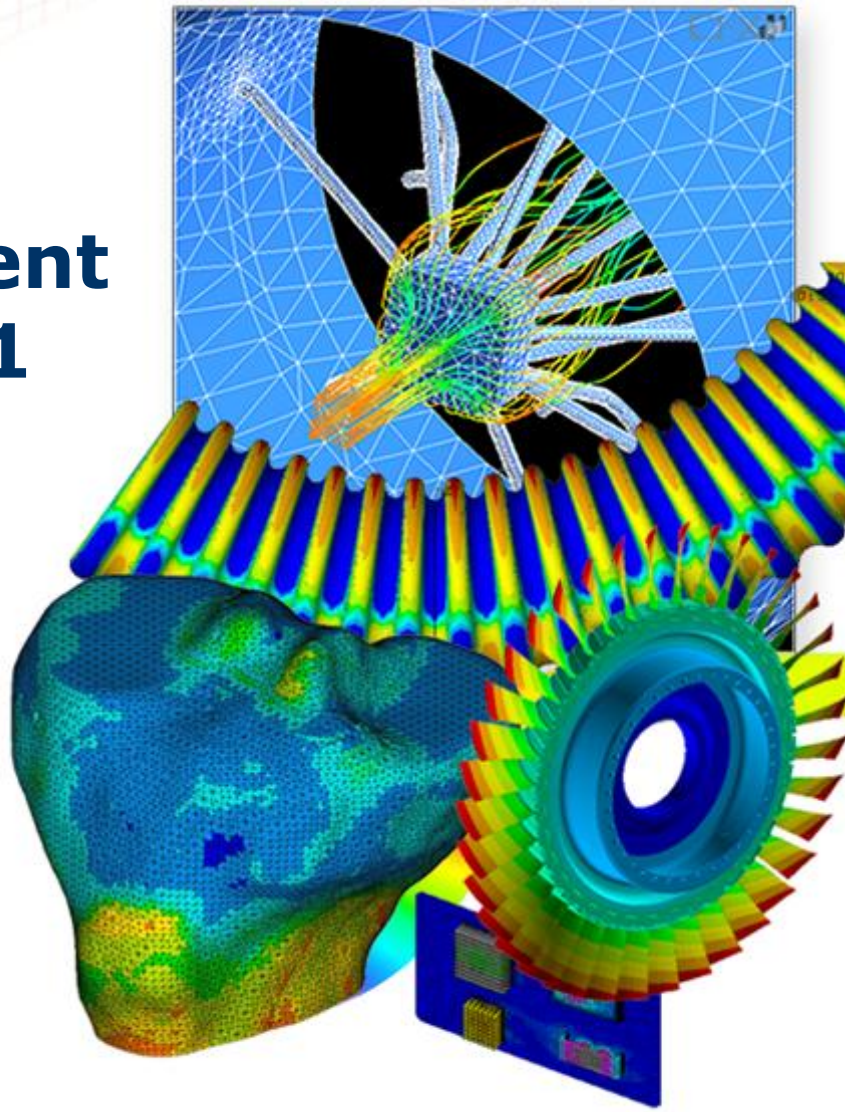
CAE ASSOCIATES

ENGINEERING ADVANTAGE

Initial State Enhancement in ANSYS Version 12.1

January 18, 2010

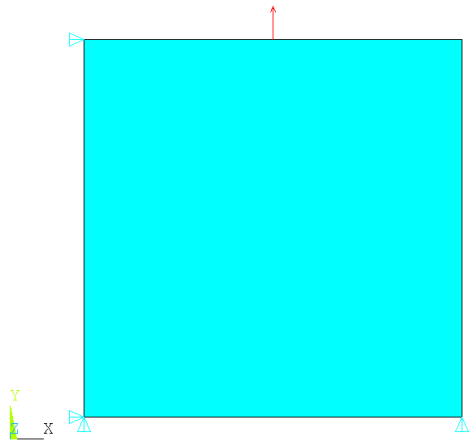


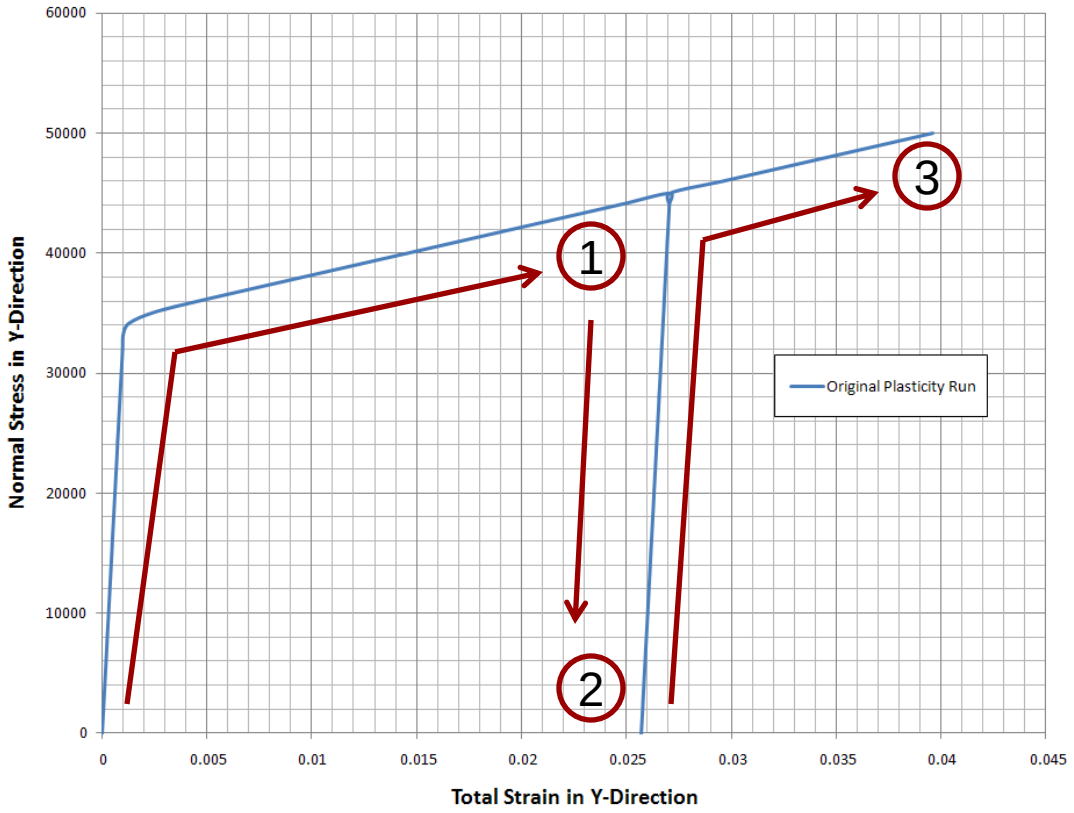
Initial State

- The term *initial state* refers to the state of a structure at the start of an analysis.
  - By default the initial state in ANSYS is an undeformed, unstressed structure.
- Previous versions of ANSYS were limited to defining an elastic stress state.
- The current version 12.1 of ANSYS has added the capability to define initial strain, initial plastic strain, and initial stress.
- This new enhancement provides a valuable capability to incorporate a residual stress/strain state from previous loadings such as manufacturing processes.

One Element Plane Strain Example

- To illustrate the new initial state procedure, consider a one-element plane strain model with BKIN plasticity:
  1. Load applied beyond the yield stress, accumulating plastic strain.
  2. Load removed, defining a residual stress state.
  3. Additional load applied, accumulating additional plastic strain.





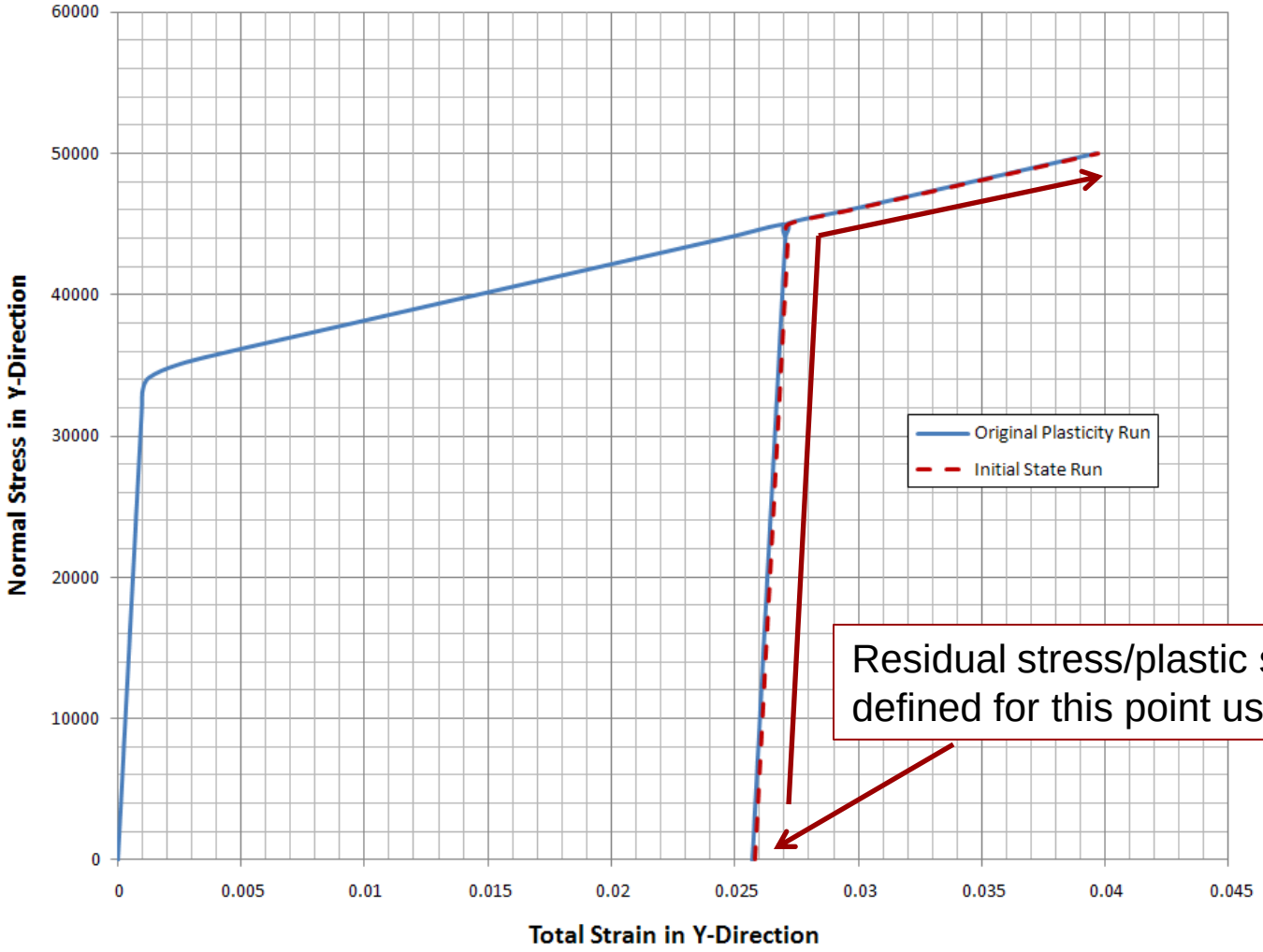
Repeating With Initial State

- The analysis is repeated by defining the residual stress and residual plastic strain at point 2, and applying the loading to point 3.
  - In this problem, the residual stress and plastic strain are the same for all integration points, so they can be entered directly using the material-based specification.
  - Options for applying initial state to individual elements, layers, sections and integration points exist, and the initial state can be read in via a file.
  - Note that the full stress and plastic strain tensors must be defined for this plane strain case in the solution processor:

```
! indicate material-based initial state
inistate,set,mat,1
! define plastic strain tensor
inistate,set,dtyp,eppl
inistate,defi,all,all,all,all,-0.02549,0.02579,-0.000295,0,0,0
! define stress tensor
inistate,set,dtyp,stress
inistate,define,all,all,all,all,0,0,8842,0,0,0
```

Repeating With Initial State

- The results from the initial state analysis are plotted, showing the comparison with the original analysis:



Writing an Initial State File

- The initial state can be defined more generally by writing out the integration point values to a text file.
  - To create an initial state file from an ANSYS analysis, enter commands in solution processor:

```
!   Generate initial state file
!      FLAG = 0 do not write, =1 write
!      CSID = coordinate system, DTYPE = S, EPEL, EPPL
inistate,write,FLAG,,,,CSID,DTYPE
```

  - For example, to establish and write out an initial state representing the results from a plasticity analysis:

```
!   Generate initial state file
inistate,write,1,,,,0,eppl
inistate,write,1,,,,0,s
```

Writing an Initial State File

- The initial state information is written to ***<filename>.ist***.
- For a multi-substep load step solution, the last result in the load step is written out.
- Once toggled on, any additional load steps will generate and append initial state data to the ***IST*** file.
  - Can turn off writing of the initial state using additional commands with the ***FLAG*** option:

```
! Generate initial state file
inistate,write,1,,,,0,eppl
inistate,write,1,,,,0,s
! Solve
solve
! Turn off initial state file writing
inistate,write,0,,,,0,eppl
inistate,write,0,,,,0,s
```

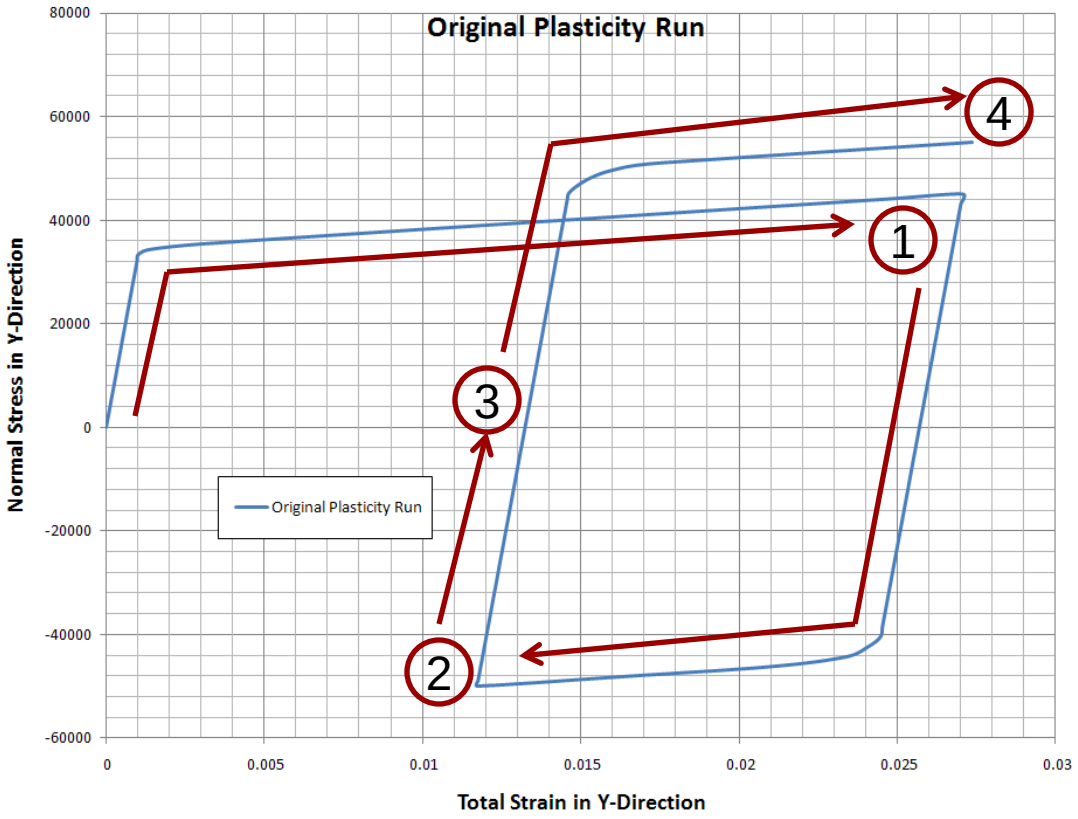
Writing an Initial State File

- The initial state file is a comma-delimited ASCII file format that can be created using any ANSYS-defined or external data.
- The file contents for the one-element test case:

```
!*************************************************************     INITIAL STATE FILE    ************************************************************
!*************************************************************     no_inis_bkin.ist      ************************************************************
!*************************************************************     HEADER INFORMATION    ************************************************************
/ETYP,DEFA
/COLINF,ELEM,ELIN,,,SX,SY,SZ,SXY,SYZ,SXZ
/ETYP,LAYE
/COLINF,ELEM,ELIN,LAYE,SECT,SX,SY,SZ,SXY,SYZ,SXZ
/ETYP,BEAM
/COLINF,ELEM,ELIN,CELL,SECT,SX,SY,SZ,SXY,SYZ,SXZ
!*************************************************************     INITIAL STATE DATA    ************************************************************
!     OUTPUT FORMAT FOR STRESSES
!     ELEM ID     ELEM INTG      LAY/CELL     SECT INTG      SX          SY          SZ          SXY         SYZ         SXZ
!     OUTPUT FORMAT FOR STRAINS
!     ELEM ID     ELEM INTG      LAY/CELL     SECT INTG     EPSX        EPSY        EPSZ        EPSXY       EPSYZ       EPSXZ
 /dtyp,eppl
 /csys,0
          1,          1,          1,          1, -0.254922E-01,  0.257869E-01, -0.294717E-03,  0.887820E-07
          1,          2,          1,          1, -0.254920E-01,  0.257867E-01, -0.294717E-03,  0.783295E-06
          1,          3,          1,          1, -0.254916E-01,  0.257863E-01, -0.294717E-03,  0.331339E-06
          1,          4,          1,          1, -0.254918E-01,  0.257866E-01, -0.294717E-03, -0.363174E-06
 /dtyp,s
 /csys,0
          1,          1,          1,          1,  0.133487       , -0.134201       ,   8841.51    ,  0.124463E-02
          1,          2,          1,          1,  0.501218E-02, -0.504554E-02,   8841.51    ,  0.137224
          1,          3,          1,          1, -0.133489       ,  0.134203       ,   8841.51    ,  0.464484E-02
          1,          4,          1,          1, -0.501011E-02,  0.504345E-02,   8841.51    , -0.131334
```

Non-Proportional Loading

- Since the initial state is defined using plastic strain EPPL, which does not represent accumulated plastic strain, its use is limited to proportional loading.
- For example, consider the non-proportional loading sequence assuming isotropic hardening below:

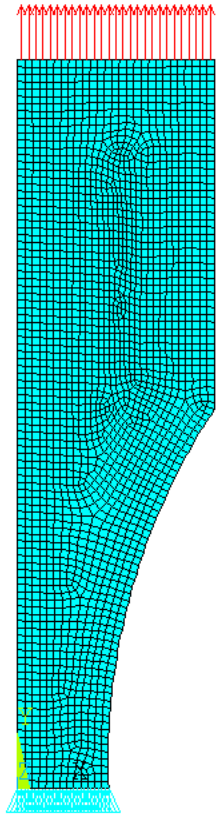


Non-Proportional Loading

- The load sequence:
  - The initial loading to point 1 accumulates plastic strain.
  - The load is reversed and additional plastic strain is accumulated to point 2.
  - The load is removed at point 3.
  - Additional load is applied to point 4.
- If the state at point 3 is to be used as an initial state, the plastic strain and stress state are identical to that which would have been generated in a proportional load case with no load reversal.
  - However, due to isotropic hardening, yielding in compression has increased the yield in tension.
  - More importantly, since only EPPL is stored, the total accumulated plastic strain will be incorrect.
- Therefore, the initial state feature is not appropriate for non-proportional loading problems.

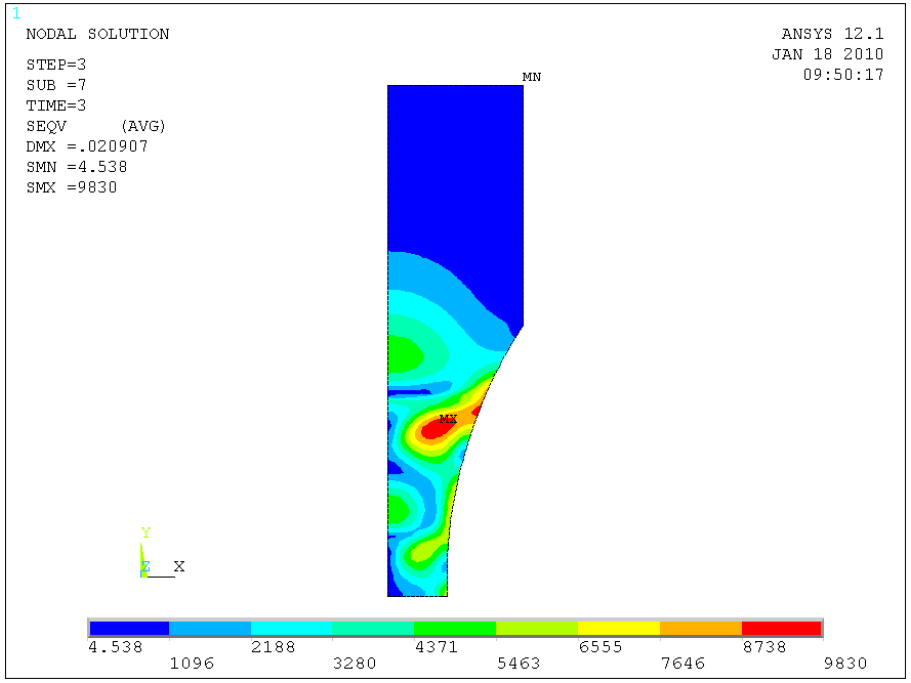
General Case

- To illustrate the general case of a non-uniform residual stress state, consider the axisymmetric tension specimen model.
  - Assume an initial material yield stress = 30,000 psi, kinematic hardening.
  - Run two cases:
    - Apply total load of 3206 lb. assuming no initial state.
    - Apply total load of 3206 lb. assuming an initial state derived from pre-stressing the specimen to 2750 lb. which results in a residual plastic strain and stress field.
  - Observe hardening behavior.

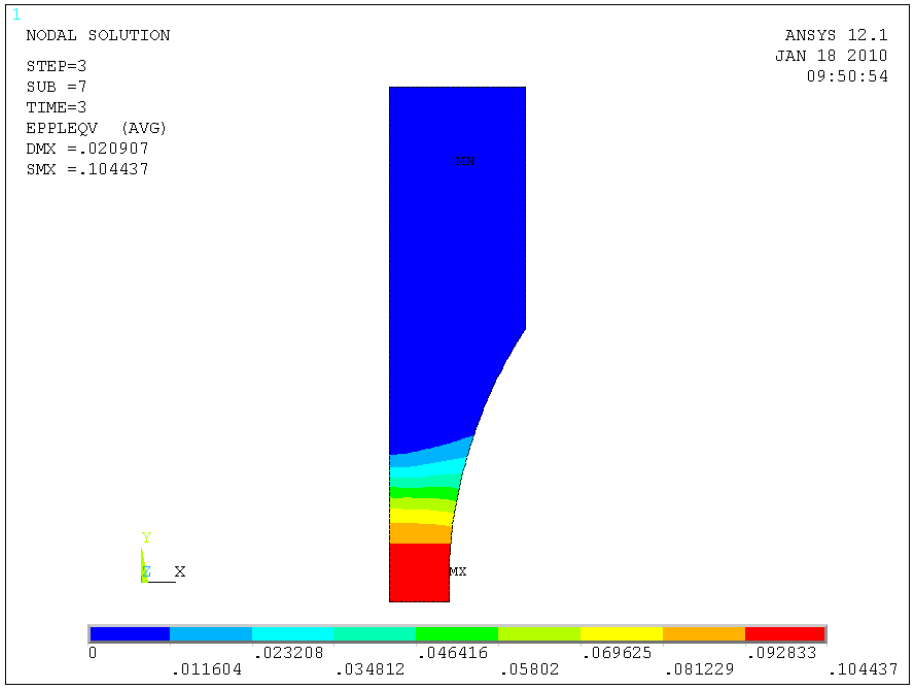


General Case

- The residual stress and plastic strain state:
  - Written out using: ***inistate,write,1,,,,0,eppl***
    inistate,write,1,,,,0,s
  - Read in using: ***inistate,read,no_inis_specimen,ist***



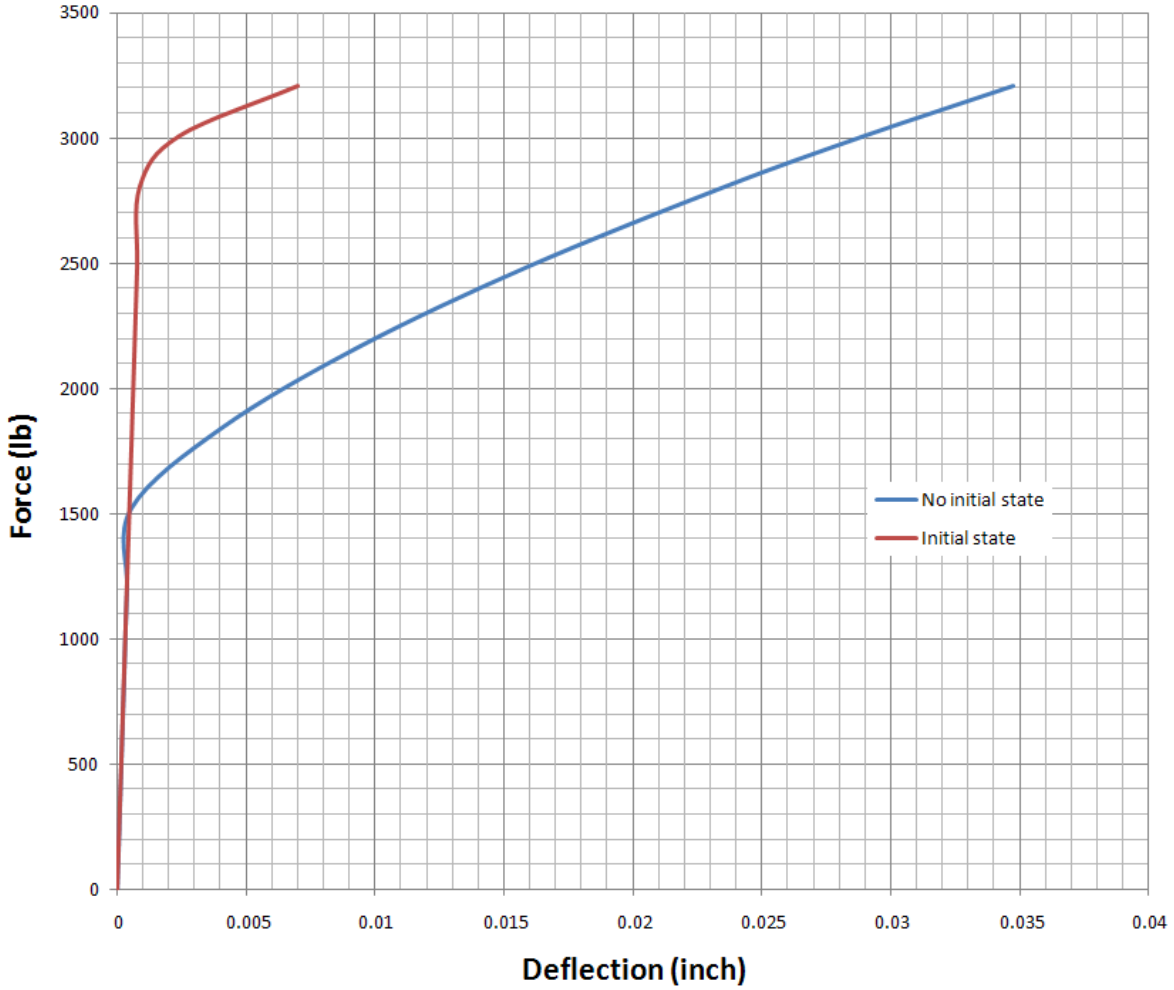
Residual stress state



Plastic strain state

General Case

- Comparing the force-deflection behavior for the base specimen versus the specimen with an initial state defined:



Summary

- The current version 12.1 of ANSYS has added the capability to define initial strain, initial plastic strain, and initial stress.
- This new enhancement provides a valuable capability to incorporate a residual stress/strain state from previous loadings such as manufacturing processes.
- The **INISTATE** command provides several ways to define the initial state:
  - Constant values entered via command for materials, elements, layers, sections and integration points.
  - Automatic generation of **IST** file from ANSYS analysis.
  - Manual generation from data defined outside of ANSYS.
- Cannot be used with non-proportional loading cases.