

Finite element modelling of plate with stiffeners using FEMAP and ABAQUS

Structure to be modelled

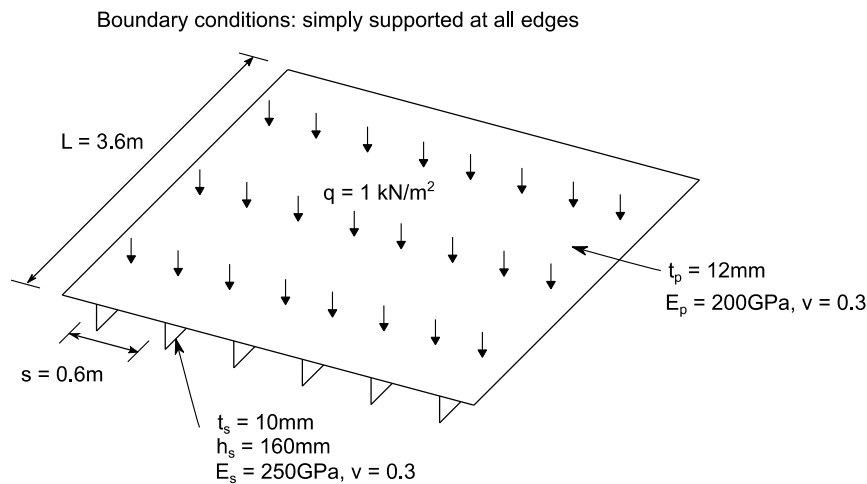


Fig 1. Stiffened panel with arbitrary dimensions

General steps:

- 1- Creating surfaces:** define surfaces that compose the structure; in FEMAP, this can be done, for instance, by defining points that limit the area (*Geometry/Points*) and linking them through the *Geometry/Surface/Corners* command. Draw only one stiffener with half plate spacing and use the *Geometry/Copy/Surface* tab to simplify the work. Note that the structure to be modelled has a symmetry plane, thus only half model should be created.
- 2- Mesh controls:** prepare surfaces to be meshed - define mesh size at each curve or surface under *Mesh/Mesh Control/Size Along Curve* or *Mesh/Mesh Control/Size On Surface*. Assign an arbitrary mesh size for now.
- 3- Material and element properties:** under *Model/Material...* one can define the different material properties used in the model, as well as its type; for static analysis, only E and ν are required. Under *Model/Property...* one can define the element type (under *Element/Property Type...* in this Palette) and properties; as we are dealing with plate elements, only the thickness TI has to be defined. Make sure the units are compatible in these.
- 4- Mesh surfaces:** select surfaces to be meshed under *Mesh/Geometry/Surface...*, create the mesh and check for coincident nodes in their intersections under *Tools/Check/Coincident Nodes....*
- 5- Load definition:** is made under *Model/Load*. One can create pressure load if *Elemental* is selected, or assign a single or multiple degrees of freedom by selecting *Nodal*. Pressure load is transformed internally into nodal equivalents using numerical integration.
- 6- Boundary conditions:** under *Model/Constraint/Nodal*. Either single degrees of freedom or common combinations are available. Assign symmetry boundary conditions at the relevant plane, and simply supported boundary conditions at all four edges. Make sure not to overly constrain the plate edges, allowing the cross-section to rotate.

- 7- **Creating analysis file:** under *Model/Analysis/New*, select *Abaqus* and *Static* to generate an input file. All parameters can be left standard at this stage. Place the input file at the folder C:\SIMULIA\AbaqusTEMP.
- 8- **Solving with Abaqus:** select the Abaqus command prompt on windows, type the command *abacus job = inp_file_name* to solve.
- 9- **Post-processing:** select the .fil file generated under *File/Import/Analysis Results...* Press F5 to select the deformed style and contours. Under *Deformed and Contour Data...* one can define what results to display. These are, for example, normal stresses, stress resultants, etc. To read nodal quantities, it may be useful to create a data table – *List/Output/Results to Data Table...* and transfer them to a spreadsheet such as MS Excel.

Useful commands:

- *File/Rebuild* refreshes the model view in case any visual bug appears
- When selecting nodes to apply loads and boundary conditions, the *prompt Entity Selection – Enter Node(s) to Select* appears. It is useful to go for the command *Pick/Coordinate, Box, Polygon*, etc., or to select from the geometry, e.g. under *Method/on Curve, on Surface*. With practice it becomes automatic to pick the best selection.
- Still when picking a node or surface, right click on the main screen beforehand and check if *Pick Any Inside* or *Pick All Inside* is selected; this will define if a node at the limit of your selection is picked or not.
- Under *Tools/Check/Sum Forces* one can see the total forces/moments along each direction in the model.
- Under *File/Preferences/View/Resolution* one can change the resolution for printing/copying; *File/Picture/Copy* takes a print screen.
- Under F6 one can change the view options, such as text size, color, hide or display certain entities, etc.