SE1025 FEM: Homework assignment 3, Per. 3, vt2013

Information

The deadline of HW3 is Tuesday, March 12, 6pm. A total of 5 points can be obtained on HW3.

Problem (5 points)

If we consider bending deformation in the *x*-*z* plane of a straight beam extending in the *x*-direction, an applied bending moment, M_y, will give rise to a normal stress that varies linearly over the cross section as $\sigma_x = (M_y/I_y)z$, where I_y is the area moment of inertia. Likewise, a shear force, *T*, acting on the beam should give rise to a shear stress. From equilibrium considerations, we may draw several conclusions. Firstly, the shear stress must on the average be equal to $\tau_{xz} = T/A$, where *A* is the cross-sectional area. Secondly, since the upper and lower surfaces of the beam are shear traction free, the shear stress must be equal to zero there. Hence, if we consider a simple rectangular cross-section of height *h* and width *b*, we may as a first approximation assume that the shear stress will exhibit a quadratic variation, i.e. $\tau_{xz} = C\left[1-(2z/h)^2\right]$, where C is a constant that is determined such that the average shear stress over the cross section becomes $\tau_{xz} = T/(bh)$. Indeed, this is the solution that is predicted based on Euler-Bernoulli's beam theory. In this homework assignment, you will use the FEM program Ansys to investigate the shear stress distribution over a certain cross section in detail.

A suitable model problem to study this phenomenon is illustrated in the figure below, which shows a cantilever beam of length *L* subjected to a shear force *P* acting on its right end. Due to symmetry, it is sufficient to model only a symmetry half of the beam, for instance, the half where $y \geq 0$, and then applying symmetry boundary conditions on the surface $y = 0$. How should such boundary conditions be formulated? Use Ansys Workbench for the numerical analysis and carry out the computations with two different meshes, employing 20-node hexahedral elements. Use 6 elements in the *z*-direction in the coarser mesh and 12 elements in the *z*-direction in the more refined mesh. Report your analysis and results as suggested below. Morever, summarize your results using the "front end page" appended below.

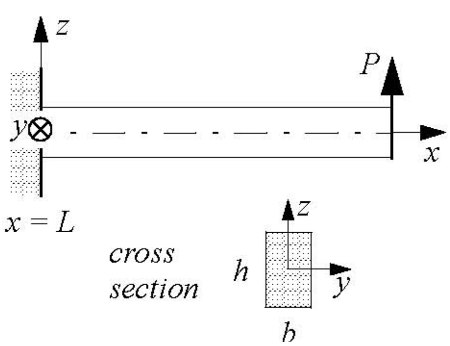


Table 1: *mm* = month

mm	*h*/mm	*b*/mm
01 –03	8	12
04 –06	10	10
07 –09	12	8
10 –12	14	6

Table 2. *dd* = day

dd	*P*/kN
01 –07	8
08 –15	10
16 –22	12
23 –31	14

(a) Describe the FEM-models you have analyzed (define all displacement boundary conditions, external loads, number of elements, type of element, etc.).

(b) Present the results in graphs, where the shear stress component τ_{xz} is plotted versus the *z*-coordinate along the line at $x = L/2$, $y = 0$. How large is the relative difference between the solutions from the two meshes? Also, compare these solutions with the solution based on Euler-Bernoulli beam theory.

Use the following values: Elasticity modulus E = 200 GPa, Poisson's ratio ν = 0.3, L = 100 mm. The values for *P*, *b* and *h* are given in Table 1 and 2, and depend on the "personal identity number" used on the "front end page" appended below.

Ansys Workbench—Guidelines

The guidelines given below should be seen as a complement to *"ANSYS Workbench—a short guideline"* used for computer workshop 2, which gives a more thorough description of how Ansys Workbench is working. How to generate the FEM model of the cantilever beam example above and how to evaluate the results are briefly outlined below.

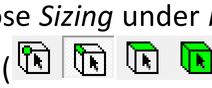
1 Generate a symmetry part of the rectangular shaped cross section in the *yz*-plane by using the modes: *sketching* and *modeling* in *DesignModeler*. (Use a symmetric model with symmetry surface at $y = 0$.) The beam can now be created by use of the extrude command as described in "*ANSYS Workbench – a short guideline".*

2 As stated above, the results from the FEM analysis should be evaluated along the line: $\{x = L/2,\ y = 0,\ -h/2 \le z \le h/2\}$. Create the line as follows. Create a sketch in the *xz-plane*; make a long horizontal straight line located at the middle of the beam. Then, go back to the modeling-mode, choose '*Extrude'*, go to the 'Details of the Extrude' window and make the following choices: '*operation = imprint faces'*; '*Extent Type = Through All'*; '*As thin surface = yes'* with the choices '*Inward Thickness = 0' and* '*Outward Thickness = 0'.* End by clicking on '*Generate*'. The desired line should now be defined.

3 Leave *DesignModeler* and go back to *Project Schematic.* Save the project and choose *Model*. Then boundary conditions can be defined and results simulated.

4 Click on *Static Structural* in the *Outline* menu. Define displacement boundary conditions by use of the *Supports*-menu.

5 Define external loads by use of the *Loads*-menu: choose *Force*, and pick *Define By = defined by components* in the *Details*-menu, and type in the force component value with a consistent unit. Leading question: how much of the total force P should be defined on a symmetry half?

6 Generate the mesh (elements). This can be done as follows. Choose *Sizing* under *Mesh Control*; make sure that you have activated the *'Edge picking'* mode (). Under the '*Details of "Sizing"':* pick appropriate edges at *Geometry*; choose *Type = Number of Divisions,* and give the required element count. Click *'Generate Mesh'* when you are ready.

7 Start the FEM-analysis (solve the equation system) by clicking on *Solve*. Notice! Ansys will not calculate the stresses or strains by default, and a target result has to be specified. Such a target result could be the shear stress, which can be specified by choosing *Shear stress* under the *Stress*-menu.

8 Evaluation of the results: click on *Shear Stress* from *Solution*, then choose *Orientation = xz-plane or xy-plane* in the '*Details of "Shear Stress"*' menu. Continue and choose *Geometry*, click on the line you want to check and then press *apply*. The graph of τ_{xz} versus the position on the predefined line can be obtained by right clicking *Shear stress* under *Solution*, and choose *Export* (you may have to click on *Evaluate results* first). To get the *z*-coordinates of the line, choose *Include node locations = yes* under the menu *Tools/Options/Export*. After that, your results will be saved as a *.txt file and you can use Excel or Matlab to plot it.

Front endpaper - HW 3: FEM for engineering applications

Name and personal number (1):

Name and personal number (2):

Name and personal number (3):

Personal number used to determine

parameters of the problem:

Summary of your results:

Maximum of the shear stress along the line {x=L/2, y=0,-h/2≤z≤h/2}:

Euler-Bernoulli beam theory: $(\tau_{xz})_{max}$ =

FEM-analysis (6 element in *z*-dir.): $(\tau_{xz})_{max}$ =

FEM-analysis (12 element in *z*-dir.): $(\tau_{xz})_{max}$ =