



EN1750: Advanced Mechanics of Solids

Homework 2: Tensor descriptions of deformation; index notation.

Due Friday Sept 27, 2019

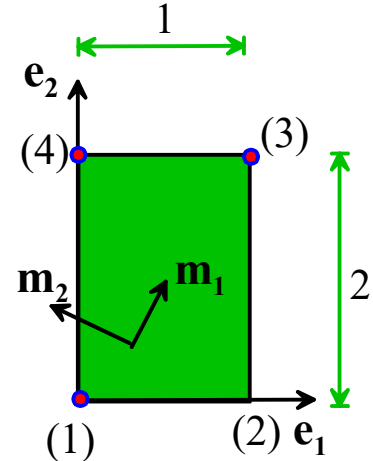
1. The figure shows a rectangular (2D) finite element. The displacement vector (as Cartesian components in the $\{e_1, e_2\}$ basis, in arbitrary units) at each of its corners is

$$\mathbf{u}^{(1)} = [0, 0]; \quad \mathbf{u}^{(1)} = \mathbf{u}^{(2)} = \mathbf{u}^{(4)} = [0, 0]; \quad \mathbf{u}^{(3)} = [0.2, 0.4];$$

The displacement at an arbitrary point inside the element is computed using a linear interpolation between values at the four corners (this only works for the rectangle shown, the general formula for an arbitrary rectangular finite element is more complicated)

$$\mathbf{u} = \frac{1}{2}(2 - x_2)(1 - x_1)\mathbf{u}^{(1)} + \frac{1}{2}(2 - x_2)x_1\mathbf{u}^{(2)} + \frac{1}{2}x_1x_2\mathbf{u}^{(3)} + \frac{1}{2}(1 - x_1)x_2\mathbf{u}^{(4)}$$

where \mathbf{x} is the position of a point in the element before deformation.



1.1 Find the components of $\nabla \mathbf{u}$ (give the answer as a 2x2 matrix, which will be a function of position) and

hence calculate the infinitesimal strain tensor $\boldsymbol{\varepsilon} = \frac{1}{2} \{ \nabla \mathbf{u} + (\nabla \mathbf{u})^T \}$

1.2 Find the components of the deformation gradient (in 2D) $\mathbf{F} = \mathbf{I} + \nabla \mathbf{u}$ and hence deduce the components of the Lagrange strain tensor $\mathbf{E} = (\mathbf{F}^T \mathbf{F} - \mathbf{I}) / 2$

1.2 Find

$$\int_A \nabla \cdot \mathbf{u} dA$$

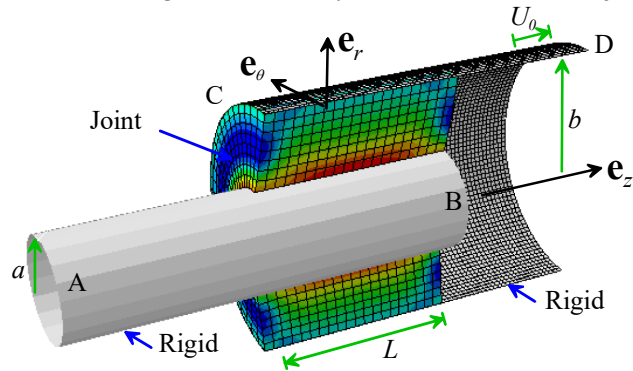
where A denotes the area of the square. Check your answer using the divergence theorem; i.e. calculate

$$\int_C \mathbf{n} \cdot \mathbf{u} ds$$

around the perimeter of the square

1.3 Let $\{m_1, m_2\}$ be unit vectors parallel and perpendicular to a line from corner (1) to corner (3) of the element. Calculate the components of $\nabla \mathbf{u}$ in this basis (use the basis change formulas for the tensor). Check your answer by calculating $\nabla \cdot \mathbf{u}$ using the components in $\{m_1, m_2\}$ (the divergence should be the same in all bases – it is said to be an ‘invariant’ of the tensor).

2. The figure shows a cross-section through a joint connecting two hollow cylindrical shafts. The joint is a hollow cylinder with external radius b and internal radius a . It is bonded to the two rigid shafts AB and CD. Shaft AB is fixed (no translation or rotation), and an axial displacement $\mathbf{u} = U_0 \mathbf{e}_z$ is applied to the hollow cylinder CD.



The displacement field in the joint will be shown (in a later HW) to be approximately

$$\mathbf{u} = \left[\frac{U_0}{\log(b/a)} \log(r/a) \right] \mathbf{e}_z$$

2.1 Find (i) the displacement gradient $\nabla \mathbf{u}$; (ii) the deformation gradient $\mathbf{F} = \mathbf{I} + \nabla \mathbf{u}$ (iii) the Lagrange strain tensor $\mathbf{E} = (\mathbf{F}^T \mathbf{F} - \mathbf{I}) / 2$ and (iv) the infinitesimal strain tensor field

$\boldsymbol{\varepsilon} = \frac{1}{2} \{ \nabla \mathbf{u} + (\nabla \mathbf{u})^T \}$ in the joint, expressing your answer as components in the cylindrical-polar basis shown in the figure (3x3 matrices or you can use dyadic notation if you prefer)

3. Let $\mathbf{a}, \mathbf{b}, \mathbf{c}$ be 3D vectors, and $\mathbf{A}, \mathbf{B}, \mathbf{C}$ be tensors. Write the following operations using index notation

3.1 $(\mathbf{a} \cdot \mathbf{b}) \mathbf{c}$


3.2 $\mathbf{c} = \mathbf{A}^T \mathbf{b}$

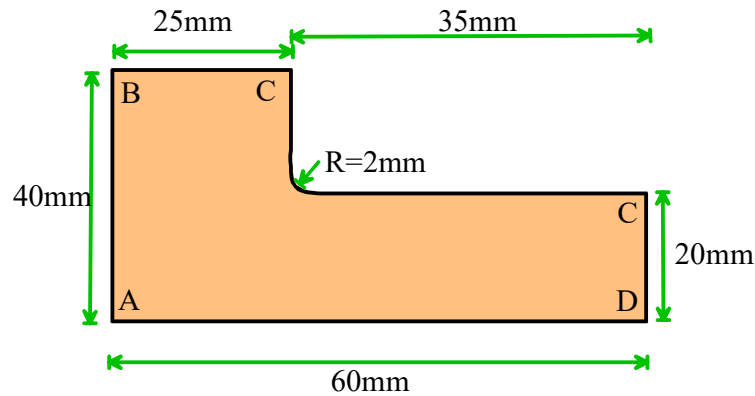
3.3 $\mathbf{a} \cdot \mathbf{B} \mathbf{C} \mathbf{b}$

4. Write out the 3x3 matrix corresponding to $\frac{\partial x_i}{\partial x_j}$

5. Write out in full the three equations represented by $\frac{\partial \sigma_{ij}}{\partial x_i} + b_j = 0$

6. FEA Mesh Design: The objective of this problem is to investigate the influence of element size on the FEA predictions of stresses near a stress concentration. Follow these steps to set up the problem in ABAQUS/CAE

1. Create the 2D part shown below (select the 2D button when creating the part, and make the little rounded radius by using the fillet radius tool  after you have drawn the profile with straight lines everywhere. Enter a radius of 2mm for the fillet radius)



2. Use a linear elastic constitutive equation with $E = 210\text{GNm}^{-2}$, $\nu = 0.3$ (use units of N for force and mm for length; this means that any quantities with stress units should be entered as N/mm^2); create a 2D homogeneous section and assign it to the part.
3. Create an instance of the part in the Assembly module
4. Create a General Static step with time period 1 and NLGEOM off (the defaults)
5. Skip the Interaction module
6. For boundary conditions, use zero horizontal displacement on AB, zero vertical displacement on AD, and apply a uniform horizontal displacement of 0.01mm on CD.

Run computations with the following meshes:

1. Seed the part using the default spacing; then mesh the instance with default mesh. Note that by default, ABAQUS will generate *plane stress* elements in the part (so it is assumed to be a thin sheet). The element type will be quadrilaterals, and ABAQUS will calculate the displacement field using a linear interpolation within each element. If you want plane strain elements, or a quadratic interpolation scheme, you have to use Mesh > Element Type to change the type of the element. If you want to change the element shape, you have to use Mesh > Controls and then select the shape you want. We will explore the different element types in more detail in next weeks HW (I use the term ‘we’ loosely – I will not be doing very much of the work). Once you have generated the mesh, create a job and then run it, and look at the contours of von Mises stress. Find the maximum value of Von Mises stress and make a note of it. (You can use Options > Contour, select the ‘Limits’ tab and check the ‘Show location’ box to have ABAQUS tell you what the max von Mises stress is, and where it occurs). You will change the mesh in the steps below, run the simulation again, and make a table showing the predicted maximum stress for each mesh.
2. Seed the part using a mesh size of 1mm; and mesh with the default mesh
3. Seed the part using a mesh size of 0.5mm; and mesh with the default mesh

4. Seed using a mesh size of 0.25mm; and mesh with the default mesh
5. Seed using a mesh size of 0.125mm and mesh with the default mesh. (this will have around 100000 elements and may take some time to run). If the mesh generator fails, use Mesh > Controls to select the 'medial axis' mesh generation method instead of the 'advancing front' method.
6. Seed the instance using a mesh size of 0.5mm, but mesh the part with quadratic elements. To select quadratic elements, go to the Mesh>Element type and check the Quadratic button
7. Seed the instance with a 0.25mm mesh size and mesh with quadratic elements.

For each mesh, calculate the maximum von Mises stress in the solid. Display your results in a table showing the max. stress, element type and mesh size.

Clearly, proper mesh design is critical to get accurate numbers out of FEA computations. As a rough rule of thumb the element size near a geometric feature should be about 1/5 of the characteristic dimension associated with the feature – in this case the radius of the fillet. If there were a sharp corner instead of a fillet radius, you would find that the stresses go on increasing indefinitely as the mesh size is reduced (the stresses are theoretically infinite at a sharp corner in an elastic solid).

ABAQUS has an option to automatically refine a mesh in regions of high stress concentration. Automatic mesh generation is not too difficult in 2D, and this process will work with both quadrilateral and triangular elements. It's much harder to generate meshes automatically in 3D, and the method only works with tetrahedral elements (it's important to use quadratic tetrahedra, as linear tets don't generally give very accurate solutions).

To try automatic remeshing, follow these steps:

1. In the Mesh module, seed the part with a 1mm mesh size, and assign a linear element type to the part (Mesh > Element type, etc). Then mesh the part.
2. In Mesh > Controls make sure the 'Advancing Front' option is checked, and the 'Element Shape' is 'quad dominated'
3. Still in the Mesh module select Adaptivity> Remeshing Rule> Create, then select the part in the viewport, and select Done. Then check the Mises equivalent stress box in the remeshing rule window (you can leave the element energy box checked as well). In the 'Constraints' tab select 0.01 for the minimum element size (the analysis should work with 0.005 as well)
4. In the Job module select Adaptivity > Create, and accept all the defaults (you can change the number of times ABAQUS attempts to generate new meshes in the menu, which is sometimes useful).
5. Select Adaptivity > submit. There is no 'monitor' option for an adaptivity analysis but ABAQUS will print messages in below the viewport, and you'll see the new meshes being generated.
6. To see the results, go to the Visualization module and select File > Open (in the instructional facility you may need to navigate to the D:\Abaqus.cae directory) – you will see 3 or 4 different output databases containing the results with the successive meshes. The one with the highest number is the last one.

As a solution to this problem, please hand in a table showing your calculated V-M stress for each of 1-7, and with the automatic mesh.

[10 POINTS]