This sample was described here: http://www.libremechanics.com/?q=node/19

(It doesn't look like real example but good for studying)

Because of using C3D4 elements with linear approximation the results can be incorrect (not only max. values, but stress distribution in general).

The using C3D10 quadratic elements gives better warranty of correct solution.

Main Steps:

1. Load .STEP file into NETGEN (see Part1.STEP in the folder).
2. Don't use default meshing option in this case (because it will produce large elements with curved surface what may cause negative Jacobian) Go to the Mesh>Meshing Options and set up mesh size grading to 0.1 (equally distributed dimensions of elements) See. Fig. 1.
3. After meshing go to Mesh>Edit Boundary Conditions and change number of faces (to find it then to apply BC).
4. Make Refinement > Second Order (C3D10) and save mesh as .vol file
5. Run CalculiXForWin and select .vol file in the starting window. The units in this problem are "mm" and the model should not be scaled in this special case. Apply pressure and fixed support (Fig. 4), save allinone.inp file and solve the problem.
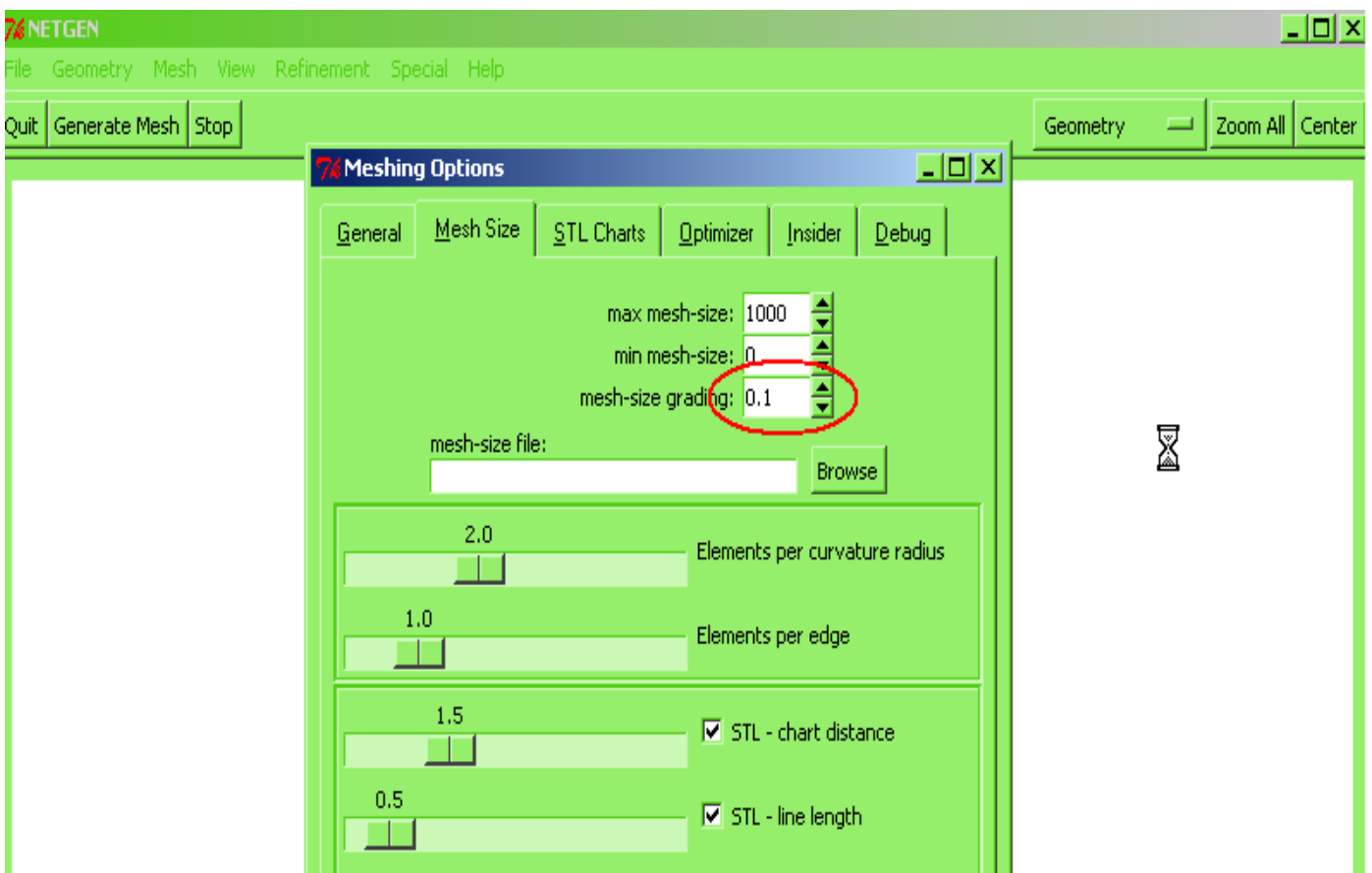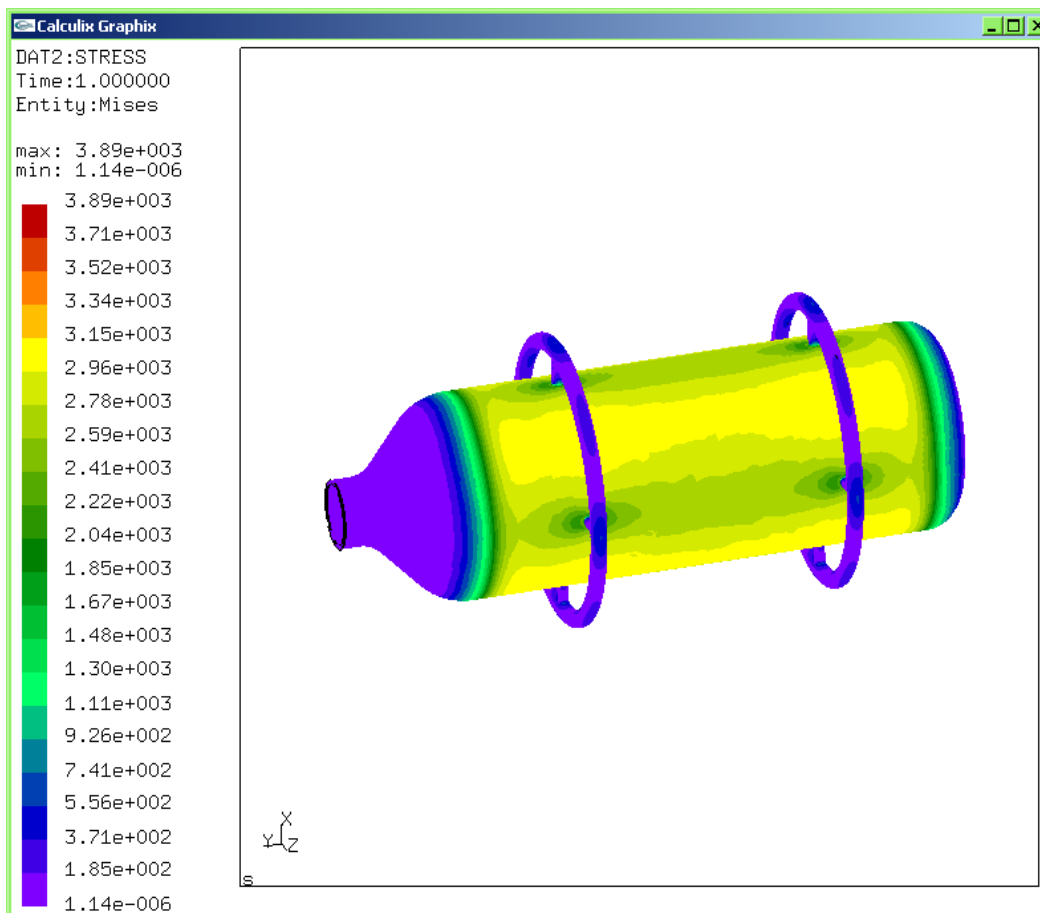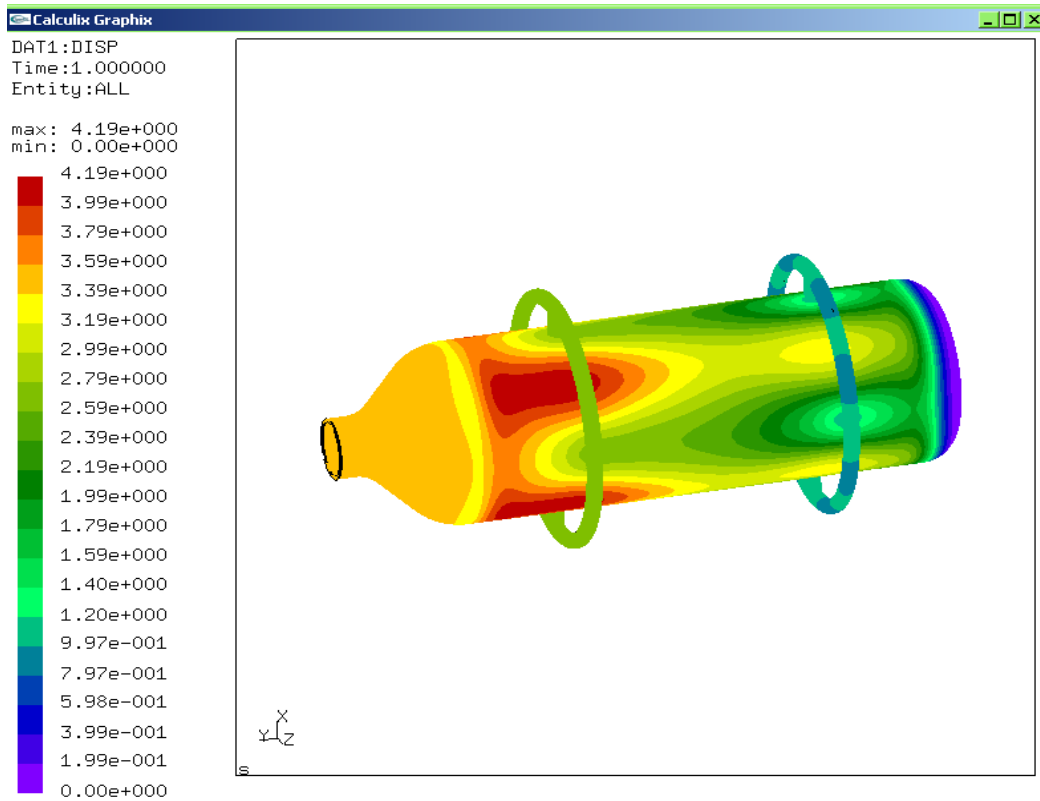

Fig. 1 – The results of calculation are shown at Fig. 2, 3 (displacements and Von-Mises stress)

Fig. 2 Max. Displacement (CalculiX) 4.19 mm

Fig. 3 Max. Stress 3890 N/mm²

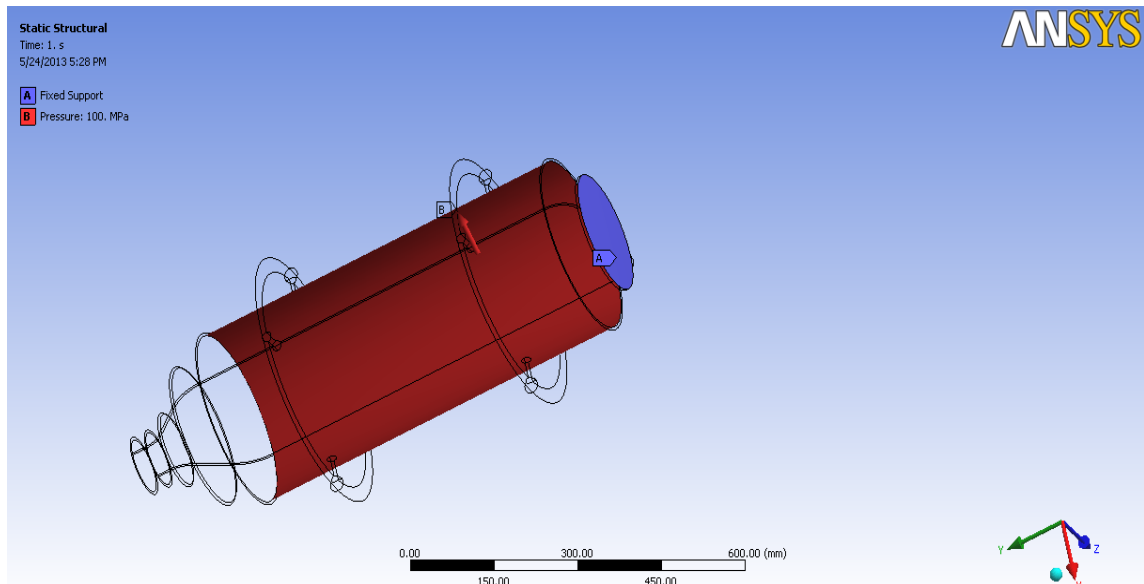The results obtained in ANSYS are shown below
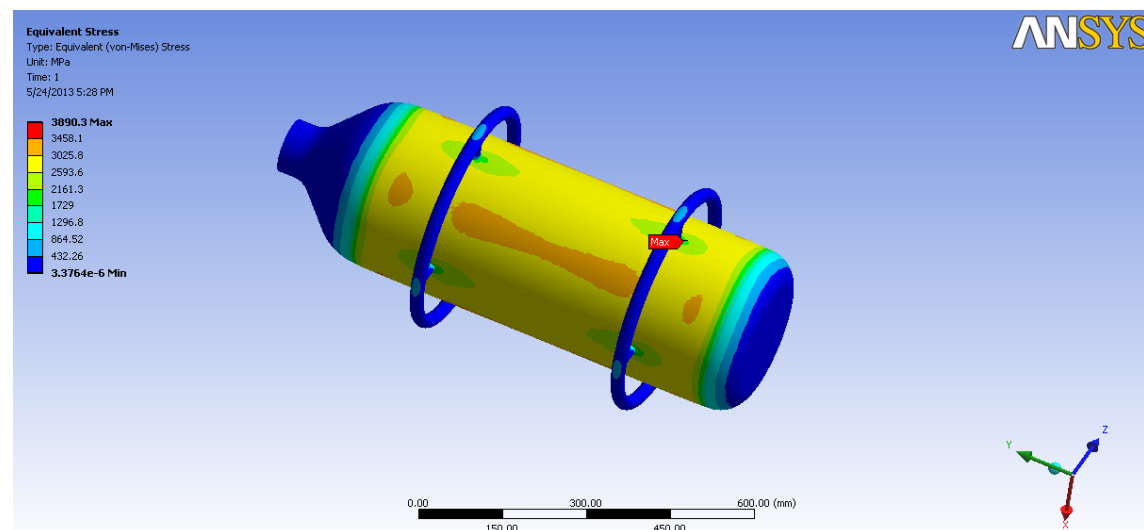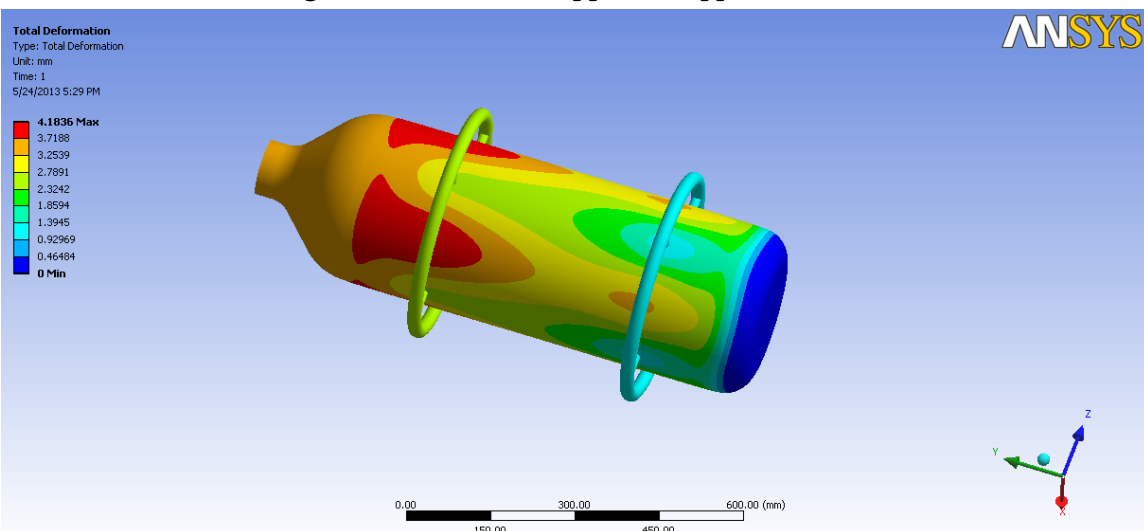
Fig. 4 Loads and supports applied

Fig. 5 Max. Displ 4.18 mm, Max. Stress 3890 MPa

Note: In this case maximal stress is more than yield stress, but it is localized. In case of real structure and loading it needs to be investigated for low-cycle fatigue, depending on possible number of cycles (local plastic zone). It is also can be a zone of singularity.