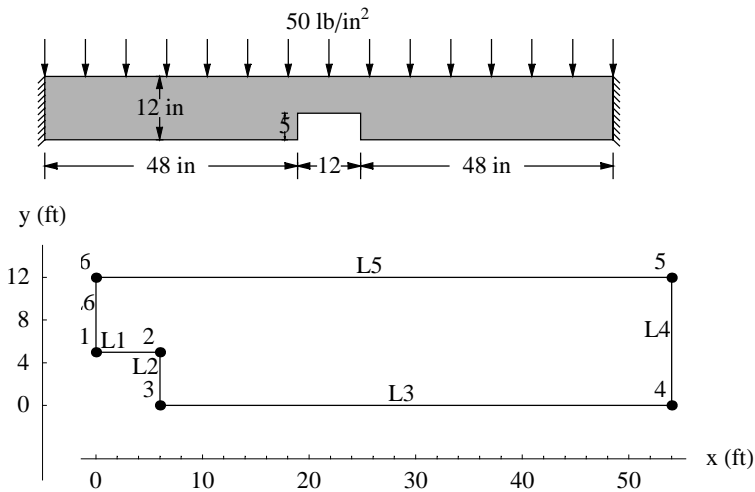


■ A.3.7 Plane Stress Analysis

As a final example consider the problem of finding stresses in a notched beam of rectangular cross-section shown in Figure A.6. The beam is 4 in thick in the direction perpendicular to the plane of paper and is made of concrete with modulus of elasticity $E = 3 \times 10^6 \text{ lb/in}^2$ and poisson's ratio $\nu = 0.2$. The beam is loaded by a uniform pressure of 50 lb/in^2 on the top surface.



Model creation using abaqus cae

1. Part module

The overall geometry is created using the Part module in the abaqus cae system. A model may consist of several different parts. Interactions among these parts are specified in a later module. For this example we create a single part as follows.

Part> Create [Name: Part-1, Modeling space: 2D Planar, Type: Deformable, Base feature: Shell, Approximate size: 60] Continue

Add> Line> Connected lines [Enter x,y coordinates of the corners (Figure A.6)]. Enter return after entering each pair: 0,5; 6,5; 6,0; 54,0; 54,12; 0,12; 0,5. Click on any other icon to get out of line define mode. Click on Done next to 'Sketch the section for planar shell'.

2. Property module

Different material and section properties are created in this module. For this example one material is defined with appropriate modulus of elasticity and Poisson's ratio values.

Material> Create> Material-1> Continue>Mechanical> Elasticity> Elastic [Enter E = 30000000 and ν = 0.2] OK

In order to use this material later, it must be associated with appropriate sections. For this example one section is defined and associated with the material as follows.

Section> Create> Section-1> Solid> Homogeneous> Continue>[Set Material to Material-1 and plane stress/strain thickness to 4] OK

The next task is to actually associate the sections to different areas of the part. In this example there is only one section and one area.

Assign> Section> [Select the area and click Done on the button next to 'Select the regions to be assigned a section']> Section-1> OK

3. Assembly module

Different contact and interface conditions can be created between parts using this module. For this example, we only have a single part and thus the assembly is just the part and is defined as follows.

Instance> Create> Part-1> OK

4. Step module

This module defines the analysis type and sets desired output solution parameters.

Step> Create> [Name: Step-1, Procedure type: General]> Static General> Continue> [Nlgeom: Off]

Nonlinear large displacement effects can be included in the analysis by setting Nlgeom: On. For a conventional small displacement linear analysis Nlgeom must be set to off.

The desired output solution parameters are defined as follows.

Output> Field output requests> Create> [Name: F-Output-2, Step-1]> Continue> [Select Stresses: S, Stress components and invariants, Displacement/Velocity/Acceleration: U, Translations and rotations, and Forces/Reactions: Loads, Uniform distributed loads and RF, Reaction forces and moments]

5. Interaction module

Contact and sliding type boundary conditions are defined in this module. These conditions do not exist in this example and thus there is no need to use this module.

6. Load module

Loads and boundary conditions are created in this module.

Load> Create> [Name: Load-1, Step: Step-1, Category: Mechanical]> Pressure> Continue. Select the line on the top of the region by clicking. Click on 'Done' next to the 'Select surfaces for the load'. In the resulting dialog box enter Magnitude = 50 for the specified load on this side and click OK.

BC> Create> [Name: BC-1, Step: Step-1, Category: Mechanical]> Symmetry/Antisymmetry/Encastre> Continue. Select the left side symmetry line by clicking. Click on 'Done' next to the 'Select regions for the boundary condition'. In the resulting dialog box click XSYMM for symmetry in the x direction and click OK.

BC> Create> [Name: BC-2, Step: Step-1, Category: Mechanical]> Symmetry/Antisymmetry/Encastre> Continue. Select the right line by clicking. Click on 'Done' next to the 'Select regions for the boundary condition'. In the resulting dialog box click ENCASTRE for fixed end condition and click OK.

7. Mesh module

The next step is to actually create a finite element mesh. We must decide on an approximate size of each element. Looking at the physical dimensions of the model, we choose a global element size of 2 in. Thus the length of the model will be divided roughly into 27 segments and the width into 6 resulting in a mesh of the order of $27 \times 6 = 162$ element mesh. We expect high stresses in the notch area and near the fixed end. To capture these stresses we use finer mesh by specifying element size of .5 in the notch region. An element size of 1 is used for line on the fixed end. The following menu paths are used to accomplish these tasks.

Seed> Edge by size> Click on the three lines in the notch region. Click on 'Done' near 'Enter edges to be assigned local seeds'. Enter element size along edges (approximate) = 0.5.

Seed> Edge by size> Click on the line on the fixed end. Click on 'Done' near 'Enter edges to be assigned local seeds'. Enter element size along edges (approximate) = 1.

Seed> Instance> Enter global element size (approximate): 2.

Mesh> Controls. In the resulting dialog box choose Element shape: Quad-dominated, Technique: Free, and Algorithm: Advancing front and then click OK and Done.

Mesh> Controls. In the resulting dialog box choose Element Library: Standard, Geometric Order: Linear, and Family: Plane stress and then click OK and Done. Make sure Reduced integration and Incompatible modes are not selected. (These options could be useful but one must understand associated theoretical details before relying on these ad-hoc measures at improving solution accuracy.)

Mesh> Instance> Click on 'Yes' button next to 'OK to mesh part instance?'.

The resulting finite model is shown in Figure A.15. The model generation is now complete. Additional information, such as boundary conditions and load symbols, can be displayed on the model by selecting View> Assembly display options. The graphics can be saved to a file by using File> Print menu and setting the Destination to 'File', giving a file name and specifying the desired format.

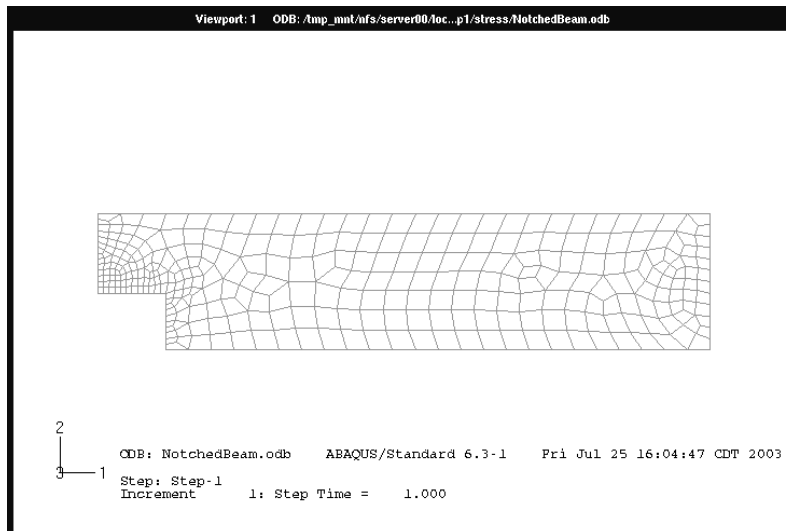


Figure A.15. Plane stress model of notched beam in Abaqus

8. Job module

We are now ready to actually create the finite element analysis input file as follows.

Job> Create> [Name: NotchedBeam, Model: Model-1]> Continue. In the resulting dialog box accept the default options [Job Type: Full analysis, Run Mode: Background, Submit Time: Immediately]

This step will create an input file named NotchedBeam.inp.

Solution

The actual analysis can be performed in the batch mode using the input file by issuing the command 'abaqus job=NotchedBeam' at the shell prompt. Alternatively we can use the menu 'Job> Submit' from within the job module of abaqus cae. With this option the progress of the solution can be monitored using 'Job> Monitor' option. Once the analysis is complete the results will be saved in a database NotchedBeam.odb and summary of the execution steps and requested output will be written to a text file NotchedBeam.dat.

Postprocessing using abaqus viewer

The postprocessing of the results is done using the viewer module. If the abaqus cae environment is already open one can simply switch to the viewer module and open the results database. Otherwise the viewer can be launched separately by issuing the 'abaqus viewer database=NotchedBeam' command at the shell prompt.

The results can now be viewed as numerical lists or plotted in various forms. For example Figure A.16 shows a contour plot of equivalent von Mises stresses obtained using the following menu path.

Results> Field output> [Select Name: S, Stress components at integration points, Invariant: Mises]

Plot> Contours

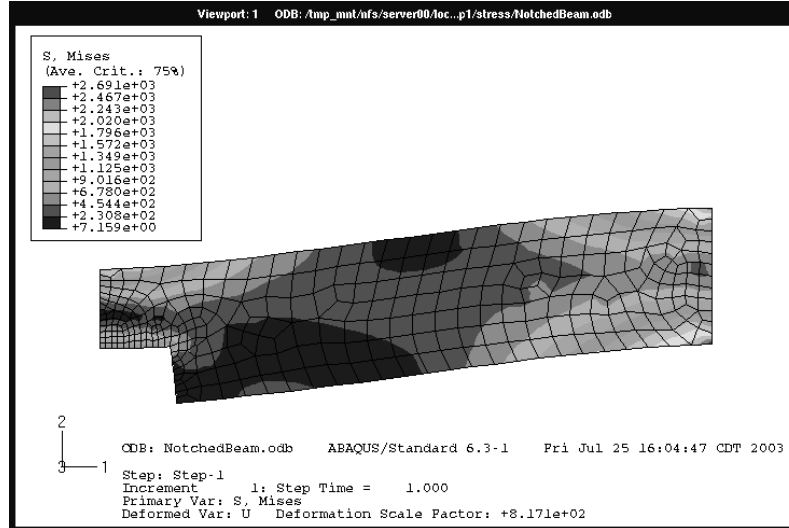


Figure A.16. von Mises equivalent stress contours