Research Paper

FREE FREE RUN ANALYSIS FOR 2D MESHING

Vikrant C. Gaikwad[1], Dr. Suresh M. Sawant[2]

Address for Correspondence

[1]Research Scholar, [2]Professor, Department of Mechanical Engineering, RIT Sakharale 415414, Sangli, Maharashtra, India

ABSTRACT

Finite Element Method reduces the degrees of freedom from infinite to finite with the help of meshing. Meshing is the most important step in finite element analysis. In meshing, mesh quality and mesh connectivity are important consideration for further analysis. In Free-Free run analysis six rigid modes indicate al1 the parts in the assembly are properly connected to each other. The mesh quality may have considerably on the analysis in terms of the quality of the solution and the time required for it. In process, meshing is done with HYPERMESH software and solution is acquired using NASTRAN solver.

KEYWORDS: Finite element analysis (FEA), Meshing.

1. INTRODUCTION

Meshing of the domain into finite elements is the first step in the finite element method. This is equivalent to replacing the domain having an infinite number of degrees of freedom by a system having finite number of degrees of freedom. The shape, size, number and configuration of elements have to be chosen carefully so that the original body or domain is simulated as closely as possible without increasing the computational effort needed for the solution.

Meshing is (at least optionally) a highly automated process, mesh quality, its connectivity (i.e. compatibility).

HyperMesh enables to receive high quality meshes with maximum accuracy in the shortest time possible. A complete set of geometry editing tools helps to efficiently prepare CAD models for the meshing process. Meshing algorithms for shell and solid elements provide full level of control, or can be used in automatic mode. HyperMesh offers the biggest variety of solid meshing capabilities in the market, including domain specific methods such as Noise Vibration and Harshness (NVH), Fatigue, crash and Optimization.

2. FINITE ELEMENT ANALYSIS (FEA)

The finite element method is used in a wide variety of disciplines and engineering applications. In the beginning, the use of these techniques was customary only in the aerospace and nuclear fields. Subsequently, the use has spread to a variety of products, physical situations, and manufacturing processes. Some of the interesting features of FEA are structural analysis, Noise Vibration and Harshness (NVH), Fatigue, crash and Optimization.

In order to obtain better design in FEA, following procedure is given below.

2.1 Initial design and geometry generation

Initial design of the model is a planning decision and the geometry is generated depending on these initial design considerations, using modelling tools.

3.2 Mesh Generation

Basic theme of FEA is to make calculations at only limited (Finite) number of points & interpolate the results for entire domain (surface or volume). Any continuous object has infinite degrees of freedom & it's just not possible to solve the problem in this format. Finite Element Method reduces degrees of freedom from Infinite to Finite with the help of discretization i.e., meshing (nodes & elements).

HYPERMESH is a powerful tool that lets designers and analysts of component create high-quality meshes, while preserving the underlying geometry. Following steps in hypermesh is given below.

3.2.1 Geometry Cleanup

Generally CAD data is provided in igs format. Geometry cleanup is an integral part of meshing activity. Before starting the meshing, geometry should be carefully checked.

Before starting any mesh geometry, it is a good to look over the part and develop a strategy for how you are going to go about meshing the part.

1. Spend sufficient time in studying the geometry
2. Time estimation
3. Geometry check
4. Symmetry check
5. Selection of type of elements
6. Type of meshing
7. Joint modeling

3.2.2 Decide the Element Type

Element type selection is based on Geometry size and shape, Type of analysis, Time allotted for project, Hardware configuration. The geometry can be categorized as 0-D, 1-D, 2-D, or 3-D based on the dominant dimensions and then accordingly that type of element is selected.

3.2.2.1 Types of Elements

There are different types of elements for discretization. Fig.1. shows the different types of elements used for meshing.

- 0 D Element: Scalar element
- 1D Element: Rod, Bar, Beam
- 2D Element: Shell, Membrane, Plane stress, Plane strain
- 3D Element: Solid

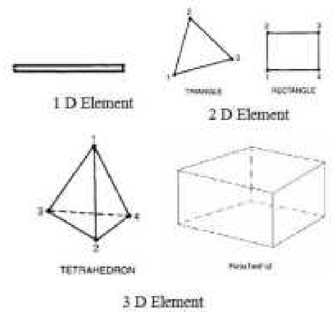

Fig.1. Different types of elements

Once the geometry is in an appropriate state, a mesh will be created to approximate the geometry. Either a beam mesh (1-D), shell mesh (2-D) or a solid mesh (3-D) will be created. This meshing step is crucial to the finite element analysis as the quality of the mesh directly reflects on the quality of the results generated. At the same time the number of elements (number of nodes) affects the computation time. That is the reason why in certain cases a 2D and 1D mesh is preferred over 3D mesh.

3.2.3 Meshing the Geometry

Once the geometry is in an appropriate state, a mesh will be created to approximate the geometry. Either a shell mesh (2-D) or a solid mesh (3-D) will be created. This meshing step is crucial to the finite element analysis as the quality of the mesh directly reflects on the quality of the results generated.

3.2.4 Quality parameter for meshing

Acceptance criteria of model quality are considered when meets the body mesh model Quality check list concerning the various mesh quality parameters like skew, aspect ratio, Jacobian etc. are the measures of how far a given element deviates from ideal shape. Some of the qualities checks are based on angles (like skew, included angles) while others on side ratios & area (like aspect, stretch).

Some of the terms used when checking element quality include:

3.2.4.1 Warpage

It is a measure of how to close a QUAD element is to being planer. A perfect planer element will have the warpage of zero. Warpage of up to five degrees is generally acceptable.

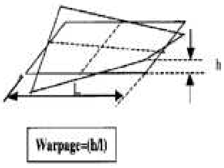

Fig.2. Warpage

3.2.4.2 Skew

The angles between the lines join opposite midsides. It Measure the angle created as square is turned into parallelogram or rhombus. Typical required values are to have less than 45^0 or 60^0.

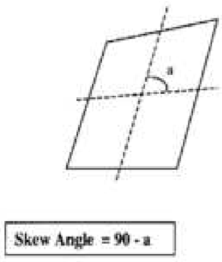

Fig.3. Skew

3.2.4.3 Aspect Ratio

It is the ratio of max length side of element to min length side of an element. This should be reduced as much as possible.

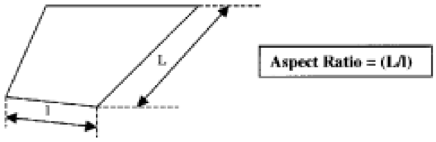

Fig.4. Aspect ratio

3.2.4.4 Quad Angle

The angle between two sides of a quad element should be 90^0 as much as possible. Typical required values are to have all angles between 45^0 and 135^0.

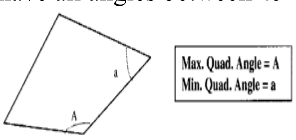

Fig.5. Quad angle

3.2.4.5 Tria Angle

The Angle between two sides of a tri element should be 60^0 as much as possible. Typical required values are to have all tria angles between 20^0 to 120^0. Some time smaller angles are require to model geometry with small angle.

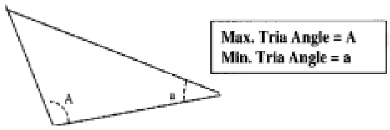

Fig.6. Tria angle

3.2.4.6 Jacobian

It is measure of close the shape of element is to ideal shape. Jacobian is really the best measure of finite element mesh quality. Typically less than 5% of element in a mesh should have Jacobian less than 0.7 within the minimum value of 0.5.

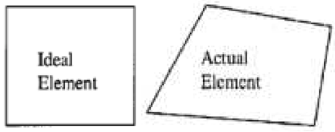

Fig.7. Jacobian

In process, two dimensional 2-D meshing have used. It is used when two of the dimensions are very large in comparison to the third one. 2-D meshing is carried out on a mid surface of the part. By creating 2-D elements, the software knows 2 out of the 3 required dimensions. The third dimension, thickness, has to provide as an additional input data. Mathematically, the element thickness specified by the user is assigned half on the element top and half on the bottom side. Hence, in order to represent the geometry appropriately, it is necessary to extract the mid surface and then mesh on the mid surface. [1]

For meshing, Element type selection is based on Geometry size and shape, type of analysis, time allotted for project and hardware configuration. In this process we have used 2D meshing used with shell element.

The assumptions for FEA,

1. Weight of the radiator with coolant
2. Weight of the fan

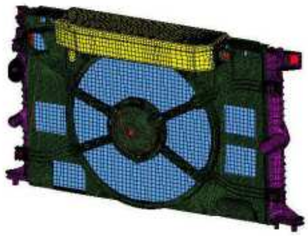

Fig.8. 2D meshing of engine cooling system model

2D meshing of engine cooling system model gives total number of elements are generated is 54368 in number. Total no. Of elements are categorised as below:

Trias-CTRIA3 : 3560
Quad-CQUAD4 : 50808
Rigid-RBE2 : 11
Mass-CONM2 : 2

3.2.5 Material and Property Information

After meshing is completed, material (e.g. Young's Modulus) and property information (e.g. thickness values) are assigned to the elements.

In Model basically Expansion tank, Radiator tank, shroud all parts are made up of plastic and radiator is made up of aluminium.

Table 1 - Material of Component

Material	Young's modulus(E) ($N/mm2$)	Passions ratio (μ)	Density (tonne/ mm^3)
Plastic	6100	0.35	1360e-9
Al	7e4	0.33	2.8e-9

2.3 Pre-Processing

Pre-processing involves creating the FE model and applying the necessary loads and boundary conditions.

In Free-Free run analysis has used with No constraints and No force as a boundary condition.

2.4 Processing (Solver)

The FEM model (consisting of nodes, elements, material properties, loads and constraints) is then exported from within the pre-processor Hypermesh. The exported FEM model, typically called solver input deck, is an ASCII file based on the specific syntax of the NASTRAN solver. In process, after applying the boundary condition in the pre-processor the model is solved in the NASTRAN-Solver.

2.5. Post-Processing

Post-processing provides an in depth view of data with visualization of various loading conditions.

The model which is solved shows results in Hyperview software.

3. RESULTS AND DISCUSSION

Element quality and Free free run analysis results are discussed below.

3.1 Element Quality Check

The mesh quality may have considerably on the computational analysis in terms of the quality of the solution and the time required for it.

Acceptance criteria of model quality are considered acceptable when it meets the Quality check list with various mesh quality parameters such as aspect ratio, warpage, percentage of trias in the mesh and the most important parameter, the Jacobian or distortion of the mesh from an ideal shape.

Table 2- Two dimensional (2D) Quality report

Quality Check;	Limiting Value;	% of Failed Elements
Mesh type	→	Manual mesh
Warpage >	10	1
Aspect Ratio >	5	0
Skew Angle >	60	0
Jacobian <	0.7	2
Min Angle For Quads <	40	0
Max Angle For Quads >	135	0
Min Angle For Trias <	13	7
Max Angle For Trias <	140	0
% Of Trias	13	4
Total Of Trias & Quads Checked	**Total=**	54368

From the above table 2 of different mesh quality report shows that how much percentage of failed elements. The above mesh parameters which have been generated for 2D mesh are within acceptable range of failure [1].

3.2 Check the free-free run analysis for assembly

Method used to check the connectivity of the mesh is the free-free run method. With this method we can checking of the connectivity of the meshing is valid or not.

So, according to table 3 first Six Mode shapes have a negative or zero frequency and after that the mode shapes have a positive value, which satisfies the condition for the Free-Free run analysis [1].

Table 3- Frequencies for various mode shapes

Mode Shapes	Frequency in Hz
1	6.286937E-04
2	7.852287E-04
3	8.415252E-04
4	9.716590E-04
5	1.007735E-03
6	1.188928E-03
7	3.095597E+01
8	4.190256E+01
9	4.560462E+01

4. CONCLUSION

In this research work following software like Hypermesh for meshing, Nastran as a solver and Hyperview for visualized result are used. Free-Free run analysis, is carried out for checking the mesh continuity of the model and it is found that it is valid for the further analysis.

ACKNOWLEDGEMENT

I wish to thank Behr India limited, Pune-410501, and Maharashtra, India for giving me an opportunity to work in this field. The guidance, cooperation, practical's approach & inspiration given by company especially by Mr. K. Parmeshwar (Engine Cooling Head), Mr. S Prakash (Manager), and Mr. Abasaheb Gaikwad (Manager at Tata Motor's Limited, Pune) provided me the much needed impetus to hard work.

REFERENCES

1. Nitin S Gokhale, Sanjay Deshpande, Sanjeeve V. Badekar, Anand N. Thithe, "Practical Finite Element Analysis", 1st Edition, ISBN978-81-906195-0-9.
2. Users Reference Manual for the Hypermesh.
3. Users Reference Manual for the MYSTRAN (NASTRAN) General Purpose Finite Element Structural Analys is Computer Program (Nov 2011).