

# Exercise 3

## Nonlinear material-

elastoplastic bending of a simply supported beam  
(residual stress after unloading)

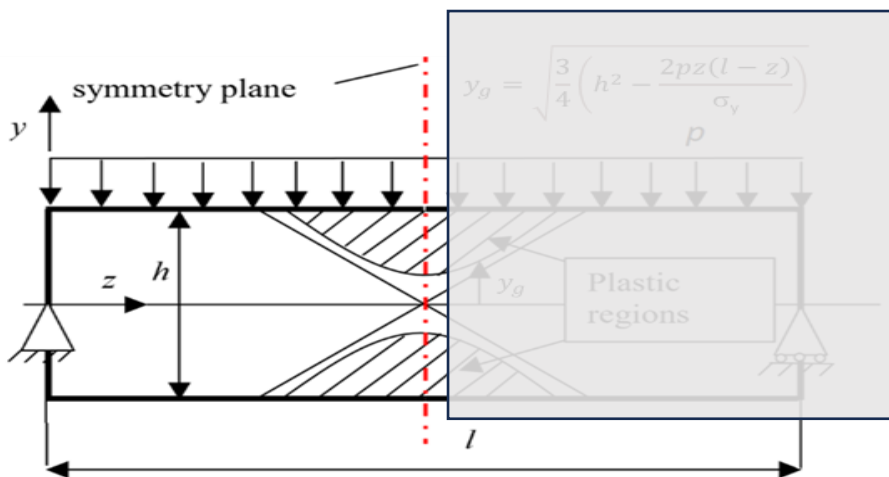
Solid beam , L= 400 mm

rectangular cross-section 20 x 10 mm

(vertical cross-section arrangement, h=20, b=10)

Simply supported.

Symmetry used – so **ONLY** left part is modelled



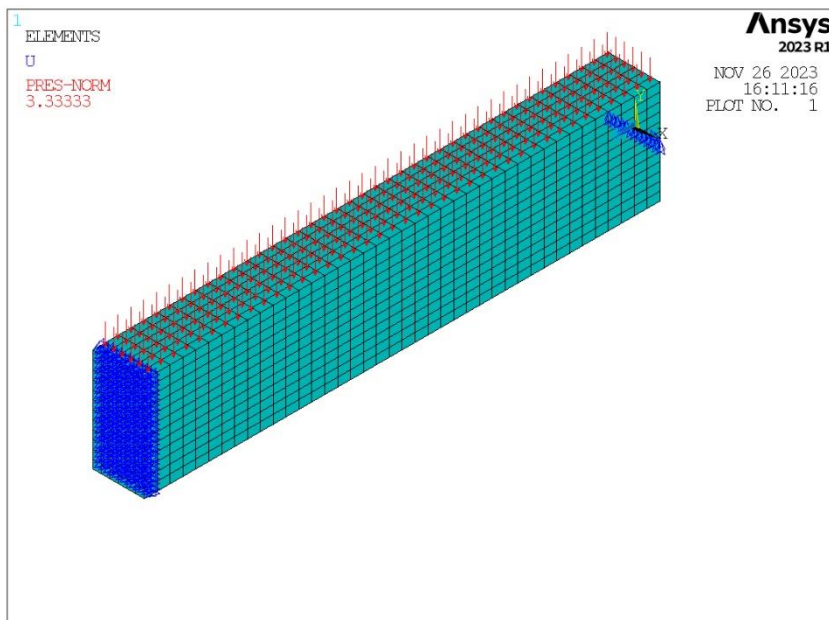
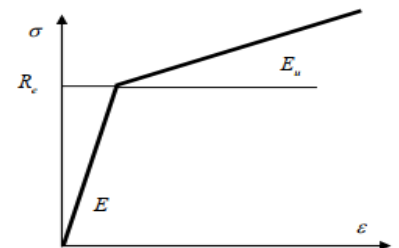
Material:

Steel, **Elasto-plastic** behaviour.

$E=200$  GPa,  $\nu=0.3$

$\sigma_y=R_e=250$  MPa,  $E_t=100$  MPa

bi-linear approximation



## A .Create GEOMETRY

1. Blocks creation (upper and lower part of the LEFT side)

*Preprocessor>Modeling>Create>Volumes>Block>By Dimensions*

create two blocks (Fig. 1):

$X1 = -\frac{1}{2}b, X2 = \frac{1}{2}b, Y1 = 0, Y2 = \frac{1}{2}h, Z1 = 0, Z2 = \frac{1}{2}l$ , and

$X1 = -\frac{1}{2}b, X2 = \frac{1}{2}b, Y1 = -\frac{1}{2}h, Y2 = 0, Z1 = 0, Z2 = \frac{1}{2}l$

1. merge the two blocks

*Preprocessor>Numbering Ctrl>Merge Items>All*

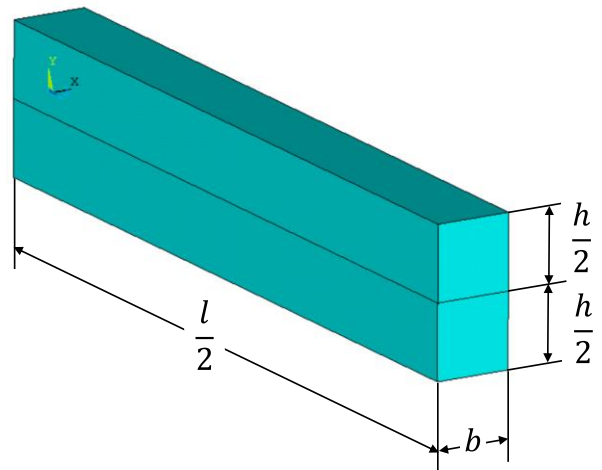
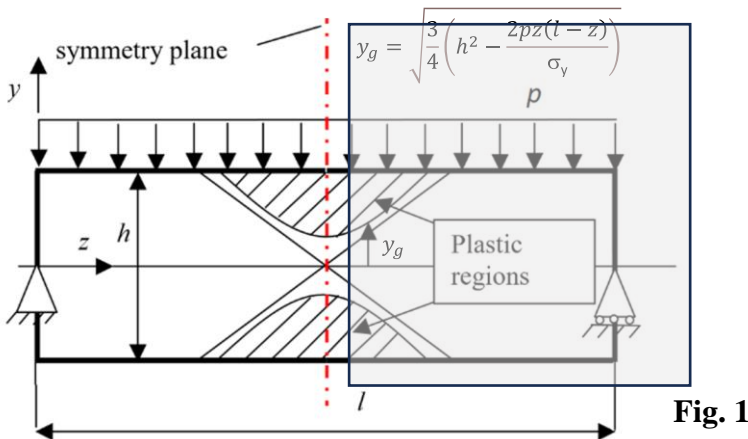
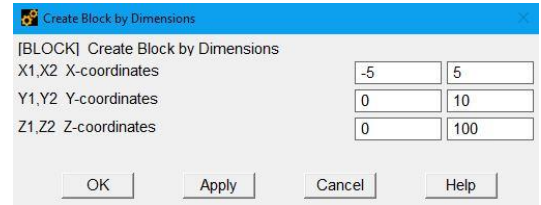
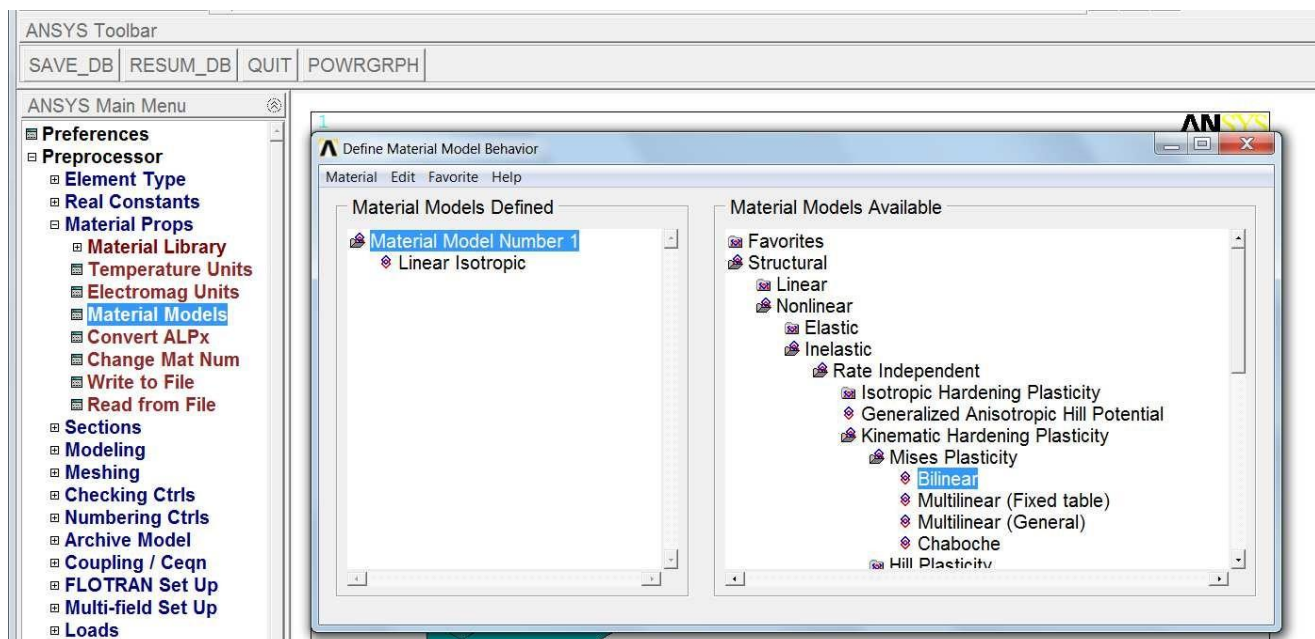
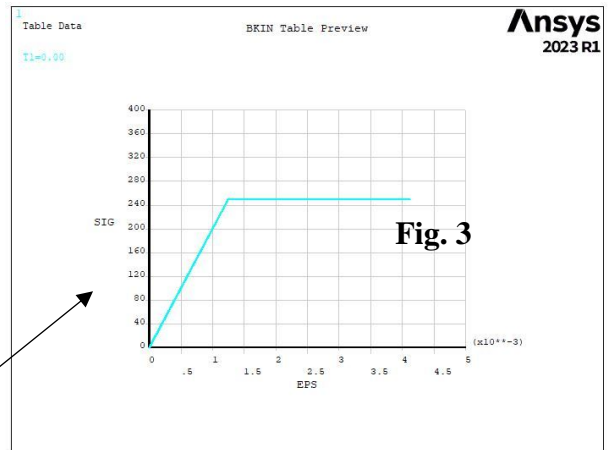
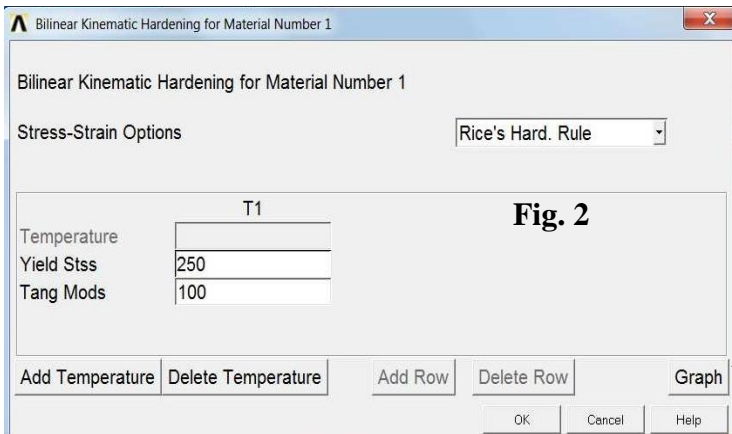


Fig. 1

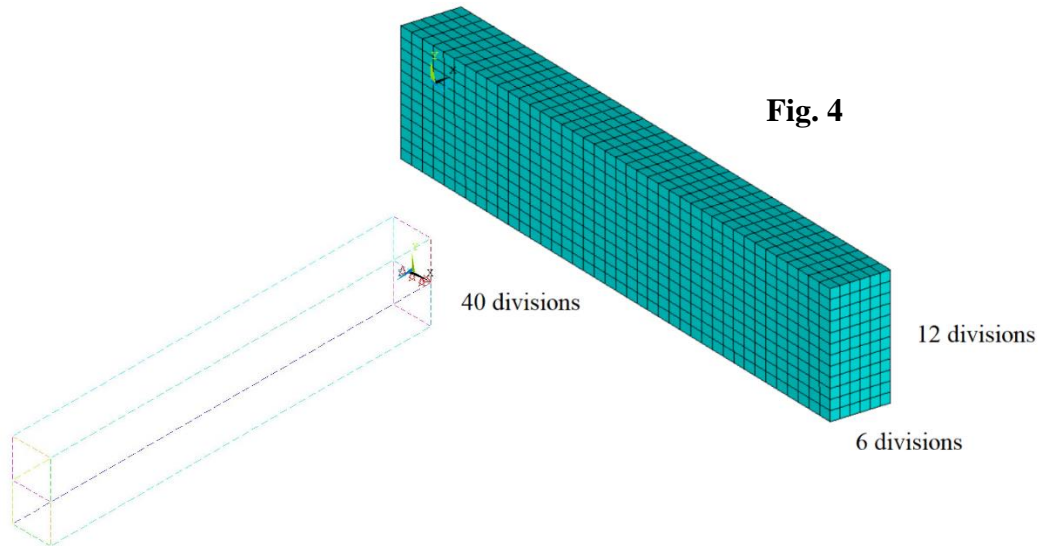
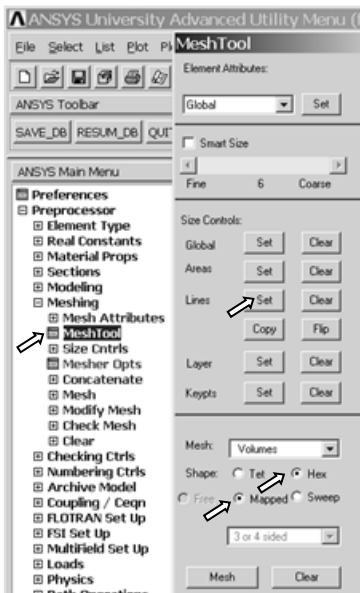
## B . FE DATA – preparation (element, material, meshing, LBC =Loads and Boundary Conditions)

2. Choose *SOLID186* finite element (Brick 20 node, element technology: *Reduced integration*)
3. Define linear elastic, isotropic material properties:  $E = 2 \cdot 105 \text{MPa}$  (Young's modulus) and  $\nu = 0.3$  (Poisson's ratio) and nonlinear properties: a *bilinear kinematic hardening* model with  $\sigma_y = 250 \text{MPa}$  (Yield stress) and  $E_t = 100 \text{MPa}$  (tangent modulus) (Fig. 2 and 3).

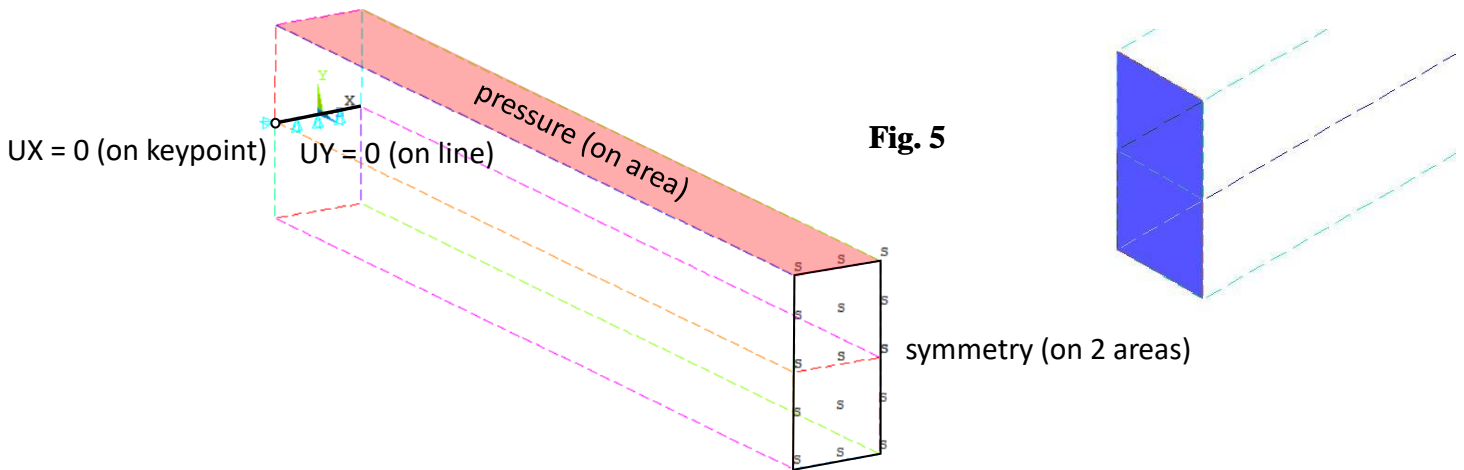




- Specify the density of discretization on lines and create a **mapped hexahedral mesh** (Fig. 4)  
(Main Menu>Preprocessor>Meshing>Mesh Tool)

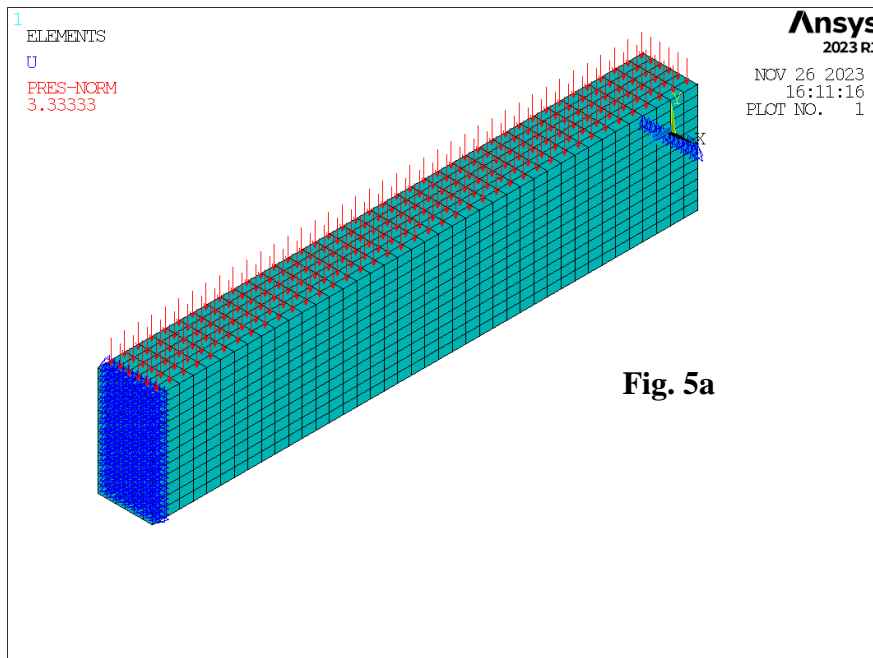


- Save the mesh as an image  
(Plot> Elements, PlotCtrls> Redirect Plots-> To JPEG File ...)
- Apply boundary conditions: displacements on one **keypoint** (UX) and on a **line** (UY)  
(Solution> DefineLoads>Apply> Structural> Displacement> On ... )  
and symmetry conditions on **TWO Areas**  
(Solution> DefineLoads>Apply> Structural> Displacement>Symmetry B.C. )



**Fig. 5**

and a pressure of  $10/3$  MPa on the top Area(Fig. 5 and 5a)  
*(Solution> DefineLoads>Apply> Structural> Pressure>On Areas )*



**Fig. 5a**

### C. Set the analysis options , create Load Steps and run the Solution

**Description of the analysis strategy:**

**A nonlinear static analysis of the elastoplastic beam will be done in three LoadSteps:**

- **step 1:** to reach the yield stress at the top and bottom surfaces of the beam for  $z = \frac{1}{2}l = 10$  mm.
- **step 2:** the limit pressure is applied so that the theoretical stress in z direction at each point of the cross-section located in the middle of the beam length is equal to the yield stress (+/-).
- **step 3:** the pressure is set to zero to capture the residual stress and permanent deformation.

7. Define a new analysis as **STATIC**

*(Main Menu>Solution>Analysis Type> New Analysis>Static)*

a) And specify options in two forms (read below about the options):

*(Main Menu>Solution>Analysis Type> AnalysisOptions)* – Large Deformations

*(Load Step Options>Time/Freq>Time and Substeps)* – Automatic Time Stepping

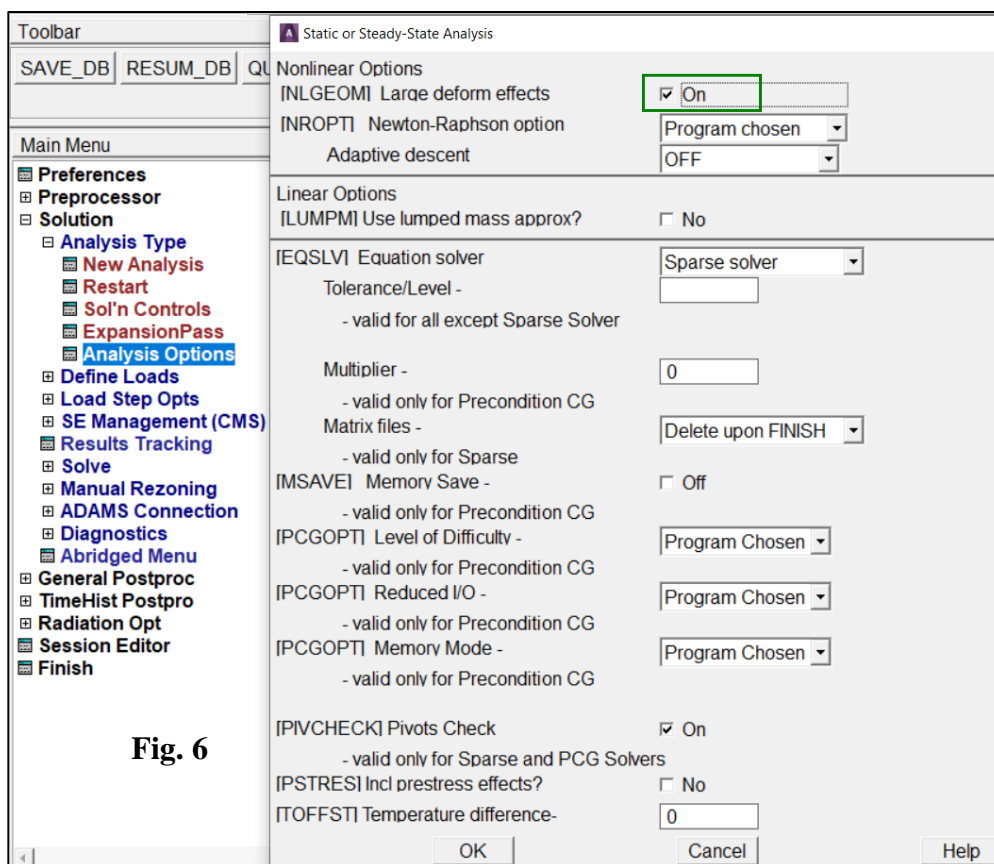
The pressure in each load step is applied as ‘ramped’: 0->10/3, 10/3->5, 5->0 (MPa). In the ‘Analysis Options’ (Fig. 6) the Large Deformations are turned ON and in the ‘Time and Substeps’ the automatic time stepping is turned OFF (to get results in chosen time moments) – see Fig 7.

NOTE: In a static analysis, time has only **formal** meaning. The time value at the end of the first and second load steps reflect the pressure magnitude to make results more readable.

b) Set and write Load Steps (apply Pressure as in point 6, and Options as in Fig 7 and 8):

- LS1: apply  $p = 10/3$  MPa, specify proper TIME, NSUBST, FREQ (Fig. 7, 8 and 9), and **LSNUM = 1**,
- LS2: apply  $p = 5$  MPa, specify proper TIME, NSUBST, FREQ, and **LSNUM = 2**,
- LS3: apply  $p = 0$  MPa, specify proper TIME, NSUBST, FREQ, and **LSNUM = 3**,

## 8. Settings below



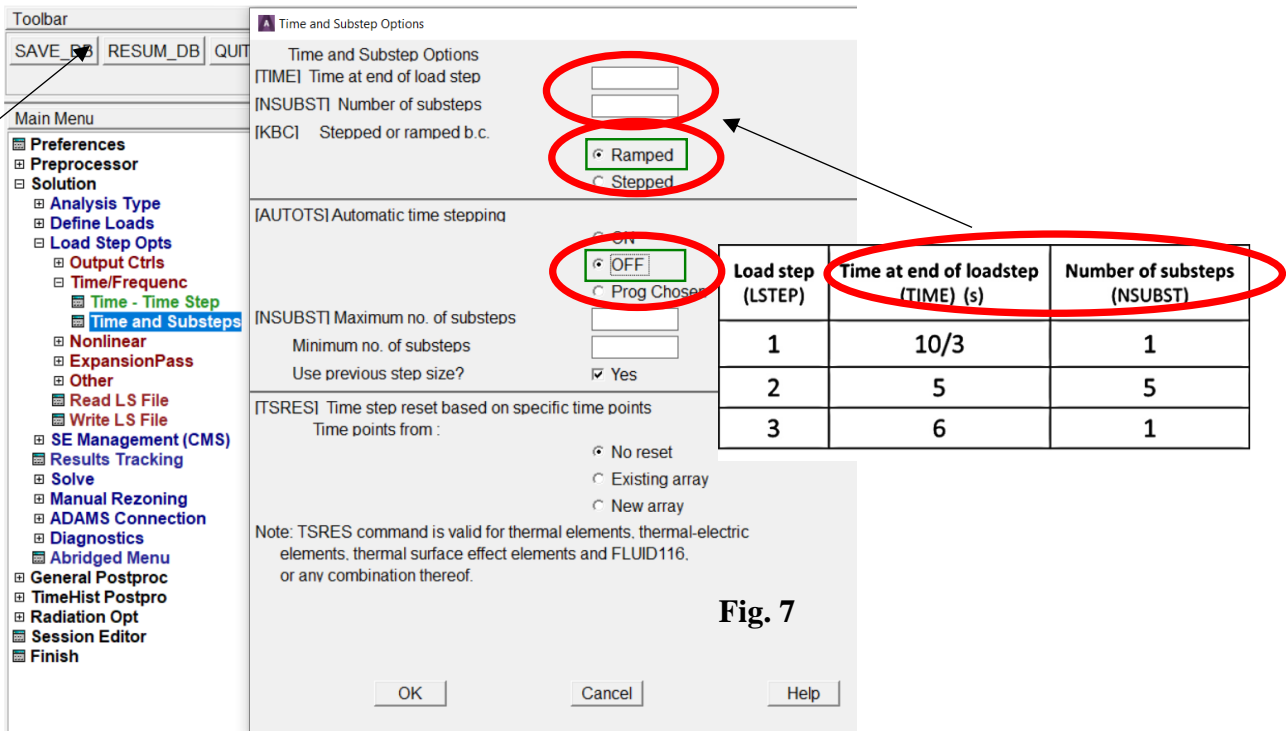


Fig. 7

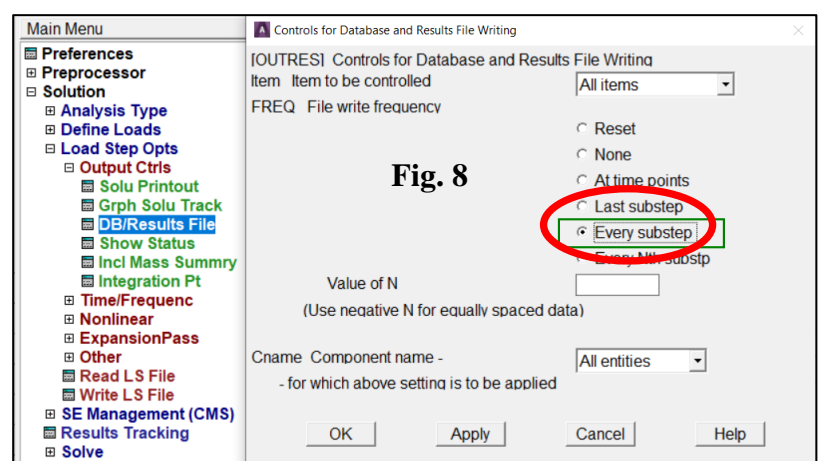


Fig. 8

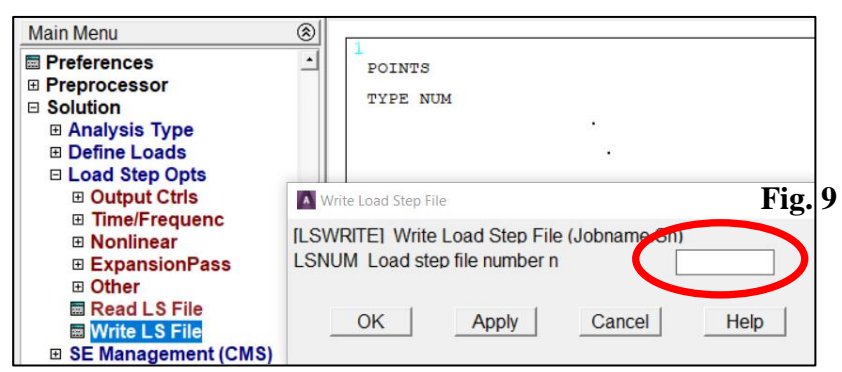


Fig. 9

9. Solve the problem

(Solution-> Solve-> From LS Files; LSMIN = 1, LSMAX = 3, LSINC = 1)

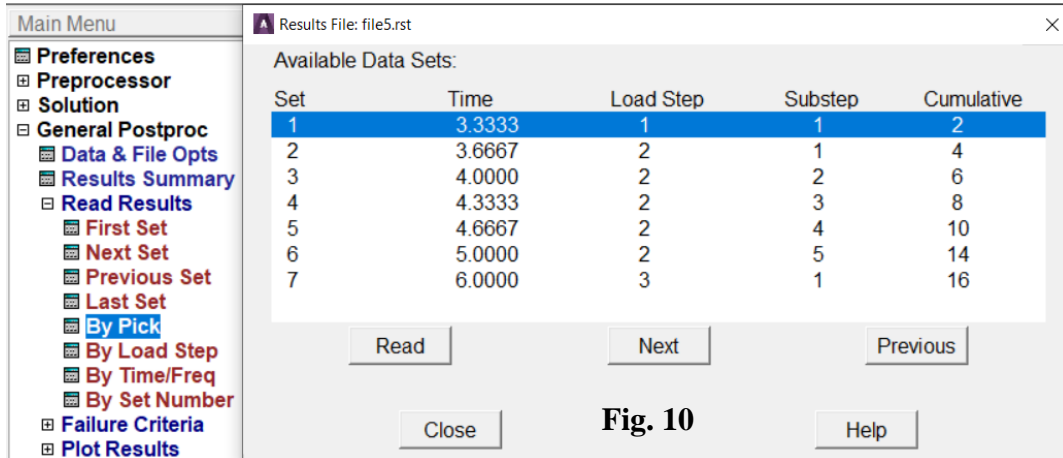
**D . POSTPROCESSING – results visualization**

10. Read the results from selected Step

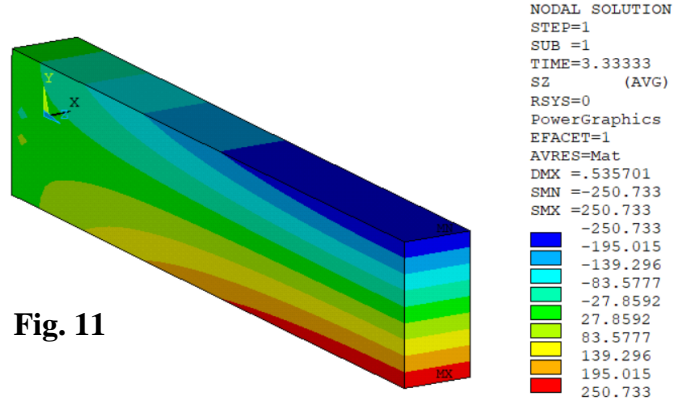
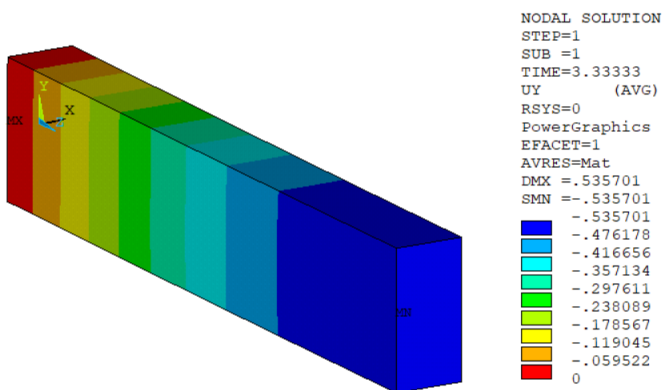
(Main Menu> General Postproc> Read Results> By Pick>) (Fig. 10)

11. Plot and save as images contour maps of the vertical displacement UY and stress component SZ at the end of each load step

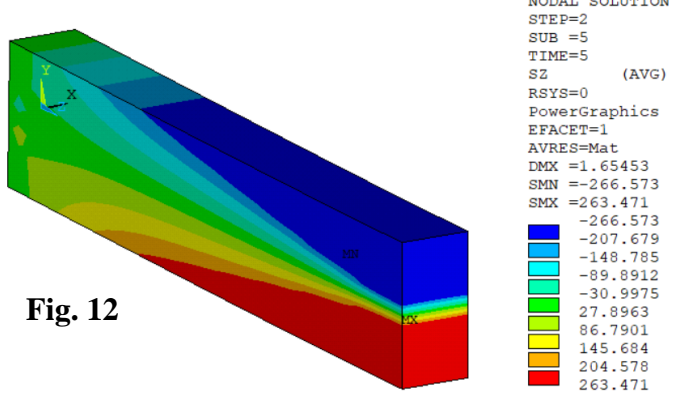
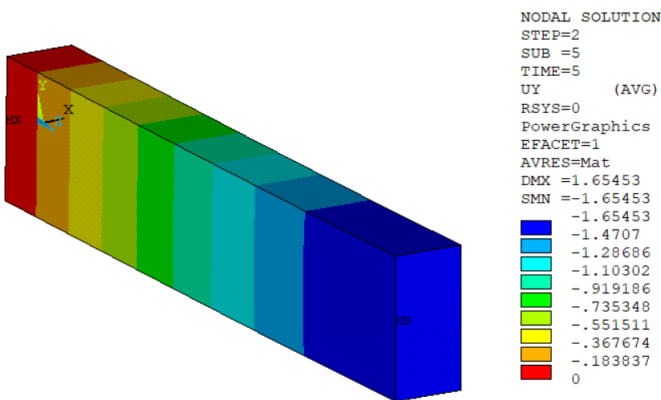
(Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu> ...) (Fig. 11-13).



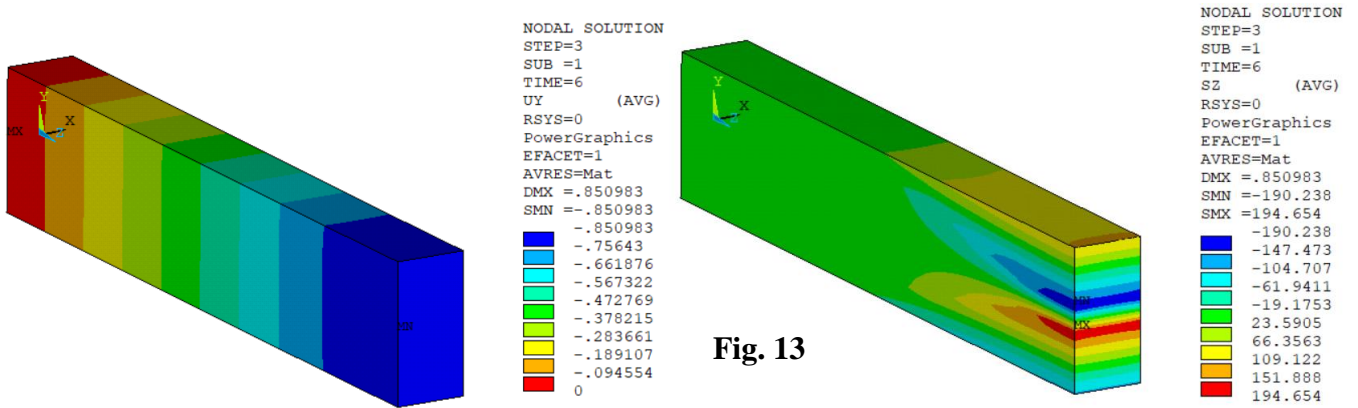
**Fig. 10**



**Fig. 11**

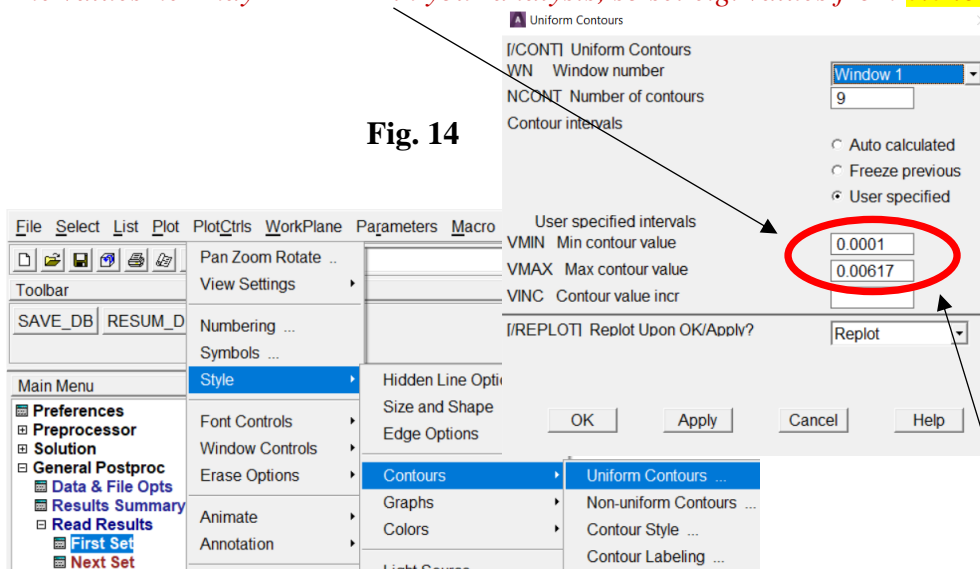


**Fig. 12**



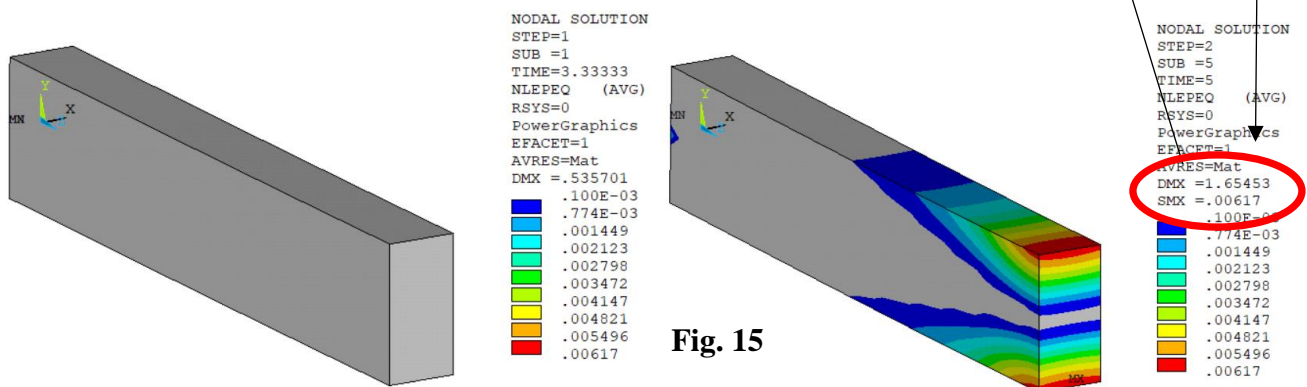
12. Set a user specified (uniform) range of the contour scale (Fig. 14). Take the values from next plot. Setting user scale from **top Menu**: *PlotCtrl>Style>Contours>UniformContours*

*The values her may DIFFER in your analysis, so set e.g. values from 0.0 to 0.0055*



13. Plot and **save as images** contour maps of the equivalent plastic strain **NLEPEQ** at the end of the **LS1** (left) and **LS2** (right) (Fig. 15).

*Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu>Plastic Strain> Equivalent plastic strain ...)*





**General remark – CREATING graphs of some values along a path**

- Create a PATH (*General Postproc>PathOperations>Define Path>ByNodes*)
- Read the LS results and choose the value to graph (*General Postproc>PathOperations>MapOntoPath*). Remember to set the **USER LABEL** for the created graph curve.
- Post the curve (*General Postproc>Path Operations>PlotPathItem>onGraph*) – selecting the **LABELS** of curves

14. Plot graphs of the displacement UY and stress component SZ at the end of LS1, LS2, LS3 as a function of the distance **ALONG the length of BEAM** and along **THE HEIGHT** and save the graphs (Fig. 16).

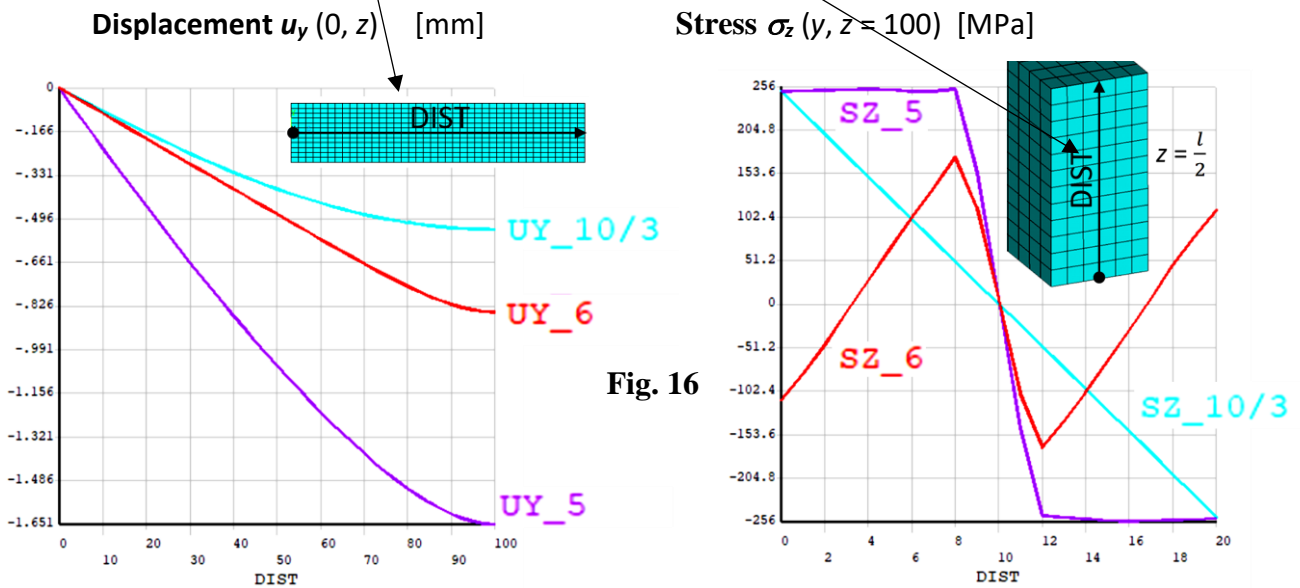


Fig. 16

15. Time/History graphs (as it was done in Ex2\_Transient\_dynamics)

Read the displacement UY and stress component SZ in the cross-section  $z = 100$  as a function of time, (*Main Menu>TimeHist Postpro>Variable Viewer*) and save the graphs as images (Fig. 17).

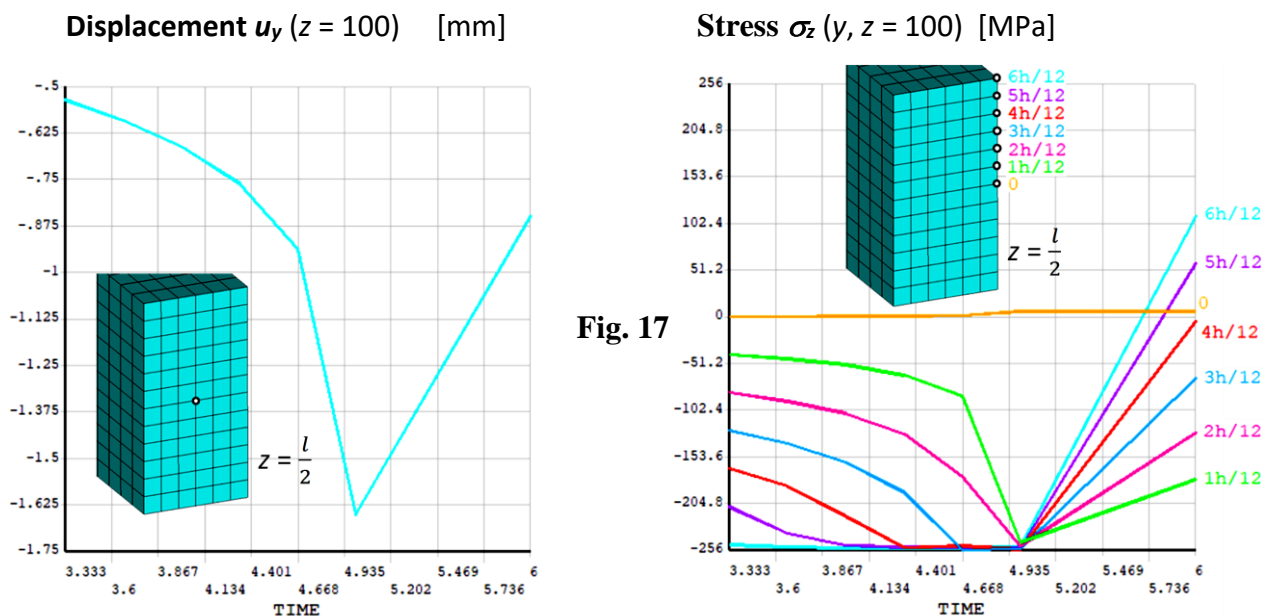
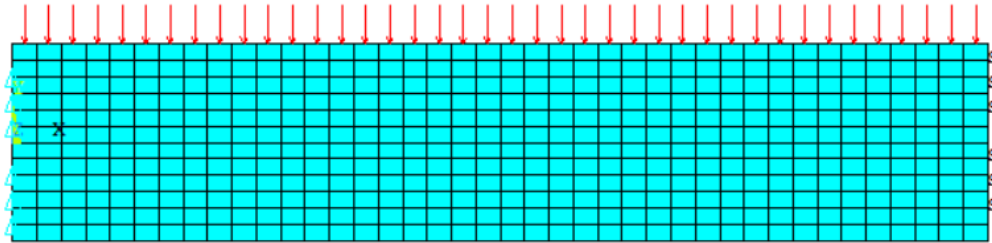


Fig. 17

**OPTIONAL:**

15. Repeat the calculation for a 2D model of the beam in a plane stress state using PLANE183 element (Fig. 18). Apply displacement  $UY = 0$  on the left edge instead of at a keypoint.

**Fig. 18**



16. Discuss results and write conclusions.