BOLTED OR WELDED ASSEMBLY SIMULATIONS – OVERCOMING SERIOUS OBSTACLES TO ADVANCED FEA VALUE

presented at NAFEMS 2003 World Congress, Orlando, Florida

reprinted 2015 by the author &
former owner of Integra Engineering, Inc.

Keith J. Orgeron, P.E.

now of Orgeron Engineering, PLLC
Spring, TX 77389 · 731-397-8625
keith.orgeron@OEngineer.com

Abstract –

Given that finite element analysis (FEA) value is relatively proportional to the complexity of the valid solution obtained, then multi-body assembly simulations represent great value. Such models (Figure 1) inherently involve component contact interaction and typically other nonlinear geometric or material and dynamic event parameters.

The advanced FEA technology required for assembly simulations is not only complex but also remains difficult to wield cost-effectively toward establishing validity and accuracy.

There are many and serious obstacles in achieving valid, converged solutions. Often, several models processed multiple times are required. It is no wonder that experience, foresight and adequate resources can be critical to successfully delivering advanced FEA value.

Most nonlinear solutions require achieving convergence for the simulation's event duration of interest. Unfortunately, there are literally hundreds of ways to inadvertently cause spurious assembly model behavior that can grind the solution to a halt with excessive numerical iterations.

This paper attempts to outline the most common and offensive spurious behaviors defeating solution convergence. By way of example it provides general guidance and details several novel techniques to help the analyst overcome these serious obstacles.

The case is made that validation of the FEA work, as opposed to the verification of the FEA software, provides the final basis for establishing the net FEA value.

The "7-Step FEA" process model is introduced as a system for performance and validation. These structured decision trees help to organize and thus simplify the complex FEA efforts at hand.

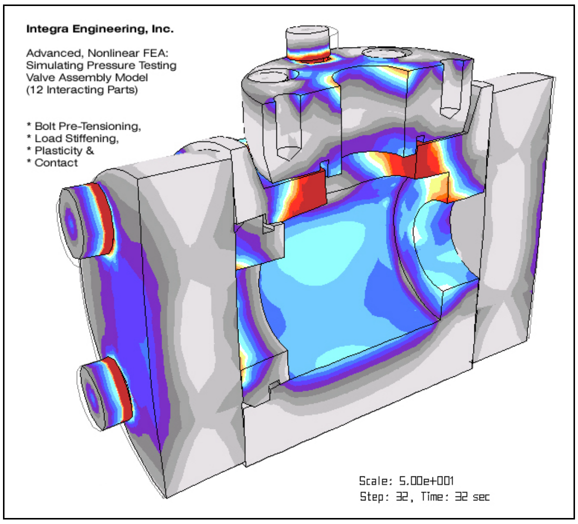


Figure 1 – Pressure-test of ball valve assembly, Mises stresses, 50x

INTRODUCTION

Most manufactured products are assemblies, subassemblies or parts that makeup an assembly. Assemblies inherently require joining methods such as bolting, welding or use of other fasteners or bonds. Bolted products such as the ball valve assembly represented by the half-model below (Figure 2) are probably the most common. (Note that the threaded holes are all blind.)

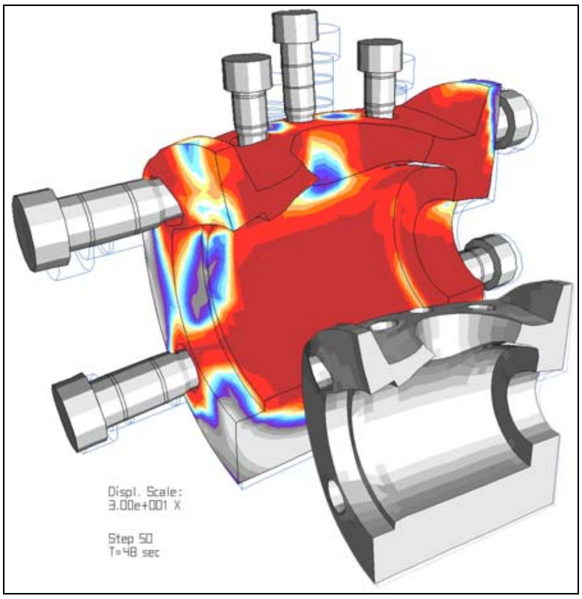


Figure 2 – Valve body & bolt stresses at limit load, 30x

Industrial and commercial equipment manufacturers comprise a large or the primary source of funding for engineering analysis activities. Most industrial and commercial products are candidates for advanced FEA simulations of at least mechanical or thermo-mechanical physical events in one or more of the following areas:

· Prototype virtual testing, · design liability containment, · failure analysis, · sensitivity studies, · design optimization, · model rating and sizing, · cost reduction, · component interactions, · code compliance, · preload adjustment, · survival conditions and · installation, bolt or weld residual effects.

FEA, as more than a 50 year old technology, still appears to favor and focus upon the traditional method of simulating a single part, regardless of the inherent and sometimes huge idealization errors. This is where advanced, nonlinear FEA excels, in achieving a much better representation of the physical event idealized, through more elaborate and often coupled, motion, material, loading and constraint set representations.

Some of the technology developed 30 years ago is today not in common use, such as nonlinear material and geometric solution capabilities. Then, besides computer hardware improvements, the last decade has seen the addition of FEA surface contact capabilities, CAD interfaces, auto-meshing, fast solvers and high-powered graphics to truly increase our efficiency and potential for accurate simulations.

So, the question remains, "Why do so few engineers/analysts perform multi-body simulations of even the most common mechanical events, such as bolt pre-loading for instance?" The advanced FEA technology required for assembly simulations is not only complex but remains difficult to wield such that validity and accuracy can be cost-effectively established to yield valuable results

Hopefully, the system and techniques conveyed in this paper will help analysts to overcome some of these serious obstacles to advanced FEA value.

The model in Figure 3 below successfully addressed the critical bolting preload issues that prevail in many assembly simulations. This validation model incorporates several of the techniques elaborated within this paper to cost-effectively refine key parameters prior to processing the larger model.

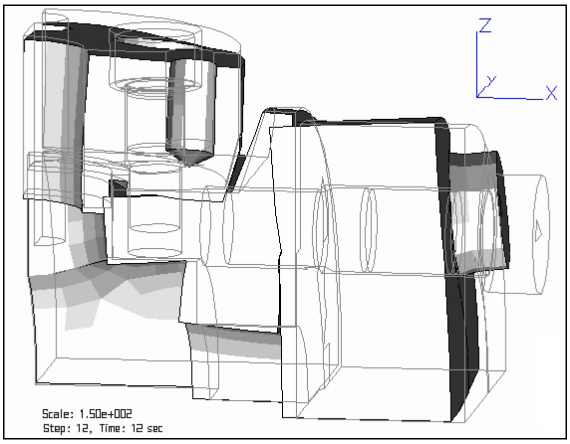


Figure 3 –Eighth-model of bolt-preloaded valve, 150x

This paper seeks to identify the key obstacles in achieving FEA value for assembly simulations, focusing on the issues of convergence of a nonlinear time-history direct integration solution and validation of the work.

Modeling and validation techniques will be presented that have been used successfully in achieving convergence and confirming FEA value, while cutting net analyst and processing time expenditures.

The scope of this paper is limited to focusing upon the modeling, processing, evaluation and presentation of nonlinear dynamic or pseudo-static time-history simulations employing direct implicit time integration. Unless noted, continuum element use is considered.

FEA VALUE = COMPLEXITY + ACCURACY

FEA value is often perceived as the timely generation of complex analysis results and conclusions. The presented results are also often believed to be "valid" and "accurate enough", but these definitions are highly subjective.

The very nature of undertaking a complex analysis implies the need for validation in order to assure both accuracy and relevance of the model's results to the idealizations sought for satisfying the project objectives. Simply stated, more complex models have more opportunities for error.

Yet, in spite of the pitfalls for these errors, advanced FEA offers more exact, sometimes far more exact representations of the idealized problem at hand.

Novice analysts often have initial difficulties achieving converged solutions, and would be more challenged to validate their many decisions. However, these same novices may also lack the engineering experience to justify the many simplifying assumptions inherent in the simpler linear static or linear dynamic analysis.

Though verifying linear loading and constraint sets is a more common engineering task, the question of accuracy must address the missing content of nonlinear behavior. Today, this issue can best be addressed by comparison with nonlinear FEA or by physical testing. Figure 4 above illustrates a very nonlinear behavior of the center bolt as it stretches.

ADVANCED NONLINEAR FEA FOR ASSEMBLIES

Assembly simulation virtually requires multi-body nonlinear contact and interaction for the most accurate behavior results. Component contact surfaces provide part or all of the loading and boundary conditions required to stabilize most or all components of these FEA models.

Many of the various classes of contact, such as finite sliding and friction, can only be analyzed using nonlinear FEA capabilities. These model features are some of the most critical, the newest and the least used.

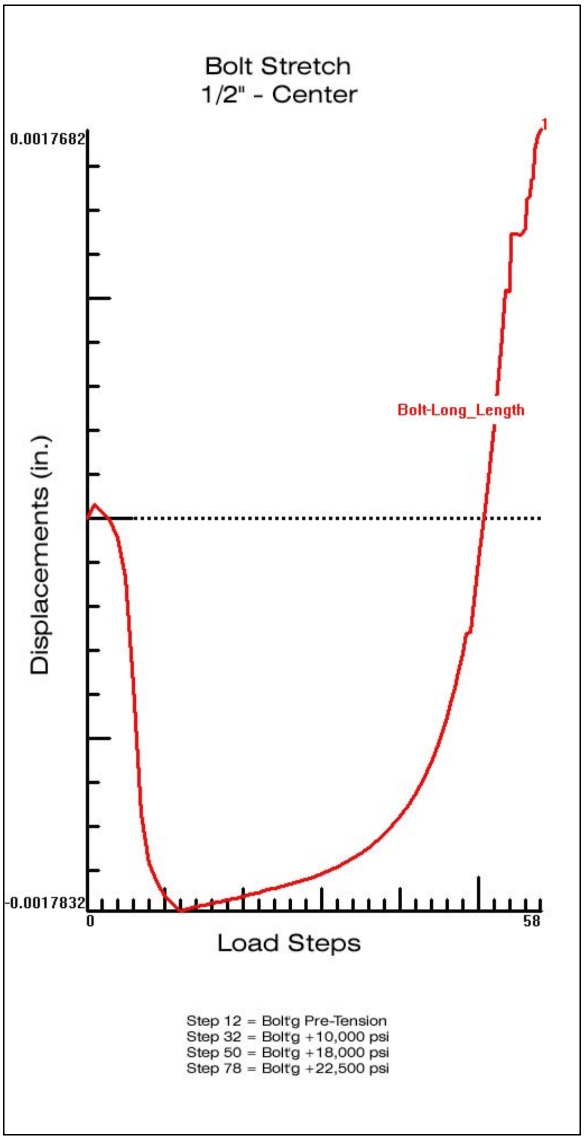


Figure 4 – Highly nonlinear behavior of center bolt

Nonlinear geometric solutions are also required for assembly simulations when they involve large displacements, rotations or strains. Figure 5 illustrates the valve bonnet's nonlinear reaction behavior as bolt elongation and plasticity cause load redistribution, clearly requiring a nonlinear geometric solution. Figure 6 presents some of the details of the geometry included in the bolted assembly model, partially detailing the blind holes.

Assembly simulations involving impact or shock loading require dynamic solutions coupled with other nonlinear capabilities, otherwise known as nonlinear dynamic solutions.

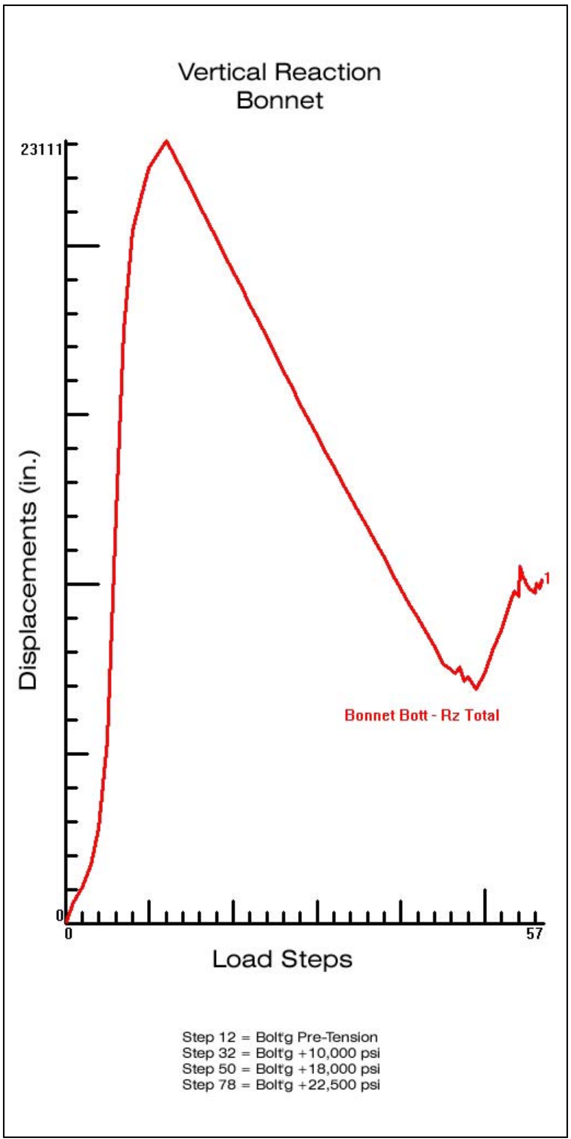


Figure 5 – Valve bonnet's nonlinear contact loading behavior (Note: the ordinate label should read "Surface Reaction Force, lbs")

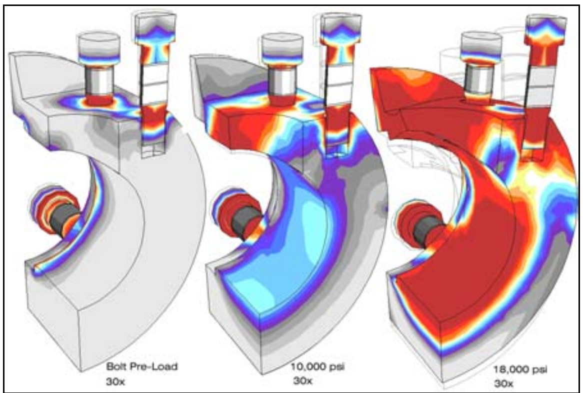


Figure 6 – Center-cut valve section Mises stresses, 30x

A time history direct integration FEA processor can readily solve nonlinear dynamic or pseudo-static solutions of nearly any duration.

Larger companies with long histories of FEA use rely upon their model validation database for cost-effective support of their analyst decisions.

IDENTIFYING CONVERGENCE & VALIDATION OBSTACLES

In general, regardless of the number of modeled components or the complexity of the many nonlinear features employed for a simulation, the following philosophy can help guide both the identification and resolution of convergence difficulties:

> Given that only smaller portions of valid, nonlinear FEA simulations actually involve significant nonlinear behavior… If each increment or phase of the event simulated can be made to behave as linear as the model's idealization, there should be no convergence difficulties related to spurious processor calculations, such as surface contact chatter, yielding or resonant vibrations of non-critical model components.
>
> Essentially, all convergence challenges should be concentrated on the truly critical model components and their features having the nonlinear behavior originally idealized. Phenomena such as surface impact, plasticity, large displacements or motion of critical model components will justifiably demand the most processing time.

Bolt Preloading & Residual Effects

The primary reason for including a preload of any kind is to achieve the residual effects that comprise deflections or distortions, internal reaction loads and stresses.

Achieving accurate representation of these residual effects is or should be one of the main focuses of any assembly simulation model, as in the case of threaded fasteners.

One study concluded that "… initial bolt stress remains a key parameter in the joint performance…" [1]

Threaded fasteners of any kind represent a significant modeling challenge, even with the simpler headed bolt and nut.

Also common but much more difficult to model are the threaded connection designs for engagement of male fasteners with blind holes. This special case is almost nonexistent in FEA literature.

Figure 7 at right depicts the cast body of the featured ball valve test assembly model detailed earlier (Figures 1, 2 & 6). The foreground image illustrates the preload condition and the background image, the subsequent pressure loaded condition. Both images have their displacements scaled ~300x for clarity.

Note the extruding and flaring effects achieved at the visible rims of all threaded holes, the closure plate seating surface stresses during preload, the concavity of the preloaded barrel and the convexity of the pressure-loaded barrel.

This is the basic behavior of such a component. It is also relatively intuitive, except for the significance of the hole geometry distortions. The behavior of these model features may also significantly affect clearances, seating surfaces and stress risers.

The inclusion of thread axial and radial stiffness is critical to achieving such results. Yet, modeling of the 3D helical thread or even individual concentric rings of threads is too laborious to include in such simulations.

Other more obvious modeling difficulties include the many options for execution of and validation of fastener preloading. Bolt shortening by volumetric or thermal shrinkage is a common approach.

Part-to-part contact and load transfer without physically bonding the fastener to its mating threaded part is necessary to preclude erroneous tensile and shear effects, especially for blind threaded connections.

Surface contact of bolt head or nut with their mating surfaces must not be too "soft" as to inadvertently impart a controlling spring into the connection's stiffness system.

Frictional surface contact is intuitively required for such assemblies, anticipating the friction to stabilize the sliding tendencies. But, such contact features generates additional processing and collectively more convergence error.

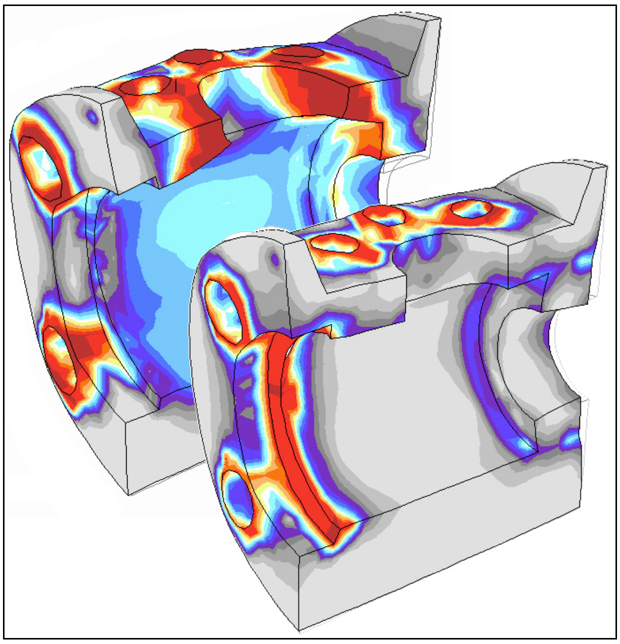


Figure 7 – Cast valve body under preload (foreground) and subsequent pressure (background), 300x

"Earth Entry" Events

An "Earth entry" event such as the onset of gravity loading is one of the most common loading obstacles encountered for the more flexible and larger scale models, especially when simulating dynamic behavior.

It is often taken for granted in nature that gravity is constantly acting upon the mass of all objects whose behavior comprise residual deflections and loading. Our musculoskeletal system is good example of this.

Idealization of the physical event must account for this transient loading as if the model was just entering the Earth's gravitational field.

The problems are that gravity is not automatically preloaded into the model, and that there is a need to have sustained gravitational acceleration prior to initiating other events in the time history.

Unless many small load steps are used, the chance of exciting one or more modes of the structure's natural vibrations is quite high.

These vibrations alone can destroy a solution's convergence, drastically cutting the time step while attempting to capture each oscillation.

What a shame, when typically this data is of no interest!

For smaller scale models gravity is not significant relevant to the other load intensities and is often excluded.

Other Body Accelerations

Centrifugal acceleration imparted to a static part, initial rotation of a dynamic part, or other initial acceleration required for simulating the event is often a similar FEA obstacle as well.

Multi-body Contact & Interaction

Establishing multi-body contact between component parts of an assembly simulation model is probably the most significant obstacle of any nonlinear FEA issue.

Surface contact permits the sensitive simulation of gear dynamics, as shown in Figure 8. Profile errors effects are evident with acceleration plots.

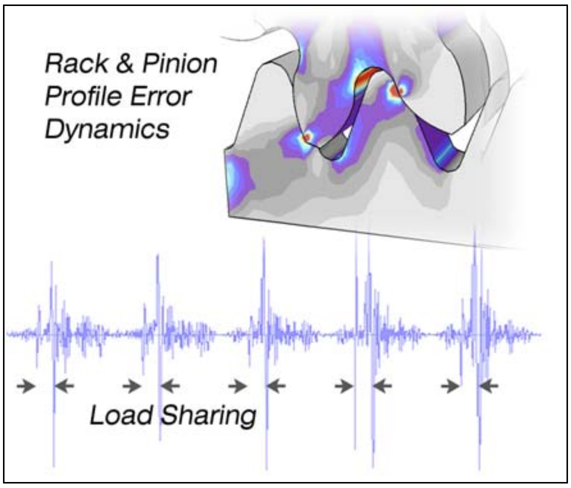


Figure 8 – Evidence of profile error dynamics and load sharing for a rack-pinion gear set.

Multi-body contact cannot be established without assembly component reaction loads and deflections. These residual component displacements cannot usually be imparted as an input.

Typically, the nonlinear processor attempts to solve for these residual loads and deflections on the very first time step of the simulation. Thus, time step size can be a significant parameter.

The most accurate and flexible nonlinear contact method to employ is surface contact. Contact calculations are usually implemented using a lagrangian or penalty stiffness approach.

The penalty stiffness approach is considered as the easier and more stable approach to use, though the lagrangian methods can be more accurate. However, the analyst's procedure dominates the error.

Algor's nonlinear FEA suite provides multi-body surface contact capability that is implemented using the penalty stiffness approach.

Surface Contact Chattering

Surface contact chattering is often a spurious model behavior requiring possibly millions of contact algorithm calculations yielding results of no interest to the analyst. This chatter is form of vibration that can be caused by a lack of surface smoothness, due to the mesh topology, and/or by excessive surface contact stiffness.

It can also be caused by inadequate restraint during contact and allowing excitation of the free-body or other modes of vibration. This chattering is can be developed for finite sliding with friction or for impact events of multi-body models (either intentioned or unintentional impact) during initial contact engagement.

Weld Shrinkage & Residual Effects

The simulation of welded assemblies is usually focused upon the residual effects of the welding process' heat transfer, weld freezing and the interactions of the welded parts.

Originally, all parent metal components begin as cool, loose and virtually unsupported parts. The weld deposition of molten metal to these parts injects a thermal shock load to the system and begins the transient thermo-mechanical event of joining them together as one.

Assuming that the heat transfer shock loading is not too extreme to achieve solution convergence, then the coupled stiffness solution must be able to carefully support but not over-constrain the loose parent material as the freezing welds stiffen and provide their own relative constraints.

Developing physically valid material properties and event parameters is no small task for this multi-physics simulation. Typically, there exists gaps between some parts having no weld metal, which may require surface contact modeling.

The above issues collectively form a set of major obstacles to performing welded assembly simulations. Issues related to appropriate weld geometry, mesh densities, mesh gradients and three-dimensional, moving heat load application to simulate the welding torch further complicates this advanced FEA work. (See Figure 9.)

Contact Friction & Stick-Slip

Use of the nonlinear contact friction capability can be a very accurate modeling decision, but under certain conditions it can also be wrought with untimely and large solution errors causing havoc with convergence.

The stick-slip friction phenomenon is often perceived as one body's acceleration during "slipping" over another one, just after a period of "sticking." This is a rather global view of the event. When, especially for the more flexible bodies, accelerations and decelerations occur at a node level.

Imagine each body divided into many parts, possibly as fine as at each node. For each node, normal reactions and frictional resistance to relative motion are calculated. Obviously, adjacent nodes are coupled to each other by the stiffness of their common finite element.

The resulting behavior can be as odd as that of a caterpillar crawling along the ground, with each set of legs taking limited small strides in sequence or in tandem. The author believes that this is in fact representative of nature's physical behavior for contacting surfaces of some classes of materials with certain stiffness, geometry and frictional characteristics.

OVERCOMING CONVERGENCE & VALIDATION OBSTACLES

The "7-Step FEA" System

7-Step FEA™ is a structured system for performing and validating finite element analysis in engineering design developed and presented by the author for customized training. A flowchart of the upper level categories in the decision tree is shown in the Appendix.

Two, nested iteration loops within the flowchart focus on 1.) Computational models and 2.) Project objectives.

1.) Model and solution parameters are evaluated against the idealization to establish the computational model validity and accuracy.

2.) Project data input, loading and constraint set are evaluated against the event idealization and project objectives to establish validity, accuracy and relevance of the results.

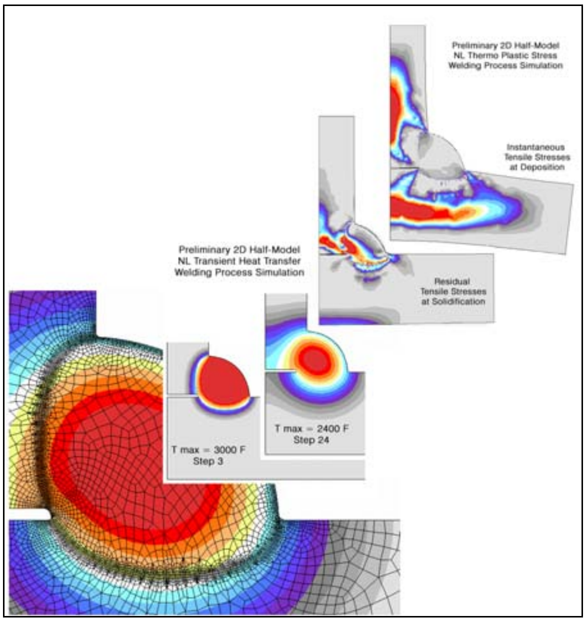


Figure 9 – 2D coupled thermo-mechanical study of molten weld deposition and freezing., 300x

7-Step FEA™ was conceived as an educational vehicle to nurture rapid, competent growth of a company's core resource, the engineer/analysts performing advanced FEA. Hinton [2] described a basis for the four computational model components, steps 2 – 5, in 1992.

Just as in engineering design, the optimization of the process of FEA cannot be effectively undertaken until a complete, technical description or "model" is generated which clearly defines how such analysis work is or should be organized, performed and delivered. This model should consider how it affects and is affected by its component parts – the analyst, client (or management), data, software, hardware, physical product or process and the project environment.

Thus, the 7-Step FEA™ system was developed as to model the FEA project process, a structured and professional set of decision sets and guidelines for performing and validating all linear, nonlinear, static and dynamic, steady state and transient physics-based events comprising most main stream types of FEA.

This system provides an organized framework for the many decisions required of the analyst. Initially, it may be used as a checklist to help plan the primary tasks for an FEA project. This framework could then be custom-fitted with guidelines and techniques specific to the industry, product and analysis types required to achieve convergence and facilitate validation.

The structure inherent in 7-Step FEA™ is rarely found in such specialized fields as engineering analysis, and the author hopes that NAFEMS and other relevant institutions will consider incorporation of this system model into their global training programs.

"Melting Support" Techniques

Variable stiffness components can be added to the assembly simulation model in order to provide a temporary level of support and stability control for component contact interactions.

Figure 10 below presents an assembly model that utilized melting supports or "tabs" between the cover and housing, as well as a common one joining all bolt heads together.

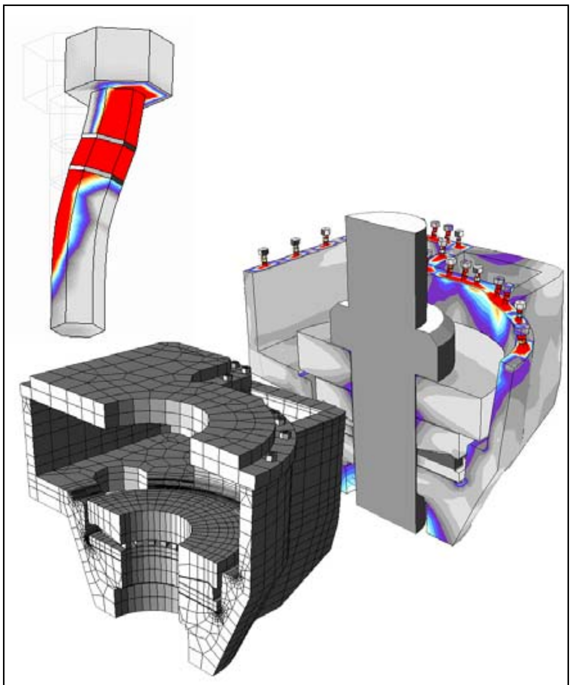


Figure 10 – Drive assembly model with preloaded bolts stabilized by "melting supports" at the bolt head and between cover and housing.

These support components are comprised of new continuum elements attached to the target parts and sharing common nodes. Their shape may take any convenient form. However, care must be taken to consider the rather large strains experienced as energy is absorbed, so not to develop invalid elements.

They may be added between assembly model parts experiencing initial surface contact for instance. Or, they may be added for initially fixing a component in space. Their stiffness will ultimately be removed, gradually after the temporary stability objectives are achieved.

Such stiffness control can be simply afforded by use of thermo-elastic material property definitions. This inspired the label "melting support" as an artificial temperature rise is used to reduce and remove their stiffness.

The melting supports themselves do not require "birth" and "death" states a capability available in some FEA codes. Instead, they simply remain connected to one of the parts being stabilized.

Relative to stiffness matrix compilation, the model components connected and stabilized by these melting supports actually become part of the same stiffness system and behave as one part, temporarily.

The above technique can provide support and stability for any duration. These supports are controllably "melted" as slowly as needed. Thus, the convergence problems associated with initial surface contact requiring solution convergence on the very first time step are overcome.

Material damping features may also be added to these supports if energy dissipation is sought.

One version of the featured ball valve assembly model also included a melting support fashioned as an internal annulus portion of the blind hole. Preloading of thread axial and radial strains normally absorbed on the very first time step were thus relaxed to facilitate convergence.

"Super Modeling" Technique

"Super models" were first introduced by the author at a NAFEMS seminar presentation [3], and represent an extremely efficient method of both achieving convergence while addressing validation.

In general, a super model as opposed to a "super element" is a collection of various types of elements into a functional model component, such as a bolt connected to an annular portion of its blind hole.

The super model of a bolt and hole was initially tested, refined, duplicated and finally inserted into the featured ball valve model. However, its status as a complete model allowed independent processing of the preload condition solution, including surface contact loading and melting supports.

A super model's size and processing time are always fractional relative to the assembly model. Thus, experimentation on super model behavior is possible. This fact is critical in light of relying upon its proper function, such as preloading, in the assembly.

Experimentation of supermodel behavior includes all solution parameters that prepare the analyst to predict numerical convergence behavior and use appropriate settings for the assembly.

Welding is another formidable use of super models, as the modeling of welding processes is extremely challenging without considering the huge geometric burden of the joined materials.

Surface Contact Studies

Individual test model studies of less than five or ten elements should be conducted as a matter of course for all analysts aspiring to assembly simulations involving surface contact.

Such studies should address the intended contact of a single node upon another element's edge (2D) or surface (3D). It should be possible to predict the exact magnitude and reaction force vector experienced at any point in time.

NAFEMS benchmarks by Feng and Prinja [4] should be studied, along with Hertz contact behavior which affords an exact analytic solution (Figure 11).

The author has performed several in depth studies of Algor's surface contact capability, implemented using a penalty stiffness approach. Results indicated the expected outcome of convergence difficulties at nodes aligned with contacting element boundaries.

Recent testing [5] indicated vast improvement in surface contact convergence and predictable behavior resulting from implementation of the author's suggestion to allow marginal extension of surface contact boundaries.

Key to surface contact success for assembly simulations is the nonalignment of all participating nodes. However, unstructured meshes may leave this possibility somewhat to chance.

The most common error is simply the specification of nodal contact stiffness that is too soft or too hard. The stiffer material should generally be used to calculate the stiffness.

Contact stiffness is also proportional to the contact distance modeled between the participating parts. Thus, this parameter is used in calculating the nodal contact stiffness mentioned above.

Bilinear contact stiffness is also available and is most valuable when gapping is not required. An initial soft stiffness is used to establish contact on the first time step, actually representing a gapped condition. Then, as the parts attempt penetration, stiffness is increased, representing the realistic material contact stiffness.

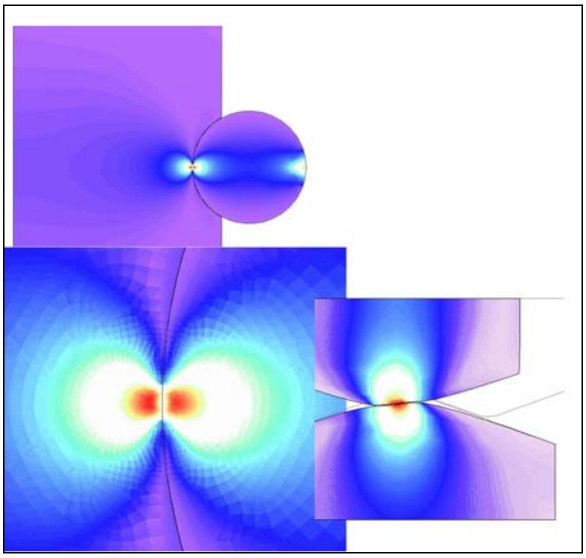


Figure 11 – Hertz contact studies for bearings and sliding over retainer features. Subsurface peak stresses govern case hardening decisions.

Mesh density should be quite uniform for each surface so defined, as each participating node in contact will contribute its normal reaction load. Thus, for surface-to-surface contact, as opposed to node-to-surface contact, nodes on both surfaces are contributing stiffness.

Definitely, initial contact surface interference is the most significant technique to facilitate convergence for surface contact solutions. There is absolutely no reason to process an impact event when such dynamics generate no valuable data.

“Precision” Surface Contact

“Precision” surface contact goes beyond the normal expectations of simply transferring gross body loads from one surface to another, it can reflect realistic sliding, load redistribution, damping and frictional resistance.

Gearing is a perfect example of the need for precise surface contact, especially for analysis of dynamic gearing events such that depicted in the “extreme gearing” example shown in Figures 12 & 13. This is a hypothetical gear set of five and ten teeth having approximately a 1.0 contact ratio, a parameter known to initiate and support damaging gear dynamics.

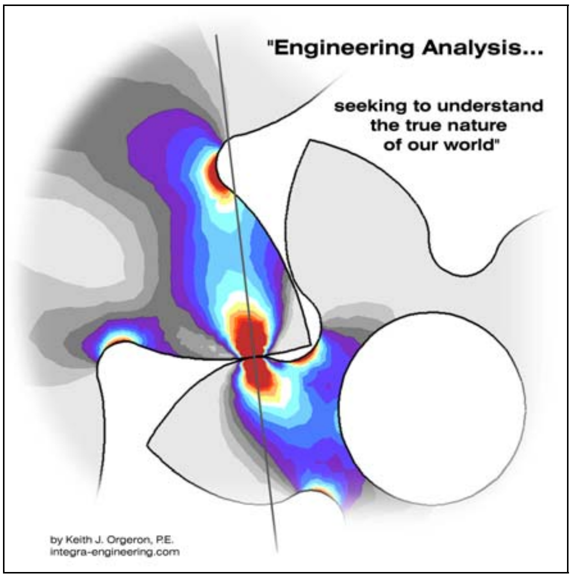


Figure 12 – Nonstandard 5-tooth pinion and 10-tooth gear having 43 degree pressure angle.

The surface contact contours of the mating parts inherently contain a piece-wise linear “surface roughness” feature. This parameter can be manipulated primarily via mesh adjustments to investigate surface finish variation effects.

Important to recall and to have studied, as in the surface contact testing discussed earlier, that surface contact elements may have discontinuous stiffness (=0) at convex surface element boundaries.

At concave surface element boundaries, stiffness may be duplicated such that stiffness of 200% - 800% or more may theoretically exist, given that all surface elements are triangular and never quadrilateral.

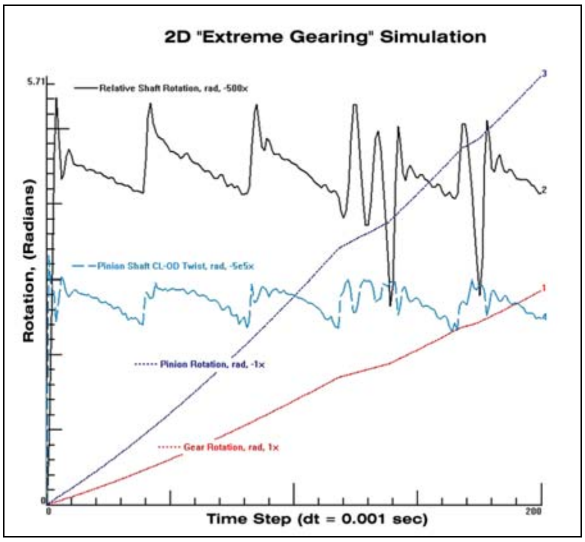


Figure 13 – Dynamics of an accelerating “extreme” gear set (Figure 12), evidencing non-constant velocity due to stiffness variations, profile error and surface roughness, sensitized by its design with a contact ratio of 1.0.

Contact Interference Techniques

If possible, all surface contact should be established as an initial interference. The key issue here is that the value of this interference can be on the order of 1.0E-6 inch or so!

Essentially, since convergence is facilitated by the smaller values of nodal displacement or rotation, it is wise to plan interference parameters with this in mind. The author typically places contacting parts 0.001” or less apart in model space, and provides for interference of only 0.1% of this value.

Validation can be facilitated by such tight clearance of a model’s contacting surfaces, as it allows for larger magnification during post-processing without undue distortion (due to motion across the gap). This supports visual verification of otherwise undetectable part contact behaviors.

Figures 14 & 15 present nonlinear dynamic results of a gas compressor assembly model utilizing such tight clearances (except in one region.)

Interference schemes are routinely used in conjunction with “melting support” techniques.

For assembly models, control over the relative motion of each part after initial contact, can also facilitate convergence. Fewer nodes moving shorter distances are always better.

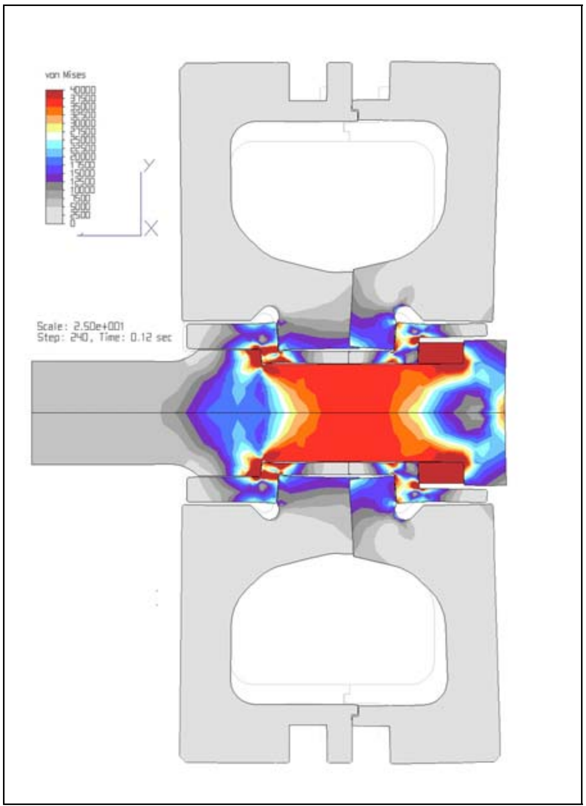


Figure 14 – Gas piston assembly simulation reciprocating at 600 rpm with bolt rod tension, pressure and thermal loading, and no friction at contacting surfaces, Mises stresses, 25x

Frictionless Stability Techniques

Due to the problematic nature of frictional surface resistance, it is highly recommended to omit this simulation feature from the initial assembly model solutions.

Instead, super models or other test models are a better venue for investigating this capability, until ultimately, the project objectives may require that it be included.

The ball valve model featured in this paper was processed with no surface friction. However, incremental bolt head shifting was noted to hamper convergence once or twice during small periods of the event's duration. Increased mesh refinement under the bolt head could have improved this phenomenon.

Frictionless bolted connections can be quite unstable during preloading without friction, attempting to spin on their axes or slide off of the clamped part. Melting supports, detailed earlier, can resolve this issue, but only until their stiffness are melted away.

The most detailed approach for the sliding would be to model the shank surface contact with the hole ID, but this is not usually necessary.

An efficient technique to overcome both problems is to use one or more embedded beam elements of low stiffness connected to the bottom of the blind hole. All but the axial rotation and transverse translation degrees of freedom can be maintained to restrain spinning and sliding.

Ultimately, the elastic deflections of both mating parts will develop a pair of annular dish-shaped surfaces that fit one into the other, thus, nominally restraining the sliding tendency.

Orthotropic melting supports located to fit as a wrench could also be used to limit rotation while a much softer directional stiffness allows for the bolt's axial stretch.

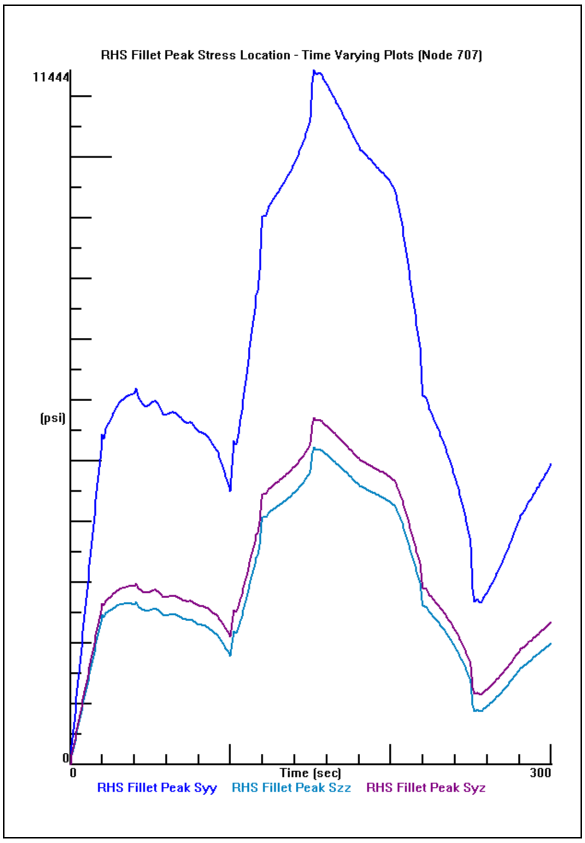


Figure 15 – Cyclic loading develops these stresses in the concave fillets of the piston halves.

Over-damping Techniques

For nonlinear dynamic simulations, such as the gas piston model discussed earlier, a temporary over-damping of the event can be imparted to help stabilize the convergence of a difficult solution. Then, a restart analysis would be processed with either no damping or an acceptable level to meet the FEA project objectives.

Figure 15 below depicts results from model testing to validate the implementation of Raleigh damping within the Algor software for a simply supported reinforced concrete beam.

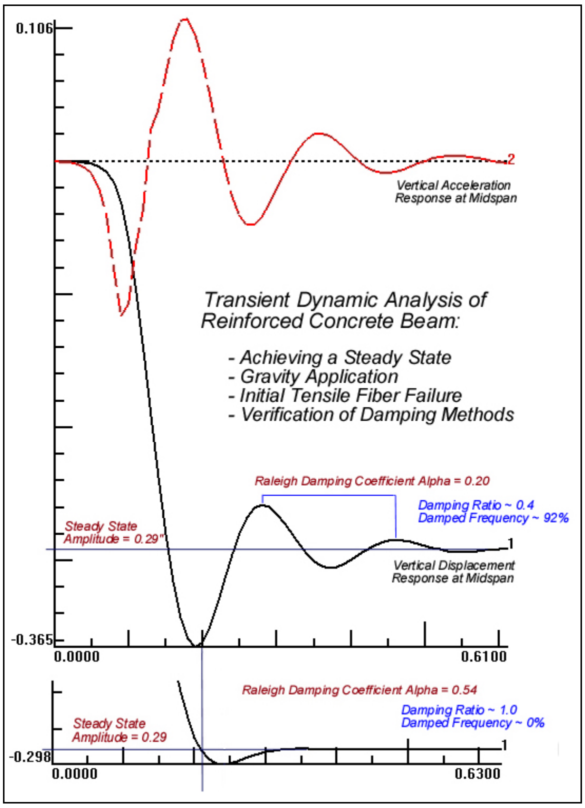


Figure 16

Load Curve Refinement Techniques

It is worth noting that even the use of 1000 point sinusoidal load curves will not guarantee smooth harmonic motion if the numerical values are truncated to just a few significant digits.

Focused testing on this topic confirmed that upon automatic time step reduction during the processor's attempt to achieve convergence the incremental changes in curve slope excited model natural frequencies! Such spurious harmonics are to be avoided at all cost.

Incompatible Mode Element Use

Use of incompatible mode or "nonconforming" elements can often generate results as accurate as higher order elements, with much less processing time due to less nodes and lower convergence error values. Higher order elements can then be used in the final solutions if needed.

Transition Element Meshing

Transition elements include those having less than a full compliment of mid-side nodes at each edge of the element. This allows for nodal compatibility with higher order elements on one side and lower order elements on the other side of the transition element set.

Judicious use of higher order element by combining them with lower order and transition elements will permit the processing of the most accurate model for a given degree of freedom budget.

However, such transition elements are not universally supported for this use. The author appeals to the FEA vendors for such tools.

Relative or Hierarchal Model Feature Validation

Relative or hierarchal forms of validation are possible for each model feature. This is especially valuable when having access to a company's FEA model database.

The relative approach is one of relating project model features to validated, established model features. Basically, geometric, elemental, constraint set and solution parameter model features are individually validated using small models. These validation models are then modified by one feature at a time as a control parameter. This captures quantitative relationships for the common feature variations.

Such feature-based validation may be similar to the lab testing of nonlinear plastics or rubber, whereby characteristic modes of loading for specific constraint sets will yield predictable behavior.

Hierarchal variations of the relative model feature validation approach involve parent-child features and are not as broadly applicable as the relative approach.

A set of model geometry, mesh density, element features and other parameters will generate similar results for similar loading and constraint sets.

CONCLUSIONS

Advanced, nonlinear FEA has been cost-effectively used to model assembly simulations, some of the most demanding but valuable analysis opportunities facing engineers today.

Some of the most common and offensive spurious behaviors found to defeat nonlinear FEA solution convergence have been identified. General guidance and novel techniques have been described for overcoming such obstacles.

As well, FEA model and results validation has been labeled as the process to confirm, establish and help quantify FEA value. Such efforts are notably difficult. Several approaches for accomplishing such validation have been described.

REFERENCES

1. Marchand, L., Laviolette, D., Derenne, M., NPS 4 Class 150 Bolted Flanged Joints Subjected to Pressure and External Bending Loads, WRC Bulletin 450, April 2000, pg 18.

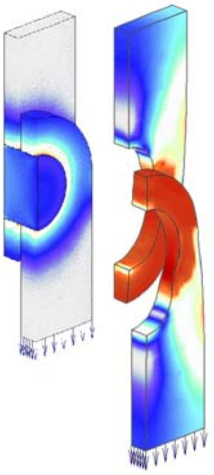
2. Hinton, E., NAFEMS Introduction to Nonlinear Finite Element Analysis, 1992, Bell and Bain Ltd, Glasgow, pp 9-14.

3. Orgeron, K J – Validation of FEA Optimization Projects: Cost Effective Procedures, presented at NAFEMS Seminar: Advances in Optimization Technologies for Product Design, October, 2001, Chicago, pg 9.

4. Prinja, N K and Feng, Q – Benchmark Tests for Finite Element Modeling of Contact, Gapping and Sliding, Proc. NAFEMS 2001 World Congress, April, Lake Como, Italy, pp 1005-1016.

5. Orgeron, K J – Cover letter of Draft Report: Contact Surface Extension Testing, Integra Engineering, Inc., December 5th, 2001, pp 1-30.

APPENDIX



7-Step FEA™

Finite Element Analysis Performance & Validation System

1. Objectives – Per Company Goals & Data

a. Physical Reality & Data Collection
b. Company Goals & Project Objectives
c. Project Constraints

Revisions

COMPUTATIONAL MODEL

2. Idealization – Of Physics-based Event

a. Structural Idealization
b. Spatial Idealization
c. Event Idealization
d. Nonlinearities

3. FE Model – The Numerical Approximations

a. Solution Type Selection
b. Element & Formulation Types
c. Discretization, or "Mesh"
d. Material Property Functions
e. Multi-Body Contact Interactions
f. Kinematics of Rigid or Flexible Bodies
g. Loading & Constraint Set Schema

4. Calculation – The Nonlinear Solution Strategy

a. Solution Iteration Scheme
b. Solution Convergence Criteria
c. Processing Completion Criteria
d. Solution Monitoring, Real-time
e. Output Variables & Frequency

5. Validation – Of Computational Model

a. Software Benchmark Testing
b. Software Operation Competence
c. FEA Procedure Qualification by Experience Base
d. Partial Post-processing & Interpretation
e. Computational Model Evaluation & ***Iterations***
f. Computational Model Acceptance by Analyst

Model Validation

Solution Evaluation

6. Evaluation – Of FEA Project Solution

a. Partial Post-processing & Interpretation
b. FEA Project Solution Evaluation & ***Iterations***
c. FEA Project Solution Acceptance by Analyst

7. Presentation – Of Credible Project Conclusions

a. Complete Post-Processing of Final Results
b. FEA Project Conclusions & Design Details
c. Graphical & Numerical Proof of Analysis Validity