

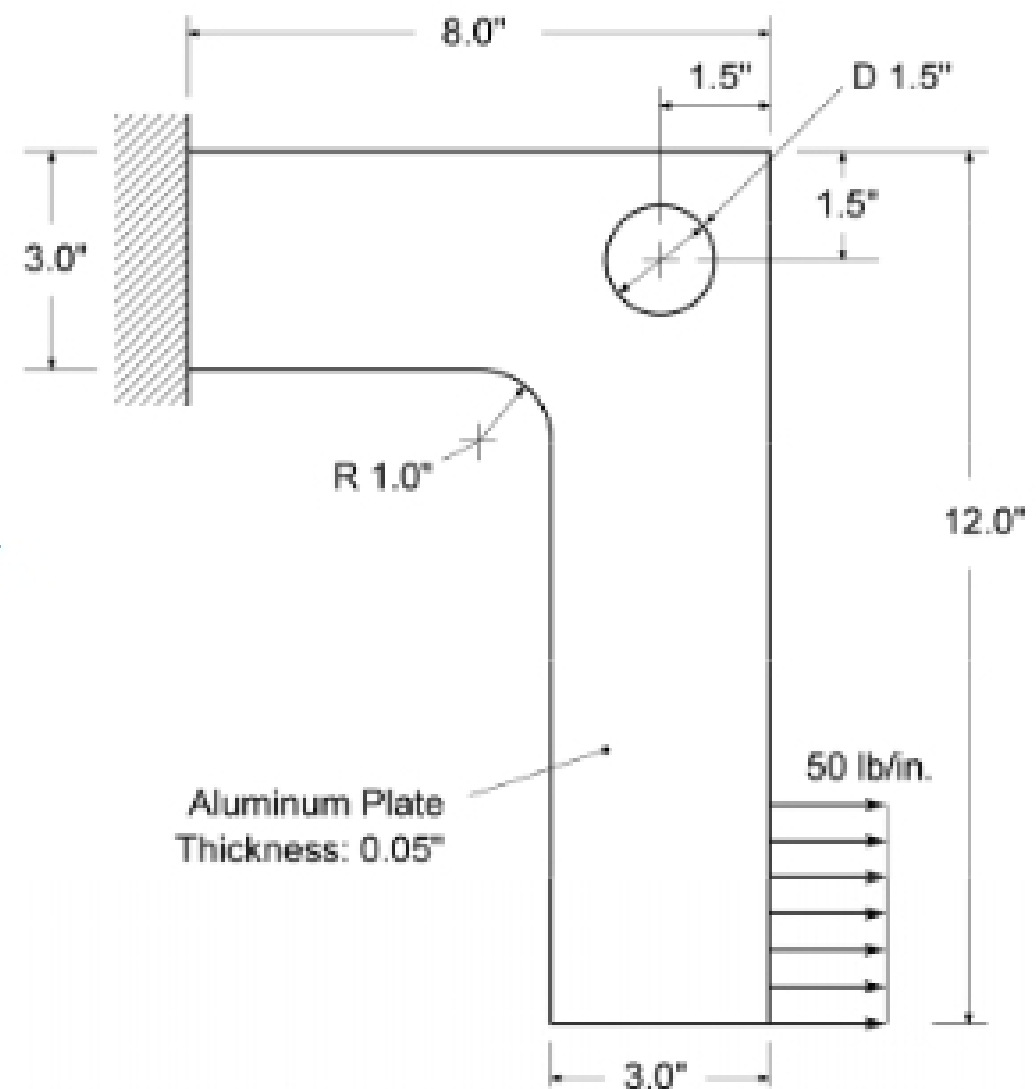
ABAQUS/CAE Tutorial: Analysis of an Aluminum Bracket

Hyonny Kim

last updated: August 2004

In this tutorial, you'll learn how to:

1. Sketch 2D geometry & define part.
2. Define material properties.
3. Apply loads and boundary conditions.
4. Mesh.
5. Run analysis.
6. View results.



Helpful Tips Before Getting Started

Use **Exceed 9.0** or equivalent PC terminal software.

HELP

Online help manuals: *abaqus_cae doc &* - there is a “book” for CAE: “ABAQUS/CAE User’s Manual”. Context sensitive help is also available within CAE.

CAE creates the *.inp* file which you can edit and run by the command line, or you can submit jobs from within CAE. Other files are *.cae* (CAE model file), *.odb*, *.dat*, *.log*, *.msg*, and *.sta*. The *.dat* is the text output file that will contain results. The *.odb* file is the binary output file that will be read during post-processing to view graphical results. The *.log* file keeps a text record of all processes and is useful for checking the status of the analysis. The *.msg* lists the progress of the analysis, as well as provides some messages about why an analysis might have crashed (this information is often within the *.dat* file as well). The *.sta* file is a summary of the information contained in the *.msg* file, and is useful for monitoring the status of long-running jobs during their computation.

MOUSE

Use of the Mouse:

- button 1 (left) is select, button 2 (right) gives menu, button 3 (middle, if available is “enter” or “done”)
- multiple items can be selected by: “dragging” a window or holding the SHIFT key while picking
- items can be de-selected by holding the CTRL key.

ABAQUS/CAE: Getting Started, Create Part

- To run ABAQUS/CAE, first go to the directory you wish your files to be located, then type:
`abaqus_cae cae`
or
`/usr/site/cae/bin/abaqus_cae cae`
- click **Create Model Database**
- In the **Module** dropdown box, select **Part** (this takes about 30 seconds for the program to initialize)
- Note the locations of **Tool Bar**, **Menu Bar**, **Toolbox Area**, **Prompt Area**. These will be referred to repeatedly in the future.
- In the **Toolbox Area**, click **Create Part** button. The **Create Part** window will pop up.
- Enter in name, e.g., *bracket*
- Under **Modeling Space**, choose **2D Planar**
- Base Feature**, **Shell**
- Approximate size**: 20
- click **Continue...**

