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

TKT4142 Finite Element Methods in Structural Engineering

CASE STUDY 2

SLEIPNER A - CONDEEP PLATFORM

Sleipner A. Illustrasjon: Norwegian Contractors

Introduction

On the 23rd of August 1991, the Sleipner A Condeep platform suffered a catastrophic collapse during lowering into Gandsfjorden, outside Stavanger. When the concrete hull reached a depth of 67 meters, loud noises were heard by the workers. A cell wall had failed, which caused a large crack in the hull. This eventually led to the platform sinking to the bottom of the fjord. We have acquired some old drawings from the design phase of Sleipner A that we will use to investigate parts of the concrete structure.

We will revisit the finite element analysis (FEA) of critical parts of the Sleipner A concrete structure. The purpose of this case study is to become familiar with and gain some experience in the process of creating, analysing, and post-processing the results in terms of stresses to obtain resultants (M, V, and N) of a 2D plane strain problem using Abaqus. A workshop on how to model the different aspects addressed in this case study is uploaded to Blackboard (see "Workshop2.pdf" in the folder "Case studies").

We encourage you to read Chapter 17 about Practical use of FEM in the textbook by Bell[1].

Learning outcome

- Modelling of 2D plane strain problems using Abaqus.
- Integrating stress quantities at Gauss points over the thickness to obtain stress resultants at user-defined surfaces.
- Perform convergence studies and assess which element type and element mesh size provide a sufficient degree of accuracy regarding the maximum von Mises stress in the structure.

Problem description

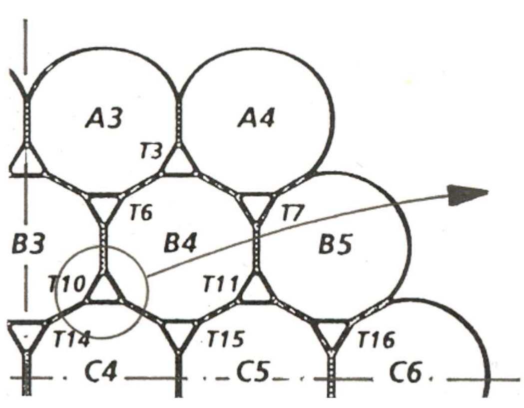

Figure 1a

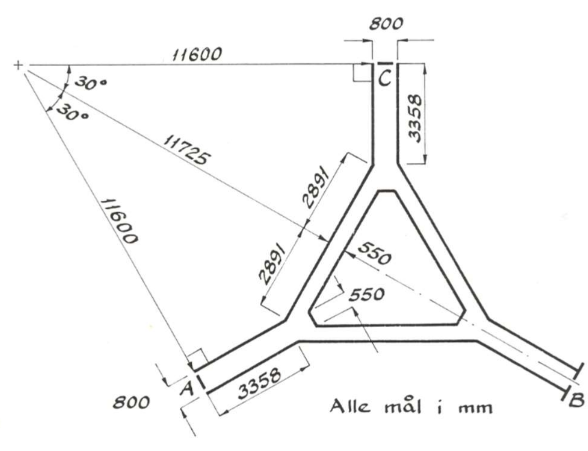

Figure 1b

Figure 1a shows ¼ of a horizontal cut through the bottom structure of the Sleipner A Condeep Platform. We will investigate the stress resultant computed in a so-called "tricell" (see Figure

---

[1] If interested, you can find more information about the Sleipner A platform at https://en.wikipedia.org/wiki/Sleipner_A.

1b) for a given load case where we have an overpressure in the tricell in relation to the main cells. Exploiting symmetry in load and geometry, it is sufficient to consider a reduced computation *plane* model in which the model can be reduced to 1/6 of the tricell shown in Figure 2. Pay attention to the boundary condition for the reduced, symmetric model.

Computational model and geometry

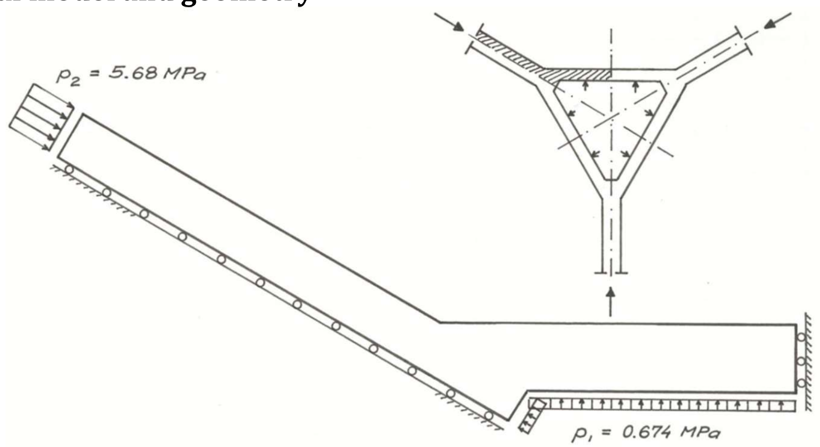

Figure 2a: Computational model of the tricell.

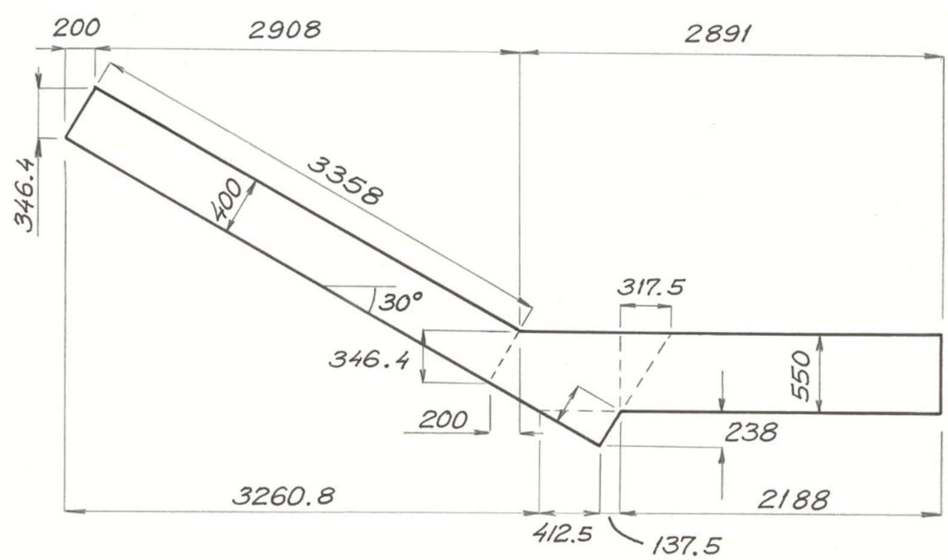

Figure 2b: Geometry of the computational model.

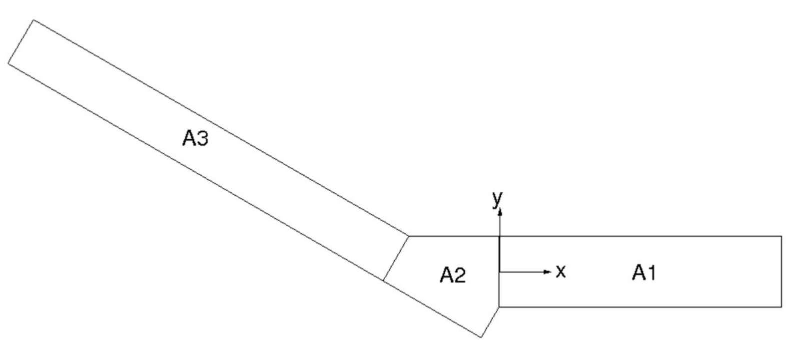

Figure 3: Computational model divided into three partitions with coordinate system.

Material data
The material data for the concrete used in the structure is shown in Table 1.

Density [kg/m^3]	Young's Modulus [MPa]	Poisson's ratio [-]
2500	30000	0.15

Table 1

Load
The load on the structure represents an overpressure p in the tricell corresponding to the 67 m water column (the pressure in the main cells is set equal to 0). The density of the seawater can be assumed equal to 1025 kg/m^3. The external water pressure on the bottom structure imposes a horizontal ring pressure in the main cell walls that is approximately constant in each horizontal section. The resultant of the pressure on the circumferential direction can be set equal to 0.55 *pr,* where r is the mean radius (12.25 m). The resultant forces are applied on Sections A, B, and C in Figure 1b as normal forces in the direction against the center in the tricell. Substituting the information provided above, we obtain the load as shown in Figure 2a:

$$\begin{aligned} p_1 &= \rho g h = 0.674 \text{ MPa} \\ p_2 &= 0.55\, p_1 \frac{r}{w} = 5.68 \text{ MPa} \end{aligned} \quad \text{where} \begin{cases} \rho = 1025 \text{ kg/m}^3 \text{ (sea water)} \\ g = 9.81 \text{ m/s} \\ h = 67 \text{ m} \\ r = 12.25 \text{ m} \\ w = 0.8 \text{ m} \end{cases}$$

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 stresses 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.

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).

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.

d) Elaborate on the physical characteristics of plane strain states. Can you give examples of 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 plane strain in this case study?

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 5 shows the stress distribution for S11 over the end section of the tricell wall ($x = 0$ in Figure 3) for the reference solution. Elaborate on what happens at the bottom of the end section ($x = 0$ in Figure 5). Is this a realistic result or a feature of the computational model? Why does the FEA not predict 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?

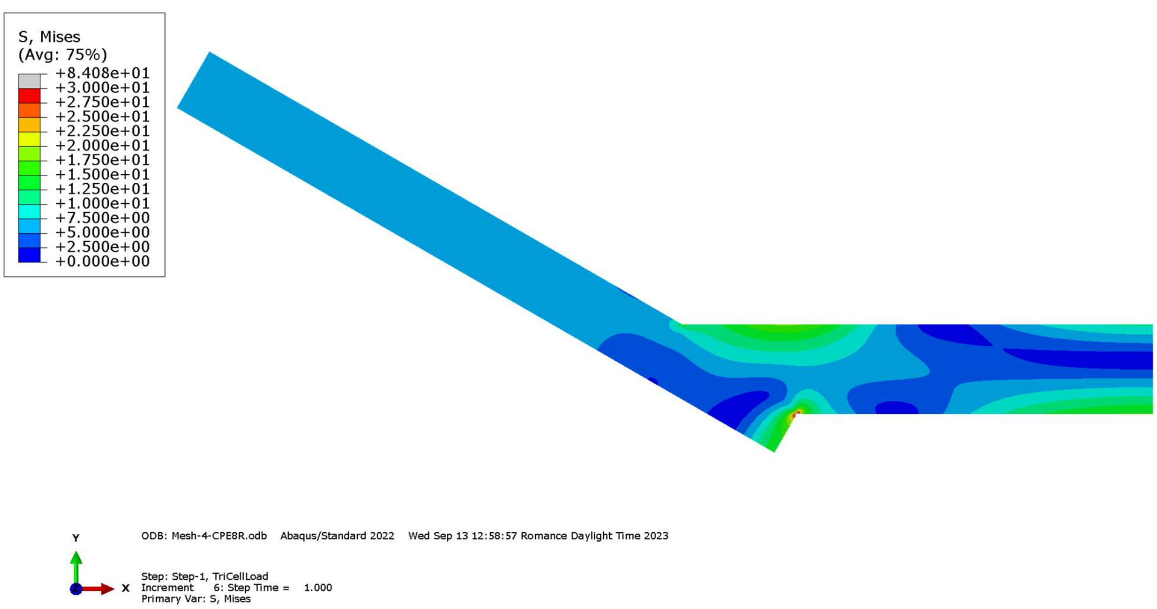

Figure 4: von Mises stress in reference solution.

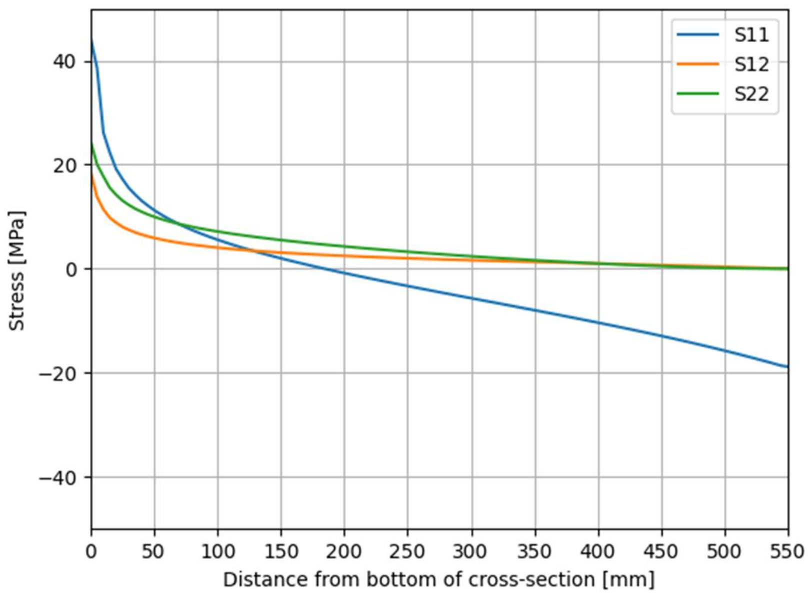

Figure 5: S11, S22 and S12 plotted over the end of the tricell wall in reference solution.