International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

FEA Best Practices Approach

Dr. S. Rajadurai[1], M. Guru Prasad[2], R.Kavin[3], M.Sundaravadivelu[4]

[1]*Head-R&D, Sharda Motors Industries Limited, Mahindra World City, New Chennai*
[2,3]*Senior Engineer,* [4]*P.D. Engineer, CAE - R&D, Sharda Motors Industries Limited, Mahindra World City, New Chennai*

***Abstract*— The growing computer capabilities, human skills, aggressive cost effective product development has made analyst to rethink a lot of parameters before proceeding into the simulations. The focus of this paper is to explain the principles of understanding the concepts, meshing guidelines, defining elements types and order for successful, time efficient solutions, reducing learning curve and avoid reinvention of cycle. By performing these check list, most of the queries are cleared in the initial phase by 30% rather than in latter stages which outlay time and rework if necessary.**

***Keywords*— Element types, Finite element analysis, FEA best practice,**

I. INTRODUCTION

With the advent of computers, FEM has become a powerful tool for solving practical industrial problems. FEM is applicable to a wide variety of engineering problems. Many general purpose FEM software packages are available commercially.

The Finite Element Method is a numerical technique for solving complex engineering problems by breaking them into smaller manageable problems.

The FE model is a way getting a numerical solution to specific problem. A FE analysis does not produce a formula as a solution, nor does it solve a class of problems. Also, the solution is approximate unless the problem is so simple that is if convenient exact formula is already available.

To analyze a structure by the FE method, the engineering problem is re-defined as a numerical model where the structure is broken down into finite number of regions or parts, called elements. The elements are connected to each other at grid points, also called nodes. The assembly of elements interconnected at nodes, is called the finite element mesh.

For example, a uniformly loaded arch could be idealized as four straight beams with three "lumped" nodal loads as shown in the figure 1 below.

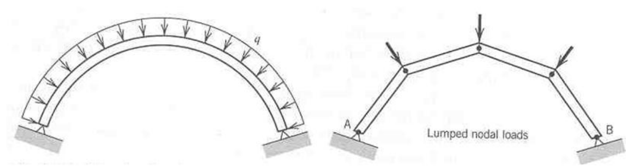

Figure 1. Uniformly loaded arch

The attributes of the structural system (materials, physical properties, loads, constraints, etc.) are added to the finite element mesh to represent the engineering problem as closely to reality as possible.

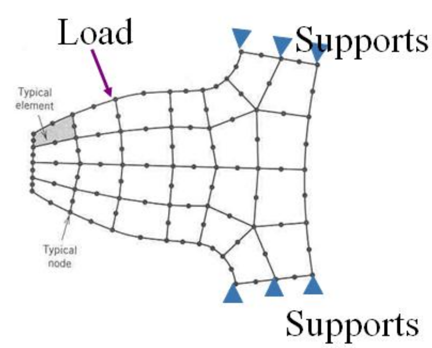

Figure 2. Attributes of structural system

Elements are assigned with thickness and material properties such as Modulus, Poisson's ratio etc.

The engineering equation solved by linear static FEM is Hooke's Law F = KX, where F is the applied force, K is the structural stiffness and X is the structural displacement.

In order to apply this equation to a structure, it discretizes using nodes and elements. The structural displacements and the applied forces are defined in terms of degrees of freedom at each node.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

In three dimensional space, there are six degrees of freedom at each node - three translations and three rotations. These are in the directions of predetermined coordinate axes.

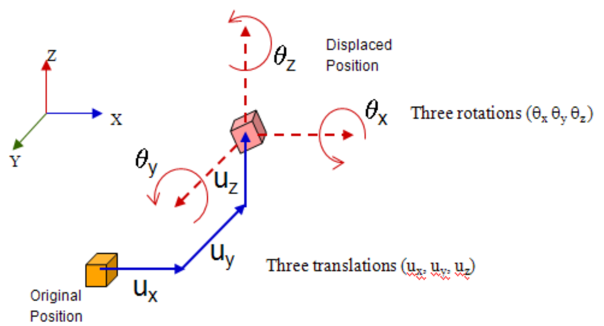

Figure 3. Nodal displacement vector (u_x, u_y, u_z, θ_x, θ_y, θ_z)

II. INITIAL PHASE

A. Prevention

FEA cannot be used carelessly; the best way to use it is in a thorough manner.

- Thorough planning.
- Careful modeling.
- Accurate loading and modeling of supports.
- Thorough verification of results.

B. Depending parameters

It is critical to understand the following parameters which will shorten the computational time required to provide solution.

- Understanding the physics of the problem and behavior of the elements.
- Selecting the correct element, the number of elements, and their distribution.
- Critically evaluating the results and making modification in the conceptual model to improve their accuracy.
- Understanding the effects of the simplifications and assumptions used.

C. Verification & Validation (V&V)

It is inherent to validate the methodology for a specific usage of loads over some uncertainties. It is necessary to verify the calculation to get overall uncertainty estimate and compare results with the test in the interested domain. The quantifying parameters need to be verified before developing a plan

- Intrinsic variability of parameters.
- Lack of knowledge of the parameters.
- Model form

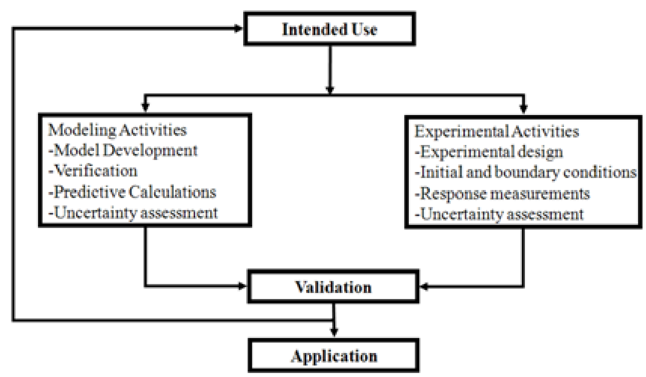

Figure 4. Flow chart for V&V

III. BEST PRACTICES APPROACH

List of parameters to be check listed at each stage are mentioned below

- Plan your analysis.
- Materials.
- Model geometry.
  - Element choice.
  - Meshing.
  - Simplifications.
- Supports and loads.
- Model calibration.
- Verification.

A. Plan your analysis

Just jumping in will cause you to go down a lot of dead ends. Sit down and plan your approach before you even start up your FEA program.

- What are the design objectives?
- What do you need to know?
- Why are you doing FEA?
- What are the design criteria?
- What engineering criteria will be used to evaluate the design?
- What are you trying to find out?
- How much of the structure needs to be modeled?
- What are the boundary conditions and loads?
- Do you need to know stresses, displacements, frequency, buckling or temperature?

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

- Get ballpark figures through hand-calculations or test data, so you have an idea of how the structure will behave and what numbers are reasonable.

IV. Analysis Decisions

Decision making at critical junctures are gained through training, expert's opinion and experience. Decision at these junctures will implicate the effects of the result's computational time and accuracy.

- Analysis type.
- How to idealize material properties.
- Geometry details/simplifications.
- Element type/options.
- What are the supports or constraints?
- What are the loads?

A. Type of analysis

Static analysis assumes that inertial and damping effects are negligible. You can use time-dependency of loads as a way to choose between static and dynamic analysis. If the loading is constant over a relatively long period of time, choose a static analysis. In general, if the excitation frequency is less than 1/3 of the structure's lowest natural frequency, a static analysis may be acceptable. Cyclic loads can be modeled by a harmonic analysis rather than full transient.

Nonlinear structural behavior is a changing structural stiffness which depends on the following considerations, that needs to be checked at each junctures mentioned above in analysis decisions.

- Types of nonlinearities.
    - Geometric (e.g., large deflections)
    - Material (e.g., plasticity, hyper elasticity)
    - Changing Status (e.g., contact)

Geometric Nonlinearities:

- Large deflections & rotations.
- Stress stiffening.
- Spinning structures

Material Nonlinearities:

- Plasticity
- Creep/Visco-elasticity.
    - Rate dependence
- Visco-plasticity.
    - Time dependent
- Hyper-elasticity.

Contact Nonlinearities:

- Bonded vs. nonlinear contact.
    - Welded/glued parts.
    - Gaps in model.
    - Will parts separate from each other?
    - Is de-lamination possible?
- Large vs. small sliding.
    - Determines type of element to use.
    - Determines type of contact
- Contact stiffness
    - Is the contact hard, or is there some softening?
    - Is contact pressure an important value?
- Does friction need to be modeled?
    - What value for the coefficient?
    - May need to run model with different values.

B. Materials

Material properties have to be decided, depending upon the boundary and loading condition of the problem. Being homogenous or undergoing phase change with respect to rate or time has to be determined before applying the material properties.

Material Information:

- For linear isotropic material, need modulus of elasticity, density and Poisson's ratio for a static analysis and inertial loads.
- For thermal analysis, thermal conductivity, specific heat, film coefficient is needed. It also requires coefficient of thermal expansion for thermal stress.
- Need test data for nonlinear materials.

Poisson's ratio:

Required when these controls expansion /contraction in direction perpendicular to load direction. For models constrained from expansion, the value of ν is very important. If using $\nu = 0.5$, need to use element with hyper elastic ability.

Multiple Materials:

- Model a boundary wherever material properties change. Make sure the appropriate material property is assigned to each part of the model.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

- Consider interaction between properties which affects contact stiffness.

C. Linear or Nonlinear

If no stress-strain data is given, the program will assume the analysis is linear, and will use Young's Modulus even if the part yields. This gives erroneous results when the loads cause the model to exceed yield.

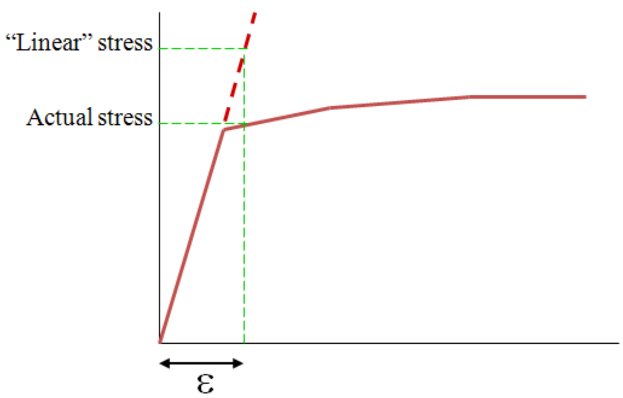

Figure 5. Stress-Strain curve

- Graph stress-strain curve to check inputs which requires entering Young's modulus.
- Enter stress-strain curve using true-stress and true-strain.
    - True strain = ln(1 + engineering strain)
    - True stress = eng. stress (1 + eng. strain)
- Make sure Young's modulus matches yield stress and strain

D. Units

Many general purpose FEA codes allow the user to enter a consistent unit set.

- Use mass = force/area to get proper mass units.

TABLE 1.
CONSISTENT UNITS

Mass unit	kg	kg	lbf-s^2/in
Length unit	m	mm	in
Time unit	s	s	s
Gravity const.	9.807	9807	386
Force unit	N	mN	lbf
Pressure/Modulus of Elasticity	Pa	kPa	psi
Density Unit	kg/m^3	kg/mm^3	lbf-s^2/in^4
Mod. Elasticity Steel	0.2E12	0.2E9	30E6
Mod. Elasticity Concrete	30E9	30000	4.5E6
Density of Steel	7860	7.86E-6	7.5e-4

- Set loads to zero and run. Check mass and center of mass. Turn on gravity and check reactions.
- If using small dimensions (e.g. microns, millimeters) use smaller base unit to reduce round-off problems.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

E. Singularities

- FEA uses the theory of elasticity: stress = force/area. If the area=0, then stress=infinite.
- Theory of FEA: as mesh is refined, the stresses approach the theoretical stress. For a singularity, you would try to converge on infinity.
- A stress singularity is a location in a finite element model where the stress value is unbounded (infinite).
- A point load, such as an applied force or moment.
- An isolated constraint point, where the reaction force behaves like a point load.

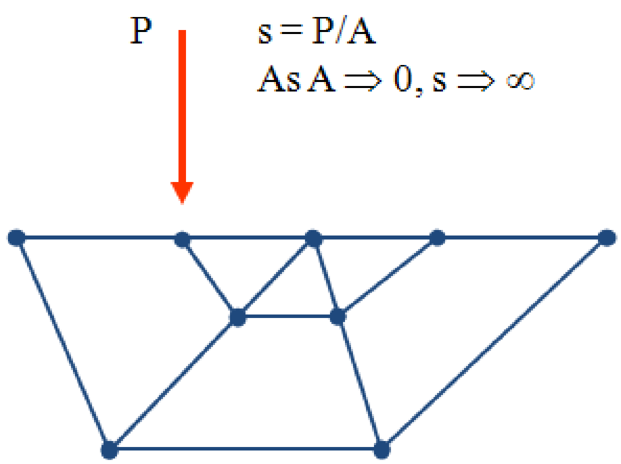

Figure 6. Singularities

- A sharp re-entrant corner (with zero fillet radiuses).
- Real structures do not contain stress singularities. They are a fiction created by the simplifying assumptions of the model.
- Point loads are best used for line elements.

F. Geometry Check

- If importing model, do some checks of the dimensions – don't assume its right.
- Make sure the model is in the required units system.
- If the model was created in a system different from the material data and loads, you need to scale the model by the proper conversion factor.
- Check for duplicate surfaces and delete it. Remove unwanted lines, fillets, holes, beads before start meshing the component.

G. Choice of Elements

Line Elements:

- Beam elements have bending and axial strength. They are used to model bolts, tubular members, Cross sections, angle irons, etc.
- Spar or Link elements have axial strength. They are used to model springs, bolts, preloaded bolts, and truss members.
- Spring or Combination elements also have axial strength, but instead of specifying a cross-section and material data, spring stiffness is entered. They are used to model springs, bolts, or long slender parts or to replace complex parts by an equivalent stiffness.

Shell Elements:

- Use shell elements, when maximum unsupported dimension of the structure is at least 10 times the thickness. Shell elements can be used to model thin panels or tubular structures
- "Thick" shell elements include transverse shear, "thin" shell elements ignore this. Shell elements can be 2D or 3D; 2D shells are drawn as a line, 3D as an area

Solid –Shell Elements:

- 3D Solid brick (or prism) element without bending locking. Nodes have same DOFs as 3D elements–can connect thin and thick structures without constraint equations or MPCs.
- Can model varying thickness bodies without using multiple real constants.

Solid Elements

- Used for structures this, because of geometry, materials, loading, or detail of required results, cannot be modeled with simpler elements.
- Also used when the model geometry is transferred from a 3-D CAD system, and a large amount of time and effort is required to convert it to a 2-D or shell form.

H. Element Order

- Element order refers to the polynomial order of the element's shape functions. It is a mathematical function that gives the "shape" of the results within the element. Since FEA solves for DOF values only at nodes, we need the shape function to map the nodal DOF values to points within the element.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

- The shape function represents assumed behavior for a given element. How well each assumed element shape function matches the true behavior directly affects the accuracy of the solution.

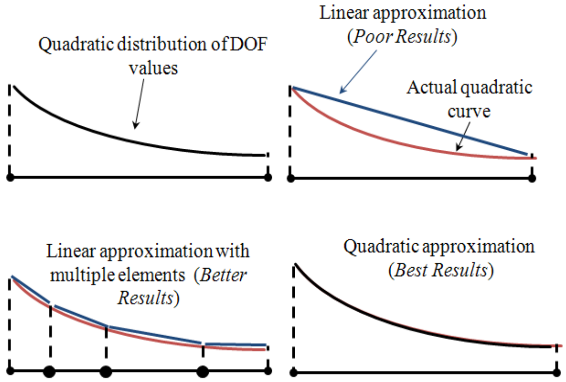

Figure 7. Element Order Comparison

TABLE 2.
ELEMENT ORDER COMPARISON

Linear Elements	Quadratic Elements
Can support only a linear variation of displacement and therefore (mostly) only a constant state of stress within a single element.	Can support a quadratic variation of displacement and therefore a linear variation of stress within a single element. Can represent curved edges and surfaces more accurately than linear elements.
Highly sensitive to element distortion.	Not as sensitive to element distortion
Acceptable, if you are only interested in nominal stress results.	Recommended, if you are interested in highly accurate stresses.
Need to use a large number of elements to resolve high stress gradients	Give better results than linear elements, in many cases with fewer numbers of elements and total DOF.

Selecting Element Order:

- When you choose an element type, you are implicitly choosing and accepting the element shape function assumed for that element type. Therefore, check the shape function information before you choose an element type.
- Typically, a linear element has only corner nodes, whereas a quadratic element also has mid-side nodes.
- For shell models, the difference between linear and quadratic elements is not as dramatic as for solid models. Linear shells are therefore usually preferred.
- Besides linear and quadratic elements, a third kind is available, known as p-elements. P-elements can support anywhere from a quadratic to an 8th-order variation of displacement within a single element and include automatic solution convergence controls.

I. *Meshing*

Mesh Considerations: For simple comparison, coarse mesh is ok. But for accurate stresses, finer mesh is needed. It is recommended to have finer mesh for fatigue. Invest elements at locations of interest to reduce computational time. It is necessary to avoid connecting quadratic and linear elements.

Element connectivity: Make sure there are no 'cracks or free edges' in the model. Can also use a shrink plot to check connectivity. Add density and perform Free – Free Modal analysis i.e. without boundary condition and check for first 6 rigid body modes and deformations in the model. Apply a dummy load and solve, then view the displacements.

Mesh Convergence: In FEA Theory: as mesh gets finer, it gets closer to real answer. Mesh once, solve, mesh finer, solve again; if results change within a certain percentage, the mesh is converged, otherwise, and repeat. Perform a mesh convergence on a problem with a known answer to get a better understanding. Displacement results converge faster than stress results.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

V. Quality Checking

A. Mesh Distortion

Elements distorted from their basic shape can be less accurate. Higher distortion indicates greater the error. These limits are subjective, and a 'bad' element might not give erroneous results, and a 'good' element might not give accurate results.

- Four types of distortion:
  - Aspect ratio (elongation).
  - Angular distortion (skew and taper).
  - Volumetric distortion.
  - Mid node position distortion (higher order elements).

B. Connections

- Meshing your entire structure is not always feasible– it's nice to model some parts with simpler elements.
- Can embed shells in solid elements to connect them, but be careful of doubling the stiffness– better to use MPC connection or Solid-Shell element.
- Use constraint equation or MPC to connect shell to solid, beam to solid or beam to shell.

C. Element Normals

Shells have a 'bottom', 'middle' and 'top'. Bending through thickness means stress on 'top' and 'bottom' will differ. In order to make sense for stress results, element normals should be orientated "top". Positive pressure is oriented opposite to the element normal (i.e., into the element).

VI. Boundary Conditions

- Do the boundary conditions adequately reflect 'real life'? Because there are no single-point or line supports; these are approximations we use. Real life has some small area.
- Be wary of singularities. If deformation of support isn't negligible, model support with coarse elements and use bonded contact to tie support to model

VII. Loading Conditions

- Do the loads match real life? Because there are no point loads in real life, just really small areas with pressures on them. So, investigate all possible combinations of loads. Think about the load path through the structure.
- Consider range of load values for parametric analysis of different values.
- Consider long-term analyzed for creep and fatigue vs. short-term loads analyzed for yielding.

VIII. Inspection

- Check for reactions and buckling.
- Watch your errors and warnings.
- Large differences in stiffness.

IX. Results

A. Evaluating Results

- Stress criteria differ for each type of analysis in order to validate. Is stress greater than yield? Don't assume the results are correct. Questions need to be asked by the analyst whether displacement, stress or deformations makes sense.
- Compare to tests or theory, when possible. Define factor of safety depending upon the model and solution.
- Use linearization, if needed and plot un-averaged stresses. Check the whole model– don't focus so much on one spot, you miss a problem elsewhere.
- Check reactions against applied loads, contact pairs for penetration and element error to assess mesh.

B. Results Verification

- Use deformed animation to check loads and look for cracks in model.
- Combined load behavior is sometimes difficult to predict– consider separating each load into its own load case to check.

C. Sensitivity Analysis

Process of discovering the effects of model input parameters on response. It can provide insight into model characteristics and also an assist in design of experiments.. A probabilistic analysis using your model and statistical data of input parameters to see how much variation there is in output. It should be subjected to same scrutiny as all V&V.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

- Generally requires several analyses– very time consuming.
- Positive sensitivity indicates that increasing the value of the uncertainty variable increases the value of the result parameter.

D. *Validation*

- Making sure the FE model will be accurate for a specified range of loads.
- Use experimental data (different from calibration data).
- Engineering experience.
- Hand calculations.
- Don't just assume the model is correct.

E. *Lessons Learnt*

- Detail all decisions made and explain simplifications.
- Detail material data, loads, supports, and test data.
- Document as much results data as possible.
  - List reaction forces.
  - Stresses.
  - Displacements.
- Create documentation for common analysis problems.

X. CHECKLIST

- Gravity pushes downward and spinning objects move radically outward.
- Heated objects grow and no real object has 1,000,000 psi stress.
- Axis symmetric objects rarely have zero hoop stress.
- A bending load causes compressive stress on one side, tensile stress on the other.
- Can use error estimation.
- Plot un-averaged stress and compare to averaged stress to check mesh. Do mesh convergences study.

A. *Peer Review*

Having a fellow engineer review your analysis can help you catch problems in the model. Can be informal, one-on-one, or a formal review, with a team looking over the analysis. Either way, it's better to be embarrassed in front of your colleagues, than in front of your customer.

XI. FEA SINS

1. Forgetting to ratio the load(s) correctly when using symmetry.
2. Not using a consistent set of units, e.g. Ton/mm^3 density.
3. Incorrectly mixing units in a model, e.g., inputting plate thickness in mm and building model in meters.
4. Trying to constrain or load degrees of freedom that nodes don't have.
5. Quoting FE stresses at re-entrant corners and point loads.
6. Trying to analyze a 'flying structure', including a lack of understanding as to why a constraint is needed in a direction for which there is no out-of-balance force.
7. Not considering convergence issues or verification checks.
8. Getting the axis wrong in axis symmetric models.
9. Not applying a degree of common sense to the results: does it look sensible, are the "field stresses" correct, is the mass correct.
10. Relying wholly on graphics and not double-checking input data that cannot always be shown in graphical form (materials, beam properties).
11. Not archiving model files.
12. Carrying out analyses when you have little or no understanding of the theoretical underpinnings of the analysis type.
13. Carrying out analyses when you have little or no understanding of the engineering significance of the results produced.
14. Underestimating the resources and timescale required for jobs.
15. Deliberately selecting views on fringe plots that avoid hot-spots caused by approximations in the model, but have no relevance to the areas of interest.
16. Actually 'painting out' hotspots with a paint program where you can't avoid the view.
17. Not proof-reading your reports properly because of time pressures.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 2, Issue 3, March 2014)

18. Not listing the assumptions made in the analysis.
19. Not making it clear where your responsibilities lie and what you are accepting liability for. E.G., not accepting design responsibility for products simply being analyzed; not accepting responsibility for ensuring that what is built was what you analyzed.
20. Problems in ascertaining the boundary conditions at the interface between structures or components being supplied by different contractors.
21. Overstating your capabilities and perhaps the software's to potential clients.
22. Quoting results to an accuracy which is not warranted.
23. Accepting bad meshes because of timescales, sheer recklessness or ignorance.
24. Including too much detail and/or too many elements in a model, without question.
25. Using averaged or continuous tone fringe plots when assessing results.
26. Thoughtlessly solving the idealized problem and not the real one, including not considering the scatter inherent in materials properties, loads, geometry, etc.
27. Simplifying material constitutive laws, without consideration of the implications.
28. Making the display color of your finite elements black with a black background.

XII. CONCLUSION

To make an analyst realize that, you must try the guidelines first before making a final decision on analysis. This tells you about the tricks, trade, pitfalls and checks of the finite element tools. Therefore, it is not an easy task to choose the most suitable guidelines for your needs. But if you follow the above guidelines you may by a gainer.

Acknowledgement

The authors wish to thank Sharda Motor Industries Ltd - R&D Centre for offering and supporting the opportunity to document and present this paper.

REFERENCES

[1] D. Baguley & D. R. Hose, How to Model with Finite Elements, NAFEMS, 1997
[2] K.A. Honkala, "Adequate Mesh Refinement for Accurate Stresses", NAFEMS Benchmark, p. 4-9, January 2000
[3] D. Baguley & D. R. Hose, "How to Model & Interpret Results Part 3: Validation & Interpretation Considerations", NAFEMS Benchmark, p. 4-8, January 2001
[4] G. Roth, "Pitfalls in Determining Analysis Accuracy", ANSYS Solutions, Vol. 2, No. 3, p. 30-31, Summer 2000
[5] Nithin.S.Gokhale, Sanjay.S.Deshpande, Sanjeev.V.Bedekar &Anand.N.Tite, Practical Finite Element Analysis, p.27-29,p. 63.67 p. 161-168, 2008

Abbreviations

FEM – Finite Element Method
FE – Finite Element
P – Force
A – Area
D – Dimension
CAD – Computer Aided Design
kg – Kilogram
mm – Millimeter
m – Meter
N – Newton
in – Inches
lbf – Pound force
psi – Pound per Square Inch
DOF – Degrees of Freedom
ν – Poisson's ratio