NORWEGIAN UNIVERSITY OF SCIENCE AND TECHNOLOGY NTNU
FACULTY OF ENGINEERING
DEPARTMENT OF STRUCTURAL ENGINEERING

TKT4142 Finite Element Methods in Structural Engineering

CASE STUDY 2

SOLUTION PROPOSAL

SLEIPNER A - CONDEEP PLATFORM

Sleipner A. Illustrasjon: Norwegian Contractors

Tasks

In this case study, we will use FEA and a reduced computational model in Abaqus to determine the stresses and the distribution of their resultants in the length direction of the tricell wall. We are interested in the stresses and their resultants in the transition between the tricell walls and the main cell wall.

It is recommended to use 2D plane strain elements with a thickness equal to 1.0, where the loads p_1 and p_2 are modelled as line or edge loads with a unit N/mm instead of N/mm^2. Since we are reducing the 3D physical model to a 2D planar model, we have to account for the 3D model having a large extent in the plane. In such cases, it is recommended to use plane strain elements (CPEn), where n denotes the number of nodes per element.

a) Perform the analyses with 3 different FE discretizations, with 1, 2 and 4 elements over the thickness of the tricell wall. Use both 4-node fully integrated compatible (CPE4) and 8-node (CPE8) plane strain elements to run a total of 6 analyses. Report your model in Abaqus by generating figures of the meshed models.
For each analysis, plot the distribution of the averaged stresses (75%) σ_{xx}^* (S11 in Abaqus) and τ_{xy}^* (S12 in Abaqus) over the end section of the tricell wall (corresponding to $x = 0$ in Figure 3a).
For the finest model with the 8-node plane strain elements, report contour plots of both the unaveraged stresses σ_{xx} (S11 in Abaqus), σ_{yy} (S22 in Abaqus) and τ_{xy} (S12 in Abaqus) and the averaged stresses (75%) σ_{xx}^*, σ_{yy}^* and τ_{xy}^* for the entire model.

Solution:

Use **File** → **Print** to make a *.png, *.ps, *.eps, *.tif, or another format to plot the model and include this in your report.

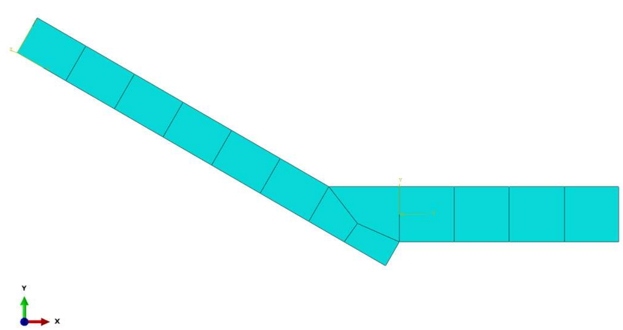

Figure 1: Mesh 1 – with 1 element across tricell wall thickness

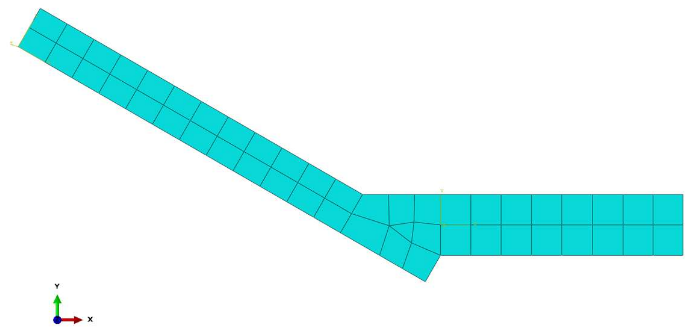

Figure 2: Mesh 2 – with 1 elements across tricell wall thickness.

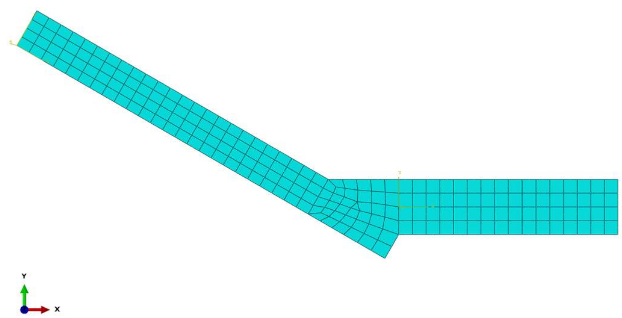

Figure 3: Mesh 3 – with 4 elements across tricell wall thickness.

Plotting the distribution of σ_{xx} (S11 in Abaqus) and τ_{xy} (S12 in Abaqus) over the end section of the tricell wall for all six analyses.

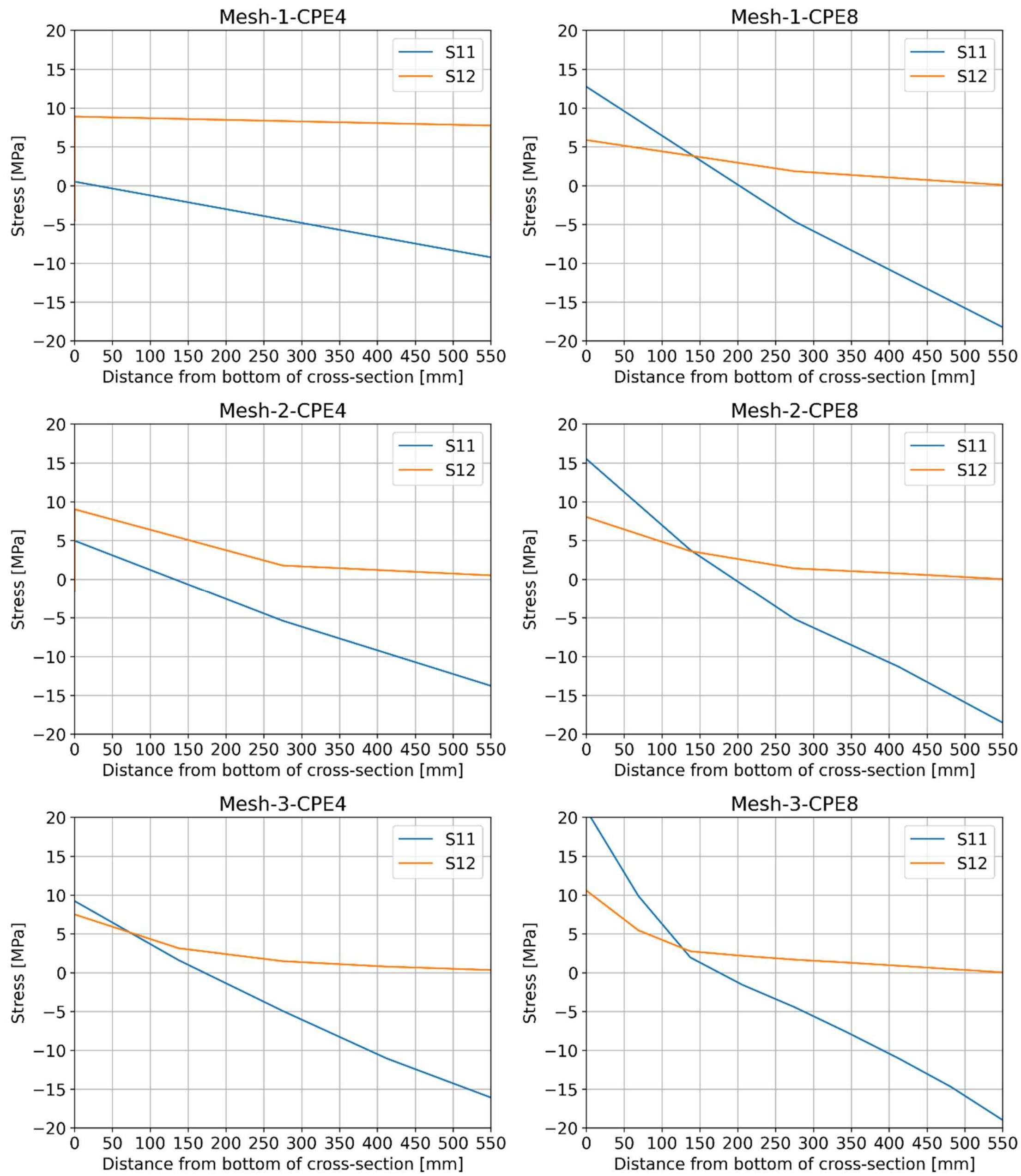

Figure 4: Averaged (75%) plots of S11 (σ_{xx}) and S12 (τ_{xy}) over the end section of the tricell wall for three meshes and two element types for a total of six analyses.

Now, we show six contour plots of unaveraged stresses σ_{xx} (S11), σ_{yy} (S22) and τ_{xy} (S12) and the averaged stresses (75%) σ_{xx}^{*}, σ_{yy}^{*} and τ_{xy}^{*} for the entire model of the finest mesh with CPE8 elements.

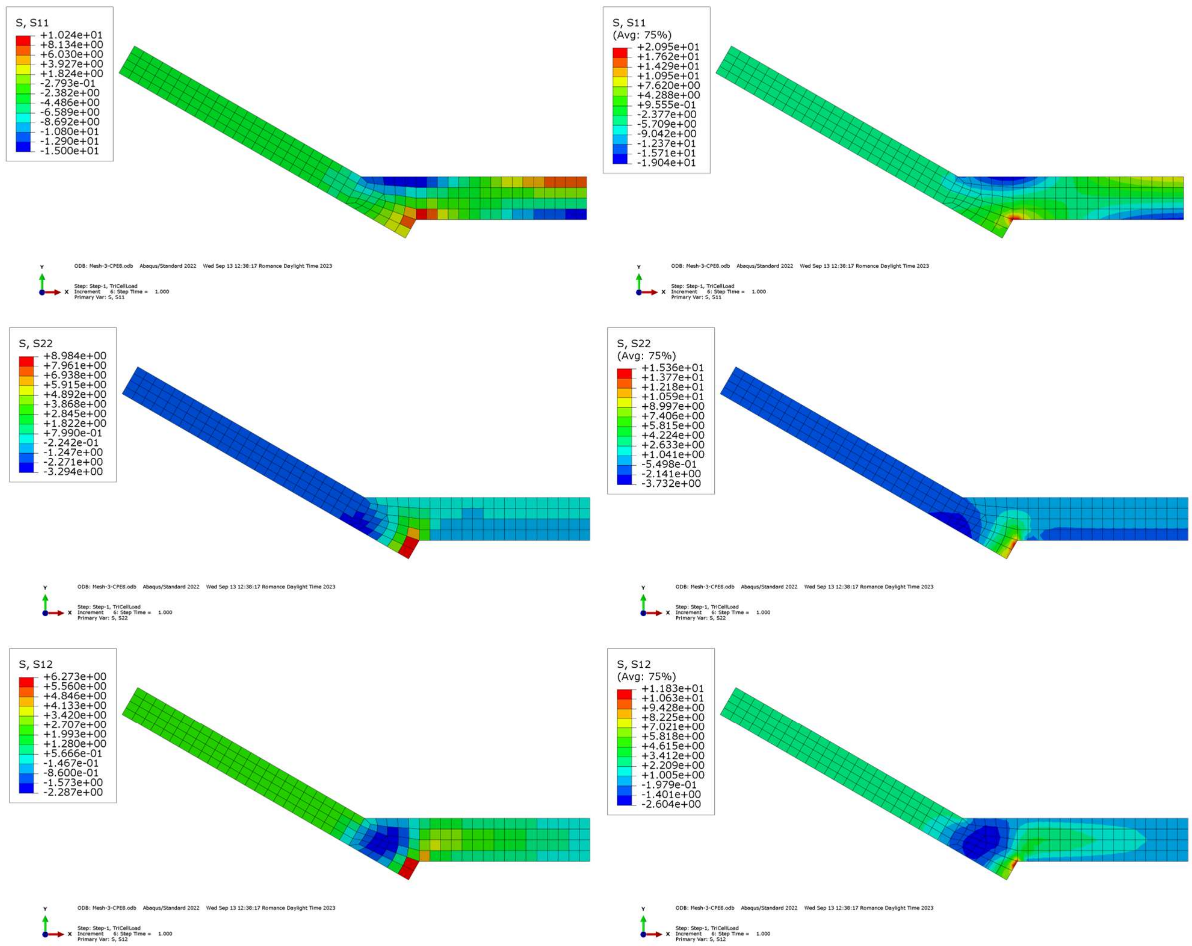

Figure 5: Contour plots of S11 (σ_{xx}), S22 (σ_{yy}) and S12 (τ_{xy}) for the entire part for mesh 3 with CPE8 elements. Left plots are unaveraged, while right plots are averaged with a value of 75%.

b) We will now use the FEA results to obtain stress resultants and corresponding forces. For each of the analyses carried out in a), the stresses σ_{xx} (S11 in Abaqus) and τ_{xy} (S12 in Abaqus) should be integrated to obtain their corresponding stress resultants axial force (N), shear force (V), and bending moment (M) at the end section of the tricell wall ($x = 0$ in Figure 3).
Compare the distribution of the moment (M) and the shear force (V) along the tricell wall for the 6 analyses in 2 diagrams (one for the moment and one for the shear force).

Solution:
Shear and moment diagram comparisons.

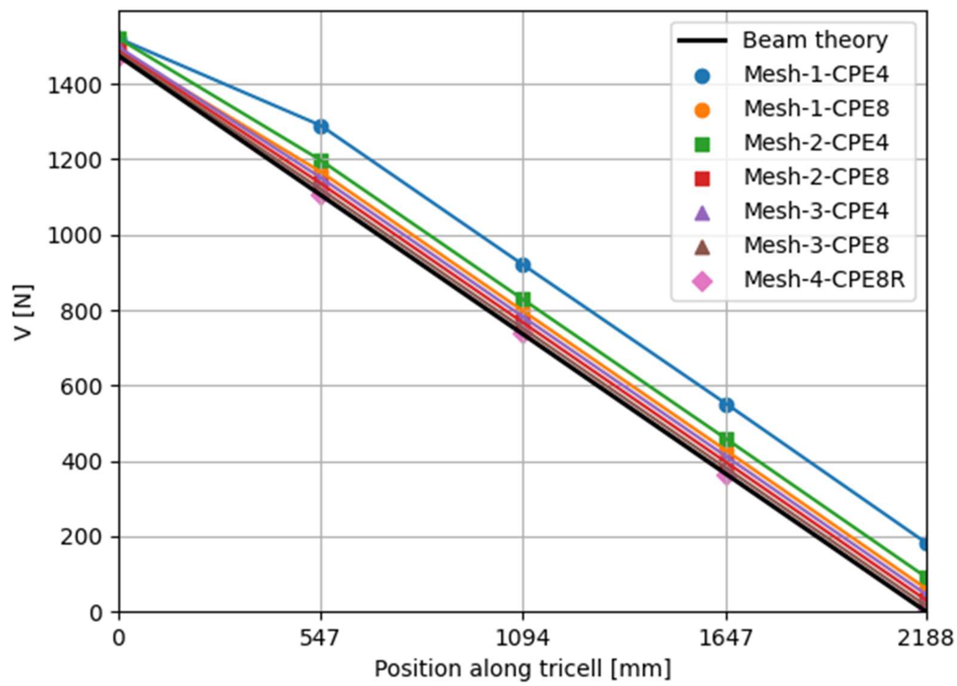

Figure 6: Shear diagram along tricell wall (SOF2 in Abaqus).

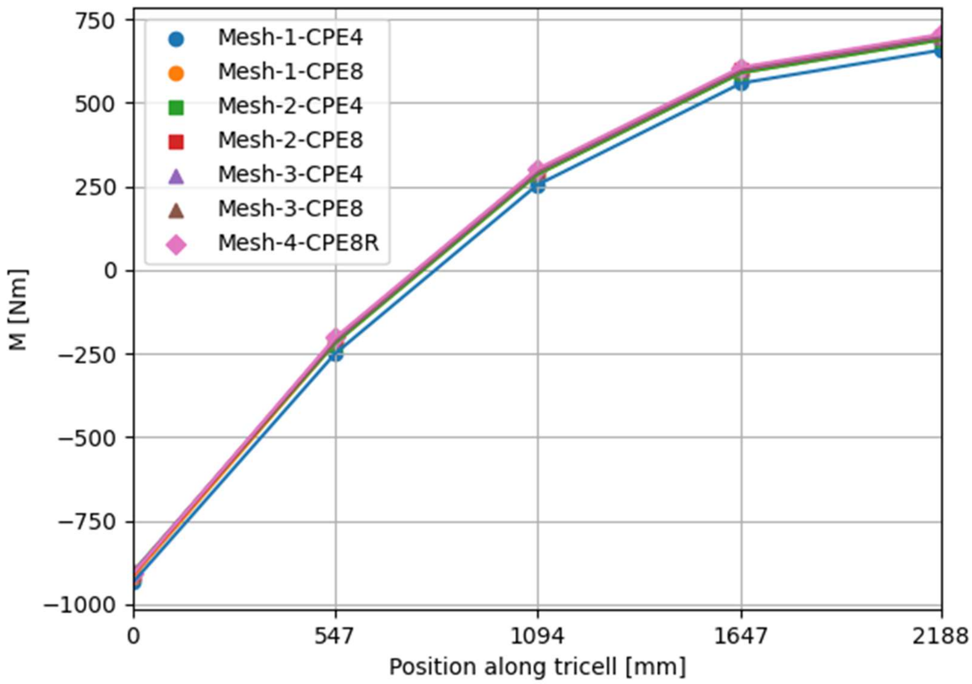

Figure 7: Moment diagram along tricell wall (SOM3 in Abaqus).

Figures 6 and 7 show the shear and moment diagrams of the tricell wall for different combinations of mesh and element type compared to the reference solution (Mesh-4-CPE8R). The results are found by using the sum of forces (SOF) and the sum of moments (SOM) in Abaqus in five different sections/surfaces along the tricell wall. Apart from the shear force at the end section ($x = 0$) where in particular the solution obtained for the coarsest mesh with CPE4 deviates somewhat from that obtained for the equilibrium considerations, the shear force distribution for all discretizations converges towards the refined reference solution. Figure 7 also shows that the moment distribution quickly converges towards the refined reference solution. Note that the shear force at $x = 2188$ mm is zero for the beam theory solution.

A note on SOM and SOF:

Using SOF and SOM, the stress resultants (M, V and N) are not calculated from the element stresses σ nor the smoothed stresses σ^*, but on the basis of the complementary part of the element nodal forces (NFORC in Abaqus), which expresses the internal forces in the element due to element nodal displacements $\mathbf{v}$. This procedure provides more accurate values for the resultants compared to integrating the element stresses over the thickness. This is because the FE solution represented by the internal nodal forces will always satisfy nodal point equilibrium with the externally applied forces exactly. We will now examine this:

We consider the model with Mesh 1 and element type CPE4, and we can plot the stress distribution on the end of the tricell wall, shown in Figure 8.

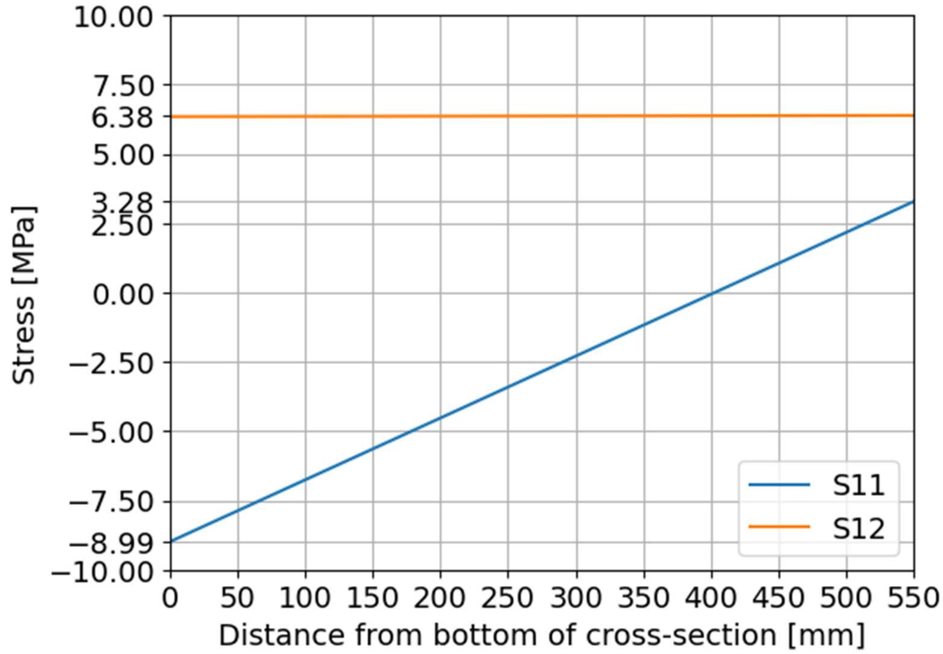

Figure 8: S11 and S12 plotted over the end of the tricell wall (x=2188 mm).

We can now integrate the element stresses to obtain the stress resultants for $x = 2188$ mm.

$$N = \int_A \sigma_{xx}\, dA = \frac{550}{2}(3.200 - 8.908) = -1570 \text{ N}$$

$$V = \int_A \tau_{xy}\, dA = \frac{550}{2}(6.335 + 6.377) = 3496 \text{ N}$$

$$M = \int_A \sigma_{xx} z\, dA = \frac{550^2}{12}(3.200 + 8.908) = 305.2 \text{ Nm}$$

We can now probe the internal forces for the element on the right edge (Element 4). We then get the values presented in Table 1.

Element	Node	Nodal forces	
		$NFORC1$	$NFORC2$
4	2	237.249	190.077
	1	-1795.3	-7.75986
	9	1974.64	-182.317
	10	-416.593	0

Table 1: Nodal forces (NFORC in Abaqus) in Element 4 for Mesh 1 with CPE4.

Table 1 shows the internal forces NFORC ($\mathbf{S} - \mathbf{S^0} = \mathbf{kv}$) for Element 4. This results in the following stress resultants for this element:

$$N = -[(NFORC1)_4^9 + (NFORC1)_4^{10}] = -[1974.64 - 416.593] = -1558 \text{ N}$$

$$V = -[(NFORC2)_4^9 + (NFORC2)_4^{10}] = 182.317 + 0 = 182 \text{ N}$$

$$M = \frac{550}{2}[(NFORC1)_4^9 - (NFORC1)_4^{10}] = \frac{550}{2}[1974.64 + 416.593] = 675.6 \text{ Nm}$$

where the subindex denotes the element number, and the superscript denotes the node number. V and M are in accordance with SOF and SOM at $x = 2188$ mm in Figures 6 and 7. However, the stress resultants (V and M) obtained by integrating the stresses in the element (Element 4) are inaccurate.

We know that the FE solution fulfils nodal point equilibrium and element equilibrium. Hence, SOF and SOM ought to be good bases to find the stress resultants. Stresses are related to strains, which are the first partial derivates of the displacements. Due to the differentiation, accuracy is lost.

With finer meshes, the stress distribution will be more accurate, and the two methods will be in better agreement.

c) Compare the results in b) with hand calculations where the tricell wall is considered to be a straight, two-sided clamped beam with length $l = 4376$ mm.

Solution:
As a control, the results of the finite element analyses will be compared with the results obtained from simple equilibrium considerations where the tricell wall is represented by a two-sided clamped beam with length 4376 mm subjected to a uniformly distributed load $p_1 = 0.674$ MPa. At the clamped end ($x = 0$), we get the following shear force (V), bending moments and the corresponding stresses:

$V = \frac{p_1 L}{2} = 1475$ N (per mm in the thickness direction. $\Rightarrow \bar{\tau}_{xy} = \frac{V}{t} = \frac{1475}{550} = 2.68$ MPa
$M = \frac{p_1 L^2}{12} = 1.08 \cdot 10^6$ Nmm (per mm in the th. dir.) $\Rightarrow \bar{\sigma}_{xx} = \frac{M}{I} \cdot \frac{t}{2} = \frac{1.08 \cdot 10^6}{\frac{550^3}{12}} \cdot \frac{550}{2} = 21.33$ MPa

The calculated V was used to plot the "Elementary beam theory" solution presented in Figure 6. From force equilibrium, we obtain the axial force N and average axial stress $\bar{\sigma}_{xx}^N$ in the tricell wall:

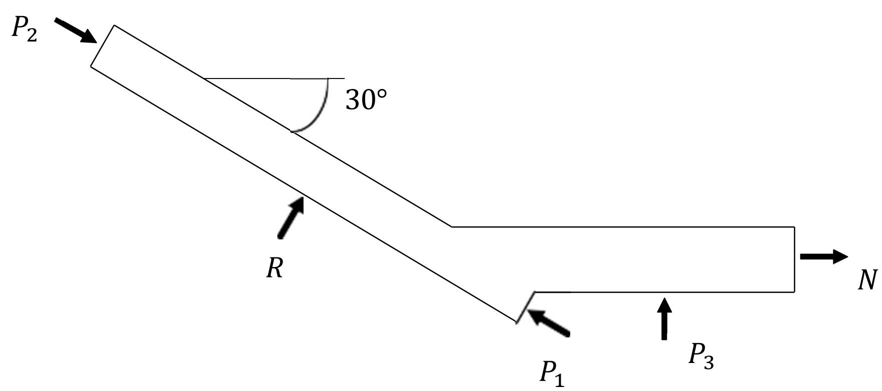

Figure 8: Free body diagram of simplified model for hand calculations.

$$P_1 = p_1 \cdot 275 = 0.674 \cdot 275 = 185 \text{ N}$$
$$P_2 = p_2 \cdot 400 = 5.68 \cdot 400 = 2272 \text{ N}$$
$$P_3 = p_1 \cdot \frac{L}{2} = 0.674 \cdot 2188 = 1475 \text{ N}$$

$$\left.\begin{aligned} \Sigma F_x &= 0: (P_2 - P_1)\cos 30° + R \cdot \sin 30° + N = 0 \\ \Sigma F_y &= 0: (P_1 - P_2)\sin 30° + R \cdot \cos 30° + P_3 = 0 \end{aligned}\right\} \Rightarrow \begin{aligned} N &= \frac{P_2 - P_1 - R \cdot \sin 30°}{\cos 30°} = 1558 \text{ N (per mm)} \\ \bar{\sigma}_{xx}^N &= \frac{N}{t} = \frac{-1558}{550} = -2.83 \text{ MPa (pressure)} \end{aligned}$$

d) Elaborate on the physical characteristics of plane strain states. Can you give examples on situations where these assumptions are valid? Why is plane strain useful for the structure we have examined in this case study? Are there some limitations to the assumption of plane strain in this case study?

Solution:
Plane strain states are defined by $\varepsilon_z = \gamma_{yz} = \gamma_{xz} = 0$. Appears in very thick (or very long) structures, where the thickness (in the z-direction) is much larger than the dimensions in the x- and y-directions and the geometry, load and boundary conditions must not be a function of the z-coordinate. Some examples include: Embankment dam (norsk: fyllingsdam), retaining walls (norsk: støttemur) and concrete tunnel wall (norsk: tunnelvegg i betong). We recognise some of these features in the structure in this case study; we have a structure that has large thickness, much larger than the dimensions in the x- and y-directions. Even though we know that a water pressure load varies linearly with the depth of the water, we simplify the load to be a linear pressure along the edges of the 2D planar computational model.

e) Figure 4 shows the von Mises stress for a reference model using a very fine element mesh ($h = 5$ mm $\Rightarrow N_{\text{els}} = 184575$) using CPE8R elements, where the range for the von Mises stress has been set to 0-30 MPa. Figure 9 shows the stress distribution for S11 over the end section of the tricell wall for the reference solution. Elaborate on what happens at the bottom of the end section ($x = 0$ in Figure 10). Is this a realistic result or a feature of the computational model? Why does the FEA not give a linear distribution of the σ_{xx} stress component as we would expect from elementary beam theory? What is σ_{yy} according to elementary beam theory?

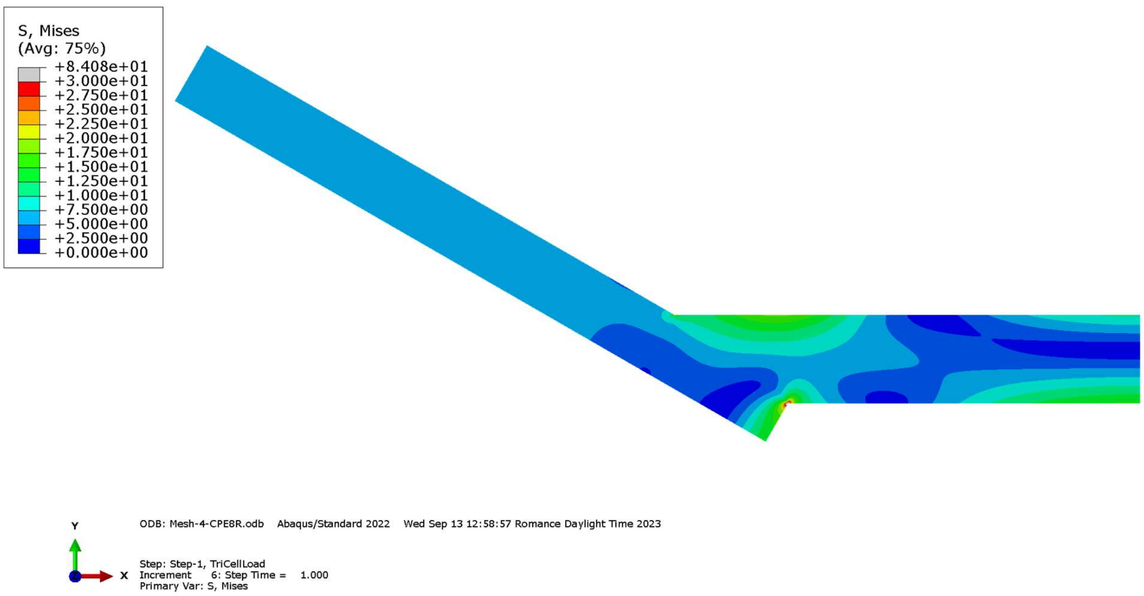

Figure 9: von Mises stress in reference solution.

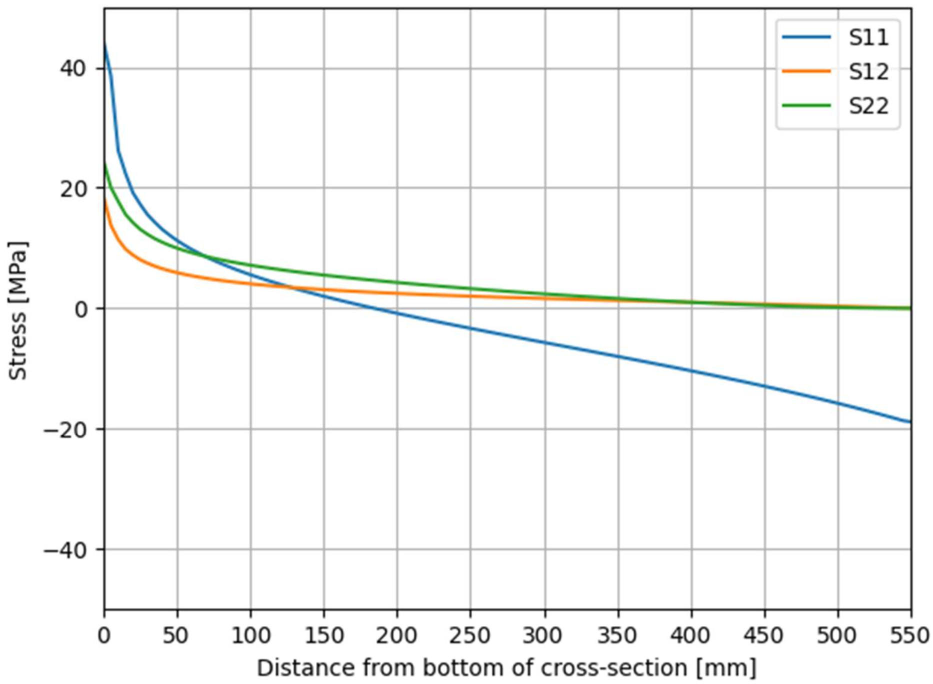

Figure 10: S11, S22 and S12 plotted over the end of the tricell wall in reference solution.

Solution:
We have a stress concentration at the bottom of the cross-section. This is a limitation of our computational model, where the stresses will increase in the kink if we decrease the element size. The stress state (S11, S22 and S12) at $x = 0$ is a result of a stress singularity at this point due to change in geometry (i.e., a sharp corner). Sharp corners are corners of a mesh where the external angle is less than 180°. Theoretically, the stress at the singularity is infinite, and does not converge towards a specific value at $x = 0$.

Although the stress at these singularities is infinite, the model results are not incorrect overall. The displacements are correct even at the singularity point. However, the stress at the singularity will pollute the stress results near the singularity, but some distance away from the singularity the stress results will be fine. This is an immediate consequence of St. Venant's Principle.

In reality, no corner can be perfectly sharp. Even if desired, a manufactured sharp corner will always present a small fillet radius. This means that the stress will not be infinite anymore, the corner singularity will disappear. Instead, a stress concentration takes over.