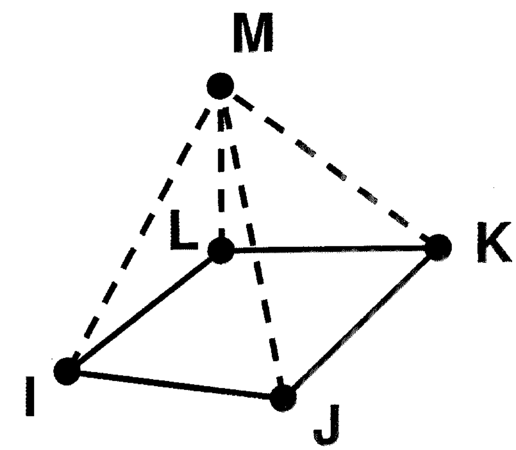
- Contact12 is oriented by the real constant THETA. Contact12 can also act as a hook or a gap, whereas Contact52 will only act as a gap. (Note that Contact52 can be used in a 2-D problem by constraining the UZ degrees of freedom.)
- Both Contact12 and Contact52 allow the specification of an initial interference or a gap by setting KEYOPT(4) and the real constants INTF and GAP.
- Refer to Chapter 4 of the *ANSYS Elements Reference* for a detailed description of Contact12 and Contact52 element options and real constants.

Node-to-Surface Elements

The node-to-surface contact elements:



Contact48 - 2D Node to Surface



Contact49 - 3D Node to Surface

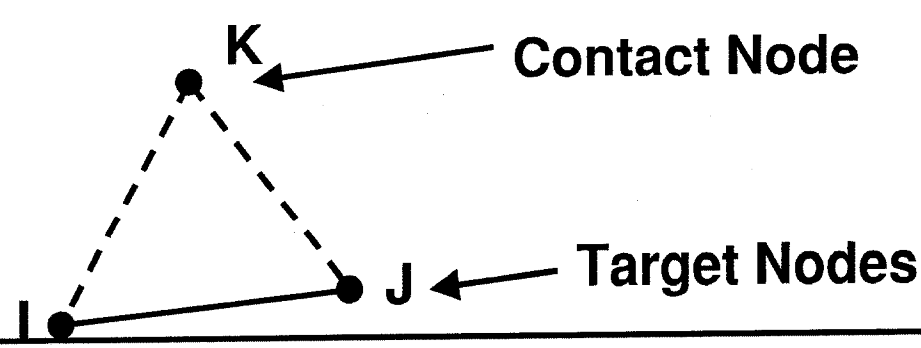
These elements also use the penalty method for contact compatibility as the default option. As an option, a combined penalty and Lagrange multiplier can be used.

Node-to-Surface Elements

- Node-to-surface contact elements are typically used to model point-to-surface contact applications such as two beams contacting (at beam tip) and the corners of snap fit parts.
- These elements can also be used to model surface-to-surface contact by defining multiple elements. These surfaces can be rigid or deformable.
- The location of the contacting area does not have to be known. Large deformation, large relative sliding, and different meshes between contacting surfaces are allowed.
- *Note that node-to-surface contact may be used only with lower order elements.*

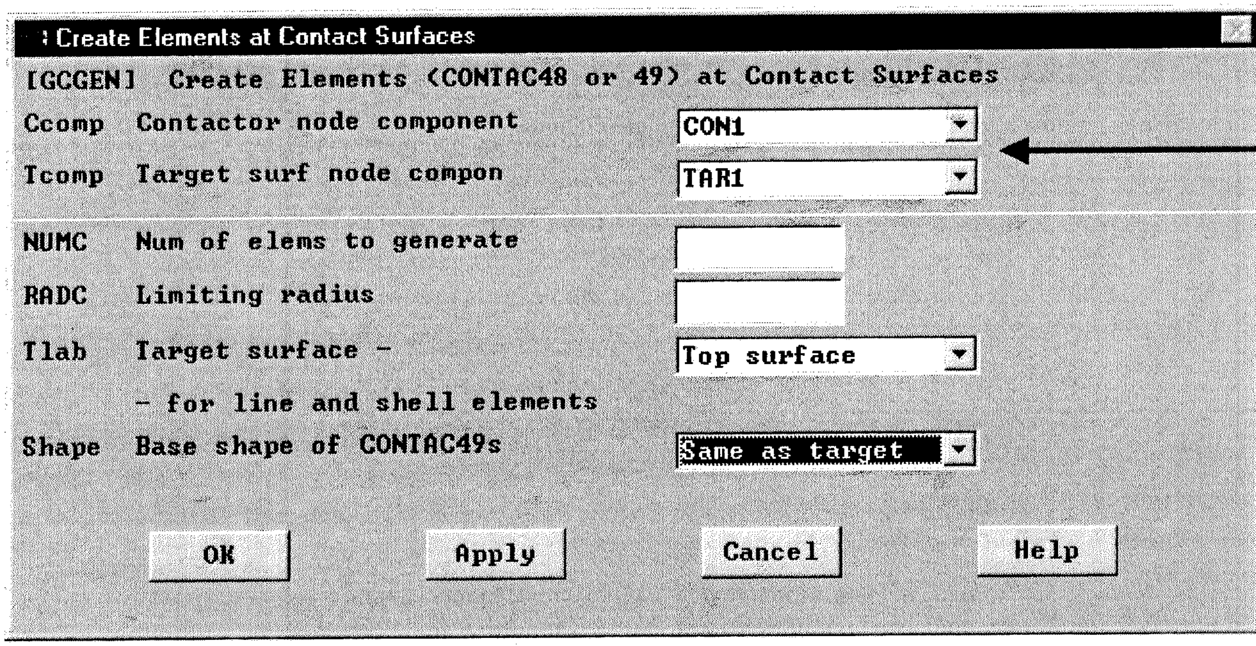
Node-to-Surface Procedure

- Contact48 and Contact49 are generated by identifying nodal components which define the target and contact surfaces.
- Select the desired nodes for the contact and target surfaces and create nodal components.
- Specify the element type and real constants and generate the contact elements. Each separate contact area should reference a different real constant set.



Node-to-Surface Procedure

Main Menu > Preprocessor > Create > Elements > Node to Surf



Nodal Component Names

Refer to the *ANSYS Structural Analysis Guide* for more detail on using the node-to-surface contact elements.

Contact Stiffness

- Both the node-to-node (Contact12 and Contact52) and node-to-surface (Contact48 and Contact49) contact elements require the specification of a penalty stiffness.

- The higher the penalty stiffness, the less penetration occurs at the contact surface. However, too high of a value can cause convergence difficulties due to ill-conditioning.

- Some experimentation may be required to determine an appropriate value for the contact stiffness which generates a converged solution with an acceptable amount of penetration.

Choosing a Contact Stiffness

- The contact stiffness is a function of the relative stiffness of the areas in contact.

- For bulky solids the *Hertz contact stiffness* is usually an appropriate value for the penalty stiffness, and can be estimated from:

$$k = fE$$

- where f is a factor between 0.1 and 10, and E is the elastic modulus of the softer contacting material. Setting f=1 is usually a good starting value.

Choosing a Contact Stiffness

- For flexible components (beam and shell models) the stiffness of the system may be much lower than the Hertz contact stiffness.

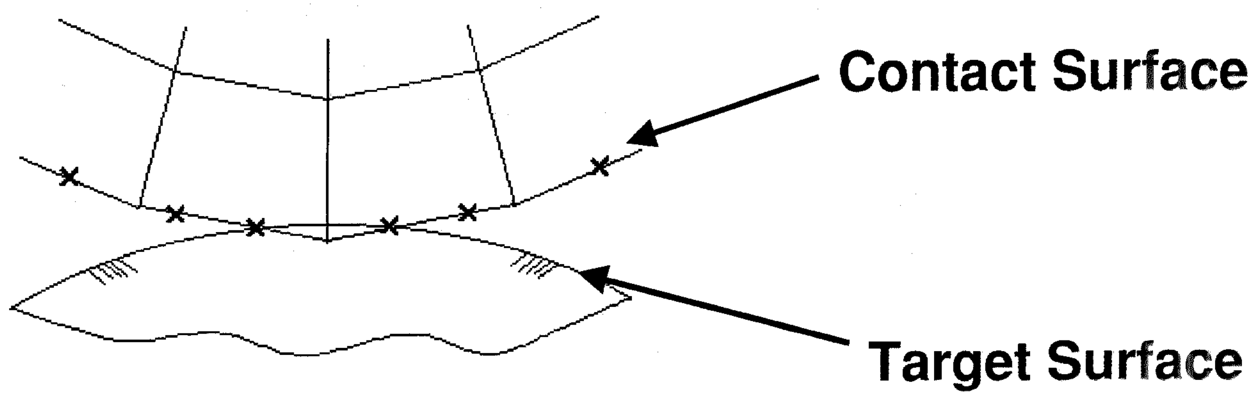
- In this situation you might run a static analysis with a unit load applied to the expected area of contact to determine the local stiffness of the model. The contact stiffness can then be estimated from:

$$k = f(P/\Delta)$$

- where for flexible body contact, f is a factor between 1 and 100. Again, setting f=1 is usually a good starting value.

Surface-to-Surface Elements

ANSYS supports both *rigid-to-flexible* and *flexible-to-flexible* surface-to-surface contact elements. These contact elements use the concept of a "target surface" and a "contact surface" to form a contact pair.



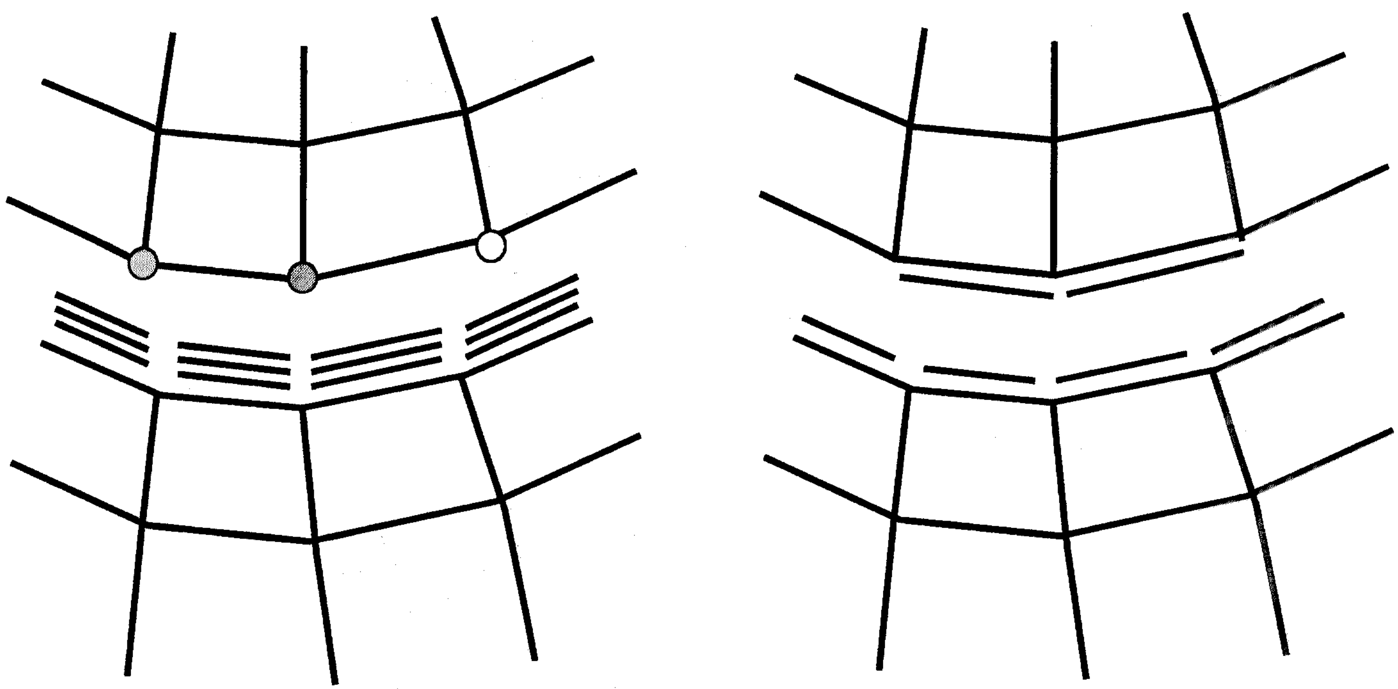
The surface-to-surface contact elements use the *augmented Lagrangian* to enforce contact compatibility (default). As an option they also may be used with the penalty method.

Surface-to-Surface Advantages

- Compatible with both lower order and higher order elements.
- Supports large deformations with significant amounts of sliding and friction efficiently.
- Provides better contact results (easier to postprocess contact pressure and frictional stresses).
- Can account for shell and beam thickness, as well as shell thickness changes.
- Semi-automatic contact stiffness calculation.
- "Pilot node" control of rigid surface (more description on the pilot node later).
- Intelligent default settings, Contact Wizard (easy to use).

Surface-to-Surface Advantages

Far fewer elements are required than with the node-to-surface contact elements (CONTAC48 and CONTAC49).



12 CONTAC48s

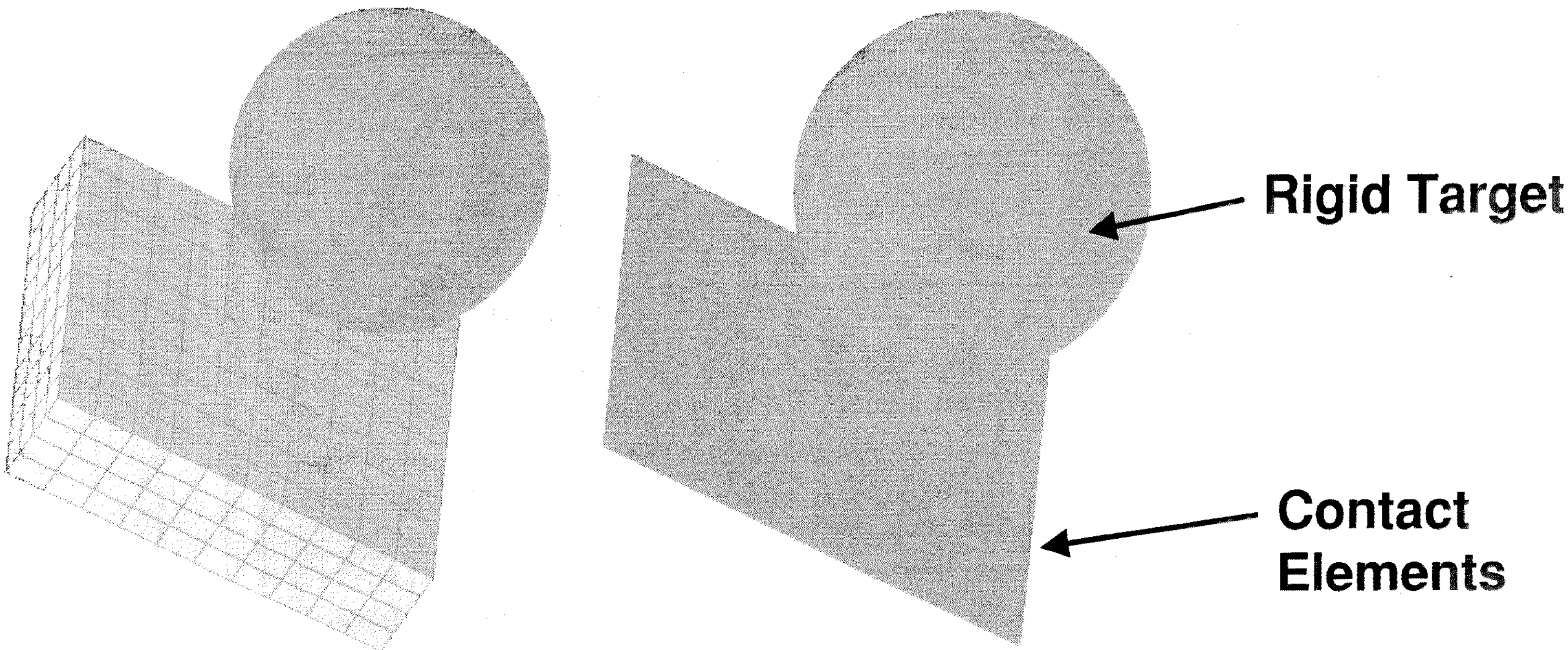
2 CONTAC171s & 4 TARGE169

Surface-to-Surface Elements

- The surface-to-surface contact elements consist of separate element types to define the target and contact surfaces. The contact pair is identified through a shared real constant set.
  - Target169 and 170 - Rigid or Deformable Target Surfaces
  - Contact171 to 174 - 2-D and 3-D Contact Elements
- The surface-to-surface contact elements overlay the underlying finite element model like a skin. These elements are well suited for surface-to-surface contact applications such as interference fit assembly contact, entry contact, forging, and deep drawing.

Surface-to-Surface Elements

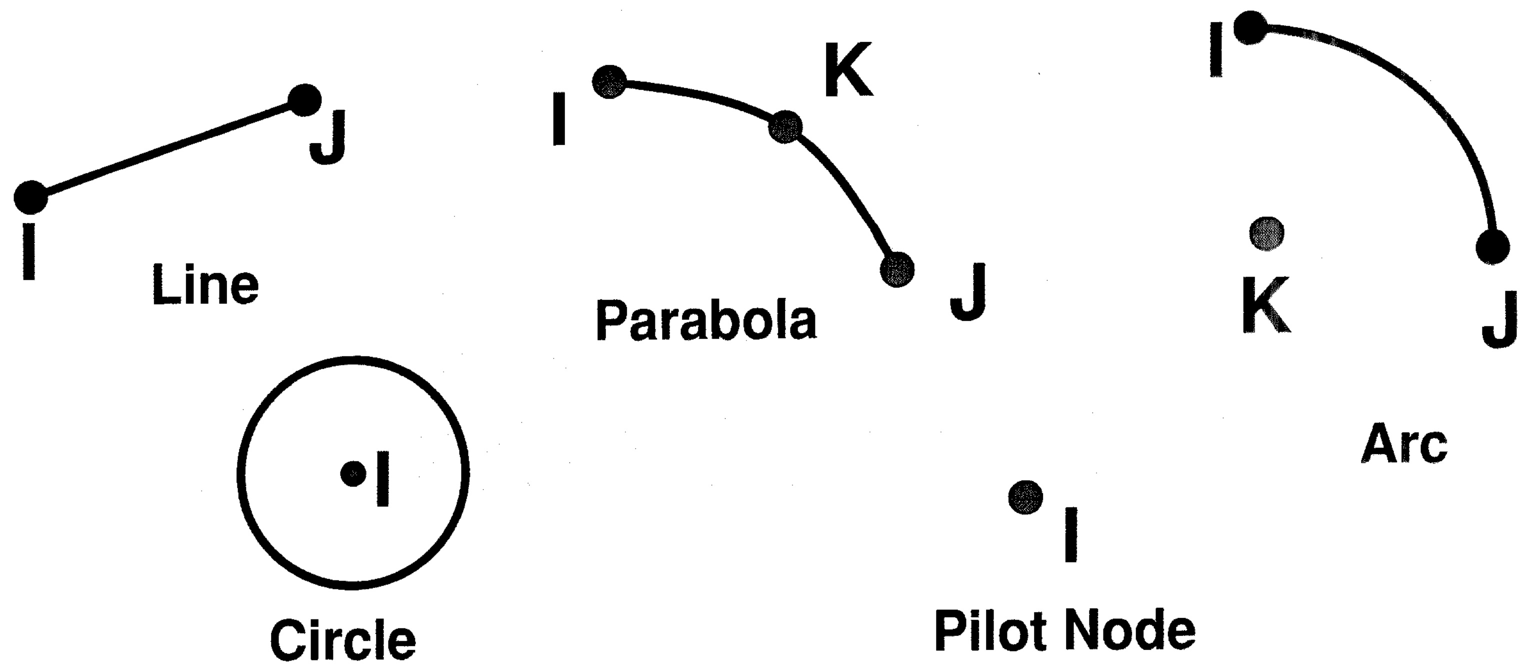
Contact Pair



The contact pair is identified by real constant set 1.

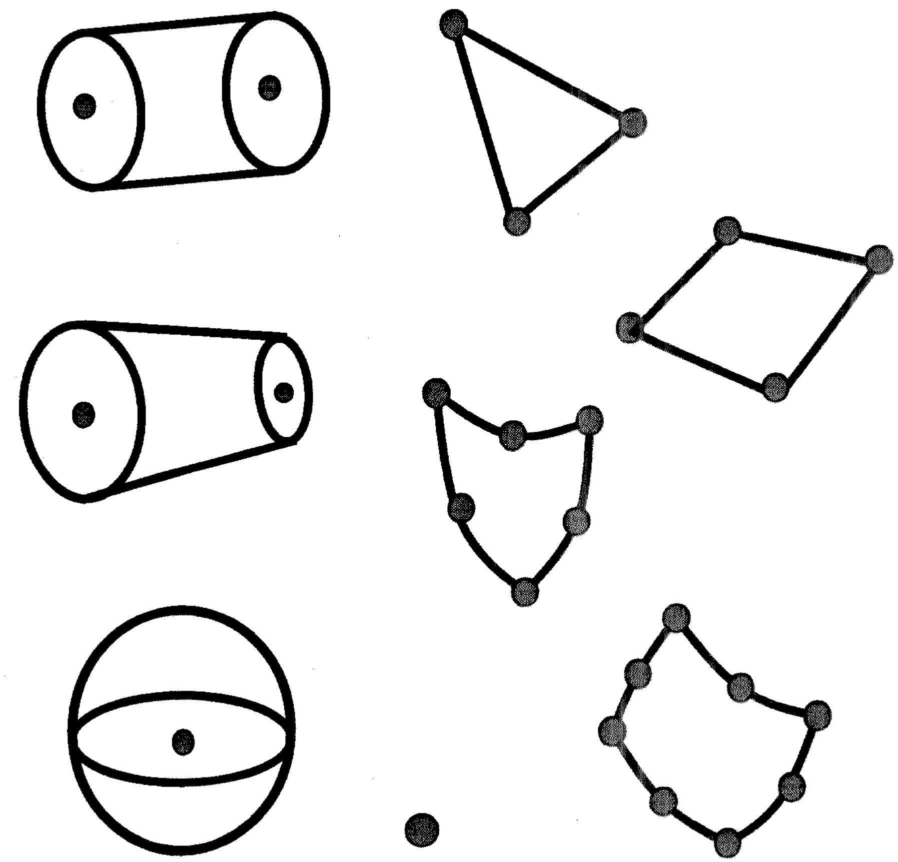
2D Target Elements

Targe169 can have the following target segment types: line, parabola, arc (clockwise or counterclockwise), circle, or pilot node.



3D Target Elements

<u>Targe170</u> can have the following target segment types: three node triangle, four node quad, six node triangle, eight node quad, cylinder, cone, sphere, and pilot node.



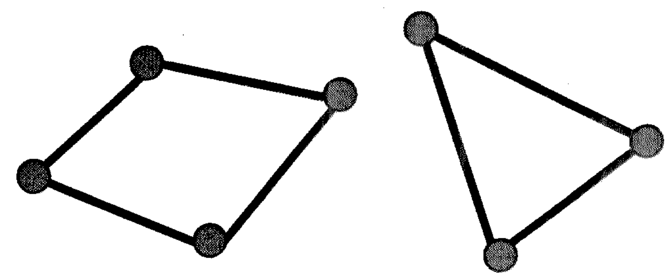
2D Surface-to-Surface Contact Elements

Conta171 is a 2-D, 2-node lower order line element that can be located on the surface of 2-D solid, shell, or beam elements.

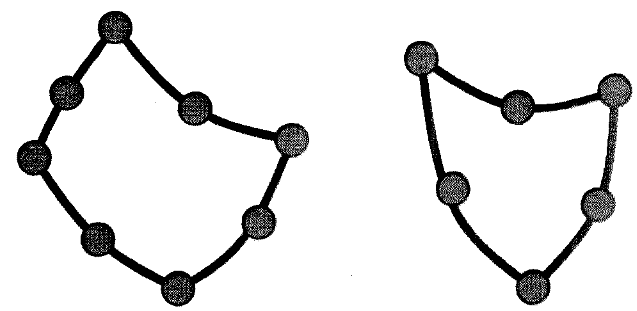
Conta172 is a 2-D, 3-node, higher order parabolic element that can be located on the surfaces of 2-D solid elements with midside nodes.

3-D Surface-to-Surface Contact Elements

<u>Conta173</u> is a 3-D, 4-node lower order quadrilateral element that can be located on the surface of 3-D solid or shell elements. It also can degenerate to a 3-node triangle.



<u>Conta174</u> is a 3-D, 8-node, higher order quadrilateral element that can be located on the surfaces of 3-D solid elements with midside nodes. It can also degenerate to a 6-node triangle.



Surface-to-Surface Procedure

- For the surface-to-surface contact elements one surface is designated as the "target" and the other as the "contact" surface.

- For rigid-to-flexible contact the rigid surface is *always* designated as the target surface. For flexible-to-flexible contact both the contact and target surface are associated with the deformable bodies.

- Contact elements are constrained against penetrating the target surface. However, target elements can penetrate through the contact surface.

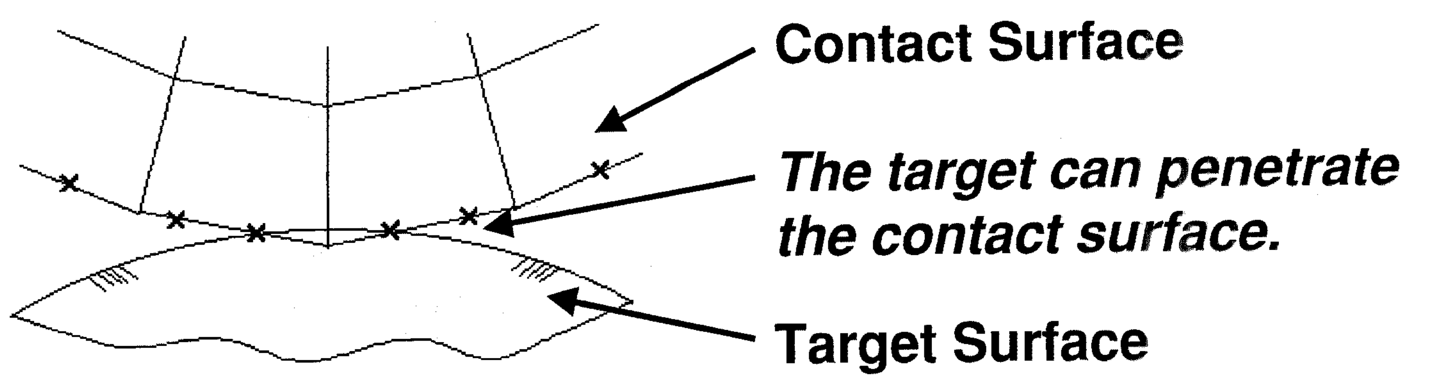
Surface to Surface Procedure

General Steps in a Contact Analysis:

1. Create the underlying finite element model.
2. Designate contact and target surfaces.
3. Set element options and real constants.
4. Create the target surface elements.
5. Create the contact surface elements.
6. Apply boundary conditions.
7. Define solution options and load steps.
8. Solve.
9. Review the results.

Designating Contact and Target Surfaces

The goal is to maximize the number of contact detection points. For rigid-to-flexible contact the target surface is always the rigid surface. For flexible-to-flexible contact the choice of the target and contact surface can cause a different amount of penetration and thus affect the solution accuracy.



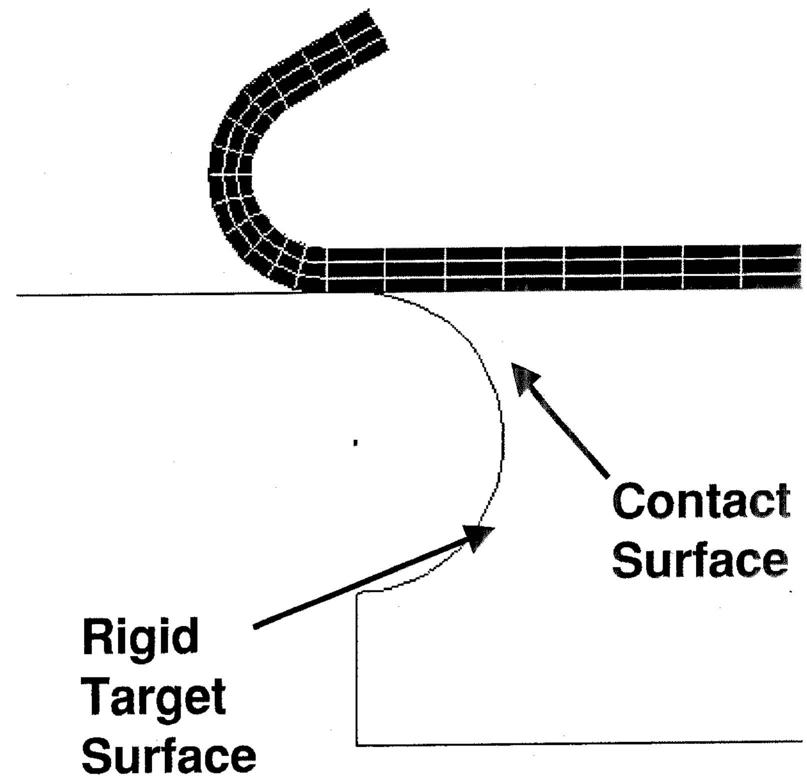
Guidelines for Target/Contact Surface

- If a convex surface comes into contact with a flat or concave surface, the flat or concave surface should be the target surface.
- If one surface has a coarse mesh and the other a fine mesh, the surface with the coarse mesh should be the target surface.
- If one surface is stiffer than the other, the stiffer surface should be the target surface.
- If one surface is higher order and the other is lower order, the lower order surface should be the target surface.
- If one surface is larger than the other, the larger surface should be the target surface.

Surface-to-Surface Procedure

Step 1. - Create the underlying solid model and its mesh.

Step 2. - Designate the contact and the target surfaces. In this example one surface is considered rigid, thus the target and contact surface designation is clear.



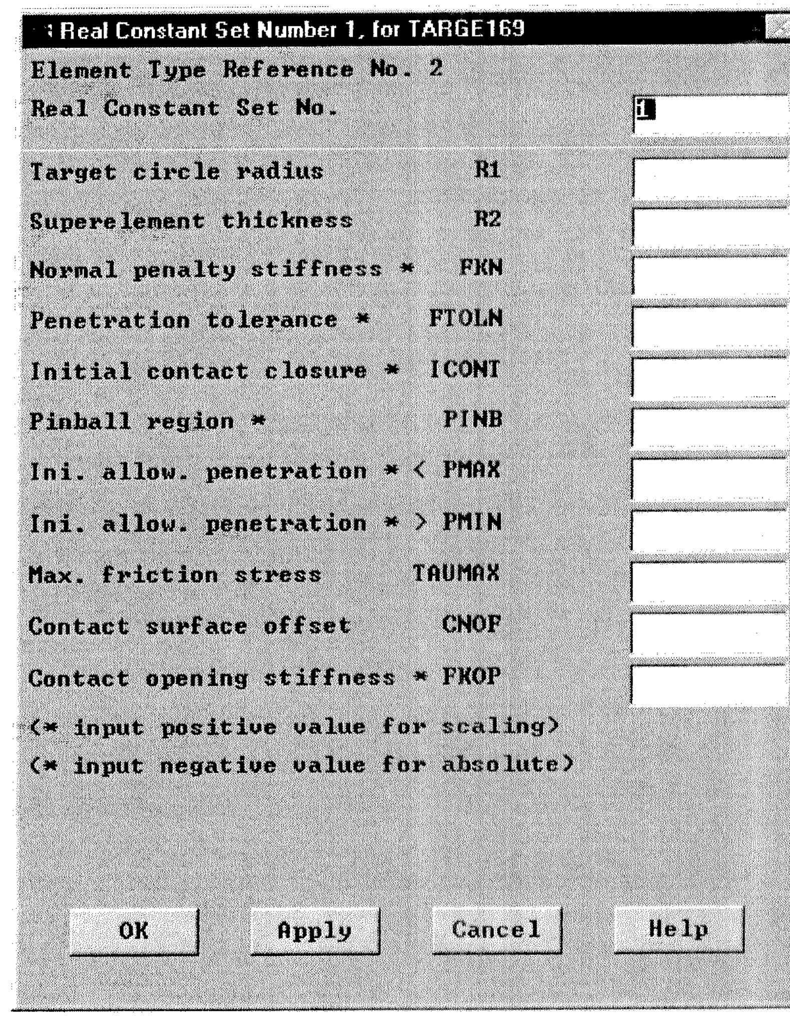
Hyperelastic Seal Insertion

Surface-to-Surface Procedure

<u>Step 3.</u> - Set the element options, and real constants.

IMPORTANT!

Contact pairs are identified by the real constant set number. The target and contact elements *must share the same real constant set*. (We will discuss in detail all the element options and real constants in a later section. The default element options and real constants settings are applicable for many problems.)



Surface-to-Surface Procedure

Step 3. - Set the element options, and real constants.

- The penalty stiffness scale factor (FKN) and the Lagrange multiplier penetration scale factor (FTOLN) should be set.
- FKN usually will be between 0.01 and 10. Use a value of 1.0 (default) for bulk deformation problems, and 0.1 for bending dominated problems.
- FTOLN defaults to 0.1. This value can be changed but too small a tolerance can cause an excessive number of iterations or non-convergence.

Surface-to-Surface Procedure

Step 4. - Create the target surface elements.

- In this step the method used will depend on whether the target surface is *rigid* or *deformable*.

For a *Rigid Target* use:

- Direct generation (E) or,
- Automatic meshing (LMESH, AMESH)

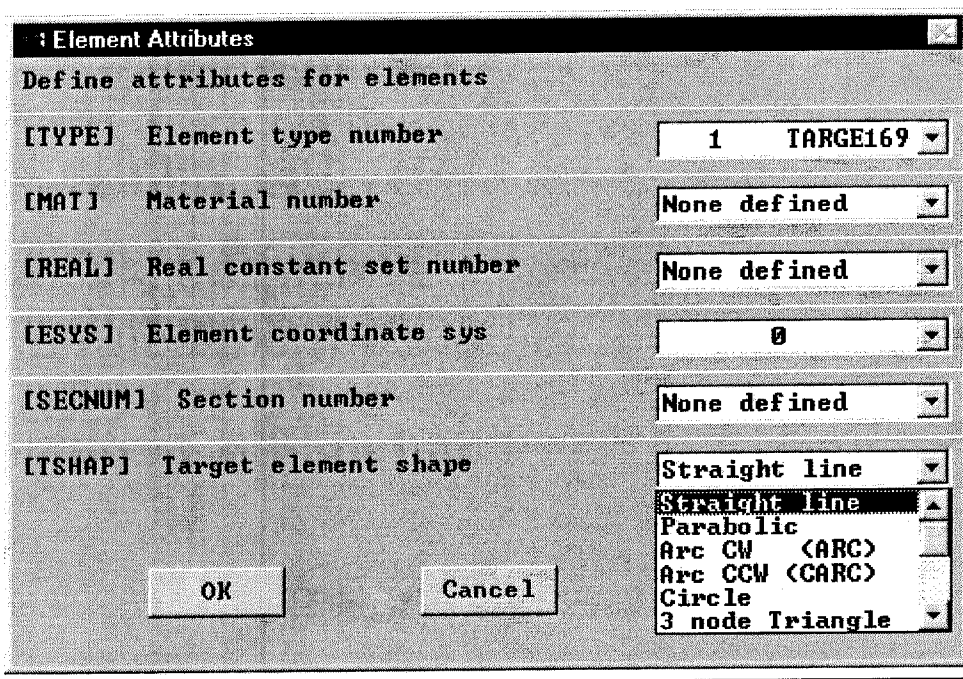
For a *Deformable Target* use:

- Create > Elements > Surface-to-Surface (ESURF)

Surface-to-Surface Procedure

<u>Step 4.</u> - Create the target surface elements.

- For direct generation of a *rigid target* surface, an additional element attribute, TSHAP, needs to be set before creating the target elements.



TSHAP defines the shape of the target elements (including "Pilot Node" shape).

Surface-to-Surface Procedure

<u>Step 4.</u> - Create the target surface elements.

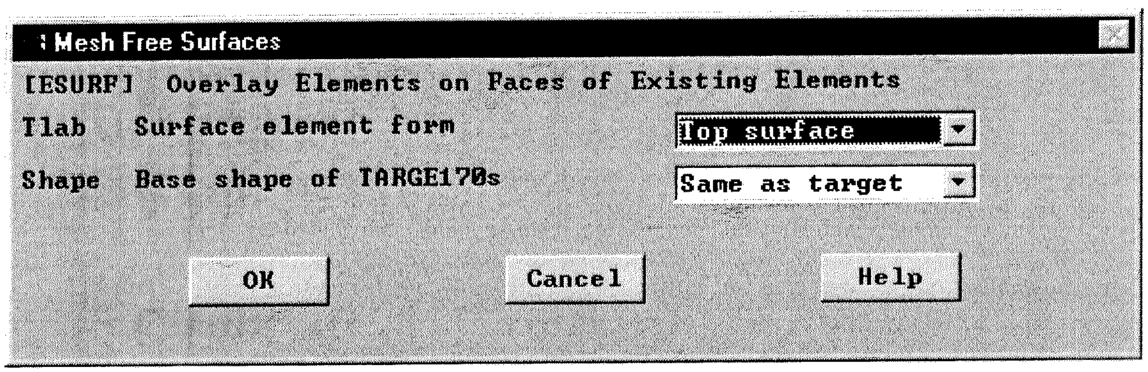
- TSHAP is *not* required for automatic meshing of a *rigid target* surface. ANSYS will determine the appropriate shape of the target element from the solid model.
  - Mesh Keypoints (KMESH) - pilot node
  - Mesh Lines (LMESH) - 2-D rigid target surface
  - Mesh Areas (AMESH) - 3-D rigid target surface

Surface-to-Surface Procedure

<u>Step 4.</u> - Create the target surface elements.

- **To automatically create target elements over a deformable mesh, select the deformable surface nodes, and create target elements on the deformable body:**

Preprocessor > Create > Elements > Surf to Surf



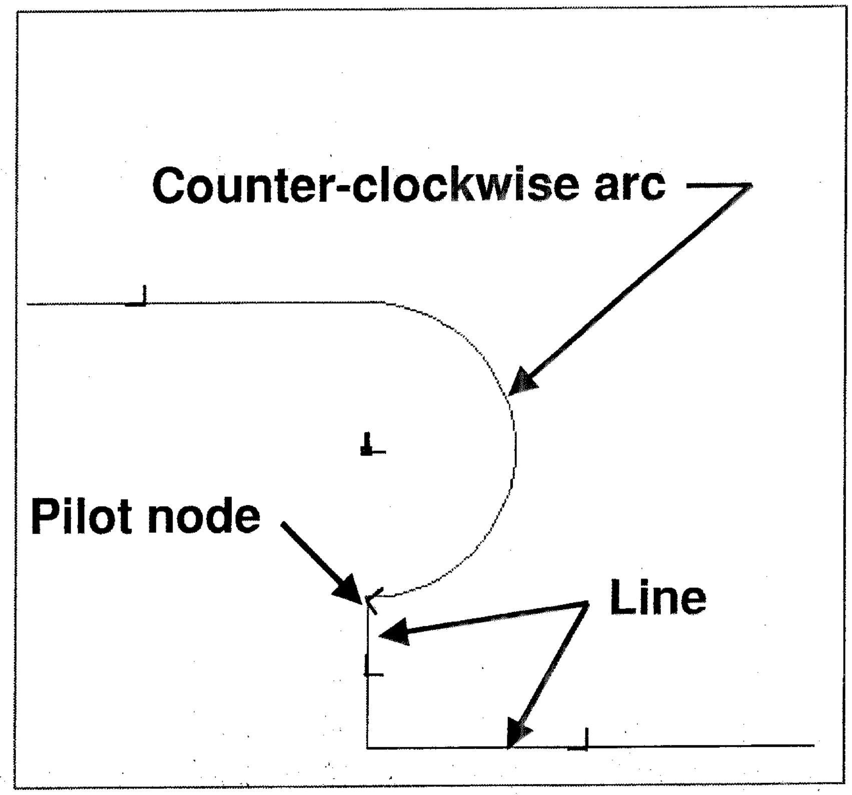
(ESURF)

ANSYS will determine the shape and outward normal direction of the target elements from the underlying mesh.

Surface-to-Surface Procedure

Step 4. - Create the target surface elements.

- Check outward normal direction (very important for rigid targets if automatic meshing was used):
  - Plot elements with element coordinate system symbol turned on.
  - Fix with ESURF (Node Num reverse). Acts on *selected* elements

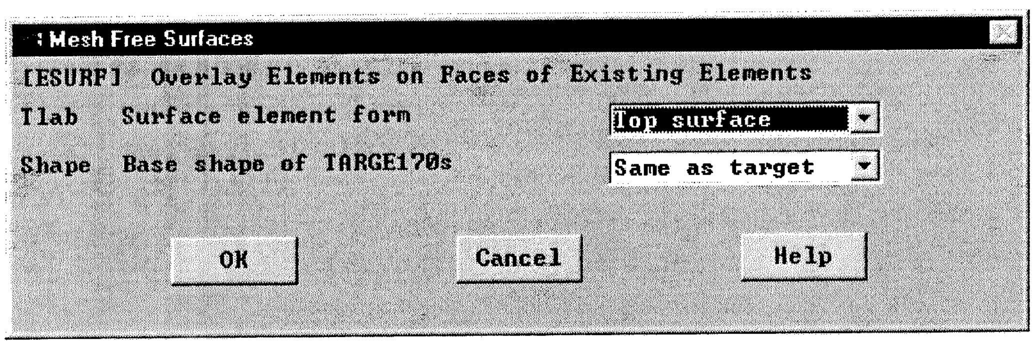


Surface-to-Surface Procedure

<u>Step 5.</u> - Create the contact surface elements.

- Set contact element attributes, select deformable surface nodes, and create contact elements on the deformable body (same procedure as for the target elements on a deformable mesh):

Preprocessor > Create > Elements > Surf to Surf



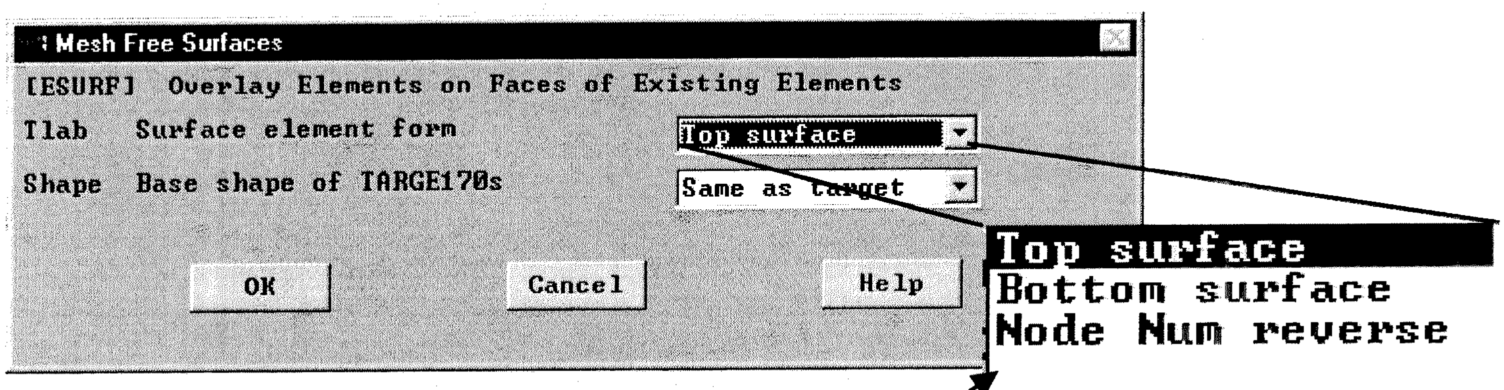
(ESURF)

These contact elements have the same order (lower or higher order) as the underlying elements.

Surface-to-Surface Procedure

Step 5. - Create the contact surface elements.

- Note that for target or contact element creation over shell or beam elements, there is an option to create the elements on the top or the bottom surfaces of the beam or shell elements.

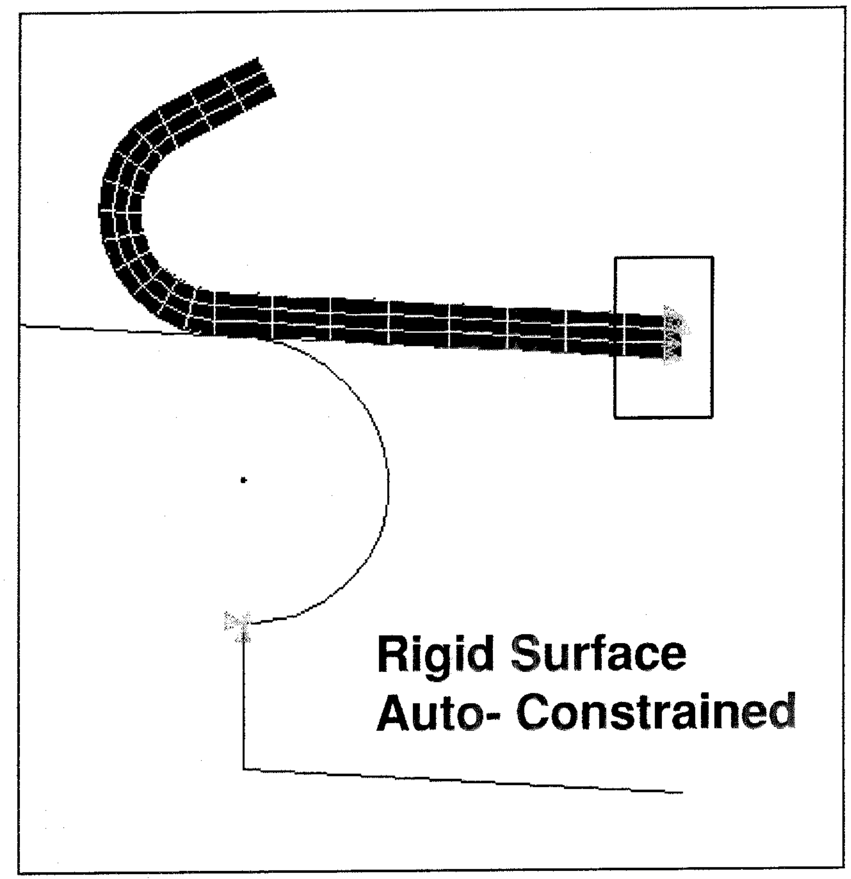


Can control element location

Surface-to-Surface Procedure

Step 6. - Apply boundary conditions on the finite element model.

Note: If the target surface is a rigid surface, the target surface will be fixed automatically. (We will discuss the rigid surface options and how to control the motion of the rigid surface in a later section.)



Surface-to-Surface Procedure

<u>Step 7.</u> - Define solution options and load steps.

- Use of solution control is strongly recommended for solving contact problems. The guidelines given below are typical, and are the *default settings* with solution control.
- A discontinuity can occur if the contact status changes during the iteration process. To avoid slow convergence use the full Newton-Raphson without adaptive descent.
- Time step size must be small enough to provide for a smooth transfer of contact forces. A reliable way to ensure an accurate time step is to use automatic time stepping.

Surface-to-Surface Procedure

Step 7. - Define solution options and load steps.

- Set the number of substeps accordingly:
  - The starting substep size (NSBSTP) can be large for assembly contact, small for large sliding contact.
  - The maximum number of substeps (NSBMX) should be large for severe discontinuities.
  - The minimum number of substeps (NSBMIN) should be small.

Note that all equilibrium iterations are carried out before bisection for the first substep to establish initial contact conditions.

Surface-to-Surface Procedures

Step 7. - Define solution options and load steps.

- Equation Solver Choices:
  - The *sparse solver* is the default, and this is the best choice for a medium size contact model (under 100,000 DOF). The sparse solver also has an unsymmetric option for difficult sliding problems.
  - The *PCG solver* is an option for large contact models (over 100,000 DOF). This option is for symmetric stiffness matrices only.

Surface-to-Surface Procedure

Step 7. - Define solution options and load steps.

- *Do not use adaptive descent.* Adaptive descent does not provide any convergence help for surface-to-surface contact problems.
- Use a reasonable number of equilibrium iterations, usually 25.
- Because iterations can become unstable for large time increments, use line search to stabilize the calculations.
- Turn the predictor on, *except for large rotations or dynamic analyses.*

ANSYS

Surface-to-Surface Procedure

<u>Step 7.</u> - Define solution options and load steps.

- Many convergence failures in a contact analysis occur because of too large of a penalty stiffness or too small of a penetration tolerance. Be sure to follow the recommendations for the penalty stiffness.

  Note: For small strain, small displacement, and small sliding problems turn large deformation effects off (NLGEOM,OFF). This setting will speed up the contact searching time.

<u>Step 8.</u> - Solve the problem.

Surface-to-Surface Procedure

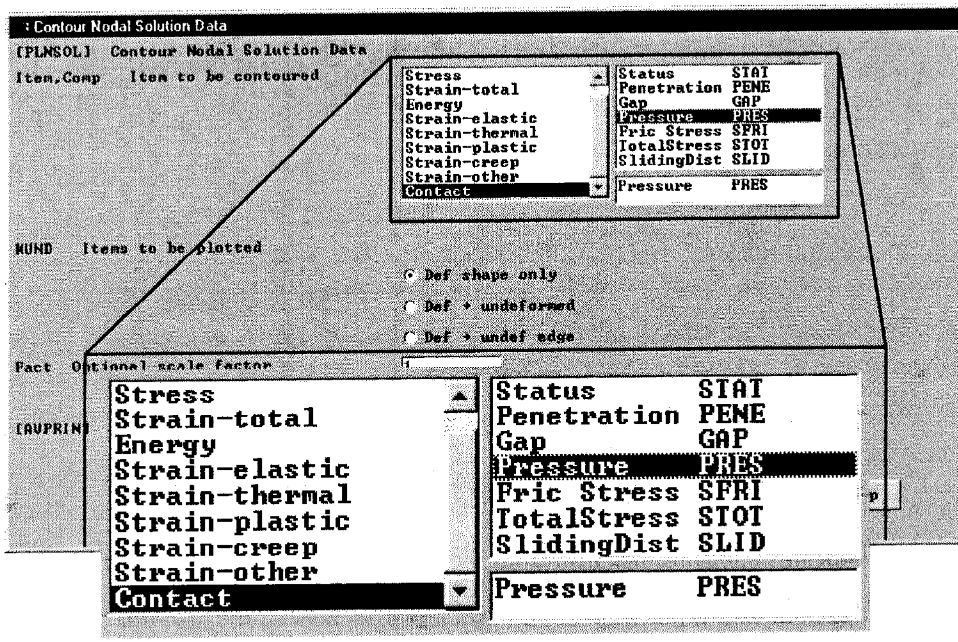
Step 9. - Review the results.

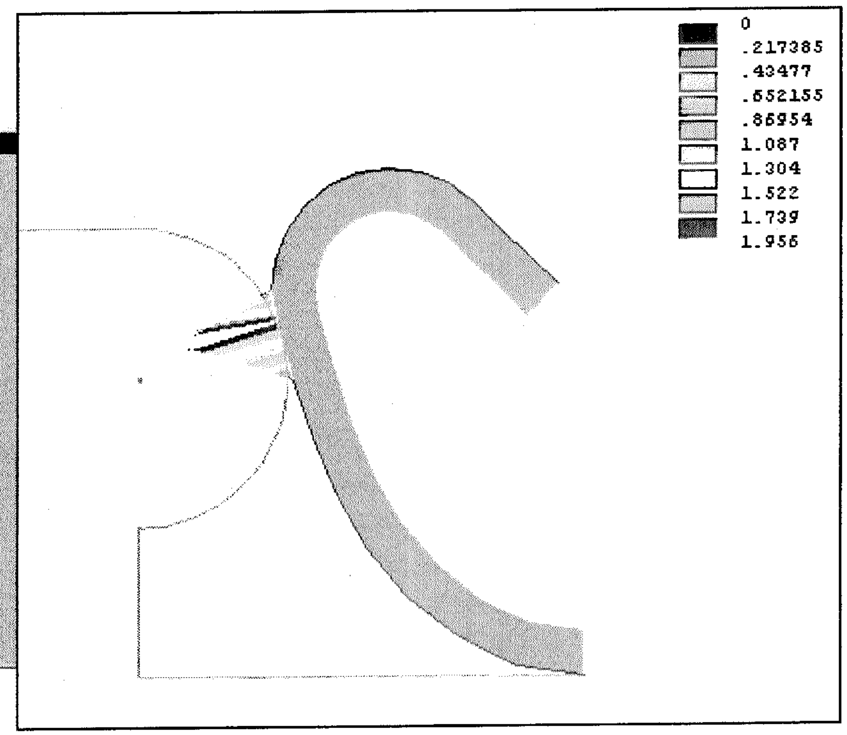
- Results of a contact analysis consist mainly of displacements, stresses, strains, and contact information.
- The contact pressures, frictional stress, total stress, contact penetration, contact gap distance, sliding distance, and the contact status are all available for postprocessing directly from the General or Time History Postprocessors.

Surface-to-Surface - Procedure

Step 9. - Review the results.

Plot Contact Pressures:

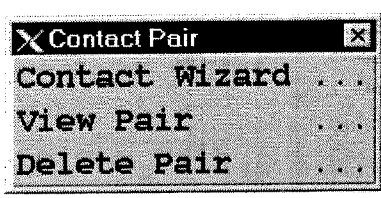


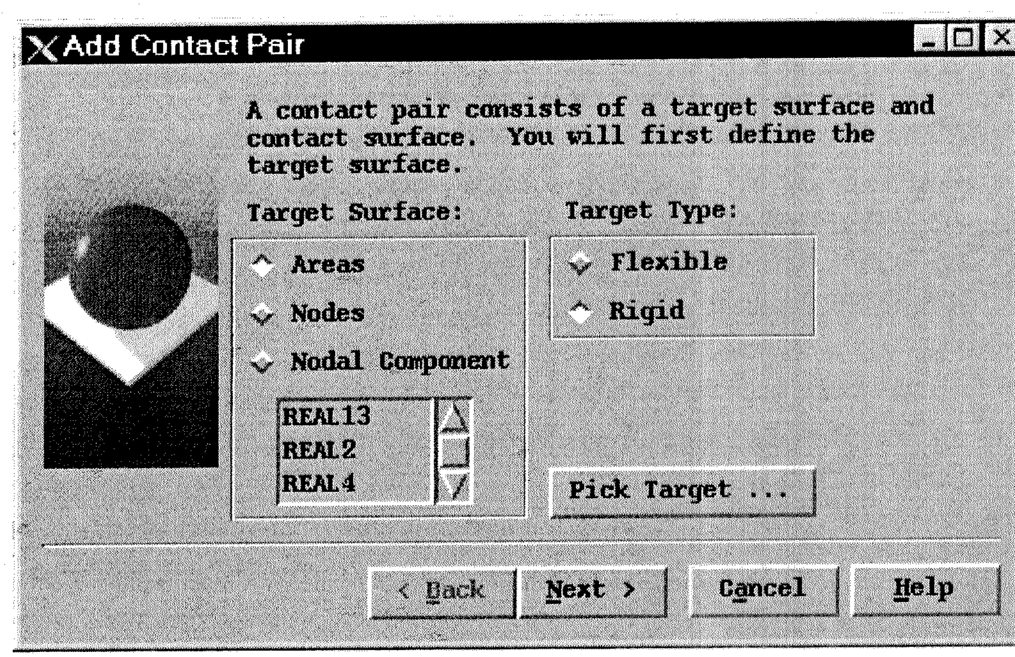


Contact Wizard

The Contact Wizard provides a simple way to construct a contact pair for most contact problems. The Contact Wizard will guide you through the process of creating a contact pair.

Preprocessor > Create > Contact Pair > Contact Wizard



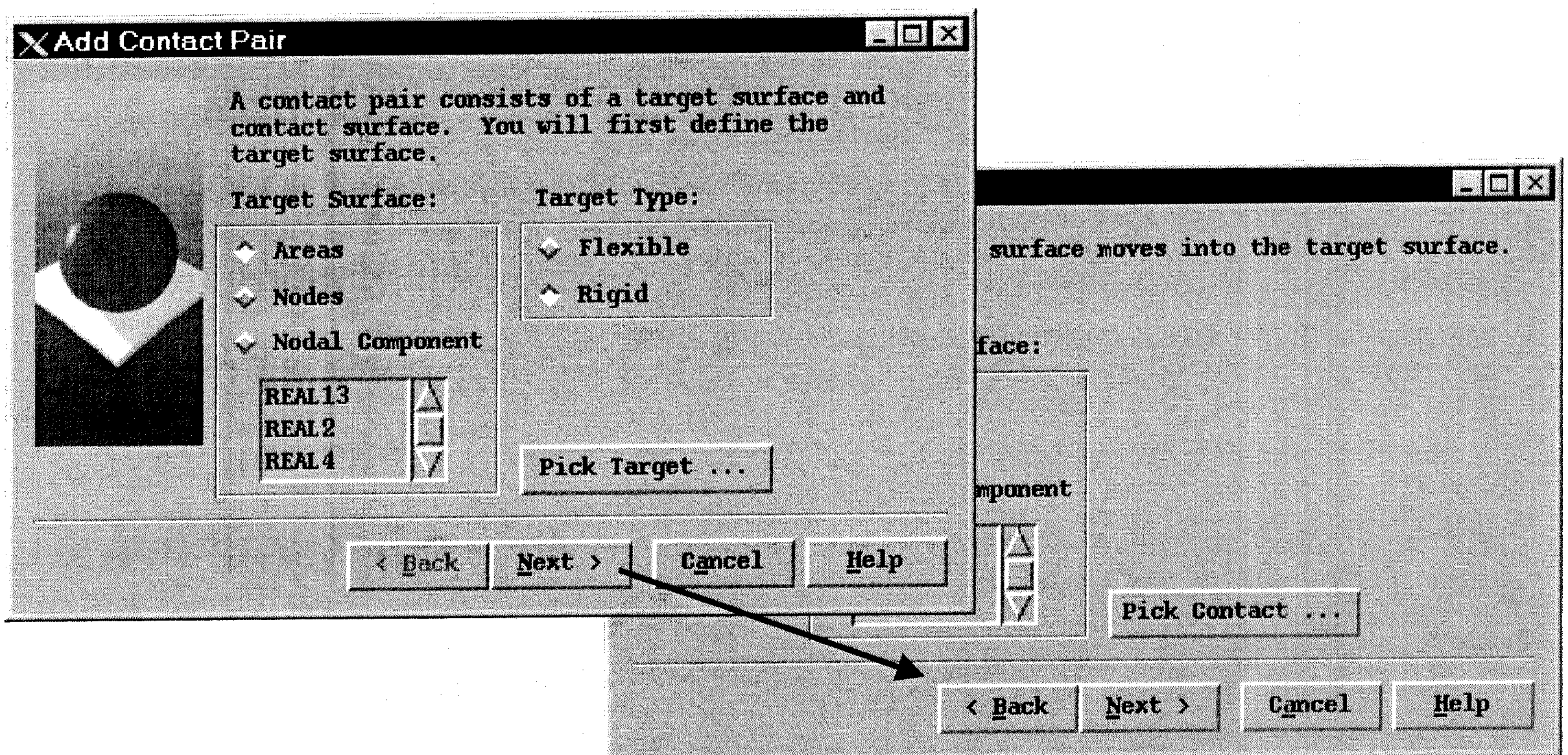


Contact Wizard

- **Benefits of the Contact Wizard:**
  - Automatically defines element types and real constant sets
  - Quickly accesses the contact options and parameters
  - Contact pair viewing tools
  - Quickly displays and reverses contact normals
- The Contact Wizard remains unavailable if you have not meshed any portions of your model.
- Before launching the wizard to create a flexible-to-flexible model, mesh all parts of the model which will be used as contact surfaces. To create a rigid-to-flexible model, mesh only the parts of the model which will be used as flexible contact surfaces.

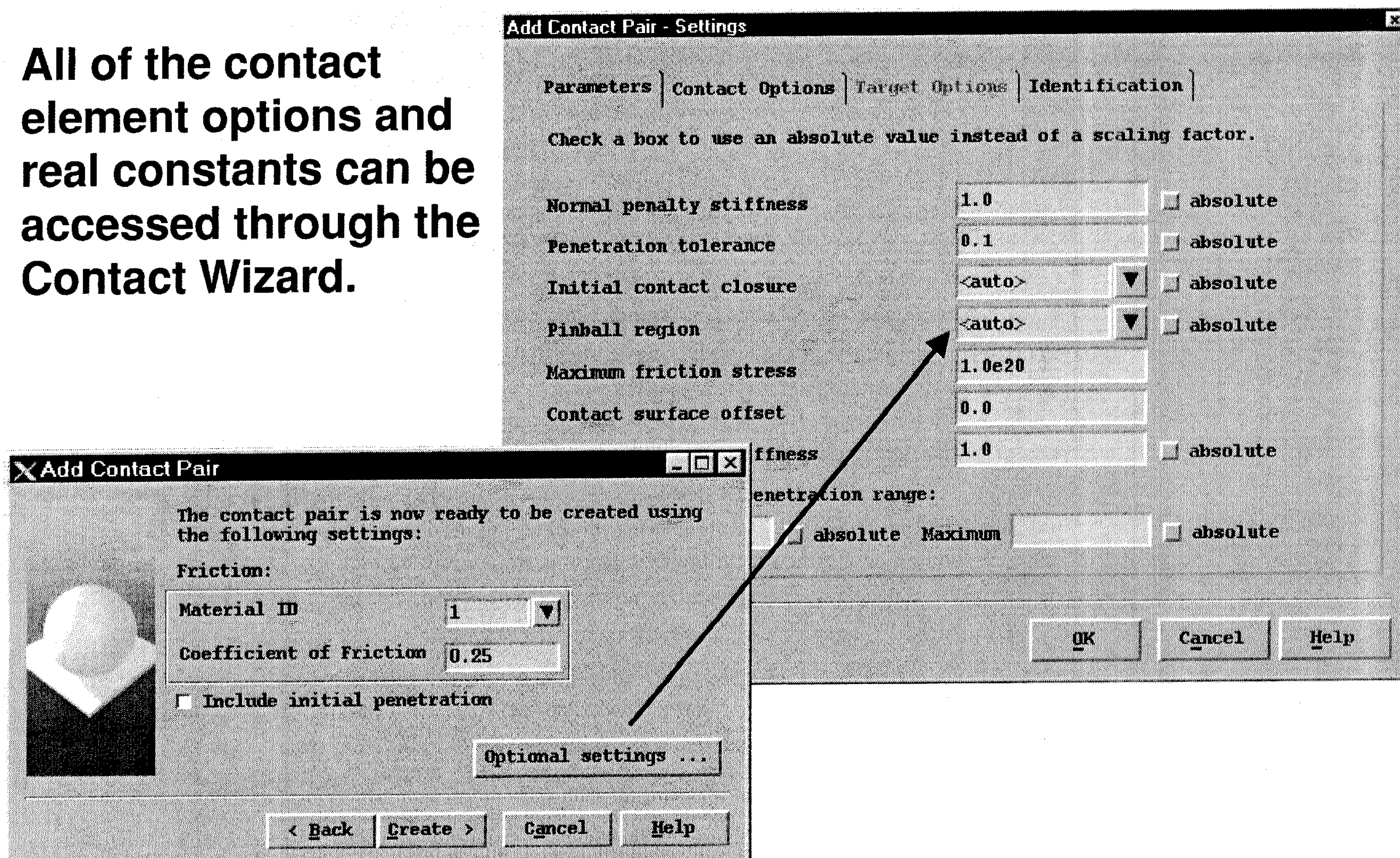
Contact Wizard

- Simply follow the wizard through the contact pair creation steps. First you will identify the target surfaces and then the contact surfaces.



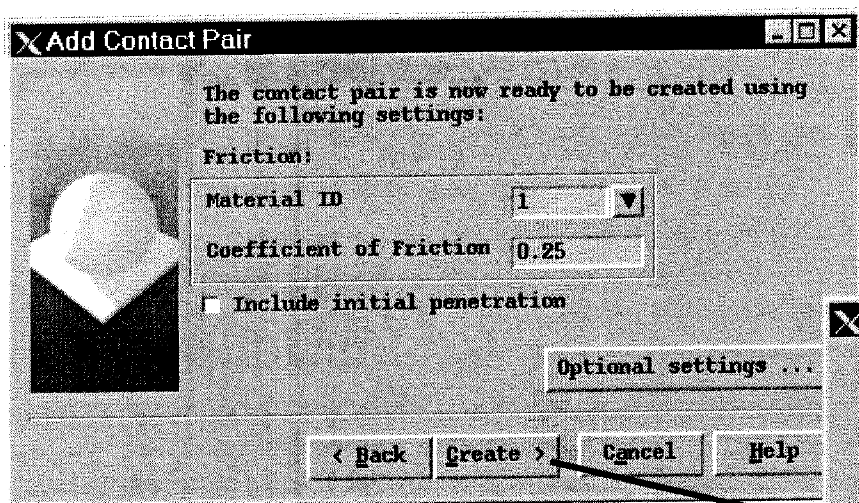
Contact Wizard

All of the contact element options and real constants can be accessed through the Contact Wizard.

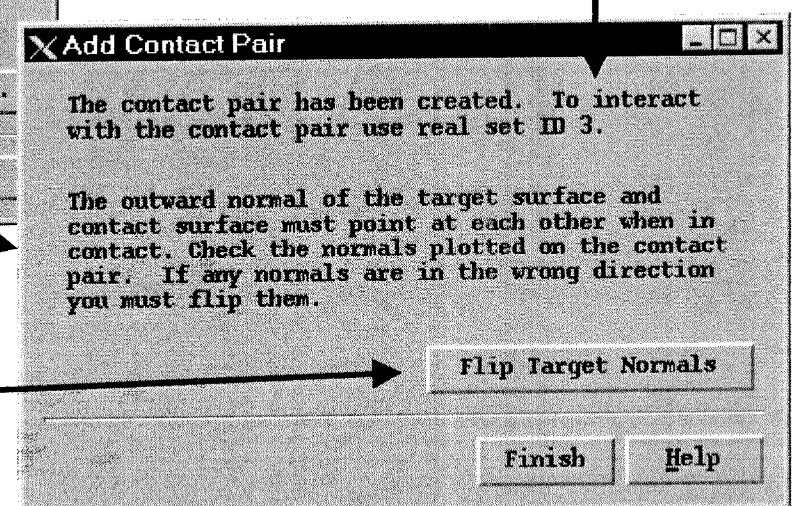


Contact Wizard

- The Contact Wizard will create the contact elements and the corresponding real constants as the final step.



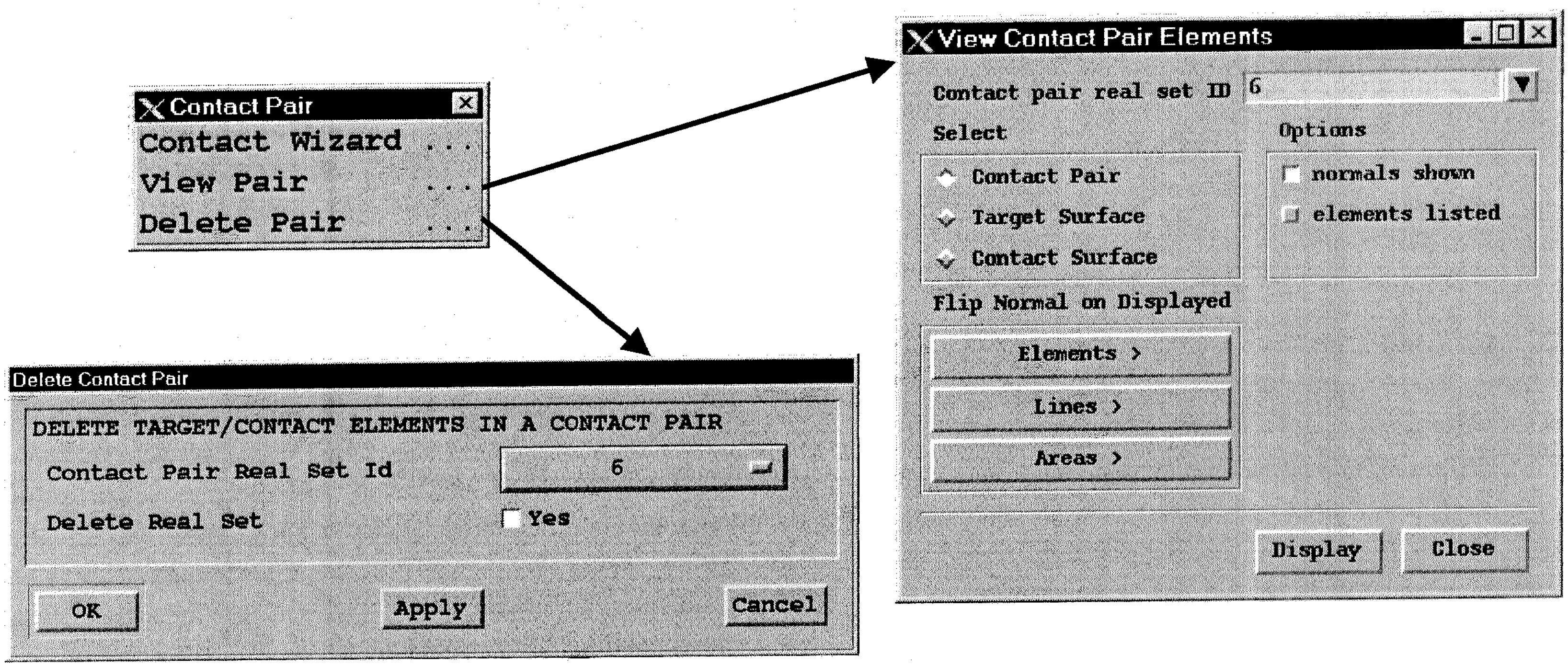
The real constant set created is identified.



If the target normal directions are incorrect, they can be reversed.

Contact Wizard

- The Contact Wizard also allows you to view, list and delete contact pairs, and quickly display and reverse normal directions.



Notes

Contact Nonlinearities

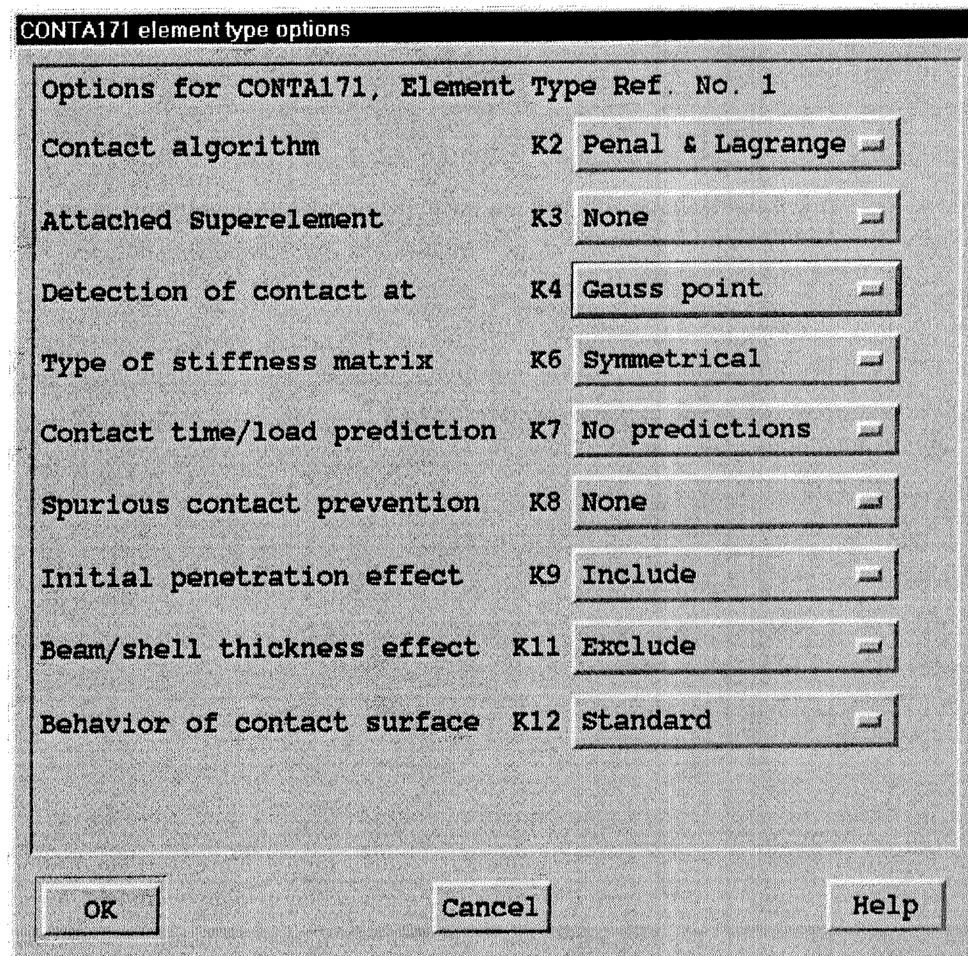
Modeling and Solution Strategies

Session Objective

- At the completion of this session you will be able to describe and demonstrate the following:
  1. Surface-to-Surface Element Options
  2. Rigid Surfaces
  3. Determining a Contact Stiffness
  4. Friction
  5. Rigid Body Modes
  6. Symmetric versus Asymmetric Contact (Self Contact)
  7. Initial Interference
  8. Trouble Shooting

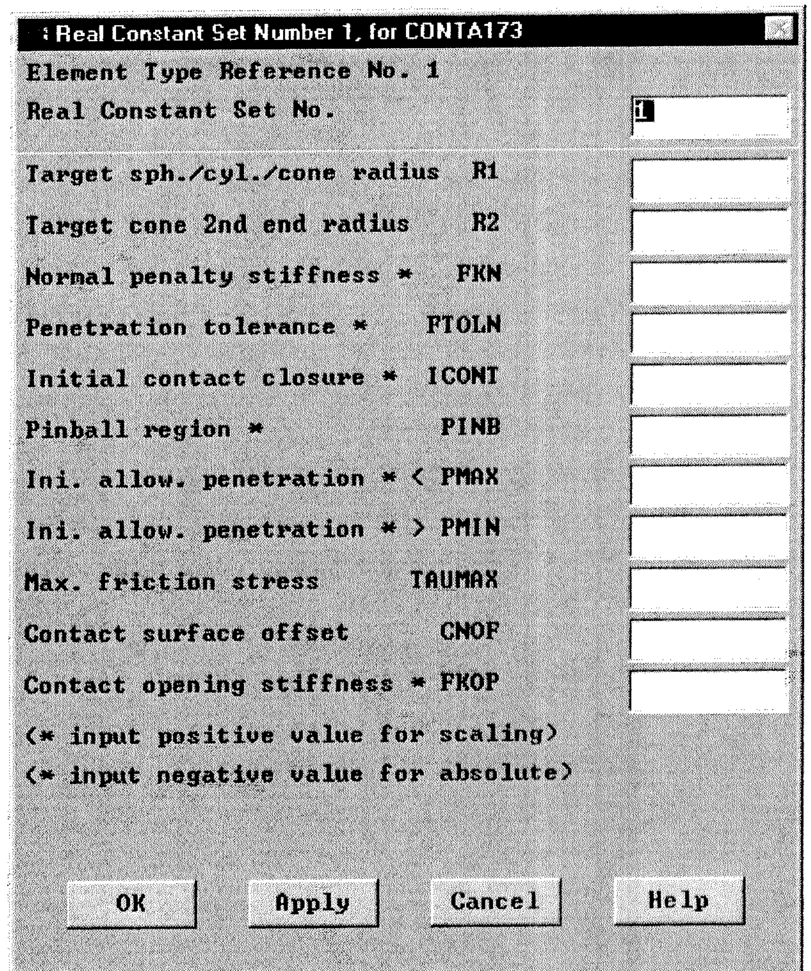
Surface-to-Surface Options

The remainder of this section will discuss the various surface-to-surface contact element options (KEYOPT's) and real constants which control the behavior of the surface-to-surface contact elements.



Surface-to-Surface Options

We will also discuss modeling techniques and strategies for commonly encountered contact problems. Including rigid surfaces, friction, rigid body modes, initial interference, surface interaction models, and trouble shooting.



Rigid Surfaces

There are some additional modeling considerations for using a rigid surface in a contact problem. These include:

- Pilot Nodes
- Meshing Rigid Surfaces
- Changing the Shape of the Target Surface
- Connecting other Elements to Rigid Surfaces

Rigid Surfaces

Pilot Node

- The rigid surface can be associated with a "pilot node" whose motion governs the motion of the target surface. Forces, displacements and/or rotations for the entire surface can be specified on the pilot node. You can think of the pilot node as a handle for the entire rigid surface.

- If you define a pilot node, ANSYS checks for boundary conditions only on the pilot node and ignores the constraints on the other nodes of the target surface.

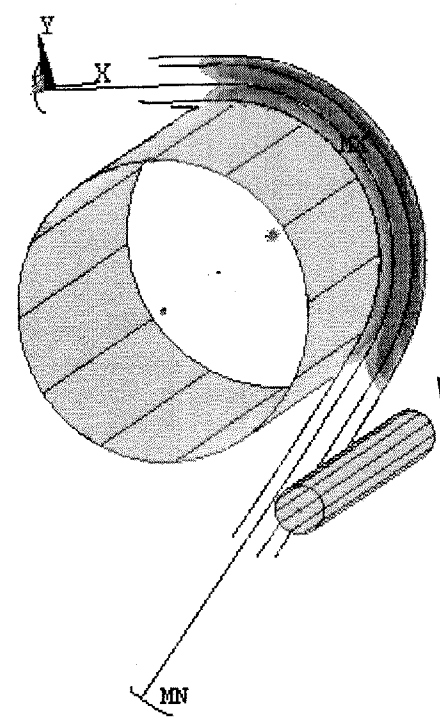
- *The pilot node can be generated either by meshing a keypoint, or by direct generation using the same element attributes as the target elements.*

Rigid Surfaces

Pilot Node

- The pilot node can be specified at any location. This allows for general rotations of the rigid target surface.
- Only the pilot node can connect to other elements. To account for the mass of a rigid body, define a mass element (MASS21) at the pilot node.
- Each target surface can have only one pilot node.

Rigid surface rotated

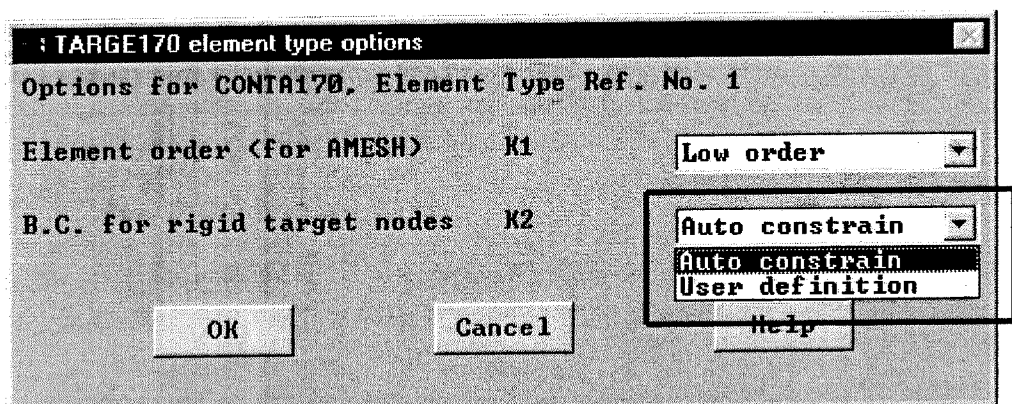


Rigid Surfaces

Constraining a Rigid Target

By default a rigid target is *auto constrained* (KEYOPT(2)=0), if the following conditions are met:

- **No explicit boundary conditions are defined.**
- **The target surface is not connected to other elements.**
- **No couples or constraint equations are defined.**



In this case, all of the degrees of freedom on the rigid target surface are constrained.

Rigid Surfaces

Constraining a Rigid Target

- If the *auto constrain* (KEYOPT(2)=0) option is selected, the internal settings for the constraints on the rigid surface are reset at the end of each load step. Check the model carefully when restarting the solution.

- If *user definition* (KEYOPT(2)=1) is selected, ANSYS expects that all appropriate boundary conditions for the rigid target surface are defined by the user.

Rigid Surfaces

Meshing Complex Target Surfaces

- A rigid target can be a combination of primitive target segments (cylinder, cone, sphere, etc.) and general segments. For an arbitrary surface, the quality of the mesh shape is not important. *However, it is important that the target elements represent the geometry well.* Excessively coarse discretization of the target surface can result in convergence problems.

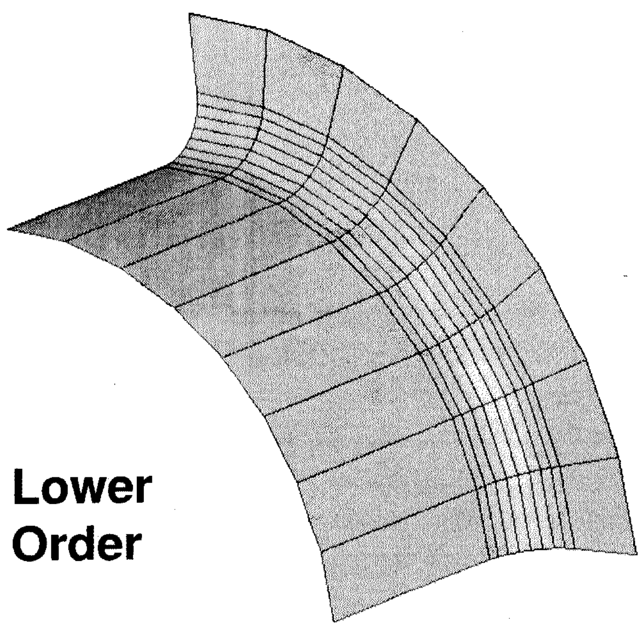
Rigid Surfaces

Meshing Complex Target Surfaces Recommendations

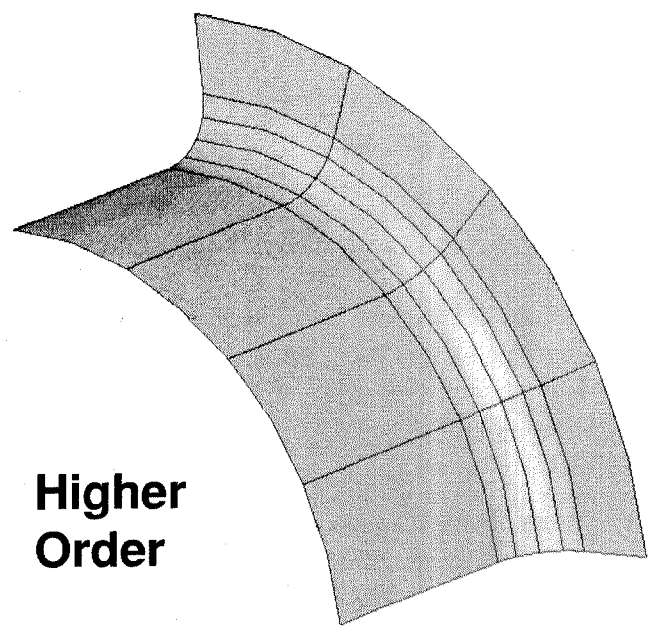
- If the rigid target can be represented by primitives, use primitives. (Fewer elements are required, more efficient.)
- For complex surfaces, use mapped meshing wherever possible.
- If there is no curvature on one edge of the surface, assign one element division on that edge.
- Use lower order targets for flat or nearly flat surfaces. Use higher order elements if the target surface is curved.

Rigid Surfaces

- Below are examples of meshing patterns for a rigid surface using both lower order and higher order elements. Lower order elements are less expensive computationally but more elements are required. Higher order elements are expensive, but fewer elements are necessary.



Lower Order



Higher Order

Rigid Surfaces

Changing the Shape of the Target Surface

- For a target surface meshed with either higher order or lower order target segments (not primitives), the target nodes can be rotated into a local coordinate system. This allows a target surface to change shape within an analysis.

- For example, if an expanding rigid cylinder was desired. The cylinder could be meshed with higher order quadrilaterals, the target nodes rotated, and radial displacements imposed on the rigid target to expand the cylinder.

Contact Algorithm

Selecting a Contact Algorithm KEYOPT(2)

- The augmented Lagrangian method (KEYOPT(2)=0) is the default option and is recommended for general usage. It is less sensitive to the penalty stiffness, but also requires the specification of a penetration tolerance.

- The penalty method (KEYOPT(2)=1) can be selected as an option. Its use is recommended for problems with very distorted elements, a high coefficient of friction, and/or poor convergence behavior with the augmented Lagrangian.

Determining a Penalty Stiffness

- For the surface-to-surface contact elements, ANSYS determines the contact stiffness based on the element type, material properties, and element sizes of the underlying finite element mesh. You can use the real constant FKN to specify either a scaling factor or absolute value for contact stiffness.

- The penalty stiffness (FKN) should be large enough that contact penetration is small, but low enough such that the problem does not have convergence problems.

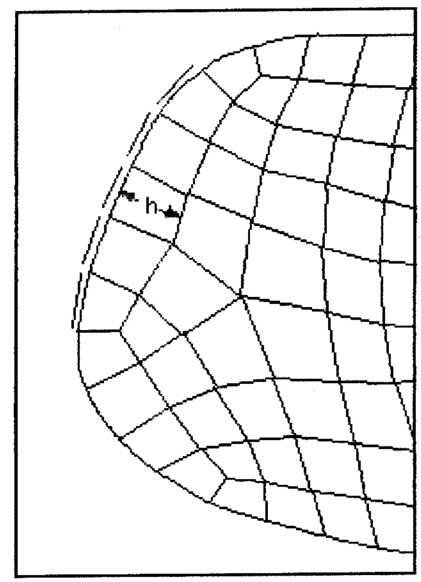
- FKN usually will be between 0.01 and 10. Use a value of 1.0 (default) for bulk deformation problems, and 0.1 for bending dominated problems.

Determining a Penetration Tolerance

- The penetration tolerance (FTOLN) is a scale factor multiplied by the depth (h) of the underlying element.

- Too small of a value will lead to convergence difficulty. *Never use too small of a tolerance!* Increasing the penalty stiffness (FKN) will reduce the penetration.

- However, increasing FKN by 100 times will reduce the penetration accordingly, but may only change the contact pressure by 5%.



Determining a Penalty Stiffness

Determining a good stiffness value may require some experimentation. The following procedure may be used as a guideline:

1. **Use a low value of FKN to start.**
2. **Run the analysis to a fraction of the final load.**
3. **Check the penetration and number of equilibrium iterations used in each substep. If convergence is being driven by the penetration tolerance, FKN may be underestimated or FTOLN may be too small. If convergence requires many iterations to converge the residual forces, rather than the penetration, FKN may be overestimated.**
4. **Adjust FKN or FTOLN and run the full analysis.**

Note: Both FKN and FTOLN can be modified from one load step to another, and can be adjusted in a restart.

Friction

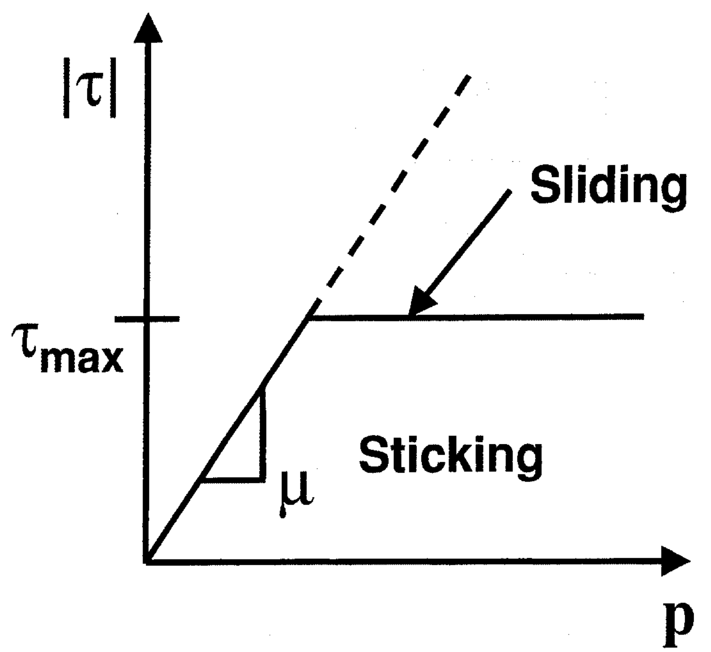
- Including friction in a contact model allows shear forces to develop between the bodies in contact. In a Coulomb friction model the two surfaces carry shear stresses up to a certain magnitude before they start sliding relative to each other. This state is called sticking. The Coulomb friction model defines the equivalent frictional stress τ at which sliding occurs as:

$$\tau = \mu \cdot p$$

- Where, p is the contact pressure and μ is the coefficient of friction (defined as the material property MU). Once the shear stress is exceeded the two surfaces will slide relative to each other. This state is known as sliding. The coefficient of friction μ can be any non-negative value.

Friction

- **ANSYS provides a real constant TAUMAX for defining the maximum equivalent shear stress such that regardless of the magnitude of the contact pressure, sliding will occur.**



The shear limit is usually used in cases where the contact pressure becomes very large. A reasonable upper estimate for TAUMAX is:

$$\frac{\sigma_y}{\sqrt{3}}$$

Where σ_y is the von Mises yield stress of the material adjacent to the surface. Empirical data is often the best source for TAUMAX.

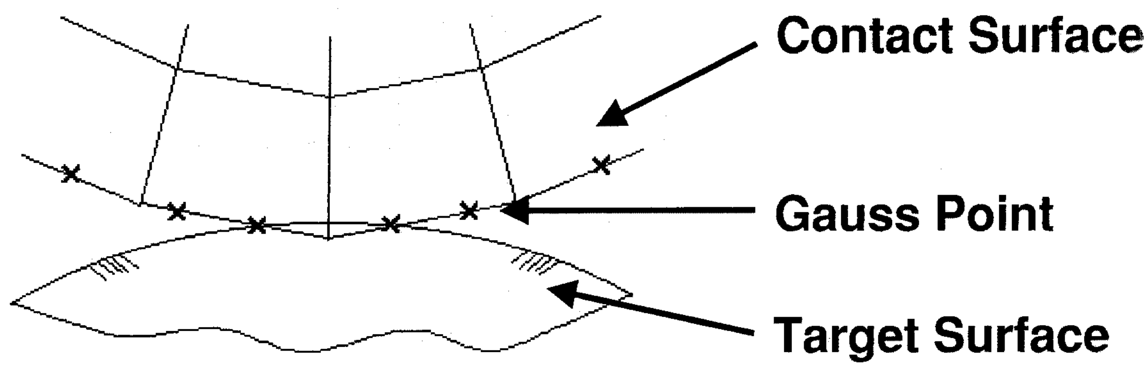
Friction

- Contact problems involving friction produce an unsymmetric stiffness matrix. However, using an unsymmetric equation solver is more computationally expensive than a symmetric solver. For this reason ANSYS uses a symmetrization algorithm by which most contact problems involving friction can be solved.

- If you are experiencing a slow rate of convergence, an unsymmetric solution option (KEYOPT(6)=1) is available. In this case you would need to use either the *sparse* (preferred) or the *frontal* solver.

Contact Detection Point

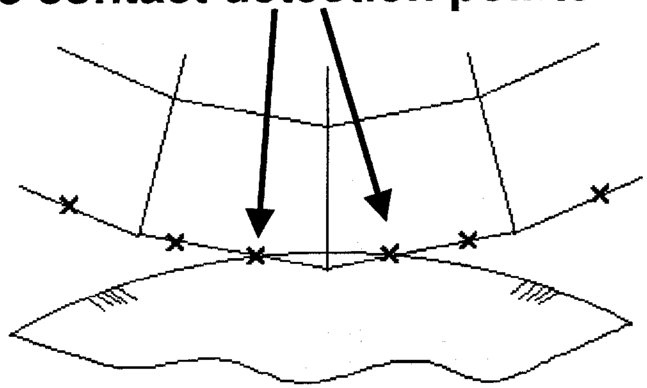
- **ANSYS surface-to-surface elements by default use the Gauss integration points as the contact detection points.**

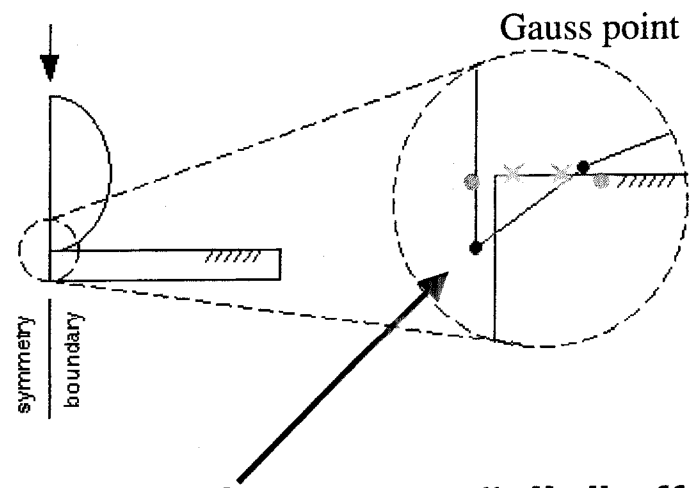


- **ANSYS allows moving the contact detection points to the nodes by setting KEYOPT(4)=1.** ***It is recommended for both convergence and accuracy to keep the default setting.***

Gauss Point Advantages

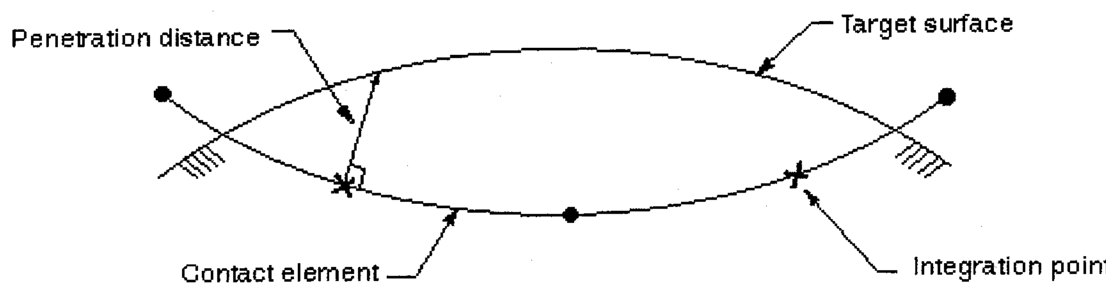
More contact detection points.





Consistent nodal forces for higher order elements.

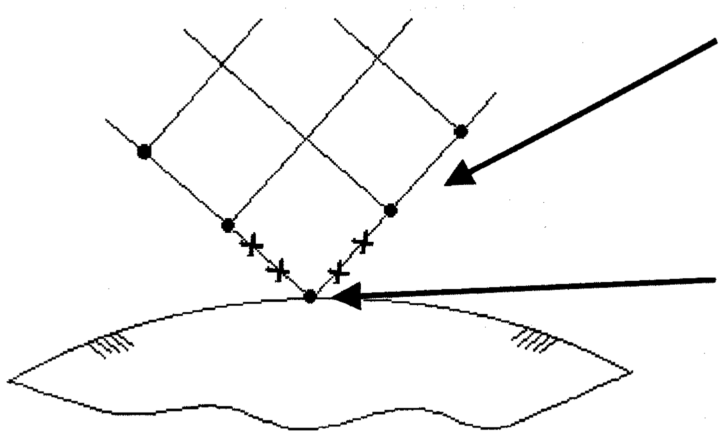
Gauss point can not "slip" off the edge of a target.



A unique penetration is defined normal to the Gauss point, no smoothing of the target surface is required.

Contact Detection Point

- If you have a problem which includes contact at a corner, using the Gauss point as the contact detection point can result in an over penetration at the corner. Rather than moving the contact detection point to the node, you can mix the surface-to-surface contact elements with node-to-surface elements.



Symmetric versus Asymmetric Contact

- In the previous section we discussed having one surface designated as the *target surface* and one surface designated as the *contact surface*. Having all contact elements on one surface and all target elements on another surface is referred to as *asymmetric contact.*
- Asymmetric contact is generally the most efficient way to model surface-to-surface contact. However, under some circumstances asymmetric contact will not perform satisfactorily. In such cases, you should use symmetric contact.
- In *symmetric contact,* you designate each surface to be a target *and* a contact surface. You then create *two* sets of contact elements between the contacting surfaces.

Asymmetric Contact

Recall, that the contact element cannot penetrate the target, but the target can penetrate the contact element. The following are guidelines for asymmetric contact:

Contact surface:

- Convex surface
- Fine mesh surface
- Softer surface
- Higher order surface
- Smaller surface

Target surface:

- Concave/flat surface
- Coarse mesh surface
- Stiffer surface
- Lower order surface
- Larger surface

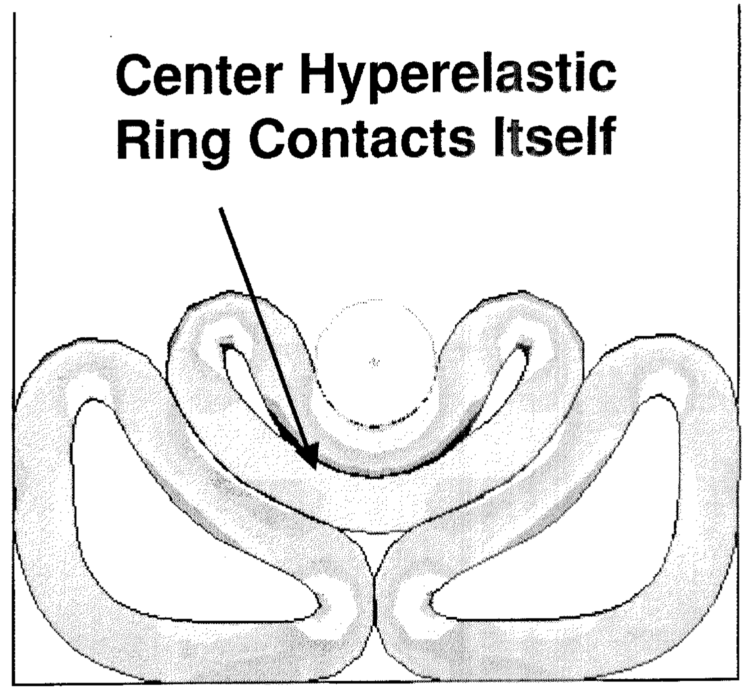
The guidelines above are determined to maximize the number of contact detection points.

Symmetric Contact

- Symmetric contact is less efficient than asymmetric contact. However, many analyses will require its use (typically to reduce penetration). *Symmetric contact increases the number of contact detection points.*
- Guidelines for symmetric contact:
  - No clear distinction exists between target and contact surfaces.
  - Both target and contact surface have coarse meshes.
- *Note that when using symmetric contact, postprocessing is more difficult. The contact pressure is the average value from both sets of contact elements.*

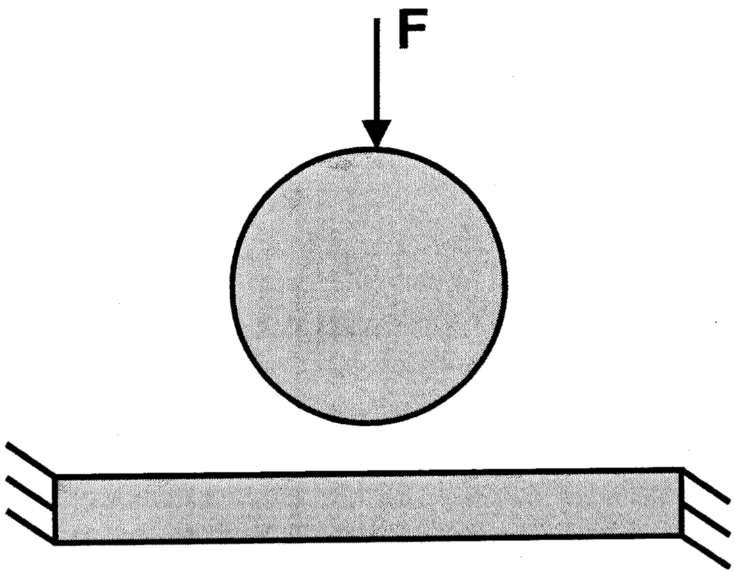
Self Contact

Using asymmetric contact is more efficient for self contact, but it may be difficult to predict the contact and target surfaces. To use symmetric contact for self contact, simply place the target elements and contact elements on the same surface.



Rigid Body Modes

In a static analysis of two (or more) bodies that are initially unconnected, a rigid body motion can result before the contact is established.



In this force controlled example the cylinder has no applied displacement constraints. The constraints on the cylinder are established by the contact between the cylinder and the plate.

Rigid Body Modes

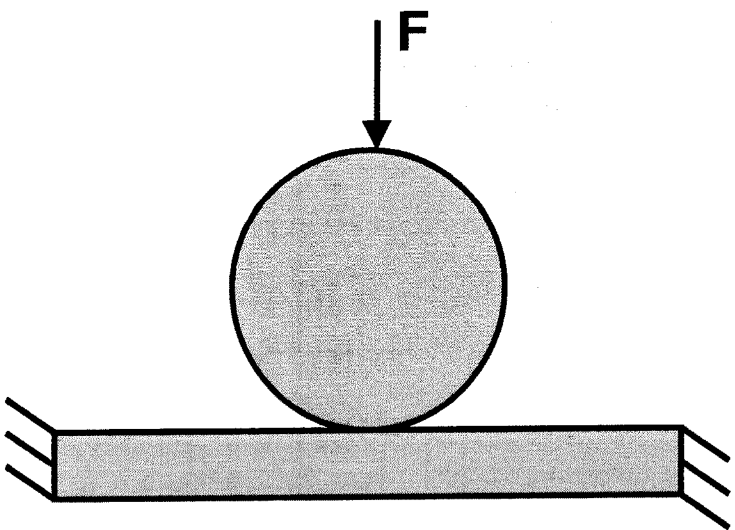
If at any point in the solution the two bodies are disconnected, the stiffness matrix will become singular. ANSYS will issue a *negative pivot* warning message. There are several options for overcoming rigid body modes due to initially unconnected bodies:

- Build the Geometry in the "Just-Touching" Position
- Dynamics
- Displacement Control
- Weak Springs
- Use No-Separation Contact (KEYOPT(12) discussed later)
- Adjusting Initial Contact Conditions

Rigid Body Modes

"Just Touching"

This requires you to know what the "just touching" position is. This can be difficult if the surfaces are curved or irregular.



Small gaps or penetrations can exist between the bodies due to numerical round-off in the finite element meshes. This can cause non-convergence or breaking away of the bodies in contact.

Rigid Body Modes

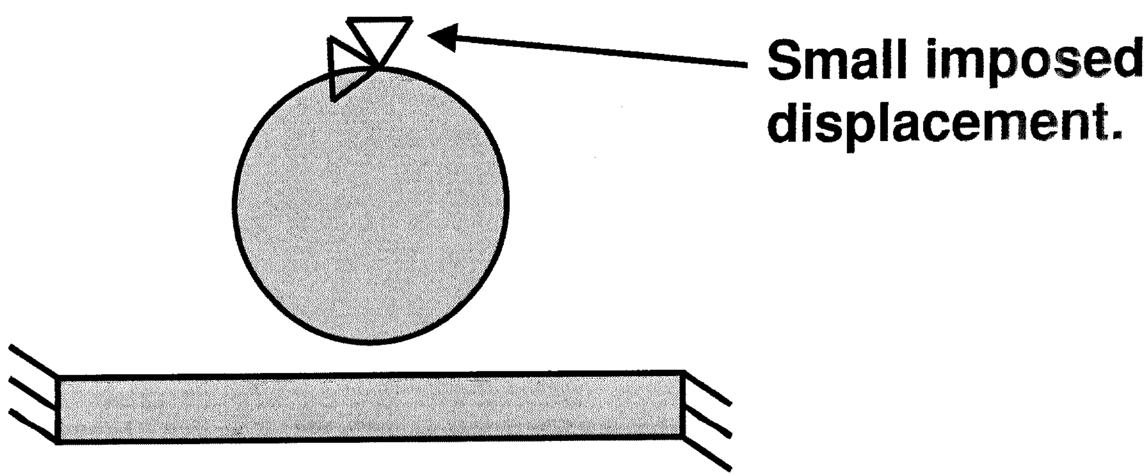
Dynamics

In a dynamic analysis inertial effects prevent a rigid body motion. One option to overcome rigid body motion is to solve the problem dynamically. You will need to add mass and damping in order to convert the solution from a static to a dynamic solution.

Rigid Body Modes

Displacement Control

This technique uses imposed displacements to move the two bodies into contact. The solution can then be switched from displacement control to force control through the use of a null load step.



Rigid Body Modes

Displacement Control

Load Step 1

Use a small imposed displacement to initialize contact.

Load Step 2

Switch from displacement control to force control. Delete the imposed displacements, apply the reaction forces, and solve in one substep. (This load step should converge in one or two iterations, there has been no change to the system.)

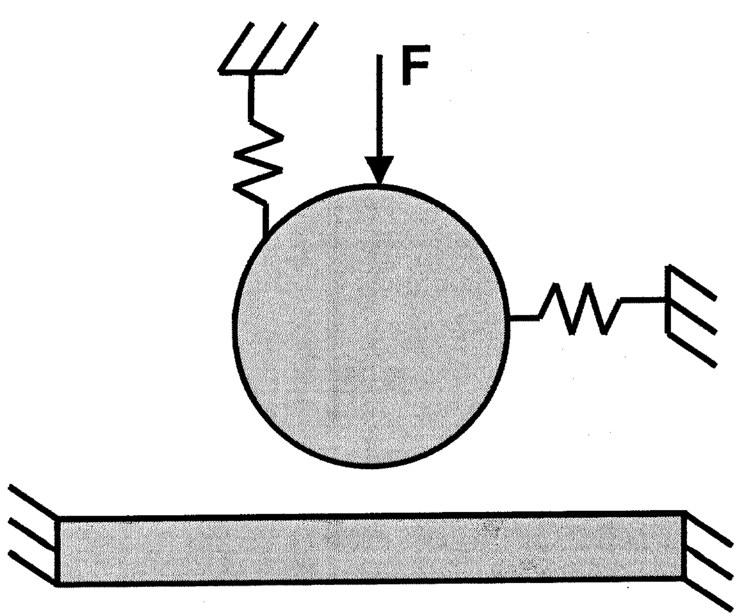
Load Step 3

Continue with the load history.

Rigid Body Modes

Weak Springs

This technique uses weak springs to ground to prevent the rigid body motion.



The spring stiffnesses should be negligible compared to the stiffness of the system. By connecting the springs to ground, the reactions at the grounded nodes can be compared to the total reaction forces to ensure that the springs have no effect on the solution.

Rigid Body Modes

- Although "just touching", dynamics, displacement control, and weak springs are all valid analysis techniques, they can be difficult to implement.
- "Just Touching" - Small gaps or penetrations can exist because of numerical round-off in the finite element mesh.
- Dynamics - Not always easy to damp out unwanted dynamic effects in a "static" model.
- Displacement Control - Not always obvious which displacements to impose for a complicated loading.

Rigid Body Modes

- Weak Springs - The initial load has to be small enough to cause a small deflection on the springs such that the contact elements recognize the penetration. Can require some experimentation as to not "pass through" the contact surfaces.

- Therefore, it is also possible to *adjust the initial contact conditions* to prevent rigid body modes using the real constants ICONT or PMIN and PMAX.

- ICONT moves the nodes on the contact surface, within the adjustment band, to the target surface. PMIN and PMAX will physically move the rigid target surface into the contact surface.

Rigid Body Modes

Adjusting the Initial Contact Conditions (ICONT)

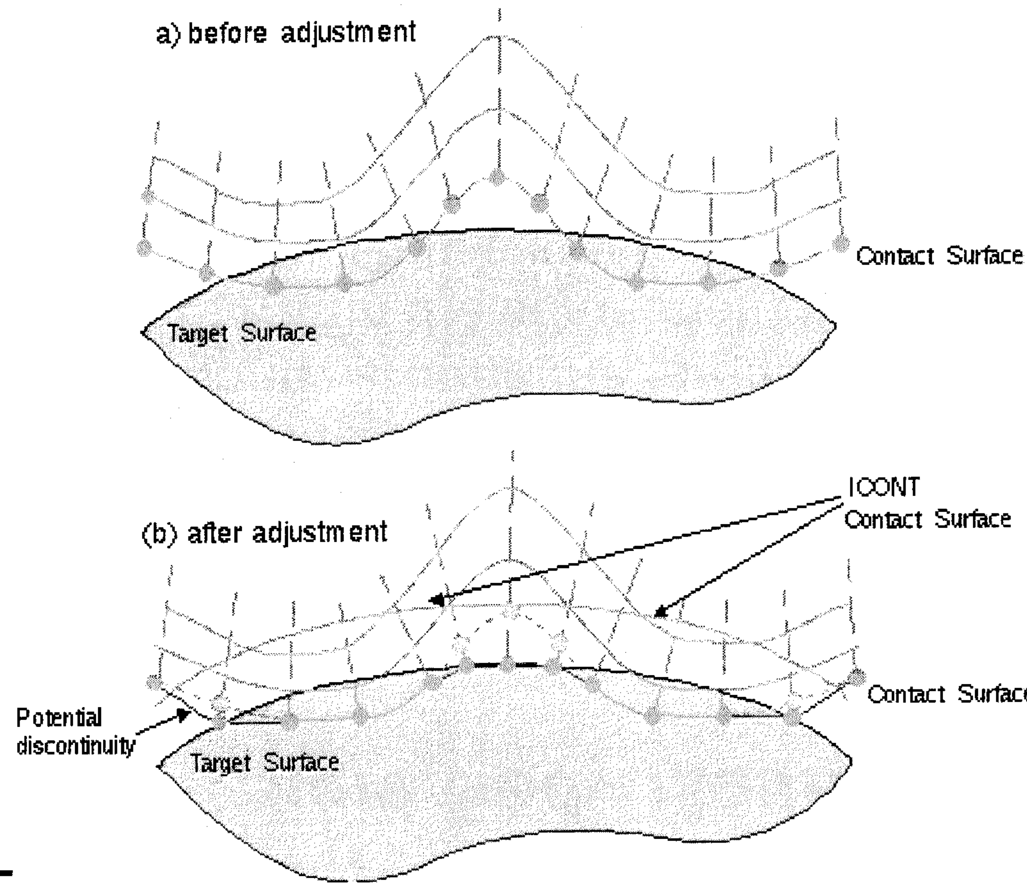
- The real constant ICONT can be used to specify an "adjustment band" around the target surface. Any contact points within the adjustment band are shifted to the target surface. Only a small correction is recommended, as discontinuities can occur as a result of too large a value of ICONT.

- *If the real constant ICONT is not specified ANSYS provides a small default value of ICONT according to the dimensions of the model.*

Rigid Body Modes

Adjusting the Initial Contact Conditions (ICONT)

Contact surface adjustment, shown before and after adjustment:



Rigid Body Modes

Adjusting the Initial Contact Conditions (PMIN and PMAX)

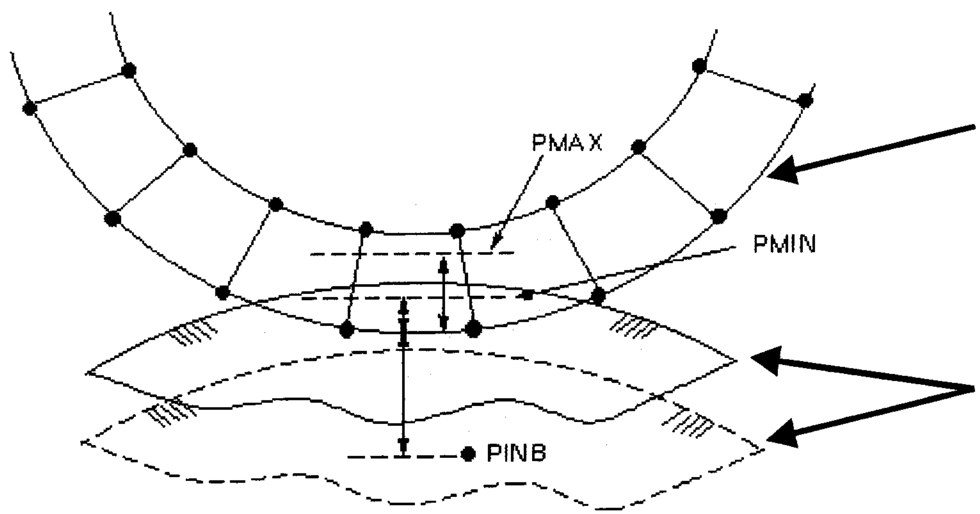
- The real constants PMIN and PMAX specify an initial penetration range. *ANSYS physically moves the entire target surface (and the attached deformable body) to be within the range of penetration specified by PMIN and PMAX.*

- If the target surface has prescribed constraints of zero, the initial adjustment using PMIN and PMAX will not be performed.

- The initial adjustment is an iterative process. ANSYS uses a maximum of 20 iterations to bring the target surface within the range of PMIN and PMAX.

Rigid Body Modes

Adjusting the Initial Contact Conditions (PMIN and PMAX)



Contact Surface

Target Surface before and after adjustment

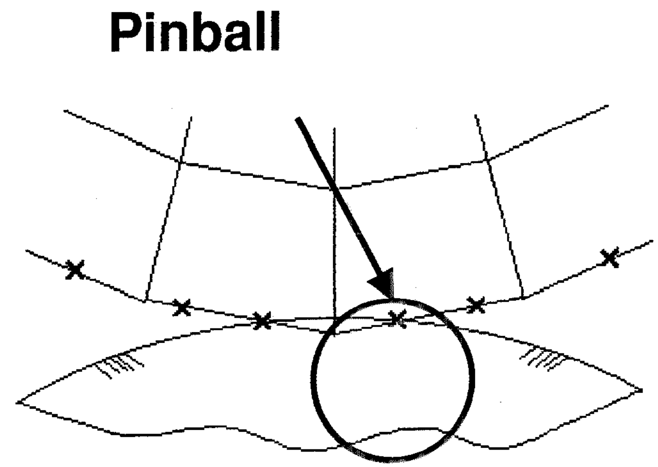
Pinball Region

The position and motion of a contact element relative to the target determines the contact status. ANSYS monitors each element and assigns it a status which can be postprocessed.

Contact Status	Relative CPU Cost
0 - Open Far-Field Contact	Inexpensive
1 - Open Near-Field Contact	Moderate
2 - In Contact Sliding	Expensive
3 - In Contact Sticking	Expensive

Pinball Region

A contact element is considered in near-field contact when it enters the pinball region. The pinball region is the circle (2-D) or sphere (3-D) which surrounds the contact element.



The real constant PINB can be used to adjust the pinball region (this may be necessary for a problem with a large initial penetration).

By default ANSYS defines the pinball region as 4×depth (rigid-to-flexible) or 2×depth (flexible-to-flexible) of the underlying finite element.

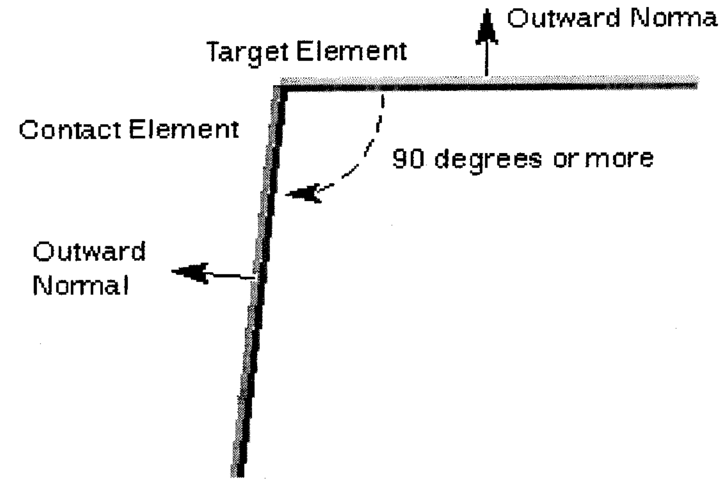
Prevention of Spurious Contact

KEYOPT(8)=1

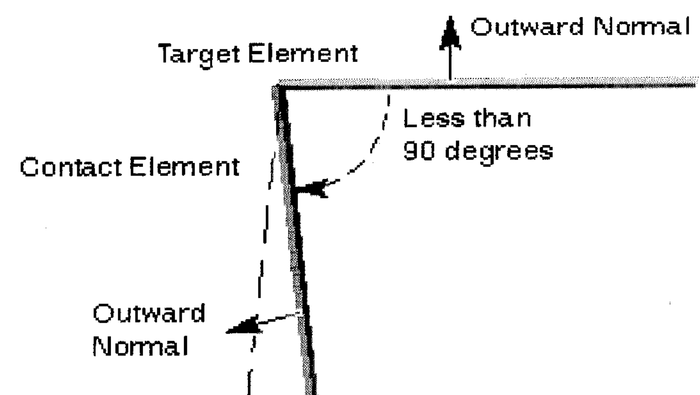
- **Only used if the real constant PINB can not handle the problem since this option may result in missing the true penetration.**

- **KEYOPT(8) prevents spurious contact from initial over-penetration, or over-penetration from deformation.**

- ***Use for symmetric contact including self-contact problems.***



No contact is detected



Contact is erroneously detected

ANSYS

Initial Interference

There are several techniques for modeling initial interference contact problems (such as shrink fit assemblies):

- Include an initial interference from the geometry.
- Include an initial interference from the real constant CNOF to specify the contact surface offset.
- Use a combination of an initial interference from the geometry and the real constant CNOF.

To overcome convergence difficulties caused by a large initial penetration, ANSYS allows for the initial penetration to be ramped over the first load step.

Initial Interference

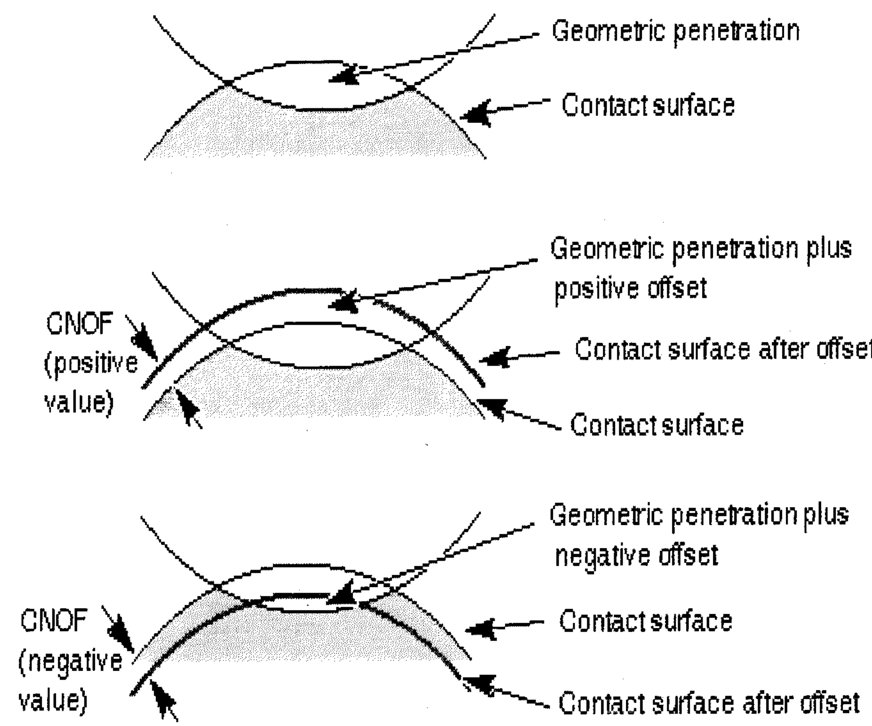
CNOF

The real constant CNOF is the contact surface offset.

+ CNOF increases interference.

- CNOF decreases interference or results in gap.

CNOF can be combined with a geometric penetration.



Initial Interference

<u>Initial Penetration Control KEYOPT(9)=</u>

0 - Include geometric penetration and CNOF.

1 - Ignore both geometric penetration and CNOF.

2 - Include both geometric penetration and CNOF. *Ramp the initial penetration on over the first load step.*

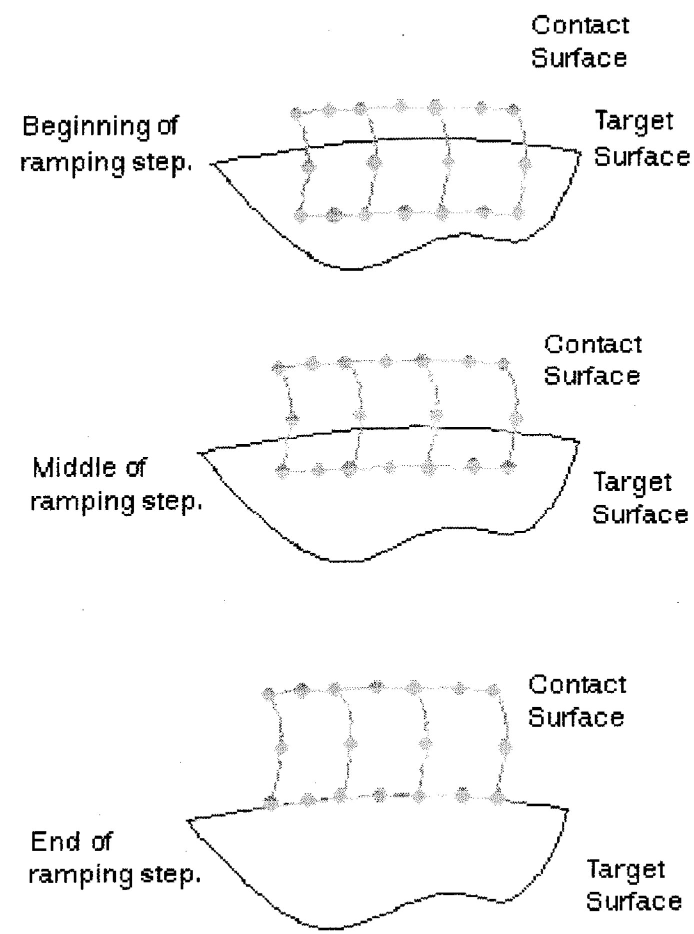
3 - Ignore the geometric penetration and include CNOF

4 - Ignore the geometric penetration and include CNOF. *Ramp the initial penetration over the first load step.*

Initial Interference

Ramping the Initial Interference

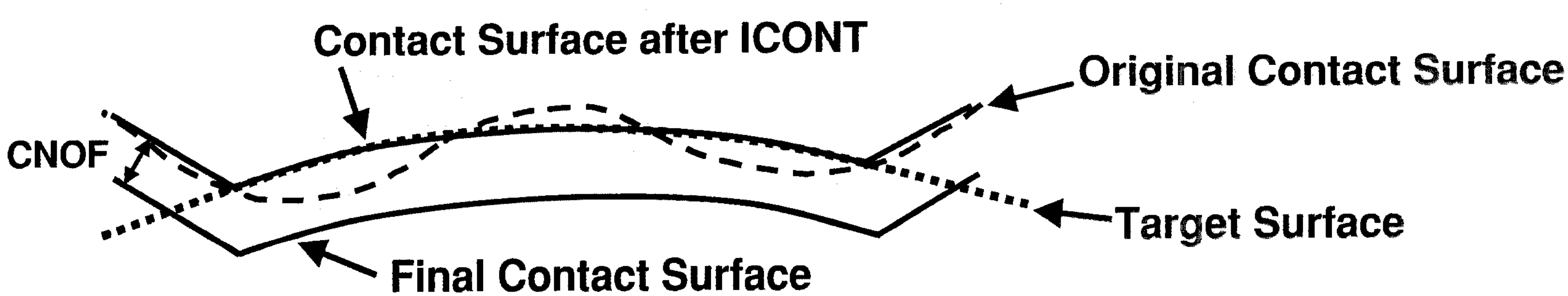
If you have a large initial interference, it usually can not be step applied. However, it can be ramped over the first load step with the KEYOPT(9) settings. (Make sure the pinball region is large enough!) For the ramped capabilities no other loads should be specified within the first load step.



Initial Interference

The techniques for adjusting the initial contact conditions and offsetting the contact surface can also be combined. For example, you may want a very precise initial penetration with an imported mesh and the initial coordinates of the nodes do not provide for sufficient precision. To accomplish this:

1. Use ICONT to move the contact points onto the target.
2. Use CNOF to specify an initial penetration.
3. Use KEYOPT(9) to resolve the penetration.



Surface Interaction Models

- KEYOPT(12) can be used to model different contact surface interactions:

0 - Standard unilateral contact behavior (default).

1 - Rough Contact: No sliding once contact is established, this corresponds to an infinite coefficient of friction.

2 - No Separation Contact: Target and contact surfaces are tied once contact is established (sliding is permitted).

3 - Bonded Contact: Target and contact surfaces are "glued" once contact is established.

Surface Interaction Models

- KEYOPT(12) description (surface interactions) continued:

4 - No Separation Contact "Always": Any contact detection points inside the pinball region or that come into contact are tied in the normal direction (sliding is permitted).

5 - Bonded Contact "Always": Any contact detection points inside the pinball region or that come into contact are bonded for the remainder of the analysis.

6 - Bonded Contact "Initial Contact": Bonds surfaces ONLY in initial contact, initially open surfaces will remain open. *Similar to CEINTF but can be used for nonlinear problems.*

Surface Interaction Models

Contact Opening Spring FKOP

- For modeling no separation or bonded contact, you may need to set the real constant FKOP. This real constant provides a stiffness value for open contact.
- For no separation contact the open gap stiffness factor can be used to prevent a rigid body motion due to a gap.
- For bonded contact FKOP prevents separation of the contact surfaces.
- FKOP defaults to 1.0 which models bonding. Use a small value (1e-5) to model a weak spring.

Accounting for Thickness Effects

If you have created a model using beam or shell elements, the contact surfaces can be shifted to account for beam or shell thickness. KEYOPT(11) includes or excludes beam and shell thickness effects.

0 - Contact at the midplane (default).

1 - Contact on the top or bottom surface as specified.

Note: When using Shell181 changes in thickness during deformation are also taken into account.

Time Step Control

Time step control is an automatic time stepping feature that predicts when the status of a contact element will change and cuts back the current time step. KEYOPT(7) controls time step prediction.

0 - No Control: Time step size is unaffected. This is usually sufficient for static problems when automatic time stepping is on.

1 - Auto Cut Back: The time step is bisected if the contact status changes dramatically. For dynamic problems auto cut back usually is sufficient.

2 - Reasonable: A more expensive algorithm than auto cut back.

3 - Minimum: This option predicts minimum time increment for the next substep (very computationally expensive, not recommended).

Trouble Shooting

Output File Information

The output file will print a contact summary, identifying all contact pairs in the model.

```
*** NOTE ***
Rigid-deformable contact pair identified by real constant set 1 has
been set up.  Please verify constraints on target nodes which may be
automatically fixed by ANSYS.
Contact stiffness factor FKN                    1.0000
Default penetration tolerance factor FTOLN     0.10000
Default initial closure ICONT will be used
Pinball region factor PINB                      1.0000
Default Max. friction stress TAUMAX            0.10000E+21
****************************************
```

Trouble Shooting

Output File Information

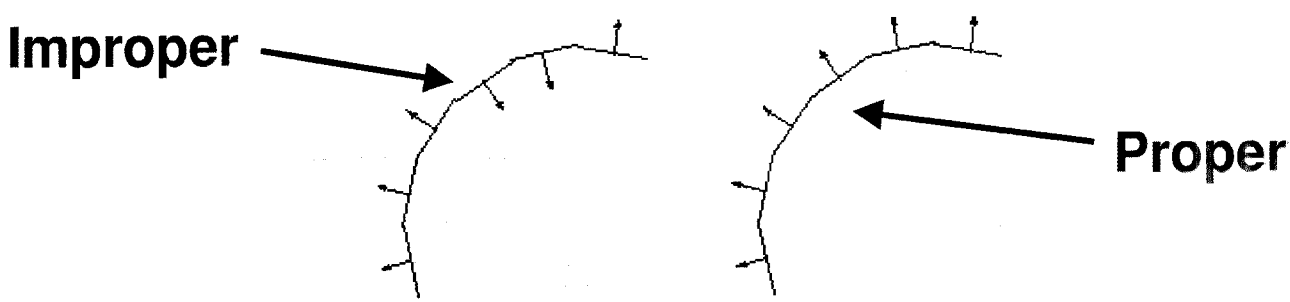
The output file will also echo the initial contact closure selected for each contact pair.

```
*** NOTE ***
Min.  Initial gap 3.914707351E-02 was detected between contact
element 603 and target element 1 specified by real constant set 1.

*** NOTE ***
You may move entire target surface specified by real constant set 1:
x= 3.902639615E-02, y= -3.071443974E-03, z= 0,to bring it in contact.

*** WARNING ***
An initial contact closure factor (ICONT) 3.E-02 has been selected
for real constant set 1.
```

Trouble Shooting

- Rigid-to-Flexible and Flexible-to-Flexible contact can be defined in the same model. Be sure to use separate real constants to define the contact pairs. The output file information can be helpful to check your contact pairs.

- Be sure to check the outward normal direction of the contact and target elements. Contact occurs on the positive outward normal side of the contact and target elements.

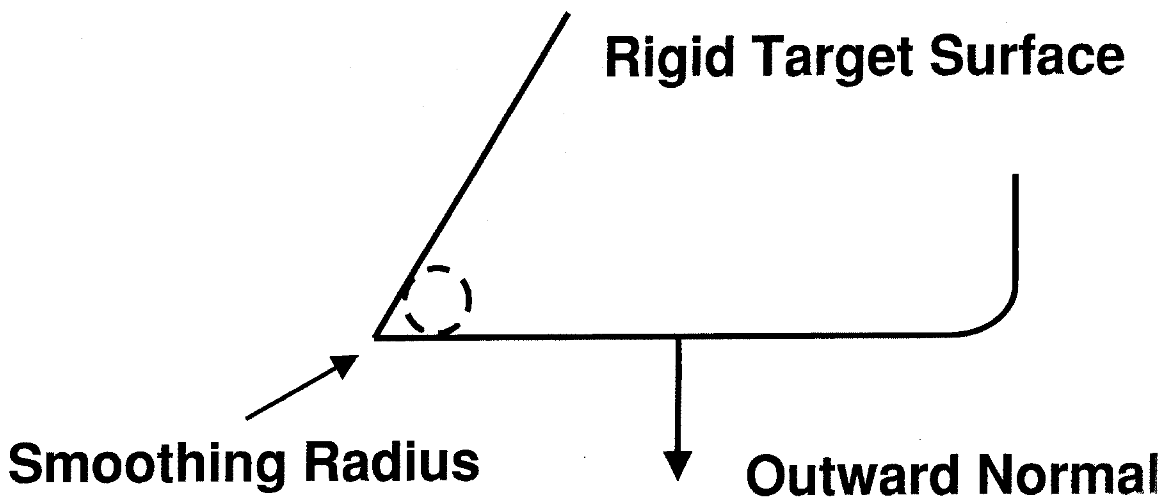


Trouble Shooting

- Ensure that the target and contact surface definitions extend far enough to cover the full expected range of motion for the analysis.
- Be sure to adequately discretize a rigid target surface. Excessively coarse discretization can cause convergence difficulties.
- Ensure that the appropriate surface pairs are initially in contact with either ICONT or PMIN and PMAX to prevent rigid body motions.

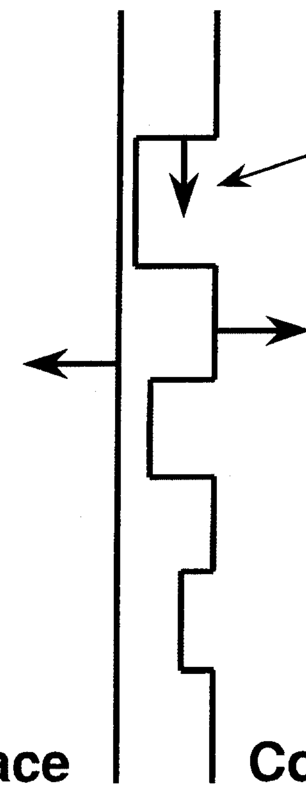
Trouble Shooting

Sharp convex corners can cause convergence difficulties in large sliding problems. Use a line or an area fillet to smooth a sharp convex corner.



Trouble Shooting

Interference fit problems with widely varying contact normal directions can result in non-physical behavior. In this case try switching the target and contact surfaces. Or, try removing problem contact elements.



Trouble Shooting

- Reset solution options, number of substeps, number of equilibrium iterations, unsymmetric solver, etc.

- And finally, FKN and FTOLN need to be set appropriately:

  - FKN usually will be between 0.01 and 10. Use a value of 1.0 (default) for bulk deformation problems, and 0.1 for bending dominated problems. Do *NOT* set FTOLN too small, too tight a penetration tolerance can result in divergence.

Chapter 9

ELEMENT BIRTH and DEATH

Session Objective

- At the end of this session you will be able to describe and demonstrate the following:

  1. Definition of Element Birth and Death

  2. Element Birth and Death Applications

  2. Implementation of Element Birth and Death in ANSYS

  3. Element Birth and Death Procedures

  4. Using ANSYS Results to Control Birth and Death

  5. Trouble Shooting

Element Birth and Death Definition

- If material is added to or removed from your structure at some time during its load history, you might want to be able to make certain elements in your model become "nonexistent" or "existent."
- In such cases, you can use the element birth and death options to deactivate or reactivate selected elements at specified times (load steps) in your model's load history.
- Element birth and death is defined as a changing status nonlinearity. (Similar to a contact problem.)

Birth and Death Applications

- Potential applications include:
  - Excavation (as in tunneling or slurry wall excavation)
  - Staged construction (as in un-shored bridge erection)
  - Sequential assembly (as in fabrication of layered computer chips, epoxy curing)
  - Weld deposition
  - Annealing
  - Other applications in which activated or deactivated elements can be identified by their known locations

Birth and Death Implementation

- Elements are deactivated (or reactivated) in the first substep of a load step, and maintain that status for the rest of the load step.
- Killed elements are not actually removed; they are deactivated.
  - Deactivated elements have their stiffnesses multiplied by a severe reduction factor.
  - Element loads (such as pressure and temperature) associated with deactivated elements are zeroed out of the load vector.
  - Similarly, mass, damping, and stress stiffness effects are set to zero for deactivated elements.
  - An element's strains are also set to zero as soon as that element is killed. (This behavior allows you to model annealing.)

Birth and Death Implementation

- Born elements are not actually added; they are reactivated.
  - All elements, including those to be born in later stages of your analysis, must be created initially in the preprocessor.
  - When an element is reactivated, its stiffness, mass, damping, and element loads return to their full original values.
  - Elements are reactivated having no record of strain history. (They are "annealed" by birth and death operations, being born with all strains and all stresses equal to zero.)

Birth and Death Procedures

- As with any other analysis, the procedure for element birth and death consists of three main steps:
  - Build the model
  - Obtain the solution
  - Review the results

Build the Model

- When you build your model in the preprocessor (PREP7), you need to create all the elements at the start of the analysis - even those that won't be activated until later load steps.
- Not all elements support birth and death. You can deactivate and reactivate only those elements that have the birth and death capability. Refer to the ANSYS Advanced Analysis Techniques for a complete listing of the element types which support birth and death.

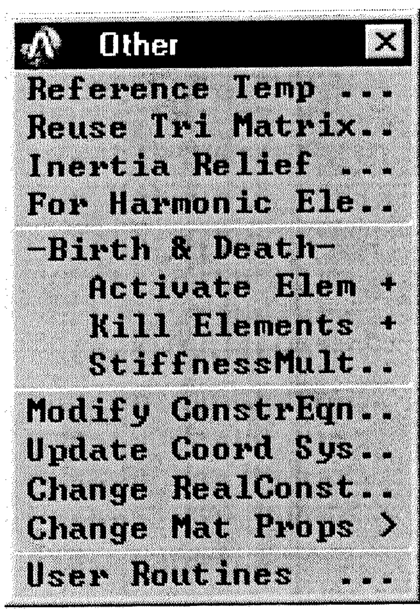
Obtain the Solution

- The birth and death load step options deactivate or reactivate elements within a given load step.
- Recommended Analysis options:
  - Large deflection effects should be activated by turning on nonlinear geometry (NLGEOM,ON).
  - Birth and death does not activate a default Newton-Raphson option. The full Newton-Raphson is the recommended option.

Obtain the Solution

- **Deactivating and reactivating the elements within a load step is performed using the menu path:**

Solution > Load Step Opts > Other > Birth & Death >



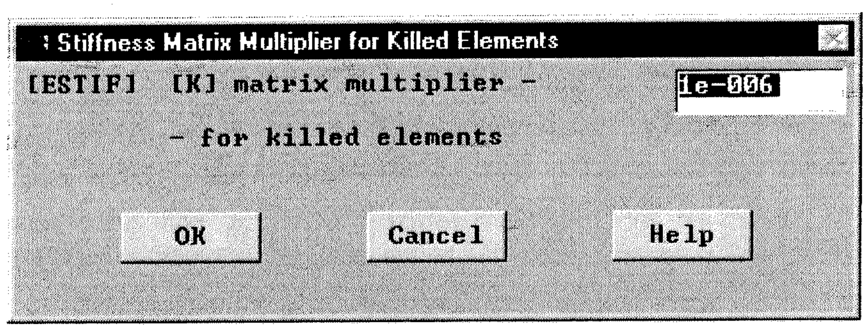
Birth and Death Options

Note: Those elements that are to be "added" in later load steps should be deactivated in the first load step.

Obtaining the Solution

- **If the default stiffness reduction factor of 1.0E-6 is not appropriate for your analysis, you can specify a different value (usually smaller):**

Solution > Load Step Opts > Other > Birth & Death > Stiffness Multiplier >



Obtaining the Solution

- Nodes that are not connected to any alive elements may "float". In some cases, you might want to constrain inactive DOFs to reduce the number of equations to be solved or to avoid ill-conditioning.
- Constraining inactive DOFs is important if you want to maintain a specific shape in elements when they are reactivated. Be sure to delete such artificial constraints when you reactivate elements.
- Note that constraint equations (CE or CEINTF) cannot be applied to inactive DOFs.

Obtaining the Solution

- Element loads are automatically zeroed out of the load vector for deactivated elements, and are automatically restored for reactivated elements. Mass is similarly zeroed out of the mass matrix. (Acceleration loads do not effect deactivated elements.)
- Concentrated forces are not automatically removed from inactive DOFs (i.e., from nodes that are not attached to any active elements). Be sure to delete such concentrated loads from inactive nodes.

Obtaining the Solution

Note on LSWRITE and LSSOLVE

The LSWRITE command cannot be used with the birth-death option. Multiple load steps need to be performed using a series of explicit SOLVE commands.

Obtaining the Solution Sample Input

```
NLGEOM,ON      ! Turn on large-deflection effects
NROPT,FULL     ! You must explicitly set the Newton-Raphson option
ESTIF,...      ! Sets non-default reduction factor (optional)
ESEL,...       ! Selects elements to be deactivated in this load step
EKILL,...      ! Deactivates selected elements
ESEL,S,LIVE    ! Selects all active elements
NSLE,S         ! Selects all active nodes
NSEL,INVE      ! Selects all inactive nodes
D,ALL,ALL,0    ! Constrains all inactive DOFs (optional)
NSEL,ALL       ! Selects ALL nodes
ESEL,ALL       ! Selects ALL elements
D,...          ! Adds constraints as appropriate
F,...          ! Adds nodal loads to active DOFs as appropriate
SAVE           ! Saves the database
SOLVE          ! Solve the problem
```

Note on Changing Material Properties in Solution

- If the strain history of the killed elements needs to be maintained, you can deactivate elements by changing their material properties in solution:

  Solution > Load Step Opts > Other > Change Mat Props

- However, this option does not remove element forces, strains, mass, specific heat, etc. Convergence problems can result from careless use of changing material properties in solution. For example if an element's stiffness was reduced to zero, but it retained its mass, a singularity could result from an acceleration load.

Review the Results

- For the most part, postprocessing a birth and death analysis follows standard procedures.
- Realize that "killed" elements are still present in your model, and will be included in element displays, output listings, etc. Use selecting to remove deactivated elements from element displays and other postprocessing operations.
- If you need to postprocess different load steps, make sure that the database is saved with the birth and death status for all elements that matches the birth and death status for that load step. (Save a copy of the database for each load step which changes the birth and death status.)

Using ANSYS Results to Control Birth and Death

- In certain problems you will need to activate or deactivate elements based on their results. For example if you wanted to kill melted elements in a thermal analysis, the elements could be identified on the basis of their calculated temperatures.
- The elements can be identified by storing results in the element table and selecting the critical elements via the element table results. The solution can then be restarted with the critical elements deactivated. Refer to the ANSYS Advanced Analysis Techniques for more detail.

Using ANSYS Results to Control Birth and Death

Sample Input for Deactivating Elements Based on Results

```
...                 ! Previous Solution Procedure
/POST1              ! Enter POST1
SET,...             ! Read in Results
ETABLE,...          ! Store criteria in ETABLE
ESEL,S,...          ! Select elements based on ETABLE item
FINISH
!
/SOLU               ! Re-enter SOLUTION
ANTYPE,,REST        ! Restart the solution
EKILL,ALL           ! Deactivate selected elements
ESEL,ALL            ! Restore full element set
...                 ! Continue with solution
```

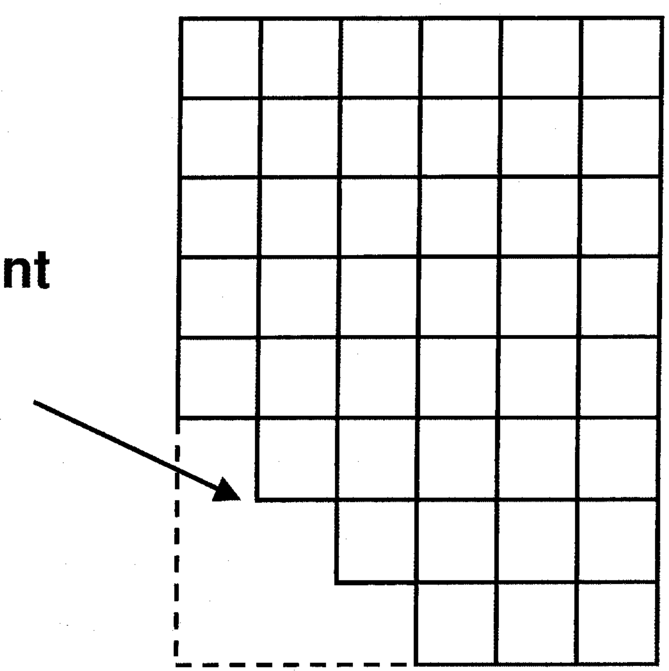
Trouble Shooting

- Deactivating and reactivating elements causes very sudden changes in your model's stiffness. If too severe, these changes can sometimes cause convergence difficulties. You may have to limit the number of elements that are deactivated or reactivated in a given load step.
- Since a large stiffness reduction during the iteration process can cause a discontinuity to occur, use the full Newton-Raphson procedure. Use of the line search as a convergence tool may also be helpful.

Trouble Shooting

- Be careful not to deactivate or reactivate elements in such a way as to create singularities in the model such as sharp re-entrant corners. This can cause convergence difficulties.

Jagged edges (re-entrant corners) should be avoided when deactivating elements.



Trouble Shooting

- Save the database with the element birth and death information for each load step which changes the birth and death status. Use the appropriate database when postprocessing individual load steps.
- When postprocessing be sure to select only the elements which are active. Including "killed" elements in contour displays will smear the results when viewing derived (stresses and strains) nodal quantities.

Chapter 10

OTHER NONLINEAR CAPABILITIES

Session Objective

- At the end of this session you will be aware of the following additional nonlinear capabilities of the ANSYS program:

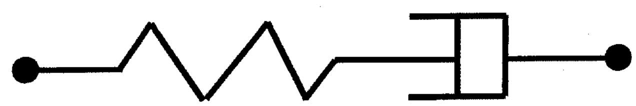
  1. Other Material Nonlinearities
     - Viscoelasticity, Creep, Nonlinear Elasticity, Concrete
  2. Other Changing Status Nonlinearities
     - Control Elements, Nonlinear Springs, Tension and/or Compression Only Spar, Revolute Joints, Membrane Shell with Cloth Option
  3. User Programmable Features

Other Nonlinear Capabilities

- Given the time frame of the training course, it is not possible to cover the entire breadth of ANSYS' nonlinear capabilities.
- This section is presented to highlight "other" nonlinear capabilities not covered in this seminar. Please refer to the ANSYS Analysis Guides for more information on any of the topics presented in this section.

Viscoelasticity

- Viscoelasticity is a nonlinear material that has both an elastic (recoverable) part of the deformation as well as a viscous (non-recoverable) part. Upon application of the load, the elastic deformation is instantaneous while the viscous part occurs over time.



A simple representation of a viscoelastic model is a spring and a damper in series.

- The viscoelastic material model is used to characterize the behavior of glass or glass-like materials. The effects of temperature changes are also included in this model, so that you can simulate heating and cooling sequences.

Viscoelasticity

- The viscoelastic material model is available with viscoelastic elements VISCO88 and VISCO89.
- VISCO88 is a 2-D, 8-node solid element with plane strain or axisymmetric analysis options.
- VISCO89 is a 3-D, 20-node solid element whose mid-side nodes may be removed.

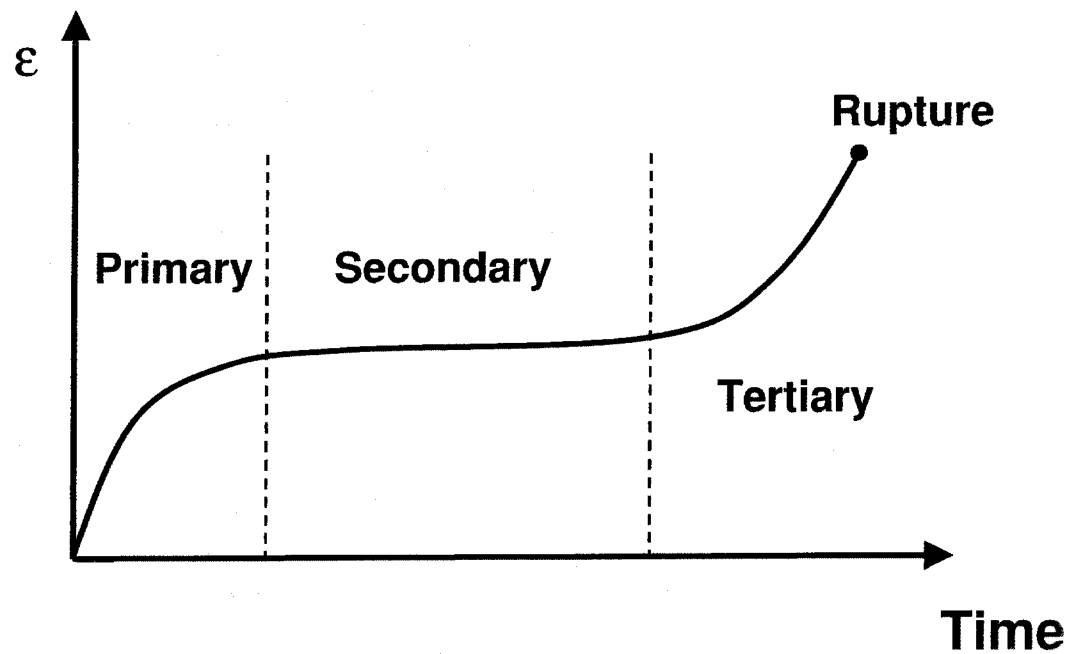
Creep

- Creep is an irreversible strain that continues to increase over time under constant load. Most metals exhibit creep at elevated temperatures, although some materials such as lead, concrete and some plastics creep significantly at room temperature.

- A component under constant load will continue to deform if the material exhibits creep behavior. Similarly, if the component is subject to a constant displacement, the stresses will decrease over time. This is sometimes called stress relaxation.

Creep

A typical uniaxial creep curve has three stages:



Creep

- In the *primary stage* the strain rate decreases with time. This stage usually occurs over a relatively short time period. The *secondary stage* has a constant strain rate associated with it, so that the creep strains develop at a constant rate. In the *tertiary stage*, the strain rate increases very rapidly until the material fails.
- The creep strain rate may be a function of stress, strain, temperature, time, or other variables.
- The primary and secondary stages are usually of the most interest. ANSYS can model the primary and secondary stages of creep.

Creep

- Be sure to use an element type which supports creep. A library of general creep strain rate equations (documented in the ANSYS *Elements Manual*) is available for use.
- Also, a number of specific creep laws are available that are used for specific materials, in particular those materials used in the nuclear power industry (types 304 and 316 stainless steels and 2 1/4 Cr-1 Mo low alloy steel).

Concrete

- The concrete material model characterizes the failure of brittle materials. Both cracking and crushing failure modes are included in the concrete material model.
- This material model is applicable to concrete, rock, and other materials which have low tensile strengths and high compressive load carrying capability.
- The concrete material model is only available with the concrete element SOLID65. This element can model concrete with reinforcing "smeared" through the element. Link elements can be used to model discrete concrete reinforcing.

Concrete

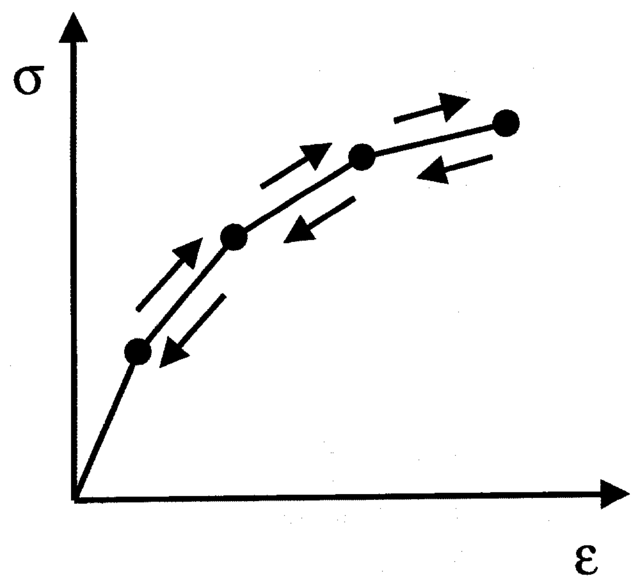
- The concrete material model is capable of cracking (in three orthogonal directions), crushing, plastic deformation, and creep.

- A crack may develop in one plane and if subsequent stresses tangential to the crack face are large enough, a second (or third) crack will develop. A crack is treated as a smeared band of cracks and not as a discrete crack.

- Crushing failure assumes complete deterioration of the structural integrity of the material, such as material spalling.

Nonlinear Elasticity

- The nonlinear elastic material option allows a multi-linear representation of an *elastic* stress-strain response:



The material will load and unload along the same path, so that no permanent deformation occurs. (The nonlinear elasticity option is only valid up to about 10% strain.)

Other Nonlinear Elements

There are other changing status nonlinear structural elements:

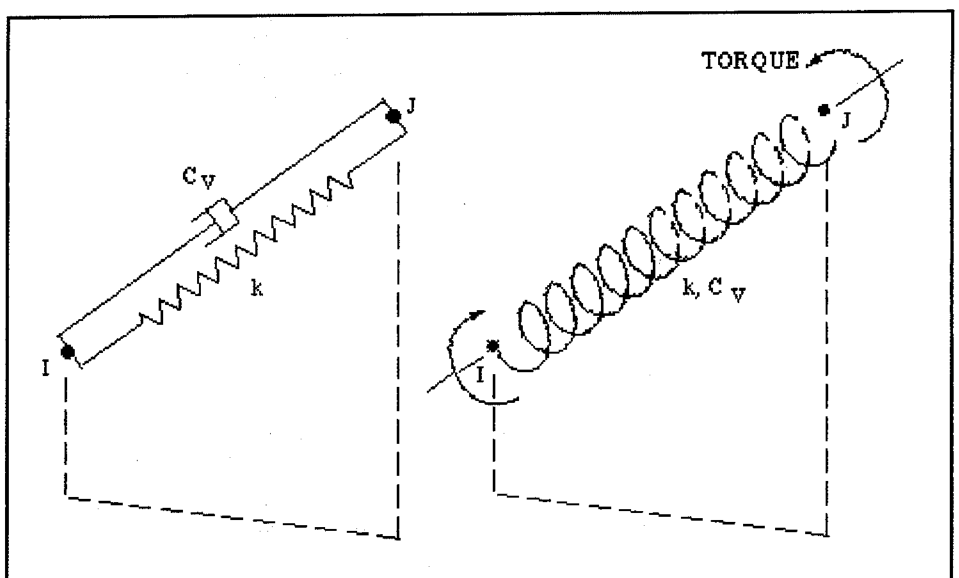
COMBIN14	Nonlinear Spring-Damper
COMBIN39	Nonlinear Spring
COMBIN40	Combination Nonlinear Spring-Linear Damper-Gap (Contact)
LINK10	Tension-Only or Compression-Only Spar
SHELL41	Shell with Cloth Option
COMBIN37	Control Element
COMBIN7	3-D Revolute Joint
LINK11	Linear Actuator

These elements are formulated to allow some type of nonlinear behavior. Their nonlinear characteristics are generally defined by element options and/or real constants.

COMBIN14 Spring-Damper

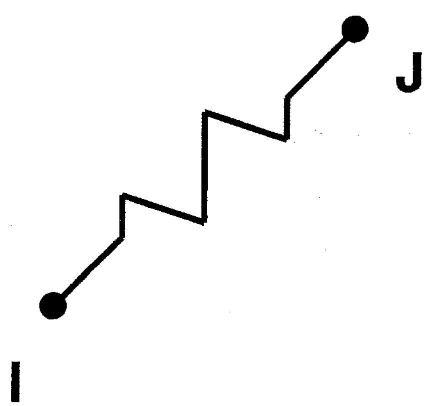
This element is a longitudinal or torsional spring-damper. Damping effects can be included in a transient analysis.

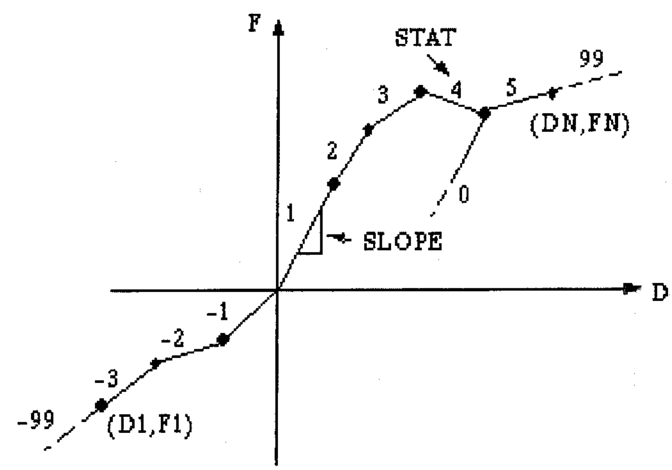
The element damping can be a nonlinear function of nodal velocity.



COMBIN39 Nonlinear Spring

This element is a longitudinal or torsional spring element which allows for a multilinear spring stiffness.

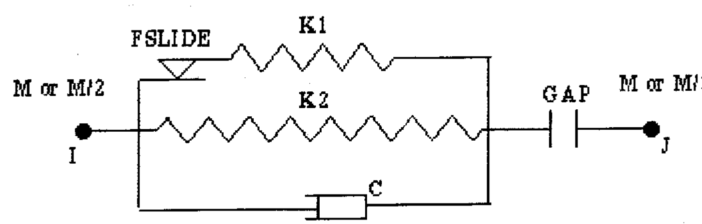




COMBIN39 has an option for conservative (same path) or nonconservative (parallel path) unloading behavior.

COMBIN40 Spring-Slider-Damper-Gap Element

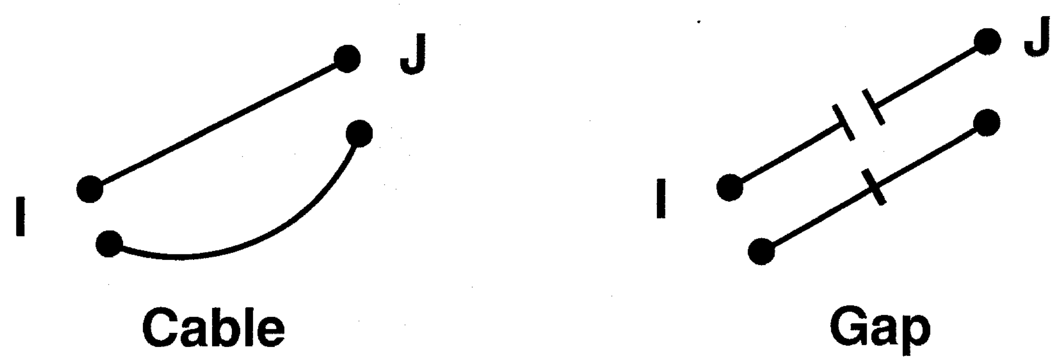
This element can model a longitudinal or torsional bilinear spring with contact (gap) behavior. The element can also model a linear damper or mass for a transient analysis.



The element can also model sliding behavior if FSLIDE $\neq$ 0.

LINK10 Tension-Only or Compression-Only Spar

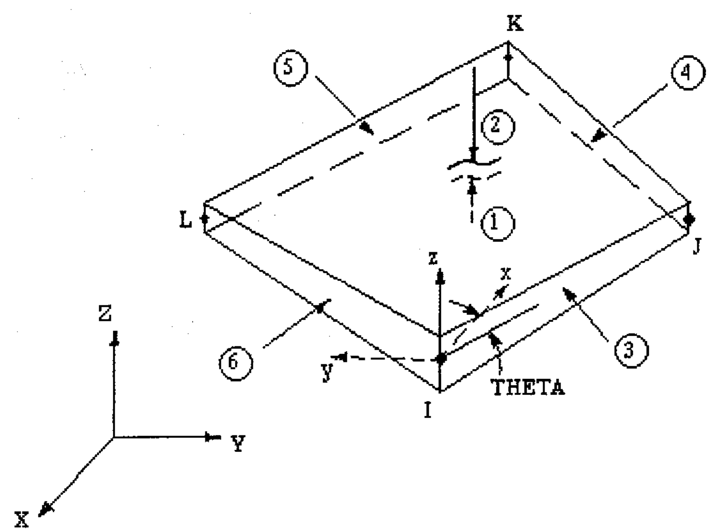
This 3-D bilinear element can be used either as a "cable" (tension-only) or as a gap (compression-only).



The status is either taut or fully slack for the cable option, or compressed or open for the gap option. An initial strain can be specified as a real constant to model pretensioning.

SHELL41 Membrane Shell with Cloth Option

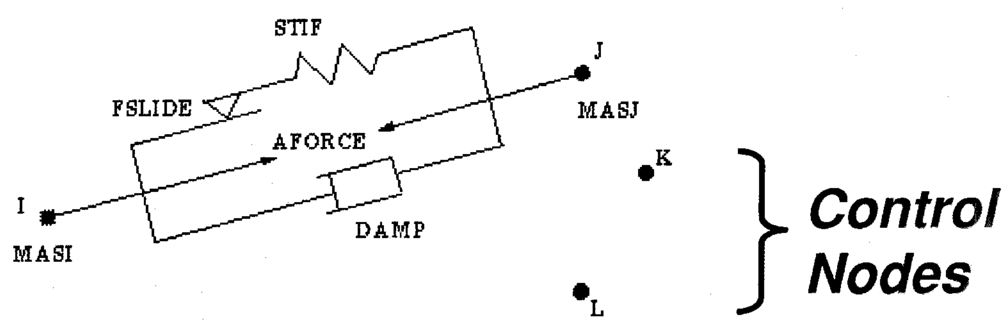
This element is a 3-D membrane shell (no bending stiffness) that can collapse, or wrinkle, in compression. Wrinkling can occur in one (or both) orthogonal directions.



Note that any warping of this element can cause an instability in the solution. To counteract this, a slight normal stiffness can be added to the the element with a real constant.

COMBIN37 Control Element

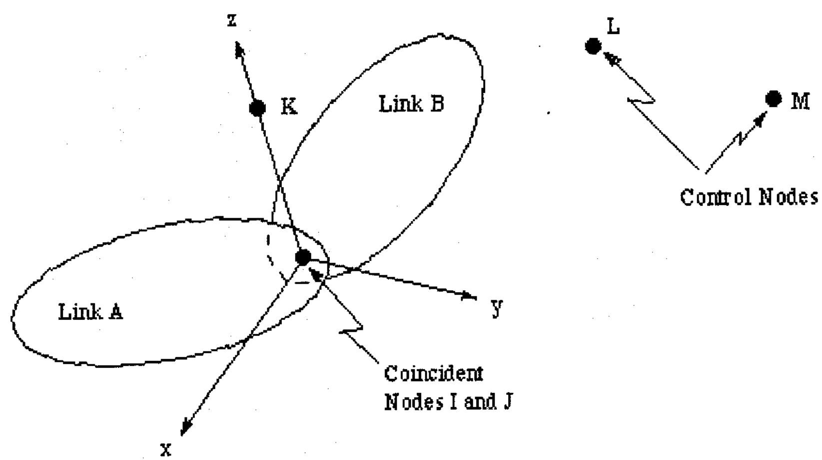
The control element is a versatile element that can be used to remotely control portions of a structure. This element provides mass, damper, and slider (nonlinear spring) capabilities.



Depending of the status of the control nodes, the element is either on or off. Applications for this element include, but are not limited to, mechanical snubbers, friction clutches, thermostats, relief valves, electrical switches, etc.

COMBIN7 3-D Pin Joint

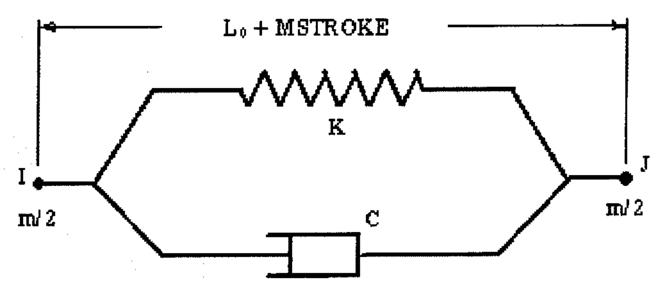
This element can represent a pin joint between two bodies that may undergo large rotations in 3-D space.



This element is useful in kinematic problems where large motion is occurring. Joint flexibility, friction, damping and rotation limits (stops) may be incorporated in the element. Additional control nodes are also available.

LINK11 Linear Actuator

LINK11 may be used to model hydraulic cylinders undergoing large rotations. This element is a spar with element damping, which can also undergo substantial prescribed length changes.



The element initial length L_0 and orientation are determined from the node locations. The measured stroke is equal to the applied stroke plus the axial deformation.

User Programmable Features

- User programmable features (UPFs) are ANSYS capabilities for which you can write your own FORTRAN routines. UPFs allow you to customize the ANSYS program to suit your needs. For nonlinear analyses this may include a user defined material option, or a user defined element.

- If you are considering writing a UPF, recommended reading is the *ANSYS Programmer's Manual.* This document contains information on the ANSYS file structures, and sample user subroutines.

User Defined Elements

- A user defined element can be added to the ANSYS element library and used as a "regular" element. You can create up to six independent element types (USER100- USER105).

- Example user defined element subroutines are included in the *ANSYS Programmer's Manual.*

User Defined Material

If you need to simulate a material behavior that is not an ANSYS program option, then you may want to consider programming the material law.

A number of user programmable features (UPFs) are offered for characterizing different types of material behavior. The available user material routines include:

- Plasticity/Viscoplasticity
- Hyperelasticity
- Creep
- Swelling

ANSYS

User Defined Plasticity/Viscoplasticity

You may define your own plasticity model as a user subroutine. The plasticity model may be either rate independent or rate dependent (viscoplastic).

Two user subroutines are available for user defined plasticity:

- USERVP is used by VISCO106, 107, and 108.
- USERPL is used by the remaining elements which support plasticity.

User Defined Hyperelasticity

- You may define your own strain energy density function (W) for the incompressible hyperelastic elements by using the subroutine USRHYP.

- You access the user defined hyperelastic routine by setting KEYOPT(7)=1 in either HYPER56, 58, 74 or 158 and activating a hyperelastic data table.

- Any user defined material constants may be entered by editing the hyperelastic data table.

User Defined Creep

- You may define your own creep strain rate equation by using the subroutine USERCR. The creep strain can be a function of a number of solution variables, stress, strain, temperature, etc.

- You may access this routine by activating a creep data table and setting $C_6 = 100$ when editing the table. Any material constants (other than C_6) may also be entered by editing the active creep data table.

User Defined Swelling

- Certain materials respond to neutron flux by enlarging volumetrically or swelling. Swelling strain can also be related to other phenomenon, such as moisture content.
- To access the swelling capabilities you *must* define your own swelling strain relation using subroutine USERSW. Swelling strain rate may be defined as a function of temperature, time, neutron flux, and/or stress.
- The user defined swelling material constants can be entered into a data table. Refer to the *ANSYS Elements Manual* for more information.

A Note on User Programmable Features

Programming a user programmable feature requires expertise in a number of areas:

- **A thorough understanding of the theory**
- **Good numerical methods skills**
- **Good programming skills**

You (and the eventual users of your UPF) are *responsible for validating and verifying* the usage and results of any user programmed feature.

NOTES

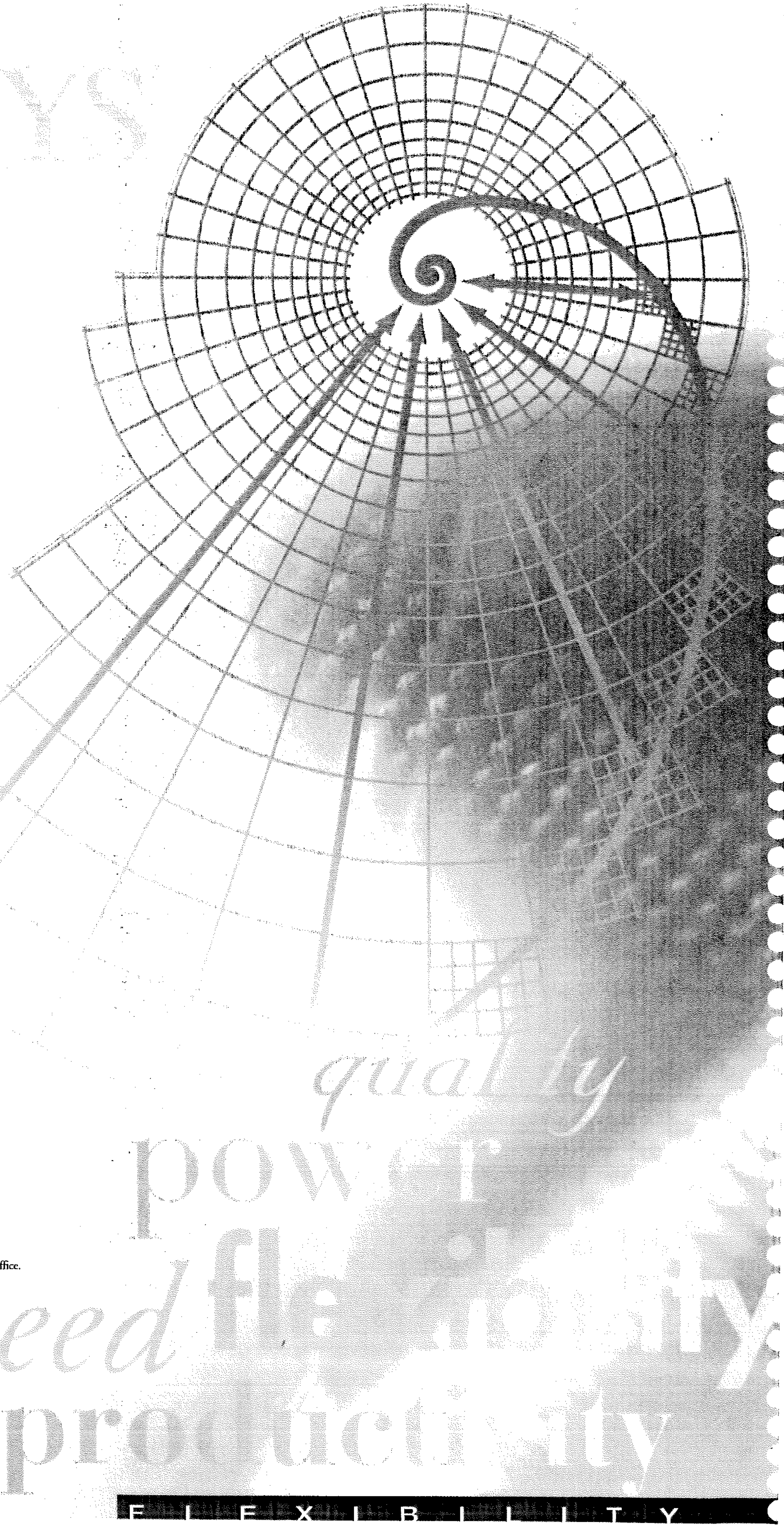
ANSYS

Southpointe
275 Technology Drive
Canonsburg, PA 15317

ansysinfo@ansys.com
T 724.746.3304
F 724.514.9494
Toll Free USA and Canada:
1.800.WE.R.FEA.1
Toll Free Mexico:
95.800.9373321

North America
len.zera@ansys.com
T 248.585.5020
F 248.585.5730
International
jim.tung@ansys.com
T 724.514.3086
F 724.514.3115
Europe
brian.butcher@ansys.com
T 44.118.9880229
F 44.118.9880925

http://www.ansys.com



001156