Meshing technique in FEA for complex shape model and result optimization

Vikas Shukla
vikas.shukla.y@gmail.com
Dr. A.P.J. Abdul Kalam Technical University, Lucknow, Uttar Pradesh

Mahendra Pratap Singh
mahendra05feb@rediddmail.com
Dr. A.P.J. Abdul Kalam Technical University, Lucknow, Uttar Pradesh

Amit Kr Srivastava
amitme.srivastava@gmail.com
University College of Engineering and Technology, Vinoba Bhave University, Hazaribagh, Jharkhand

ABSTRACT

In this article the technological application of finite element analysis, meshing is one of the most critical steps. The quality of meshing directly affects the accuracy and speed of the solution. There are three steps in meshing: defining cell properties (including real constants), defining mesh properties on a geometric model, and meshing. Dynamic interaction of a proposed methodology for flexible structures subjected to mobile loads is presented. In the first place, the modeling of a mobile finite element mesh is described, which moves on the contact of a crack in a manner that is integral to the engine that runs through the catenary and, subsequently, the comparison of this methodology is presented against a conventional fixed-mesh finite element model. The simulated case corresponds to that proposed by the standard, obtaining results according to the revalidation intervals proposed by the said standard. The main advantage of the mobile mesh method lies in the fact that for analogous precision, the calculation time is of the order of 4 times less than the classical model.

Keywords— Meshing element, FEA, ANSYS 17, Mesh optimization, Result evaluation

1. INTRODUCTION

Engineers and researchers use Finite Element Analysis (FEA) software to build predictive computational models of real-world scenarios. When using finite element analysis software, we usually start with information about Computer-Aided Design (CAD) models, material properties, applied loads, and constraints that characterize the parts of the material that need to be simulated. This information is usually very accurate and helps people predict the real world.

The accuracy obtained based on any finite element analysis model is directly related to the finite element mesh used. The finite element mesh is used to segment the CAD model into many smaller domains, which we call cells, and then solve a set of equations on these cells, which are approximated by a set of polynomial functions defined on each cell. The required governing equation. As the mesh continues to refine, these units become smaller and smaller, making the results of the solution closer to the real solution.

The mesh refinement process is a key step in our validation of the finite element model and confidence in the software, models, and calculations.

Engineering Analysis Process in ANSYS Advanced Simulation Usually for a preliminary assessment of the strength / performance of an engineer's construction the constructor applies engineering approaches based on the design representation in the form of simple nodes and elements for which there are analytical estimates of the search for the deformed state. These estimates include the use of simple formulas for the search for stresses in beams during their stretching, bending or torsion, searching relative lengthening, moments of inertia, reaction forces, etc. The design engineer has to den work with a large amount of specialized literature to find the necessary expressions and laws. These approaches have significant limitations in their application. To real complex structures and are used less and less in modern high productions and design. When using numbering systems Lennon analysis engineer gets the opportunity to simulate structures and machines of any difficulties with any necessary (rational) level of detail.

2. FEA MESHING ELEMENTS AND ITS PROPERTIES

A tool for analyzing the actual distribution of stresses and strains in a structure. ANSYS advanced Simulation based on the ANSYS Nastran industrial solver (and other Siemens PLM Software) gives the engineer the opportunity to work with various applications, toying in a single ANSY design environment. Moreover, the scalability of the module ANSYS Advanced Simulation (ANSYS Advanced Simulation) allows you to solve both simple and very complex problems from various areas of deformable solid mechanics, fluid mechanics, and gases, heat transfer mechanics, etc. The main steps in performing engineering analysis using the finite element method are Meshing overview Meshing involves the shape of the element and its topology type, cell type, mesh generator selection, mesh density, Unit number, and geometry voxel. For example,

in terms of geometric expression, the beam and the rod are the same, but they are solved from the physical and numerical values.
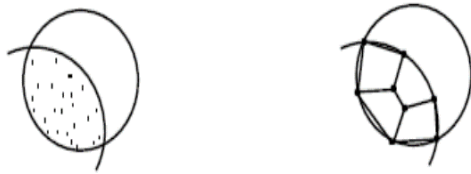

Fig. 1: View of numerical meshing

No. of Points = ∞   No. of Nodes = 8
DOF per Point = 6 Total   DOF per Node = 6
equations = ∞   Total equations = 48

FEA is to make calculations at an only limited number of points and then interpolate the results for the entire domain (surface or volume).
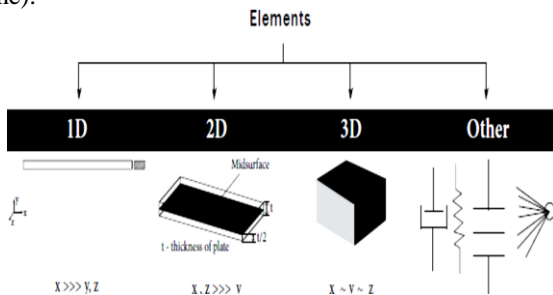

Fig. 2: Meshing Element type

2.1 Element Type Selection
- Geometry size and shape
- Type of analysis
- Time allotted for the project

There is a difference in the above. Similarly, the unit solving equations for plane stress and plane strain conditions are also different. Due to the stiffness matrix of different units is different, and the solution method using numerical integration is different. Therefore, in practical applications, it must be adopted. Simulate the solution with a reasonable unit. ANSYS provides two methods for creating finite element models, namely direct method and solid model meshing modeling. Direct method. First, create a node, then use the node to create a unit, single or multiple units form a limited.

2.1.1 The Tetrahedrons: method allows you to generate volumetric meshes with elements having the shape of tetrahedral, based on one of two methods: Patch Conforming and Patch Independent (the Algorithm section of the Tetrahedrons method).
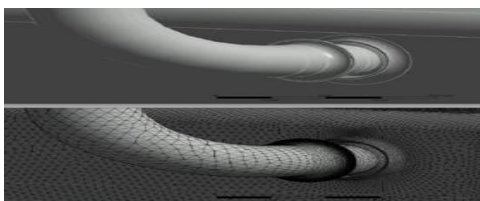

Fig. 3: Generate volumetric meshes with elements

2.1.2 The Patch conforming method: is preferable if the initial geometry contains fine details and at the same time is of high quality, that is, it lacks problem areas resulting from incorrect construction in CAD. The method takes place in 2 stages:
- First, the edges and surfaces of the model are split;
- Then the construction of tetra-elements is carried out in the volume of the model, taking into account the already constructed surface mesh. Grid construction is possible either using the Advance Front algorithm or in Program Controlled

mode with automatic selection of the more suitable of two algorithms: Front or Delaunay (based on the construction of a triangulation that satisfies the Delaunay criterion). To set the construction algorithm, you need to select the appropriate option in the global grid settings window in the Advanced-Triangle Surface Mesh section.
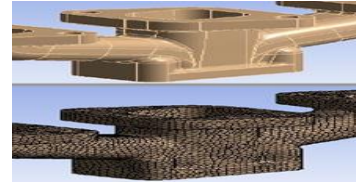

Fig. 4: Generate edges and surfaces meshes

2.1.3 The Patch Independent method: in contrast to Patch Conforming, is preferable to use when the geometric model is of low quality and contains problem areas in the form of gaps, small protrusions, etc. It allows you to select the level of tolerances (Tolerance) of the grid from the original CAD model and filter thus some of the details of the geometric assembly, the size of which is less than a specified level of tolerance. This method is based on the boundary correction algorithm. The main idea of the method is to impose a certain grid on the calculated area and then cut off all fragments that extend beyond the geometric region. Unlike the Patch Conforming method, a volume grid for the geometric region is first constructed, and then the surface mesh is corrected in accordance with the surface features of the model and the established tolerances.
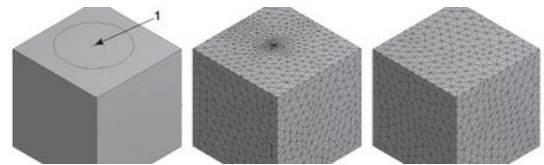

Fig. 5: Generate Patch surfaces meshes

To understand the difference, the grid on the middle die was constructed using the Patch Conforming method, and on the right, using the Patch Independent method with the Disfeaturing Tolerance option.

2.1.4 The Sweep method: allows building a computational grid on the basis of prismatic elements using the operation of pulling elements of one layer along a certain axis. This method can be used only for a certain class of geometric models, obtained as a body of rotation/pulling.
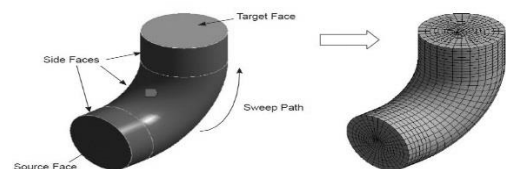

Fig. 6: Generate Patch Independent surfaces meshes

To build a grid by pulling mesh elements, the geometric model must contain a source face (Source Face), the elements of which will be pulled along a certain Sweep Path in the direction of the target surface (Target Face). The Src / Trg Selection option of the method includes five modes for specifying the Source and Target surfaces:

- **Automatic:** automatic determination of the surfaces most suitable for performing the pulling operation;
- **Manual Source:** select the source surface manually, and the receiver surface is determined automatically. This option is

useful when the model section along with the pull line changes and the cross-section sizes of the source and receiver surfaces do not match;

- **Manual Source and Target:** we select the source surface and the surface receiver manually.
- This option is useful for building a mesh by pulling for a body where the source surface and the receiving surface have common nodes or edges;
- **Automatic Thin and Manual Thin modes:** designed to build a grid on the shell-bodies (for example, sheet metal), for which you need to build a grid based on Hexa-elements with a single grid element across the body thickness.
- **There are two options in the properties of the Sweep method that are not found in other methods:** Type and Sweep Bias Type. The Type option is responsible for the size of the extricable elements (thickness) along the Sweep Path. We can set the number of grid elements along the pulling path or the thickness of the grid element along the path of pulling. By default, the thickness of all the elements when pulling is the same, that is, the grid in this direction is uniform. To thicken the grid to one of the surfaces (Target / Source) or to both surfaces at the same time, use the Sweep Bias Type option, which determines the direction of thickening, and the Sweep Bias option, which determines the degree of element thickening.
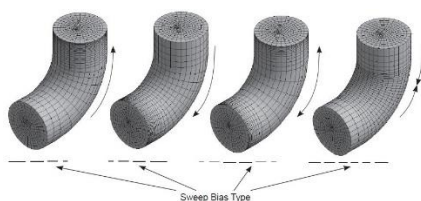

Fig. 7: Sweep Bias Type option

The Hex Dominant method allows building grids based on hexahedron elements and can be used where the use of the Sweep method is impossible. When constructing a grid using the Hex Dominant method, it is necessary to take into account that the original geometry must be of high quality and in some cases, its division into simple geometric bodies is required.
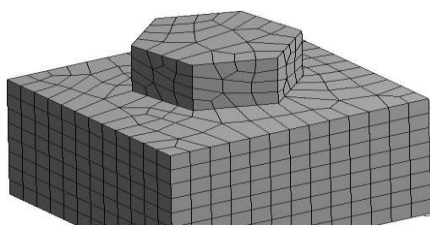

Fig. 8: Hex Dominant method

This is an example of using the Hex Dominant method with the Quad / Tri option for a surface.

The MultiZone method is based on block technology and provides automatic separation of geometry into two groups of geometric bodies: volumes to which the Sweep method can be applied to construct a Hexa net with pulling grid elements along a certain axis, and other volumes for which the Hexa net will be constructed unstructured type. For example, such a plate with pins. To apply the usual Sweep method to such a model, it would be necessary to manually divide it into several Sweepable Bodies and then, for each body, choose your own Sweep method and source surface for pulling the cells. The advantage of the MultiZone method is that it can be divided automatically and select several source surfaces in the properties of the method for applying the Sweep operation.
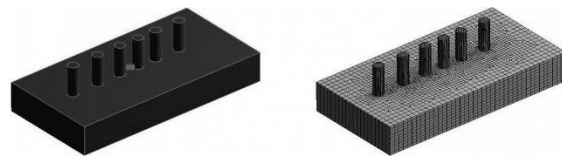

Fig. 9: Several Sweepable Bodies

We now turn to the options of the Multi Zone method:

2.2 Mapped Mesh Type
This sets the shape of the cells for areas where a structured mesh is supposed to be built. There are three modes:

2.2.1 Hexa: involves the construction of elements of exclusively hexahedral form.

2.2.2 Hexa / Prism: involves the construction of elements based on hexahedra, as well as elements of a prismatic form. At the same time, to improve the quality of individual grid cells, surface elements of a triangular cross-section can be used, which are further extended to prisms.

2.2.3 Prism: involves the construction of the grid exclusively from elements of a prismatic shape. This mode is used in cases where one geometric area for which the grid is built using the method MultiZone, bordered by another area filled with tetrahedron- based mesh.

2.3 Surface Mesh Method
This defines the model for building a grid on the surface of a geometric region. Uniform mode uses a step-by-step method to build the most uniform mesh on the surface. This method is best suited for cases where all faces of an area have the same scale, and surfaces covered by a grid have a low degree of curvature. In these cases, the method allows obtaining a grid of sufficiently high quality. Pave mode, on the contrary, is applicable for building high-quality mesh on surfaces with a high degree of curvature.

2.4 Met model. Meshing method
First, create a solid model and then use the meshing tool to divide the solid model into finite Met model. Regardless of which finite element model creation method is used, the following preparations are required. Basic steps for creating a finite element model by the direct method. The basic steps for creating a finite element model directly are as follows:
(a) Define the unit type.
(b) Define the unit real constant.

3. METHODOLOGY
The finite element model required for meta-analysis. Finite element meshing is a crucial step in finite element numerical simulation analysis. It directly affects the accuracy of subsequent numerical calculations. This chapter focuses on how to mesh a solid model. It includes preparations before the meshing, free partitioning, and mapping partitioning. The properties of the defined grid are mainly to define the shape and size of the unit. The unit size is basically defined on the line segment and can be divided by the number of line segments or the length, which can be declared immediately after the online segment is created, or declared one by one after the entire entity model is completed. When the model is built using the Bottom-Up method, it is convenient and not easy to make mistakes after the line segment is established. For example, after declaring the number and size of line segments, the properties of the object will be copied together when the object is copied. After the above operation, the gridding command can be performed.
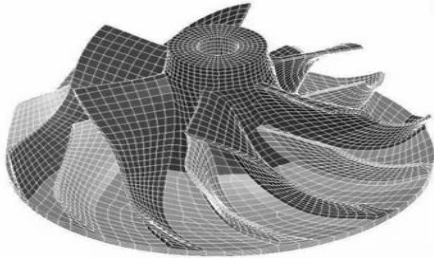
Fig. 10: Complex meshing of a model

The meshing process can also be carried out step by step, that is, the solid model object is finished to a certain stage and the mesh is spoken. If the obtained result is satisfactory, other objects are continuously built and meshed.

The division of the grid can be divided into free meshing, www.featech.com.cn mapped meshing and sweep meshing. First, free meshing is one of the most automated meshing techniques. It automatically generates a triangle or quadrilateral mesh on the surface and automatically generates a tetrahedral mesh on the body. In general, ANSYS's intelligent size control technology (SMART SIZE command) can be used to automatically control the size and density of the grid, or manually set the size of the grid (ASIZE, LESIZE, KESIZE, ESIZE, etc.) Control the density distribution and select the network algorithm, etc. (MOPT command). For complex geometric models, this method of netting saves time and effort, but the disadvantage is that the number of cells is usually large and the computational efficiency is reduced. At the same time, because this method can only generate tetrahedral elements for 3D complex models, in order to obtain better calculation accuracy, it is recommended to use quadratic tetrahedral elements if a hexahedral element is selected, this method automatically degenerates the hexahedral element into a uniform tetrahedral element. Therefore, it is best not to use a linear (first-order) hexahedral element because the unit is linear and tetrahedral, it has excessive stiffness and poor calculation accuracy. If a secondary hexahedral element is used, the number of nodes is consistent with its hexahedral prototype because it is a degenerate form. Only a plurality of nodes is in the same position. Therefore, the TCHG command can be used to change the degenerate tetrahedral unit in the model to a non-degenerate tetrahedral unit reducing the number of nodes in each unit. Improve the efficiency of the solution. And so, we continue to master meshes in ANSYS.

As part of the three-dimensional grids, they use their own methods: Tetrahedrons, Sweep, Hex Dominant, Multi-Zone, and Automatic. And now more.

3.1 Free Mesh Type
This defines the shape of the mesh elements for areas of geometry for which it is impossible to construct a structured mesh based on the Sweep. For such areas, one of the methods for constructing an unstructured grid can be applied:
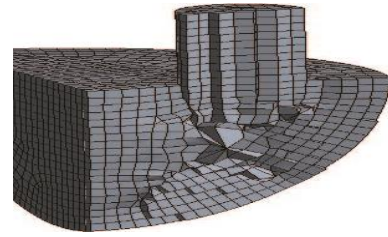
(a) Tetra (elements built on the basis of tetrahedra);
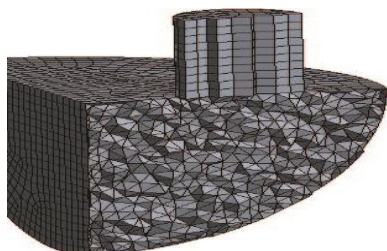
Fig. 11: Free Mesh Type

(b) Tetra / Pyramid (elements built on the basis of tetrahedral and pyramids).
(c) Hex Dominant (predominance of elements based on hexahedra);

Fig. 12: Free Predominance Mesh Type

(d) Hexa Core (the main volume of the region is filled with Hexa-elements in the Cartesian coordinate system, the rest of the region is filled with elements of a prismatic shape or tetrahedra).
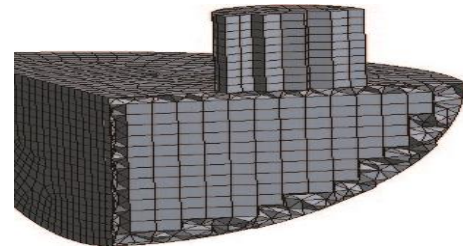
Fig. 13: Prismatic Shape or Tetrahedra

The Free Mesh Type parameter can also be set to Not Allowed, in which the construction of an unstructured mesh is unacceptable. In this case, the preprocessor will forcefully build a structured grid over the entire computational domain, and if this proves impossible, it will complete the construction process with an error.

3.2 Src / Trg Selection
This similar to the option of the Sweep method, it is necessary to select source surfaces for pulling in automatic mode (Automatic) or manually (Manual Source). In Manual Source mode, you must select in the Source field all surfaces in the geometry that will be used as source surfaces for the Sweep method.

Automatic. By default, the automatic method Automatic, which uses the Sweep method for volumetric bodies and the quadrilateral elements for surface grids, is used as the method of grid generation. In cases when the construction of the grid by the Sweep method is not possible, the Automatic mode uses a volumetric grid based on tetrahedra constructed using the front advancement algorithm as part of the Patch Conforming Tetrahedron method. I understand that this may be difficult to understand, but for writing this article I had 2 reasons:
(a) After the first article, many people became interested in the topic and wanted to study.
(b) Without understanding how to build the correct grid, the calculation accuracy will not be high.

If this is really interesting, then I will continue. Just as I began to write about the parameters of the grids - the interest is frightening, that is, it is either difficult or the people are disappointed

3.3 Grid refinement process
Often, we first need to understand the physical system under study, as well as a complete description of the geometry of the system in order to successfully perform finite element analysis. The geometry can be represented by a CAD model. A typical

CAD model can accurately describe the shape and structure of a research object, but usually also contains some modified features or manufacturing details; it turns out that this information is often independent of finite element modeling. The simulation analyst should perform some engineering judgments on the CAD model and decide whether these features and details can be removed or simplified before meshing. Starting with a simple model and gradually increasing complexity is often much easier than starting with a complex model and then simplifying it.

Analysts should also be aware of all the physics, material properties, loads, constraints, and all other factors that may affect the results of the analysis. These inputs may be subject to uncertainty. For example, we don't always know the material properties and loads accurately. It is important to keep this in mind during the modeling process, as it is not wise to try to make the analytical precision of the model exceed the effective accuracy of the input data.
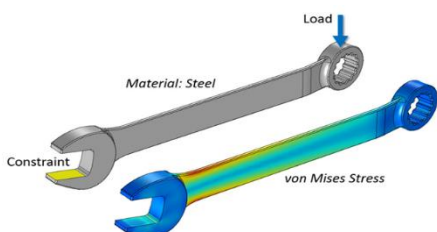

Fig. 14: Wrench finite element model and calculated stress

The grid is not shown in the figure. Once all of this information is aggregated into the finite element analysis model, the analyst can begin operations from the preliminary grid. In the early analysis process, it is better to start with the coarsest possible mesh (the grid cells are very large). The computational resources required to solve a coarse mesh are small, although the solution obtained may be very inaccurate, but it can still be used as a rough verification method and can check external loads and constraints.

Fig. 15: Starting with a very rough mesh, the first few iterations of the mesh refinement study of the wrench

After the solution is completed based on the coarse mesh, the mesh refinement process can be started. In terms of the simplest form of mesh refinement, this is a process of resolving models with an increasingly refined mesh and comparing the results obtained under different meshes. Results can be compared by analyzing the physical field at one or more points in the model, or by integrating a physics field on certain domains or boundaries.

Based on the comparison of these scalars, it is possible to determine the convergence of solutions obtained by some degree of mesh refinement. After at least three successively obtained solutions are compared, the asymptotic behavior of the solution begins to appear, and the difference between the solutions obtained by the different meshes becomes smaller and smaller. Ultimately, these differences will be small enough that analysts can assume that the model has converged. The specific criteria for judgment are always determined by the analyst because he/she knows the model input and the acceptable uncertainty in the results.

3.4 Different metrics for mesh refinement

When studying convergence, you need to choose the appropriate mesh refinement metrics. This metric can be local or global. That is to say, this metric can be defined either at a certain location in the model or as a field integral in the entire model space. The displacement or stress at a point in structural analysis is an example of a local metric. The integral of strain energy density over all domains is an example of a global metric. Both stress and strain are calculated from the solution's gradient and displacement field, and the solution's gradient is always calculated to the lower order of the polynomial approximation.
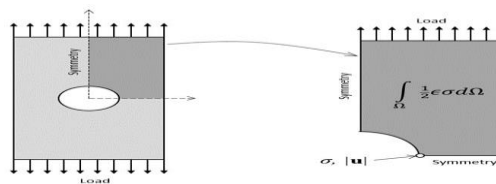

Fig. 16: A simple finite element model of a perforated carrier plate

You can reduce the size of the model by using symmetry, and you can define multiple different metrics to study mesh refinement.

When choosing metrics, it's important to remember that different metrics produce different convergence behaviors. This is illustrated in the figure below: the figure shows the use of different meshes to solve the same finite element analysis model. These grids have different cell sizes and are compared according to the Degree of Freedom (DOF) in the model. Degree of freedom node related to the number, "node" is defined for each calculation point of the finite element shape functions. The computational resources required to solve the finite element analysis model are directly related to the degree of freedom.

As you can see from the figure below, some metrics converge faster, but it should be noted that the grid convergence speed of a particular problem depends on the meshing technique used.
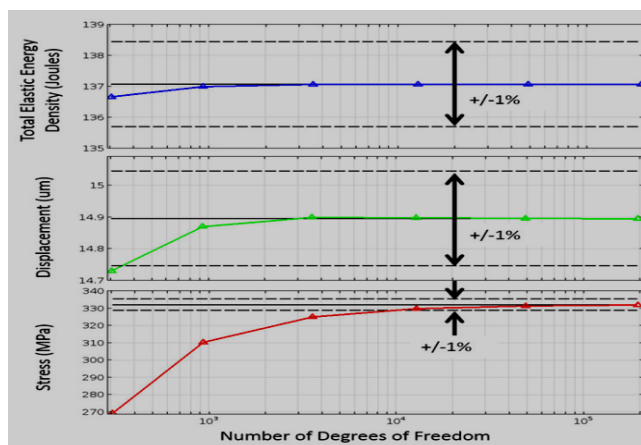

Fig. 17: Use global metrics (top), local metrics based on physics (middle), and local metrics based on solution gradients (bottom); and show % compared to the most accurate solution Error bar). The three cases used the same grid.

3.5 Different mesh refinement techniques

As far as grid refinement is concerned, a common set of technologies has been developed. Experienced finite element analysis software users should be very familiar with these technologies and understand their respective strengths and weaknesses.

3.6 Reduce the unit size

Reducing cell size is the simplest mesh refinement strategy, which essentially reduces the cell size of the entire modeling domain. Because of its simplicity and ease of use, this method is very attractive; however, it also has shortcomings. For local areas that require a finer mesh, it is not possible to selectively refine the mesh.
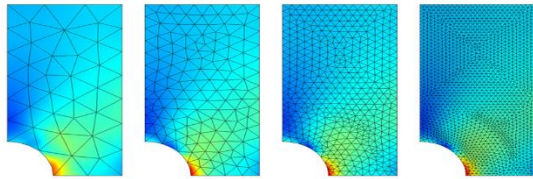

Fig. 18: Stress in a perforated plate solved by different cell sizes

3.7 Increase unit order

The advantage of the method of increasing the unit order is that there is no need to re-mesh; you can use the same mesh to change the cell order. Re-meshing is very time consuming when dealing with complex 3D geometries; on the other hand, we cannot make changes to meshes obtained from external sources. The disadvantage of this technique is that it consumes more computing resources than other meshing techniques.
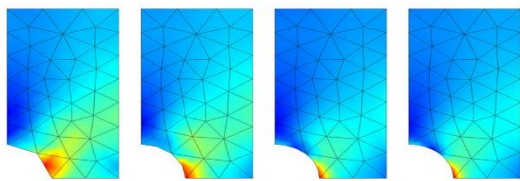

Fig. 19: The same finite element mesh solved by different unit orders

3.8 Global adaptive mesh refinement

The global adaptive mesh refinement method determines the point with the largest local error in the modeling domain by using an error estimation strategy. The finite element analysis software then generates a completely new grid based on the error estimate information. While the software takes into account the local errors of the entire model, it uses smaller cells in areas where local errors are more significant. The advantage of this approach is that all mesh refinement is done by software; the downside is that the user cannot control the mesh. As a result, irrelevant areas that can accept large local errors may undergo excessive mesh refinement.
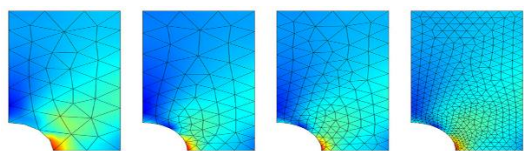

Fig. 20: Global adaptive mesh refinement changes cell size in a non-uniform manner

3.9 Local adaptive meshing

The difference between local adaptive mesh refinement and global adaptive mesh refinement is that the former only performs error estimation on a partial subset of the entire model space according to specific metrics. For example, the stress at the whole boundaries can be more accurately resolved by refining the mesh. This meshing strategy is still to re-mesh the entire model to reduce the error of a certain area. The local adaptive mesh refinement method is superior to the global adaptive method if there is a logical and applicable local metric to determine the mesh that can be refined.
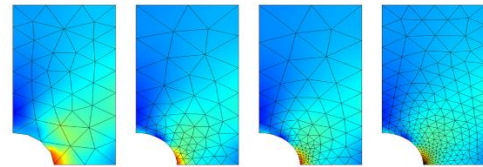

Fig. 21: Local adaptive mesh refinement corresponding to different stresses at a certain point

3.10 Manually adjust the grid

For simulation analysts, the most frequent method of manual operation is to manually create a series of different finite element meshes based on the physics of a particular problem, and intuitively determine which locations require more elaborate elements. Triangle and quadrilateral elements can be used in combination when dealing with 2D models. In the case of a three-dimensional model, a combination of tetrahedrons, hexahedrons (also known as bricks), triangular prisms, and pyramid elements can be used. Although triangle and tetrahedral elements can be used to mesh any geometry, it is more helpful to use quadrilateral, hexahedron, prism, and pyramid elements where it can be determined that the solution's solution is gradually changing in one or more directions. By stretching or shrinking the unit in certain directions, the mesh can be adjusted to accommodate field variations.
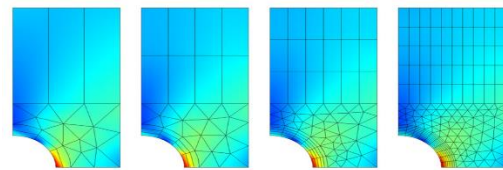

Fig. 22: Manually created mesh in a perforated plate model with different sized triangles and elongated quadrilateral elements

The manual meshing method has the highest requirements for analysts, and requires a deep understanding of the finite element method and the solved physics field and has a wealth of practical experience. However, as long as it is used properly, it can save a lot of time and resources.
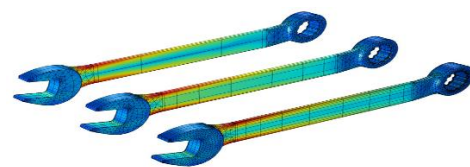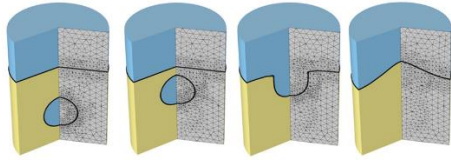

Fig. 23: Manual mesh refinement of the wrench using different unit types

3.11 Time domain and frequency domain meshing

In addition to the various meshing techniques mentioned above, there are other factors that need to be considered when meshing problems with time-varying loads. Models with a nonlinear material response or arbitrary time-varying excitations need to be solved in the time domain. On the other hand, if the applied excitation is a single frequency, or within a known frequency range, and the material properties are linear, then the preferred approach is to model in the frequency domain. For each case, there is a corresponding grid refinement strategy.
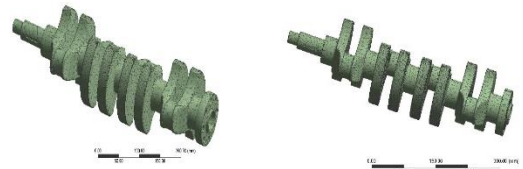
3.12 Time adaptive meshing

The time adaptive mesh refinement method re-meshing the model at different time intervals and using the error estimate of the solution for each time interval as a metric for re-dividing the model mesh. This method is very useful when the relevant area needs to maintain a high grid resolution over time.
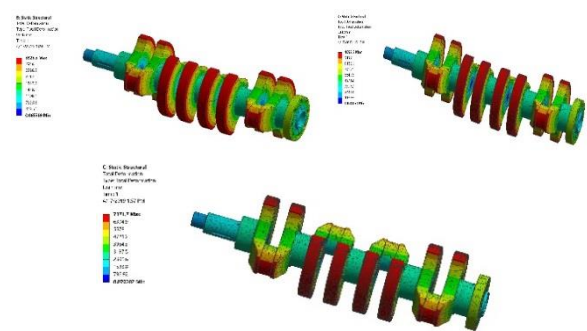
Fig. 24: Time-adaptive mesh refinement of the ascending bubble model solved by the two-phase flow model. The finite element mesh near the phase boundary is finer

3.13 Wavelength adaptive meshing

When modeling in the frequency domain, we predetermined the range of excitation frequencies and material properties. Therefore, it is possible to predict the wavelengths in all modeling domains. The size of the cell must be much smaller than the wavelength, and its polynomial basis function can resolve the wave.

Fig. 25: Microwave waveguide with a dielectric load (cross-sectional view)

Wavelength adaptive mesh refinement changes cell size based on the frequency and material properties.

3.14 The challenge and development trend of finite element mesh refinement

The key point to remember about all of the above methods is that for a well-posed problem, no matter which method is used, it should converge to the same solution. The only difference between these different methods is that they converge at different speeds. However, this is an obvious practical difference. Depending on the problem, a technique can converge much faster than other methods; but no refinement strategy can be applied to all situations. Each problem has specific technical challenges in terms of meshing, which continues to pose challenges for analysts.

Some of the changes that are taking place will help to address these difficulties. In the past few years, it has become easier to get low-cost cloud computing resources, which is one of the most important developments, thus creating conditions for running multiple different simulation instances in parallel. Because of this, analysts can study a variety of different models and meshing schemes in less time, so that all uncertainties can be quickly eliminated.

The grid generation algorithm itself is constantly improving, and the advantages brought by multi-core computing are increasingly utilized. In addition, the solver has become more and more efficient, enabling large models to be solved on clustered computers. All of these changes will result in more accurate results in less time, speeding up the analysis and design process.

4. RESULTS
4.1 Free Mesh of model
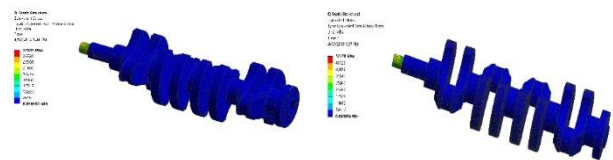

4.2 Different mesh refinement

4.3 Total Deformation result compare during meshing

Fig. 26: Total Deformation result compares during meshing part

4.4 Equivalent stress evaluation result compare during meshing

Fig. 27: Vonmises Stress Result compare during the meshing part.

Table 1: Comparison of the Result Vonmises Stress Results and Vonmises Error Percentage
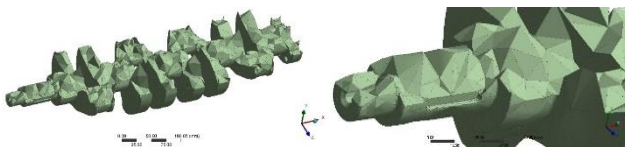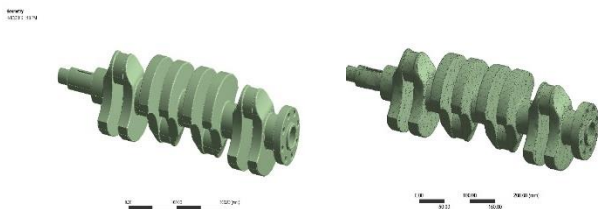
Element Size	Vonmises Stress (MPa)	Error Percentage
0.5 mm	52570	0
0.75	50045	0
1 mm	45257	7
1.5	40753	8.5
2.5 mm	39525	15.5
3	31256	18.3
5 mm	35573	24.5
10 mm	32535	35

5. CONCLUSION

The optimization mesh model exhibits a good level of results obtained using the classical finite element model. Many of the problems in Finite Element Modeling (FE) are tracked and the quality is the poor grid. In this article, powerful algorithms are used to optimize these bad meshes. Design and implementation. Inferior nets have a big impact analysis. For example, if the function approximates on a low mass grid, then a lot introducing adverse effects of interpolation and gradient interpolation errors Precision modeling. The stiffness matrix collected from the inferior mesh can because it is closely related to the main grid, the preparation is very bad. Two discusses these effects of poor mesh quality in detail. Use anisotropic elements and methods to calculate the ideal size of elements, shapes, and elements also

enter the direction of each location in the domain. As part of this paper, a very efficient mesh optimization method was developed. And implemented. Powerful algorithms are used as part of this method details the results obtained from grid optimization. The content based on this methodology will be introduced in Chapter 4. Obviously, this is a logarithmic obstacle the objective function is a very powerful tool and should be used in all situations Grid optimization is due to its effectiveness and efficiency. In the present work, the mobile mesh model shows an excellent level of agreement with the results obtained by the classical finite element model, reaching levels of relative differences in average value over 2 spans in the contact force of the order 7% and in the displacement average differences less than 2 mm, so it can be said that the accuracy of the mobile mesh model is adequate for simulation purposes; (iii) and finally, the main advantage of the mobile mesh model becomes clear when comparing the required calculation times. It is observed that the proposed methodology presents simulation time reductions around 4 times the time used by a conventional finite element model. This aspect is decisive to achieve a dynamic simulation with the finite element method, currently of wide application in research, with a substantial reduction in computational cost. This requires the development of methodologies such as the one proposed here, which, due to the high relation between precision and calculation time, opens the door to the use of optimization techniques, sensitivity analysis, the effects of element size on accuracy of finite element models and simulation results were thoroughly investigated through static analysis, impact analysis, and modal analysis. It was found that for static analysis that assumes steady loading and response conditions, each side of a plate model should be discretized into 10 divisions in order to obtain satisfying results (with approximation error < 1% in our example) consuming fewer computer resources.

6. REFERENCES

[1] Aman Dutt, "Effect of Mesh Size on Finite Element Analysis of Beam" SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 2 Issue 12 – December 2015

[2] M. Such, JR Jiménez-Octavio, A. Butcher, C. Sanchez-Rebollo. Simulation of mobile loads on structures through a mobile finite element meshing. Rev. Int. Metod. Numer Calc. Dise. Online, 2014.

[3] Structural Analysis of Tram Car Using Steel and Aluminum Honeycomb by Benyam Adane,2014

[4] P.G. Tathe, Prof. D.S. Bajaj and Swapnil S. Kulkarni, "Failure Analysis And Optimization In Yoke Assembly Subjected By Torsion And Shear", International Journalof Advanced Engineering Research and Studies E. July-Sept, 2014

[5] Sean Delfel, "Introduction to Mesh Generation with ANSYS Workbench" MECH479 coanda research and development corporate, 2013.

[6] Li Z.-P., Kindig M.W., Subit D., and Kent R.W., "Influence of mesh density, cortical thickness and material properties on human rib fracture prediction", Medical Engineering & Physics, 32(9), 2010, 998-1008.

[7] LS-DYNA Theoretical Manual, Livermore Software Technology Corporation, May 2007.

[8] Roth S. and Oudry J., "Influence of mesh density on a finite element model under dynamic loading", Proceedings of 3rd European Hyperworks Technology Conference, November 2nd-4th, 2009, Ludwigsburg, Germany.

[9] Simpson, R B 'Automatic local refinement for irregular rectangular meshes' Int. J. Numer. Meth. Eng. Vol 14 (1979) pp 1665-1678

[10] Rivara, M C 'Algorithms for refining triangular grids suitable for adaptive and multi-grid techniques' Int. J. Numer. Meth. Eng. Vol 20 (1984) pp 745-756