

The dimensions of all structural elements have been modeled as shell elements in Abaqus, as specified in Table 4. The thickness of each element has been defined in the Property Section Manager module.

Table 4 Geometric properties of structural elements in the model

DECK CEN. GIRDER	L200×100×10 mm
DECK SIDE GIRDERS	L200×200×10 mm
DECK TR	L200×100×10 mm
DECK PLATE	10 mm
DECK BEAMS	FB 100×12 mm
SIDE PLATE	10 mm
SIDE TR	L200×100×10 mm
PILLAR	PIPE400×16 & PIPE150×8 mm

To simplify the finite element model, small local features such as cutouts, lugs, scallops, air holes, and snipes were not included in the analysis (Figure 11).

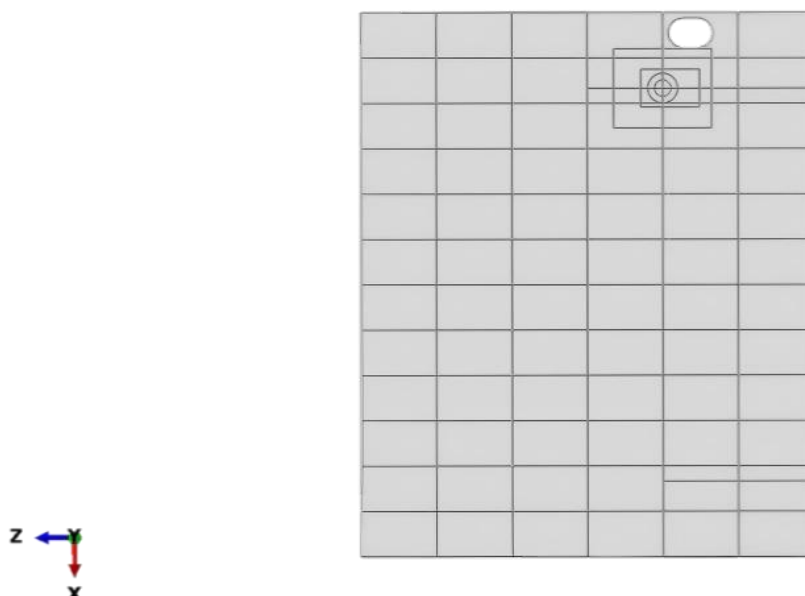


Figure 11 Deck plan showing the layout of longitudinal and transverse reinforcements and columns in Abaqus

## 2.7 Geometric Properties of the Model

The finite element mesh was generated using free meshing with the medial axis algorithm. The mesh consists of both quadrilateral and triangular shell elements, with a global element size of 40 mm. The final model contains 71 996 nodes and 72 259 elements, including 71 822 quadrilateral elements of type S4R and 437 triangular elements of type S3 (Table 5).

Table 5 Mesh properties of the finite element model

Parameter	Specification
Element Shape	Triangular and Quadrilateral
Meshing Algorithm	Medial Axis
Meshing Technique	Free
Global Element Size (mm)	40
Number of Nodes	71,996
Number of Elements	72,259
Number of Quadrilateral Elements (S4R)	71,822
Number of Triangular Elements (S3)	437