

### LAB 3. Elastoplastic deformation and residual stress in a simply supported beam

1. Geometry of the beam (width  $b$ , height  $h$  and length  $l/2$ ):

a) create two blocks (*Preprocessor>Modeling>Create>Volumes>Block>By Dimensions*) (Fig. 1):

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

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

b) merge the two blocks (*Preprocessor>NumberingCtrls>Merge Items>All*)

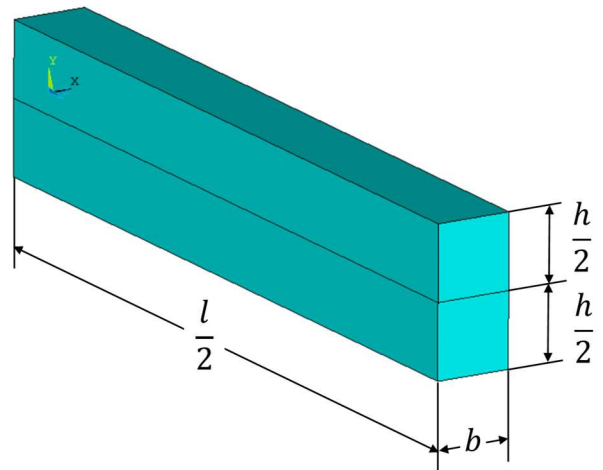
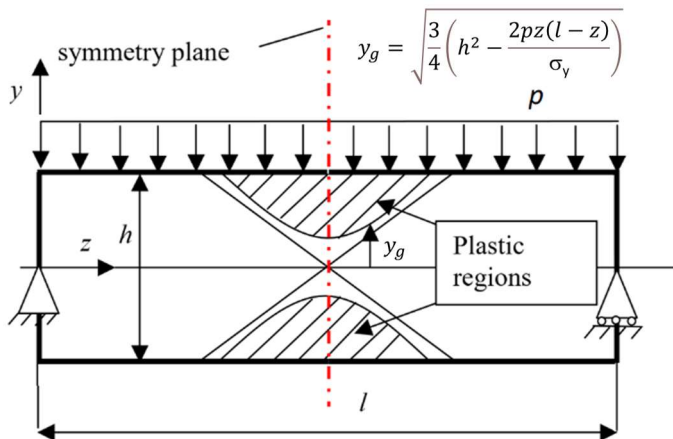


Fig. 1

2. Choose SOLID186 finite element (Brick 20 node, element technology: *Reduced integration*)

3. Define linear elastic, isotropic material properties:  $E = 2 \cdot 10^5$ MPa (Young's modulus) and  $\nu = 0.3$  (Poisson's ratio) and nonlinear properties: a bilinear kinematic hardening model with  $\sigma_y = 250$ MPa (Yield stress) and  $E_t = 100$ MPa (tangent modulus) (Fig. 2 and 3).

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

5. **Save the mesh as an image** (*Plot> Elements, PlotCtrls> Redirect Plots-> To JPEG File ...*)

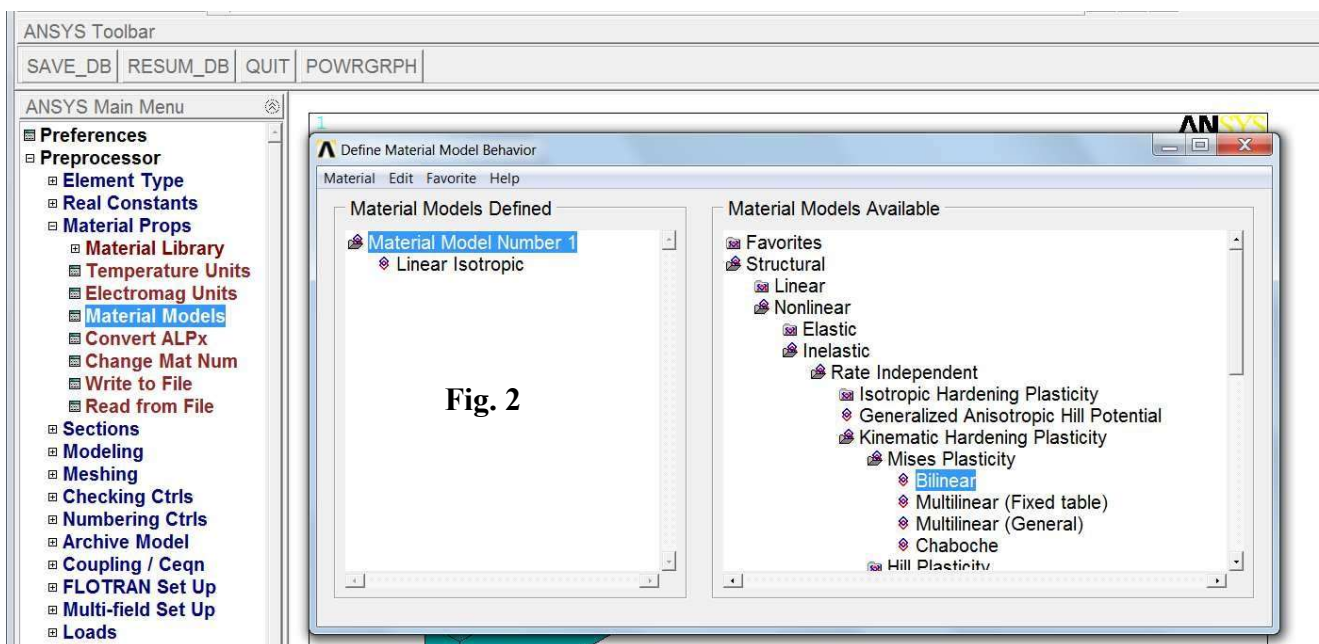


Fig. 2

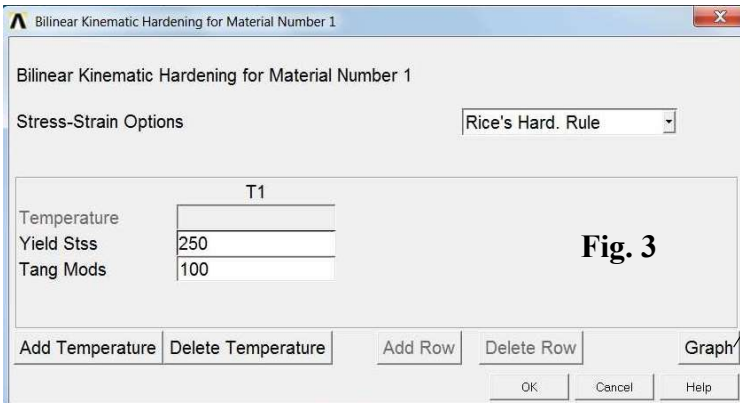


Fig. 3

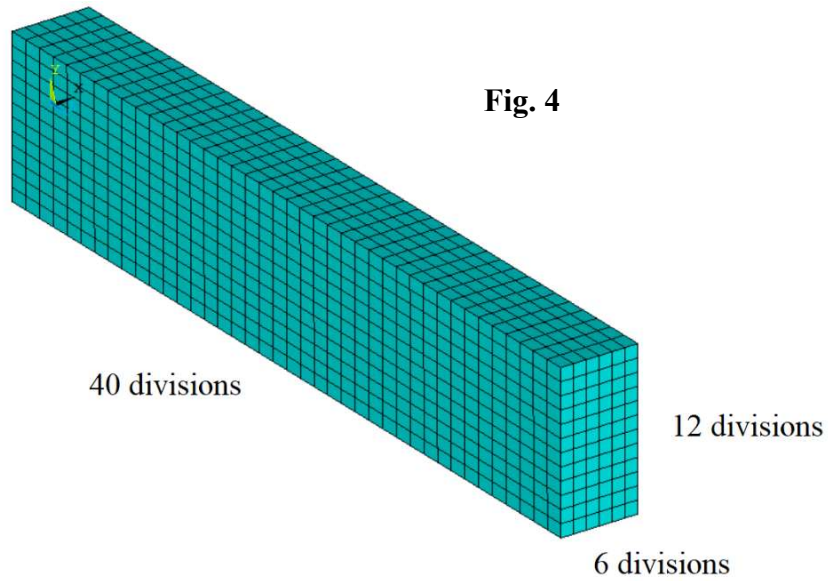
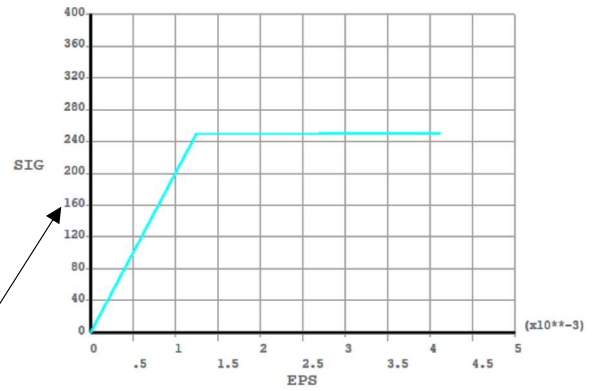


Fig. 4

6. Apply boundary conditions: displacements and pressure of 10/3 MPa (Fig. 5) (*Solution > Apply > Structural > Displacement ... and*)

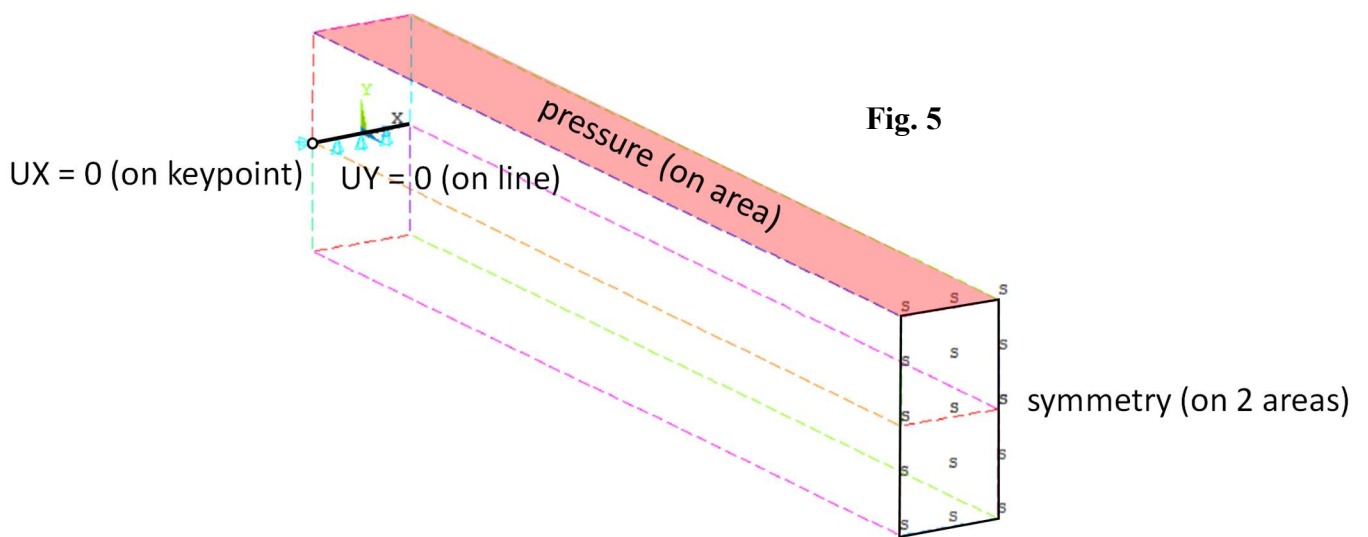


Fig. 5

7. Set the analysis options and create load steps.

A nonlinear static analysis of the elastoplastic beam will be done in three load steps:

- step 1: to reach the yield stress at the top and bottom surfaces of the beam for  $z = \frac{1}{2}l$ .
- 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.

a) Define a new analysis (Main Menu>Solution>Analysis Type> New Analysis>Static) and specify options:

The pressure in each load step is applied as 'ramped': 0->10/3, 10/3->5, 5->0 (MPa). Large deformations are included and the automatic time stepping is turned off to get results in chosen time moments, set by 'Time and Substeps Options' (Fig. 6 to 8). 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) Write load steps:

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

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

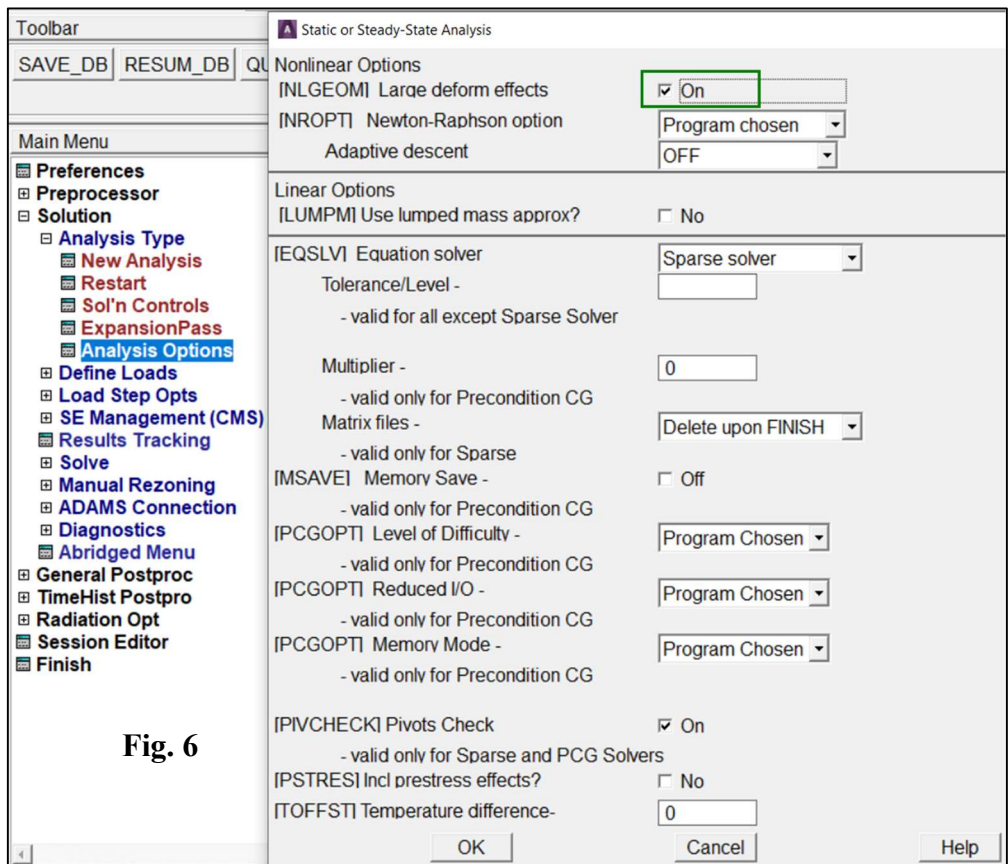


Fig. 6

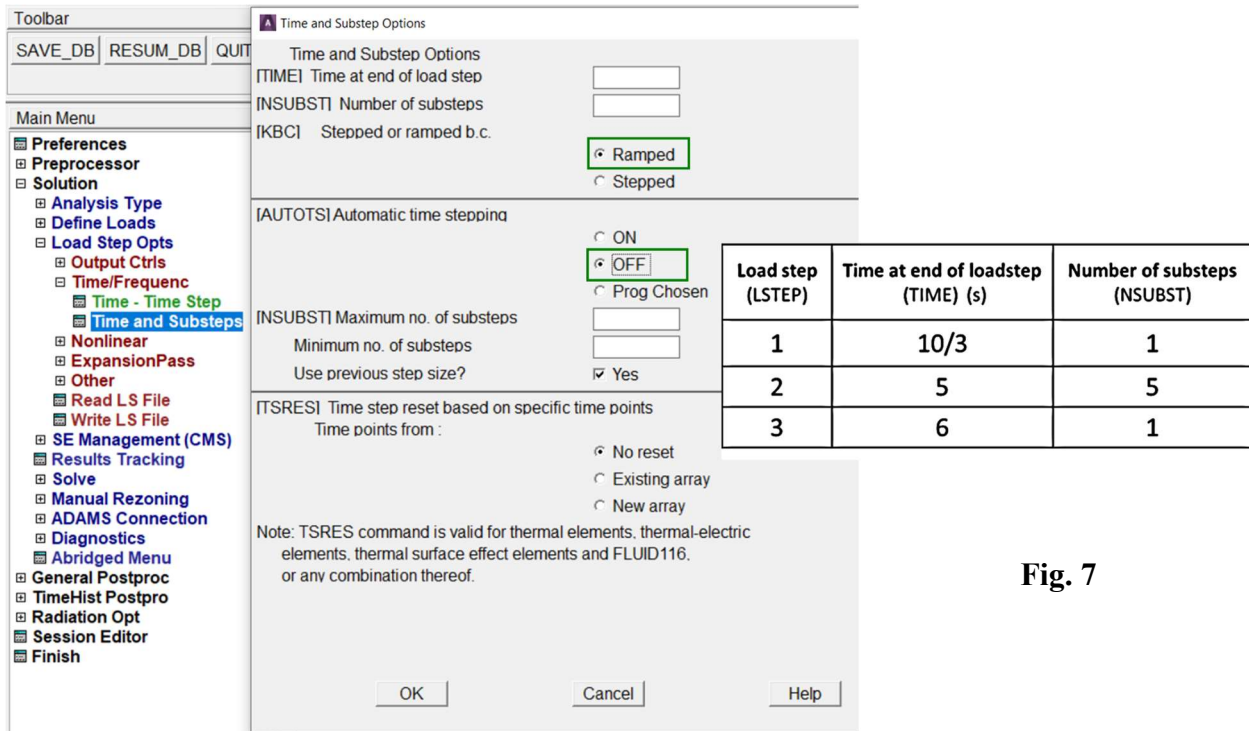


Fig. 7

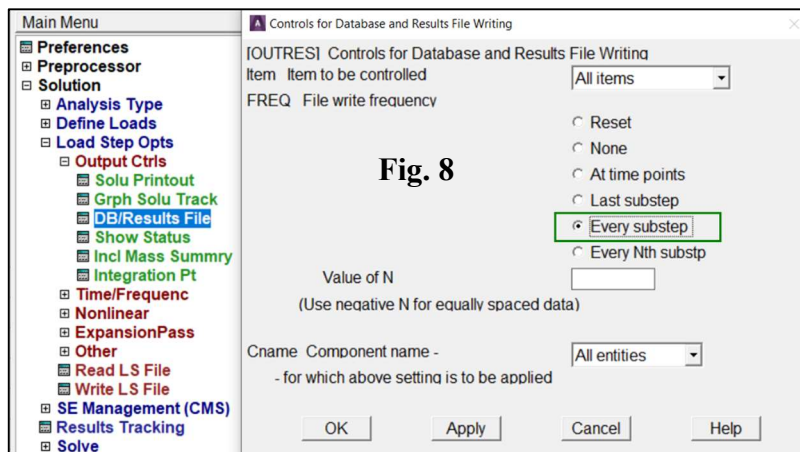


Fig. 8

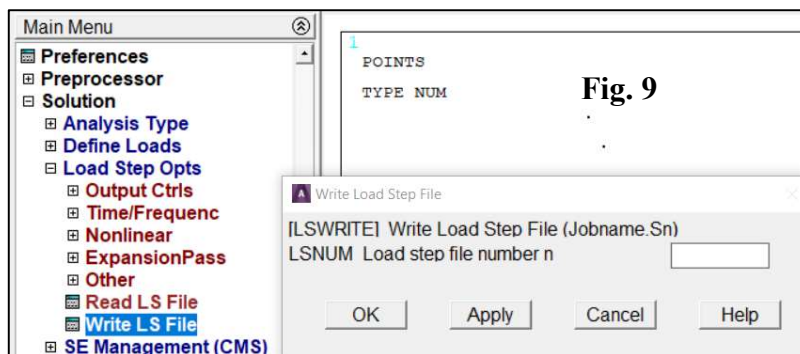


Fig. 9

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

10. Read the results (Main Menu> General Postproc> Read Results> First Step) (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). Fill Table 1.



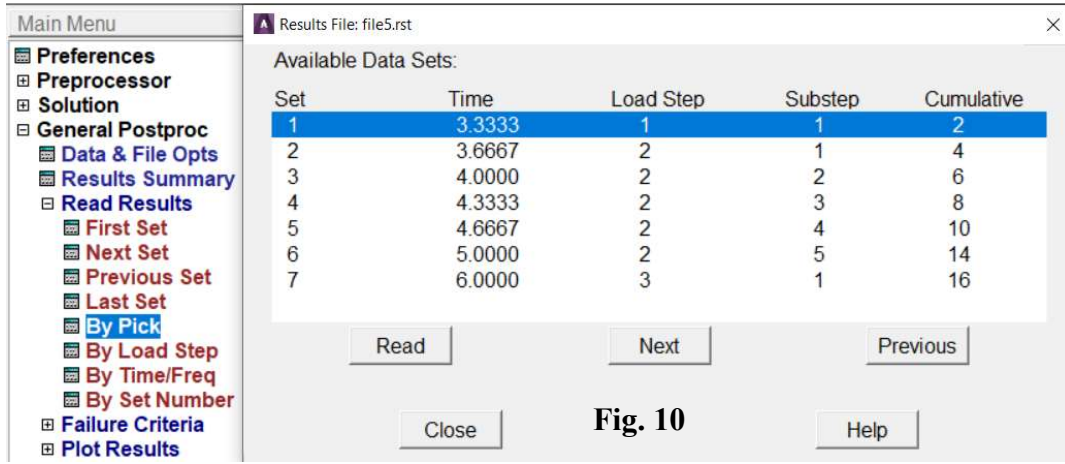


Fig. 10

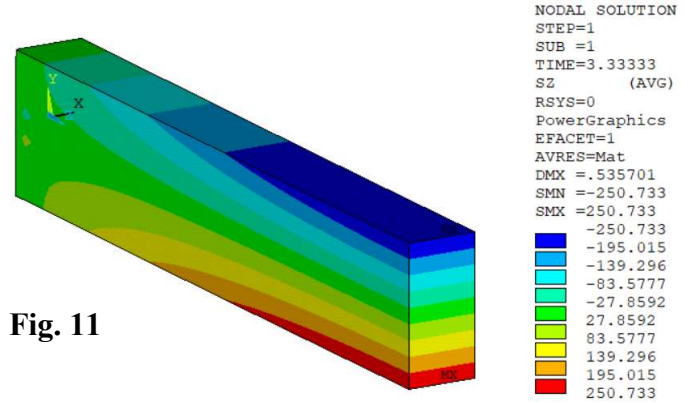
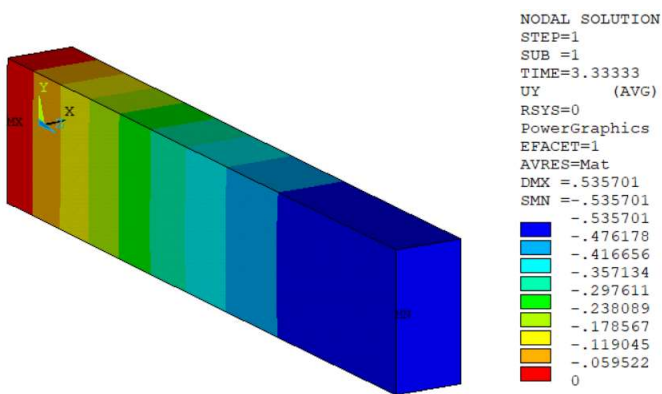


Fig. 11

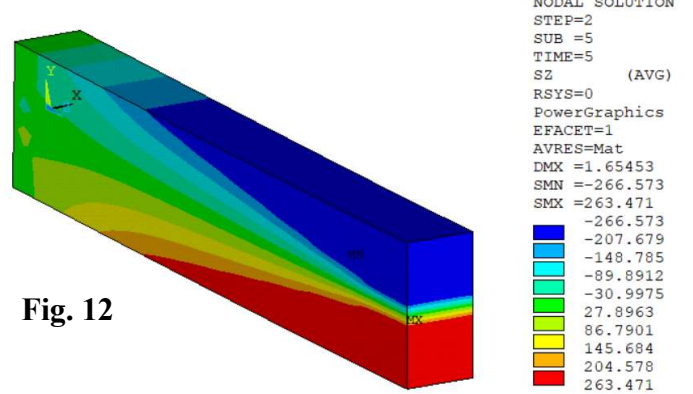
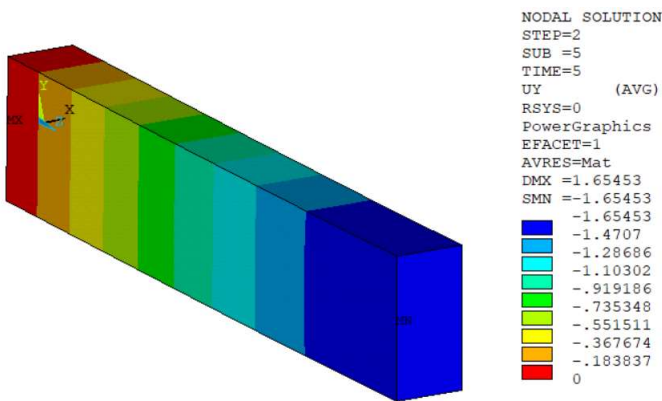


Fig. 12

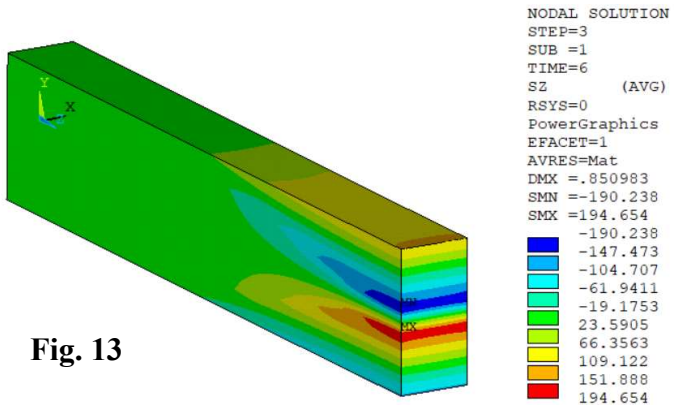
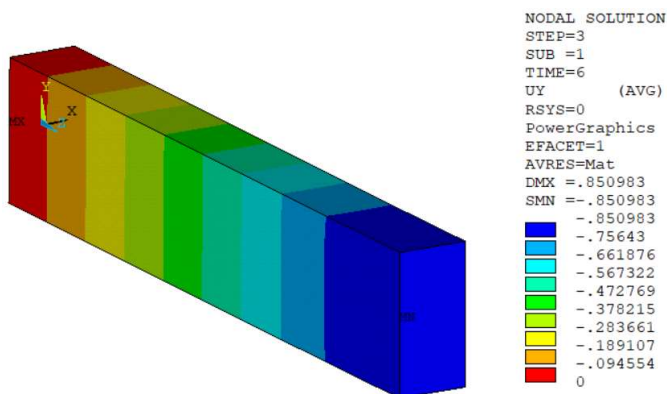


Fig. 13

12. Set a user specified range of the contour scale (Fig. 14). Plot and [save as images](#) contour maps of the equivalent plastic strain NLEPEQ at the end of the first and second load steps (*Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu>Plastic Strain>Equivalent plastic strain ...*) (Fig. 15). Fill Table 1.

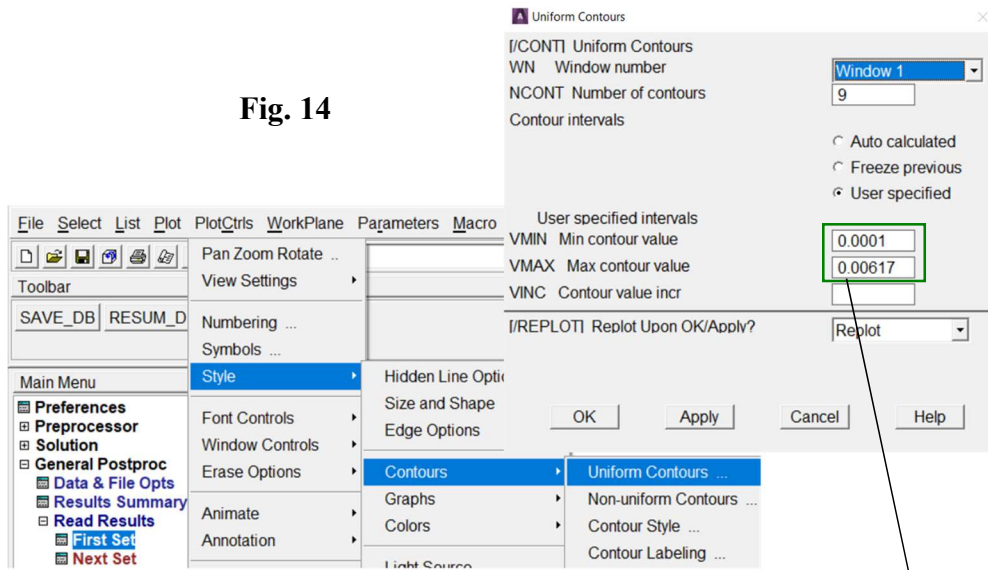


Fig. 14

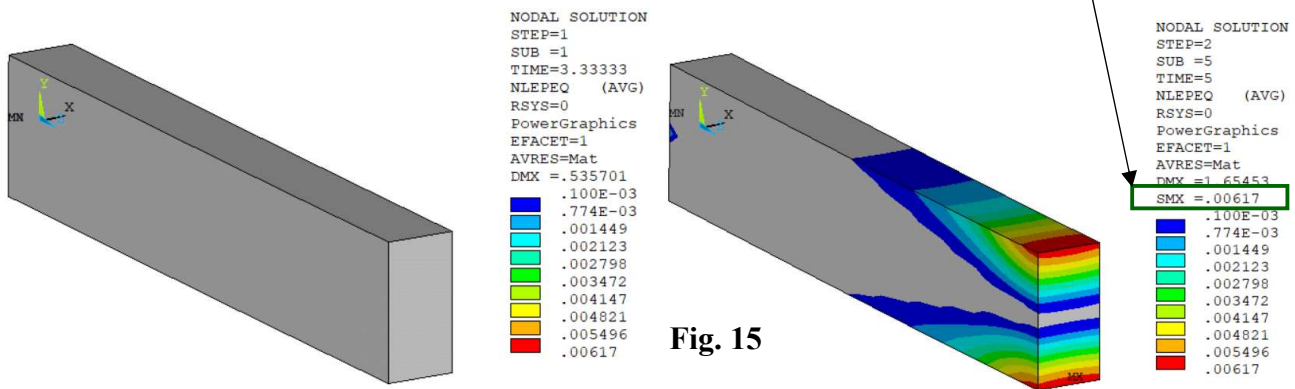


Fig. 15

13. Plot graphs of the displacement  $U_y$  and stress component  $S_z$  at the end of each load step as a function of the distance and [save the graphs as images](#) (*Main Menu>General Postproc>Path Operations>...*) (Fig. 16).

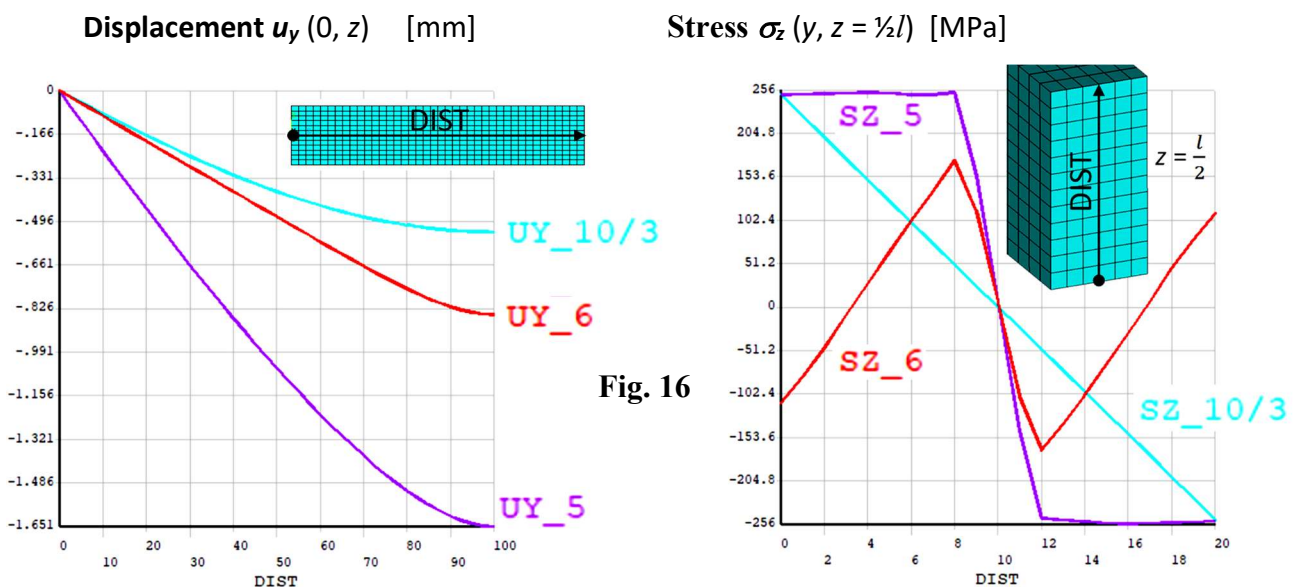


Fig. 16

14. Read the displacement  $U_y$  and stress component  $S_z$  in the cross-section  $z = \frac{1}{2}l$  as a function of time, (Main Menu>TimeHist Postpro>Variable Viewer) and save the graphs as images (Fig. 17).

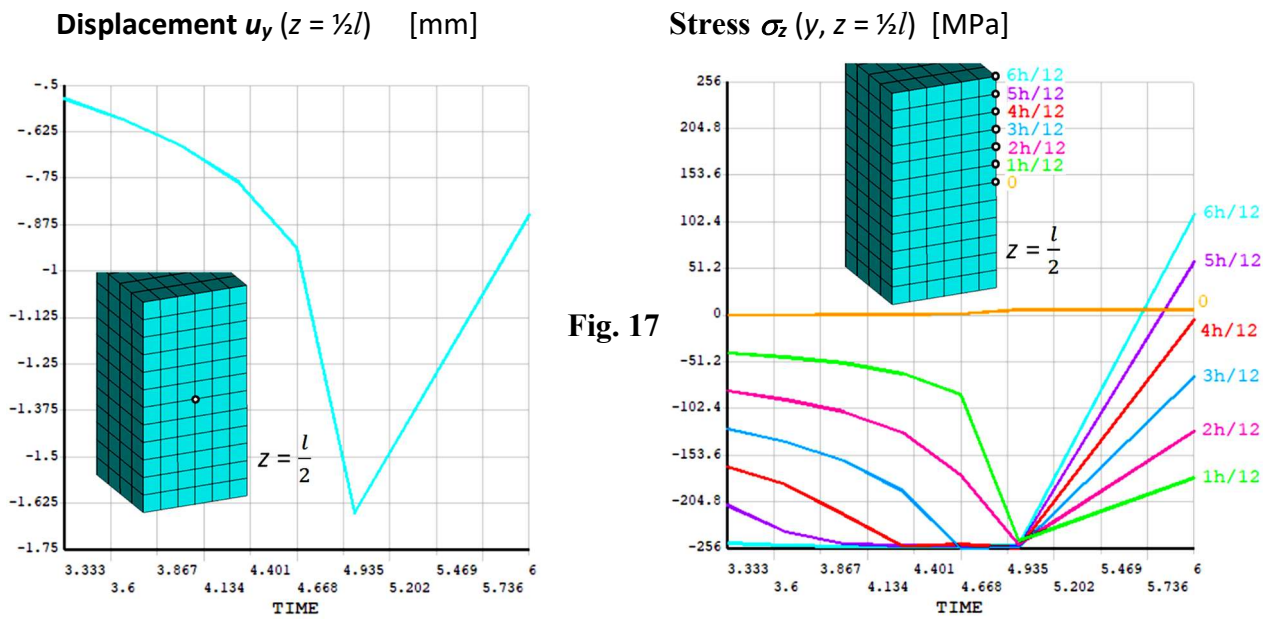


Fig. 17

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

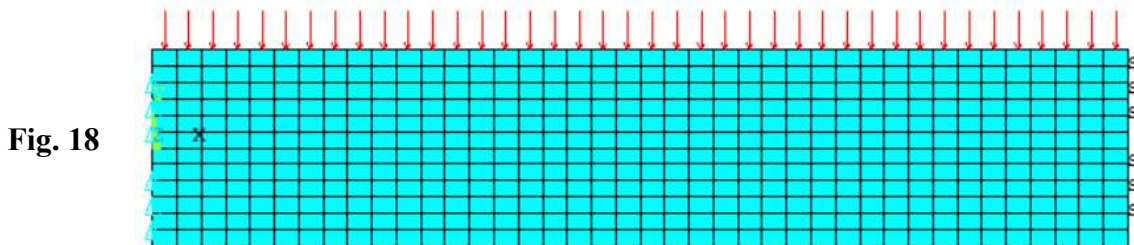


Fig. 18

16. Discuss results and write conclusions.