

ABAQUS/Standard UMAT Example Problem (Non-FEAMAC)

Description:

This is an example problem from Section 1.1.1 of the ABAQUS Example Problem Manual [1]. It is an analysis of a bolted pipe flange and gasket connection using a 3D model. The purpose of this example problem is for the FEAMAC user to ensure that their ABAQUS/Standard installation is working properly in the case of a user-defined constitutive model (UMAT) for a material. The gasket material is modeled as an isotropic material via a UMAT. A clamping force of 15 kN is applied to the bolt using the *PRE-TENSION SECTION option to associate a pre-tension node with a pre-tension section, where the pre-tension section is identified by means of the *SURFACE option. The pre-tension is then prescribed by applying a concentrated load to the pre-tension node. The applied load is 7.5 kN since only half of a bolt is modeled in conjunction with symmetry conditions [1].

Required Files:

The following files should be placed in the ABAQUS working directory:

File	Purpose
boltpipeflange_3d_usr_umat.inp	ABAQUS input file
boltpipeflange_3d_usr_umat.for	User-defined subroutine UMAT describing gasket material constitutive model

Execution:

This problem can be executed via the following command at the ABAQUS command line:

```
abaqus -j boltpipeflange_3d_usr_umat -user boltpipeflange_3d_usr_umat  
interactive
```

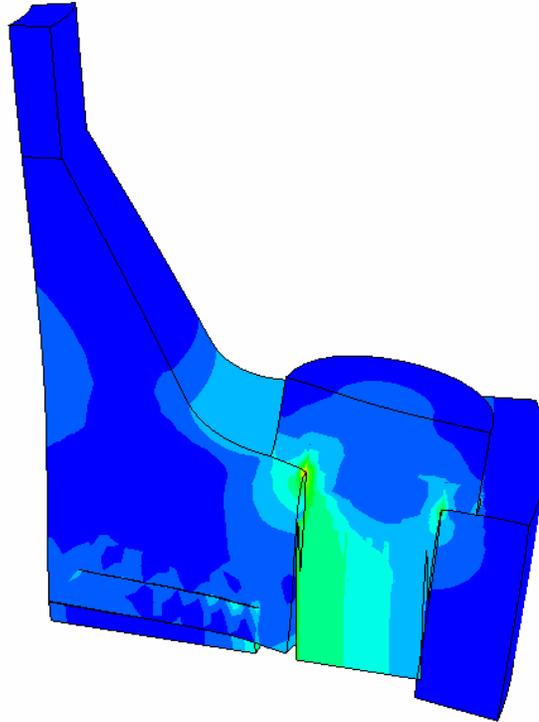
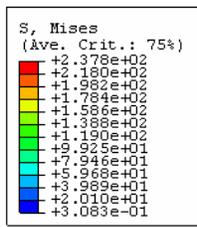
The -j specification indicates the job name (i.e., ABAQUS input file name), while the -user specification indicates the file containing the appropriate UMAT routine for the given problem. The interactive specification provides detailed information on the problem execution during the execution and is optional.

Output:

The output for this problem is written to the ABAQUS output database file boltpipeflange_3d_usr_umat.odb for post-processing in ABAQUS/CAE, ABAQUS/Viewer, or other appropriate finite element post-processing software. In addition, ASCII output is written to the boltpipeflange_3d_usr_umat.dat file.

Results:

Results in the form of the Von Mises stress field in the bolted pipe flange connection are given in Fig. 1.



2 BOLTED PIPE JOINT: 22.5 DEG. SEGMENT 3-D MODEL
 ODB: boltpipeflange_3d_usr_umat.odb ABAQUS/Standard 6.4-1 Mon Feb 07 11:16:19 Eastern Standard Time 200

3 Step: Step-1, APPLY PRE-TENSION LOAD
 Increment 1: Step Time = 1.000
 Primary Var: S, Mises
 Deformed Var: U Deformation Scale Factor: +3.239e+02

Fig. 1. von Mises stress field (MPa) in the bolted pipe flange connection modeled in the ABAQUS example problem 1.1.1 [1].