ABAQUS 3D triaxial compression tutorial
instructions to start in Bechtel lab

1. Start → All Programs → Abaqus Teaching → Abaqus CAE

   *select Create Model Database

   *File → Set Work Directory: select folder in your directory, or a folder on the Desktop, or the Desktop itself

   *remember to back up your files to a flash-drive, or an online file repository

2. Under Part, select Create Part, and type specimen for name
   -Modeling Space should be 3D
   -Type: Deformable
   -Base Feature: Solid, Extrusion
   -Approximate size: 2 (we will use consistent units of N, m, and Pa)

3. Use Create Arc: Center and 2 Endpoints
   0,0
   0,0.03 m
   0.03,0
   Esc, Done
   accept Warning

4. Use Create Lines: Connected
   0.03,0
   0,0
   0,0.03
   Esc, Done

5. Depth = 0.1 m

6. Under Part, select Create Part, and type endplaten for name
   -Modeling Space should be 3D
   -Type: Deformable
   -Base Feature: Solid, Extrusion
   -Approximate size: 2 (we will use consistent units of N, m, and Pa)

7. Use Create Arc: Center and 2 Endpoints
   0,0
   0,0.03 m
   0.03,0
   Esc, Done
   accept Warning

8. Use Create Lines: Connected
   0.03,0
   0,0
   0,0.03
   Esc, Done

9. Depth = 0.01 m

10. under Module, select Property

    (a) Click Create Material (stress-strain curve symbol)
        -name soil
        -Mechanical, Elastic, Isotropic
        -$E = 1e8$ Pa, $\nu = 0.4$

    (b) Click Create Section
        -Name: soil section
        -Category: Solid
        -Type: Homogeneous
        -click OK

    (c) Click Assign Section
        -click on specimen (quarter symmetry) in view port
        -click Done
        -in Assignment Window make sure soil section is selected
        -click OK

    (d) Click Create Material (stress-strain curve symbol)
        -name steel
        -Mechanical, Elastic, Isotropic
        -$E = 2e11$ Pa, $\nu = 0.3$

    (e) Click Create Section
        -Name: steel section
        -Category: Solid
        -Type: Homogeneous
        -click OK

    (f) Click Assign Section
        -click on load platen (quarter symmetry) in view port
        -click Done
        -in Assignment Window make sure steel section is selected
        -click OK

11. under Module, select Assembly

    (a) Click Instance Part
        -select part in viewport, in this case the specimen
        -Mesh dependent on part -click OK

    (b) Click Instance Part
        -select part in viewport, in this case the load platen
        -Independent (mesh on instance)
        -click Auto-offset box
        -click OK

    (c) Click Translate Instance
        -select each end platen to translate separately to put at each end of specimen

-select start point and end point
-click OK

12. under Module, select Step

(a) Click Create Step
-name load step
-should be General: Static, General
-click Continue to popup Edit Step window
-choose 0.1 as initial increment size
-click OK

13. under Module, select Interaction

(a) Click Find Contact Pairs (the flashlight icon)
-click Find Contact Pairs
-double click on Property column name
-select Mechanical, Normal Behavior, and accept default
-select Mechanical, Tangential Behavior, and Penalty, and 0 for Friction Coeff for now
-OK
-OK again

14. under Module, select Load

(a) Click Create Load
-Mechanical: Pressure; select curved face of specimen, click Done
-Magnitude = 100,000 Pa (confining pressure)

(b) Click Create Boundary Condition
-fix x symmetry boundaries
-click Displacement/Rotation
-click Continue, and select x faces
-in Edit BC, set U1 and leave as 0.0

(c) Click Create Boundary Condition
-fix y symmetry boundaries
-click Displacement/Rotation
-click Continue, and select y faces
-in Edit BC, set U2 and leave as 0.0

(d) Click Create Boundary Condition
-fix one platen
-click Displacement/Rotation
-click Continue, and select z face of one platen
-in Edit BC, set U3 and leave as 0.0

(e) Click Create Boundary Condition
-displacement of other platen
-click Displacement/Rotation
-click Continue, and select z face of one

platen
-in Edit BC, set U3 and enter -0.005

*this is a good time to save your Model

15. use Hex elements to mesh (note that quadratic elements will mesh the circular cross-section exactly)

16. under Module, select Mesh

(a) you can assign mesh controls to the platens, and then to the specimen

(b) Click Seed Part Instance, and select part, specimen
- Approximate global size: use default

(c) Click Assign Element Type
-Standard, Linear, 3D Stress
-unselect Reduced integration; may need to turn on if use isochoric plasticity
-OK

(d) Click Mesh Part

(e) do same for end platen instances

17. under Module, select Job

(a) Click Create Job
-name rod tension

(b) Click Job Manager and Submit, and wait until completed successfully
-click Results, and then select deformed contour

(c) you can go back to Interaction Property, and put in a nonzero friction coefficient, like 0.1 or 0.5

(d) you can also substitute out the material model for soil to be Mohr-Coulomb or Cam-Clay plasticity