Download 2d Abaqus Example Meshing

At present I am facing a memory problem in ANSYS (APDL) while meshing as well as solving the solution for the model. I know that in Abaqus there are no memory problems while meshing and solving ...I am compressing a cylinder up to half of its length. But Abaqus is giving only ~35% compression and after that it is giving convergence problem and Job agets aborted.Pycalculix is a tool I wrote which lets users build, solve, and query mechanical engineering models of parts. The tool is a Python3 library, which uses the Calculix program to run and solve finite element analysis models. With it you can see and understand part stresses, strains, displacements, and reaction forces.This paper presents a 100-line Python code for general 3D topology optimization. The code adopts the Abaqus Scripting Interface that provides convenient access to advanced finite element analysis (FEA).