Finite Element Modeling and Analysis Validation

Patrick Safarian
Technical Specialist
Federal Aviation Administration
1601 Lind Ave SW Renton, WA 98057
Tel: (425)917-6446
E-mail: patrick.safarian@faa.gov

Terminal Objectives

- **At the completion of this session you will be able to:**
  - Identify Federal Aviation Regulation requirements for having a validated finite element analysis (FEA)
  - Identify acceptable means of validating FEA results to show compliance to related structural FARs

Introduction - Finite Element Modeling and Analysis Validation

- **Identify 14 CFR, Order and Issue Paper for validation of the modeling and the analytical techniques**
- **Introduction to FEA as an analytical tool**
- **Applications of FEA as a analytical tool**
  - Complex/Detail Structures and Large Structures
- **Building a Finite Element Model (FEM)**
  - Planning an accurate FEM and early validation of results
- **Validation of FEA as part of Certification Plan**
- **Means of validation**
  - Case studies
- **Summary**
- **Check List**
- **Appendices I & II**

Finite Element Modeling and Analysis Validation *Requirements*

Introduction to FEA- Structure 14CFR

- **Requirements for having a validated FEA:**
  - **23/25.301(b)**, "*… Methods used to determine load intensities and distributions must be validated … unless the methods … are shown to be reliable or conservative…*"
  - **25.305(b)**, "*…When analytical methods are used to show compliance with the ultimate load strength requirements, it must be shown that-- … The methods and assumptions used are sufficient to cover the effects of these deformations.*"
    - Note: This is not in **Part 23**. In **Part 23/27** testing is the only option.
  - **23/25/27/29.307(a)**, "*… Structural analysis may be used only if … experience has shown this method to be reliable. In other cases, substantiating load tests must be made.*"

Introduction to FEA- Fatigue 14CFR

- **Requirements for having a validated FEA (cont):**
  - **23.571(a), 572(a)(1), 573(a) and (b), 574(b)** allow:
    - *"...tests, or by analysis supported by test evidence…"*
    - *"...tests, or by analysis supported by tests..."*
    - *"...analysis supported by test evidence..."*
  - **25.571(a), (b), (c), and (d)** allow:
    - *"Repeated load and static analyses supported by test evidence…"*
    - *"…analysis, supported by test evidence…"*
  - **27.571** does not mention analysis:
    - All qualifications *"…must be shown."*
  - **29.571(b)** allows: *"…analysis supported by test evidence…"* for fatigue tolerance evaluation only.

Introduction to FEA- Flutter 14CFR

- **Requirements for having a validated FEA (cont):**
  - **23.629**, ***"(a) It must be shown by the methods of paragraph (b) and either paragraph (c) or (d) of this section, that the airplane is free from flutter... (b) Flight flutter tests must be made to show that the airplane is free from flutter"***
  - **25.629(a)**, ***" ... Compliance with this section must be shown by analyses, wind tunnel tests, ground vibration tests, flight tests, or other means found necessary by the Administrator."***
  - **25.629(e)**, ***"... Full scale flight flutter tests ... must be conducted for new type designs and for modifications to a type design unless the modifications have been shown to have an insignificant effect on the aeroelastic stability."***
  - **27/29.629**, ***"Each aerodynamic surface of the rotorcraft must be free from flutter under each appropriate speed and power condition."***

Introduction to FEA- Order

- **Requirements for having a validated FEA (cont):**
  - **8110.4C, 2-6g**, *"…Use of a well established analysis technique is not enough to guarantee the validity of the result. The applicant must show the data are valid. Consequently, the ACO and its representatives are responsible for finding the data accurate, and applicable, and that the analysis does not violate the assumptions of the problem."*

Introduction to FEA- Issue Paper

- **Requirements for having a validated FEA (cont):**
  - **Generic Issue Paper (§25.305 and §25.307)**
    - The applicant must validate the FEM before it can become an acceptable analysis method.
    - Prior to accomplishing the appropriate tests, predicted strains are generated at strain gauge locations. These predictions are then compared to the test results. A good correlation with small deviation indicates that the model geometry, stiffness data, internal load distribution, and boundary conditions are acceptable.
    - Strain gauges are required in high stress regions and complex geometry.

Introduction to FEA- Issue Paper

- **Requirements for having a validated FEA (cont)**
  - **Generic Issue Paper (§25.305 and §25.307)** (cont)
    - Application of realistic load is used to validate the FEM. Each of the three main aspects of the modeling process should be addressed, that is, external load application, model stiffness (nodes and elements), and boundary conditions.
    - The results from each test must correlate to the predicted results within zero to ten percent for the FEM to be accepted as validated without further evaluation.

Introduction to FEA- Requirements

- **Requirements for having a validated FEA (cont)**
  - According to the Regulations, Order, and Generic Issue Paper, plus good engineering practices, acceptability of the FEA results depends on validity, suitability and reliability of the model and conservatism of the results.
  - The analytical methods and assumptions must be shown to be sufficiently accurate or conservative before they are used as means of showing compliance to Regulations.
    - Analysis must be shown reliable and correct by test evidence or other agreed upon validation methods.

Finite Element Modeling and Analysis Validation

Acceptable Methods

Introduction to FEA as a Tool

- **Structural finite element model (FEM) is a mathematical idealization of a physical structural behavior for engineering analysis.**
  - Remember that FEA is not stress analysis!
- **Some of the common applications of FEA:**
  - Proof of structure
  - Determination of deflection and flexibility or attachment stiffness
  - Distribution of structural Internal loads including fastener loads and payload interface loads (e.g. interior mods)
  - Computation of stress concentration factors
  - Computation of stress intensity factors
  - Computation of mass distribution

Introduction to FEA as a Tool

- **Some of the common applications are (cont):**
  - Static strength and deformation analyses
  - Damage tolerance analysis
  - Dynamic analysis: Modal, Transient and Steady State
  - Stability analysis; e.g. Buckling analysis
  - Nonlinear analysis
    - Implicit and explicit solvers
  - Failure analysis
  - Thermal analysis
- **Experience level of the analyst is of great importance**
- **Quality of the software is essential**

Introduction to FEA as a Tool

- FEM validation is not an event but a series of steps, which includes:
  - **Product Definition**
    - Good definition of the product to model: Dimensions, Materials, Joints, Applied Loads
  - **Analysis Types**
    - Linear, Nonlinear (Large Deformation & Plasticity), Static, Dynamic, Thermal, etc.
  - **Model Design**
    - Accurate representation of geometry and properties: Appropriate mesh size, Choice of element type, Load application, Boundary conditions, etc.
  - **Model Evaluation**
    - Compare to other models, hand analysis, check reaction forces and deformations, look for discontinuities
  - **Final Validation**
    - Validate FEM predictions by test data or other known solutions

Introduction to FEA as a Tool

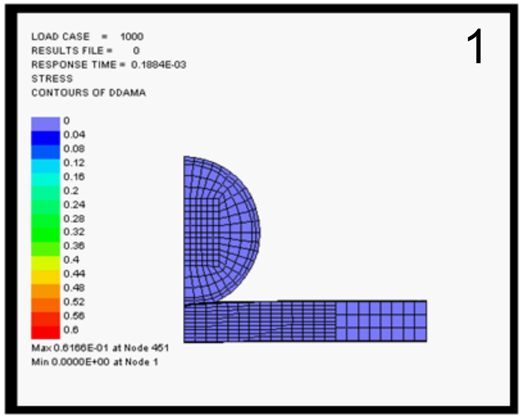
- **A word on Explicit Solvers**
  - Fundamentally used for time based solutions
    - For a state known at a specified time, i.e. displacement and velocity (nonlinear and transient), the solution at a future time step is calculated using finite difference approximations of the differential equations of motion, e.g. Newmark numerical integration method
  - Do not involve inversion of system matrices, so very quick
    - Disadvantage: generally require very small time steps to guarantee numerical stability
  - Physical phenomena such as shock wave velocities usually determine the maximum permissible time step
    - FEA packages automatically calculate the maximum time step and increment automatically based on state of conditional stability

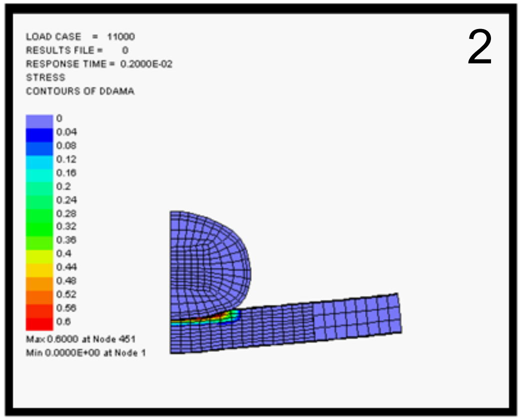
Introduction to FEA as a Tool

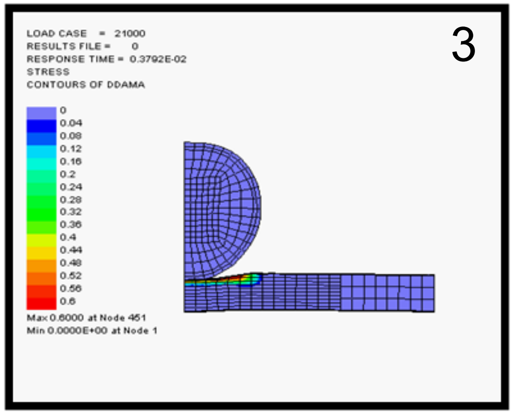
- **Example of an Explicit Solver :**
  - The following example is a soft ball impacting a Nomex honeycomb sandwich panel to simulate a soft body impact of the panel-
    - Similar to a Bird Strike Simulation
  - The analysis estimates crushing of the core and the final deformed shape.
  - The impact lasts about 6.44 milliseconds and contains 35,706 time steps!

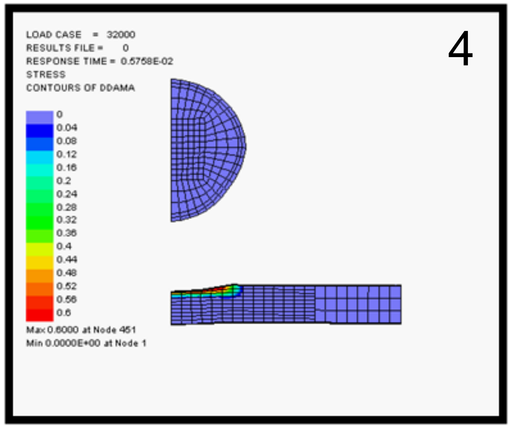
Introduction to FEA as a Tool

- **Explicit Solver Example (cont)**









Click for Video

Introduction to FEA as a Tool

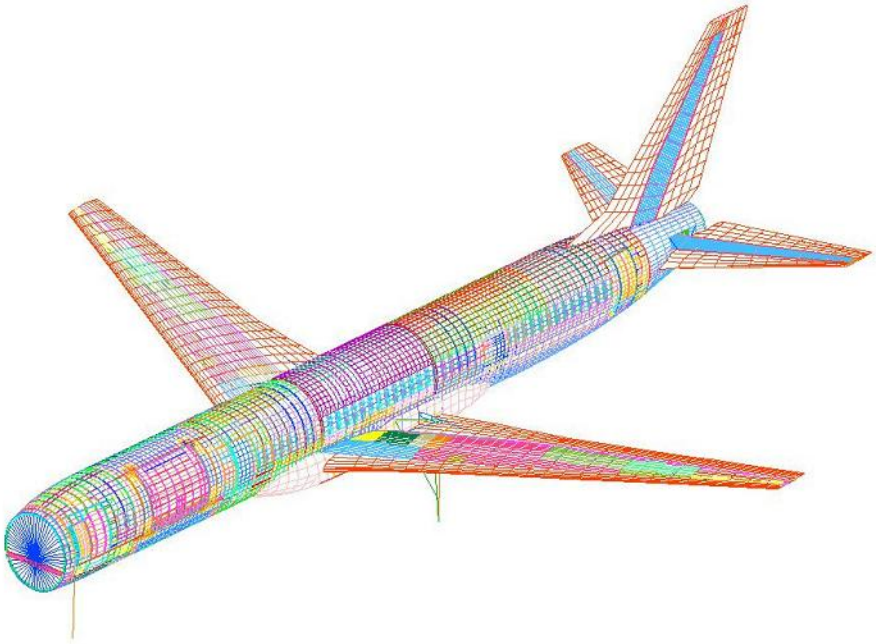
- **Some general steps for *any* FEA process**
  - Establish a clearly defined goal early on
  - Compile and qualify the inputs
  - Solve the problem with most appropriate means
    - Keep it simple- add complexity as requires
  - Verify and document the results
    - Documentation must include restraints and assumptions
- **To establish these goals ask:**
  - How accurate the results need to be?
    - Exact, ballpark, look for trends, etc.
  - What specific output is necessary?
    - Displacements, reaction forces, detail stresses/effects, etc.

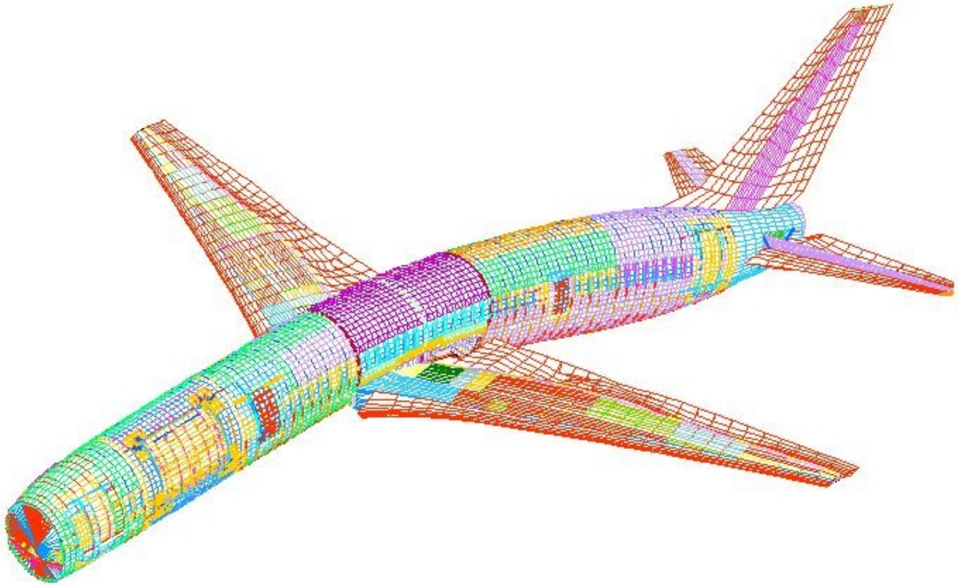
Applications of FEA as a Tool

- **The main advantage of FEA is that it can analyze Large and Complex/Detail structures with many load cases in a timely fashion**
- **Examples of large structures:**
  - Complete Aircraft, Fuselage, Main Deck Floor Beam, Wing and Center Section
- **Examples of complex/detail structures:**
  - Joints, Load transfer, Load distribution in built-up structures, Stress concentration and Stress intensity factors

Application of FEA as a Tool

- **Large Structure- Complete Aircraft:**
  - Study of effects of installation of a major STC on the airframe structural behavior; e.g. MCD, Winglets

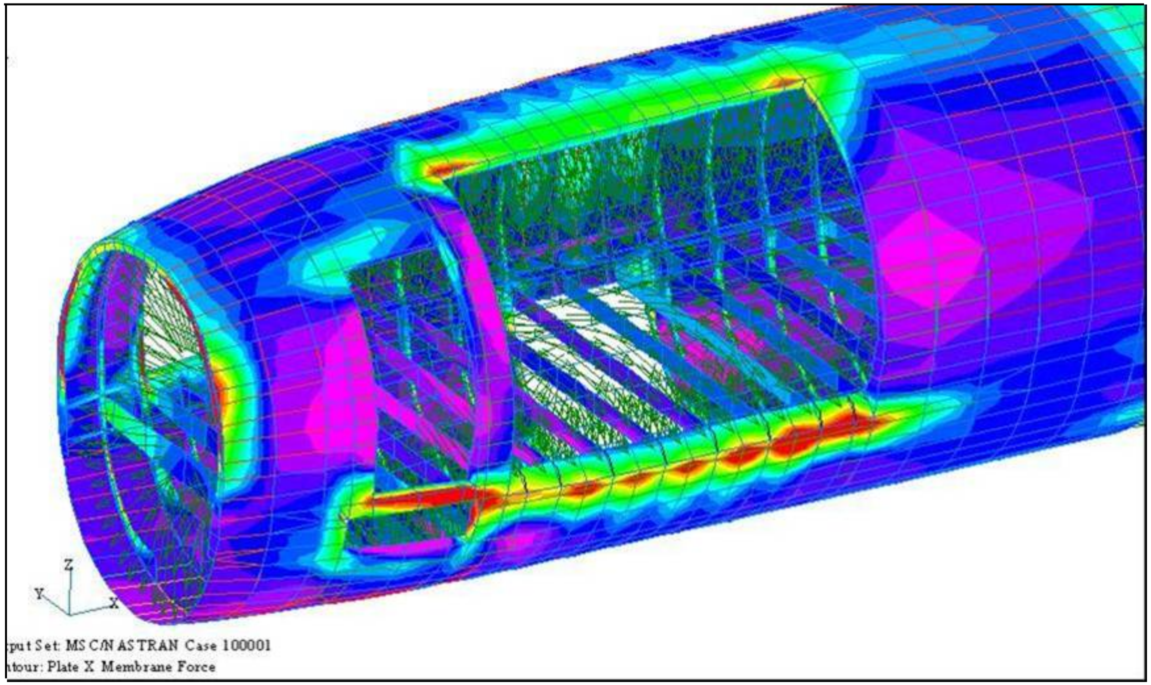




Application of FEA as a Tool

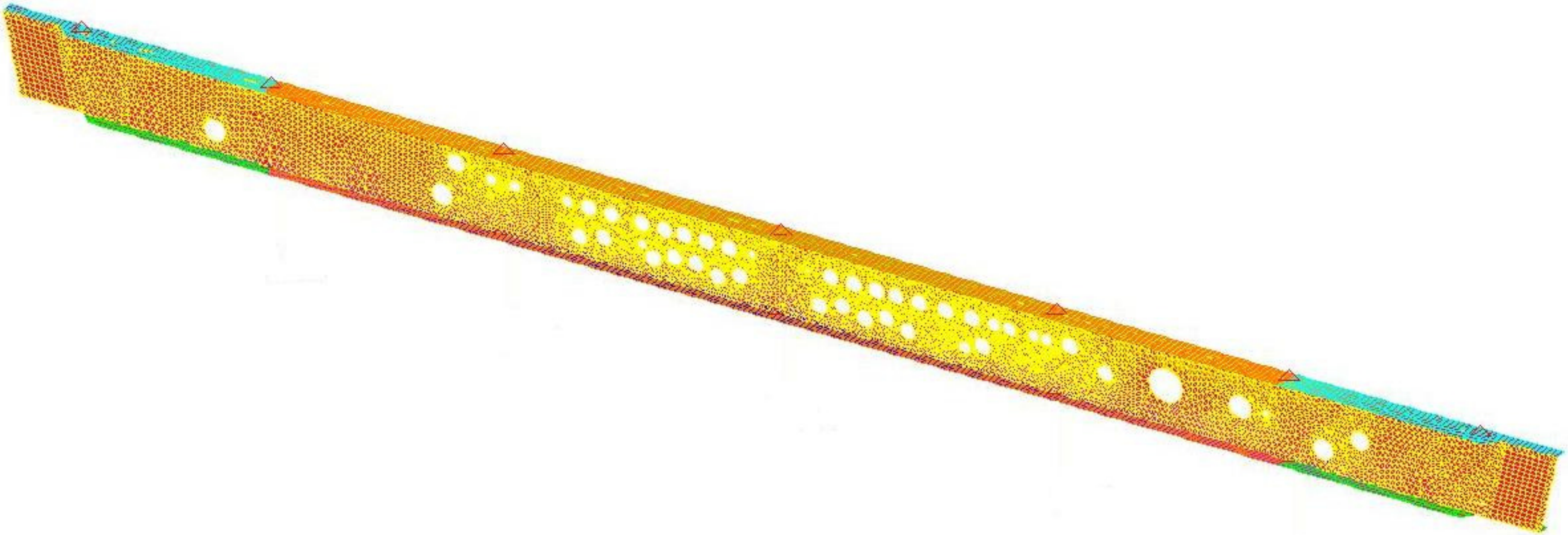
Large Structure- Fuselage

– Study the effects of Main Cargo Door (MCD) installation



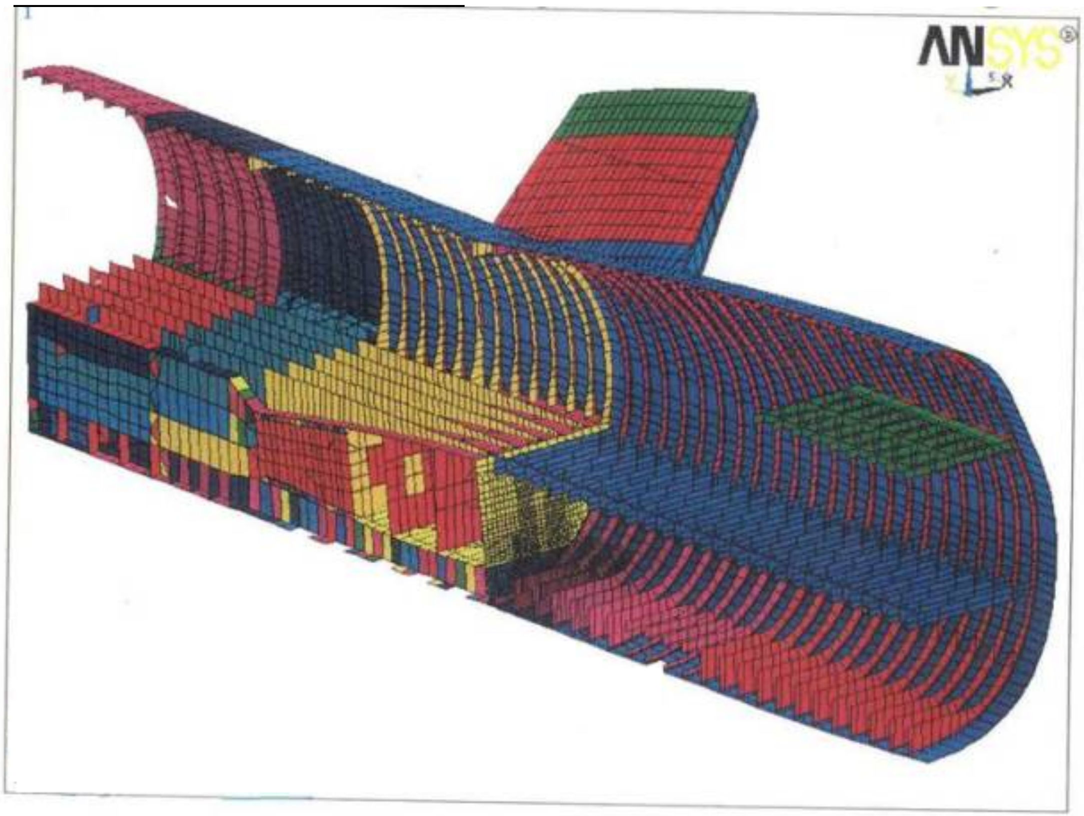
Application of FEA as a Tool

- **Large Structure- Main Deck Floor Beam:**
  - Study of major STCs such as auxiliary fuel tank installation or gross weight increase on the floor beam and fuselage frames



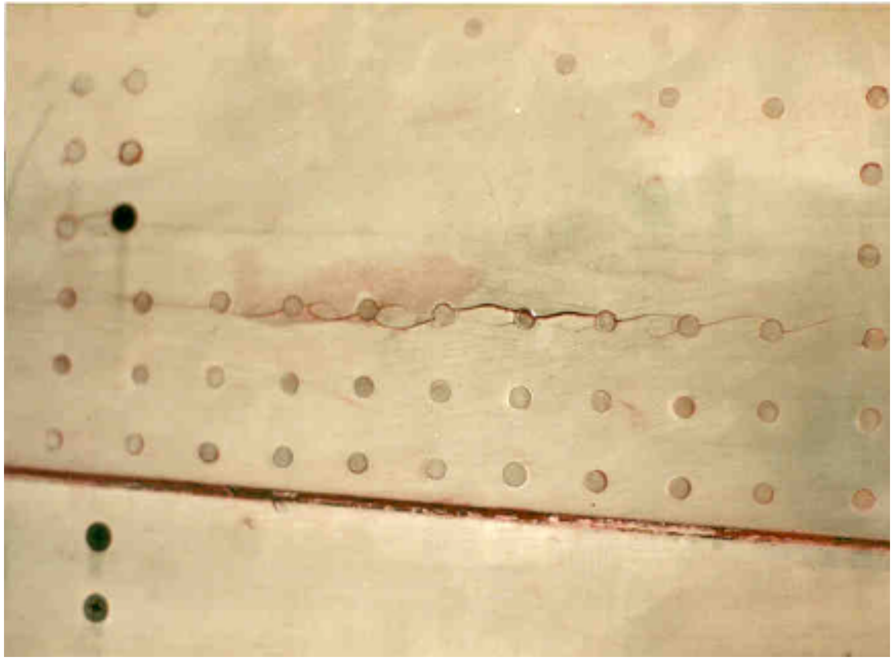
Application of FEA as a Tool

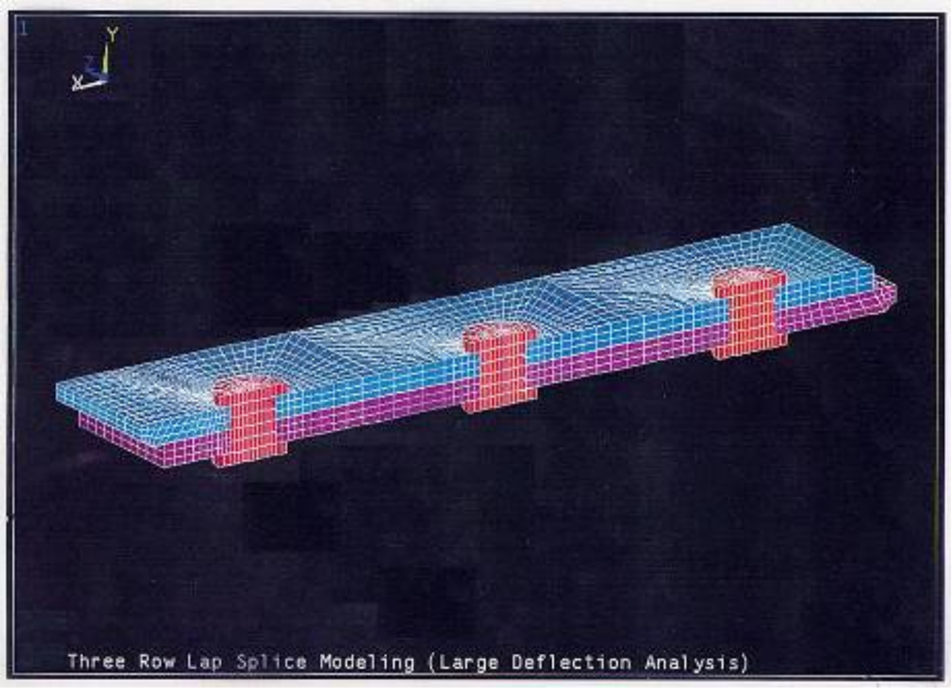
- **Large Structure- Wing and Center Section**
  - Study of effects of wing center tank over-pressurization to the overall integrity of the airframe structure



Application of FEA as a Tool

- **Complex/Detail- Joints**
  - Example of a longitudinal skin lap joint study





Application of FEA as a Tool

- **Complex/Detail- Load Transfer in Joints**
  - Example of an antenna installation load transfer study
    - ***Doubler ends at the critical row of the lap splice (0.071” thick doubler ends on 0.04” thick skin lap joint!)***

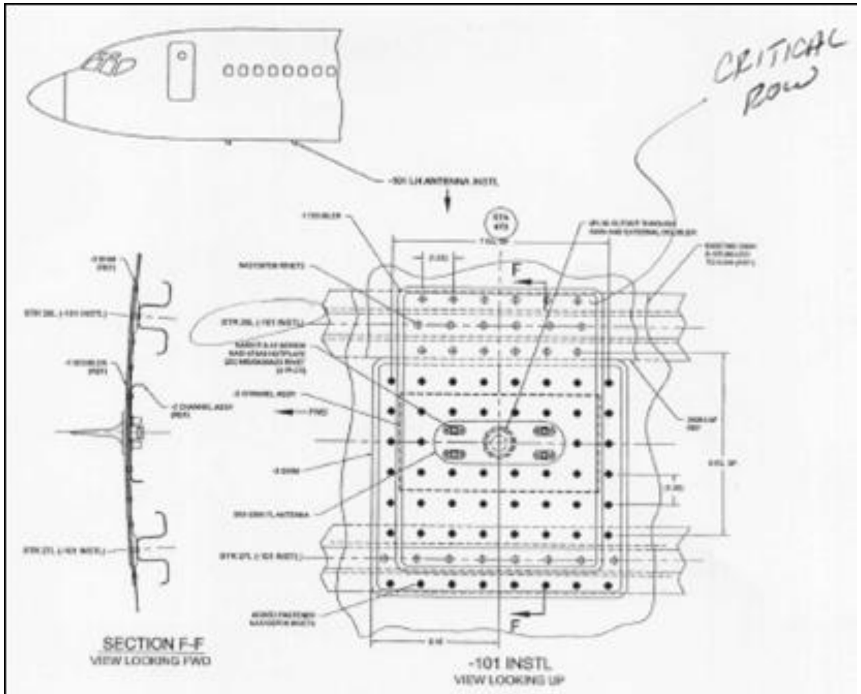
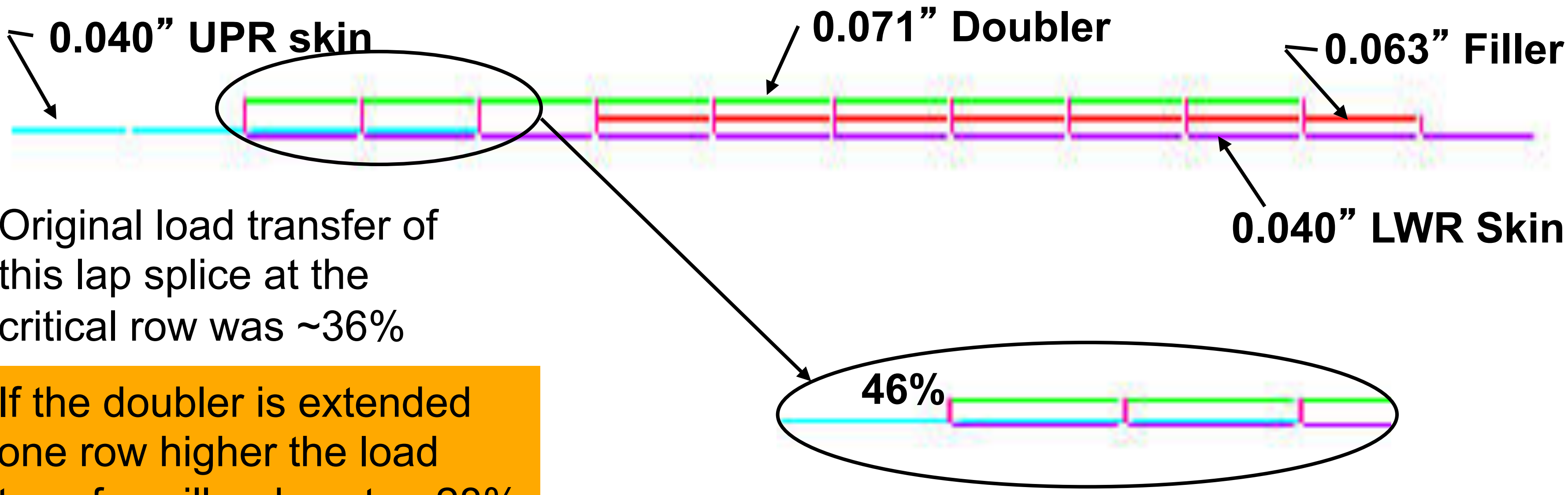


Figure 1

Application of FEA as a Tool

- **Complex/Detail- Load Transfer in Joints (cont)**
  - Example of an antenna installation load transfer study
    - Load transfer is 10% higher at the original critical fastener row



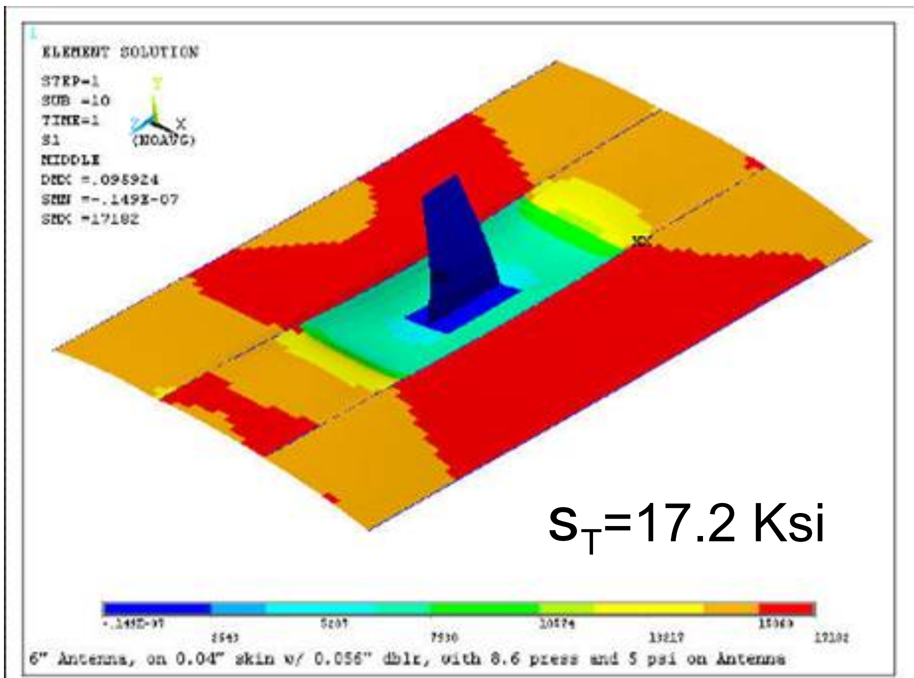
Original load transfer of this lap splice at the critical row was ~36%

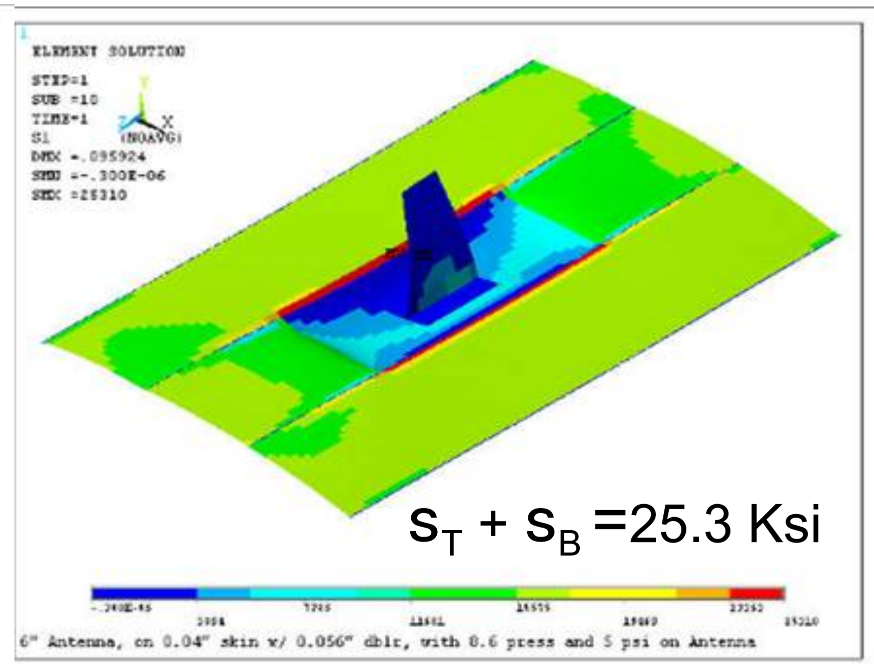
If the doubler is extended one row higher the load transfer will reduce to ~28%

46%>36%

Application of FEA as a Tool

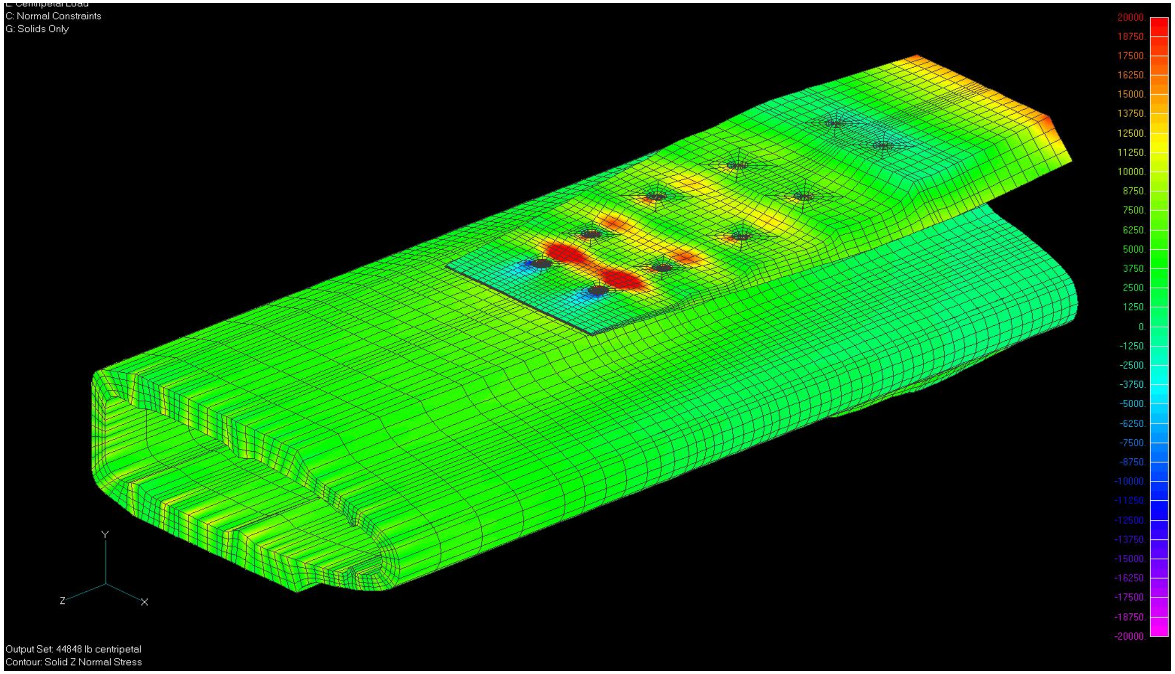
- **Complex/Detail- Skin Load Distribution**
  - Doublers increase the skin tensile stress by ~10% (pR/t=15.9 Ksi) and causes secondary bending stress at the critical row
    - Most contribution is from eccentricity, so repairs have similar effects





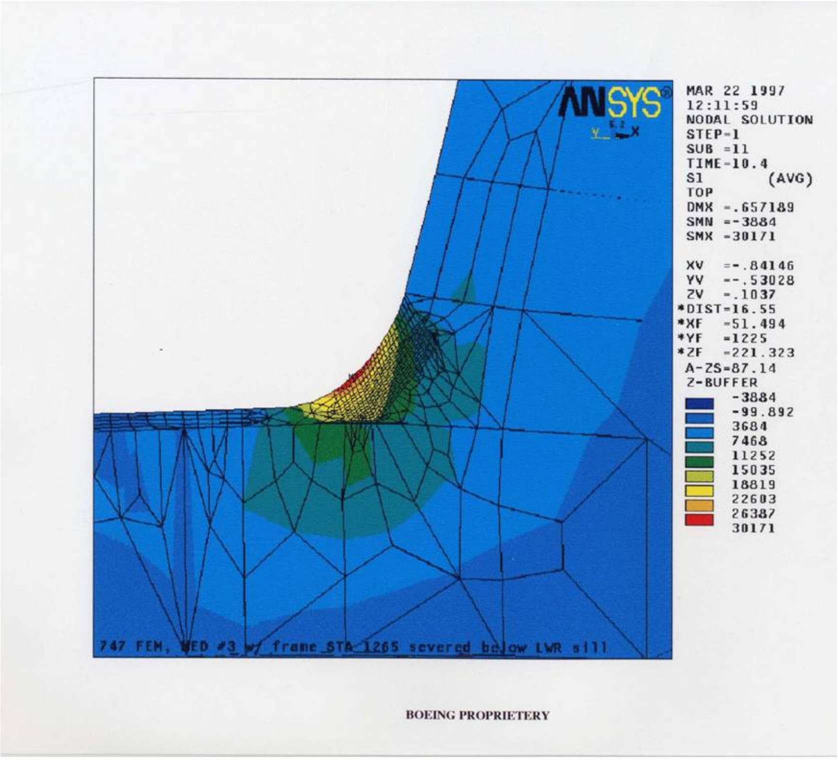
Application of FEA as a Tool

- **Complex/Detail- Stress Concentration Factor**
  - The interface between a rotor blade spar to rotor blade cuff generates stress concentrations at bolt locations.
    - This model was validated by comparison to test data



Application of FEA as a Tool

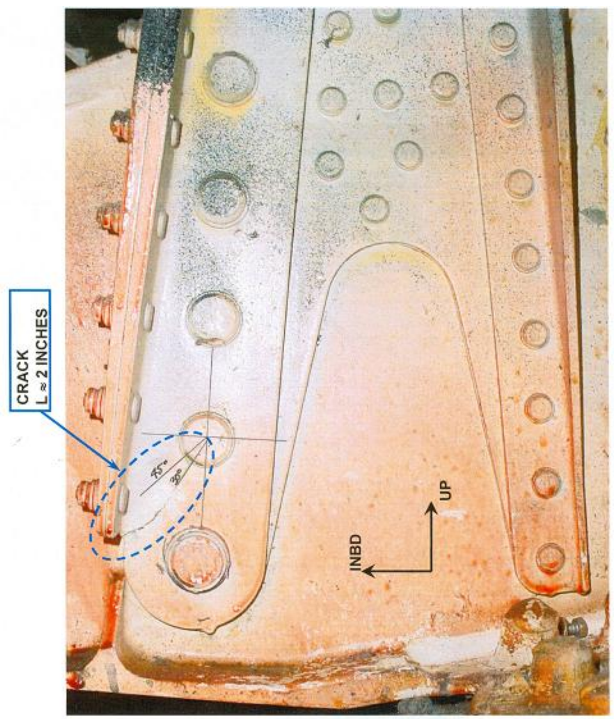
- **Complex/Detail- Stress Concentration Factor**
  - K_t at the lower fwd MED # 3 skin cutout with broken member
  - 747 Classic model major failed frame
  - 3-Dimentional FEM of the entire airframe was necessary to capture proper effects

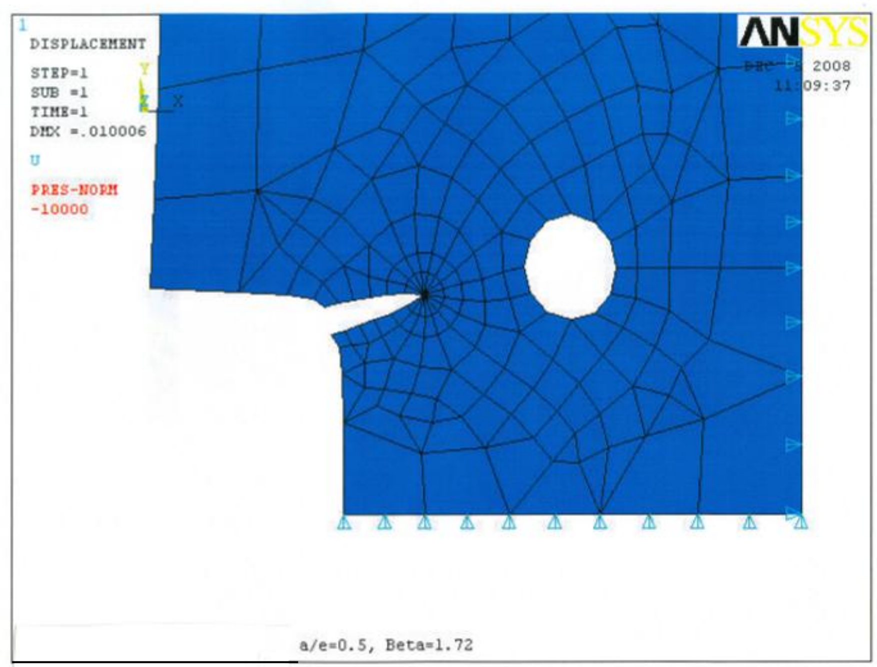


Application of FEA as a Tool

Complex/Detail- Stress Intensity Factor

– Investigation of the fatigue crack growth in a complex structure- *Crack growth results in slide 80*





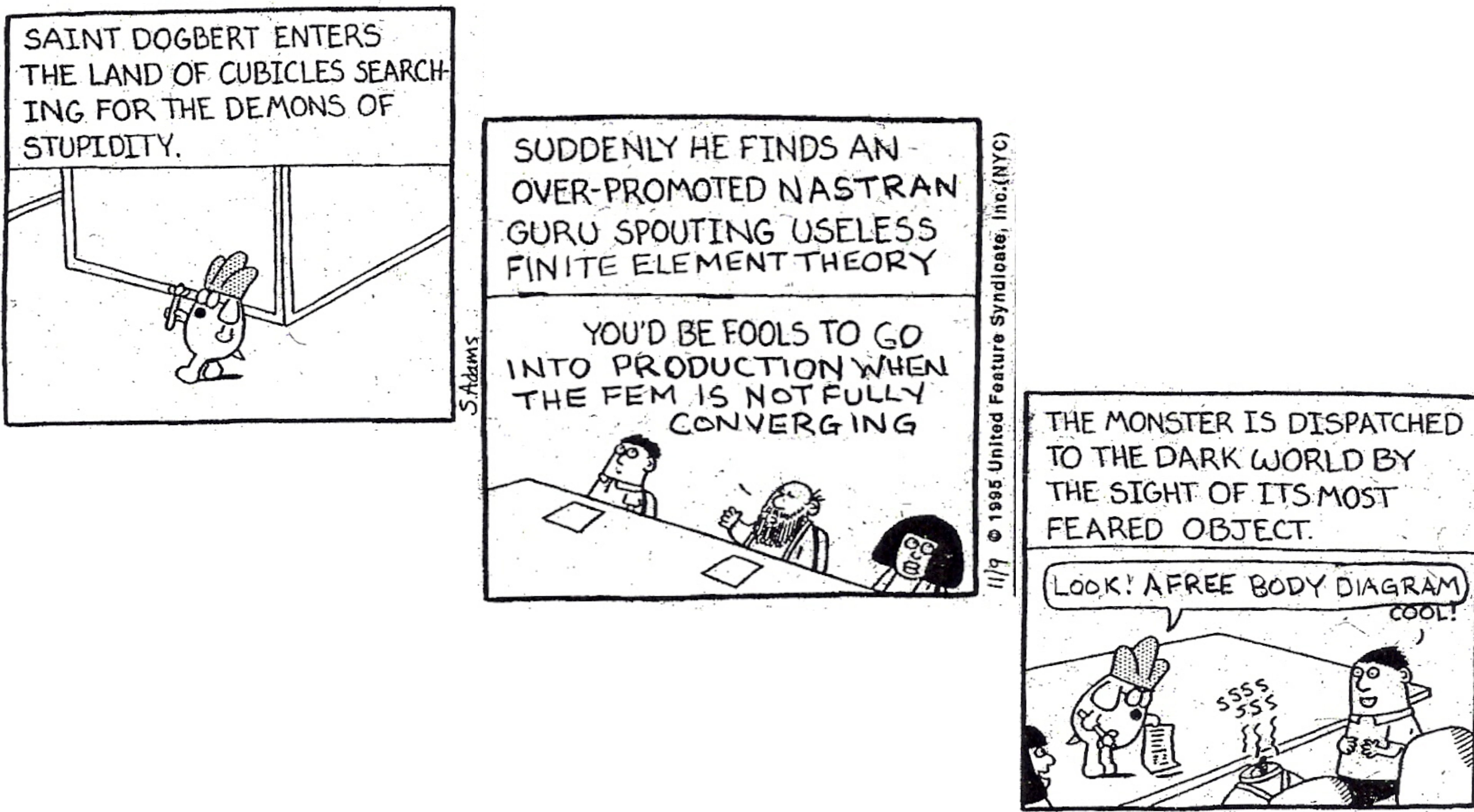
Planning for FEA

- Planning is the most basic step to avoid many future mistakes and save a lot of resources: time and money
- Quality consciousness climate points to check and verify the analysis from the outset
- How much of the idealization is already validated and how much should be validated anew
- Identify the purpose of an analysis at the early stage
  - The source of data
  - The method of idealization
  - The desired results
  - The required accuracy
  - The checking and validation required
- These will influence
  - Allocation of staff
  - Selection of the software

Building the FEM

- **Before creating an FEM, the analyst must develop a Free Body Diagram of the structure; include all loads and boundary conditions**
  - This will provide the analyst the proper idea of the structural behavior and a reasonable idea of the results.
- **Assess the sensitivity of the results to approximation of various types of data**
- **Develop an overall strategy to create the model**
- **Compare the expected idealized structure with the expected behavior of the real structure**

Building the FEM (cont.)



Building the FEA (cont.)

- For most types of FEA the following major steps in creation of FEM are essential:
  - Creation of the model geometry
  - Selection of element type: Rod/Beam, Shell/Plate
  - Idealization of material properties
  - Application of support, constraints and loads
  - Selection of analysis type
  - Solution optimization
- It is essential that in every stage verification of the input and validity of the assumptions are checked and verified

Building an Accurate FEM

- **To achieve the required level of accuracy all analyses require refinement.**
  - Accuracy can be affected by:
    - **The assumption of linearity**
    - **The representation of adjoining structures**
    - **The material properties and idealization**
    - **The accuracy of geometric representation**
    - **The loading and boundary conditions**
    - **The oversimplification of the model or behavior**
    - **The mesh density**
    - **The element types and shapes**
    - **The numerical error in the solution**
  - Global/Local analysis
    - **Use "global" model to compute internal load distributions, followed by "local" FEA or classical methods for refinements**

Sensitivity study of various parameters will help determine which ones affect the FEA results the most and thus requires more time/resources; e.g. fastener flexibility

Early Validation of FEA

- **Model validation should start before the solution stage:**
  - Material data quality
  - Representativeness of the Geometry
  - Choice of elements: derivations, shape functions, orders, types and options that affect formulation and results- e.g.
    - Shell element formulation with/without transverse shear capability
    - Linear elements with constant direct strain in their formulation
    - Shear Beam elements vs. Thin (engineering) beam elements

See A - I

  - Element properties that are assigned to the element
    - Layered material directions vs. smeared/consolidated properties
  - Composite material modeling requires Building Block Approach – do not mix calibration and validation of the FEA

See A - II

Early Validation of FEA (cont)

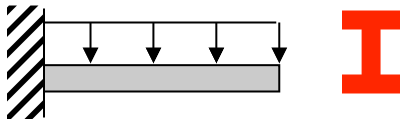
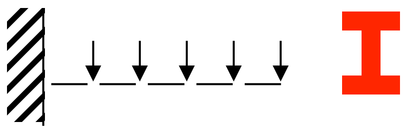
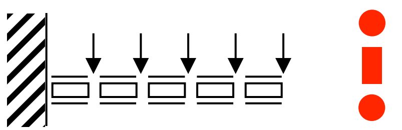
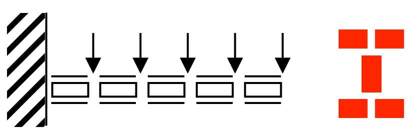
- **Model validation should start before the solution stage (cont):**
  - Connectivity of the elements
  - Consistency of element local direction
  - Constraint equations
  - Supports
  - Loading
  - Adequacy of the mesh density
  - Numerical accuracy of the solution
  - Validity of the idealization of the boundary conditions

Early Validation of FEA (cont)

- **What is <u>Calibration</u> of an FE model?**
  - **Calibration of an FE model is usually undertaken to ensure that specific features which have been modeled provide a realistic estimate of the model stiffness or other behavior.**
    - Spring rate of a bolt can be estimated, later calibrated using test data to get realistic values
    - Determining Composite Material behavior using test results from element level of BBA is <u>calibration</u>
- **What is <u>Validation</u> of an FE model?**
  - **Validation of an FE model is ensuring that the model as a whole predicts measured behavior properly**
    - This usually includes a variety of loading conditions
    - Validation looks for consistency and accuracy of behavior
    - Demonstrating the validity of results at the top of BBA is <u>validation</u>

Early Validation of FEA (cont)

Choice of Elements: Cantilever Beam Summary

Model		Deflection (inches)	Max. Stress (Ksi)	Stress % Diff
Theoretical Solution		0.2837	15.2	--
Beam Elements		0.2837	15.2	0
Rod-Plate-Rod		0.4013	18.1	+19
Plate Elements		0.2843	14.1	-7

See Appendix I for details.

Early Validation of FEA (cont)

- **Preliminary Post-Processing:**
  - Are the reaction forces and deflection as <u>expected</u>?
    - Check the equilibrium of forces against the Free Body Diagram
    - Check excessive displacements or unexpected Rigid Body Motion
    - Check if deflected shape is rational; Use of animation may be helpful
  - Error estimation
    - Comparison of average and unaverage stress values
  - Any areas with rapid changes in stress or deflection
  - Check results of the load cases and their consistency
  - Correctness vs. Accuracy
- **<u>Analyst</u> is solely responsible for the *Fidelity* of the FEM and the *Correctness* of the FEA results.**
  - FEA as a tool has limitations- more with the analysts than the tool. The results should be viewed with skepticism until proved NOT GUILTY!

Validation of FEA as Part of Certification Plan

- **Project Specific Certification Plan (PSCP) defines means of demonstrating compliance with the regulations, e.g. 14CFR Parts 23, 25, 27 and 29**
- **If analysis is the means of demonstrating compliance to 14CFR, the validation method and procedures should be specified:**
  - Compliance with 14CFR 25.305 (a), (b) and 25.307 are by analysis, through use of data generated using FEA. The FEA results will be validated by means of comparison to the test results or classical/known solutions.

Validation of FEA as Part of Certification Plan (Cont.)

- **Means of Validation, Comparison to acceptable data**
  - Test results from FAA approved test plans and conformed test articles
  - Test article is instrumented to provide data for comparison to FEA results, e.g. strain gauges, accelerometers, deflection gauges, electronic displacement indicators, pressure sensors, load cells, etc.
      - Ground Test, e.g. Static Loads Test or Ground Vibration Test (GVT)
      - Flight Test
    - Prior Certified Test- only applicable on case-by-case bases
  - Closed-form or other acceptable analytical methods
    - Analyze parts of structure for which closed-form solutions are appropriate, then compare to FEA results
- **The means of validation must be defined in the PSCP**

Validation of FEA as Part of Certification Plan (Cont.)

- **Identify test plan to validate the model**
  - Pressure, body/wing bending, vibration
- **Establish test conditions which represent similar load application methods, structural stiffness, boundary conditions**
- **Define instrumentation requirements to provide necessary data for comparison of FEA validation <u>prior to test</u>**
- **Test plan submitted and approved <u>prior to test</u>**
- **Test setup must be conformed <u>prior to test</u>**

Validation of FEA as Part of Certification Plan (Cont.)

- **Strain gauges are the typical data acquisition method for measuring the internal loads**
  - Identify and document quantity, locations and orientations of gauges in the test plan according to the analytical predictions- FAA will have to agree with these locations
    - Axial versus Rosette
    - Back-to-back strain gauges may be necessary to measure bending in structures such as thin sheets subject to pressure
  - Provide predicted values for strains in the exact locations of the gauge
- **Establish Pass/Fail criteria for the validation**

Validation of FEA as Part of Certification Plan (Cont.)

- **Documentation**
  - Test Plan
    - Purpose of the test
    - Test conditions
    - Instrumentations
    - Prediction of the results and Pass/Fail criteria
  - Test Results Report
  - FEA Report
    - Description and purpose of the model
    - Validation of the results
    - Input/Output files
  - Keep in mind the user of FEA results may not be YOU!

Validation of FEA by Test

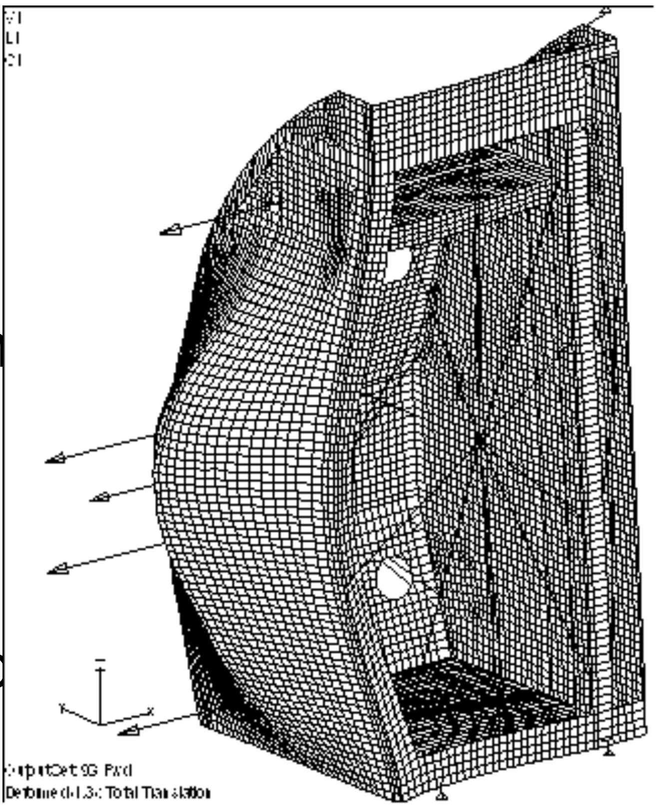
- **Validation by Test- Expectations**
  - Proposed tests will adequately demonstrate FEA ability to resolve principal internal load distributions
  - Test load cases should simulate critical load conditions, e.g. for fuselage structure hoop loading due to internal pressure and longitudinal loading due to combined fuselage bending and internal pressure.
  - Location of measuring instruments must be described in the test plan
    - Avoid placing gauges in high stress gradient areas
    - Best comparison with nominal stresses & their principal directions
  - Pre-Test predictions are included in test plan

Validation of FEA by Test

- **Validation by Test- Expectations (cont.)**
  - Post test evaluation should yield agreement between test and analysis, within an acceptable tolerance.
    - Less than 10% deviation in <u>strain</u> is typically acceptable
      - The acceptability tolerance depends on structure geometry and loads. (Refer to the generic issue paper)
    - Greater than 10% deviation in strain generally requires further evaluations
      - Unconservative results within 10% may require reevaluation
      - Shifting results between +10% & –10% may require reevaluated
    - Possibly FE model changes or FEA analysis/post-processing methodology revisions.

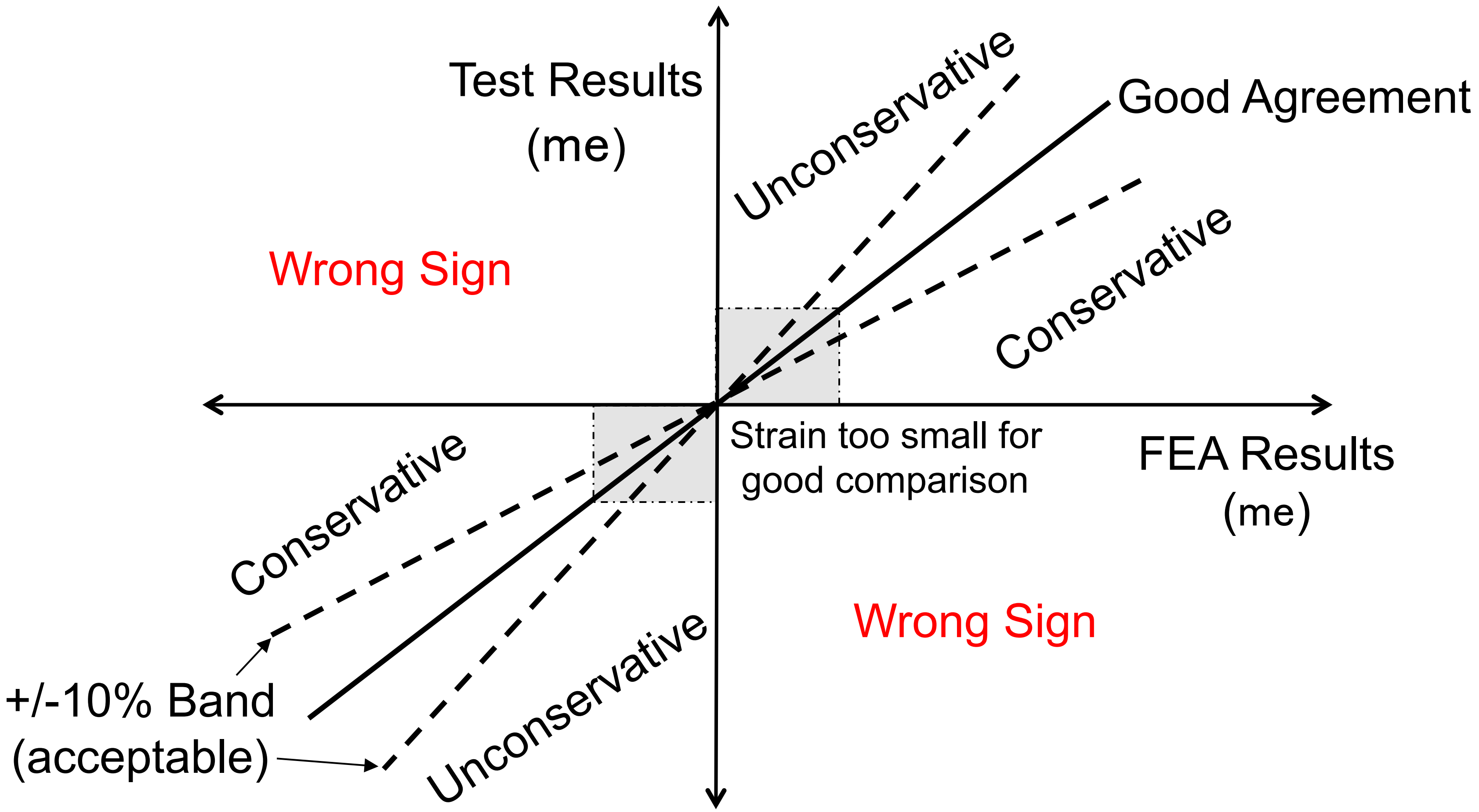
Validation of FEA by Test

- **Validation by Test- Expectation (cont.)**
  - Often in analyzing interior monuments validations includes comparison of deflections to demonstrate validity of the FEA
    - Typically less than 5% deviation in <u>displacement</u> values is acceptable
  - This model of an aircraft interior monument was validated by test to this displacement criteria



Validation of FEA by Test

- **Validation by Test- Strain Comparison**



Validation of FEA by Test

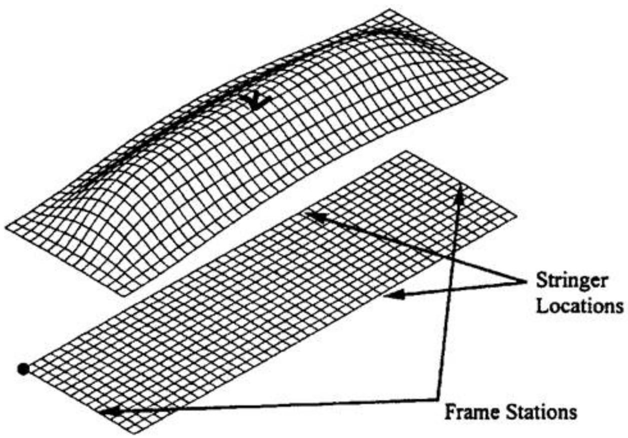
- **Considerations: FEA versus Test**
  - Realize the limitations of the FE model, i.e. is it refined sufficiently to analyze areas of high stress gradients.
    - Actual aircraft structure contains many details, which create localized stress concentrations or have second order load behavior, such as fastener holes.
  - Test data may include nonlinear effects such as hysteresis or residual stresses associated with settling into a steady state
    - These effects should be removed from the test data prior to comparison
  - Test complexity depends on confidence level
  - Confidence Level Depends on Experience

Validation of FEA by Test

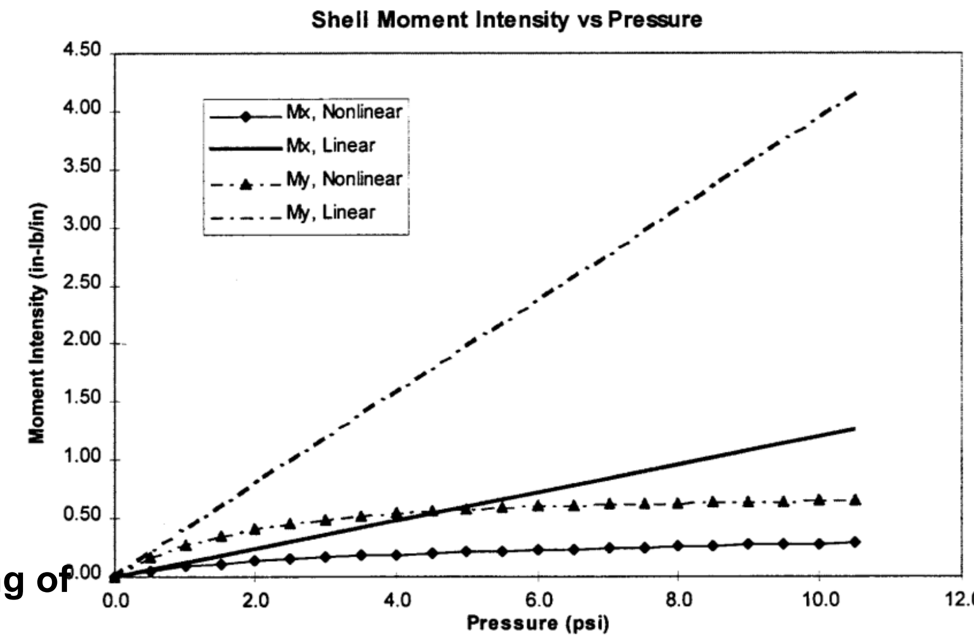
- **Case Study 1: Fuselage skin subject to internal pressure - A Geometric Nonlinear Behavior**
  - Typical "Global" FE model, element refinement of a single "skin" element between stringers and frames.
  - "Thin" skin segment, bounded by stringers and frames, is subjected to uniform pressure over its entire surface.
  - The global model cannot simulate the out-of-plane displacement caused by pressure;
  - Refinements of the model with proper element size and type is required for proper analysis
    - Consider use of "Local" modeling

Validation of FEA by Test

- **Nonlinear Behavior**
  - Strain gauge results shows an initial nonlinearity due to secondary bending, whereas linear FEA cannot predict any secondary bending.

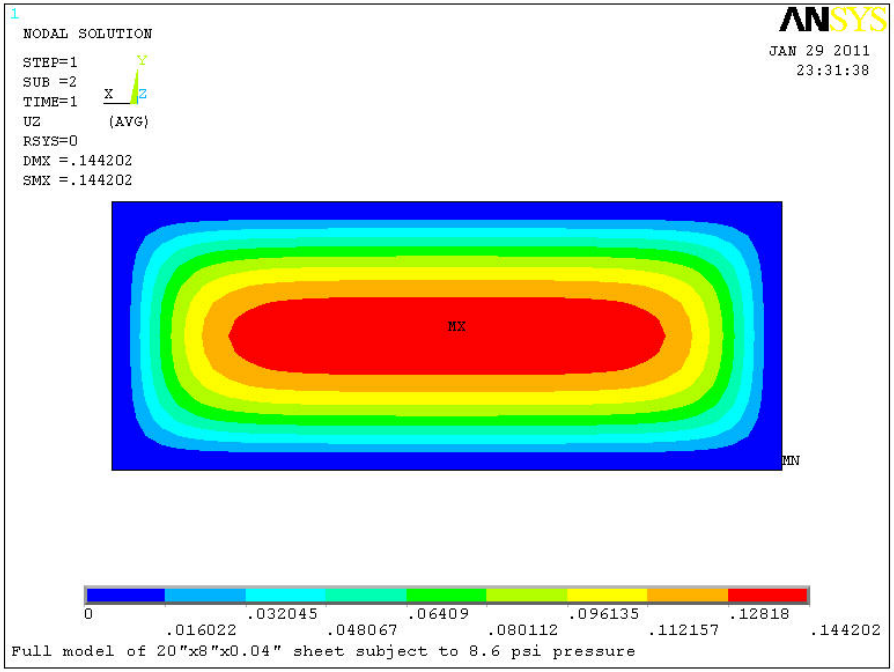


Refined Grid FEM to study ‘Secondary’ Bending of a skin panel segment under internal pressure.

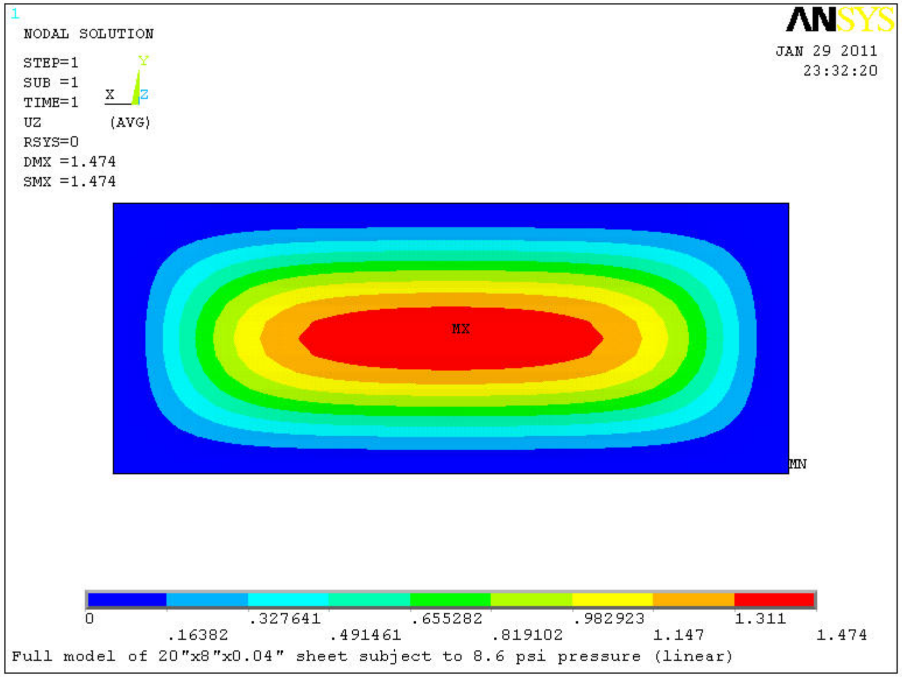


Validation of FEA by Test

- **Maximum Deflection Comparison:**
  - Note the absence of membrane effect in the linear model



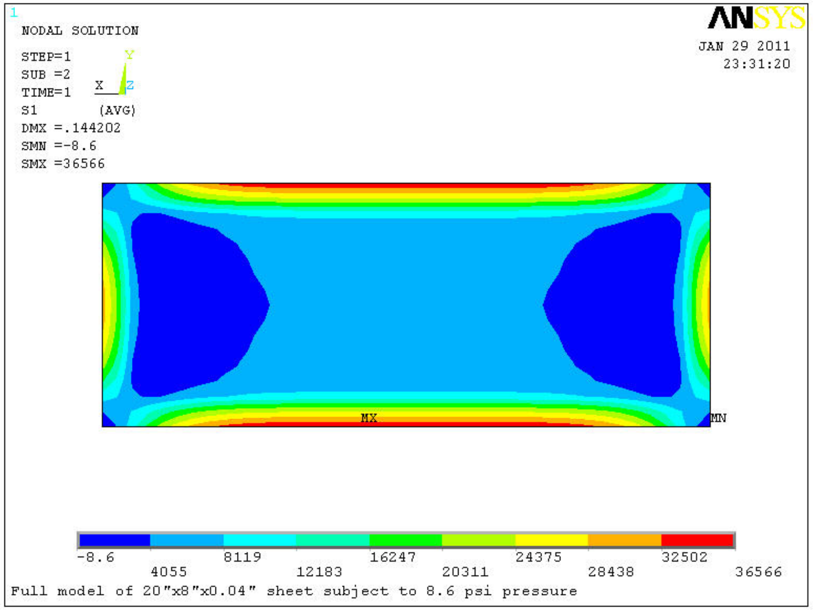
Nonlinear Soln: Uz=0.144”

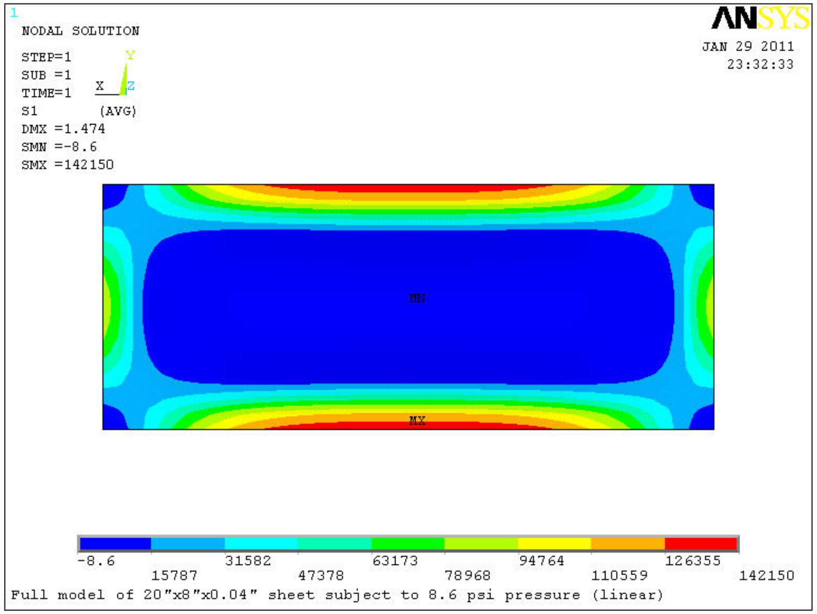


Linear Soln: Uz=1.47”

Validation of FEA by Test

- **Maximum Principal Stress Comparison:**
  - Note the incorrect simulation of stress field in the plate



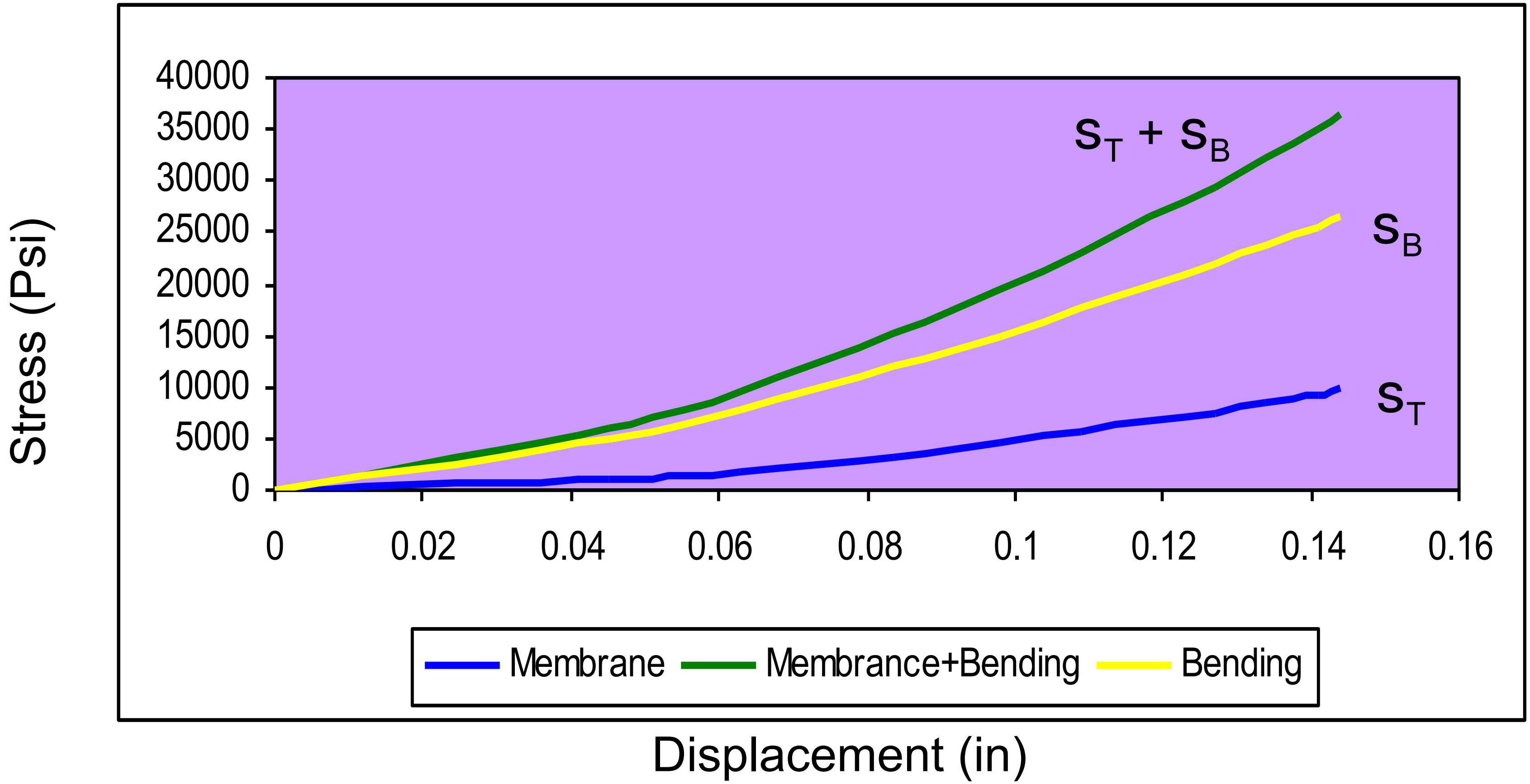


Nonlinear Soln: S_1=37 Ksi

Linear Soln : S_1=142 Ksi

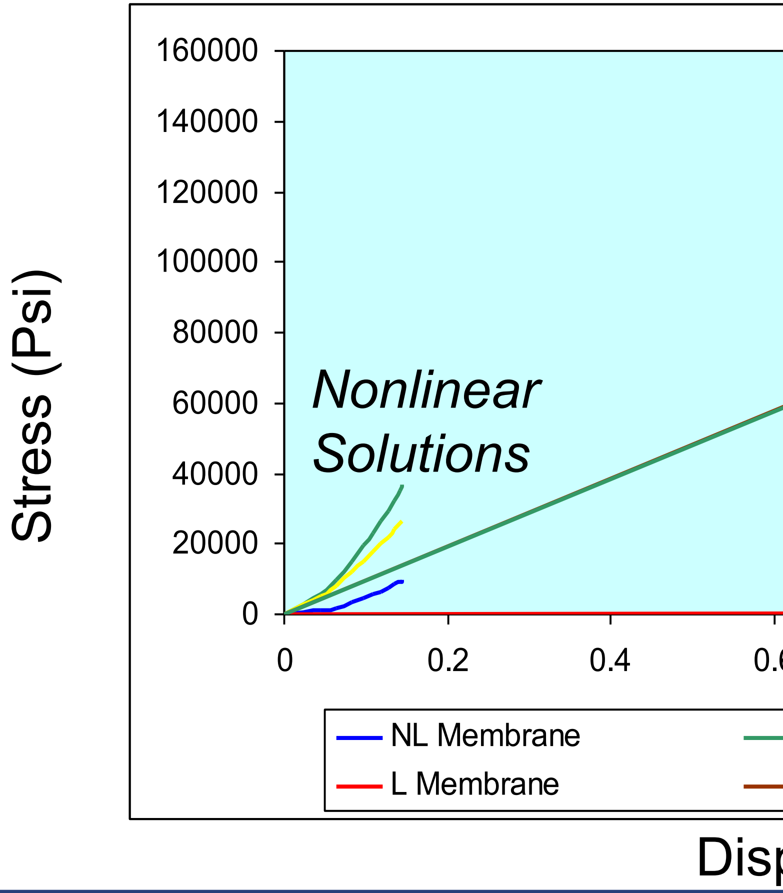
Validation of FEA by Test

- **Max. Principal Stress vs. Displacement of the Nonlinear Solution:**



Validation of FEA by Test

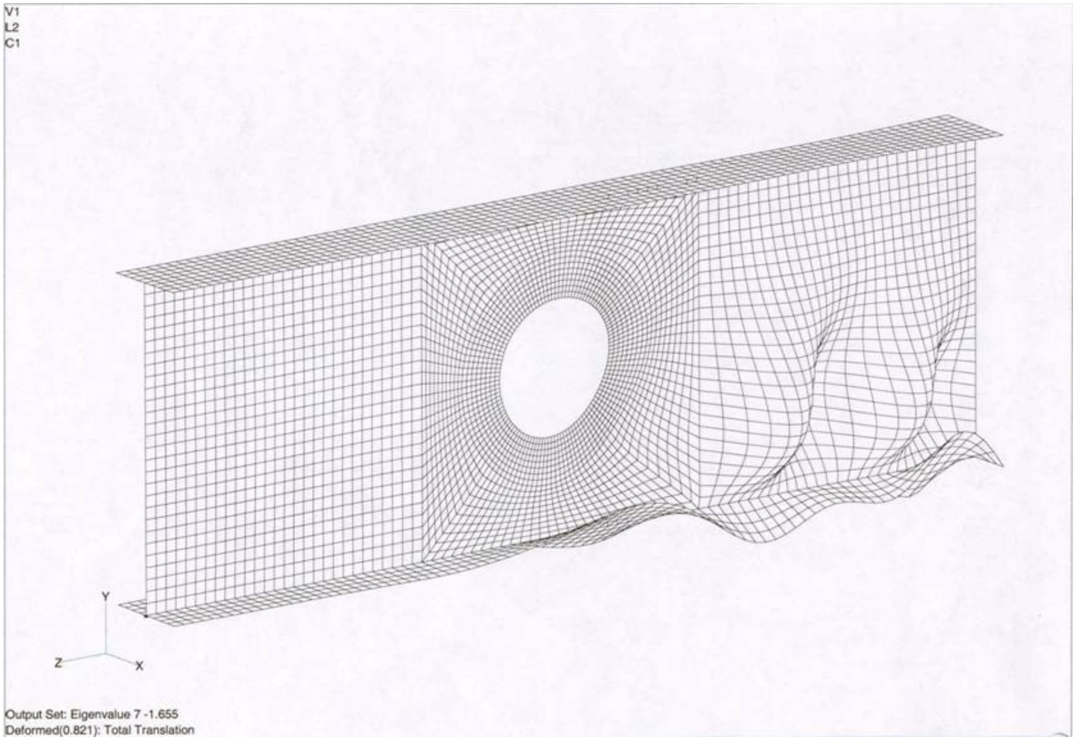
- **Max. Principal Stress vs. Displacement of the Linear and Nonlinear Solutions:**



Validation of FEA by Test

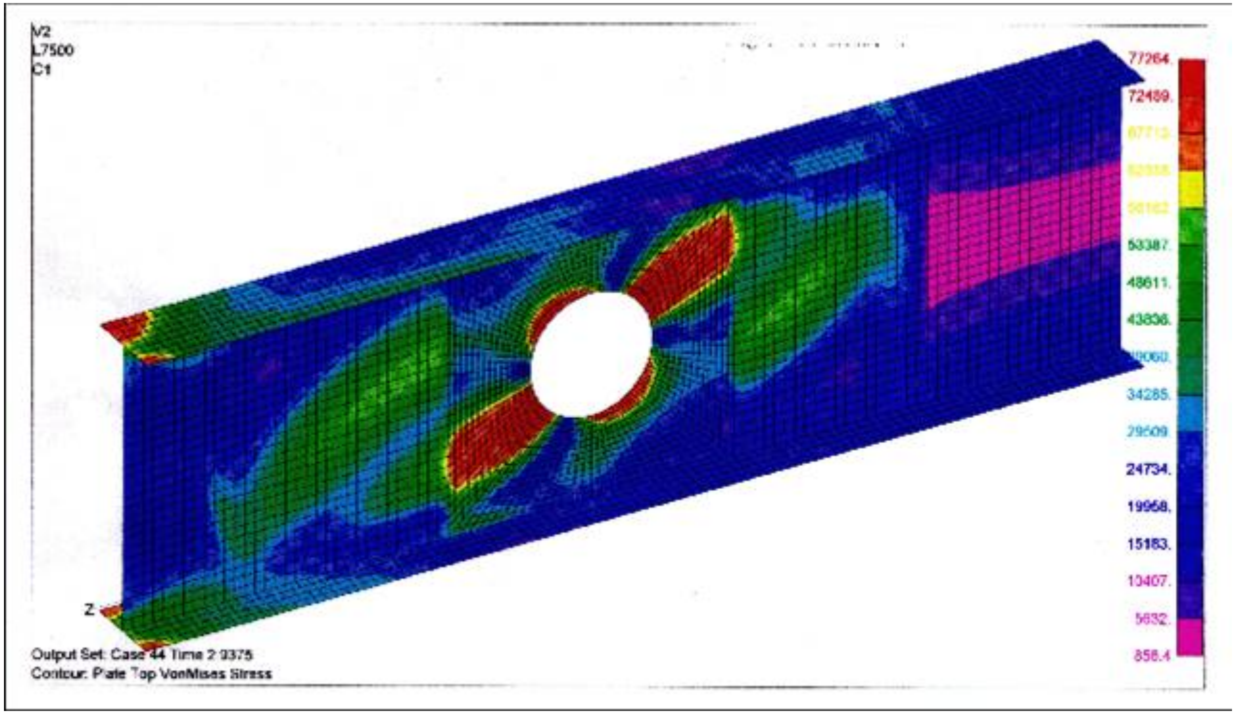
- **Case Study 2: Floor Beam Buckling Analysis**
  - Buckling and beam web fixity analyses are validated by comparison to test results by a margin of 2%

Use of global to local modeling the nonlinear behavior of this floor beam was studied



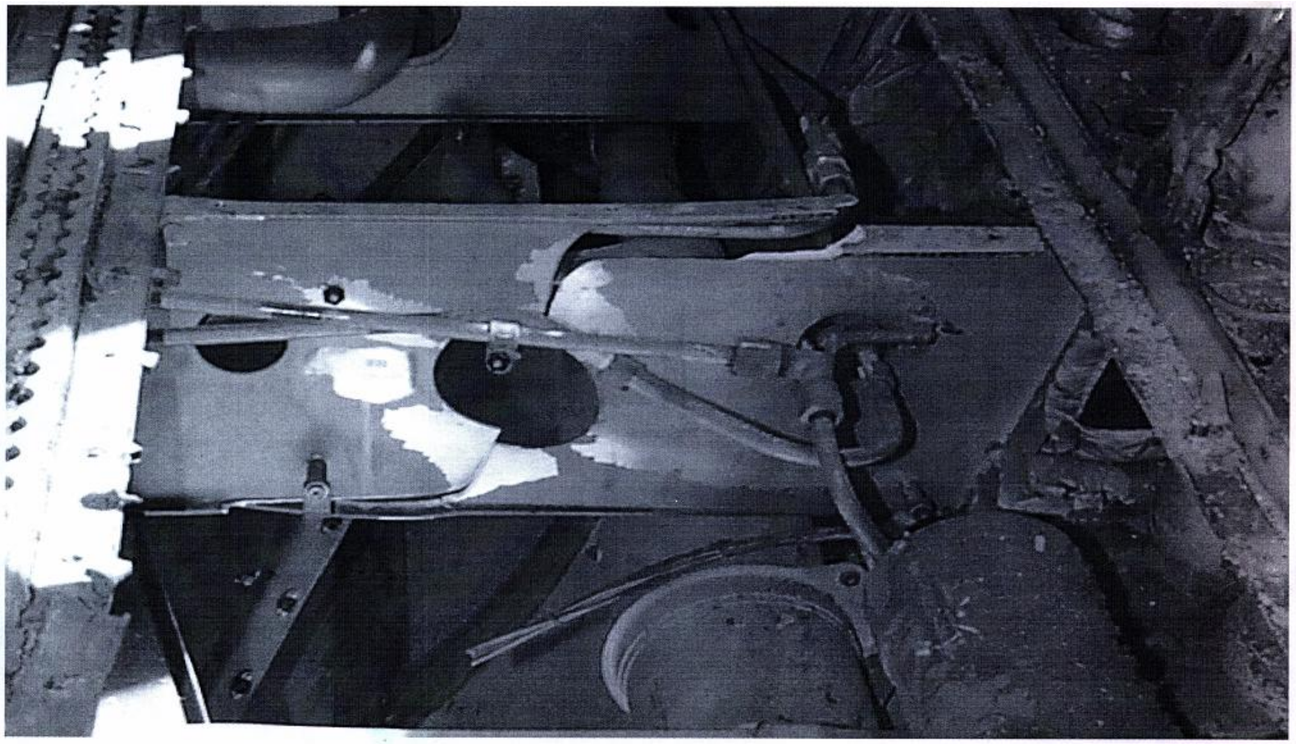
Validation of FEA by Test

- **Nonlinear behavior**
  - Failure Stress Simulation- FEA
    - Beam bending maximum shear stress capability of beam web validated by comparison to test results



Validation of FEA by Test

- **Nonlinear behavior**
  - Failure Stress Simulation- Test
    - Beam failed as predicted by the FEA



Validation of FEA by Test

- **What did we learn from these case studies?**
  - Geometric nonlinear analysis may be required to capture the physical behavior of the structure
    - Note the displacements could be off by an order magnitude!
  - A "global" linear model may not accurately predict the measured strains/failure. For proper solutions try:
    - A "local" model from the "global" model to capture the effects of nonlinear behavior- A.K.A. Sub-modeling
  - Even away from high strain gradient, FEA may not match the actual measurements.
    - Measuring the stress in the middle of a pressurized panel
    - Local buckling behavior near cutouts

Validation of FEA by Classical Soln and Test

Case Study 3: Longitudinal Lap Joint

- Load transfer causing bearing stress
- Stress concentration factors
  - Fastener holes
  - Secondary bending

- Load transfer at the critical fastener row

- Known equations such as Swift or Huth
- FEA

- Stress concentration factor at the hole

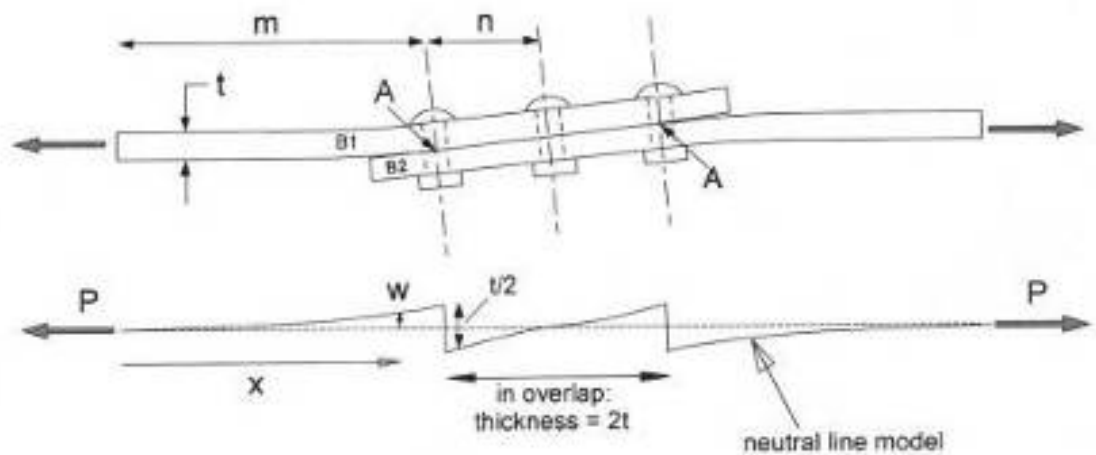
- Handbooks such as Peterson’s

Validation of FEA by Classical Soln and Test

Case Study 3: Longitudinal Lap Joint (Cont)

- Secondary Bending

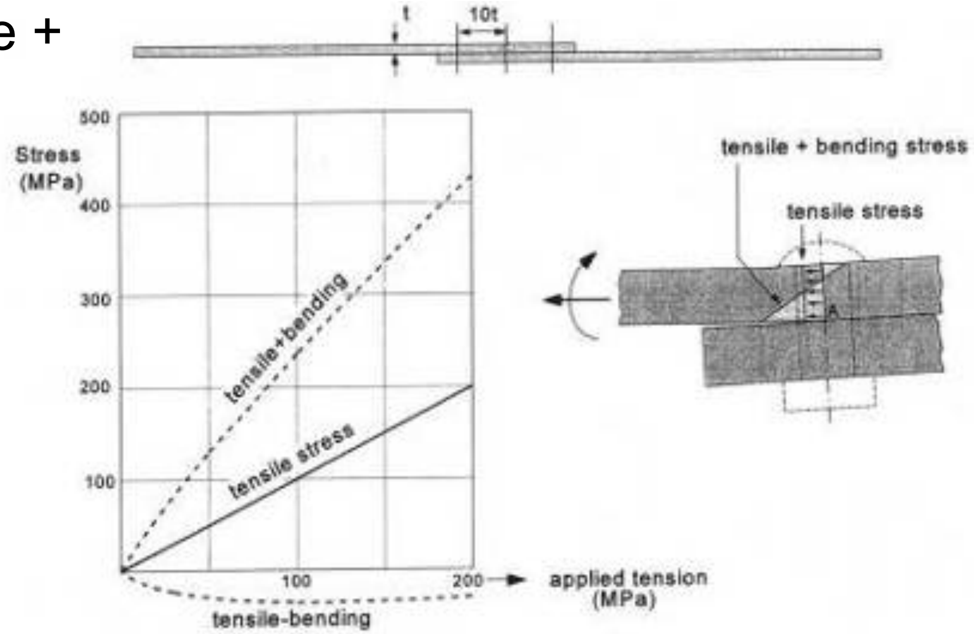
- –Caused by step in neutral line
- –Bending moment depends on
  - »Step size (eccentricity)
  - »Thickness
- –Load transfer
- –Overlap length (row distance)
- –Joint Rotation



Validation of FEA by Classical Soln and Test

Case Study 3: Longitudinal Lap Joint (Cont)

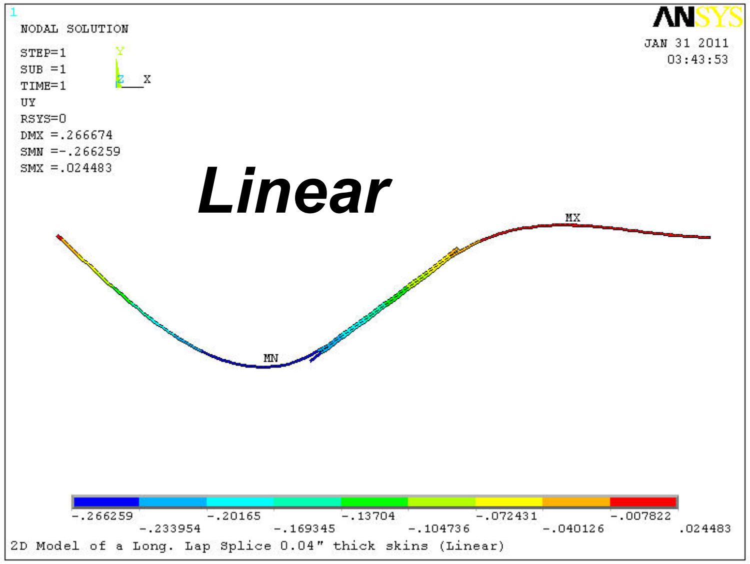
- Loads on the joint:
  1. Tensile stresses (hoop effects)
  2. Secondary bending (local effects)
     - Contact surface: Tensile + bending stress
     - Outer surface: Tensile - bending stress

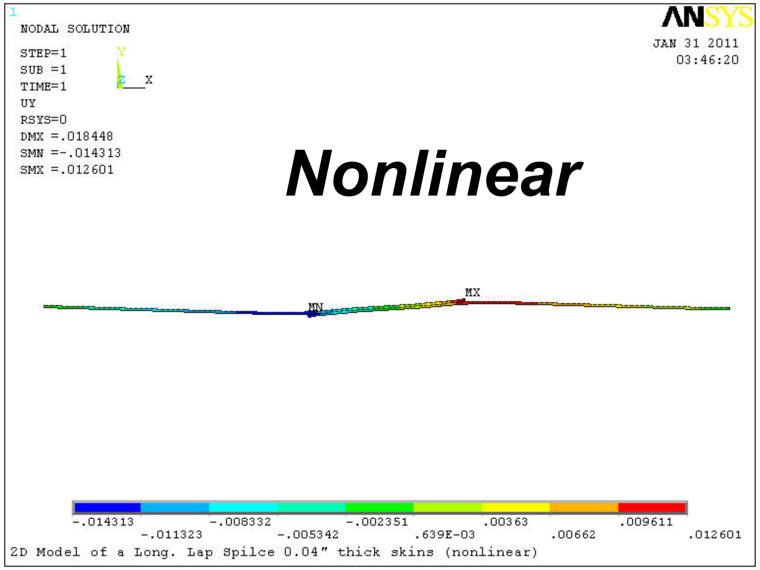


Validation of FEA by Classical Soln and Test

Case Study 3: Longitudinal Lap Joint (Cont)

- Secondary Bending
  - Know approaches by 1) Fawaz and 2) Sovar
    1. **A simplified approach for stress analysis of mechanically fastened joints**
    2. **Durability assessment of fuselage single shear lap joint with pads**
  - FEA



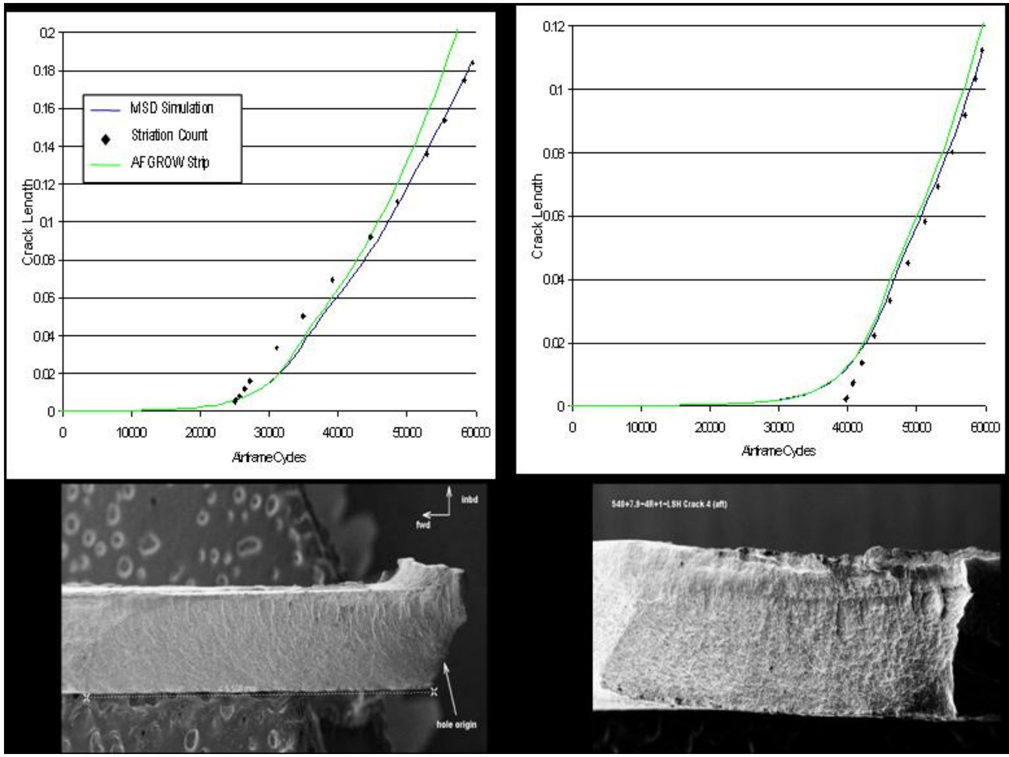


Case Study 3- Joints Eccentricity

- **To account for the combined effects of tensile, bending and bearing stress components for damage tolerance analysis use softwares such as AFGROW or NASGRO**
  - Use the computed skin tensile and bending stresses at the location of the eccentricity
  - Calculate the bearing stress due to load transfer through the critical fastener
  - Enter each of the 3 stress components in AFGROW or NASGRO to determine the crack growth life to critical length
  - Good example of Validation being a <u>process</u> not an event!

Case Study 3- Joints Eccentricity

- **Comparison of two cracks at lap joints and the respective fracture surfaces.**
  - The crack growth model using the tension, bending and bearing stresses are in a close correlation with the striation data from a fleet tear down investigation

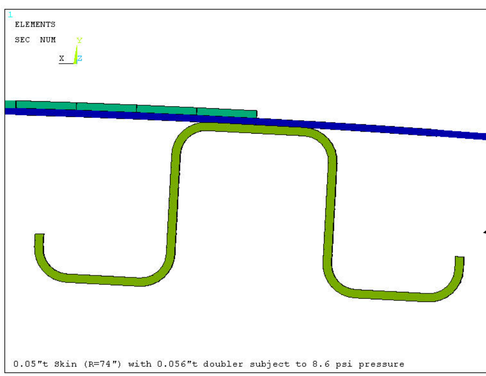


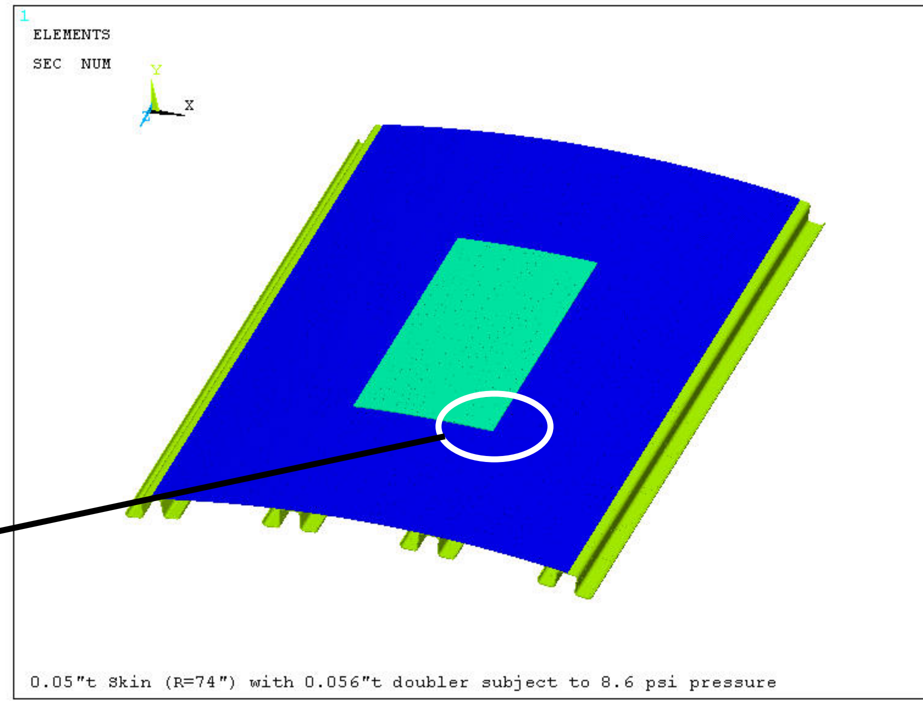
Ref: D. Steadman, R. Ramakrishnan and M. Boudreau, (2006), "Simulation of Multiple Site Damage Growth", *9th Joint FAA/DoD/NASA Aging Aircraft Conference*, Atlanta, GA., pp 12

Case Study 3- Joints Eccentricity

- **Using this approach, skin stresses around a repair doubler can be computed using FEM**

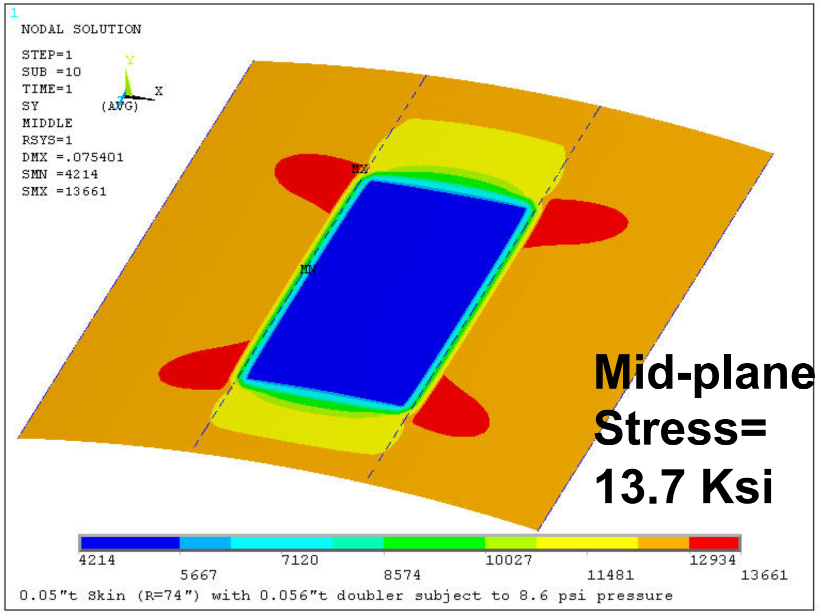
Due to excessive bending this is a nonlinear, large displacement, problem

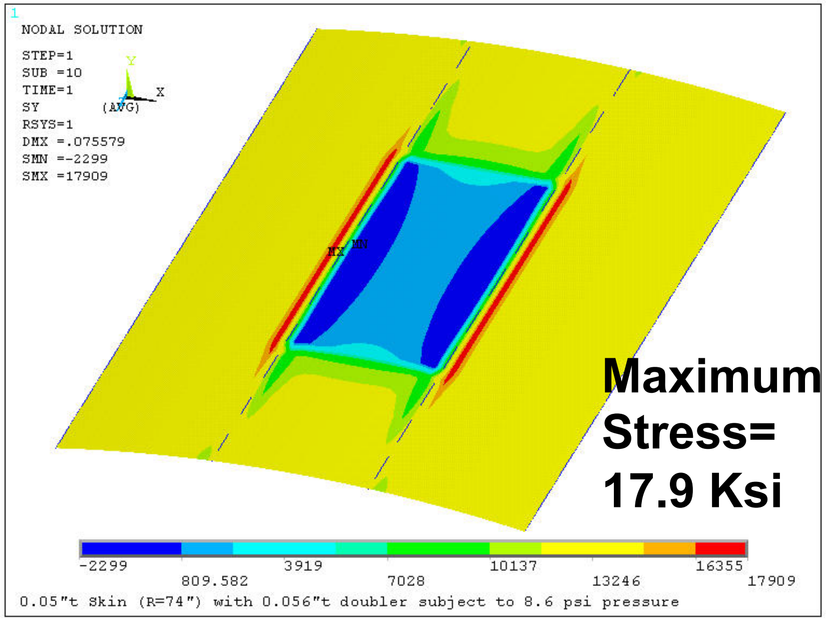




Case Study 3- Joints Eccentricity

- **Using this approach skin stresses around a repair doubler can be computed (cont.)**





Skin tensile stress (pR/t=12.7 Ksi) is increased at the edge of the doubler and bending stress is high at the skin critical location

Validation of FEA by Classical Soln and Test

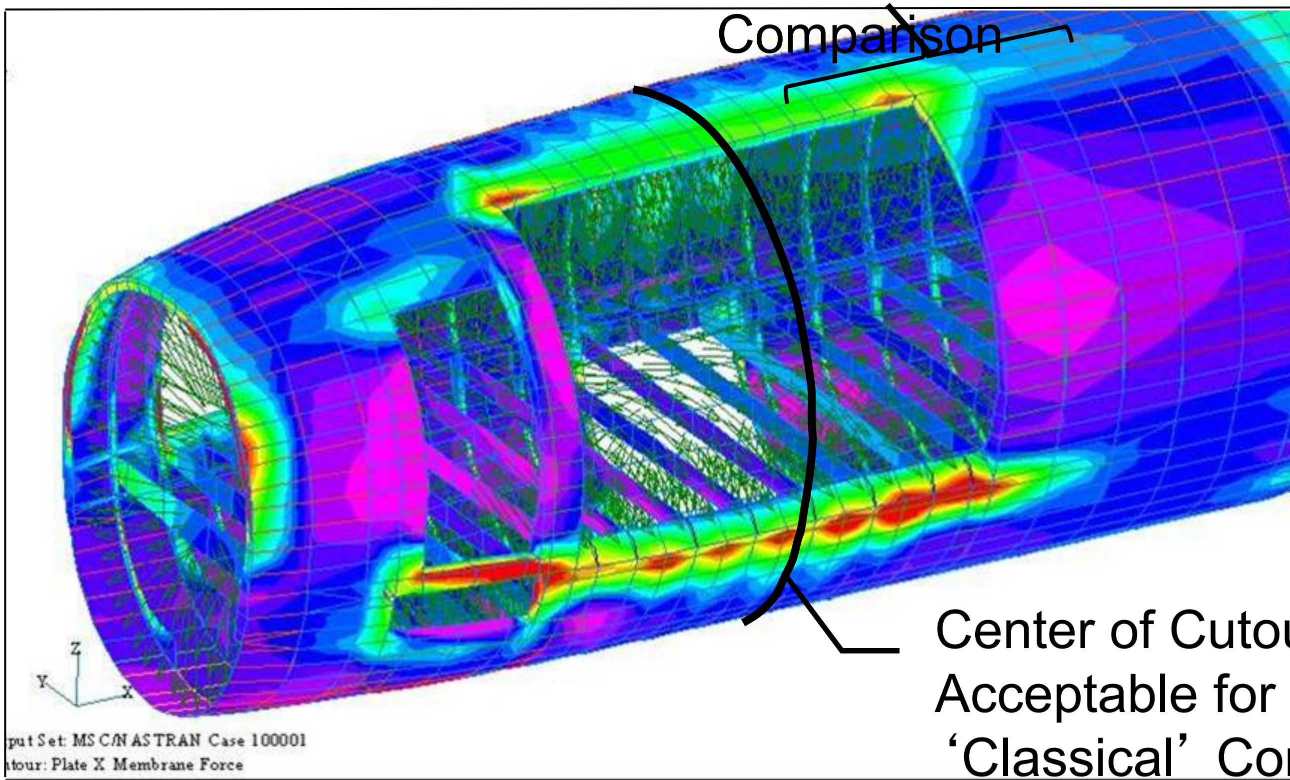
- **What did we learn from this case study?**
  - Joints are complicated structures to accurately model
  - Effects to account for are:
    - Load transfer causing bearing stress
    - Stress concentration factors
      - Fastener hole
      - Secondary bending
  - Using proper analytical tools and accounting for all the effects analytical results will be in close agreement when verified and compared with known solutions
  - Ignoring any of the effects can invalidate the results

Validation of FEA by Classical Soln

- **Considerations: FEA versus Classical Solutions**
  - Classical solutions include closed-form or other acceptable analytical methods
  - Realize the limitations of Classical methods
    - For instance bending evaluation of a fuselage cross sections assumes linear strain distribution
  - Load application to a FE model or actual structure may have differences
- **Therefore ensure that the comparison between FEA and Classical Solution reflects equivalent conditions.**

Validation of FEA by Classical Soln

- **Case Study 1: Fuselage with large cutouts**



Considerable Load Gradients, Not Suitable for ‘Classical’ Comparison

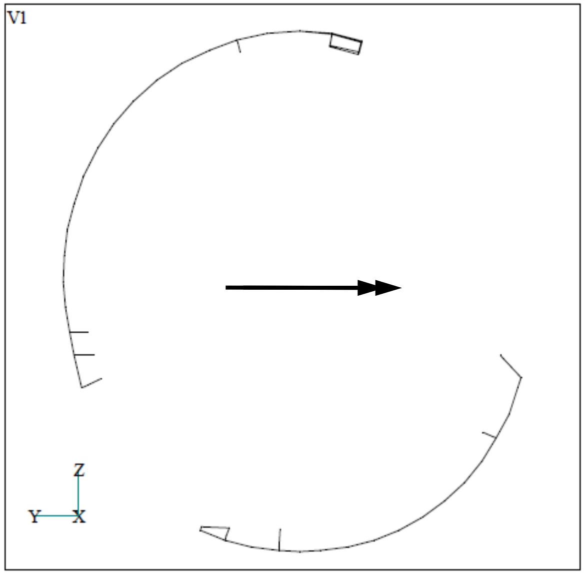
Center of Cutout Acceptable for ‘Classical’ Comparison

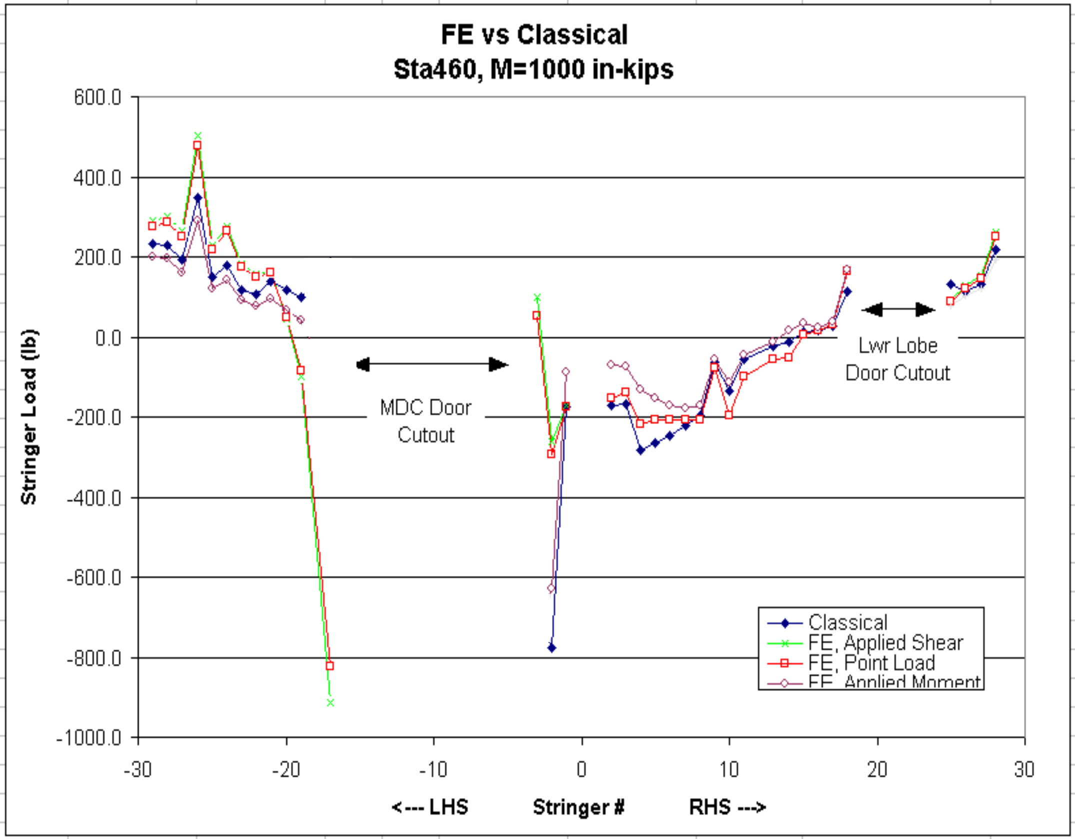
Validation of FEA by Classical Soln

- **FEA versus Classical Solutions- Fuselage with large cutouts (cont.)**
  - Classical analysis assumed pure moment at a cross-section.
  - Best comparison was achieved when FE model was loaded with an applied moment, thereby eliminating secondary shear and moment effects encountered using other loading methods.

Validation of FEA by Classical Soln

- FEA versus Classical Soln- Fuselage with large cutouts (cont.)





Validation of FEA by Classical Soln

- **What did we learn from this case study?**
  - The model/loading may need to be simplified or tailored to match the classical solutions
  - Once the model has been validated, it may be shown how it could be used for other load cases
    - There are definite limitations that must be addressed prior to this application
  - If the classical solutions were sufficient for the real model/loading why is there need for FEA?
    - See case studies 2 and 3

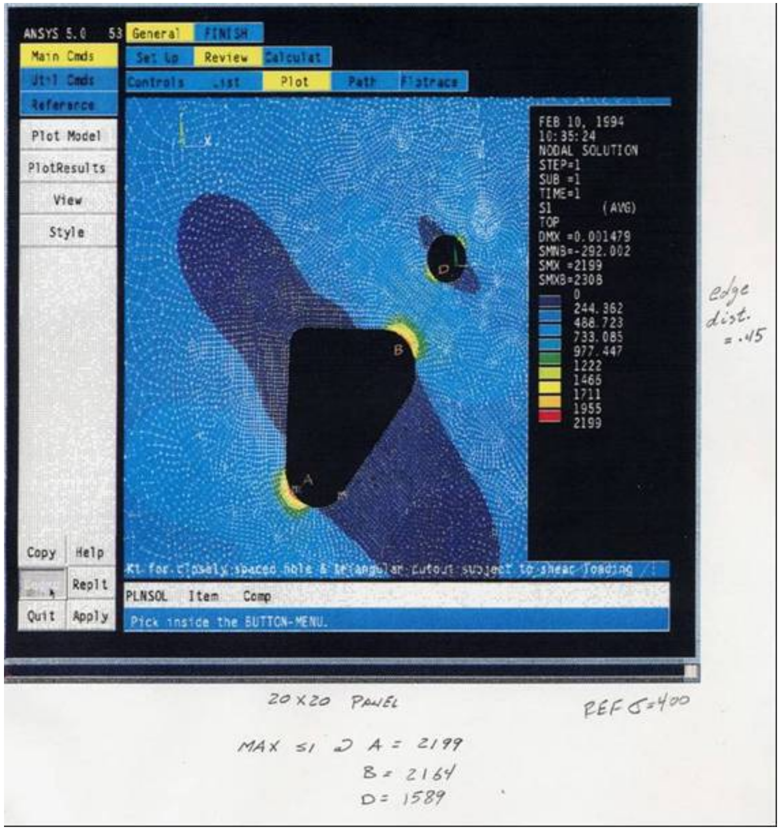
Validation of FEA by Classical Soln

- **Case Study 2 - FEA versus handbook solutions- Stress Concentration Factor**

SCF for the open hole is compared and shown to match Peterson's Handbook, so the SCF for the triangular cutout can reliably be computed and use for any analysis, in this case fatigue.

Hole K_t = 4 **(Same as Peterson's)**

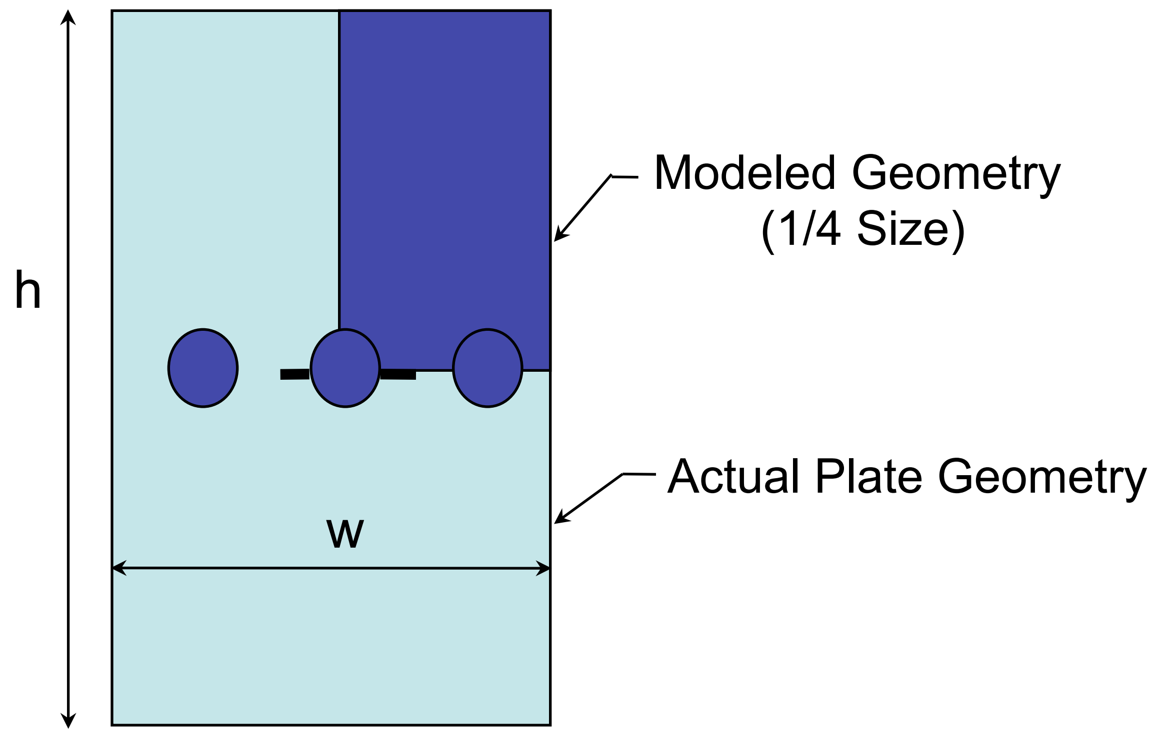
Tri cutout K_t = 5.5



Validation of FEA by Classical Soln

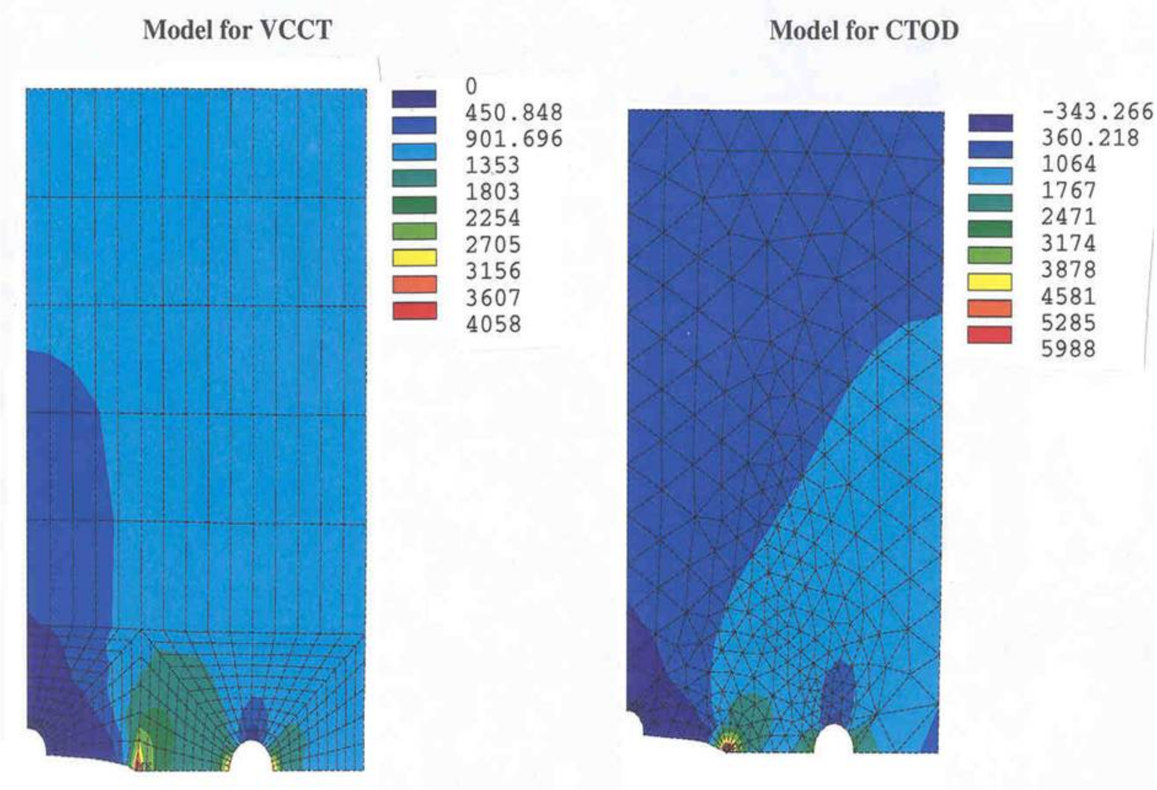
- **Case Study 3 - FEA versus acceptable numerical methods- Stress Intensity Factor**

Finite plate with collinear holes with crack in the center hole



Validation of FEA by Classical Soln

- **Case Study 3- FEA versus acceptable numerical methods- Stress Intensity Factor**
  - Two techniques are used to compute K
    1. Virtual Crack Closure Technique
    2. Crack Tip Opening Displacement

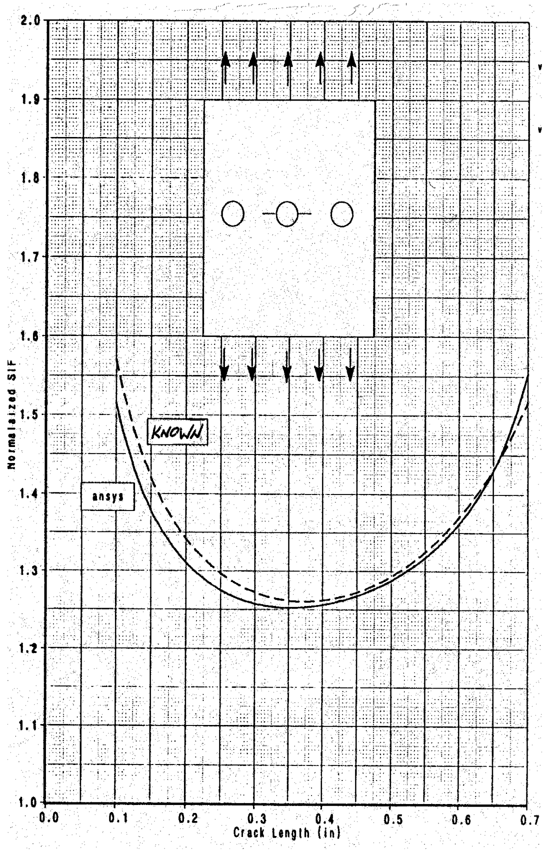


Element types and meshing techniques may be different for the VCCT and CTOD

Validation of FEA by Classical Soln

- **Case Study 3- FEA versus acceptable numerical methods- Stress Intensity Factor**

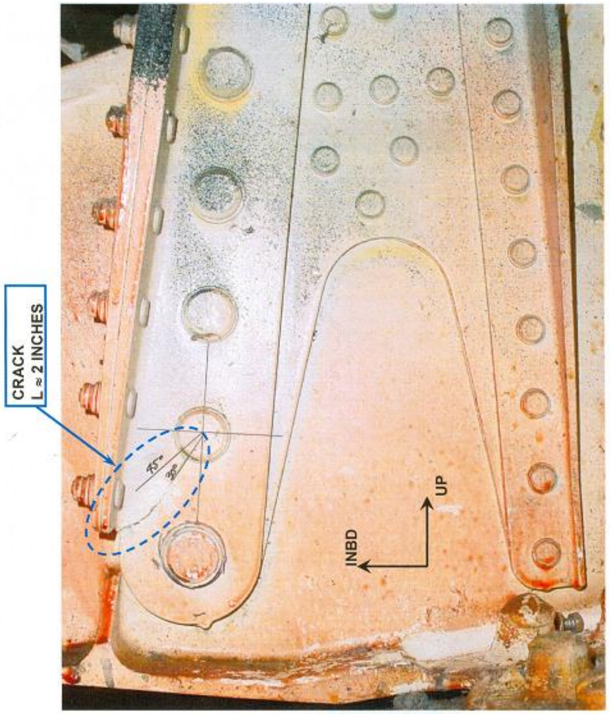
Close correlation with the acceptable numerical results

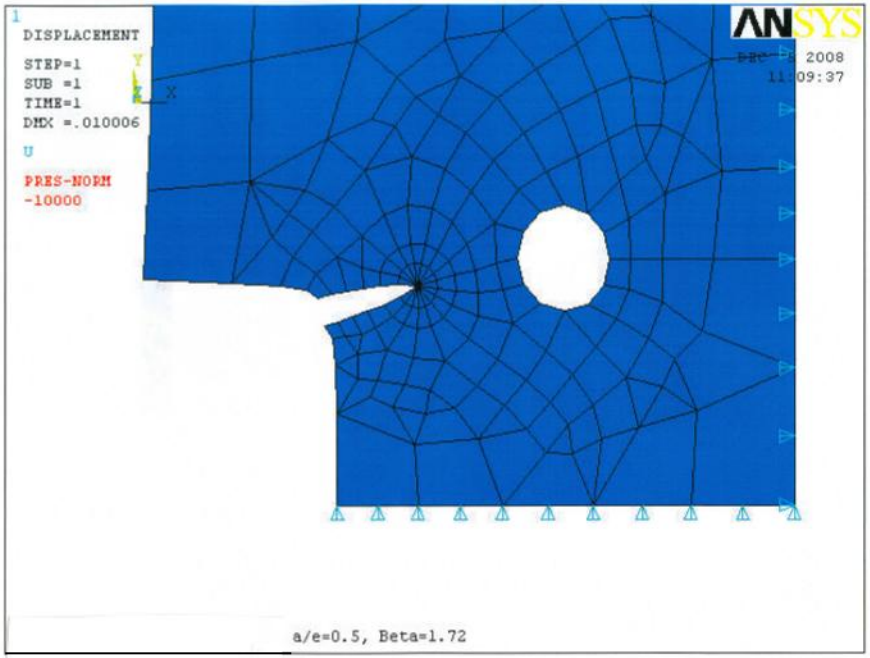


Validation of FEA by Classical Soln

Complex/Detail- Stress Intensity Factor

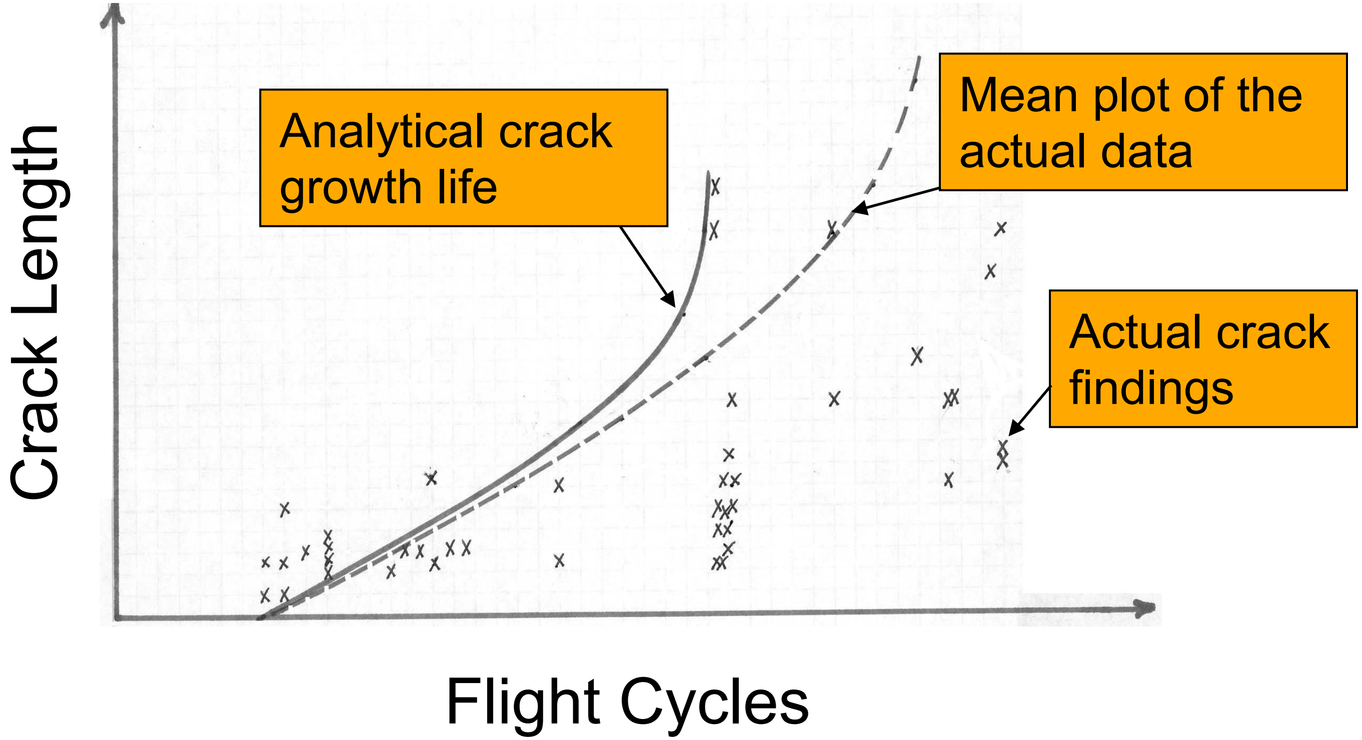
- Investigation of fatigue crack growth in a complex structure- Stress intensity factor computation using FEA





Case Study 3- Failure Analysis

- **FEA versus Classical Soln- Damage Tolerance**
  – Same technique is used to computed SIF and subsequently calculation of crack growth life- *Slide 29 geometry*



Summary - Finite Element Modeling and Analysis Validation

- In addition to good engineering practices, FAA FARs, Order and issue paper require and discuss validation requirements
- FEA is an efficient analytical tool that can be used extensively after validation, substantiation and verification of the approaches and the results
- FEA results must be documented and controlled
- Validation of FEA begins in the onset of the modeling
- Validation of FEA must be a part of Certification Plan
- Discussion of Means of Validation
  - Case studies

Check List - Finite Element Modeling and Analysis Validation

- **Is the FE model correct?**
  - Right geometry, material representations, stiffness, elements, loads, BCs
- **Is the FE model sufficiently accurate?**
  - Mesh size, type, nonlinear effects
- **Does the FE model check out?**
  - Results are as expected- Free Body Diagram
  - Displacements, reactions, stresses
- **Is the FE model in agreement with the test data or classical/known solutions?**
- **Etc…**

Acknowledgments

- **The following companies have provided extra valuable input for this presentation:**
  - Aerospace Consulting Engineering Corp.
  - The Boeing Company
  - DTA Technologies
  - TASS Inc.
  - Structural Integrity Engineering
  - Univ. of Wash. – Aeronautics & Astronautics Depart.
  - Wagner Aeronautical Inc.
- **Several individuals within branches of ACO's in the FAA provided great comments and suggestions for this presentation.**

Appendix I
Early Validation of FEA by *Correct* Element Choices

Early Validation of FEA by *Correct* Element Choices

- **Since the understanding of the structural behavior and the element formulation in FEA is <u>essential</u> the analyst must be intimately familiar with:**
  1) Stress Analysis: Fundamentals such as free body diagram, static and dynamic behaviors and strength of material
  2) Finite Element Analysis: Element formulation, assumptions, capabilities, choice of solution, limitations and restrictions of software
- **Accuracy of an FEA solution is dictated by a combination of:**
  – Correct element shape & order, element shape functions, element capabilities and mesh density

A Simple Illustration

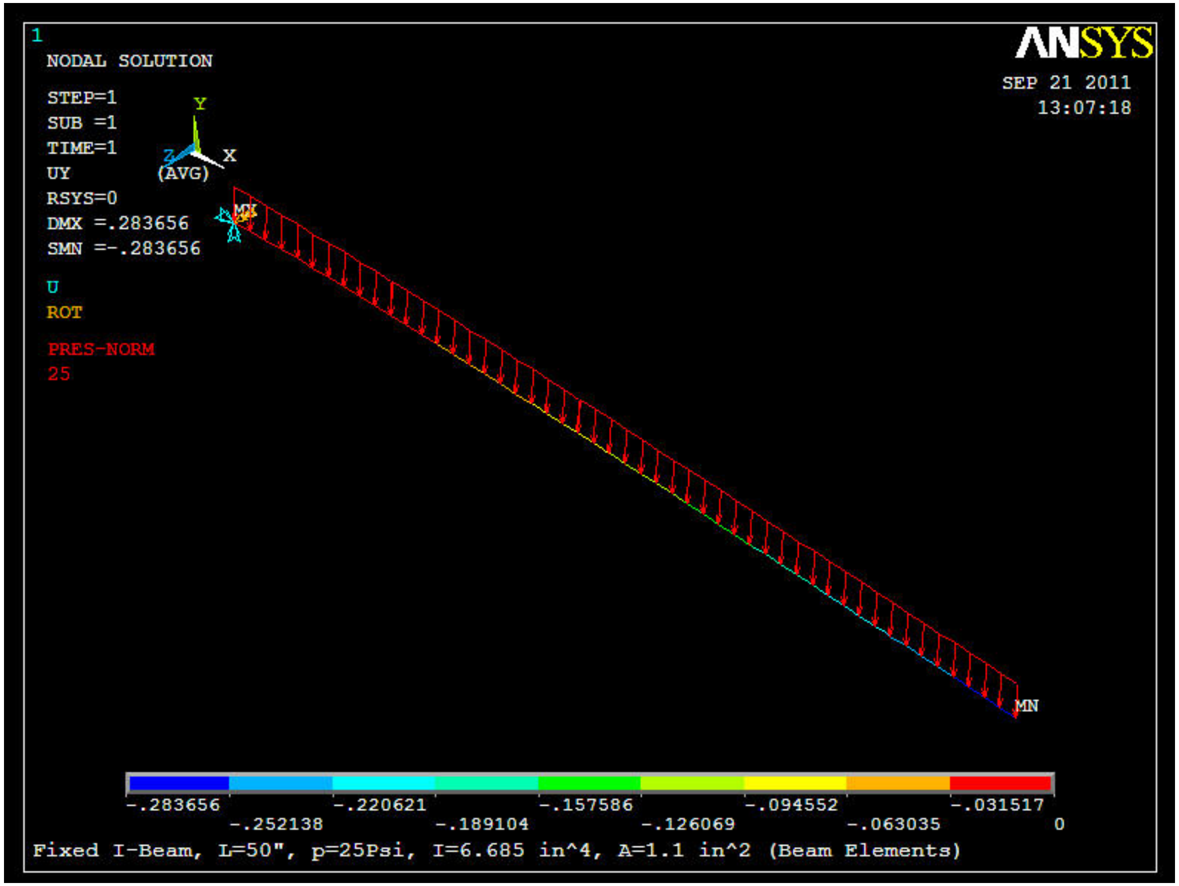
Example - 50" Long Cantilever I-Beam

– 25 Lb/in distributed load

- **The Cantilever Beam is idealized as**
  - *Beam Elements (Classical and Section)*
  - *Rod-Plate-Rod Elements*
  - *Plate Elements*
- **All three idealizations are acceptable**
- **Theoretical solution:**
  - *Vertical Deflection= 0.2837"*
  - *Maximum Stress= 15.193 Ksi*

A Simple Illustration
Beam Elements

Deflection Plot:



Deflection = 0.2837”

A Simple Illustration
Beam Elements

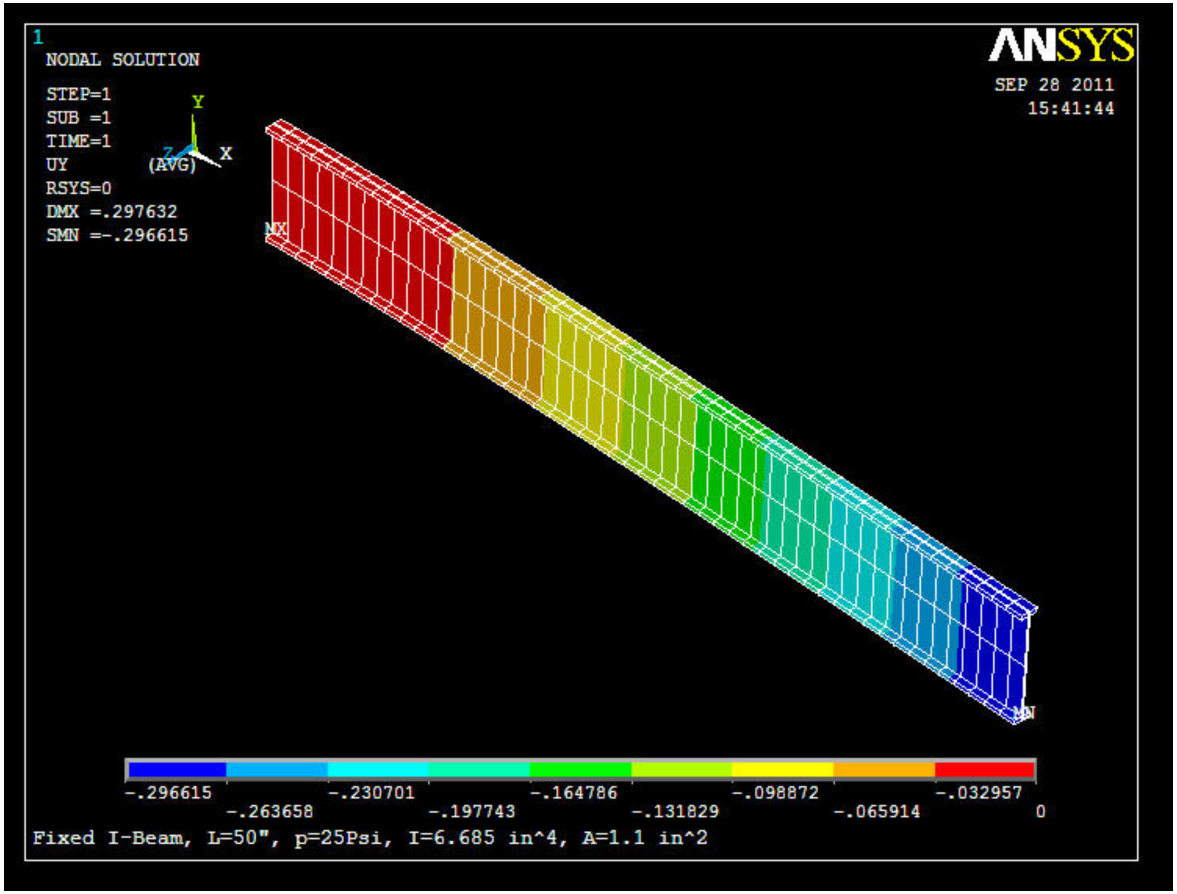
Stress Output:

```
PRINT ELEM ELEMENT SOLUTION PER ELEMENT

***** POST1 ELEMENT SOLUTION LISTING *****
  LOAD STEP     1  SUBSTEP=     1
  TIME=    1.0000         LOAD CASE=  0

 EL=       1  NODES=       1       3  MAT=  1
                         BEAM3
PRES LOAD KEY = 1  FACE NODES =    1    3
                PRESSURES(F/L) =   25.000      25.000
 LOCATION   SDIR         SBYT        SBYB
        1 (I)     0.0000       15193.     -15193.
        2 (J)     0.0000       14591.     -14591.
 LOCATION   SMAX         SMIN
        1 (I)     15193.     -15193.
        2 (J)     14591.     -14591.
```

S_{Max}=15.193 Ksi

A Simple Illustration
Beam Elements

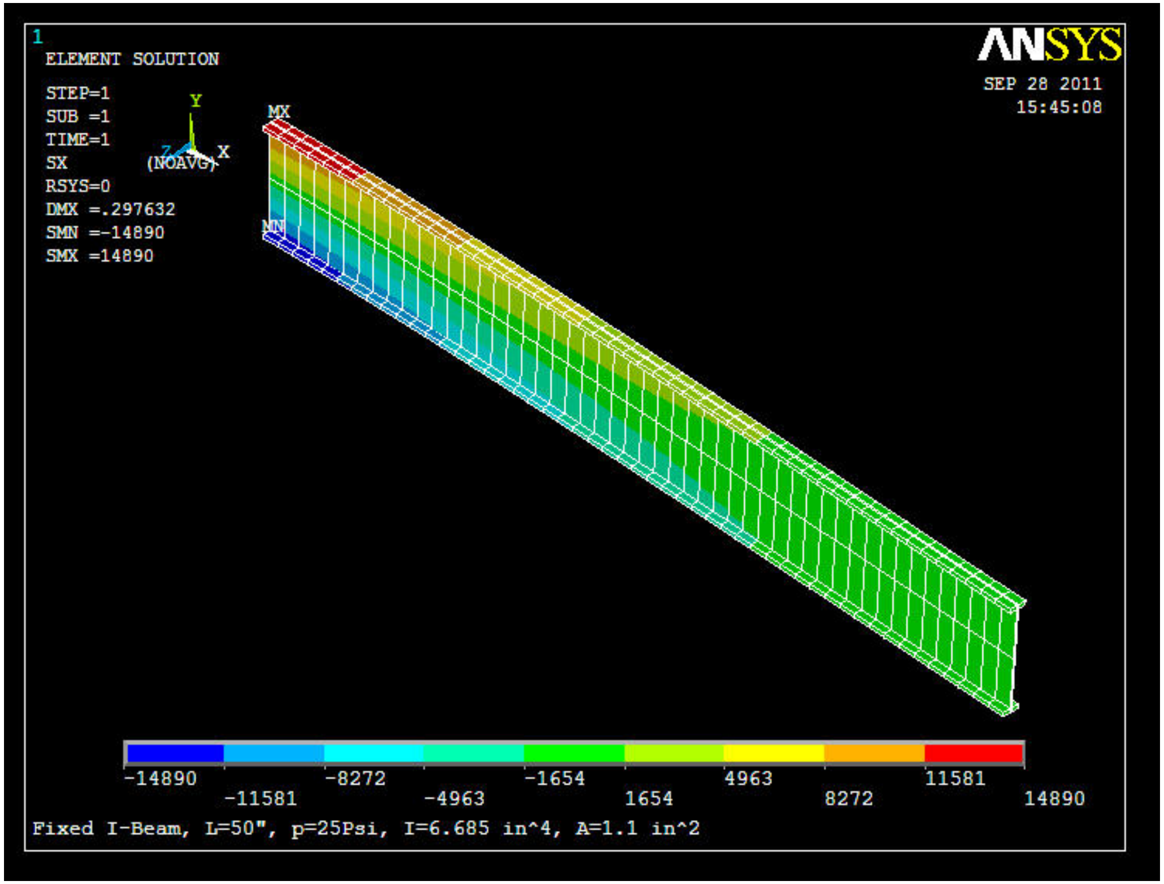
Deflection Plot:



Deflection = 0.2966”

A Simple Illustration
Beam Elements

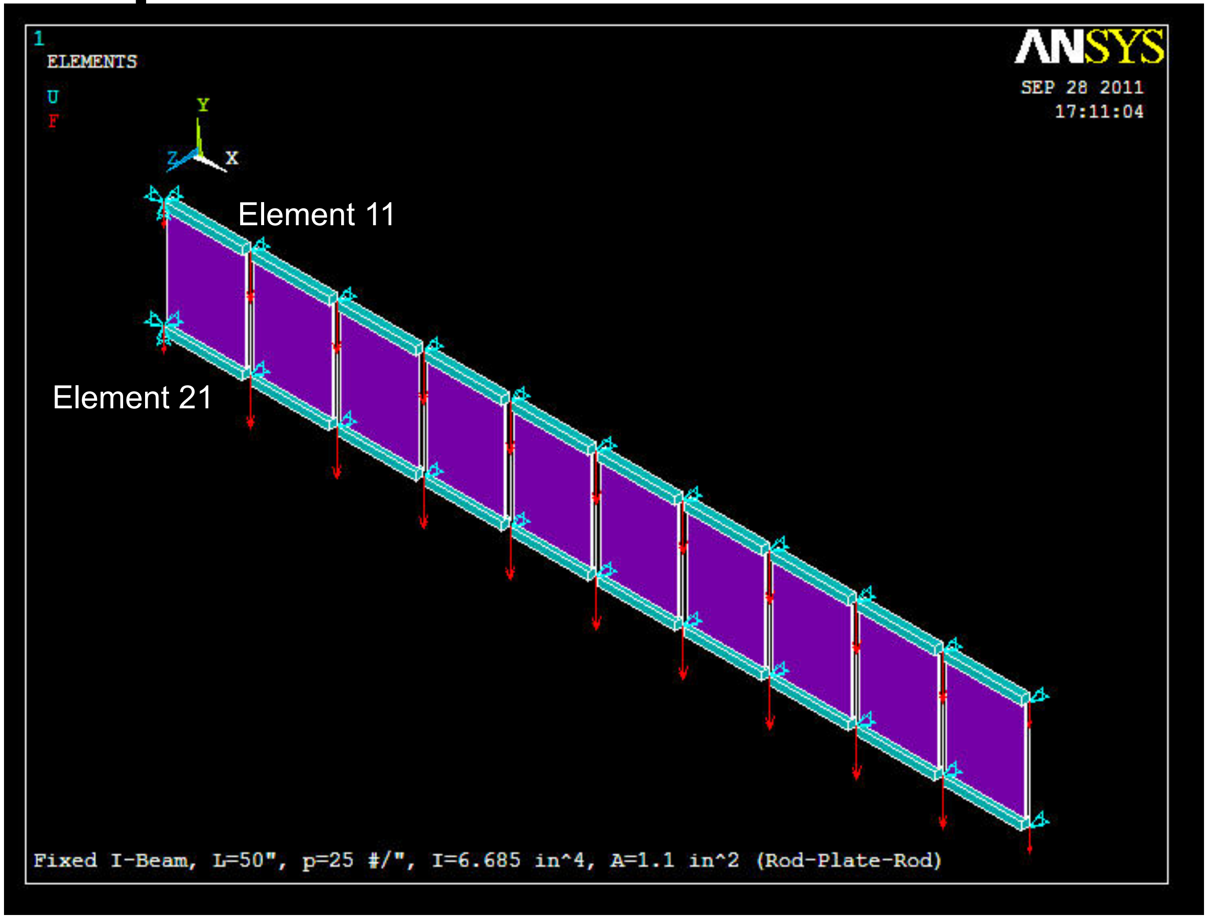
Stress Output:



S_{Max}=14.890 Ksi

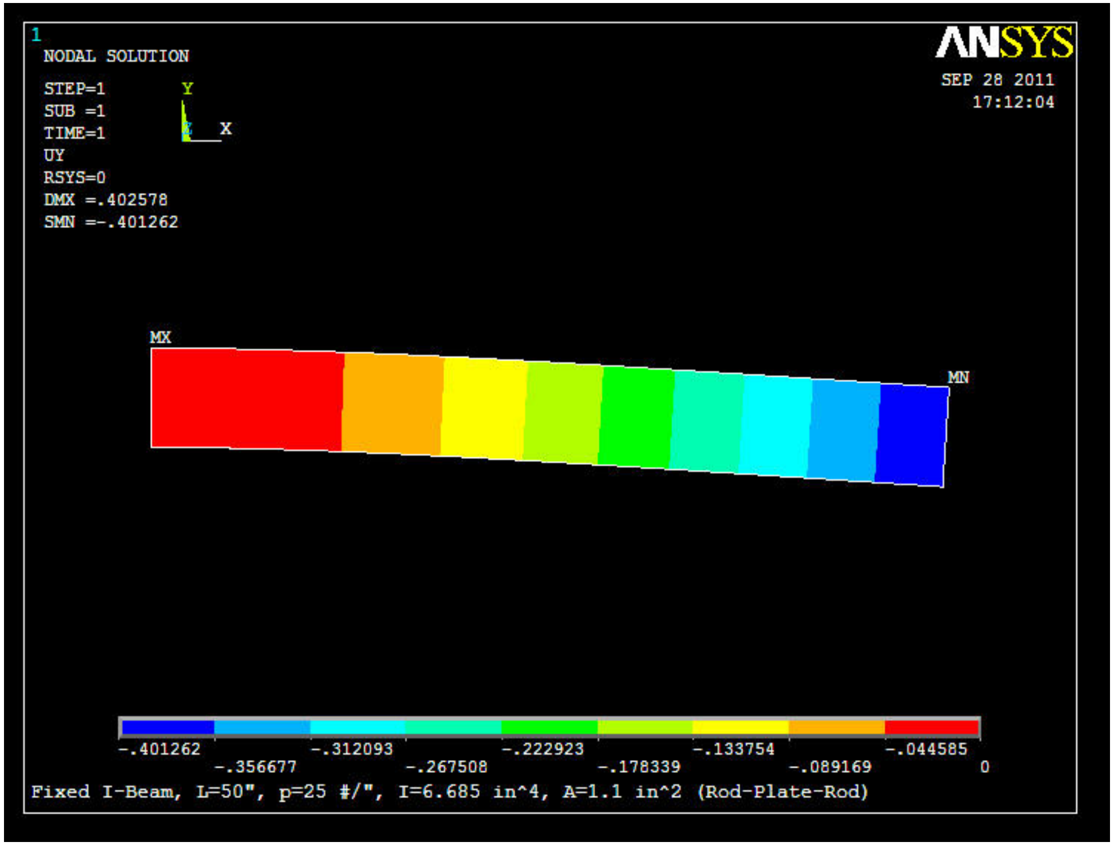
A Simple Illustration
Rod-Plate-Rod Elements

Element plot of the I-Beam



A Simple Illustration
Rod-Plate-Rod Elements

Deflection Plot:



Deflection = 0.4013”

A Simple Illustration
Rod-Plate-Rod Elements

Stress Output:

PRINT ELEMENT TABLE ITEMS PER ELEMENT

***** POST1 ELEMENT TABLE LISTING *****

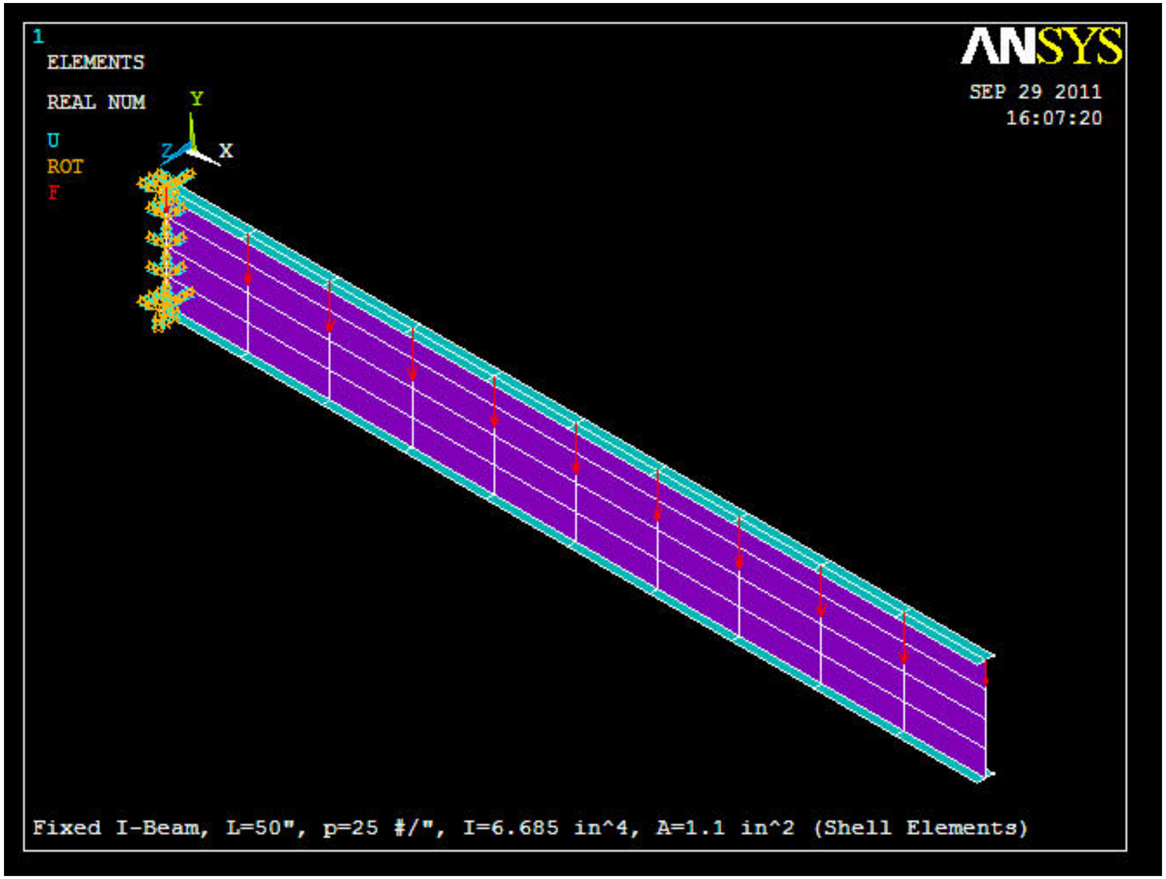
S_{Max}=18.100 Ksi

STAT	MIXED	CURRENT
ELEM	Stress	Force
11	18100.	4525.0
21	-18100.	-4525.0

This solution is very dependant on how the beam is idealizes. In this case the upr & lwr chord are modeled by 0.25 in^2 bars and the web is modeled with shear elements. No axial load is assumed to be carried by the web.

A Simple Illustration
Shell Elements

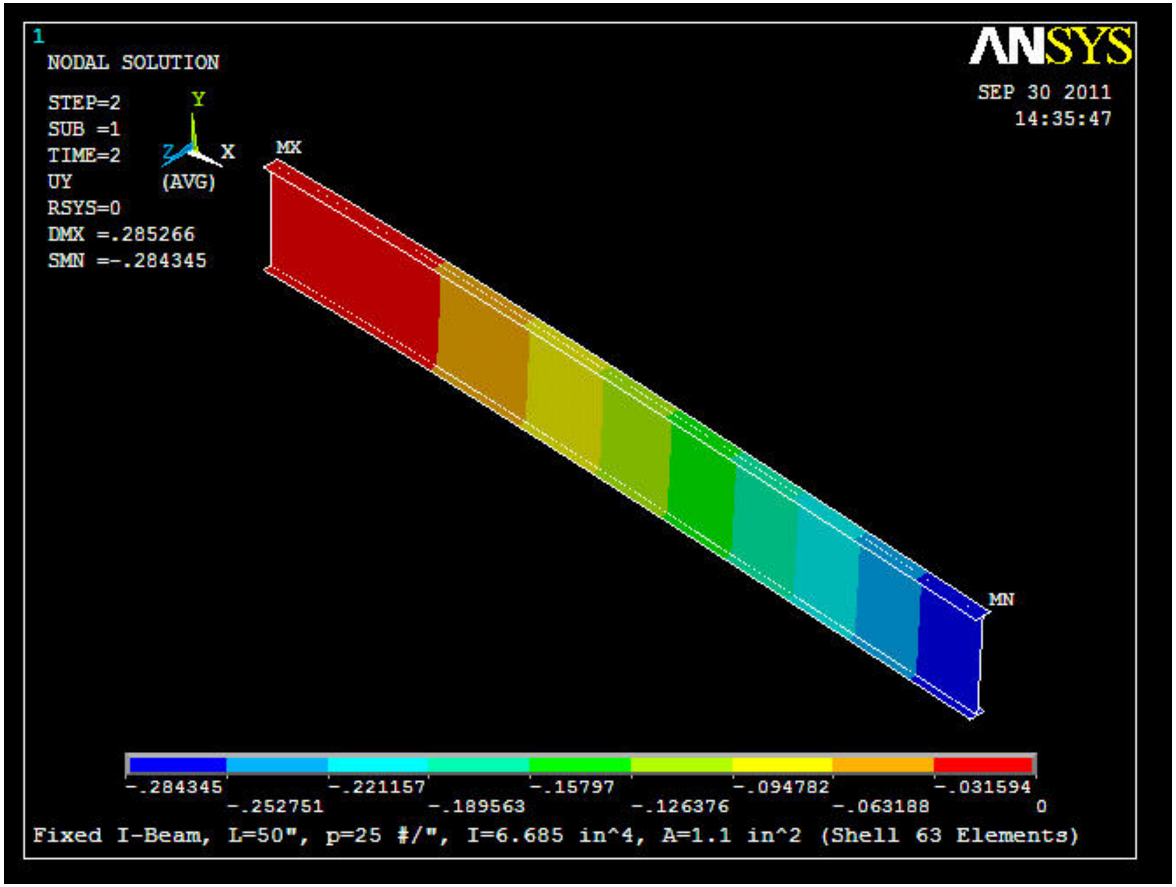
Element plot of the I-Beam



Shell element used is based on *Kirchhoff* plate theory that is suitable for modeling thin plates

A Simple Illustration
Shell Elements

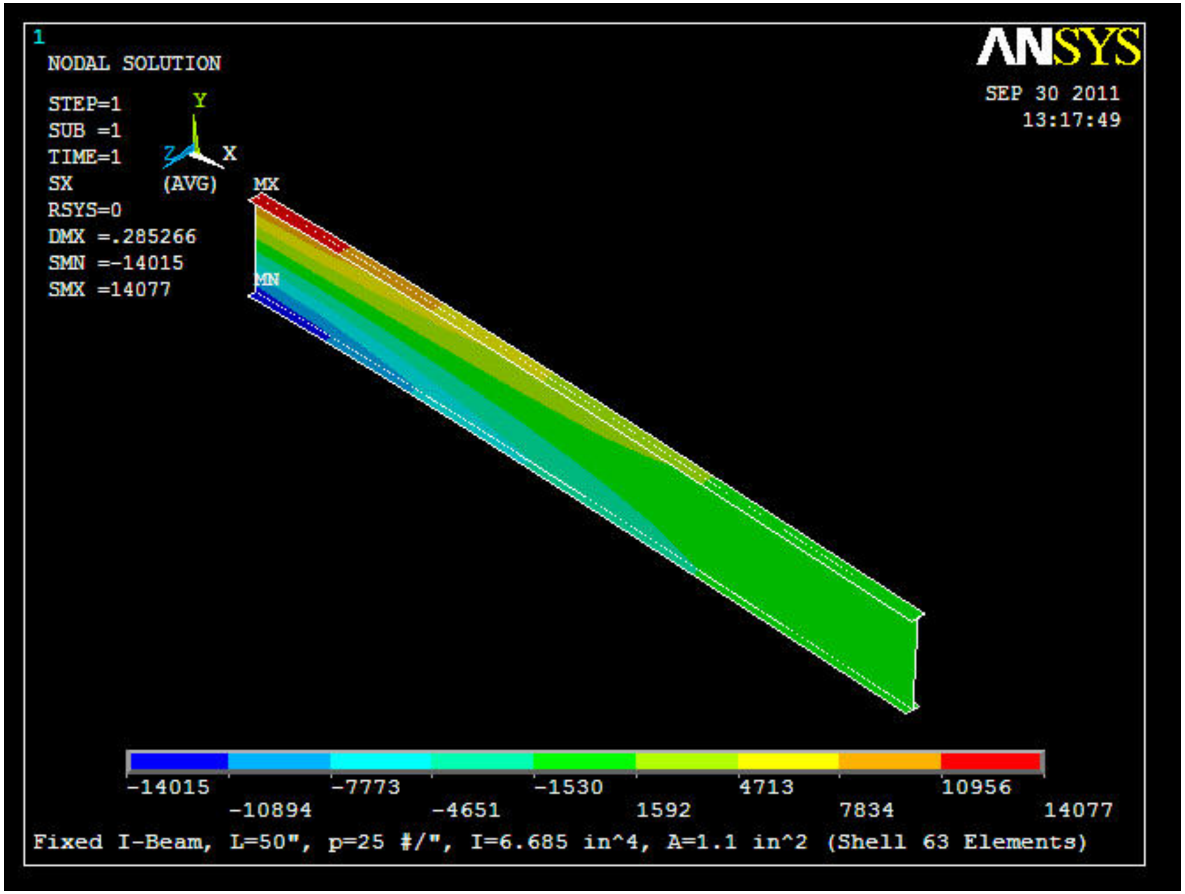
Deflection Plot:



Deflection = 0.2843”

A Simple Illustration
Shell Elements

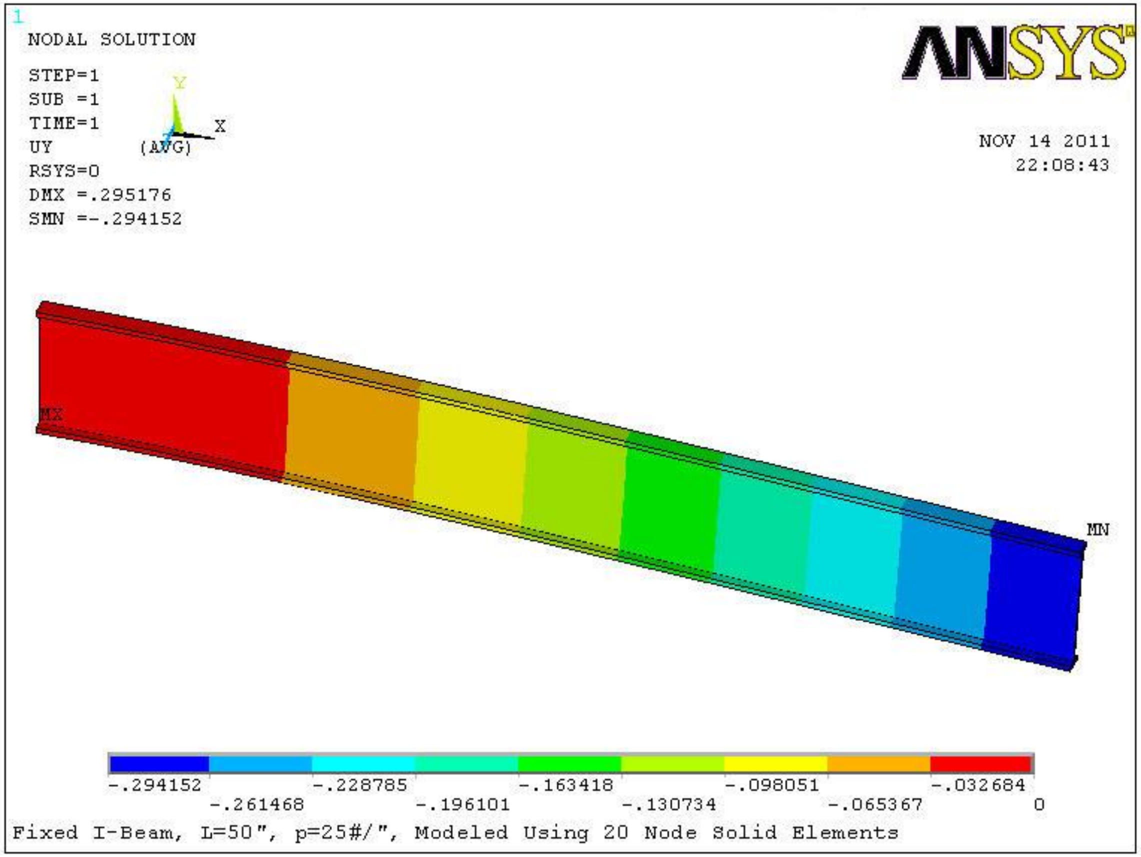
Stress Output:



S_{Max}=14.077 Ksi

A Simple Illustration 3D-Solid Elements

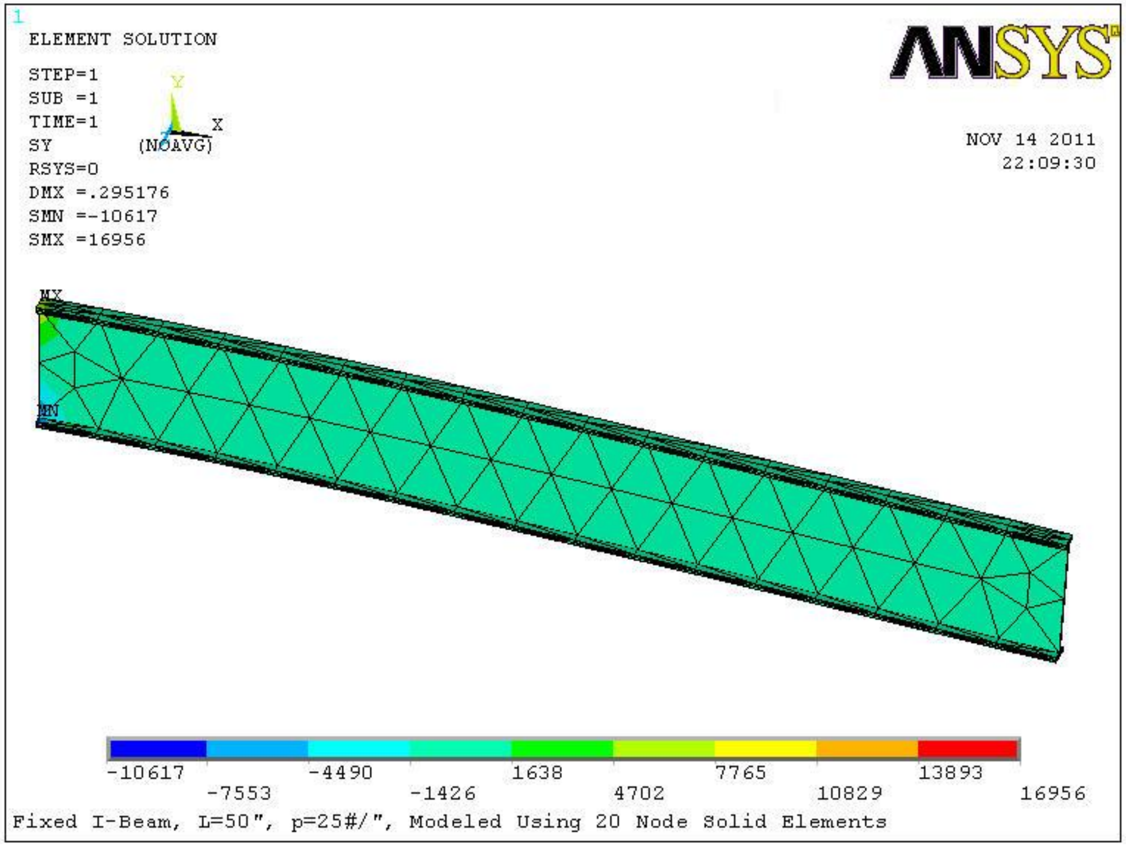
Deflection Plot:



Deflection = 0.2942”

A Simple Illustration
3D-Solid Elements

Stress Output:



s_{Max}=16.96 Ksi

A Simple Illustration
Choice of Elements- Comparison

- **Results comparison**
  - Beam element – 15.19 Ksi or 14.89 Ksi
    - 0% or 2% difference
  - Rod-plate-rod elements - 18.10 Ksi
    - +19% difference
  - Plate element – 14.08 Ksi
    - -7% difference
  - Solid element – 16.96 Ksi
    - +12% difference
- **Clearly choice of elements make a difference in the accuracy of the results!**

Early Validation of FEA by *Correct* Element Choices - Shape Function

- **In bending of a thick plate where the transverse shear effect is not negligible certain shell elements do not have extra shape functions to account for this phenomena. Thus they will produce erroneous results.**
  - Use shell elements that have the extra shape function
  - Use solid elements (Only way for very thick plates subject to bending)
- **Let us consider a thick plate with a hole in the center, subject to pure bending**
- **Compare results to *Stress Concentration Factors* by R.E. Peterson**

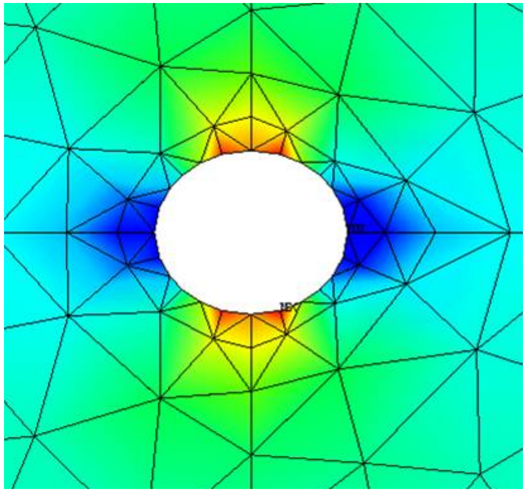
Early Validation of FEA by *Correct* Element Choices - Shape Function

- **Comparison of K_t for a plate with a hole subject to pure bending using two types of shell and solid elements**

Plate Geometry			Kt	Elements without shape Functions	Elements with shape functions	Solid Elements
D	t	W				
1	0.03	8	1.65	1.615	1.595	N/A
1	0.05	8	1.65	1.615	1.605	N/A
1	0.1	8	1.65	1.615	1.605	N/A
1	1	8	2.02	1.615	2.07	2.11
1	2	8	2.23	1.615	2.31	2.3
1	4	8	2.43	1.615	2.53	2.57
			Peterson's handbook	Kirchhoff plate theory	Mindlin plate theory	

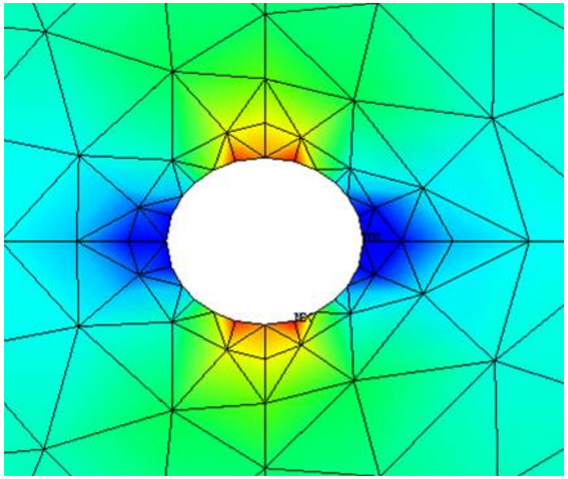
Early Validation of FEA by *Correct* Element Choices – Shape & Order

- **Elements: Shape & Order - Example**
  - Flat plate with a center hole subject to axial load
    - Theoretical value = 3000 Psi



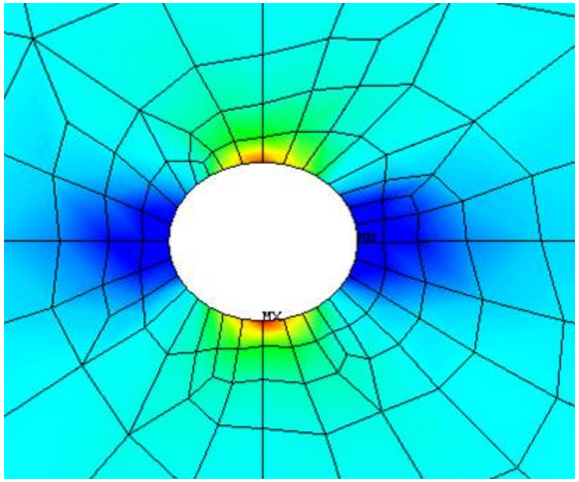
3-node triangular solid elements

s_{peak}= 2247 Psi



6-node triangular solid elements

s_{peak}= 2986 Psi

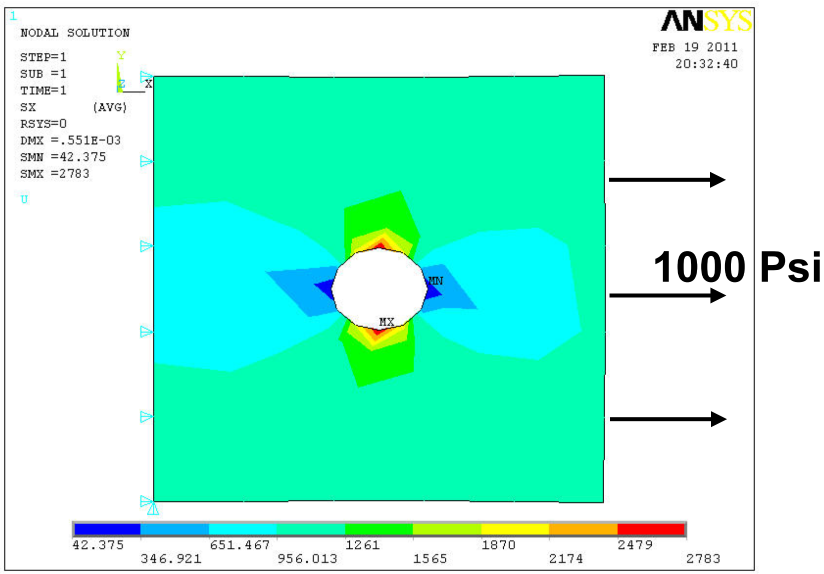


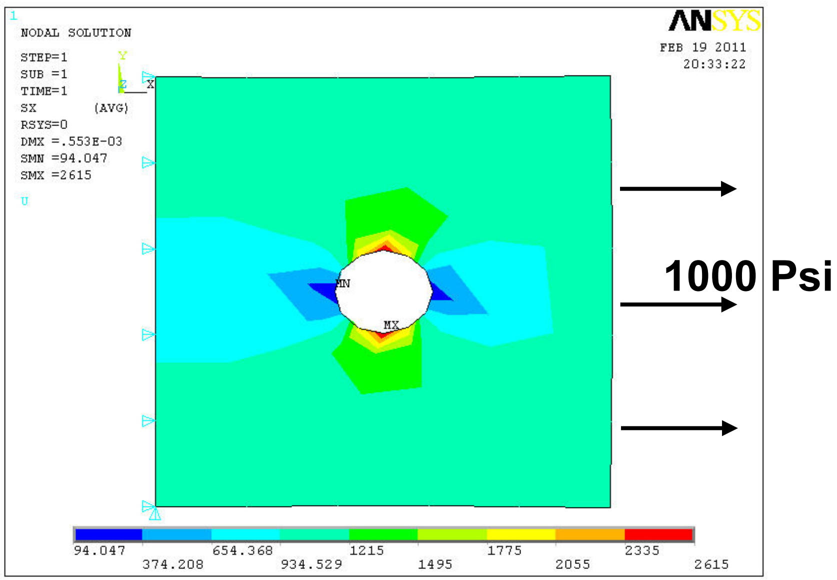
4-node solid quad elements

s_{peak}= 2940 Psi

Early Validation of FEA by *Correct* Element Choices - Shape Function

- **Linear elements with *constant direct strain* in their formulations**
  - Theoretical value of 3,000 Psi is obtained by sufficient mesh density





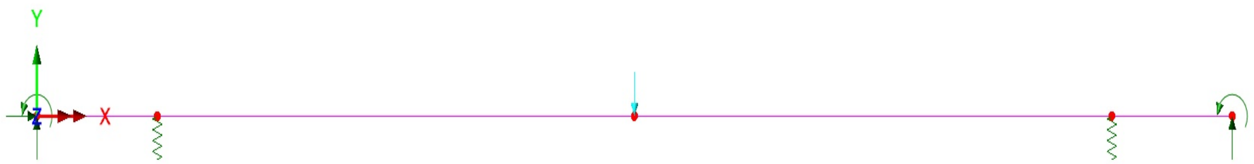
s_x=2,783 Psi (constant strain) s_x=2,615 Psi (enhanced strain)

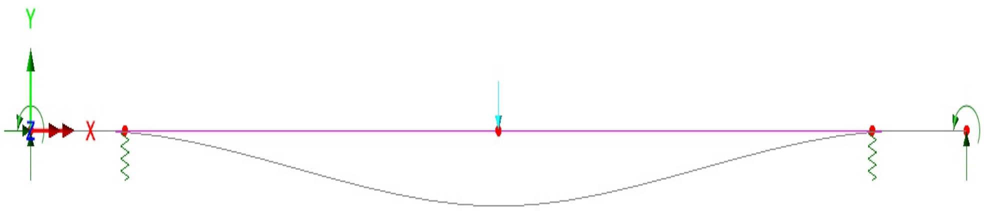
Early Validation of FEA by *Correct* Element Choices : Element Capability

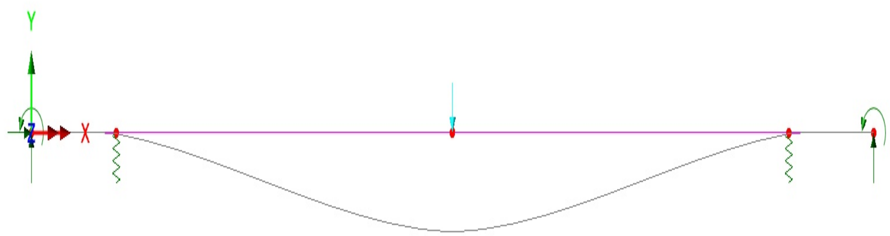
- **Thin beam element vs. Shear beam element:**
  - Example of Floor Beam representation of a 747 and the key results to note are that:
    - The deformations are quite different, particularly in the section between the stanchion and the frame because of the short distance.
    - The shear force is off by 570 # (1170 # for Thin beam, 600 # for Shear beam) or about 95% off. This changes the load estimated in the stanchion.
    - The moment is ~6% off, in this case driven by the large moments created at the built-in ends.
      - Softening the end restraints may affect this error.

Early Validation of FEA by *Correct* Element Choices : Thin vs. Shear Beam Elements

Deflection Plot







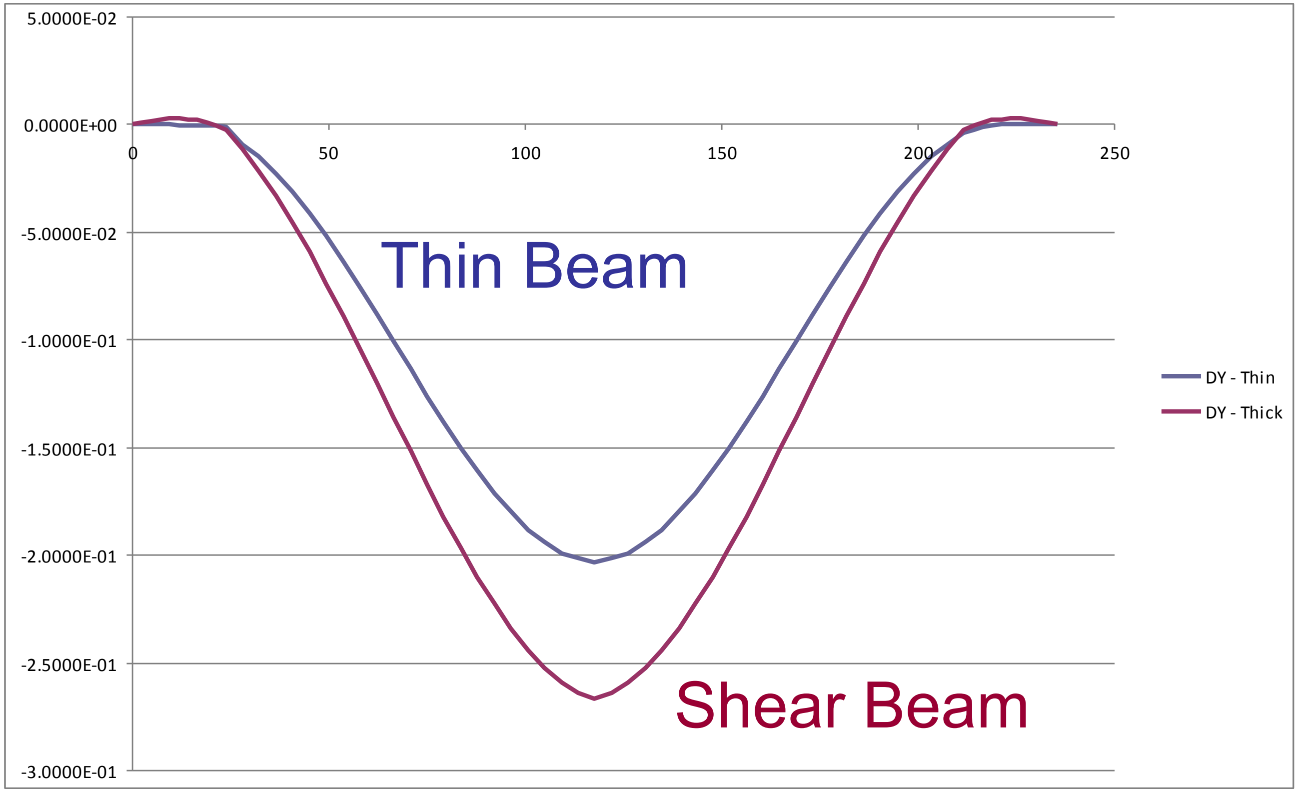
Thin Beam: U_y=0.203
(Incorrect)

Shear Beam: U_y=0.267
(Correct)

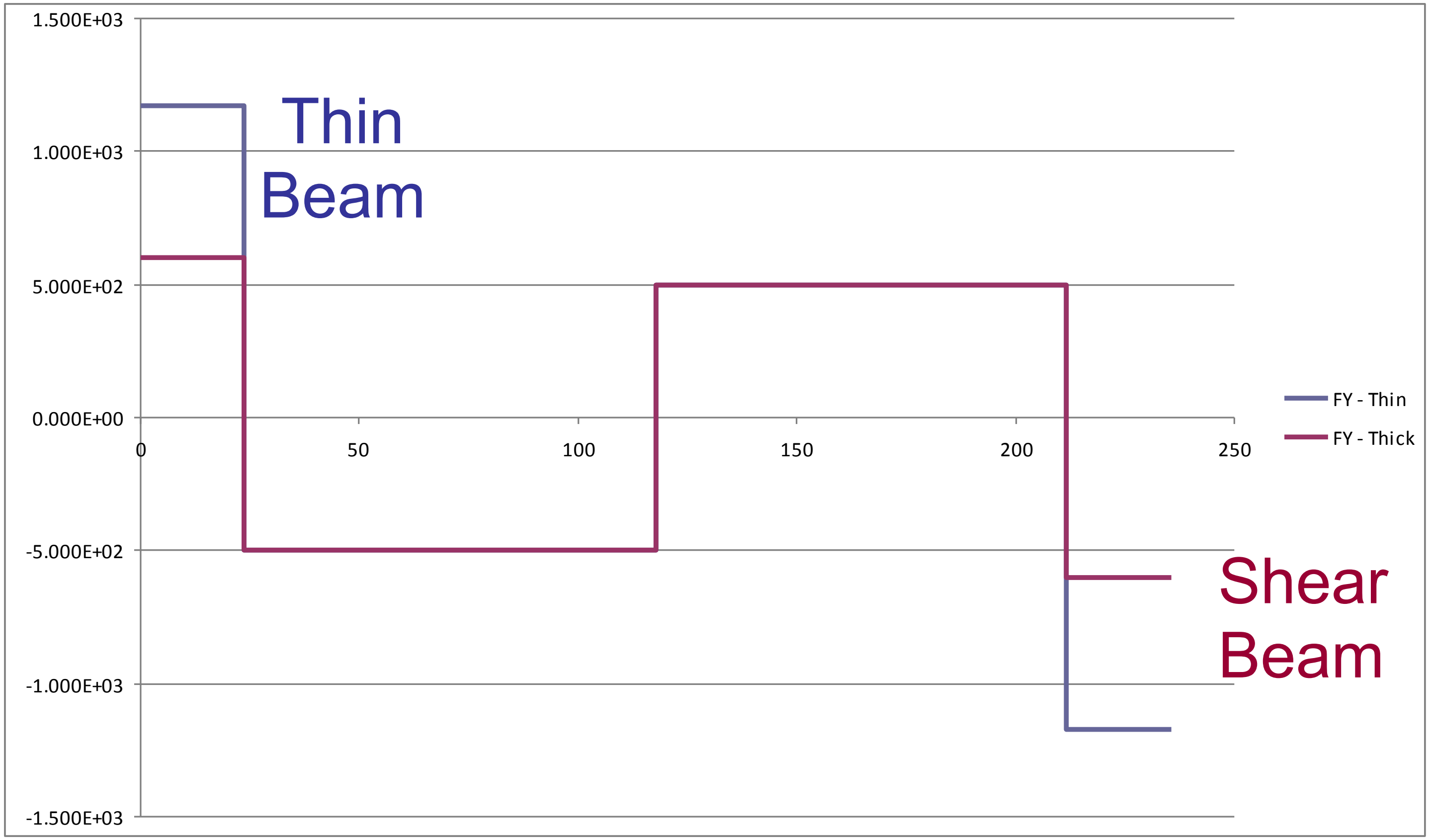
Early Validation of FEA by *Correct* Element Choices: Thin vs. Shear Beam Elements

Deflection Diagram

Note the different deflections between the frame and the stanchion

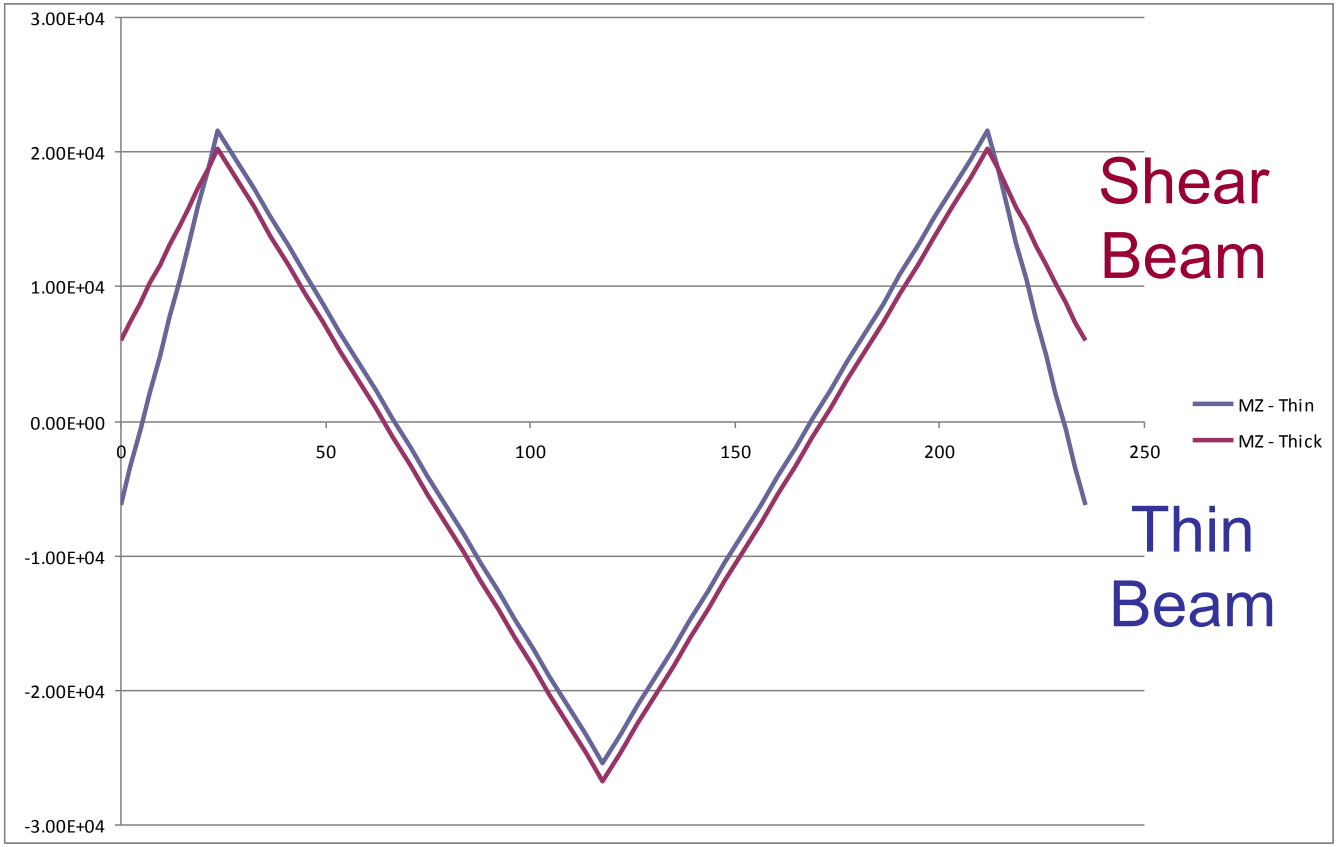


Early Validation of FEA by *Correct* Element Choices : Thin vs. Shear Beam Elements
Shear Force Diagram



Early Validation of FEA by *Correct* Element Choices : Thin vs. Shear Beam Elements

<u>Moment Diagram</u>



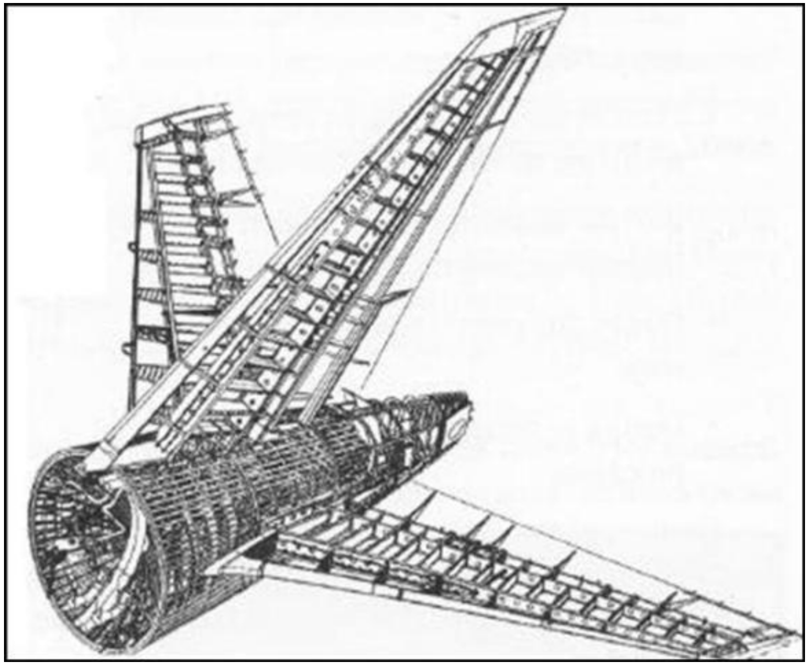
Appendix II
Building Block Approach For Composite Material Structures FEA

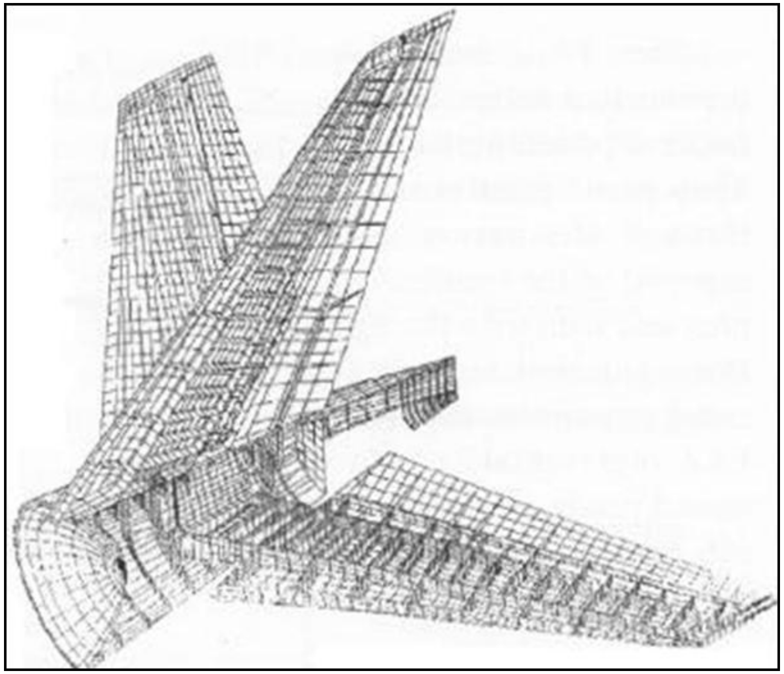
Appendix II

- **Building Block Approach for simulating composite material structures**
  1. Boeing 777 Empennage Certification Approach
  2. Certification by analysis supported by test evidence for the design of the door-sill of the new Lamborghini supercar, called Aventador

777 Empennage Certification Approach
by Fawcett, Trostle and Ward

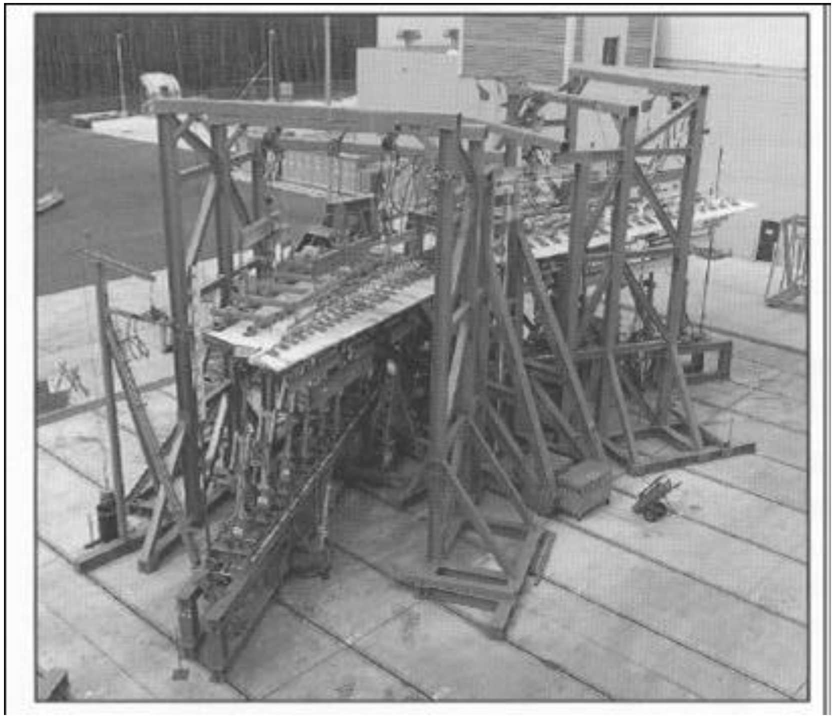
- 777-200 Empennage geometry and FEM



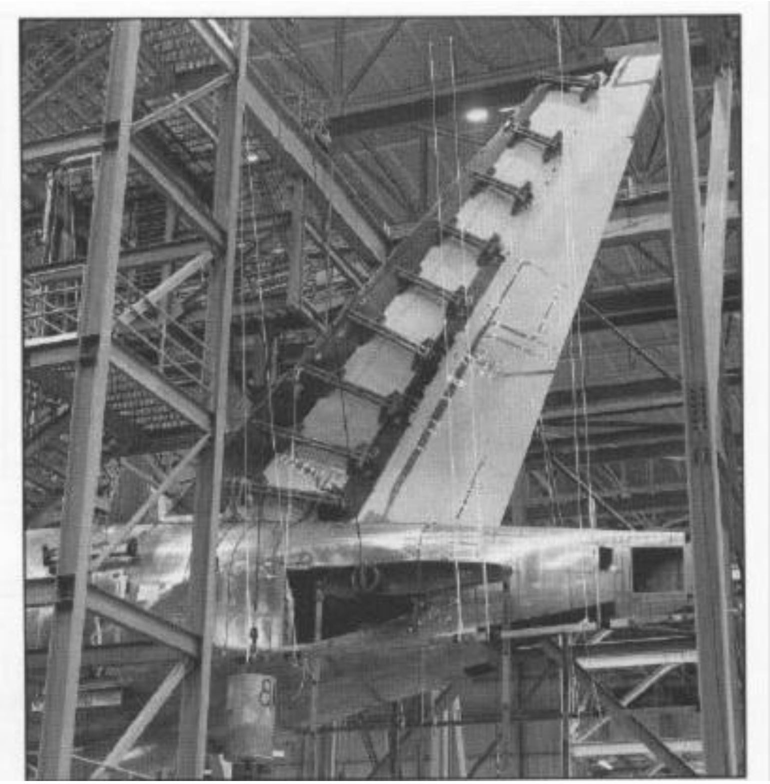


777 Empennage Certification Approach
by Fawcett, Trostle and Ward

- Test Setup



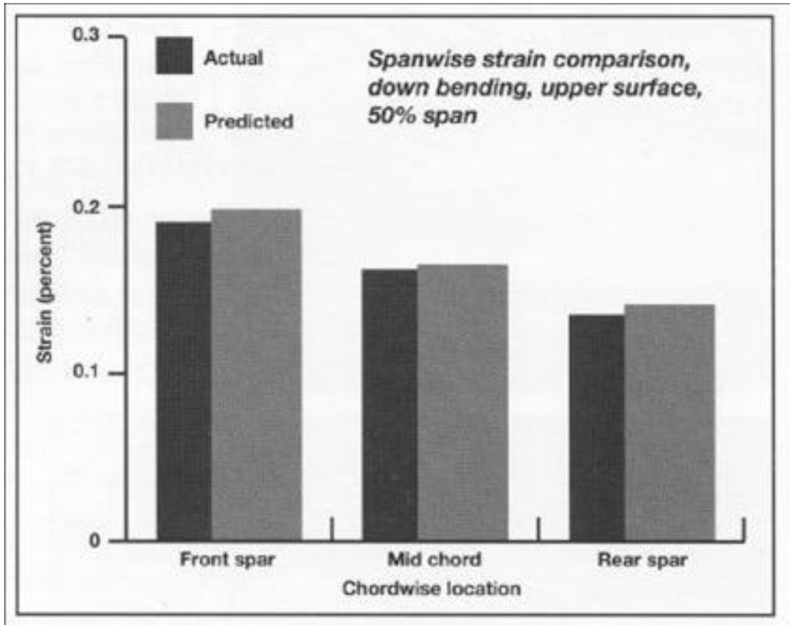
Horizontal Stabilizer

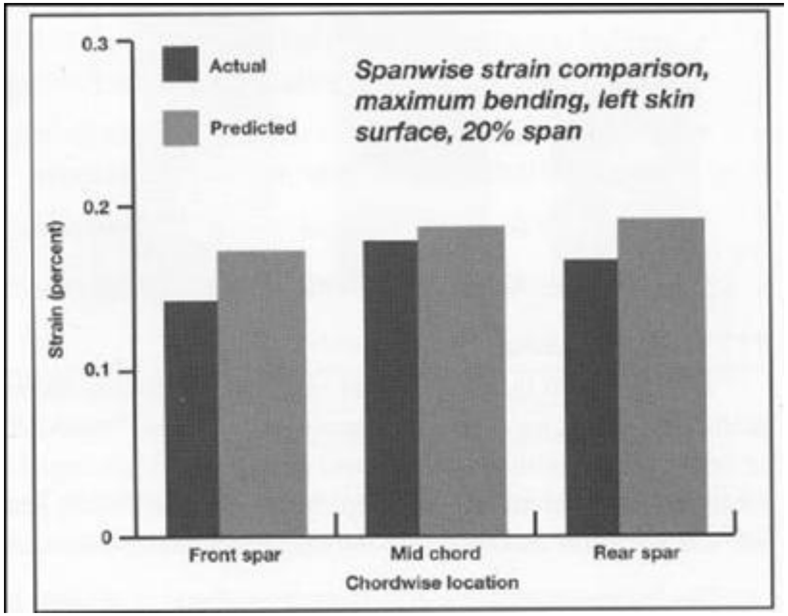


Vertical Stabilizer

777 Empennage Certification Approach
by Fawcett, Trostle and Ward

- Stabilizer Test- Predicted vs. Actual Strains



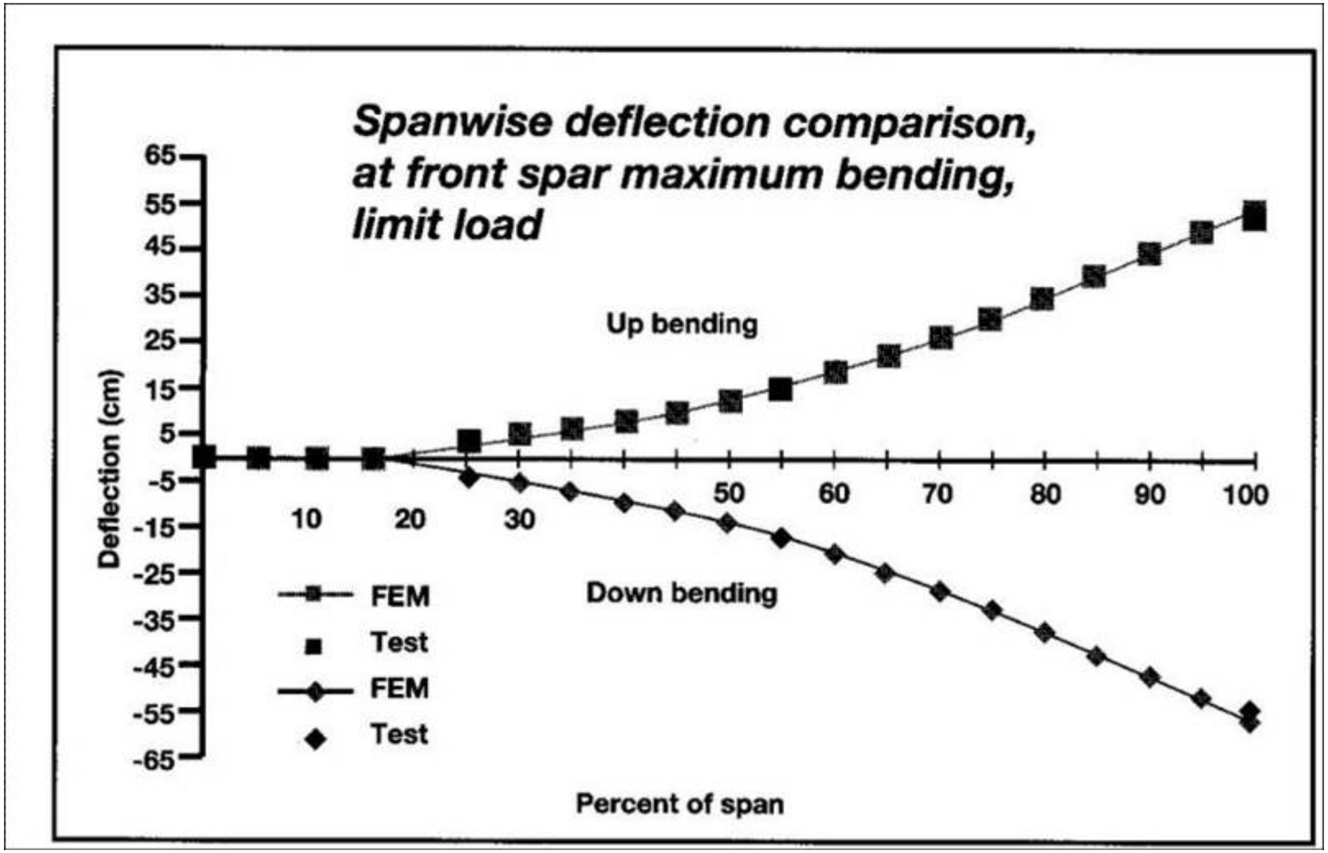


<u>Horizontal</u> Stabilizer

<u>Vertical</u> Stabilizer

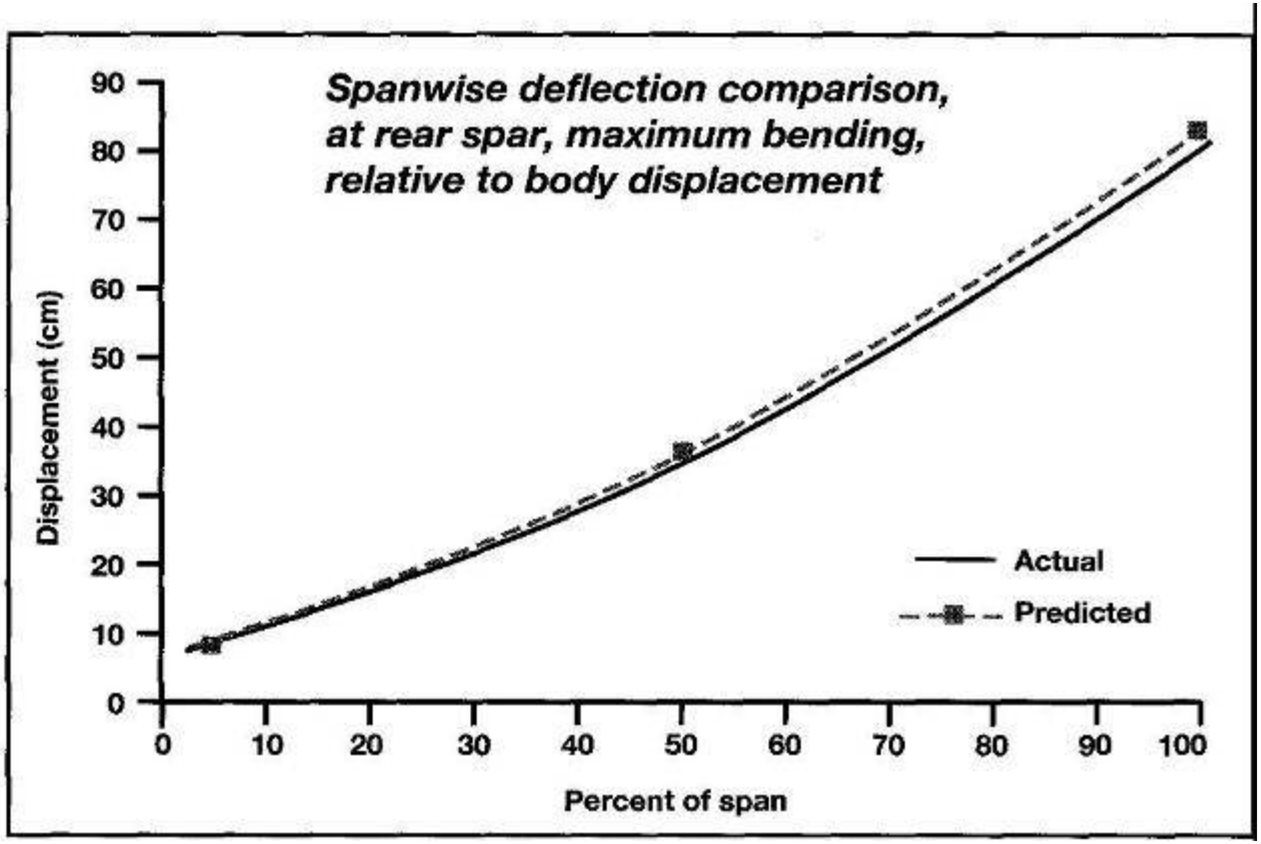
777 Empennage Certification Approach
by Fawcett, Trostle and Ward

Spanwise Deflection Comparison at the Front Spar



777 Empennage Certification Approach
by Fawcett, Trostle and Ward

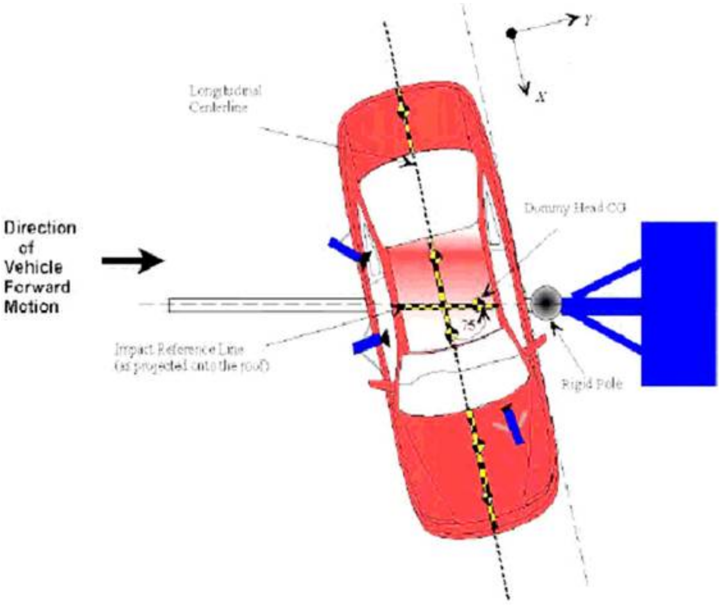
Spanwise Deflection Comparison at the Rear Spar



Lamborghini , Aventador- Univ. of Wash.
Door Sill Technology Demonstrator- Feraboli/Deleo

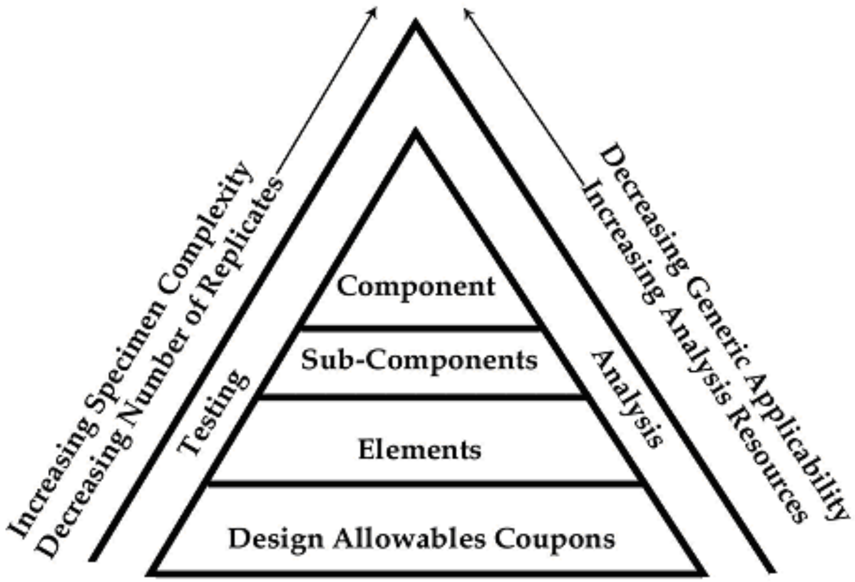
Energy–Absorbing Sandwich Structural Concept Using the building block approach (BBA): Design and Certification of Door Sill of new Lamborghini supercar, Aventador.

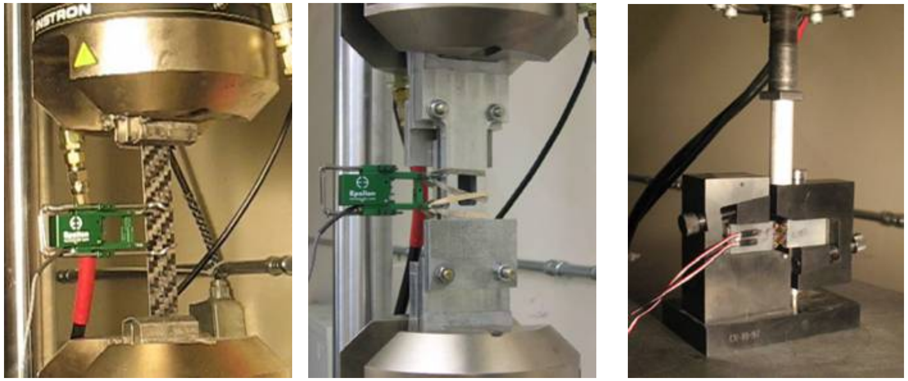
- **Certification by analysis versus testing**
  - In the automobile world a vehicle is certified for crashworthiness by testing alone
  - Costly, time-consuming, requires long lead-times for re-development
  - Analysis is used in the design/sizing stage
- **Certification by analysis supported by test evidence**
  - Derived from commercial aircraft industry
  - Adapted to automotive need by Lamborghini
  - Reduces amount of large scale testing by using a mix of testing and analysis
- **FMVSS 214 Side Impact Protection**
  - Third part: Oblique Side Pole Impact Test
  - 20 mph (32.2 km/h)
  - Fixed steel pole 10 in. (254 mm) diameter
  - 75 degrees from the axis of the vehicle



Lamborghini , Aventador- Univ. of Wash.
Door Sill Technology Demonstrator- Feraboli/Deleo

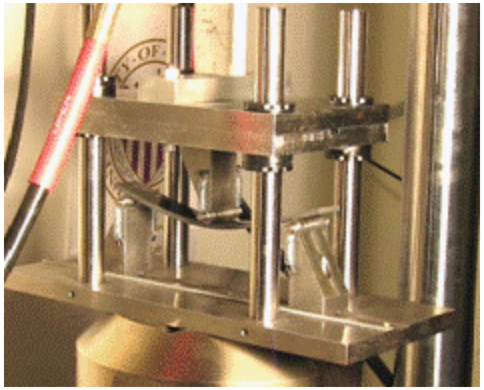
- **Door sill FEA model can be isolated in key material models**
  - MAT 54 for the composite facesheets
  - MAT 126 for honeycomb core
  - Tie-break contact for adhesive joint
- **Need to perform specific tests for each MAT model**
- **Coupon level testing** to generate allowable to assemble material model cards
  - Represent real (not nominal) production process and includes effect of damage
- **Element level testing** to calibrate the material models
  - Facesheets in bending
  - Honeycomb crushing
  - Tie-break contact for bonded joints
- **Sub-component level testing** to validate material models

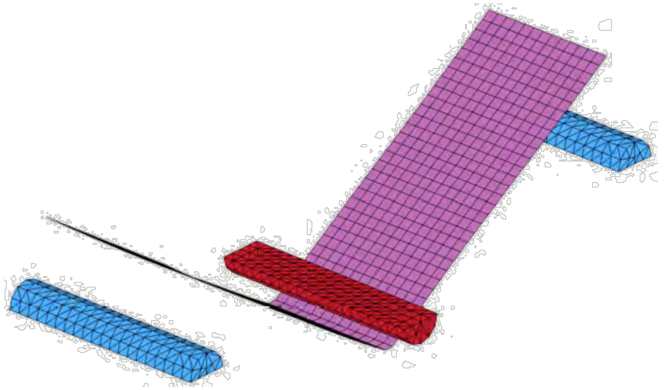


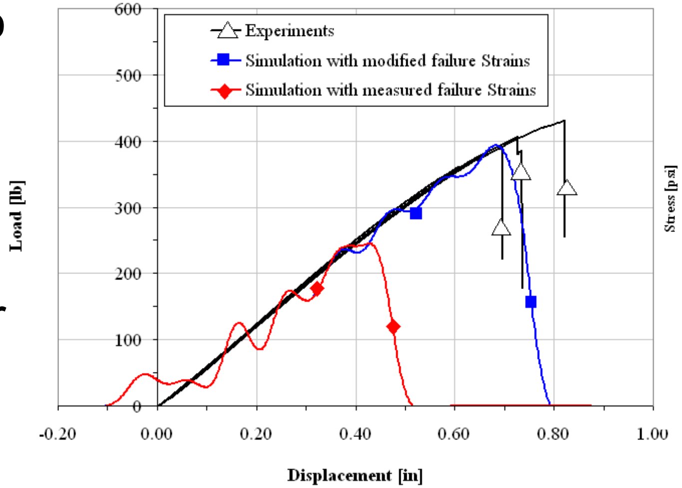


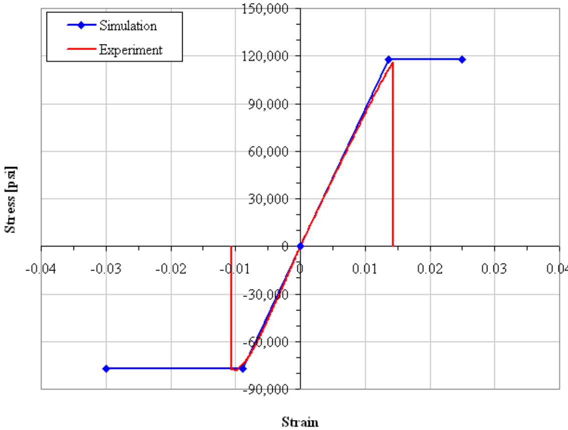
Lamborghini , Aventador- Univ. of Wash.
Door Sill Technology Demonstrator- Feraboli/Deleo

- MAT model parameters are tuned to match experiment
- Three-point bend flexure test on carbon facesheet
- Experimental stress- strain curves in tension and compression (RED) lead to low failure load and displacement for flexure test simulation
- Need to virtually increase the strain-to-failure in order to match experimental data (BLUE). **This is <u>calibration</u> test.**

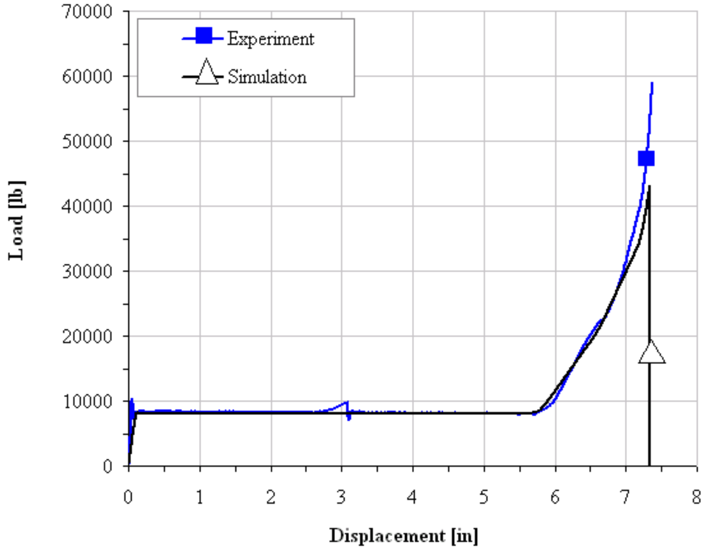


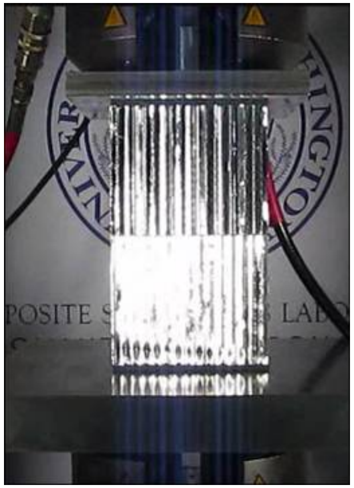


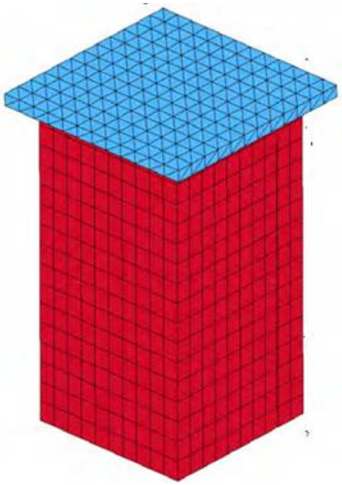


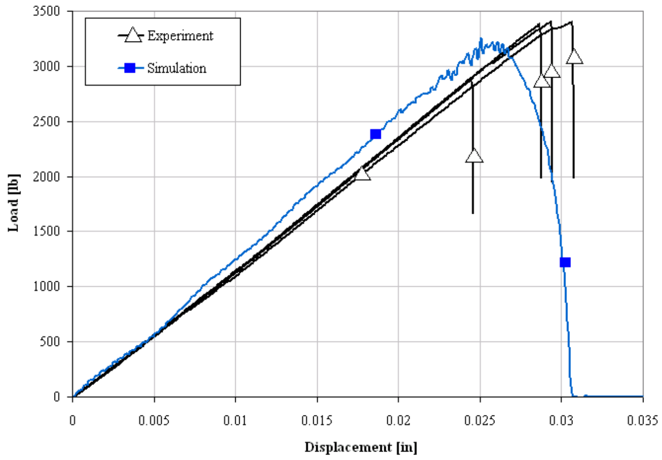


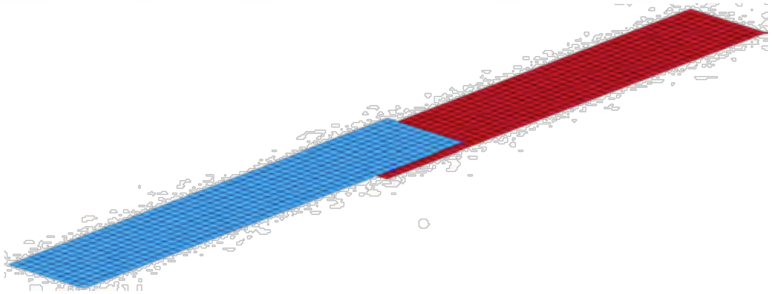
Lamborghini , Aventador- Univ. of Wash.
Door Sill Technology Demonstrator- Feraboli/Deleo





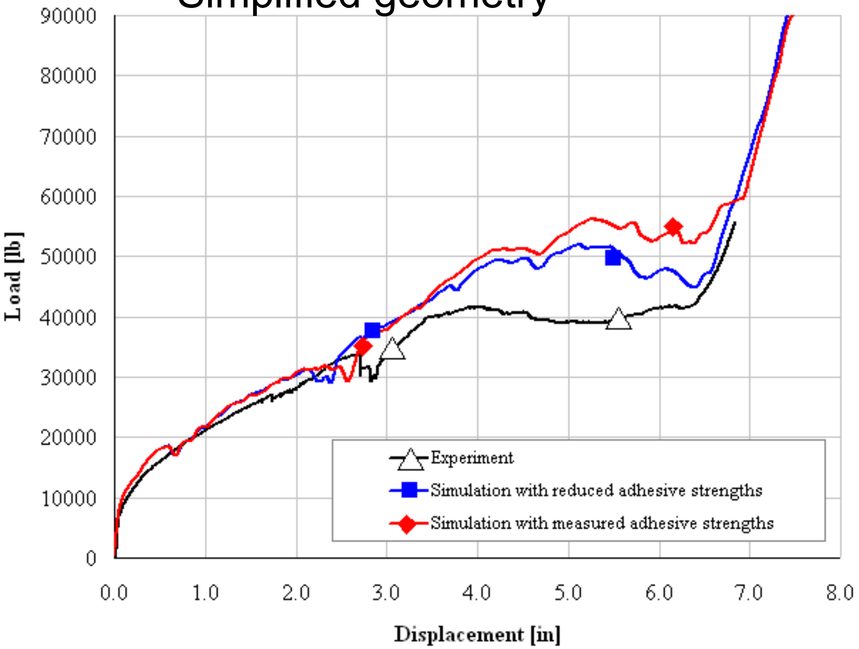


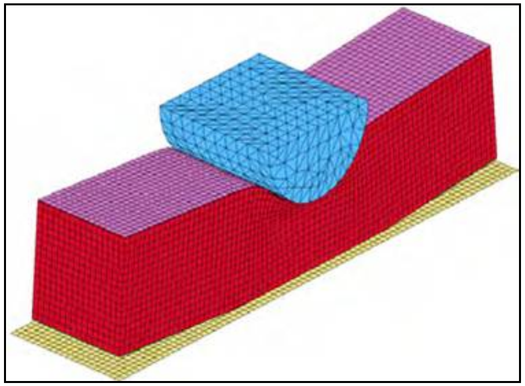


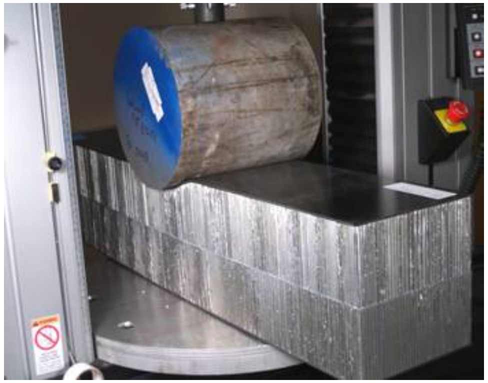


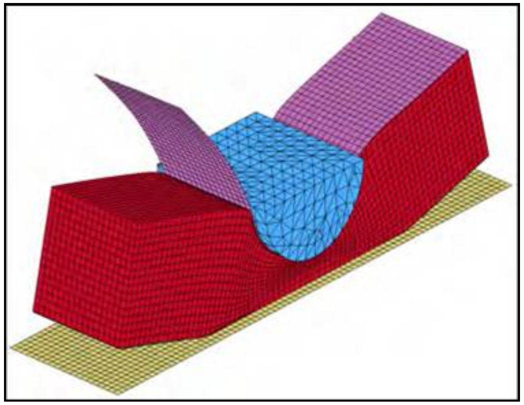
Lamborghini , Aventador- Univ. of Wash.
Door Sill Technology Demonstrator- Feraboli/Deleo

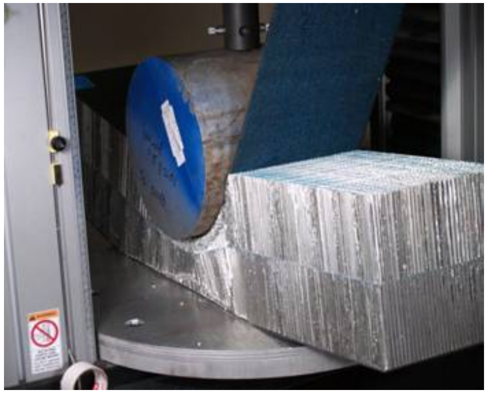
- **Full-scale model is assembled**
- **Parameters <u>cannot</u> be changed to match experiment- This is <u>validation</u> test**
- **Pole crushing of deep large beam**
    - Materials & processing are consistent
    - FMVSS pole
    - Simplified geometry











Click for Video

Questions?