Open access Journal International Journal of Emerging Trends in Science and Technology

A Computational Technique for Efficient and Accurate Stress Prediction in a Complex Wind Turbine Model

Authors

Stanlee. P[1], Dr. Velamurali[2]

[1]2012287017, ME CAD (PT), College of Engineering, Guindy, Anna University, Chennai – 600025

Email: *stanleekit@yahoo.co.in*

[2]Head and Professor, Engineering Design Division, Department of Mechanical Engineering, College of Engineering, Guindy, Anna University, Chennai – 600025

Abstract

Stress concentrations are a concern in engineering because of their implications regarding structural integrity. Efficiently and accurately computing the associated stresses is therefore important. Unfortunately it is not always an easy task. This task becomes more challenging when the complex configurations encountered in practice require three-dimensional analysis. Submodeling with finite element analysis is one means available to meet this challenge. With submodeling, a subregion is broken out from the original global region and analyzed separately. This subregion requires boundary conditions taken from the finite element analysis of the global region. Accurately estimating the errors associated with these boundary conditions is critical to controlling them and thereby enabling efficient submodeling.

Since it requires more computational time and leads to convergence issues to analyze a complex model which will be having millions of Degrees of Freedom. It is a smart choice to check the regions of interest where the stress dominance takes place and slice the portion from the global model away from the Stress concentrated regions and analyze the same using Submodeling methodology.

Keywords: *FEM, Sub modeling.*

FEM?

- FEM is a numerical technique for finding approximate solutions to boundary value problems for differential equations.
- FEM is best understood from its practical application, known as finite element analysis (FEA). FEA is a computational tool to perform analysis of any structures based on the physics of the component through applied Boundary and Loading conditions. It includes the use of mash generation techniques for dividing a complex problem into finite number of parts called as elements.
- FEA is based on the methodology of solving the problem through piecewise polynomial differential equations.

Steps in FEM

- Discretize and Select the Element Types:
  Modeling a body by dividing it into an equivalent system of smaller bodies or units (finite elements) interconnected at points common to or more elements (nodal points or nodes) and/or boundary lines and/or surfaces (discretization)
  Element type: line element, plane elements, 3D element, axisymmetric

- Select a Displacement Function: linear, quadratic, cubic polynomials
- Define the strain/Displacement and Stress/Strain relationships

  $\varepsilon x = du/dx$, $\sigma x = E\,\varepsilon x$
- Derive the Element Stiffness Matrix and Equations through direct method, energy method, weighted residual method

  $\{f\}=[k]\{d\}$
- Assemble the Element Equations to obtain the Global Equations and introduce boundary conditions. Element stiffness matrix are added together to obtain global stiffness matrix (globalization) $\{F\}=[K]\{d\}$
- Solve for the unknown Degrees of Freedom (DoF) using elimination method, iterative method etc.
- Solve for the Element Strains and Stresses Interprete the results largest displacement, largest stress etc.

Why FEM?

- Finite: Any continuous object has infinite Degrees Of Freedom (DOF) and it's just not possible to solve the problem in this format. Finite Element Method reduces DOF from infinite to Finite with the help of discretization technique known as MESHING.
- Element: All the calculations are made at limited number of points known as Nodes. Entity joining nodes and forming a specific shape such as quadrilateral or triangular etc. is known as Element. To get value of variable (say displacement) anywhere in between the calculation points, interpolation function (as per the shape of element) is used.
- Method: There are 3 methods to solve any engineering problem. FEA belongs to Numerical method category.

Advantages and Disadvantages of FEM

Advantages:

- Material properties in adjacent elements do not have to the same. This allows application to composite materials.
- Irregularly shaped boundaries can be approximated using elements with straight sides or matched exactly using elements with curved boundaries.
- The size of elements can be varied. This allows element grid to be expanded or refined as the need arises.
- Mixed Boundary conditions can be easily handled.
- Automatic mesh or grid techniques to assist model building (Batch meshing).
- FEM comes under CAE which is an integral part of CAD/CAM and CFD.

Disadvantages:

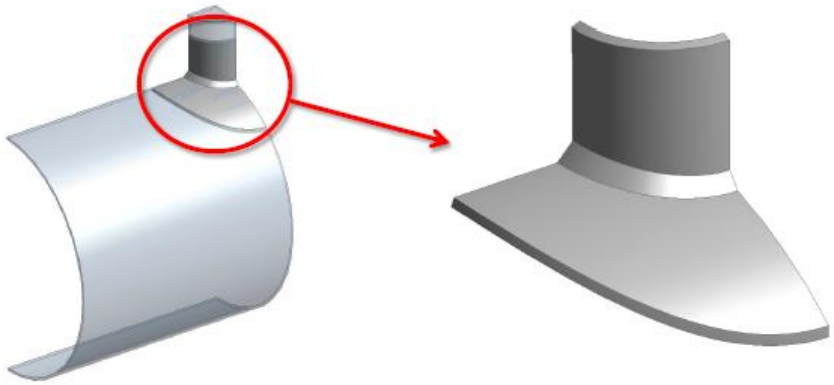
- Need for computer programs and facilities.
- Still an approximated technique.
- Gives solution only at nodal points.

Submodeling

- In simple words, Sub modeling is a modeling approach that allows solving a small part of a complex model (global model), with more refined meshes and results.
- Sub modeling is based on St. Venant's Principle.

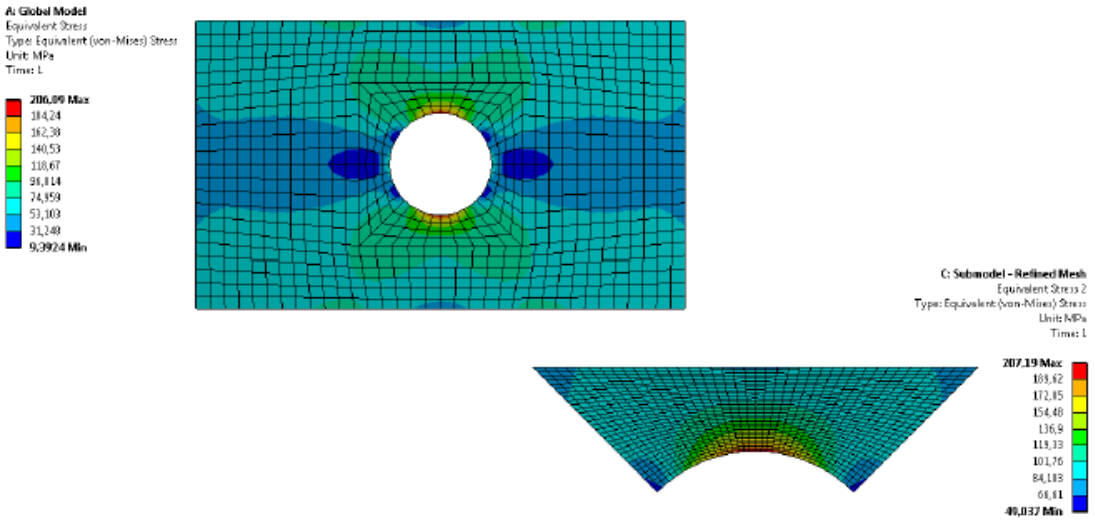
  -Far away from loading regions and constraint boundaries, so that the mechanical behavior is not affected.

  -If sub model boundaries respect this principle, good results are obtained.
- This technique allows an initial evaluation of a coarser global model, followed by a detailed analysis based on the regions of interest.



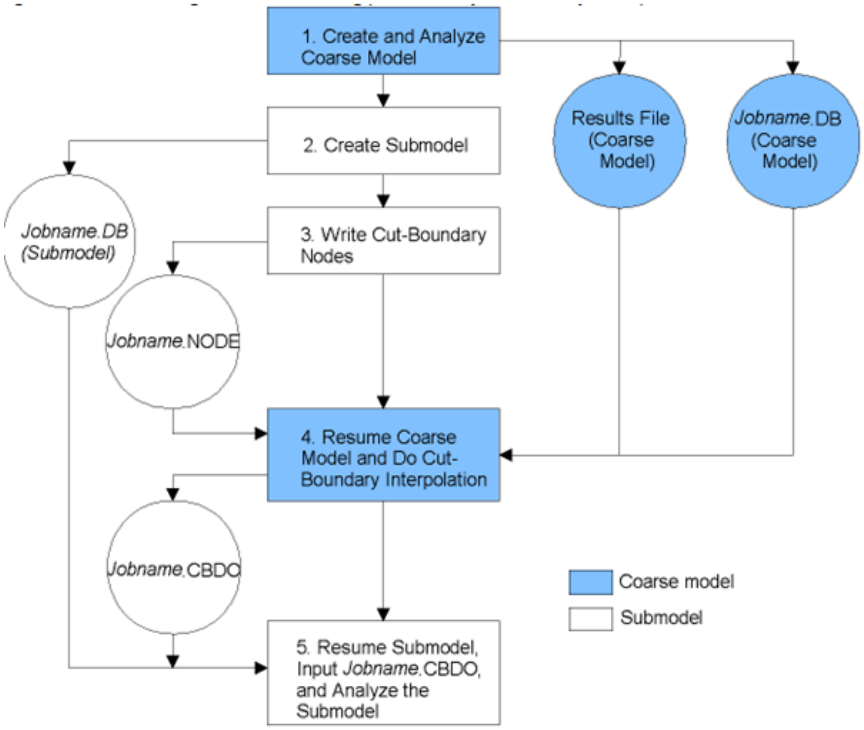
Why Submodeling required?

- Since it requires large computational time to solve a Global FE model which may lead to convergence difficulties, it is a smart choice to go for Sub modeling methodology.
- Suppose if there is a requirement to do Cost out in the existing global model, it is difficult to re-analyse and optimize the entire global model.
- So very few sub models are selected based on the fatigue life obtained whose Safety Reserve Factor (SRF) is pretty higher than the allowable limit. This helps to do the optimization (e.g., reducing the plate thickness) easier.
- Then the global model is updated with the necessary changes and rerun again to obtain the global results.
- These results are then interpolated with the selected sub models to arrive the decision of this optimized cost out.
- In general, the sub model geometry allows one to:
  -Use a more refined mesh at the critical regions to provide better results.
  -Include small geometric details (like welds) that are not represented in the global model.



Submodeling procedure?

- Create and solve the Global model.
- Create the sub models.
- Apply cut boundary interpolation.
- Solve the sub model.
- Verify the results and cut boundary position (Through Displacement comparison between Global vs. Sub models).

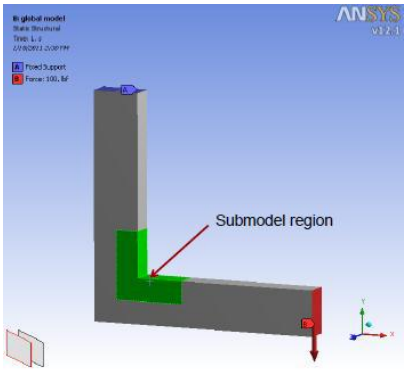


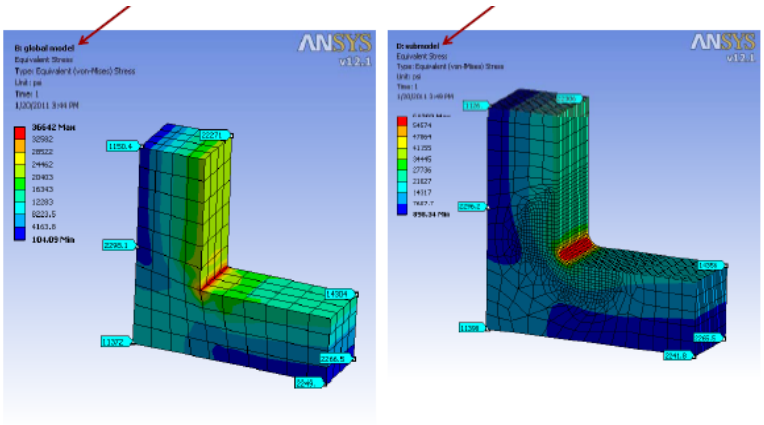
Advantages of Sub modelling

- Reduce computational time.
- Reduce convergence issues.
- Analyze the actual behavior of bolted joints, contacts and welds.
- Easy to optimize sub models.
- Cost outs can be made.
- Easy user interaction and handling.

Conclusion

Thus using Sub modeling methodology, the regions of interest can be easily evaluated to understand the behavior of bolts, contacts and welds and to obtain the exact solution.





References

1. Practical FEA (Nitin Gokulae, Sanjay, Sanjeev, Anand)
2. ANSYS 13.0 (Help).
3. https://engineering.purdue.edu
4. http://www.kth.se