Finite Element Convergence Studies Using COMSOL Multiphysics and LiveLink™ for MATLAB® with Large Assembly Models

Hoofar Pourzand[1], Abdul H. Aziz[1], Anand Singh[1]

[1]Pennsylvania State University, State College, PA, USA

Abstract

The original question of how to import a huge model which is in fact an assembly of assemblies or of several parts depends on many parameters, like system geometry, type of simulation under study and available resources. Whether to import the system in the form of several parts individually or as of a one huge part is a decision to make before performing the analysis. Also, one might consider combining few parts and reduce the number of assemblies as an alternative. How reasonable the finite element solutions prove to be is dependent on the quality of the mesh. Convergence studies are performed on a Flyer Model while the exact solution to the simulation problem is not available. COMSOL Multiphysics along with its Finite Element Convergence Studies Using COMSOL Multiphysics and LiveLink™ for MATLAB® with Large Assembly Models is used as a test bench to investigate the needed number of elements and degree of freedom as well as the order of Lagrangean p element for a defined simulation under study. Use of COMSOL Multiphysics: The Parasolid® model of five different assemblies are imported into COMSOL 4.3 for structural mechanical analysis. Statistics from the solution is gathered from the mesh engine and the MUMPS solver for evaluation the convergence rate. Results: A preliminary study is performed on the rudder where the upper and lower bars are fixed and a plan force of 290N is applied on one side of the Rudder. VonMises stress is plotted in the results. The converegence happens at the fourth refinement of the third element type, which will save us about 50% in computation time as well as lower memory requirement and CPU load on the clusters. In this report, we outlined a procedure for measuring the convergence order for the FEM solution using COMSOL 4.3 and LiveLink for MATLAB for a huge assembly model where the exact solution does not exist. Also, a convergence rate is derived for each assembly. This automated convergence study illuminates the advantages of the interconnection of COMSOL with MATLAB, which provides us autonomy in applying different levels of refinement, element type and storing statistics from the solver engine.

Reference

[1] Dietrich Braess. Finite Elements. Cambridge University Press, third edition, 2007.
[2] Matthias K. Gobbert. A techinique for the quantitative assessment of the solution quality on particular finite elements in COMSOL Multiphysics. COMSOL Conference 2007, Boston, MA, pp. 267-272, 2007

Figures used in the abstract

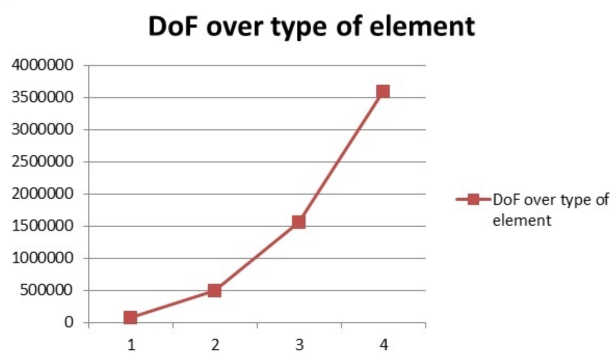

Figure 1: Cost of increasing element type in final number of Degrees of Freedom.

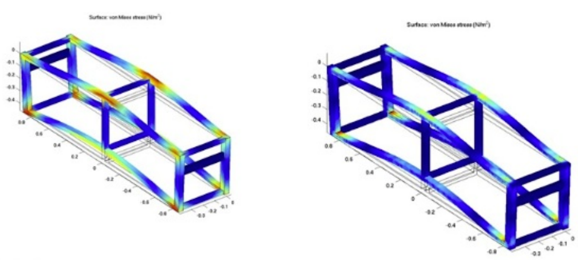

Figure 2: View of the VonMises Stress on the Rudder Frame for element type 1 and zero refinement on the left and type 1 and 4 refinements on the right.

Results:

r	N_e	N_v	DoF
0	1128	579	1737
1	3719	1515	4545
2	11052	3625	10875
3	33402	9123	27369
4	103115	24775	74325

Lagrange Elements with order p = 1 and nrefmax = 4

r	N_e	N_v	DoF
0	1128	579	8637
1	3719	1515	24141
2	11052	3625	62883
3	33402	9123	170922
4	103115	24775	491028

Lagrange Elements with order p = 2 and nrefmax = 4

r	N_e	N_v	DoF
0	1128	579	24054
1	3719	1515	69915
2	11052	3625	189150
3	33402	9123	530835
4	103115	24775	1559424

Lagrange Elements with order p = 3 and nrefmax = 4

r	N_e	N_v	DoF
0	1128	579	51372
1	3719	1515	153024
2	11052	3625	422832
3	33402	9123	1207314
4	103115	24775	3588858

Lagrange Elements with order p = 4 and nrefmax = 4

Figure 3: Reveiw of the Preleminary Results.