International Journal for Modern Trends in Science and Technology
Volume: 03, Issue No: 05, May 2017
ISSN: 2455-3778
http://www.ijmtst.com

Meshing Techniques for Finite Element Analysis in ANSYS

S B Bhatt[1] | Chirag P. Mistry[2]

[1] Assistant Professor, Mechanical Engineering Department, L .D. College of Engineering, Ahmedabad.
[2] P.G. Student, Mechanical Engineering Department, L .D. College of Engineering, Ahmedabad.

To Cite this Article
S B Bhatt and Chirag P. Mistry, "Meshing Techniques for Finite Element Analysis in ANSYS", *International Journal for Modern Trends in Science and Technology*, Vol. 03, Issue 08, August 2017, pp.-391-394.

ABSTRACT

A computational method called finite element analysis (FEA) is used to analyze the structure of complex shapes, apply complex loading conditions, and forecast stress levels. It functions according to the divide-and-concat principle, which splits a physical system with infinite unknowns into small, finite elements with a finite number of unknowns. It's known as meshing. The quality of the mesh will determine how accurate the FEA result is. The most important FEA analysis processes are element size, shape, and meshing method selection. The findings are more accurate when there are more elements, but this also means that more calculations are needed to find the solution. Verifying the convergence error for every iteration serves as a criterion for accepting the results.

Keywords: *FEA, Meshing methods, Meshing quality, Meshing refinement*

I. INTRODUCTION

At last, finite element analysis is being used often in engineering. In FEA, discretizing the continuum using different finite elements is the first stage. The geometric model's dimensions and form determine the degree of discretization. Depending on the loading state, model geometry, and type of application, this can be accomplished using a variety of elements. Various elements, such as beams, shells, and plates, are available in FEA analysis in 1D, 2D, and 3D depending on the geometry to be simulated and the amount of degrees of freedom needed at each node. From the same node, loads are converted from one element to another.

To locate stress concentration, the user should first model a structure with a coarse mesh and then refine it as needed.Meshing is done with the element's default size in the first iteration, and the mesh quality is then checked. The meshing quality and results are examined after each iteration. If the meshing quality is adequate and acceptable, the convergence error in the result is checked; if not, the mesh is refined using a different refining approach, which is covered later in this work. A percentage difference between the most recent two iterations is referred to as convergence error. The outcome is satisfactory if the convergence error is between 5% and 10%; otherwise, mesh refining is needed.

II. MESHING METHODS

A. *Tetrahedral meshing methods*

Tetrahedral meshing method used triangle element for meshing with following two types of patch method.

1) Patch conforming:

This method starts the meshing process from the edges and works its way up to the faces and volume. Every facet and its limits are reflected and interwoven. Good for CAD geometry of superior grade. Global and/or local control determine size.

2). Patch independent:

This method projects the volume mesh onto the face and edges using a top-down approach. Vertices, edges, and faces were not always in compliance.

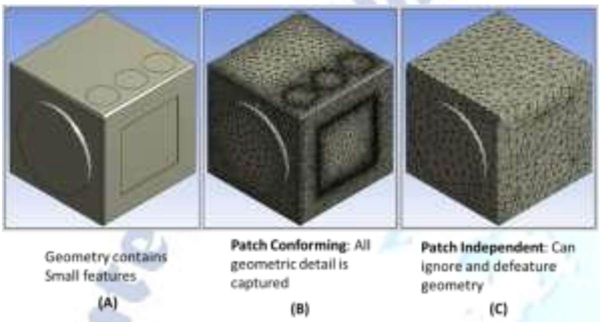

Fig. 1: Comparison of tetrahedral meshing methods

B. Hexahedral meshing methods

Because the hexahedral meshing method used fewer elements, the results were more quickly to converge. Components oriented in the flow direction Enhanced mesh quality and accuracy of results. It is Cut down on numerical error.

.

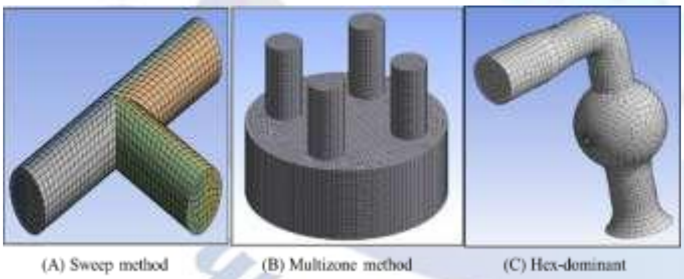

Fig. 2: Hexahedral meshing methods [3]

1) Sweep method

With this meshing technique sweeps through to the target after meshing the source surface. produces wedge/hex elements. The two ends of the body, which serve as the source and target faces, must have topologically identical faces. This method needs to be able to map side faces.

2) Multizone method

Geometry is automatically broken down by the multizone method.When feasible, it creates structured hex mesh using both structured and unstructured blocks; unstructured mesh is used to fill in the empty space. able to choose source and target faces manually or automatically. Depending on the mesh type (struct / free), the block will be meshed by either Tetra or Hex.

3). *Hex-dominant*

The mesh is made up mostly of hex cells, but there are also some tetra and pyramid cells as well. helpful for bodies that are difficult to sweep. helpful for CFD applications where inflation is not needed. beneficial for CFD within the allowable Skewed or Perpendicular Metrics for quality mesh quality

C. Surface meshing

- Default quad, quad/tri or tri
- Uniform quad or quad/tri

III. MESHING REFINEMENT

1. h-refinement (changing the element size)

2.p-refinement (changing to higher order polynomial interpolations)

3. hp-refinement (combination of h and p refinements)

4. r-refinement (keeps the number of nodes constant and adjusts their positions)

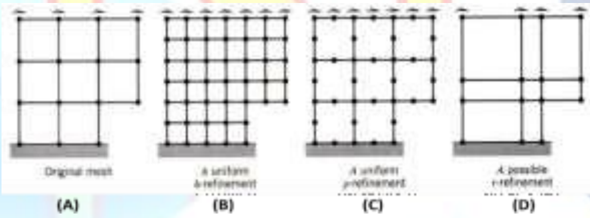

Fig. 3: Meshing refinements [7]

IV. MESHING QUALITY

The aspect ratio, skewness, orthogonality, and smoothness are the four criteria used to assess the solid mesh's properties. The most crucial criterion for assessing the attributes of each individual element is its aspect ratio. On the other hand, for two adjacent elements that share an inner face, skewness, orthogonality, and smoothness indicate the quality prediction. [9]

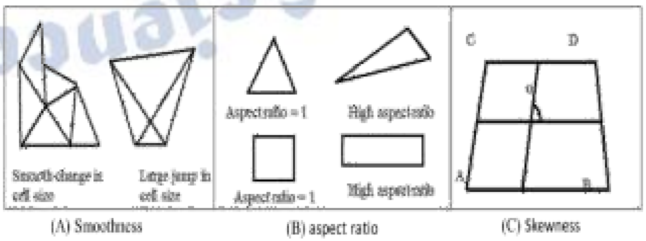

Fig. 4: Meshing quality

A. Aspect ratio

It is the proportion of a cell's longest to shortest side. To get the best results, it should ideally equal 1. If the flow is multidimensional, it should be close to one..

B. Smoothness

Additionally, the size change ought to be seamless. Sudden changes in the cell's size should be avoided as they could lead to incorrect findings at neighboring nodes.

C. Skewness

To find the skew of quad elements, first connect the midpoints of each side to the opposite side's midpoint. Then, use the smaller of the two angles to find the angle α, as shown in Figure 3-10, making sure that α is less than 180 degrees. Usually, the result is normalized by dividing the angle by 180 degrees.

V. CONVERGENCE CHECK

A precise finite element analysis necessitates the measurement and assessment of multiple error kinds. a result that differ between the current and previous run by less than 10%. Compared to first order elements, higher order elements are more accurate. A higher mesh density translates into a higher level of simulation accuracy. However, because of the increased element count, more computation time is needed in the interim. Consequently, accuracy and efficiency are always traded off.

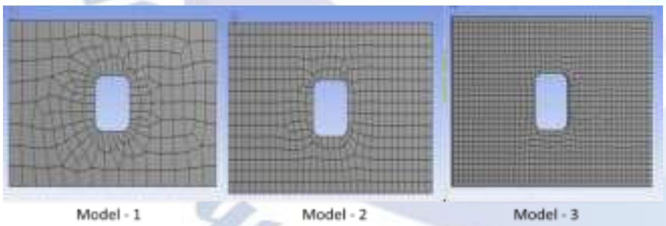

Fig. 5: Model refinements for convergence check [1]

Model no.	Element Size (mm)	Number of Element	Maxi. Principle Stress (Mpa)	Convergence error In %
1	4	834	180	--
2	2	2047	203	11.33 %
3	1	14839	212	4.24 %

Table. 1: Meshing summary for convergence error

As illustrated in Figure 5, consider a single, rectangular plate with a certain thickness. Refine the mesh in each model according to Figure 5, halving the element's size.

Verify each model's convergence error, and if it's acceptable, stop refining the mesh.

VI. ILLUSTRATION FOR MESHING REFINEMENT

Technologies for efficient heat transfer are necessary for magnetically confined fusion reactors. As a result, several high heat flux devices have been created especially for this use. The hypervapotron, a water-cooled device that maximizes heat transfer capacity through internal fins and boiling heat transfer, is one of the most promising candidates. The Indian test facility at IPR uses hypervapotron for heat transfer in its beam line component.Thermal (body temperature) and structural (water pressure) loading were applied to it.The hypervapotron's geometry is depicted in Figure 6. The element quality and connectivity of the first two iterations of ANSYS auto meshing were not good enough for acceptance.Thus, the hyper mesh meshing module was used for the last two iterations. Hypervapotron geometry is intricate, and.

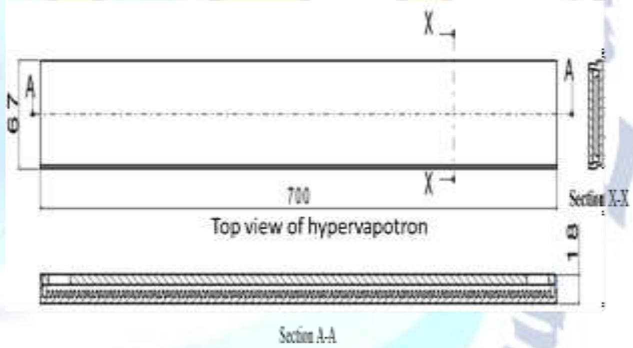

Fig. 6: Geometry of hypervapotron

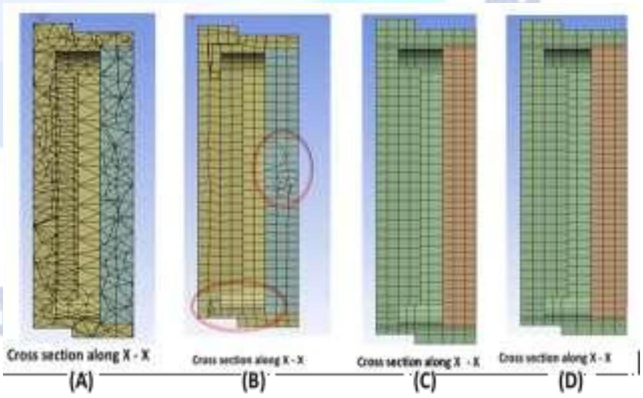

Fig. 7: Meshing refinements iteration for Hypervapotron at cross section x-x

Table. 2: Meshing iteration summary for hypervapotron

	Iteration I	Iteration II	Iteration III	Iteration IV
Meshing Method	Tetrahedral (Auto meshing)	Hexahedral (Auto meshing)	Hyper meshing single order	Hyper meshing higher order
Type of element	SOLID87	SOLID96	SOLID185	SOLID186
Element shape	Tetrahedron	Hexahedron	hexahedron	hexahedron
Nub. Of Element	185647	170147	145658	145658
Num. of nodes	329200	293100	200981	748330

VII. SUMMARY

FEA is an approximation technique for solving difficult problems. The geometry features of the part determine which element size and meshing technique to use. If at all possible, eliminate fillets from geometry to make it simpler and eliminate superfluous parts from geometry. To obtain accurate results, all of the element's edge lengths must be a small fraction of the smallest feature's size. Models created with this mesh density are frequently too complex to analyze.In these circumstances, second-order elements are highly beneficial. Second order elements can more accurately model complex solids with fewer elements because they are not limited to straight line edges.

This study's main goal was to provide meshing guidelines and acceptance criteria based on meshing quality and needed result accuracy. You can use this to apply this methodology to any type of complex meshing problem in an ANSYS simulation.

References

[1] "Avoiding Pitfalls in FEA," Paul Kurowski, Machine Design, November 7, 1994.

[2] Joseph Milne's, Alan burns, dimitrs drikakis," Computational modelling of the hypervapotron cooling technique", fusion engineering and design 87 (2012) 1647-1661.

[3] ANSYS 13 Meshing user manual.

[4] Farazdak haideri (2011),"CAD/CAM and automation", seventh edition, McGraw-Hill Companies Inc.

[5] Tirupathi R. chandrupatla,"Finite element in engineering", third edition, prentice-hall of India.

[6] The Finite Element Method in Mechanical Design, Charles E. Knight, Jr., PWS-Kent, 1993.

[7] Abdullah alsahly and shorash miro,"Adaptive finite element" short presentation at Rahr university Bochum.

[8] A. Entrekin, "Accuracy of MSC/NASTRAN first- and Second-order Tetrahedral elements in solid modeling for stress analysis"

[9] Meshing quality definition, http://help.plastics-u.com/online-help/moldex3d-pre-processing-tools/moldex3d-mesh/mesh-quality-definition/.