The Cupola Treated With Shell Elements

www.calculixforwin.com

See the movies here: | 1 | 2 |

Input data: Geometry Cupol.step (inches)

Total vertical force (applied to half of area) = 5000 lb. Material - steel.

Thickness of shell = 1''.

Static analysis (ANSYS VS CalculiX)

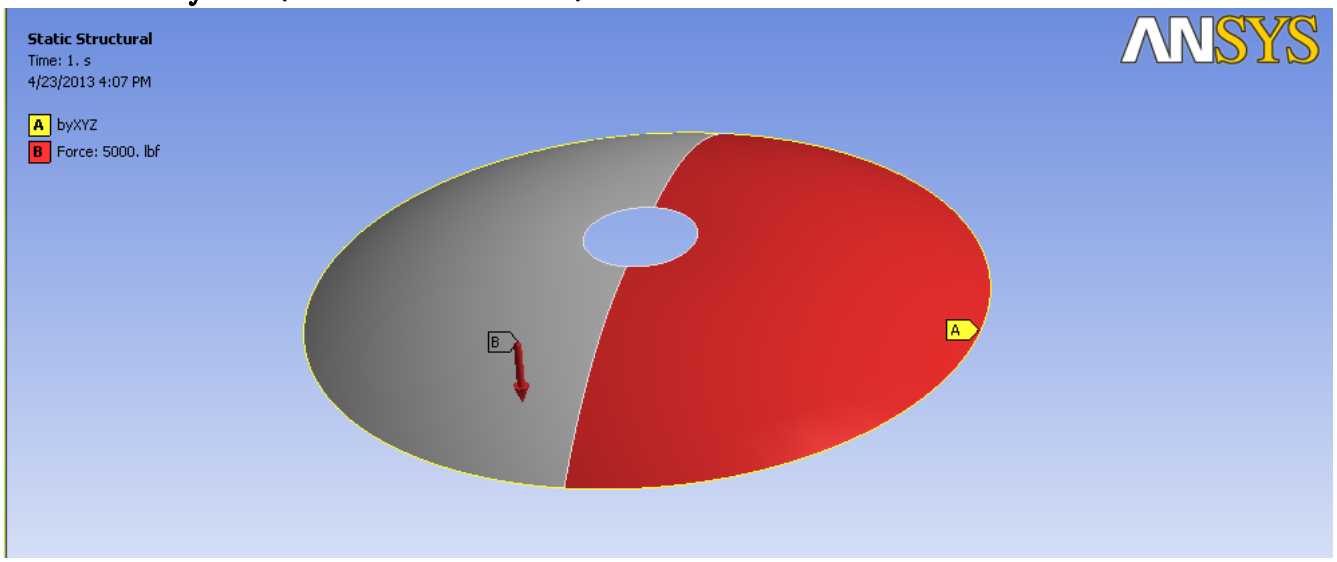

Fig. 1 - ANSYS model

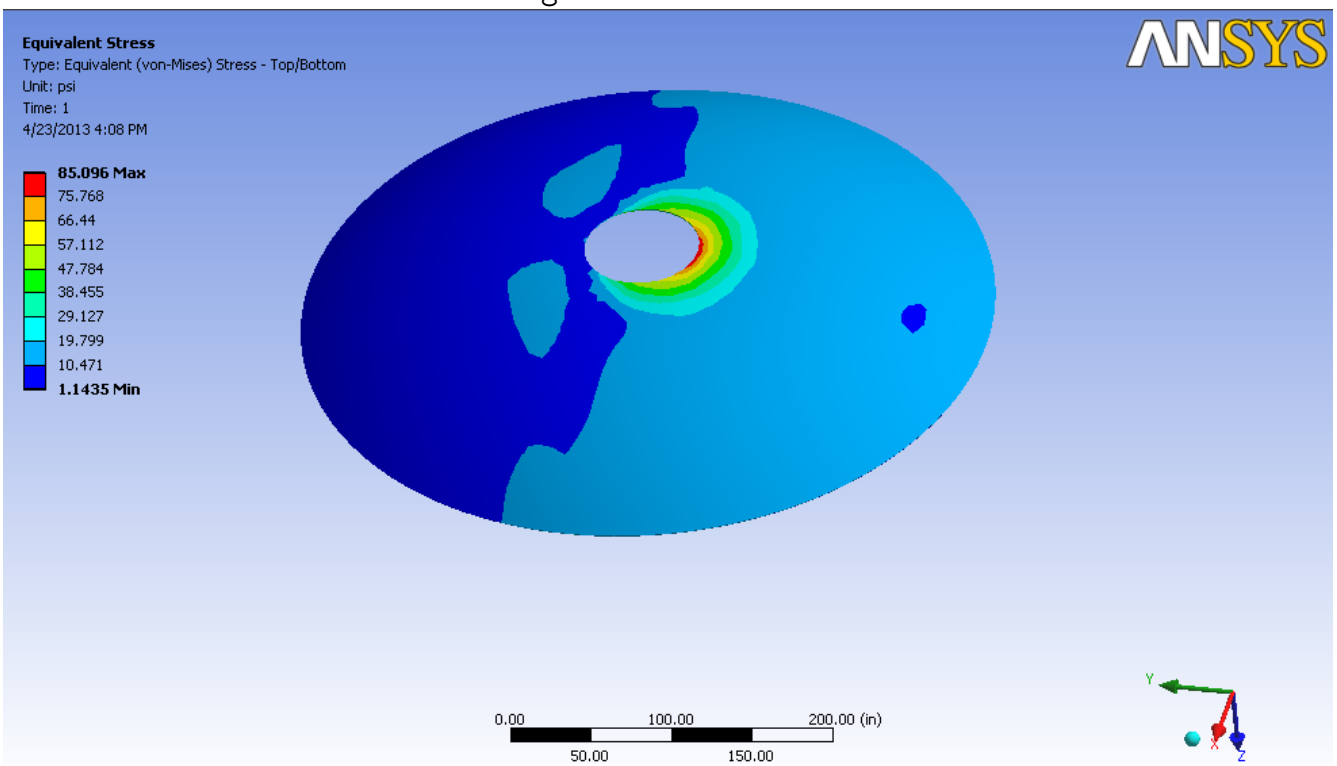

Fig. 2 - ANSYS, Von Mises stress (max. 85 psi)

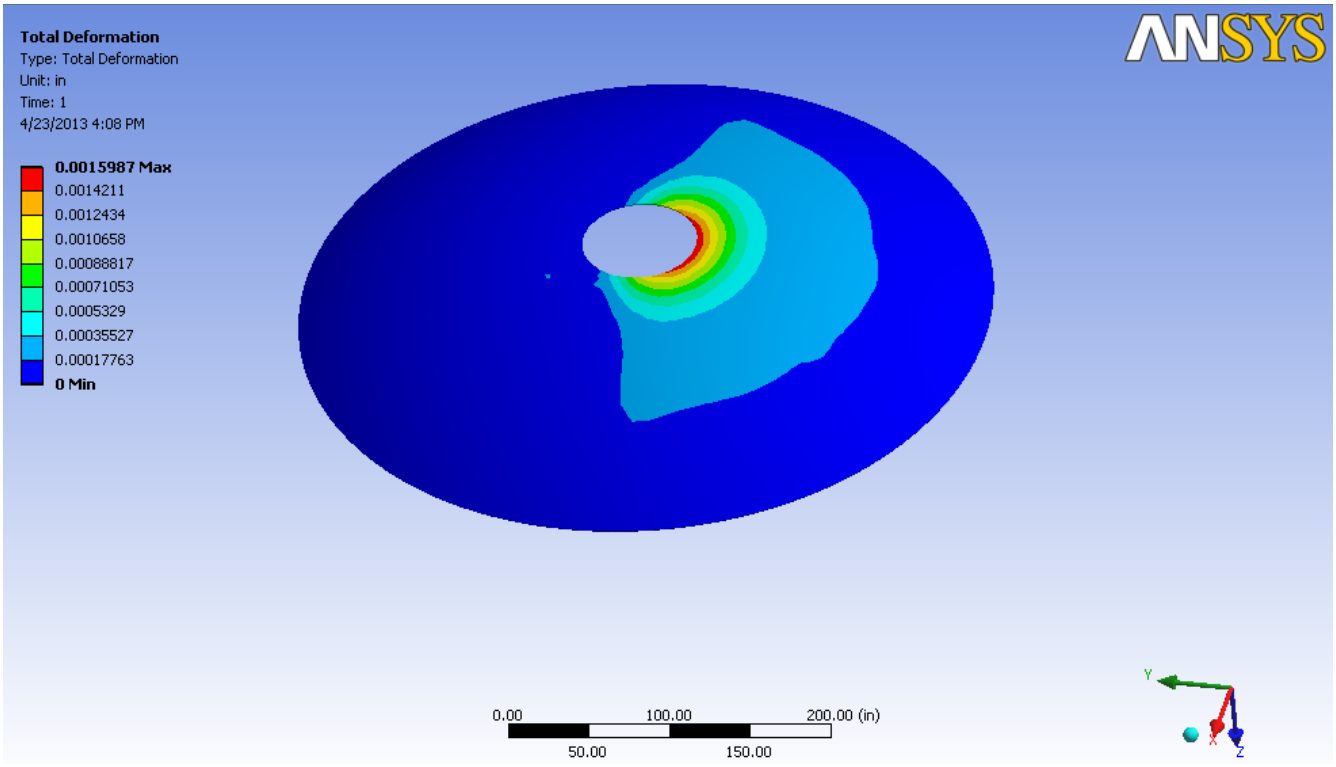

Fig. 3 - ANSYS, displacements, max. 0.0016''

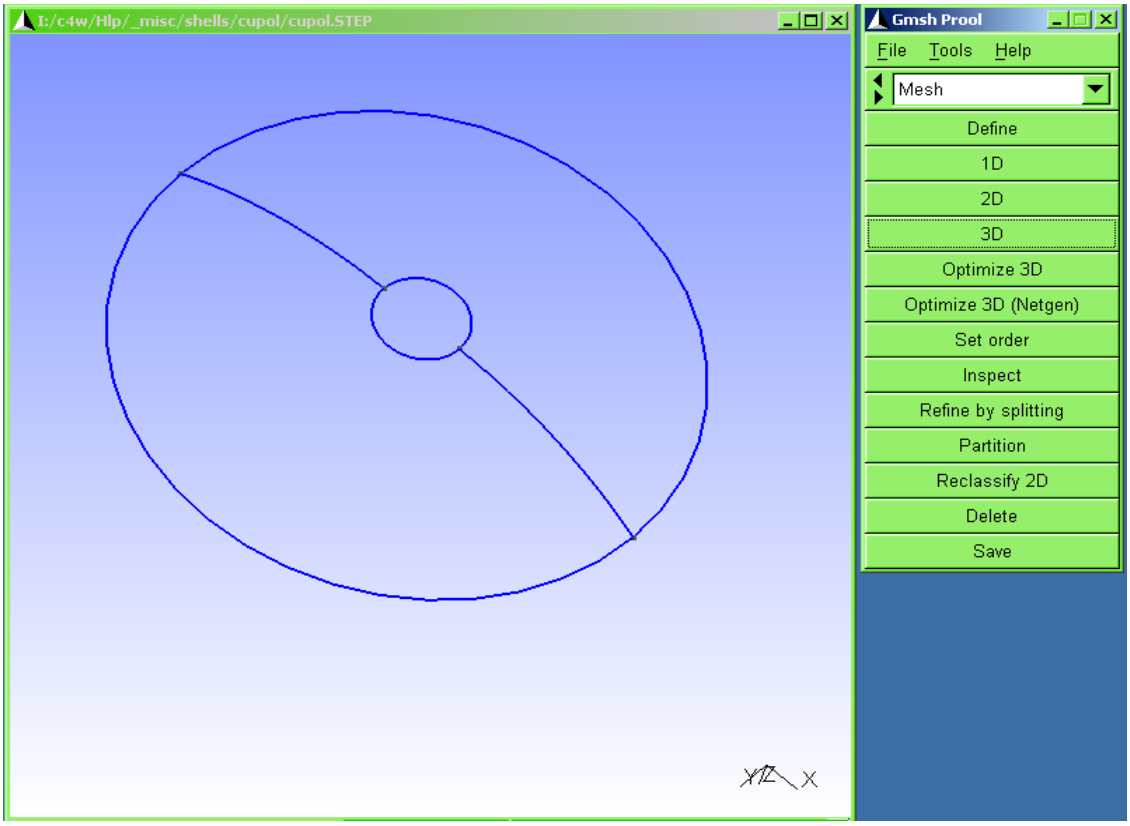

Fig. 4 - Load model into GMSH (by Prool with CalculiX .INP support)

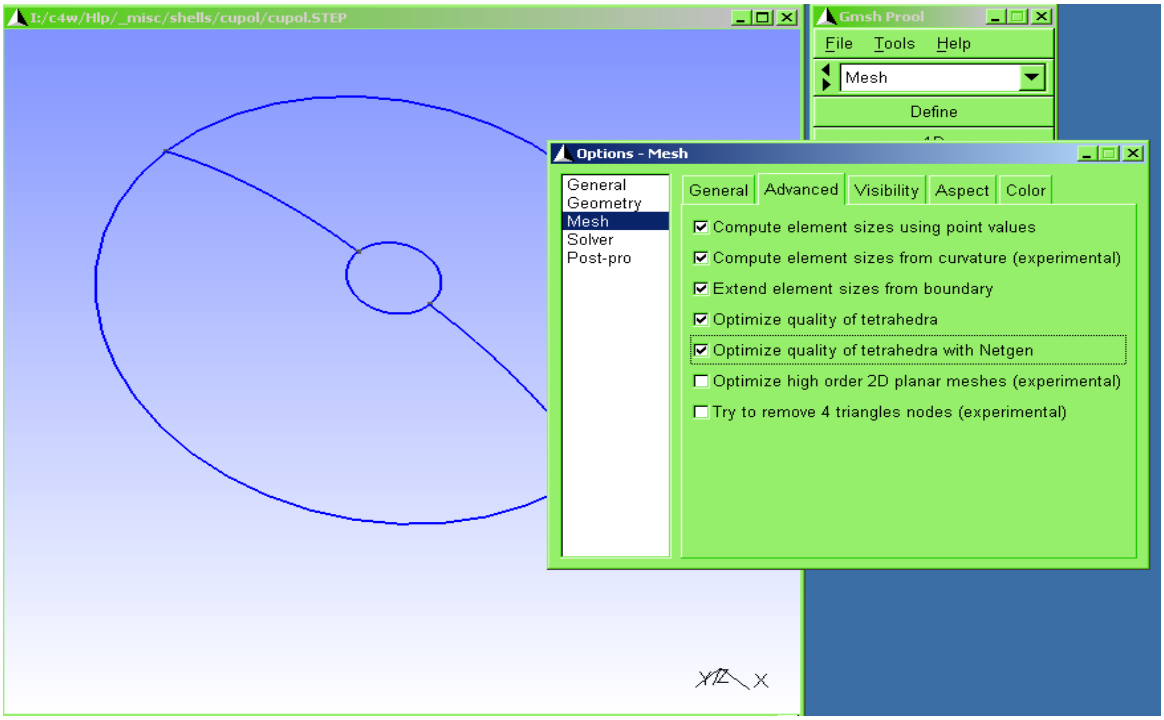

Fig. 5 - Set Up some mesh options, push Mesh>3D, then Mesh>Refine by Splitting (3 times).

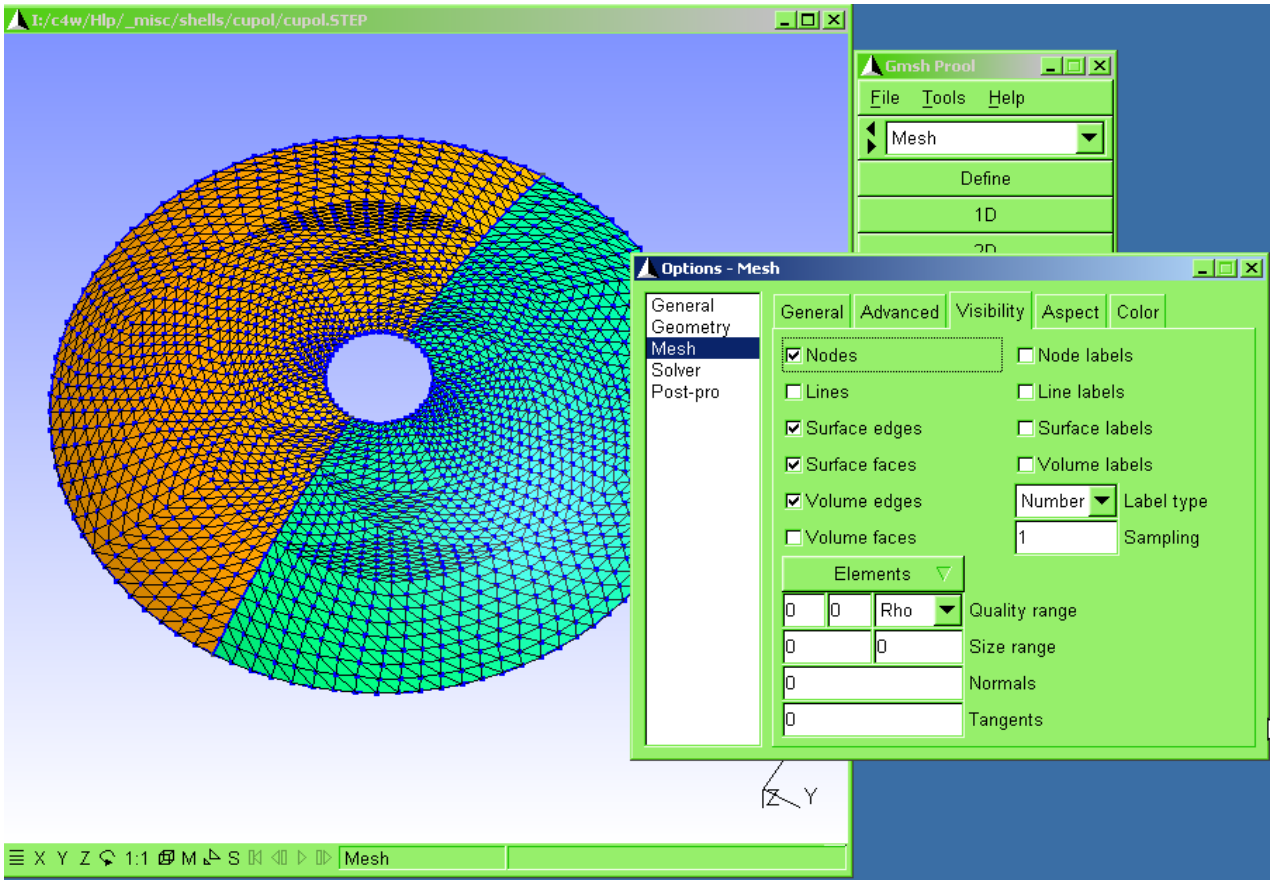

Fig. 6 - The result of meshing (**don't set second order now**, you can do it later in CGX!). Save as INP (with no options)

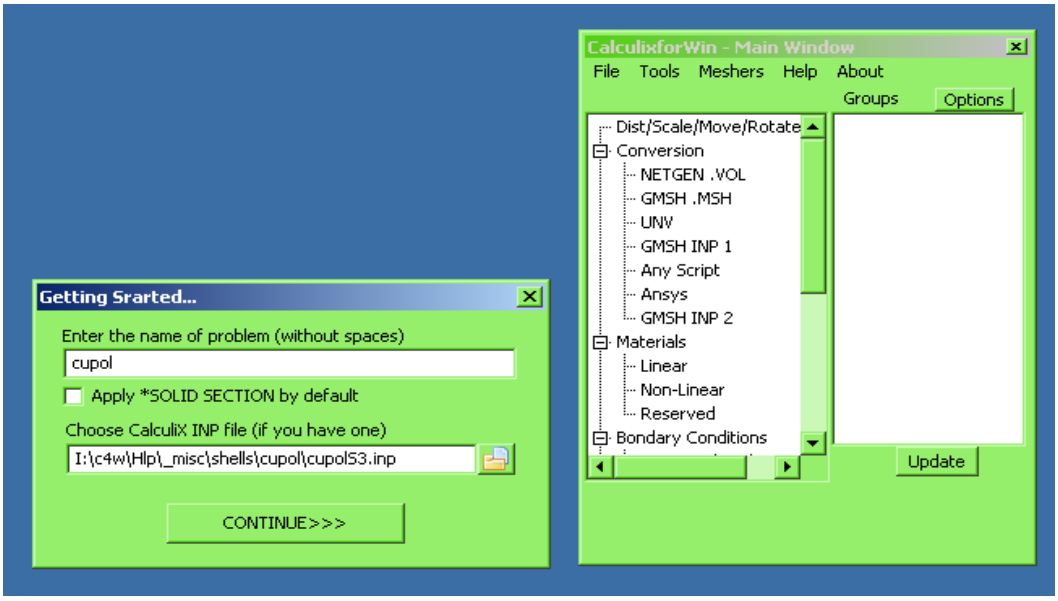

Fig. 7 - Run CalculixForWin and set path to INP file (uncheck *SOLID SECTION)

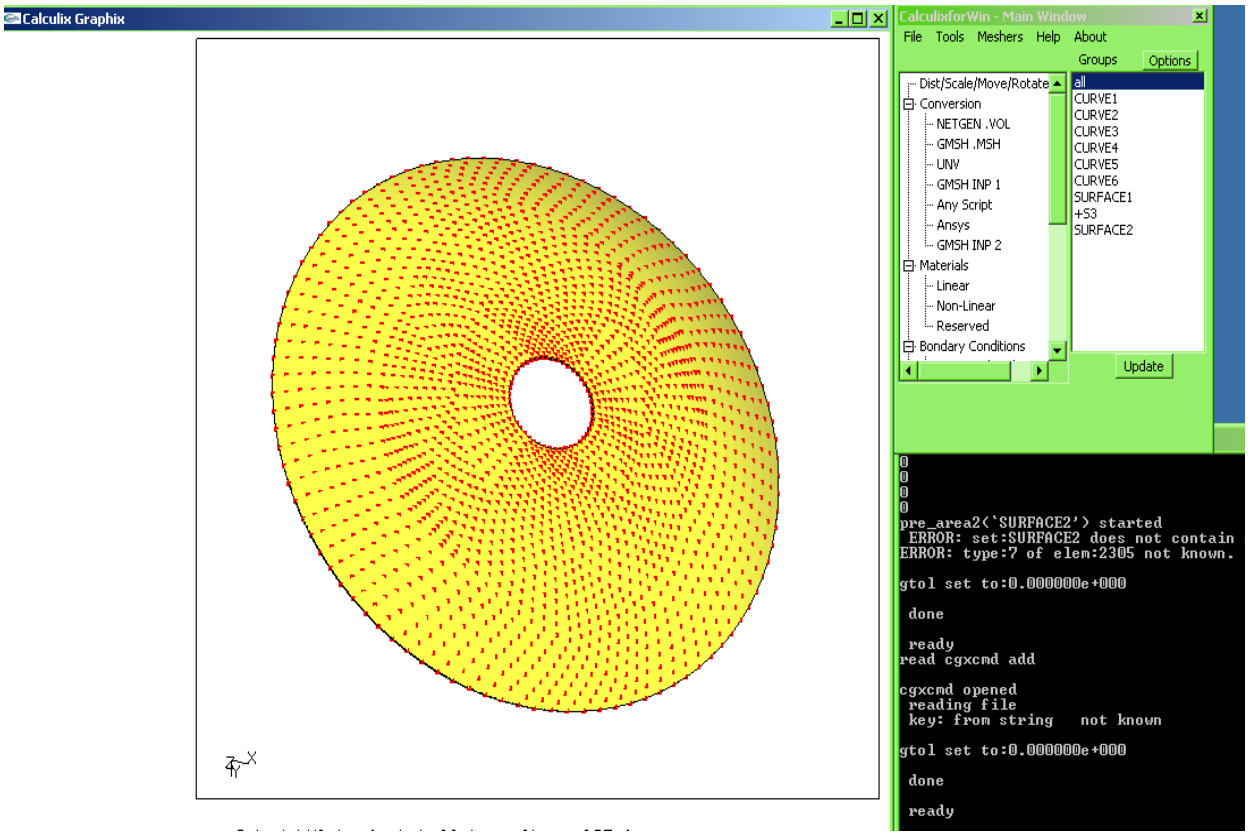

Fig. 8 - Update the groups

Go to Tree View > Materials>Linear and check material applied by default (steel, no "for solid section")

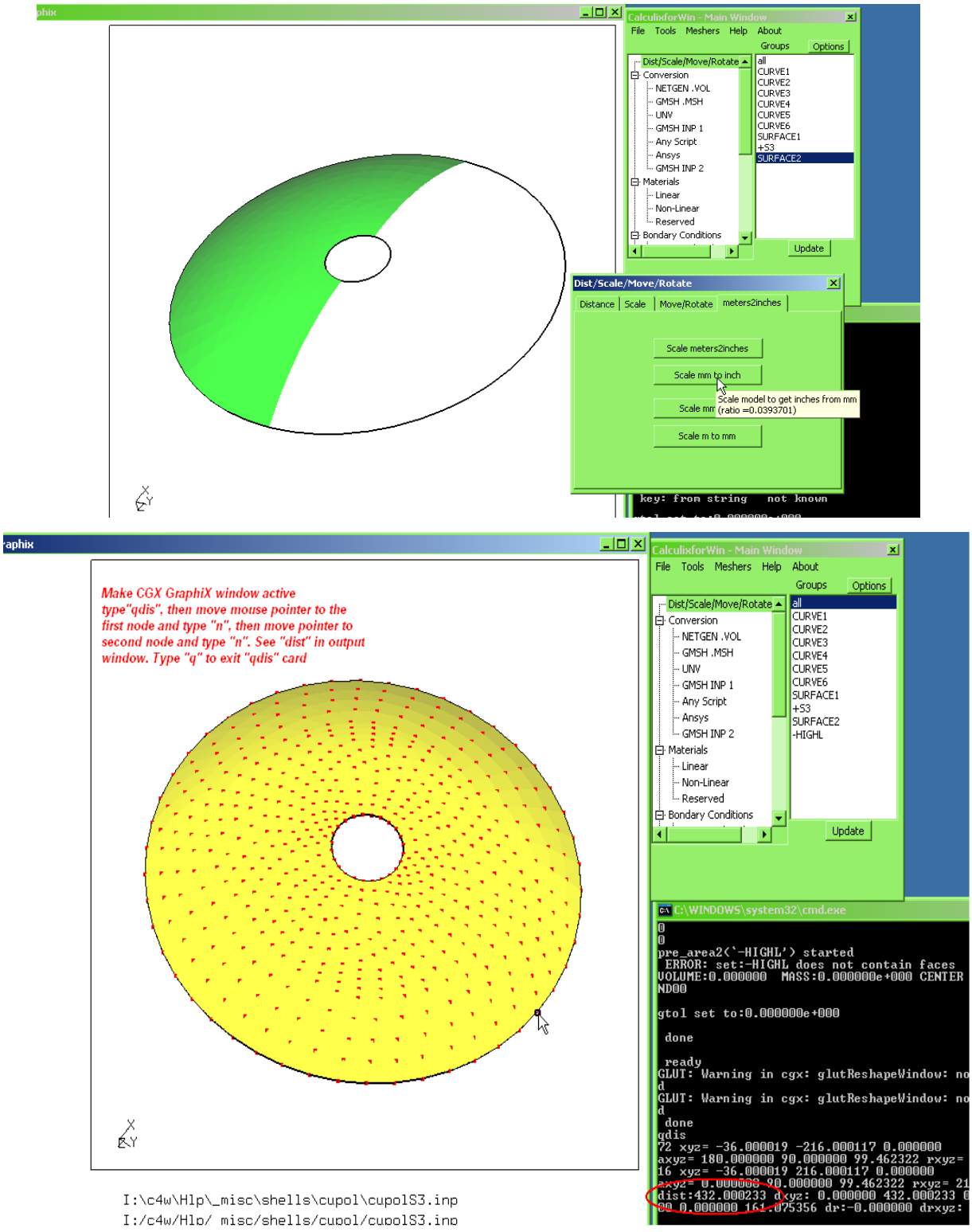

Fig. 9 - Scale model and check dimensions with "qdis" card. Scaling is necessarily due to using step file with GMSH. (due to open-cascade library option) Current unit system is Imperial and length units should be "inches" (and the model should be scaled to inches) **Update the groups after scaling!!!**

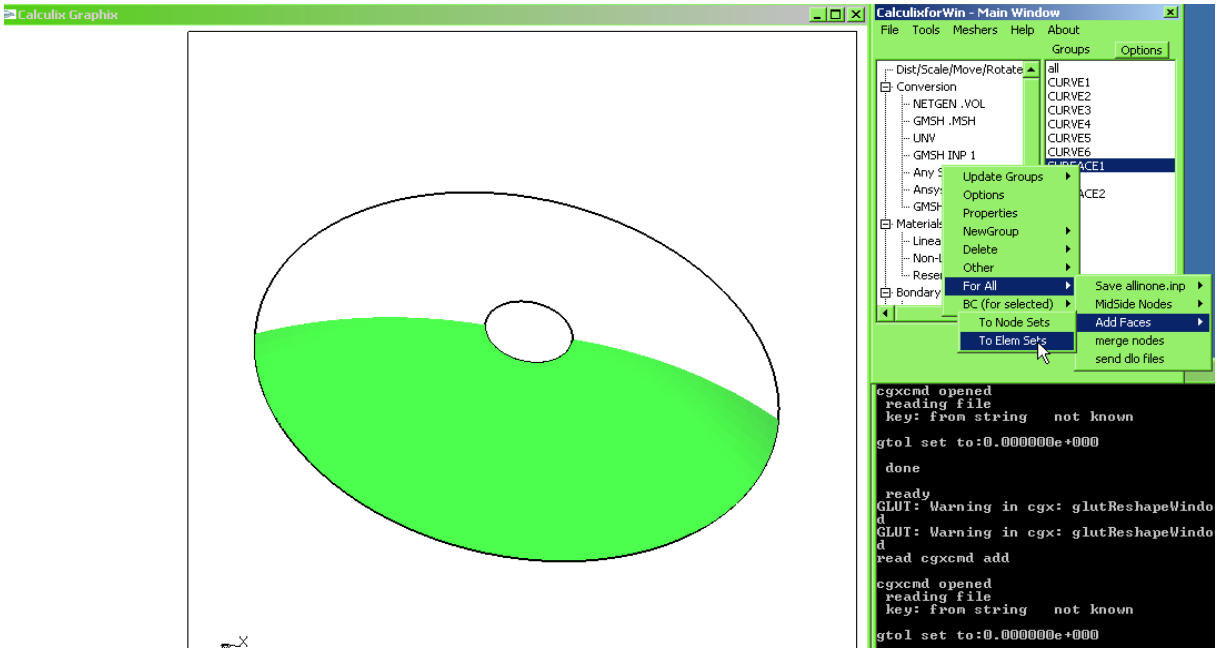

Fig. 10 - Add surfaces to element groups (this is equivalent of "comp do" card Update groups to add new information about changes into the CalculixForWin database

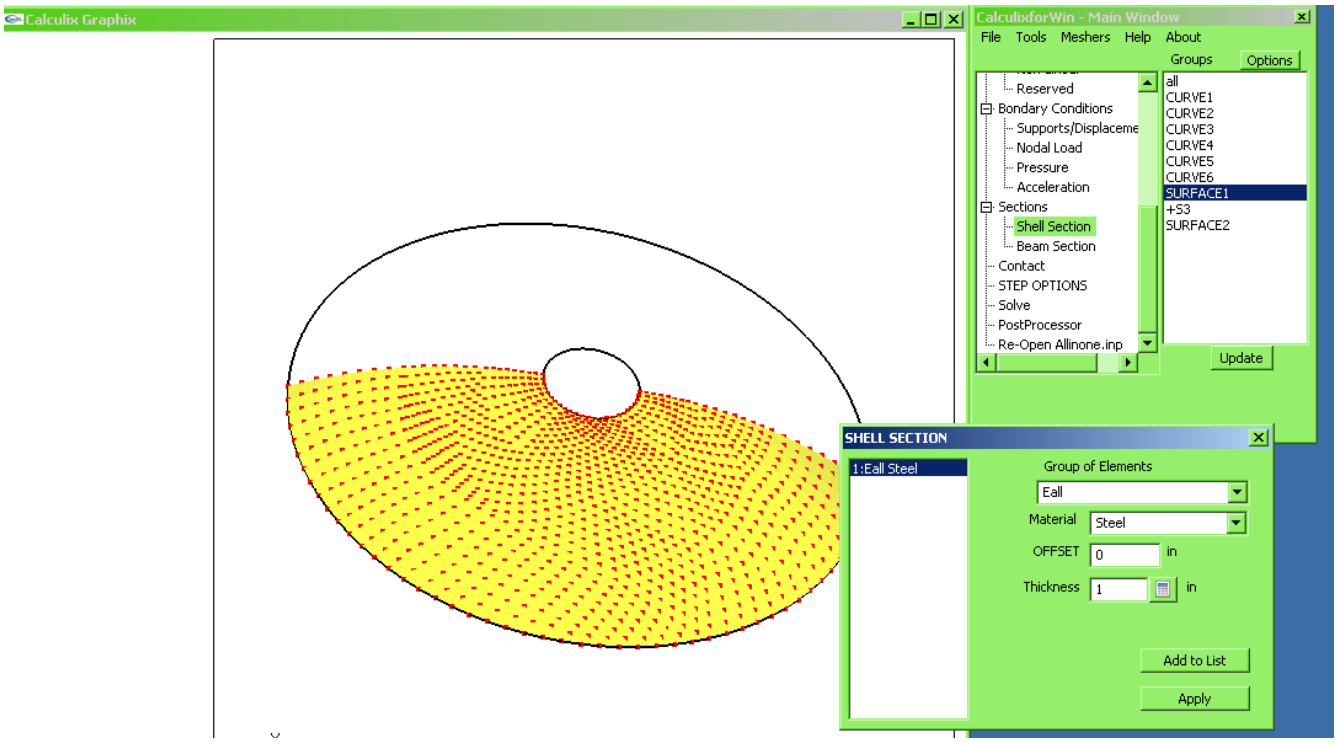

Fig. 11 Apply "Shell Section", thk = 1''

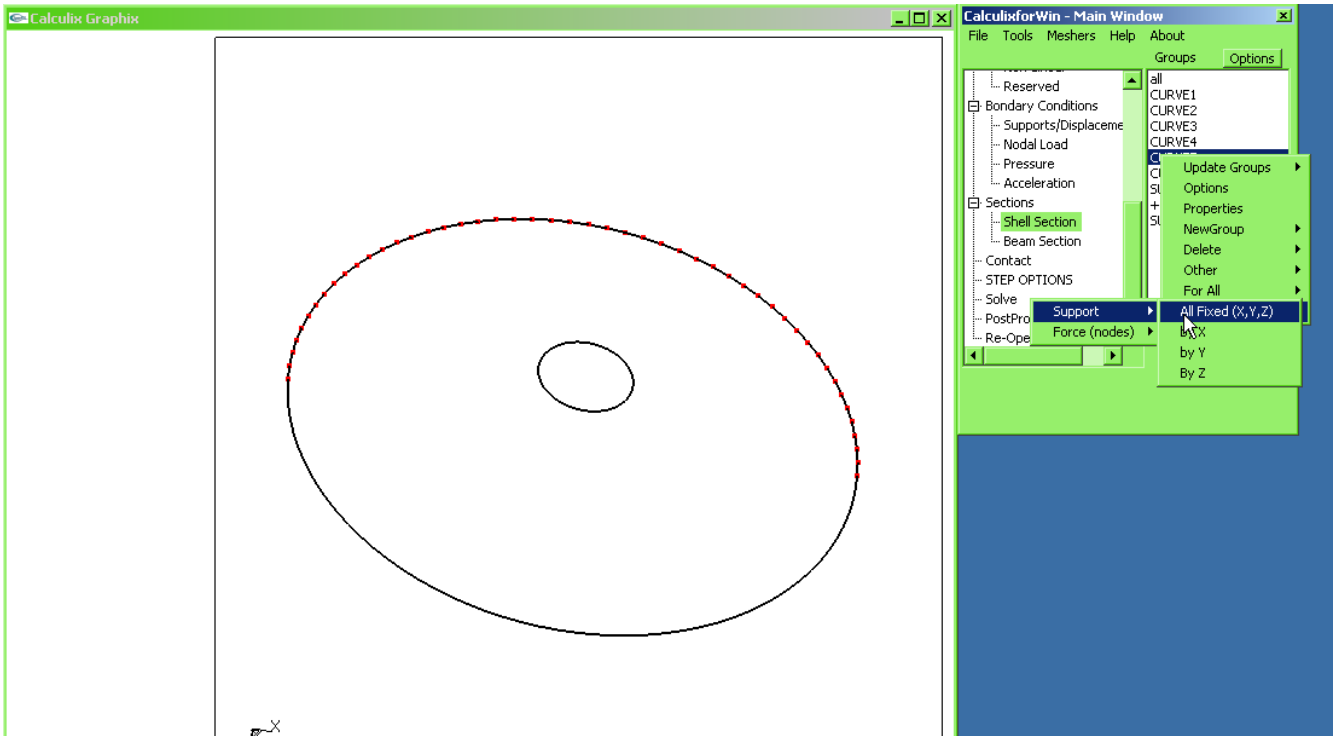

Fig. 12 Apply "Support > All Fixed" to nodal groups "CURVE1" and "CURVE5"

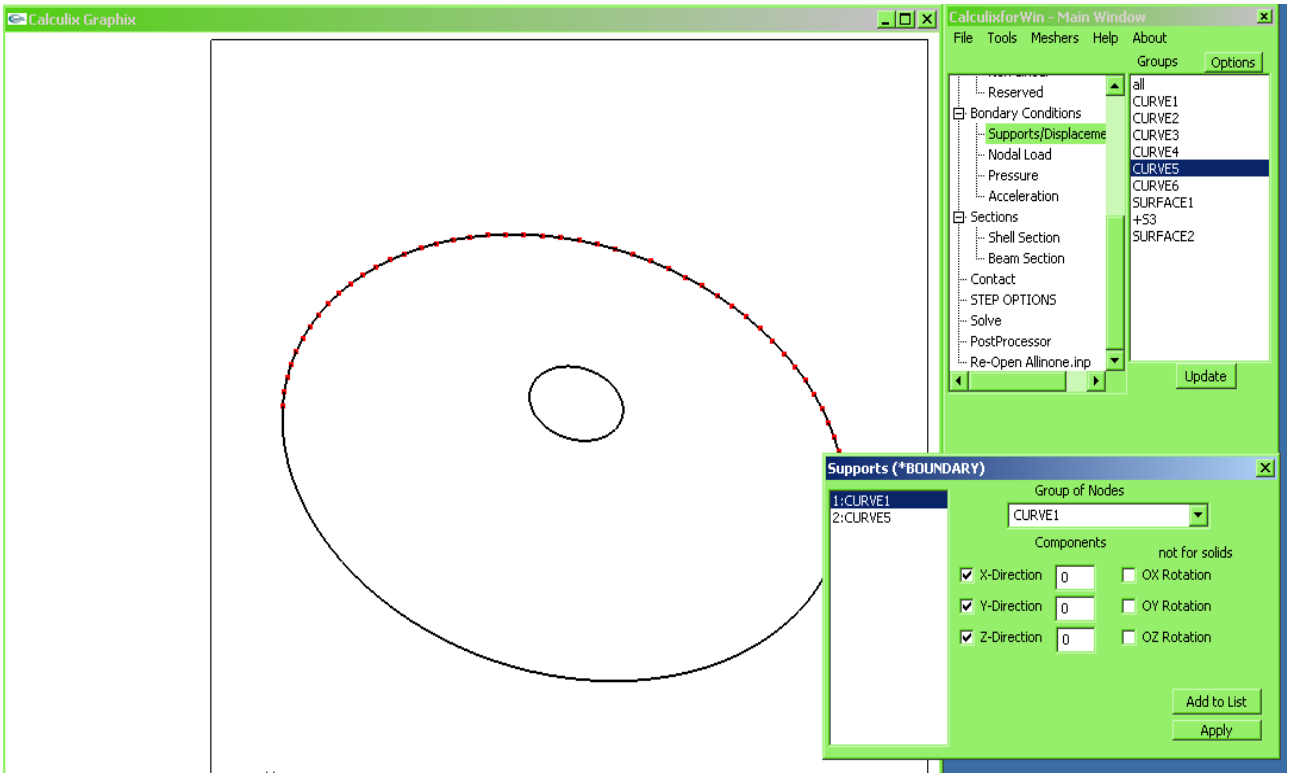

Fig. 13 - Check BC applied

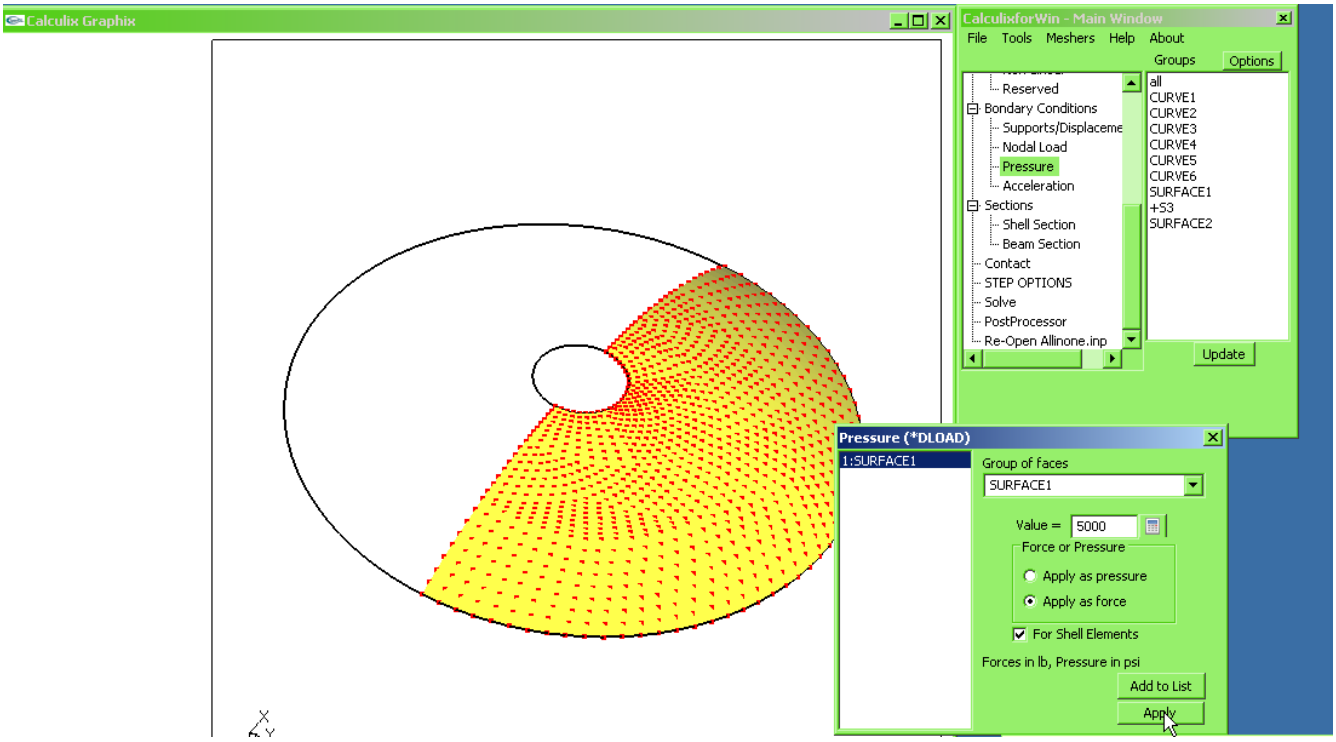

Fig. 14 - Apply Load to "SURFACE1" (to the faces as Force)
The pressure = 5000 lb/Area will be applied
(model should be scaled before using this option)

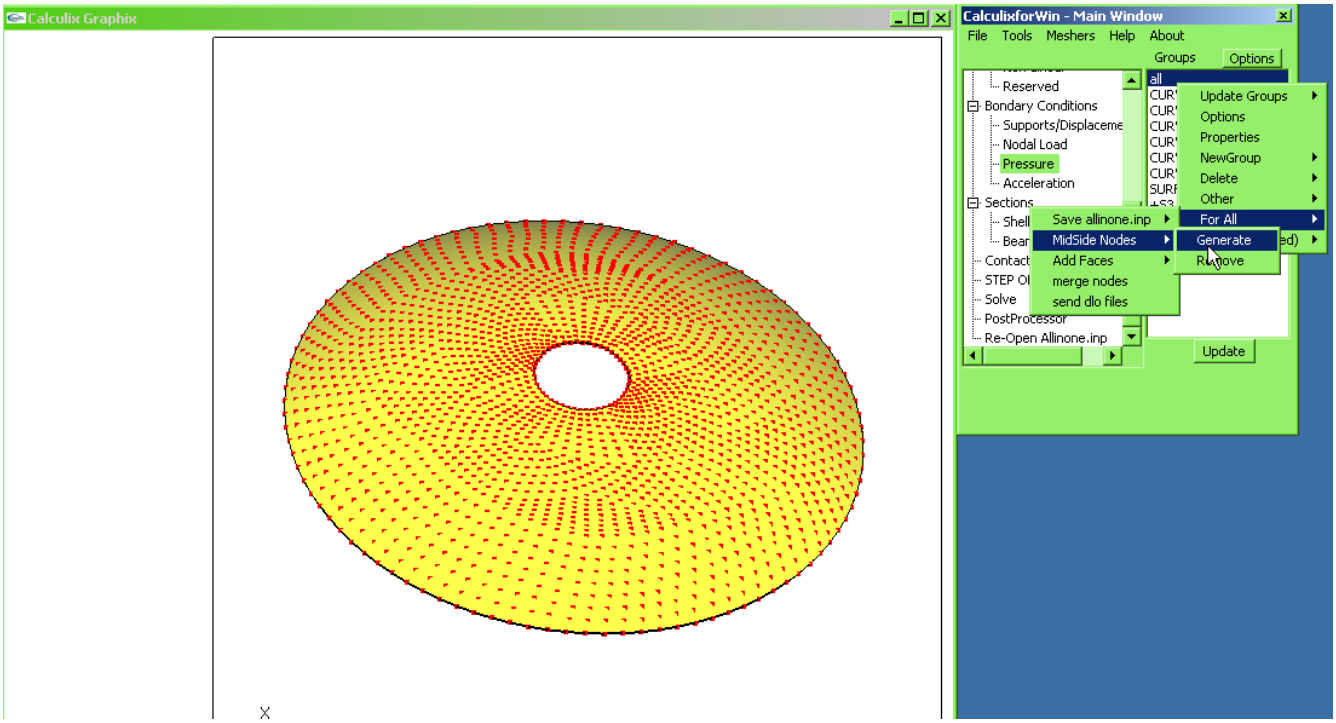

Fig. 15 - Generate mid-nodes (second order elements)

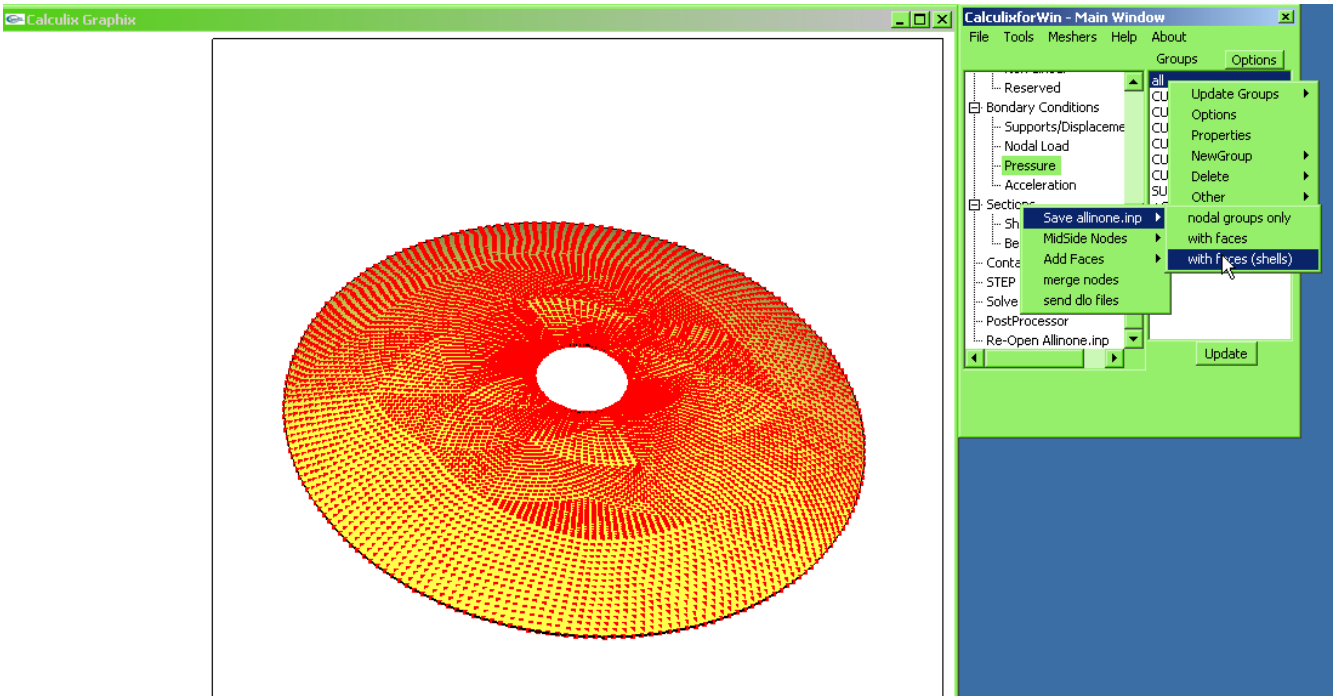

Fig. 16 Save allinone.inp (with "for shells" option)

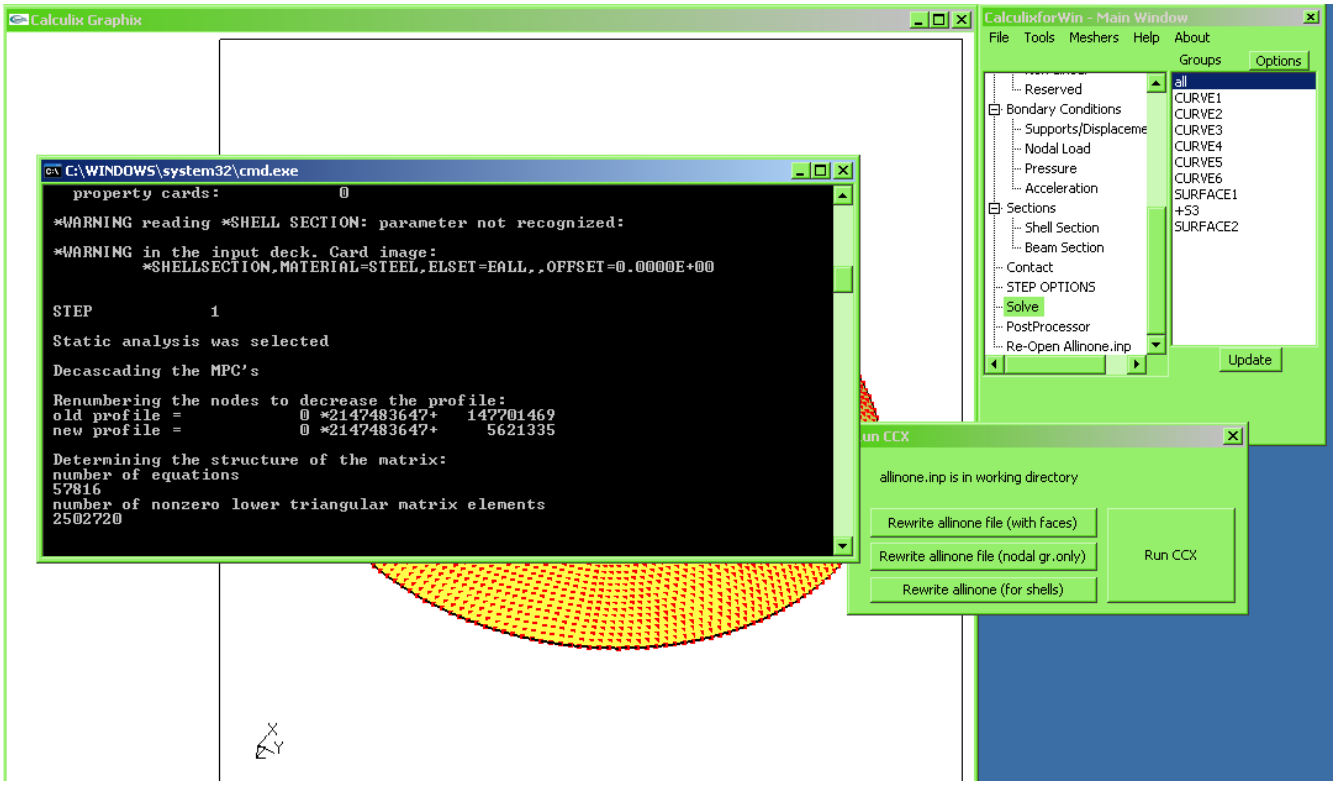

Fig. 17 Run CCX and wait when solution is done..

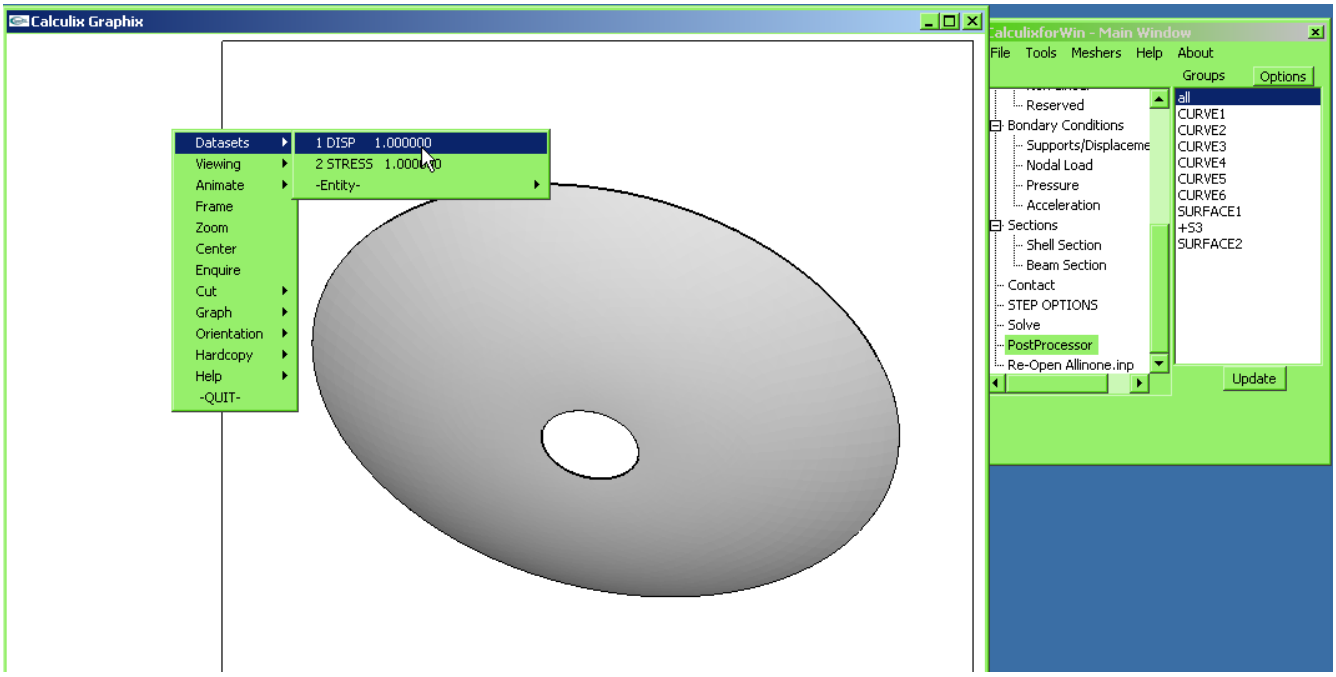

Fig. 18 Run CGX in post-processor mode

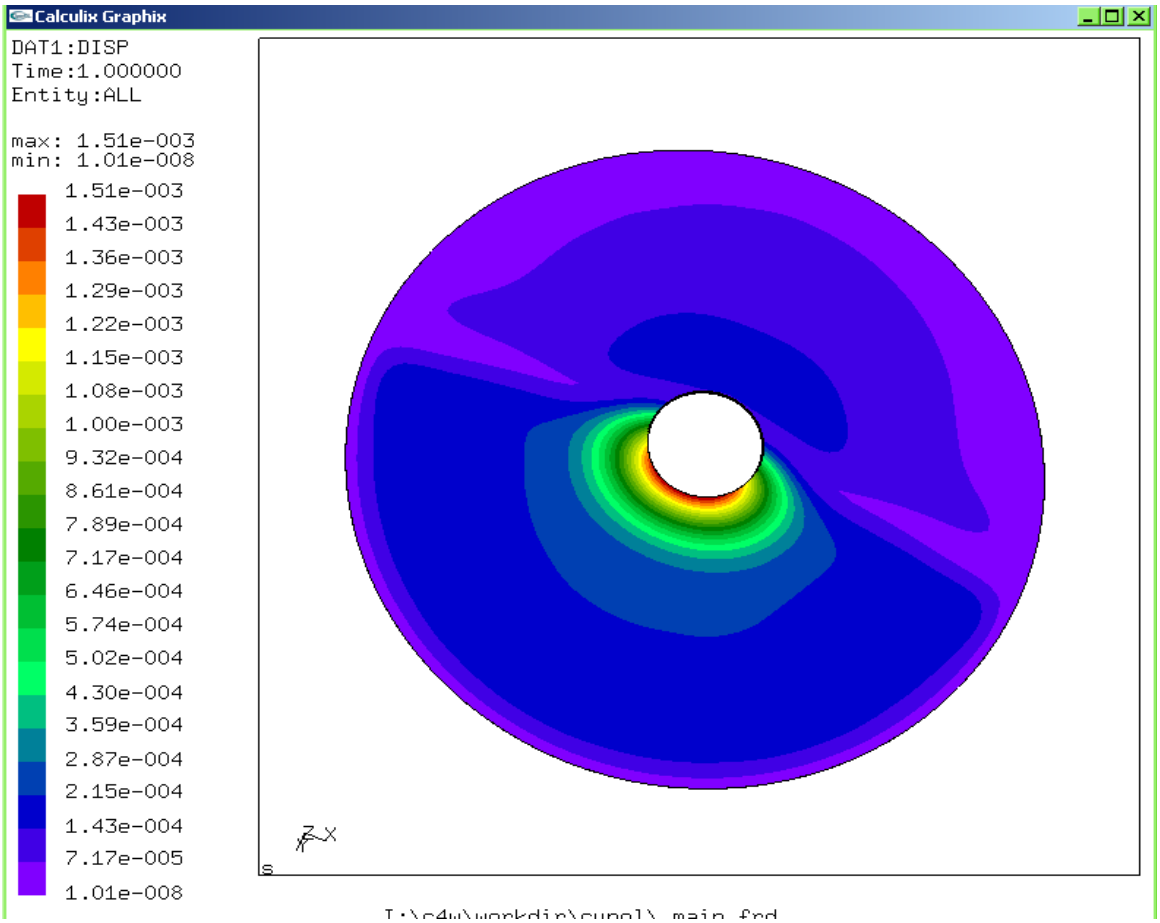

Fig. 19 Max. Displacement 0.00151''

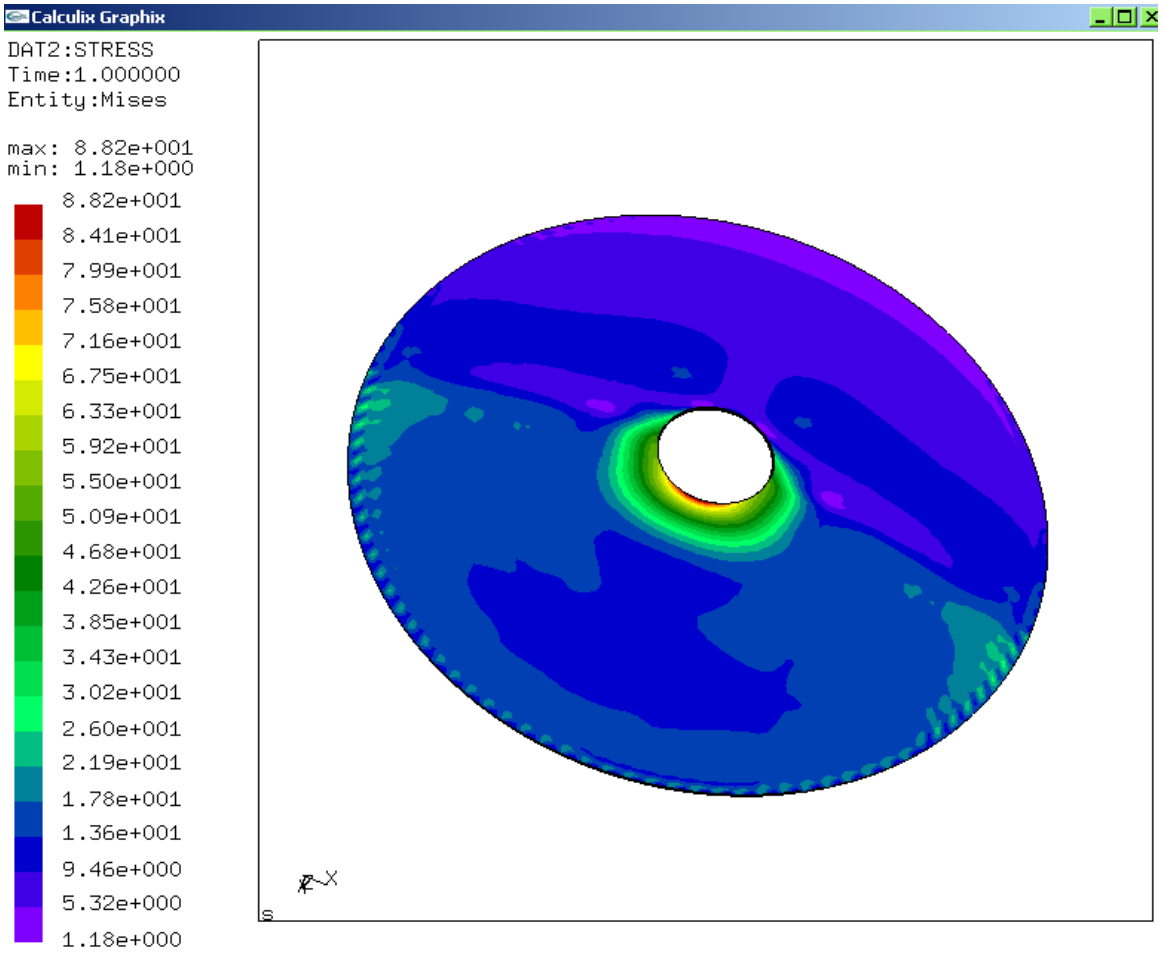

Fig. 20 -Max. Von Mises Stress 88.2 psi