



HYPERMESH BASIC

1. Introduction.
2. Pre-processing for Finite Element Analysis.
3. Opening and Saving Files.
4. Working with Panels.
5. Organizing a Model.
6. Shell Meshing.
7. AutoMeshing.
8. Checking and Editing Mesh.
9. 3D meshing.
10. Analysis Setup.
11. Formatting Models for Analysis.
12. Setting up Loading Conditions.
13. Setting up the problem for solver
14. Submitting the results to the solver
15. Post processing the result