Practical Advice for Finite Element Analysis of Your Design

February 26th, 2009

Agenda

Practical Advice for Finite Element Analysis of Your Design

February 26th, 2009

8am PST (Los Angeles) / 11am EST (New York) / 4pm GMT (London)

- Welcome & Introduction (Overview of NAFEMS Activities)
  - Mr. Matthew Ladzinski, *NAFEMS North America*
- Practical Advice for Finite Element Analysis of Your Design
  - Mr. Tony Abbey, *FETraining*
- Q&A Session
  - Panel
- Closing

Ladzinski

Abbey

An Overview of NAFEMS NA Activities

Matthew Ladzinski
NAFEMS
North American Representative

Planned Activities in North America

- Webinars
  - New topic each month!
    - March 19th - Modal Analysis in Virtual Prototyping and Product Validation
  - Recent webinars:
    - Pathways to Future CAE Technologies and their Role in Ambient Intelligent Environments
    - Computational Structural Acoustics: Technology, Trends and Challenges
    - FAM: Advances in Research and Industrial Application of Experimental Mechanics
    - CCOPPS: Power Generation: Engineering Challenges of a Low Carbon Future
    - Practical CFD Analysis
    - Complexity Management
    - CCOPPS: Creep Loading of Pressurized Components – Phenomena and Evaluation
    - Multiphysics Simulation using Implicit Sequential Coupling
    - CCOPPS: Fatigue of Welded Pressure Vessels
    - Applied Element Method as a Practical Tool for Progressive Collapse Analysis of Structures
    - AUTOSIM: The Future of Simulation in the Automotive Industry
    - A Common Sense Approach to Stress Analysis and Finite Element Modeling
    - The Interfacing of FEA with Pressure Vessel Design Codes (CCOPPS Project)
    - Multiphysics Simulation using Directly Coupled-Field Element Technology
    - Methods and Technology for the Analysis of Composite Materials
    - Simulation Process Management
    - Simulation-supported Decision Making (Stochastics)
    - Simulation Driven Design (SDD) Findings

To register for upcoming webinars, or to view a past webinar, please visit: www.nafems.org/events/webinars

Planned Activities in North America

- Training Courses for 2009:
  - Introduction to FEA Analysis
    - Los Angeles – March 24^{th} – 26^{th} – ***3* seats open!**
    - Chicago – May 26^{th} – 28^{th} – ***5* seats open!**
    - Orlando – September 22^{nd} – 24^{th} – ***11* seats open**
    - Seattle – November 3^{rd} – 5^{th} – ***12* seats open**
  - Proposed Courses for 2009
    - Dynamic FE Analysis
    - Verification and Validation (V&V)

To register for a training course, or for more information, please visit: www.nafems.org/events/

NAFEMS WORLD CONGRESS 09

JUNE 16th - 19th CRETE - GREECE

- When: June 16th – 19th, 2009
- Where: Crete, Greece
- Updates:
  - Over 200 presentations
  - Six Keynote Presentations
  - Additional Workshops and Activities:
    - Mini-symposium: Analysis and Simulation of Composite Structures Including Damage and Failure Prediction
    - Engineering Analysis Quality, Verification & Validation

NAFEMS WORLD CONGRESS 09
JUNE 16th - 19th CRETE - GREECE

- Additional Workshops and Activities (cont.):
  - High Performance Computing in Engineering Simulation
  - Multi-physics Simulation: Advanced Coupling Algorithms and Strategies
  - Crash

- Additional Workshops and Activities (cont.):
  - EC AUTOSIM Project – (one year)
  - EC FENet Project – (four years)
  - EC Multi-Scale Analysis of Large Aerostructures Project
  - NAFEMS Skills Management Initiative
  - Simulation Data Management
  - Material Data
  - Optimization/Robustness/Stochastics
  - Round Table Discussion on Business Drivers

NWC 2009

NAFEMS WORLD CONGRESS 09

JUNE 16th - 19th CRETE - GREECE

NWC09 Keynotes

- **Erich Schelkle** - Porsche AG and Automotive Simulation Center Stuttgart, Germany

- **Tsuyoshi Yasuki** - Toyota Motor Corporation, Japan

- **Martin Wiedemann** - DLR German Aerospace Center, Germany

- **Jacek Marczyk** - Ontonix, Italy

- **Louis Komzsik** - Siemens PLM Software, USA

- **François Besnier** - Principia RD, France

NWC 2009
NAFEMS WORLD CONGRESS 09
JUNE 16th - 19th CRETE - GREECE

- **For more information about the NWC09, please visit: www.nafems.org/congress**

- **Sponsorship and Exhibition Opportunities Still Available!**
  - **For more information, please visit: www.nafems.org/congress/sponsor**

Principal Sponsor
SIMULIA

Gold Sponsors
ANSYS
SIEMENS

Silver Sponsors
BETA CAE Systems S.A.
CD-adapco
ESI GROUP
Fraunhofer Institut Algorithmen und Wissenschaftliches Rechnen SCAI
LMS
MSC Software

NAFEMS

Welcome and Agenda

Introduction to FETraining

Overview of the 3 day course

Practical Advice for FEA of Your Design

Q and A

Introduction to FETraining

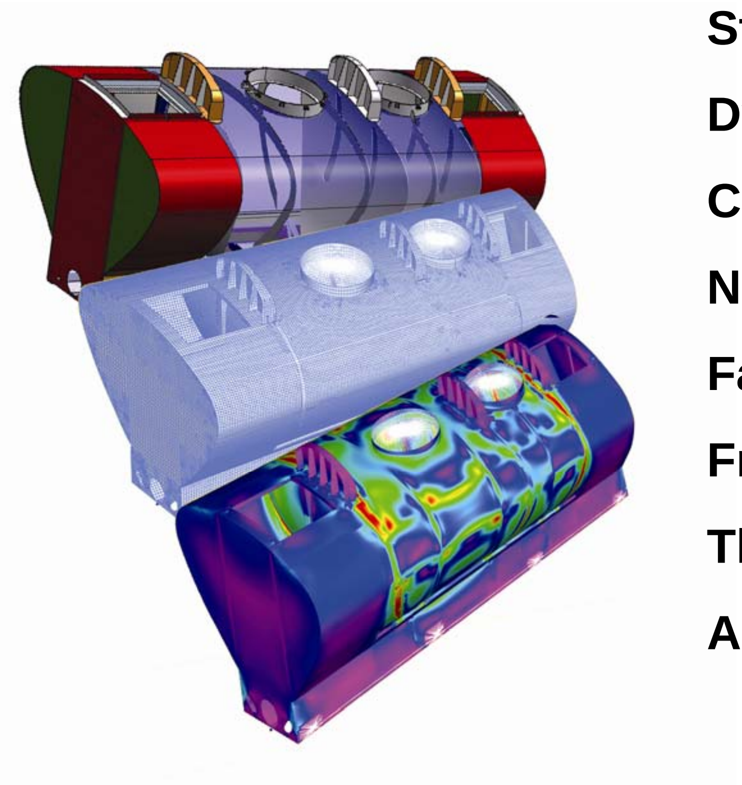
Primary Skill Set:
NASTRAN
PATRAN
FEMAP

Tony Abbey

BSc Aero. Eng. University of Hertfordshire, UK
MSc Struct. Eng. Imperial College, London

- Started at BAC Warton, UK in 1976
- Worked in UK Defence Industry for 20 years; Hunting Engineering, BAe Systems, RRA
- Joined MSC.Software as UK support and Training Manager
- Transferred to MSC.Software US in 2000
- Joined Noran Engineering 2003
- Formed FE Training in 2007

Intro to FETraining: Consultancy Solutions



Statics

Dynamics

Composites

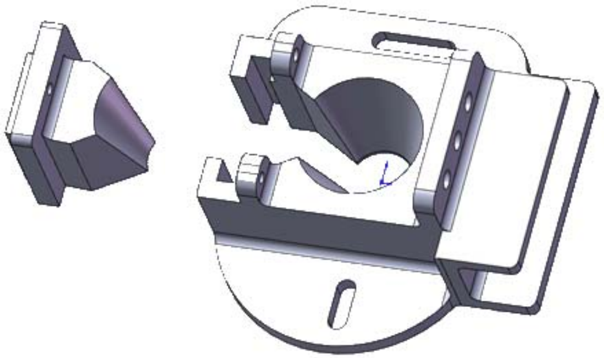
Non-Linear

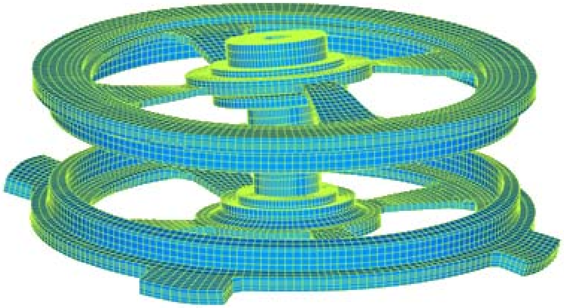
Fatigue

Fracture Mechanics

Thermal

Aero Elasticity





Details

Email : tony@fetraining.com

www.fetraining.com

Intro to FETraining: Training Solutions

Interactive DVD

Live Training On-Site or Public Courses

Webinar – multiple or one-on-one

Details

Email : tony@fetraining.com

www.fetraining.com

Overview of 3 day Class

FEA has become widely used and universally accepted in many industry sectors.

In order to derive maximum benefit from the available technology, engineers should learn about the strengths of numerical techniques and how to apply them.

At the same time, guidance on how to avoid the inevitable pitfalls will prove invaluable throughout the engineer's career.

This three day example-driven, practical course is designed to meet this requirement.

The course is accredited by NAFEMS, the only vendor neutral, not-for-profit organization with the aim of promoting the effective and reliable use of FEA.

The course is completely code independent. No software is required.

Practical Advice for FEA of Your Design

Agenda

- A Sanity Check on FEA
  – what it is and it is not

- Understanding the objective of the analysis
  – why are we doing it?

- Getting a clear view of the scope of the real world problem
  – how do we tackle it?

- Looking critically at the CAD geometry model
  – how much can we use?

Practical Advice for FEA of Your Design

Agenda (continued)

- Why not use 20 million elements
  – let the computer take the strain?
- Real world boundary conditions and loading
  – some examples of good and bad modeling
- Anticipate the load paths
  – examples of how to produce free body diagrams to use as a sanity check on your models
- Checking the results
  – FEA is guilty until proven innocent

A Sanity Check on FEA

A little bit of history!

During the 1950s Advances in Aircraft Design brought more complex structural problems

Environment Driven

- High speed
- Compressibility
- Flutter

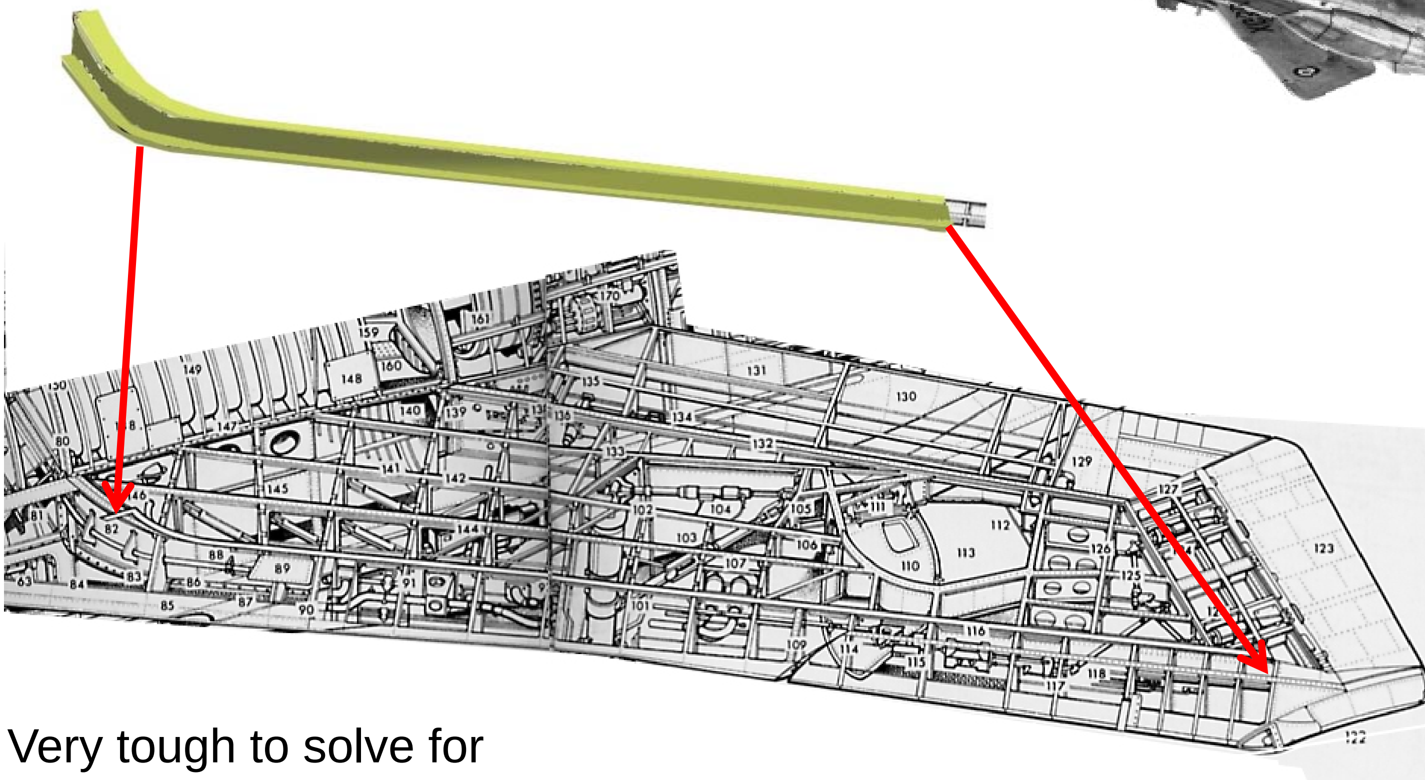
Design Driven

- Swept wing
- Advanced structural layouts
- Advanced materials and methods
- Highly redundant – complex load paths

Multiple equations to Solve
Automating Hand calculations

A Sanity Check on FEA

English Electric P1B 'Hockey Stick' Spar 1954



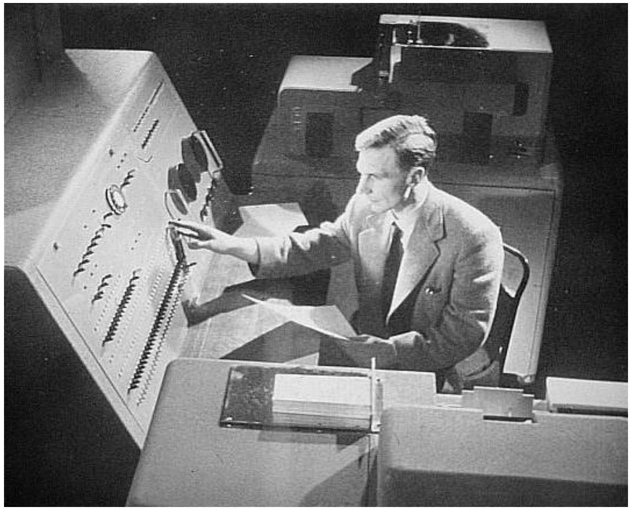
Very tough to solve for stresses by hand

A Sanity Check on FEA

Resulting system of **stress** equations put into matrix form:

- 7 by 7 matrix solution using a mechanical calculator could take a full day

- Very small models – high degree of ingenuity and **idealization**



English Electric Deuce Computer

Analog and digital computers in the mid 50's began to increase the size of problem that could be solved

However set up was very procedural, tedious and specific to the structure being analyzed

A Sanity Check on FEA

Big break through when solution was put in terms of solving for **displacement**

- Called **Displacement Method**
- Not very appealing at first , goal had been to solve for **stresses** via **Force Method**…
- But easier to formulate general purpose equations

Force = Stiffness * Displacement
F = K * d

Our FEA Foundation

Late 50's early 60's this method developed in many areas – most aircraft companies, but also marine, and civil.
The concept of a physical analogy to the equations emerged leading to FEA we know today.

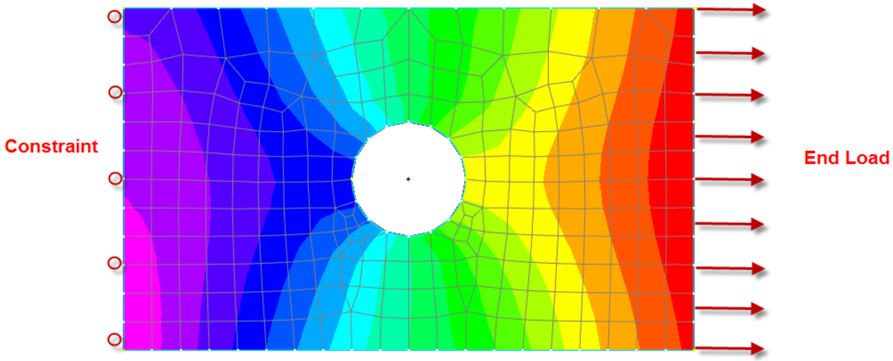
Finite Elements

- Finite - a simple mathematical representation of a specific region
- Element – a 'building block' that can be assembled into a complex representation

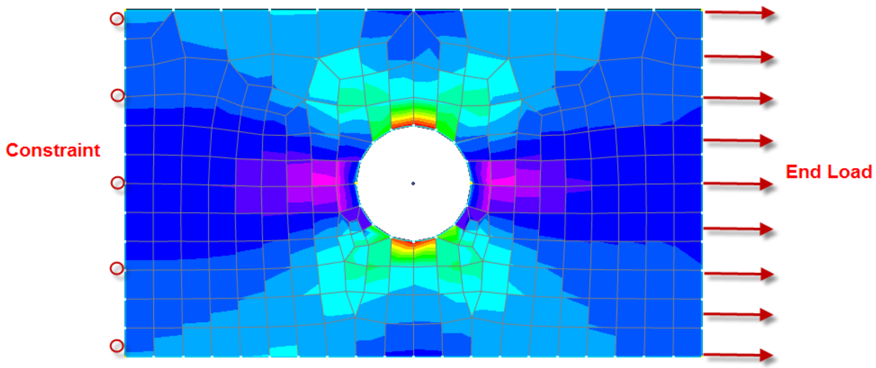
A Sanity Check on FEA

The reason for the history lesson!

- We solve for displacements
- They are continuous functions across the mesh of elements
- Stresses are calculated independently in each element
- Stresses at a node will be different for each adjacent element



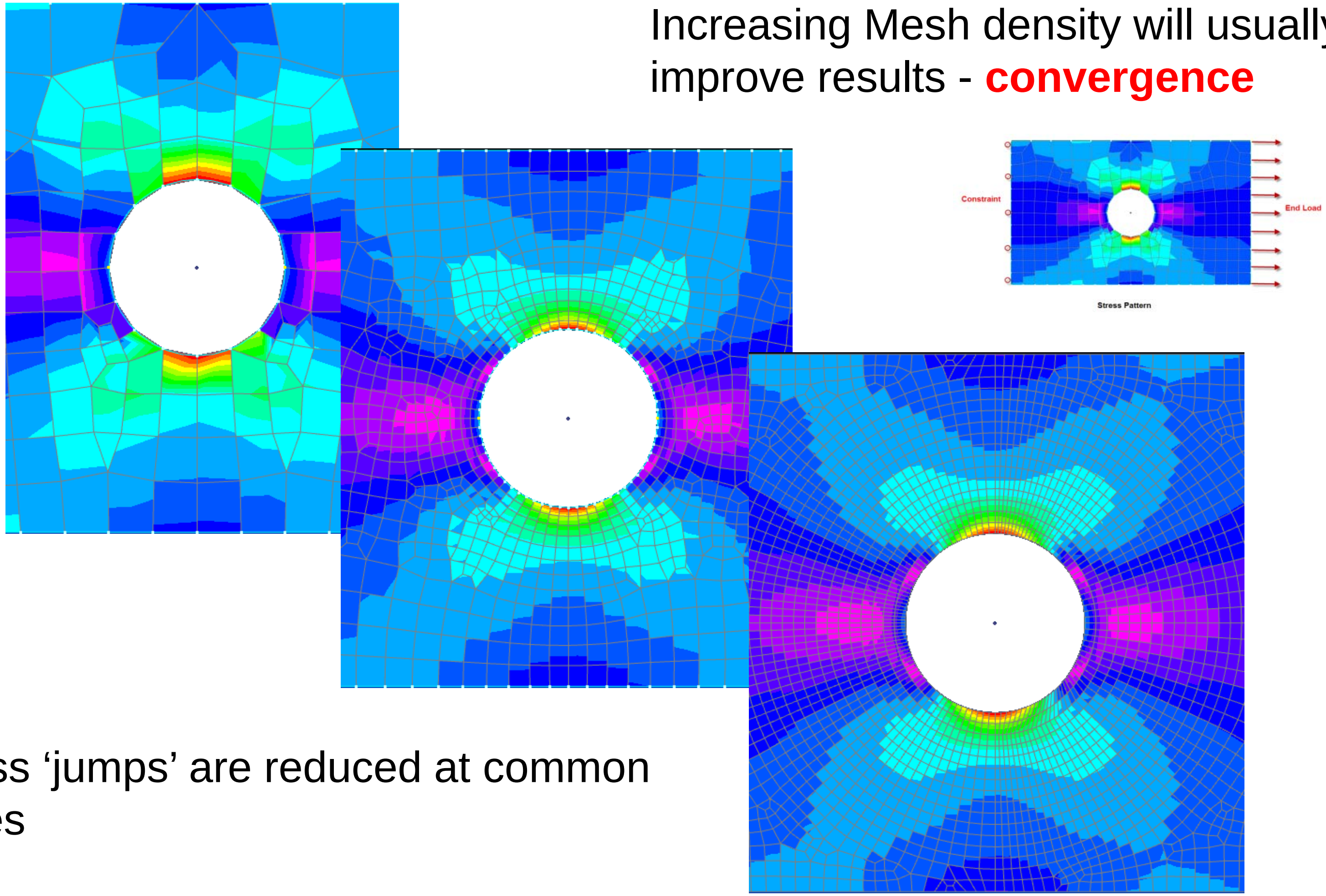
Displacement Pattern



Stress Pattern

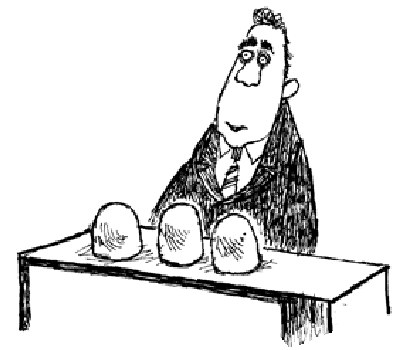
A Sanity Check on FEA

Increasing Mesh density will usually improve results - **convergence**



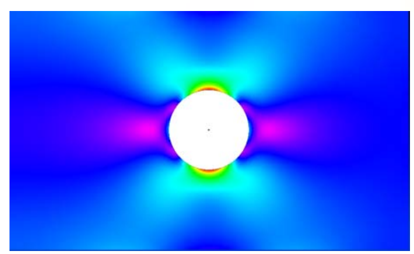
Stress ‘jumps’ are reduced at common nodes

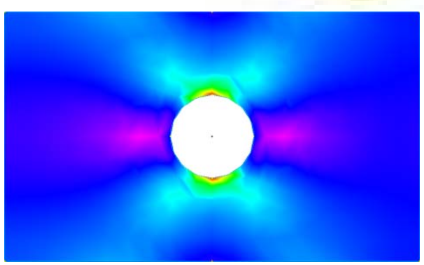
A Sanity Check on FEA

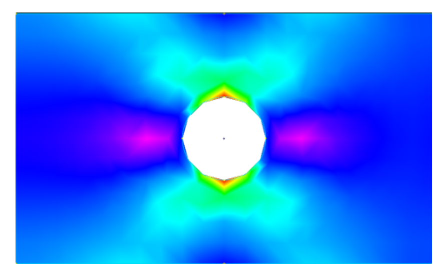


In the shell game – with smoothed post processing

Which is the bad one?







A Sanity Check on FEA

Class Topics taking this further

- Simple element matrix descriptions
- How distortion inaccuracies are introduced
- Element Quality checking measures
- Convergence checking
- 2D and 3D popular element options
- Case Studies

Practical Advice for FEA of Your Design

Agenda

- A Sanity Check on FEA
  – what it is and it is not
- **Understanding the objective of the analysis**
  – why are we doing it?
- Getting a clear view of the scope of the real world problem
  – how do we tackle it?
- Looking critically at the CAD geometry model
  – how much can we use?

Understanding the Objective of the Analysis

– why are we doing it?

In the consulting world, very often …

Something broke

Something needs certifying

These are motivations.

Now we figure out the real objectives…

Understanding the Objective of the Analysis

It's just so too easy to go eyes down and start meshing!

Why am I doing this analysis?

- Check maximum stress, maximum displacement
- Check permanent distortion
- Evaluate Fatigue life
- Establish Critical frequencies
- Check strength against shock or seismic loading
- Investigate Crashworthiness
- Etc.

Maybe it is a combination of these

Understanding the Objective of the Analysis

How accurate do the answers have to be?

- Approximate to an order of magnitude, based on loading and boundary condition assumptions with large margins
- As accurate as possible, with good material data, clearly defined loading spec and well determined boundary conditions
- Safety or mission critical requiring, adherence to regulating authority QA procedures and 3 sigma reliability

Understanding the Objective of the Analysis

What resources do I have

- Computing hardware, CPU speed, RAM, Disk space and I/O speed
- Computing software, are all needed modules available
- Timescale; long - elaborate or short - quick and dirty
- Experience levels available

Medium computer resources
Static Analysis license only
Three weeks to get solution
I have some FEA experience

Simple models to gain understanding

2000 Linux cluster
Full ABAQUS, DYNA3D, NASTRAN licenses
Nine months timescale
15 experienced analysts in the team

Comprehensive system analysis

Understanding the Objective of the Analysis

If you miss the analysis deadline and the design is frozen or metal is cut

- The resource spent on the over complex model is lost
- Maybe a simple model, showing the ball park stresses or stiffnesses would have been useful
- Keep this kind of model around

Understanding the Objective of the Analysis

How do I fit into the Design, Test, Analysis, Manufacture Process

- Right first time ethos, aggressive timescales and targets
- Working to reduce design cycles
- Forensic checking only

- Integrated team designer/analysts
- Silo mentality

Can we benefit from:

- CAD/FEA integration
- FEA/Test correlation

Understanding the Objective of the Analysis

Class Topics taking this further

- Case studies on Project Analyses from basic to complex
- Students question simulated customer's brief
- Solutions are discussed
- Analysis plan evolved

Practical Advice for FEA of Your Design

Agenda

- A Sanity Check on FEA
  – what it is and it is not
- Understanding the objective of the analysis
  – why are we doing it?
- Getting a clear view of the scope of the real world problem
  – how do we tackle it?
- Looking critically at the CAD geometry model
  – how much can we use?

Getting a Clear View of the Scope of the Real World Problem

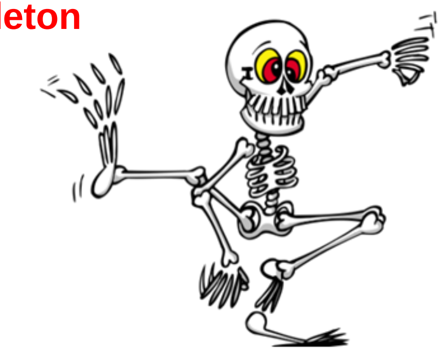
– how do we tackle it?

What is the Scope of the FEA technology Implied?

Basic strength assessment ⟷ **Low Cycle Plastic crack growth**

What is the Dimensional Scope?

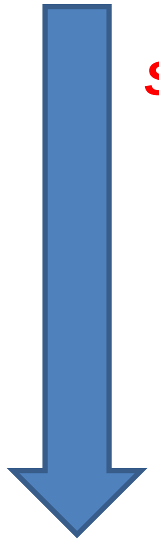
Shin Bone ⟷ **Full Skeleton**

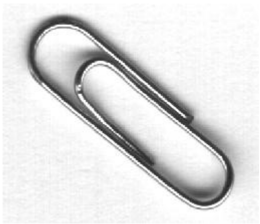


Getting a Clear View of the Scope of the Real World Problem

Scope of the FEA technology

- linear strength and stiffness
- bilinear plasticity
- general plasticity, advanced material models
- strain rate effects
- simple contact
- self seeking general contact
- large displacements and follower forces
- fatigue and fracture mechanics
- linear dynamic response analysis
- non-linear dynamic response
- damping mechanisms

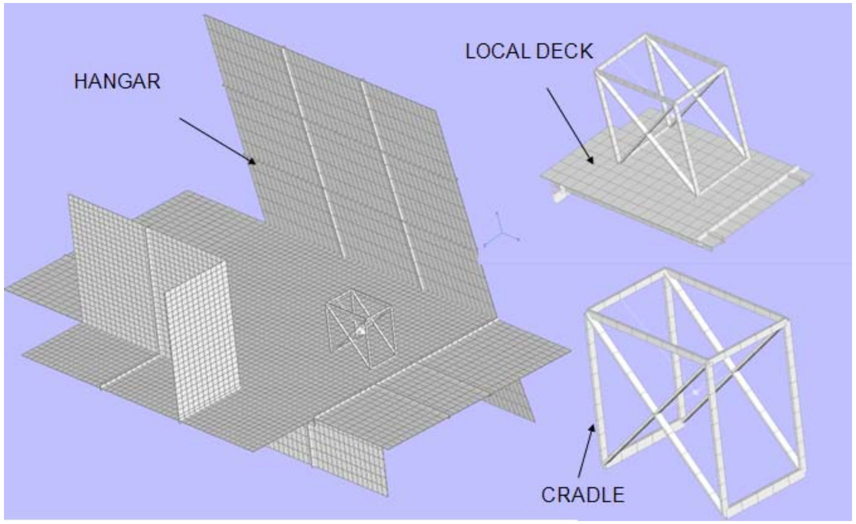


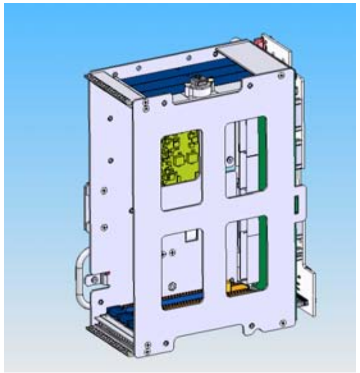


Getting a Clear View of the Scope of the Real World Problem

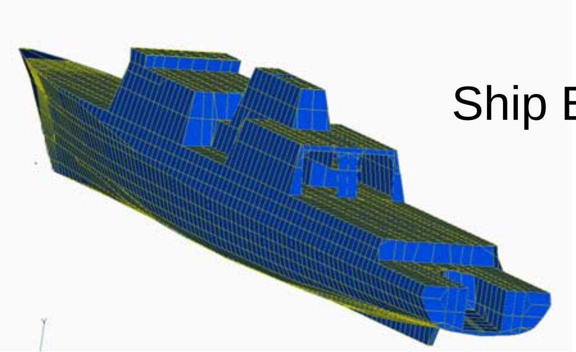
Dimensional Scope

- Assemblies
- Surrounding Structure
- Internal Structure





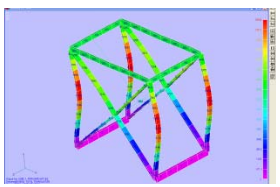
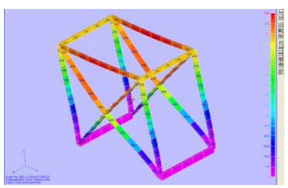
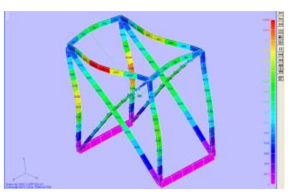
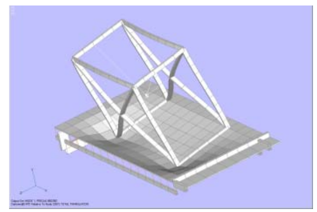
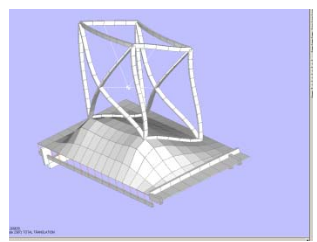
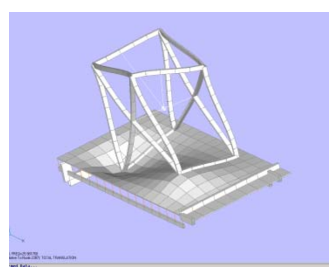
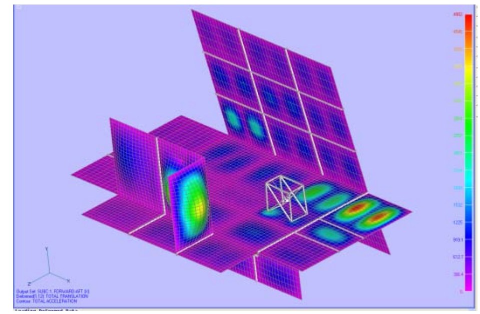
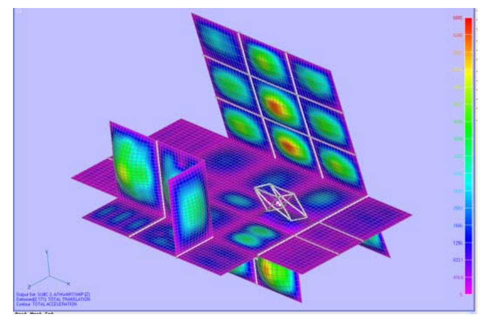
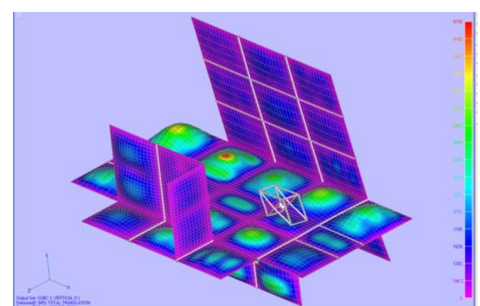
Electronic Packaging Example



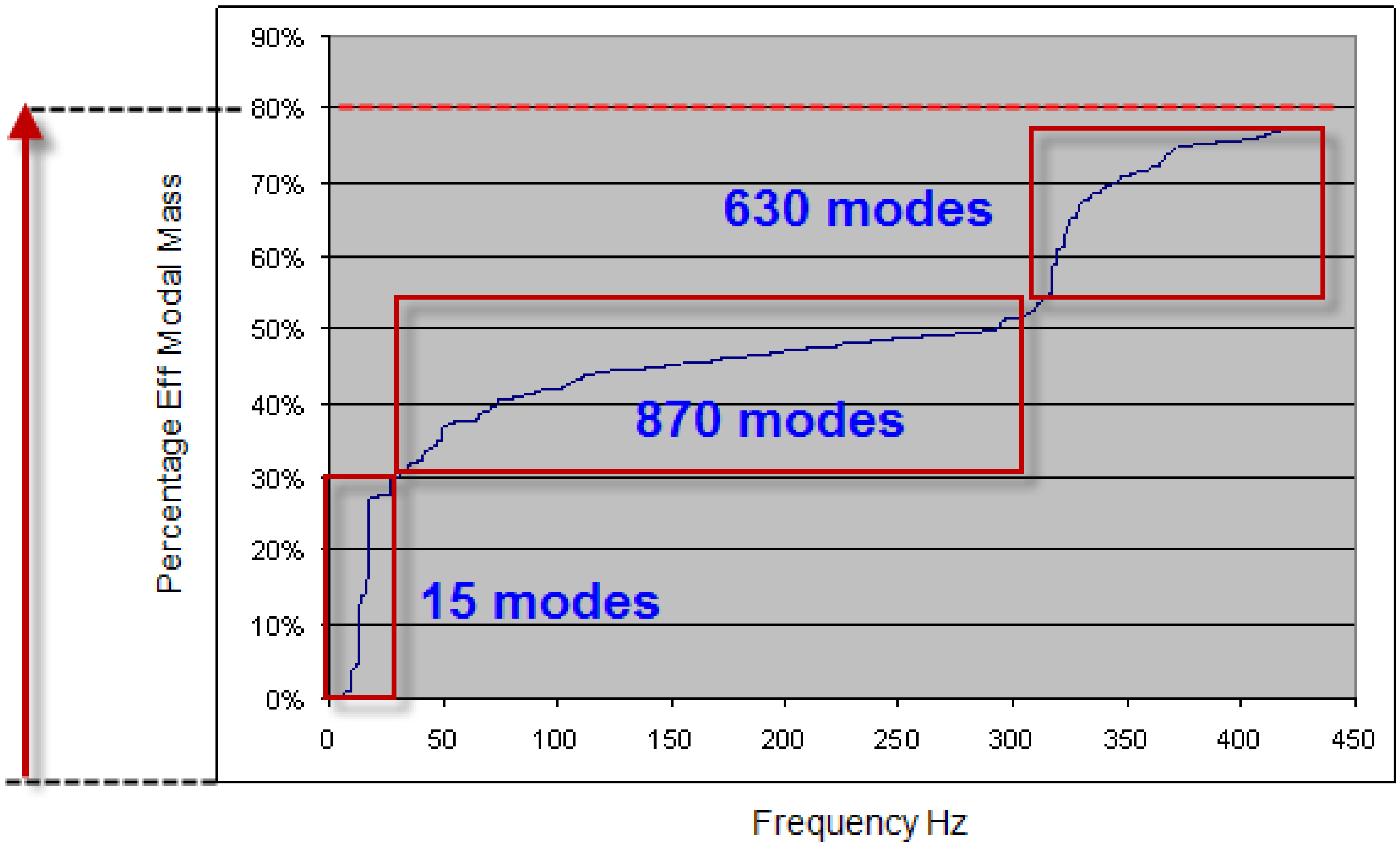
Ship Example

Getting a Clear View of the Scope of the Real World Problem

Increasing amounts of surrounding structure

Getting a Clear View of the Scope of the Real World Problem



Getting a Clear View of the Scope of the Real World Problem

Class Topics taking this further

- Review of Non-Linear Analysis
  - Elastic - Plastic
  - Large Displacement and Buckling
  - Follower Force
  - Contact

- Assessing technology levels
- Simplifying techniques
- Progressive approach to solving problems
- Case studies

Practical Advice for FEA of Your Design

Agenda

- A Sanity Check on FEA
  – what it is and it is not

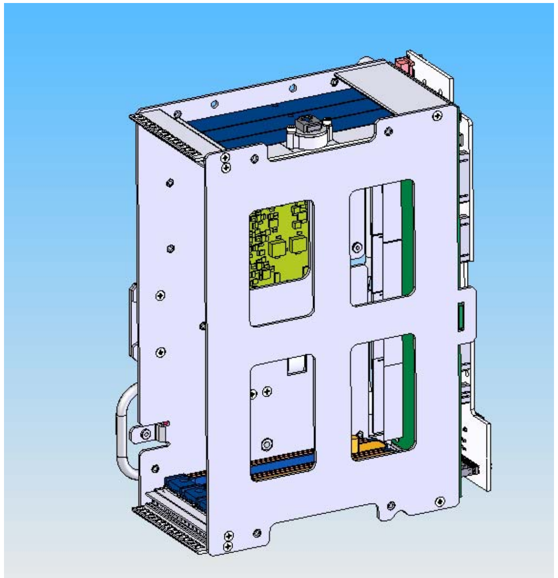
- Understanding the objective of the analysis
  – why are we doing it?

- Getting a clear view of the scope of the real world problem
  – how do we tackle it?

- Looking critically at the CAD geometry model
  – how much can we use?

Looking Critically at the CAD Geometry Model

Scope of Geometry (linked to Real World Scope)



Can we ignore:
- Bolts, washers, springs etc.
- Bolt holes, tooling holes, part numbers etc.
- Parts or features away from load paths

Can we get rid of:
- Slivers, spikes etc.
- Non-watertight volumes
- (Is CAD geometry stable enough)

Can we fix up:
- Bad geometry

Can we make CAD fit our analysis objective:
- Idealize via thin shell or beams
- Simplify loading or boundary conditions

Looking Critically at the CAD Geometry Model

Choice of element types

- It is tempting to use solid elements for all analysis.
- but shell and to a lesser extent beam elements can be ideal

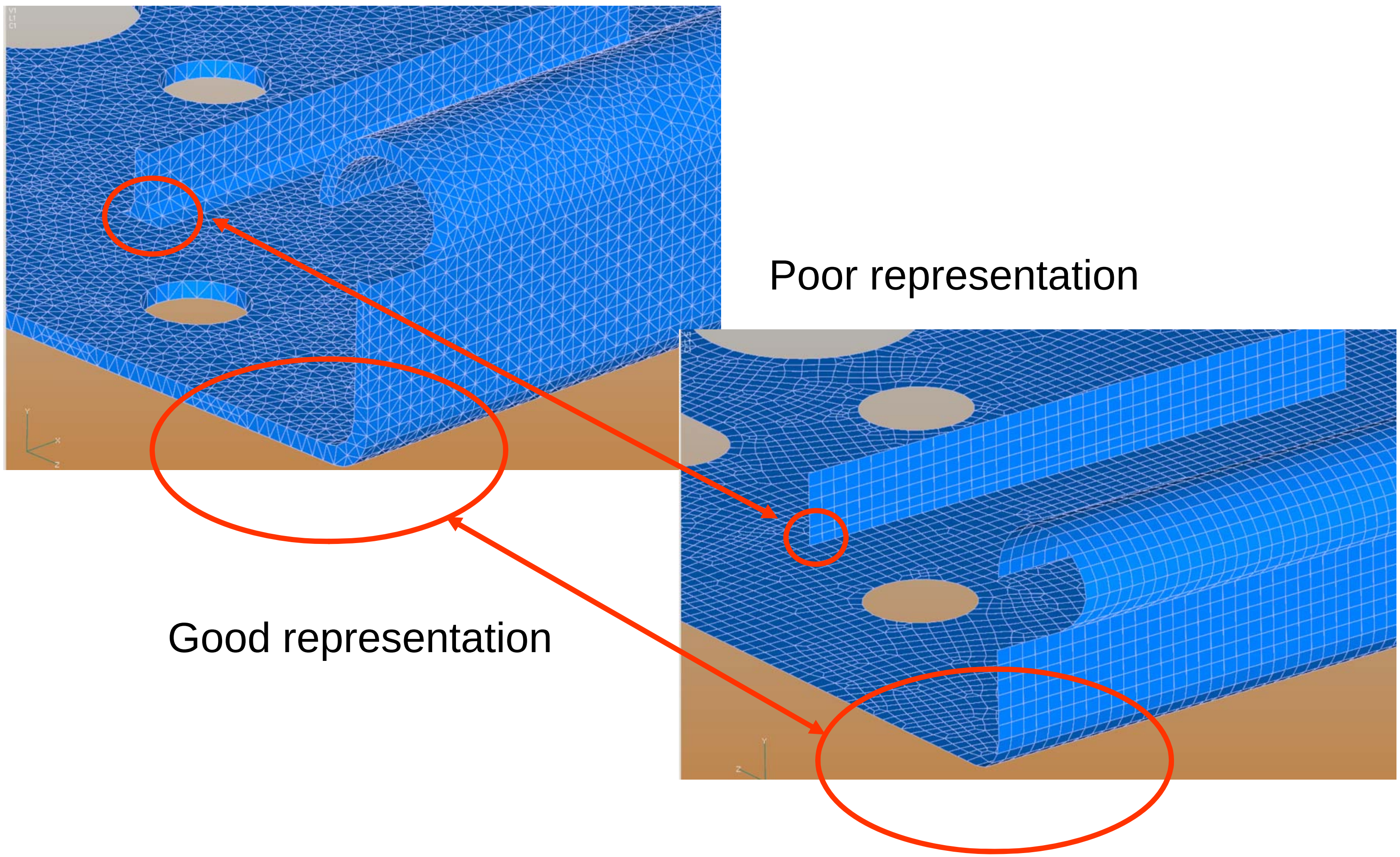
Shell Element Usage:

- Shell elements work very well in thin regions of structure such as webs, walls WHERE BENDING DOMINATES
- In fact the rule is if t/l > 15 then a good representation by a shell
- Solids have to be used with caution.
- Need several elements (3 or 4) through thickness with a TET mesh, and several (2) with a HEX mesh

- However, poor detail at intersections and joints

Looking Critically at the CAD Geometry Model

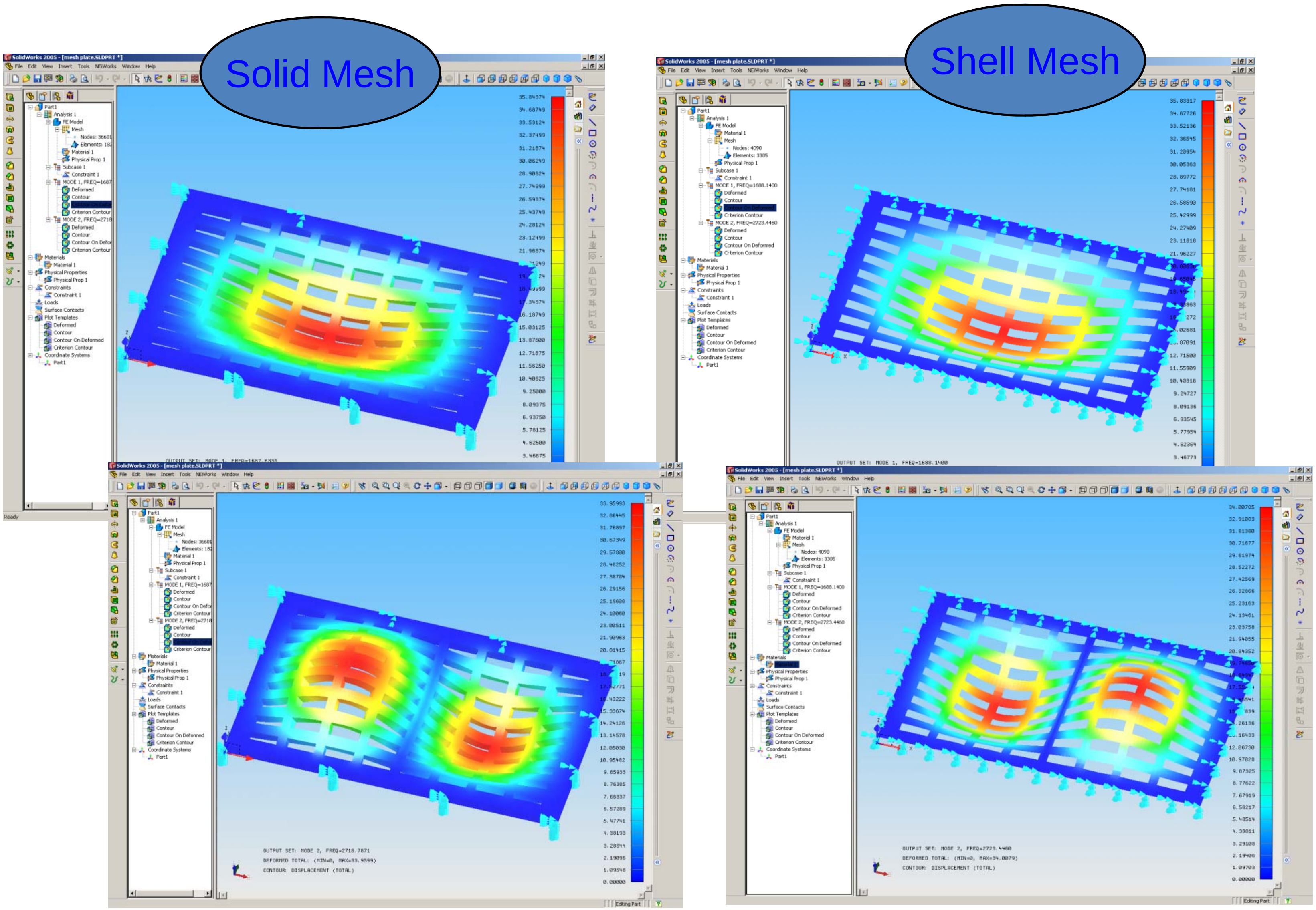
Shell Element Usage



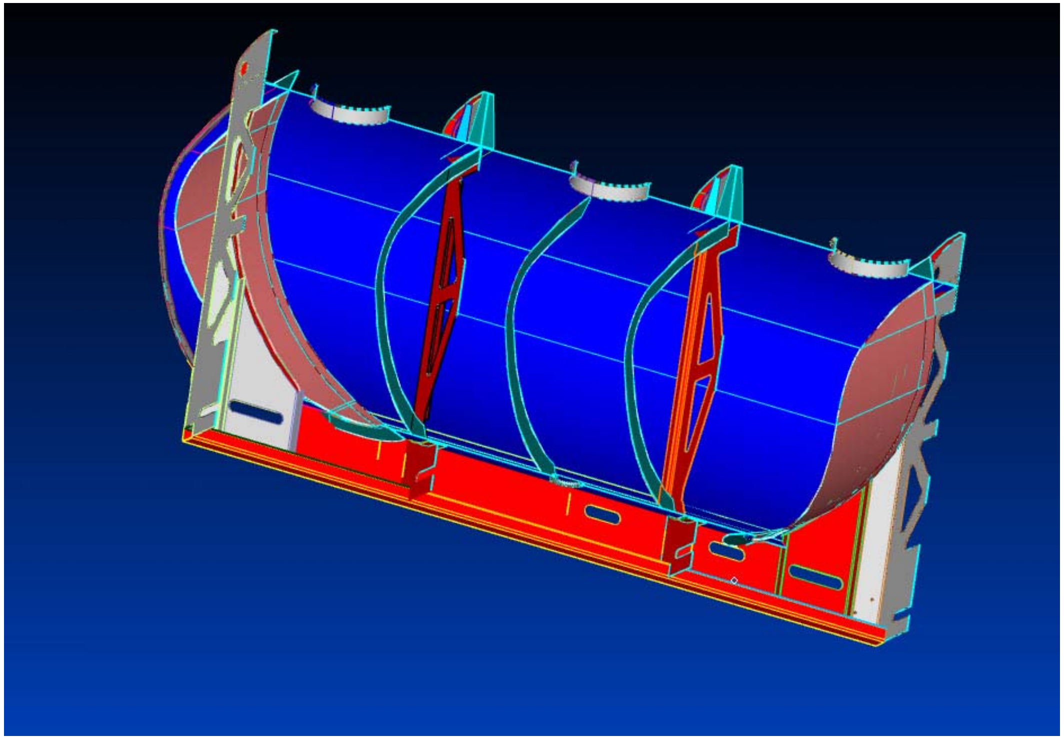
Poor representation

Good representation

Simplification of the Model

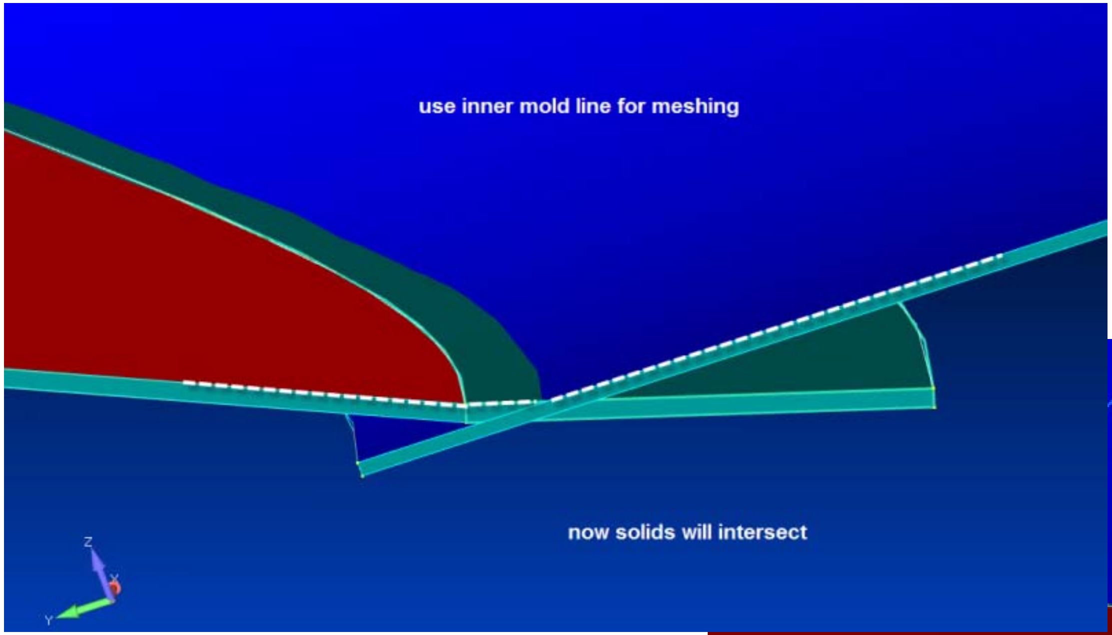


Looking Critically at the CAD Geometry Model

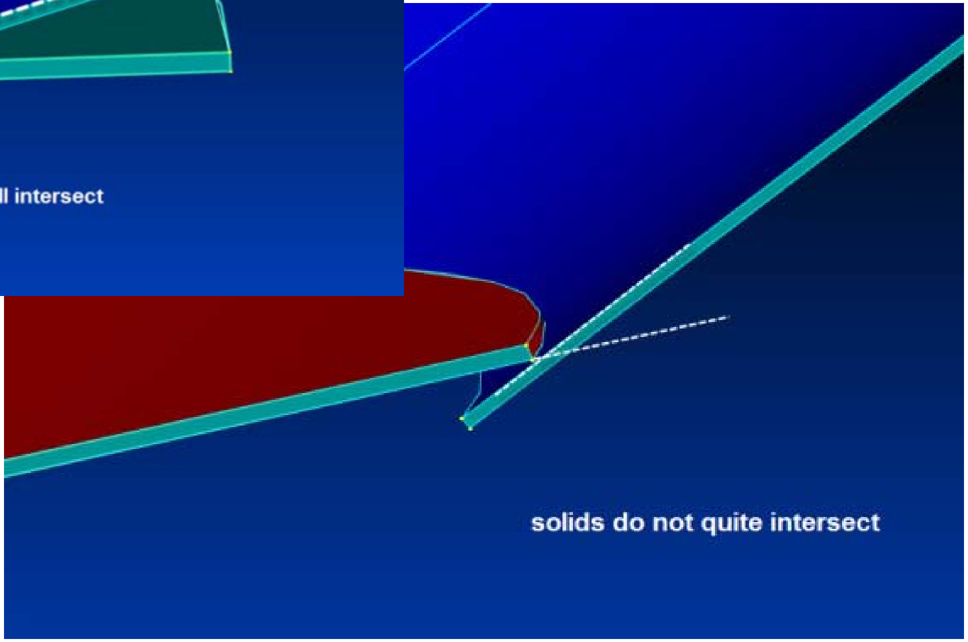


Review the solid geometry and decide how to idealize and what to simplify or eliminate

Looking Critically at the CAD Geometry Model

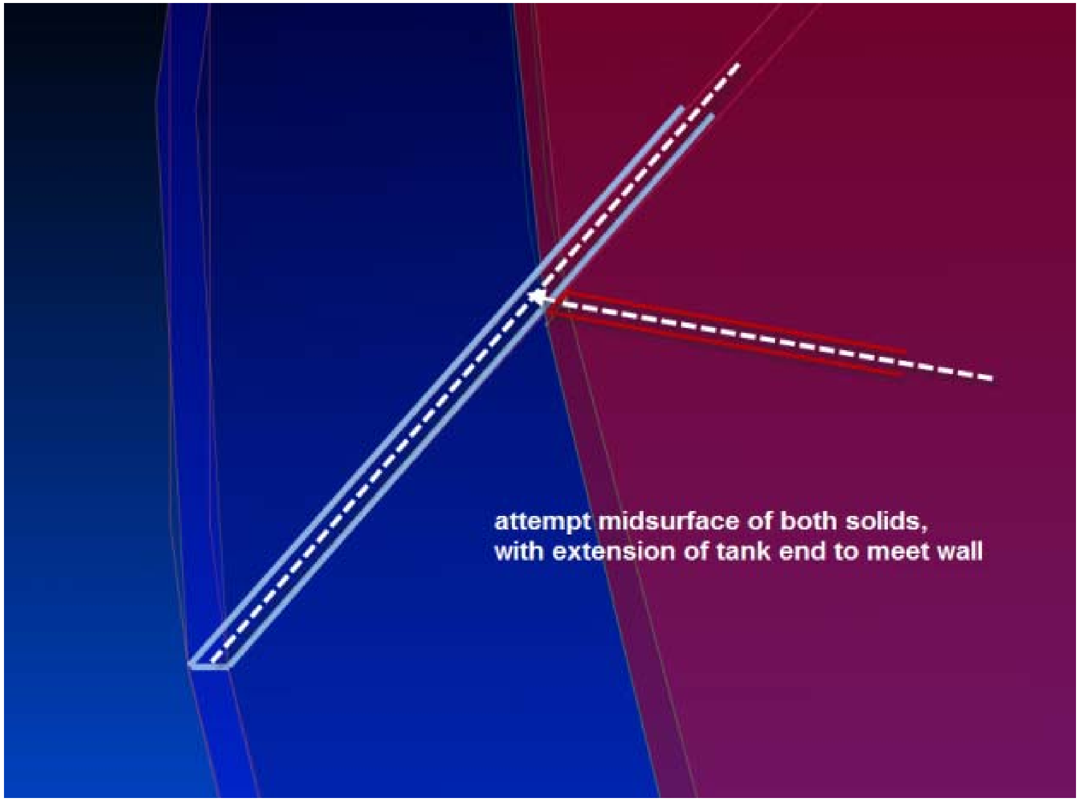


Extend geometry if feasible



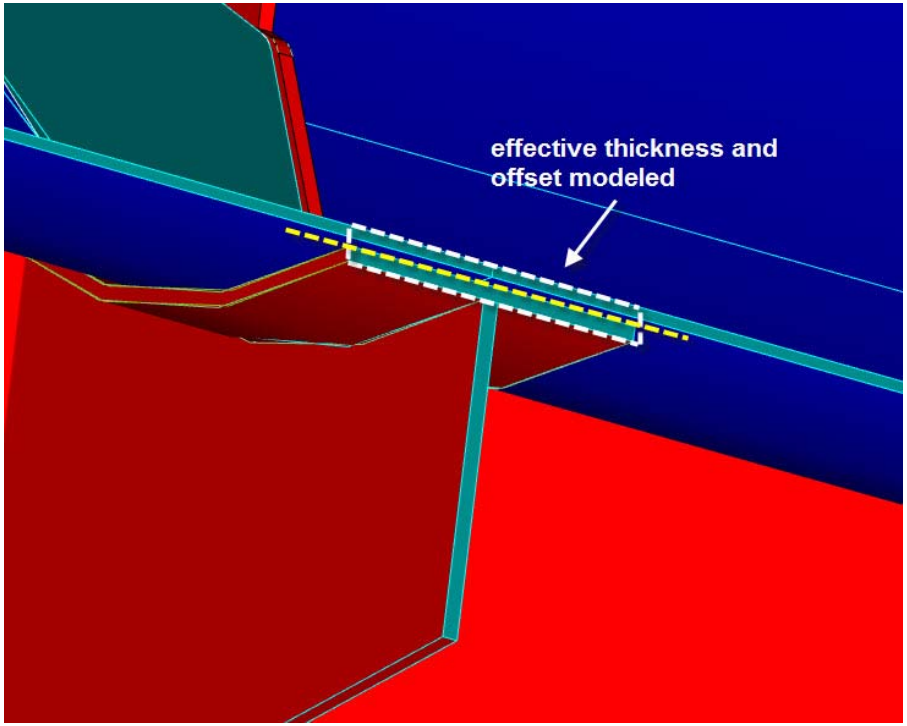
Example: anticipating welded fabrication, gaps are left

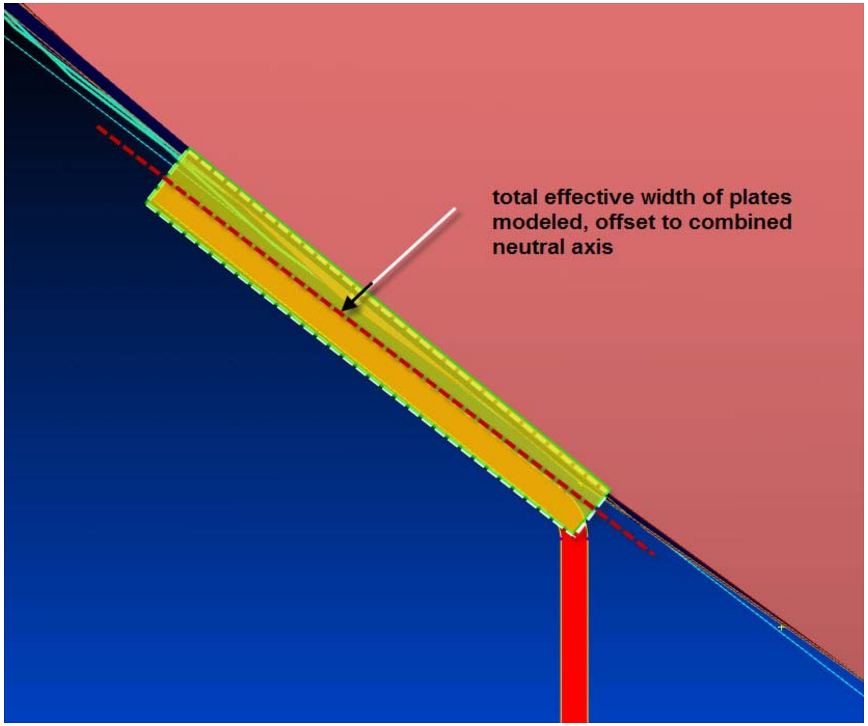
Looking Critically at the CAD Geometry Model



If using shells, pick datum mid surface, or inner/outer mold lines

Looking Critically at the CAD Geometry Model





If using shells, accommodate joggles, doublers offsets etc.

Looking Critically at the CAD Geometry Model

Class Topics taking this further

- CAD to FEA interchange
- Case Studies
- Review of idealization methods and tradeoffs
- Robustness of CAD model
- Defeaturing and meshing exercises

Practical Advice for FEA of Your Design

Agenda (continued)

- Why not use 20 million elements
  – let the computer take the strain?

- Real world boundary conditions and loading
  – some examples of good and bad modeling

- Anticipate the load paths
  – examples of how to produce free body diagrams to use as a sanity check on your models

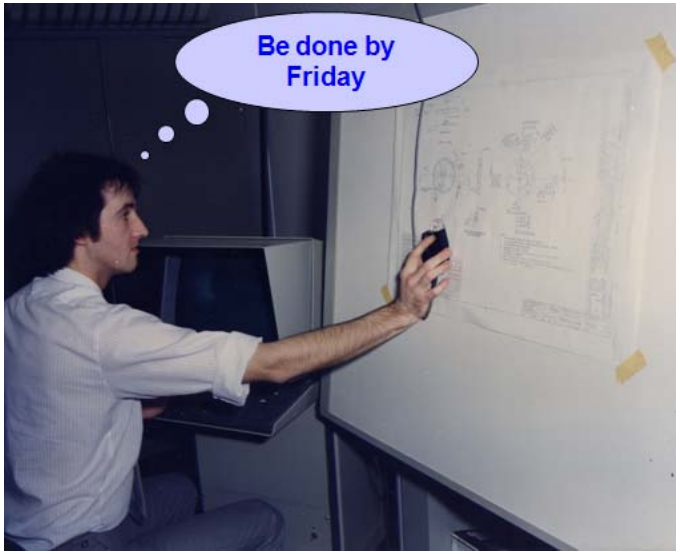
- Checking the results
  – FEA is guilty until proven innocent

Why Not Use 20 Million Elements?

Level of Idealization is a trade off

Bad old days - forced to spend a long time idealizing structure by simplifying

- Geometry (more realistically mesh layout)
- Loads
- Boundary conditions
- Post processing by hand – so models were simple



Why Not Use 20 Million* Elements?

Today we don't need to be as extreme – 'mesh what you see' has benefits

- Idealization experience not needed
- CAD manipulation reduced
- Meshing time is a minimum
- Simple stress results and displacements are easily reviewed

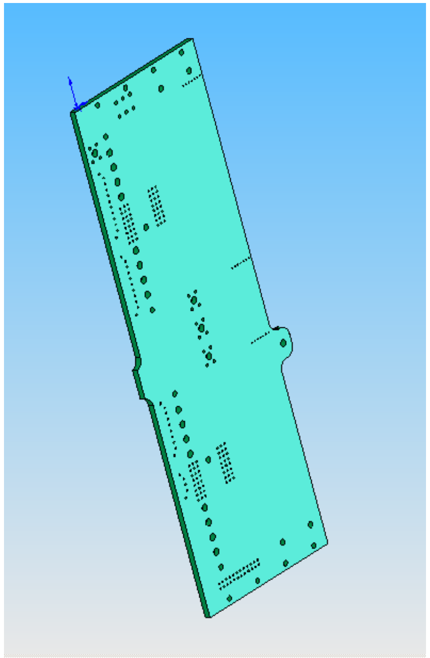
But there are disadvantages:

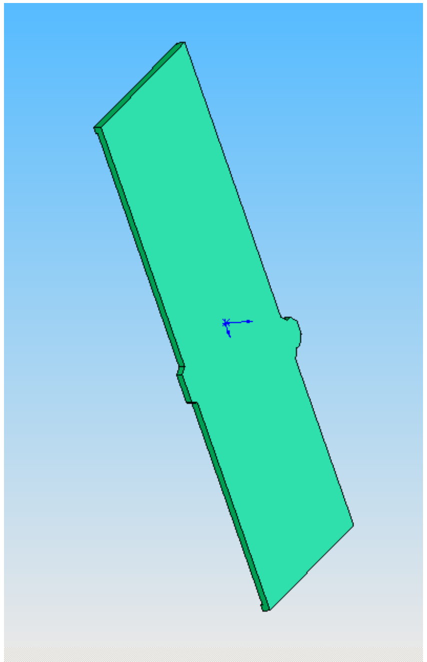
- Analysis time is longer (issue with non-linear, dynamics etc)
- Data storage is greater (issue with non-linear, dynamics etc)
- Establishing free body diagrams and explaining load paths may be difficult (tools required)
- Getting to the answers needed
- Full ship, aircraft or space models are still too complex (GDOF)

*Early 2009 – 20 Million element is still a VERY big model, pushing limits

Why Not Use 20 Million Elements?

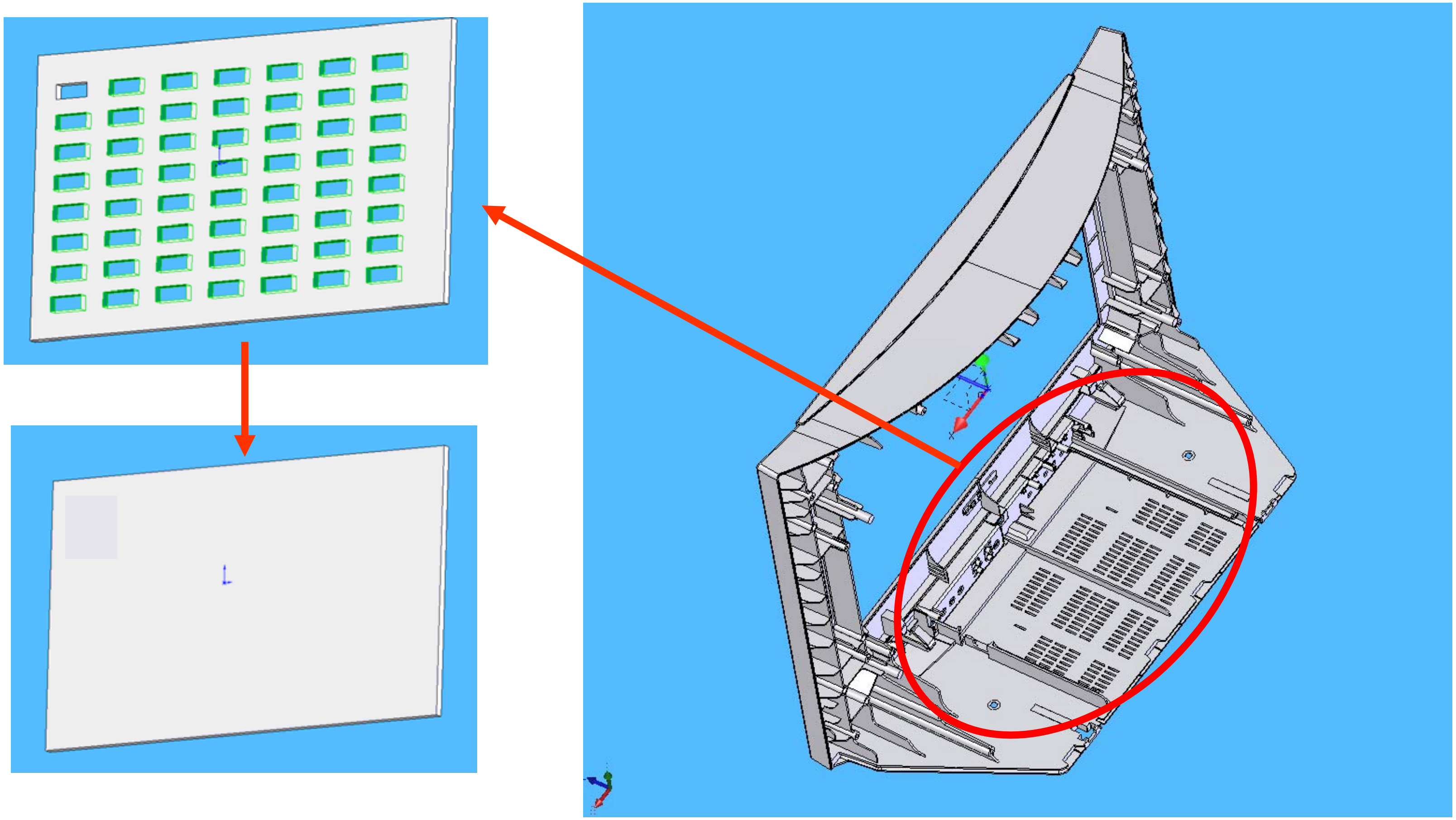
It takes little effort to achieve this type of defeaturing





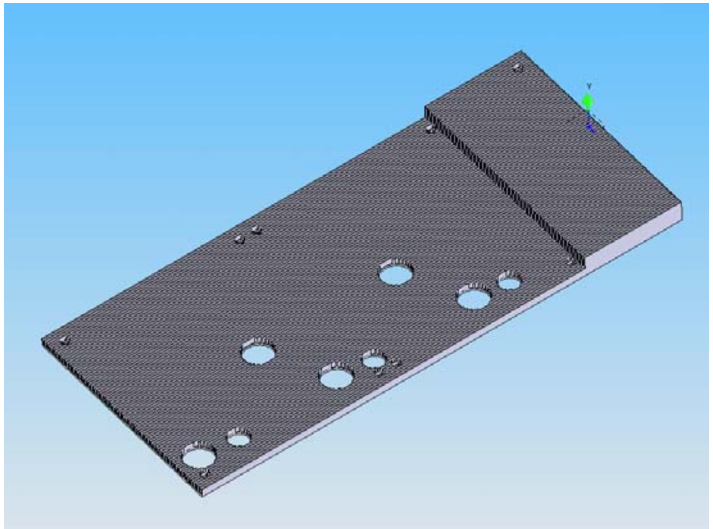
Element count can be significantly reduced

Why Not Use 20 Million Elements?

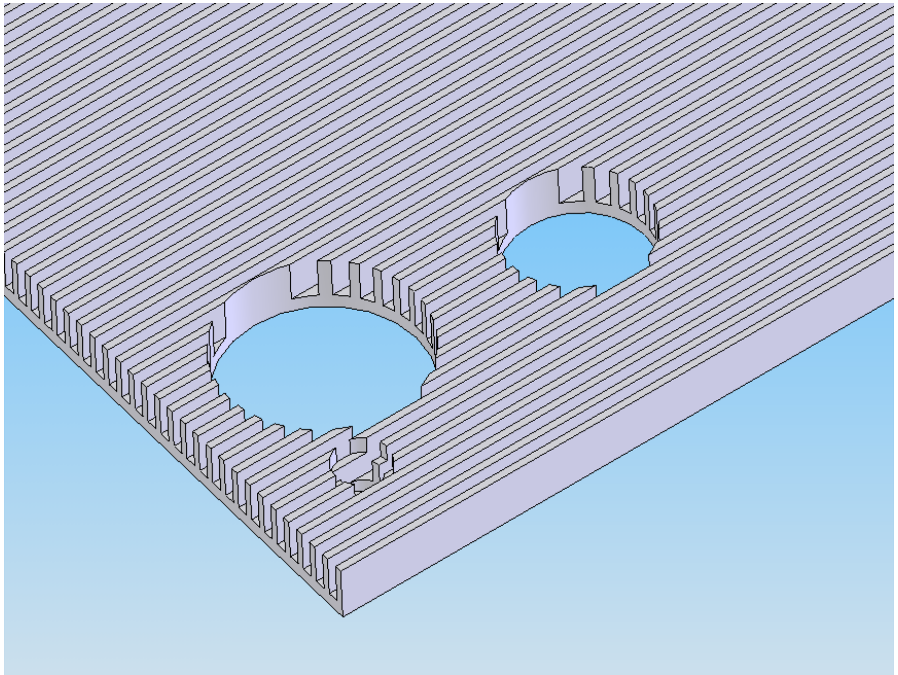


Use Smearing type techniques

Why Not Use 20 Million Elements?

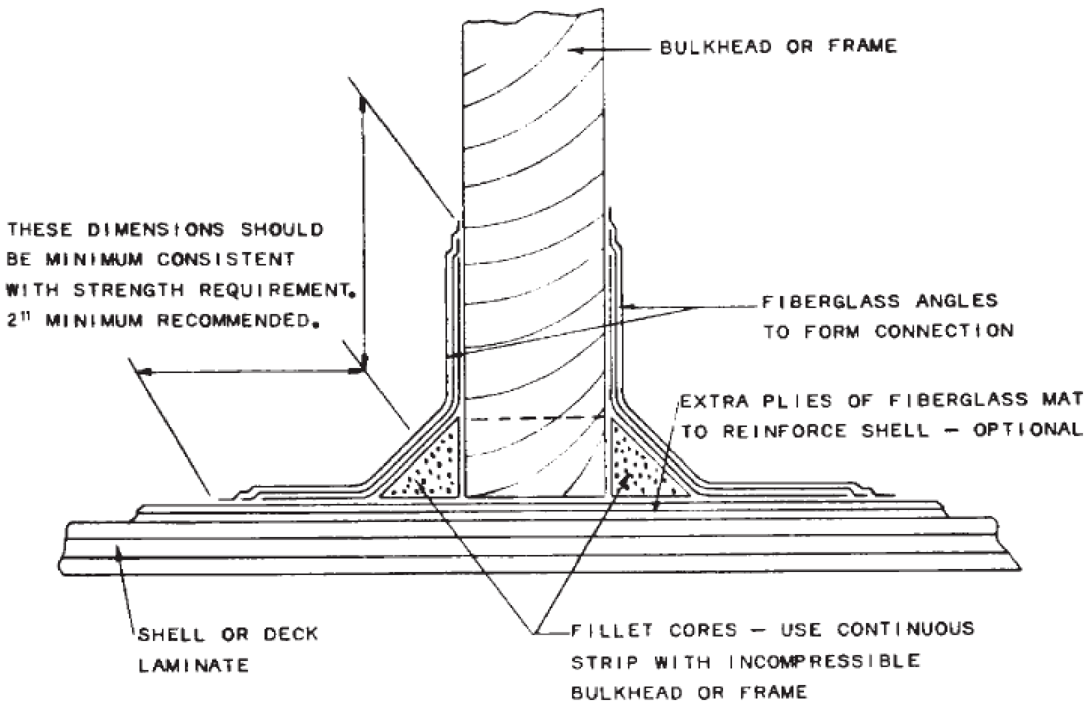


Integral cooling fins



- Huge element count to model in detail
- Use Orthotropic material Property
- Investigate hot spots with local models

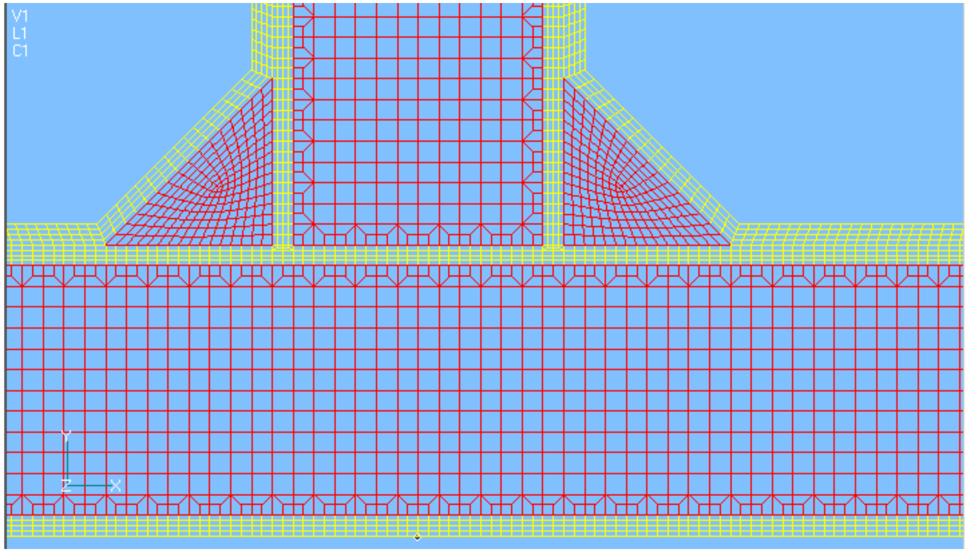
Why Not Use 20 Million Elements?

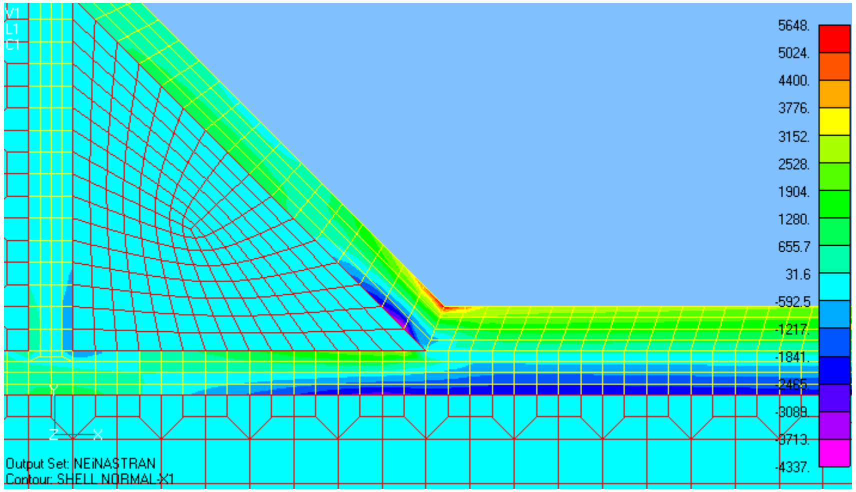


Connection of Bulkheads and Framing to Shell or Deck [Gibbs and Cox, ***Marine Design Manual for FRP***]

Too complex to model whole vessel at this level of detail

Why Not Use 20 Million Elements?

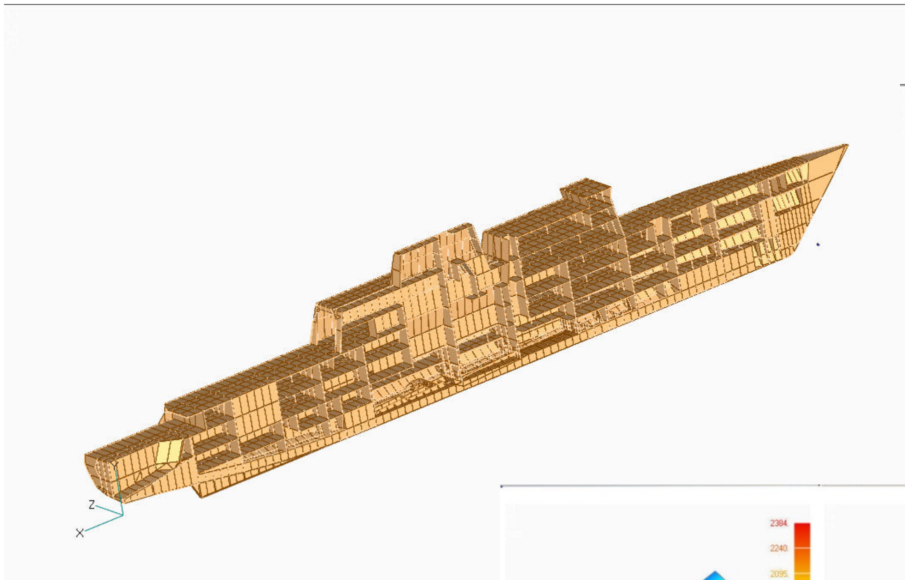


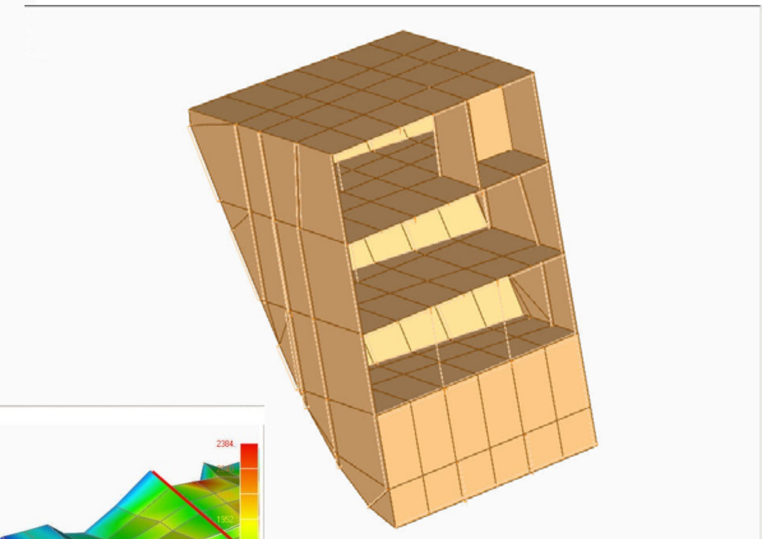


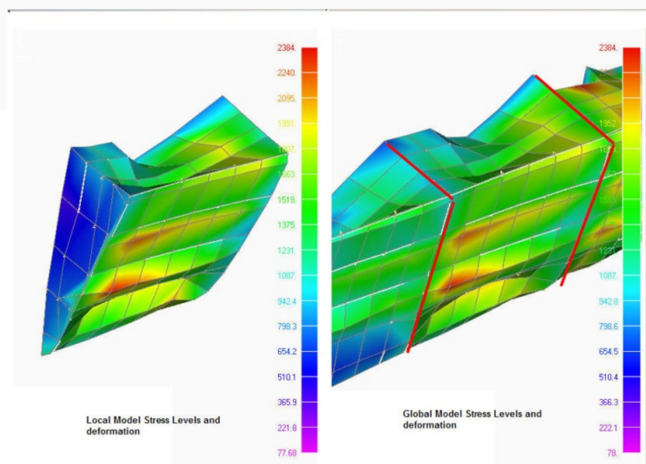
Use plane strain in highly loaded areas

Yes it can be done with orthotropic properties!

Why Not Use 20 Million Elements?

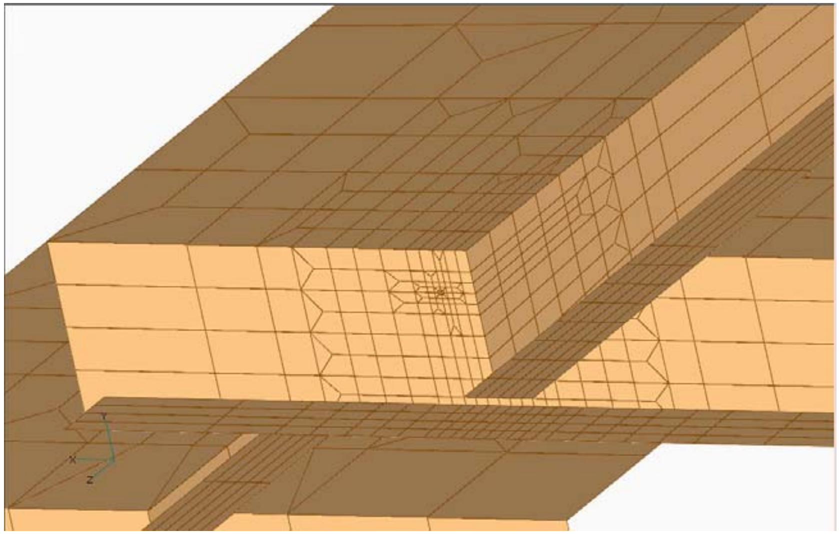




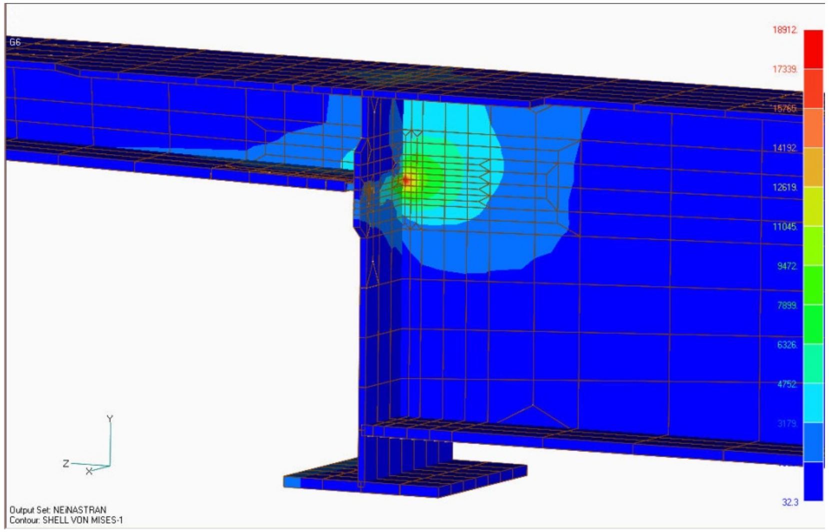


Global Local Modeling can be used

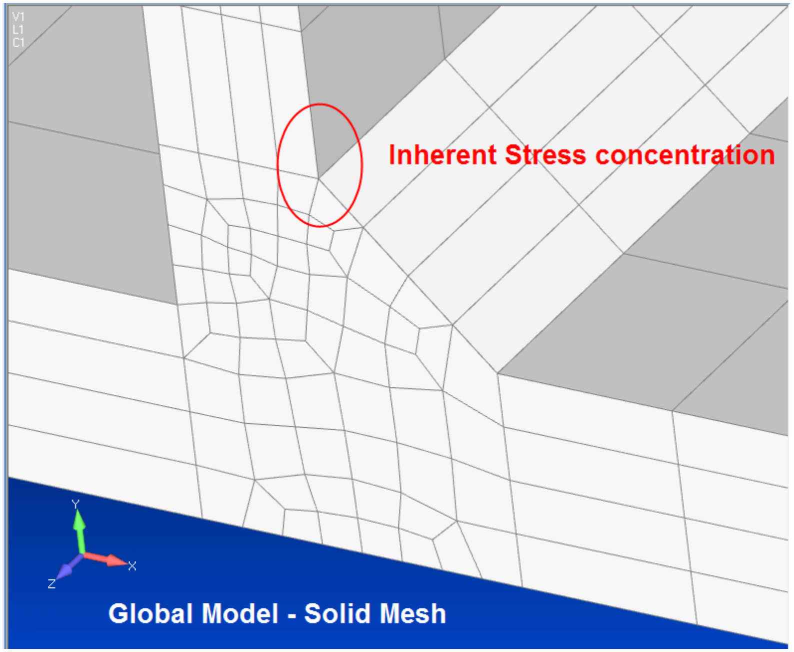
Why Not Use 20 Million Elements?

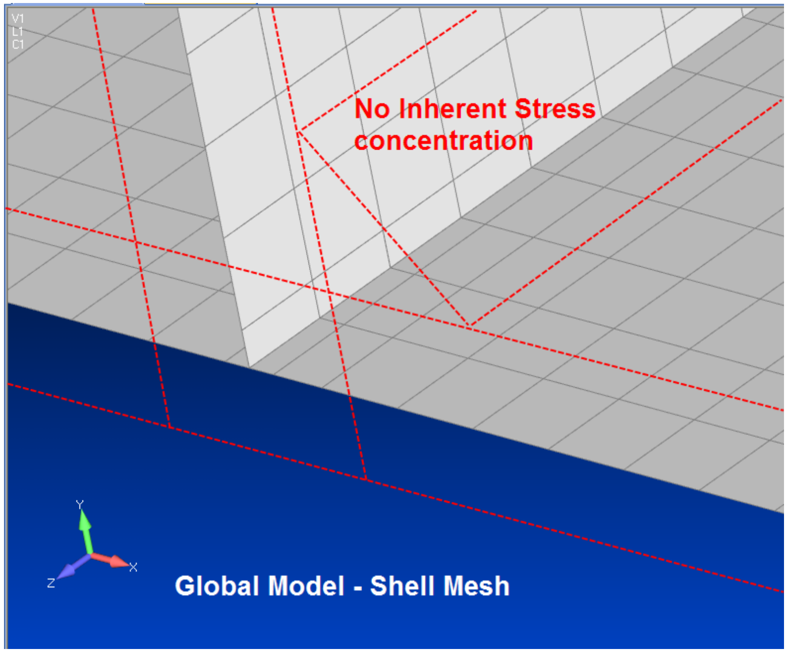


Local detail models use boundary conditions from whole vessel models



Why Not Use 20 Million Elements?





Sometimes FEA will give problematic results – here there will always be a singularity

- Solid model includes general feature stress concentration effect, need to back them out
- Shell model has no stress concentration effect
- Can apply hand calculations or certification rules to this feature

Why Not Use 20 Million Elements?

Class Topics taking this further

- Estimating CPU time and resources
- “Smearing”, rigid elements, and other tricks
- Plane strain and Plane Stress
- Global Local and substructure Modeling
- Symmetry
- Stress singularities and Stress concentrations
- Trends and Predictions in FEA technology

Practical Advice for FEA of Your Design

Agenda (continued)

- Why not use 20 million elements
  – *let the computer take the strain?*

- Real world boundary conditions and loading
  – *some examples of good and bad modeling*

- Anticipate the load paths
  – *examples of how to produce free body diagrams to use as a sanity check on your models*

- Checking the results
  – *FEA is guilty until proven innocent*

Real World Boundary Conditions and Loading

Boundary conditions:

Any structure is always attached to a neighboring part somehow (except for free flight!)

In a real structure, no such thing as:

- a fully rigid connection
- a simply supported connection
- a point support

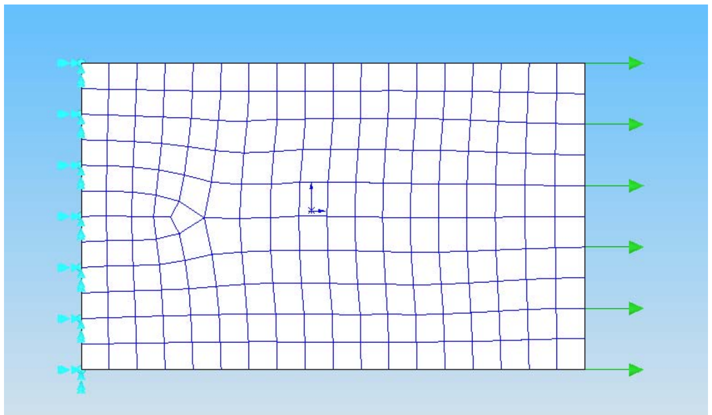
Examples:

- Car dashboard
- Deck of a ship
- Poisson's ratio in a plate

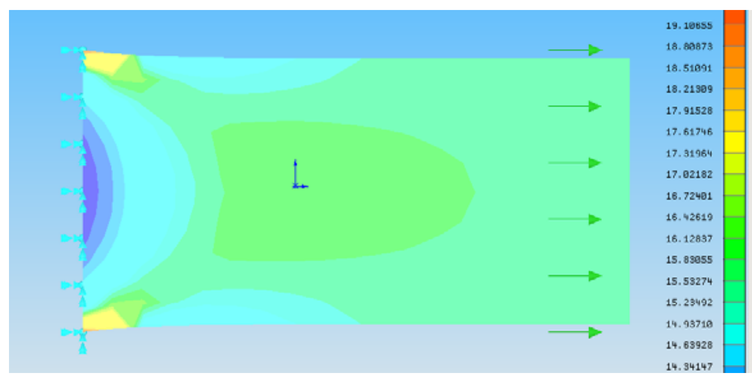
Consider upper and lower bounds
Use local boundary structure

- Detailed
- Simplified via spring stiffness

Real World Boundary Conditions and Loading

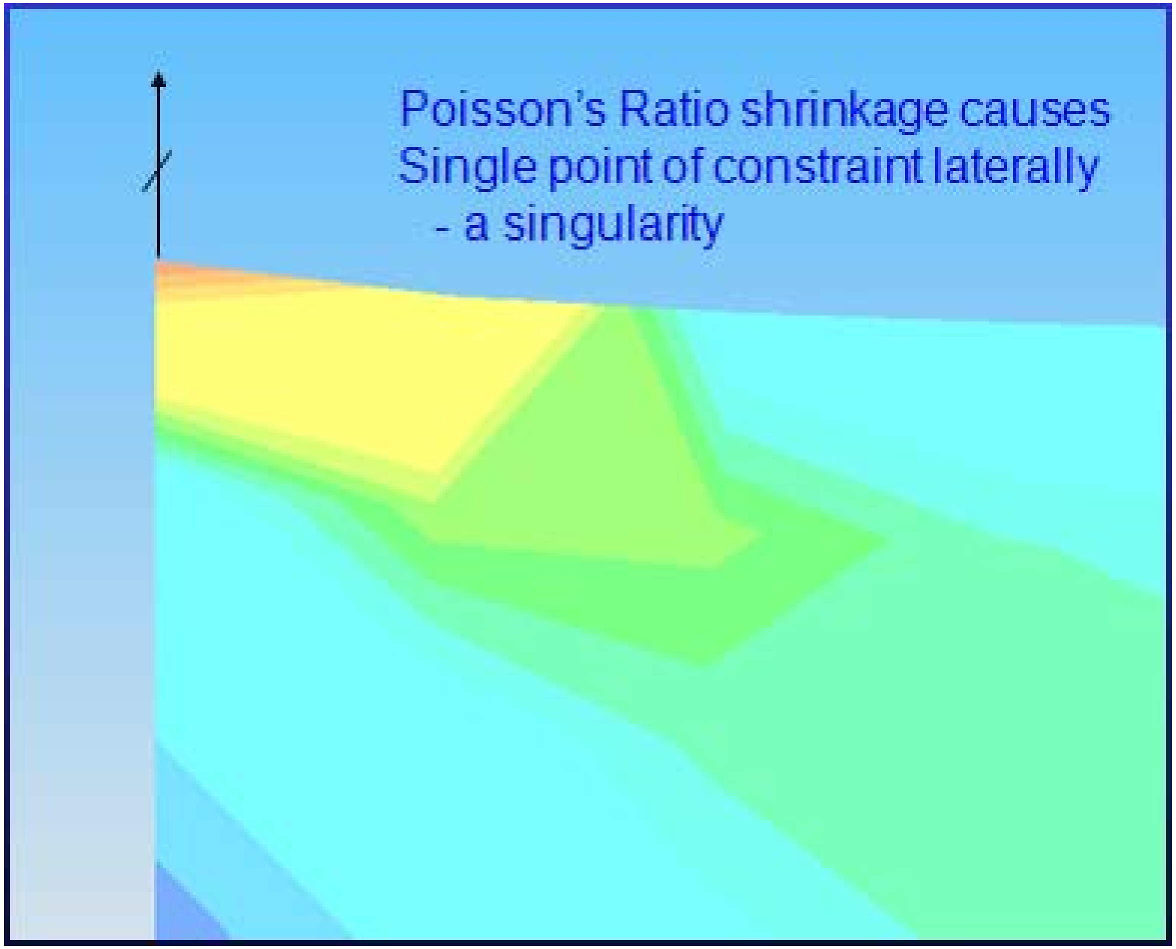


Simple tension coupon

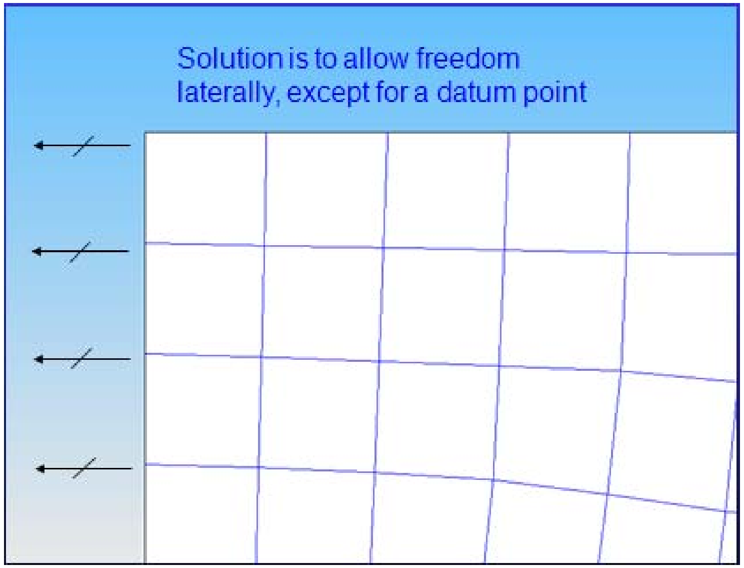


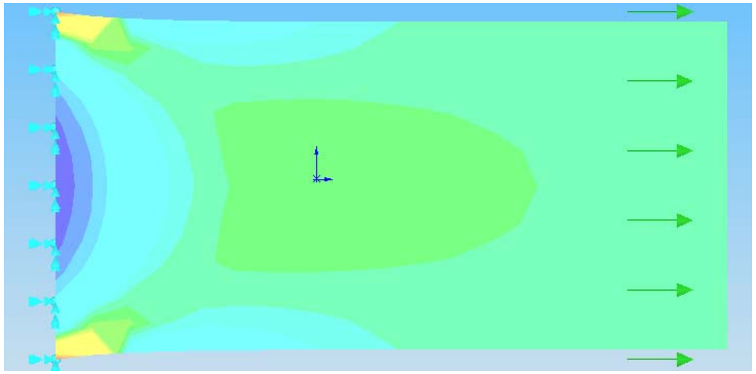
Non uniform stress

Real World Boundary Conditions and Loading



Real World Boundary Conditions and Loading

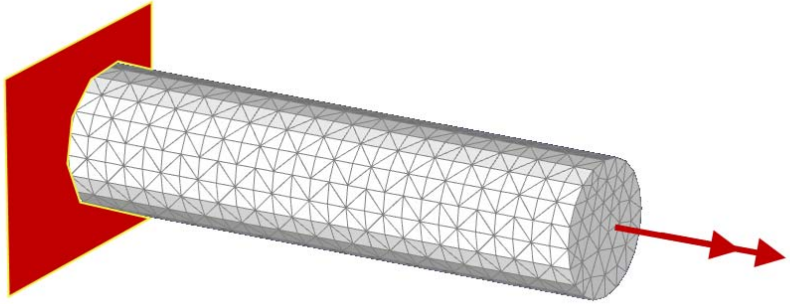




Real World Boundary Conditions and Loading

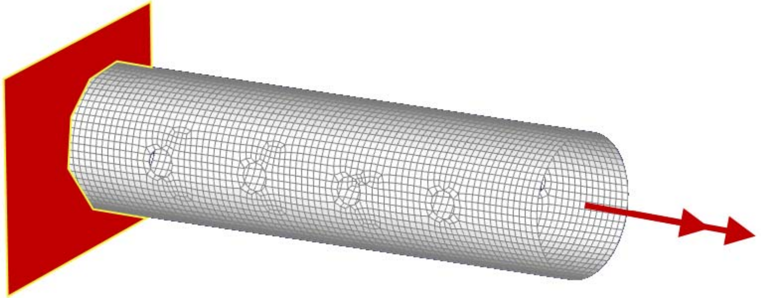
Solid Shaft under torsion

Fully fixed at end



Fuselage under torsion

Fully fixed at end



Is either doing a good enough job?

Real World Boundary Conditions and Loading

Loading:

How is the part loaded in practice

In a real structure, no such thing as:

- a point load loading

Spread the load over a ‘pad’

Pressure load, inertia loads

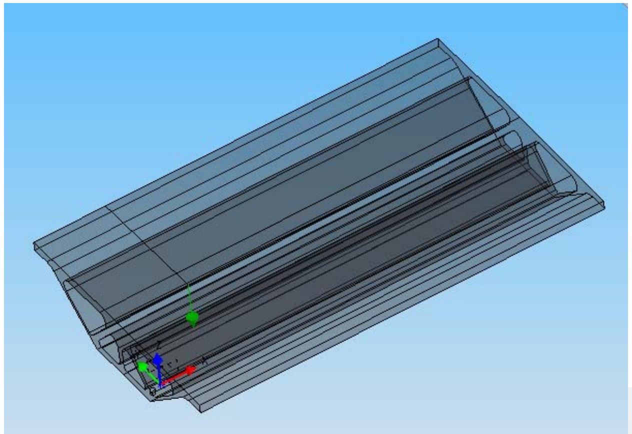
- Need to be checked very carefully

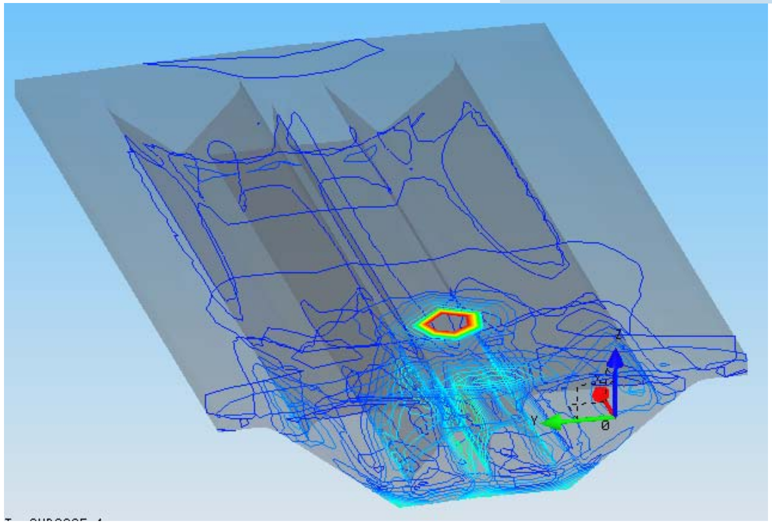
Nonlinear -

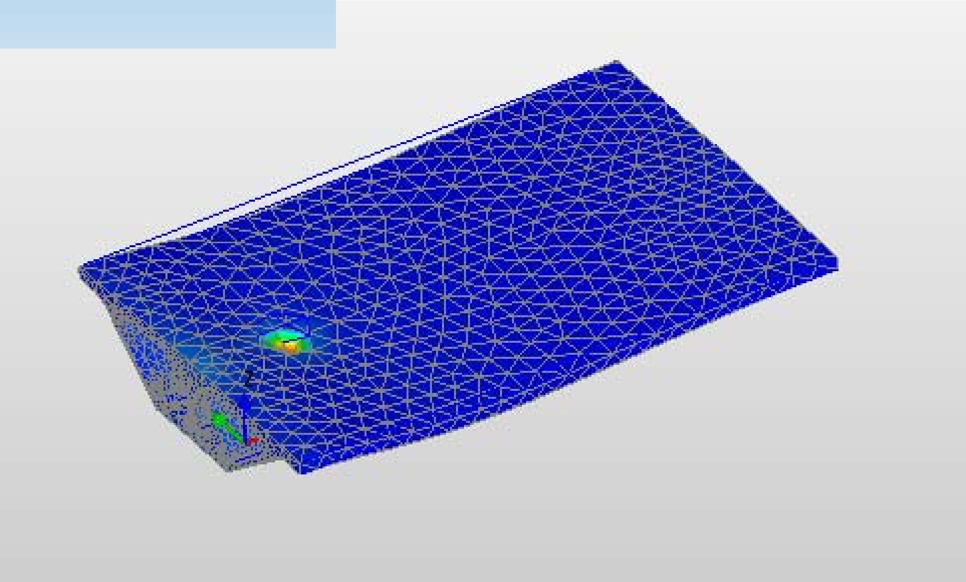
- Fixed or a follower load
- Is a displacement driven solution better (like a tensile test)

Real World Boundary Conditions and Loading

Point Load on Bridge

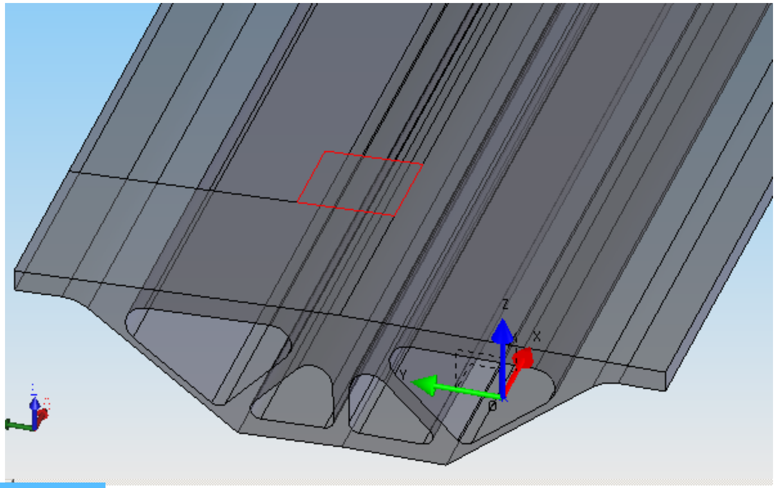


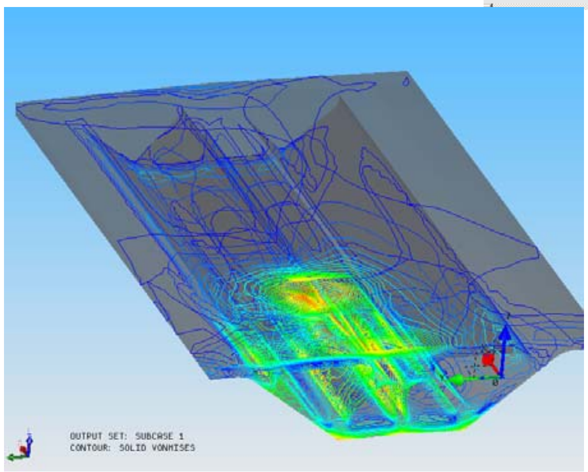


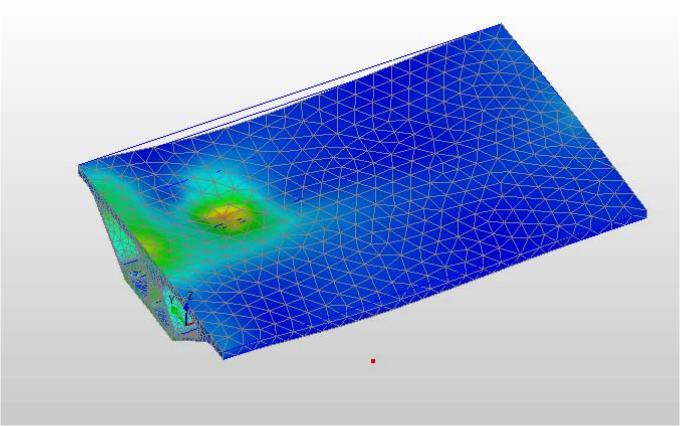


Real World Boundary Conditions and Loading

Pad Load on Bridge

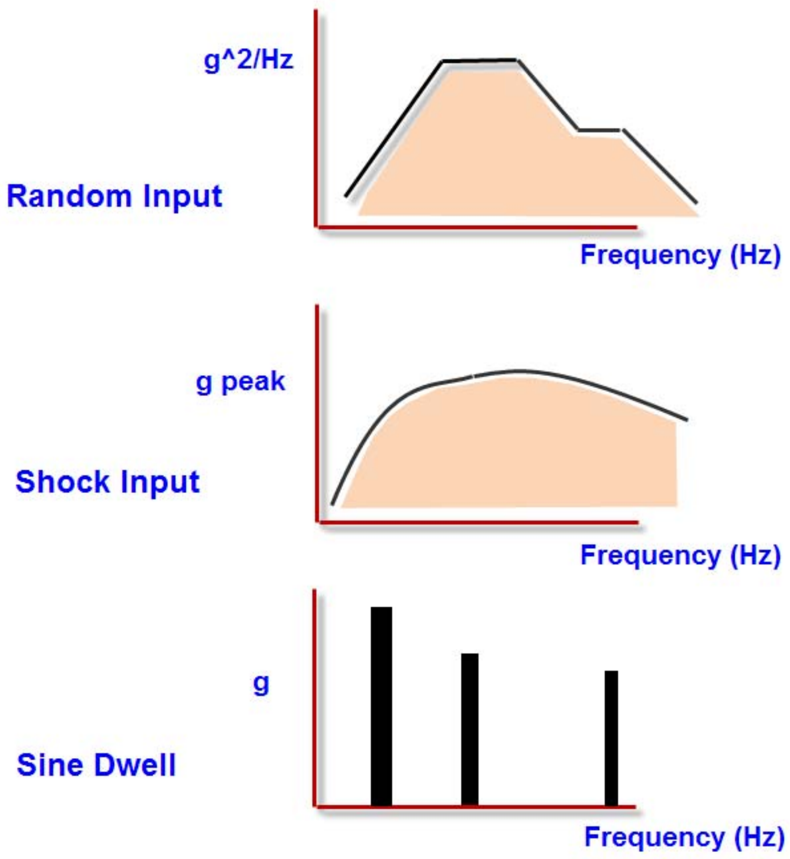


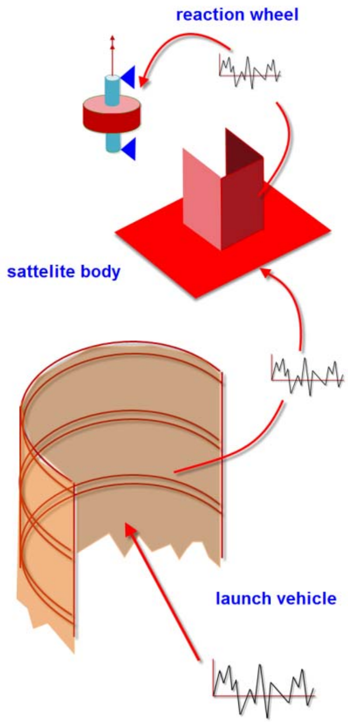




Real World Boundary Conditions and Loading

The environment





Real World Boundary Conditions and Loading

Class Topics taking this further

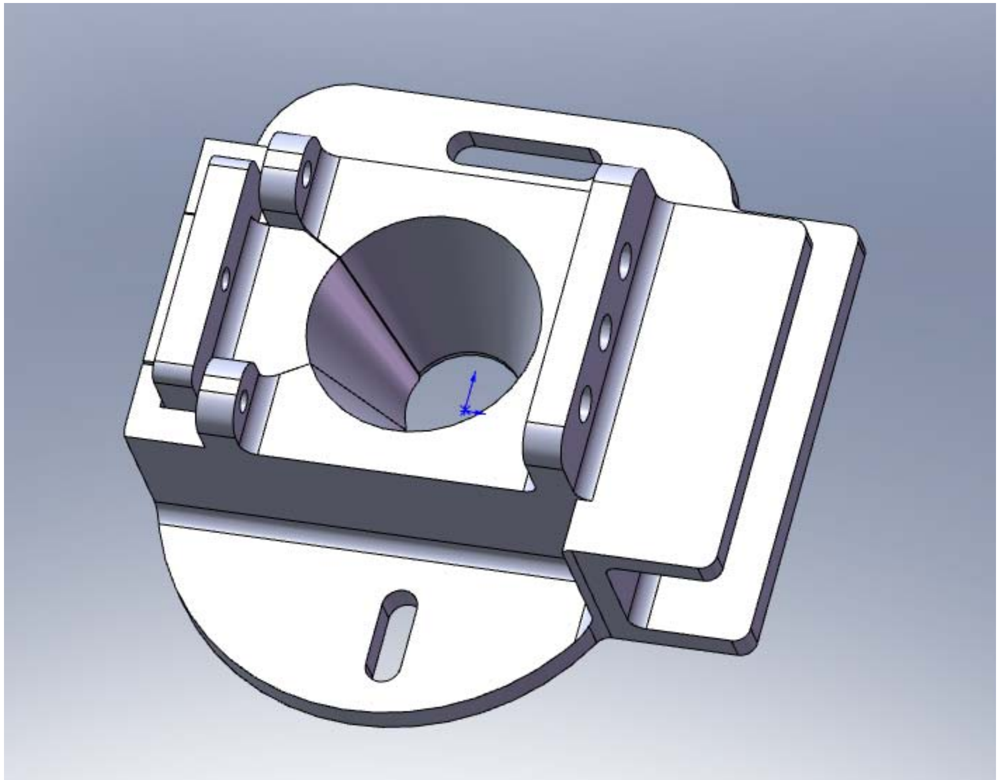
- Live Demos of bad loads and boundary conditions
- Case studies and discussion of real world lbcs
- Checking Loads and boundary conditions
- Equivalent loading systems
- Bolt loading methods
- Minimum constraint sets – avoid over constraining
- Unit load case approach

Practical Advice for FEA of Your Design

Agenda (continued)

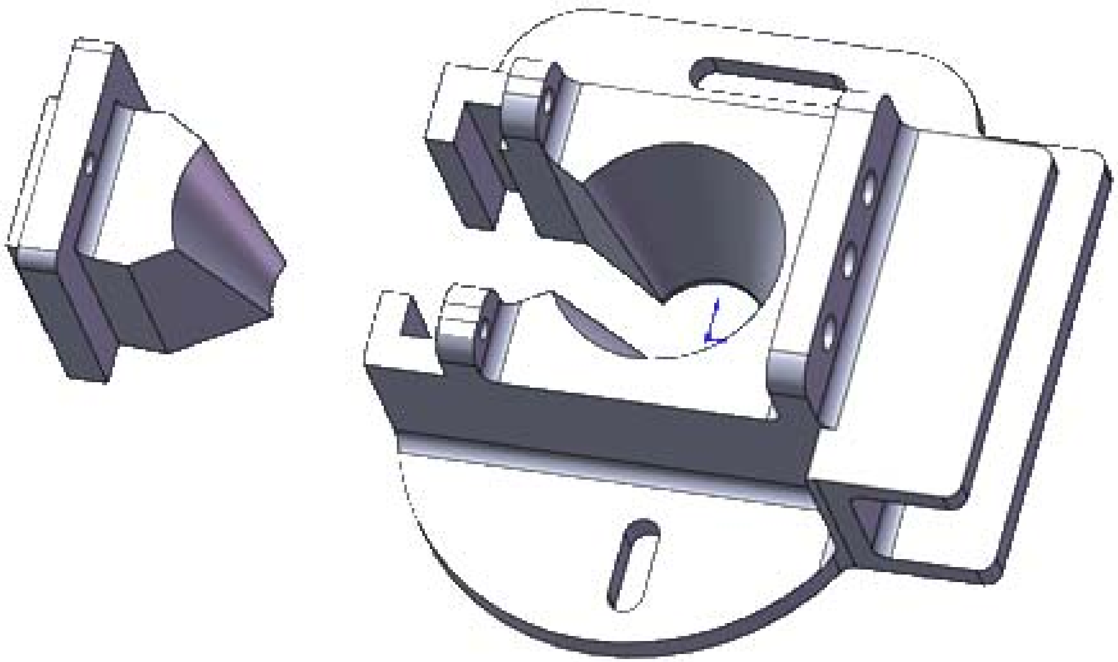
- Why not use 20 million elements
  – let the computer take the strain?
- Real world boundary conditions and loading
  – some examples of good and bad modeling
- Anticipate the load paths
  – examples of how to produce free body diagrams to use as a sanity check on your models
- Checking the results
  – FEA is guilty until proven innocent

Anticipate the Load Paths



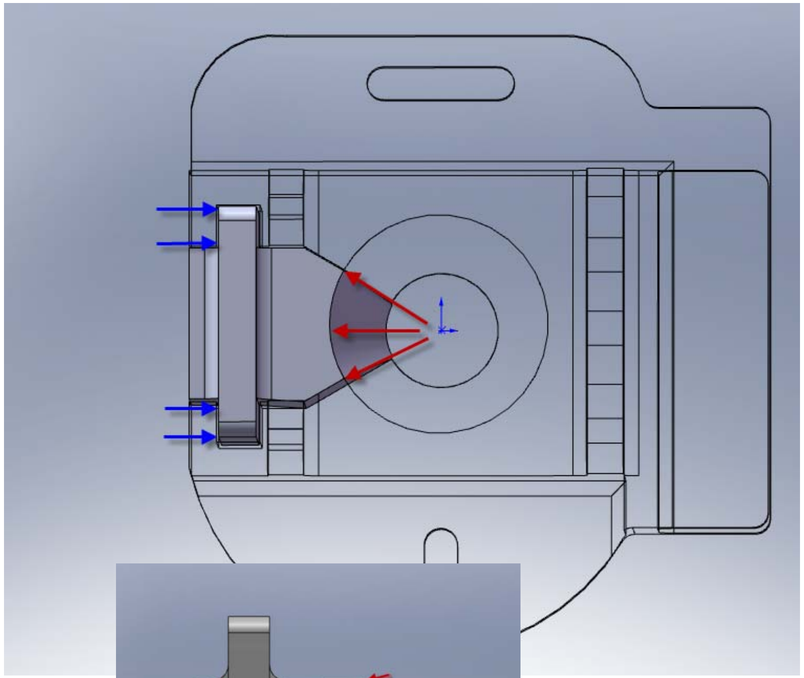
Nozzle location gear assembly – two parts

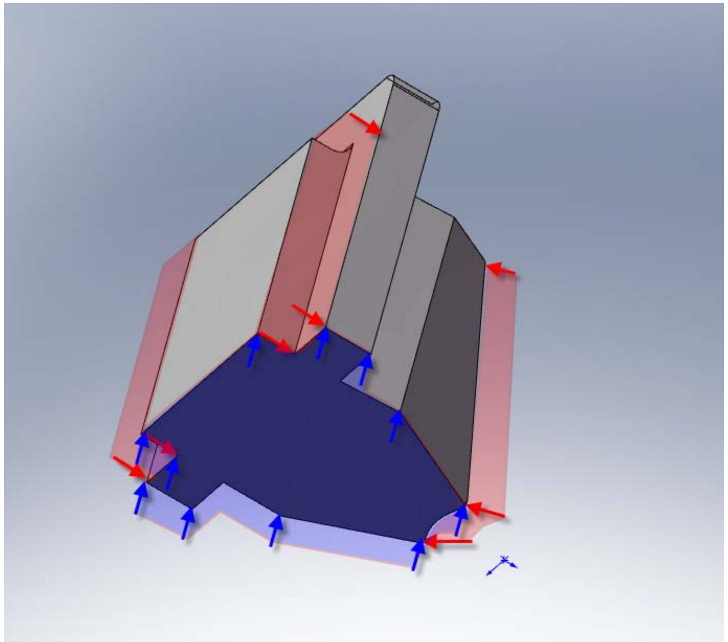
Anticipate the Load Paths

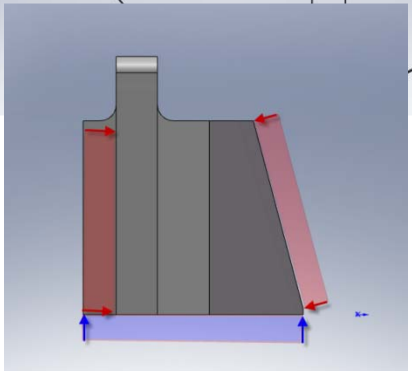


Nozzle location gear assembly – exploded parts

Anticipate the Load Paths



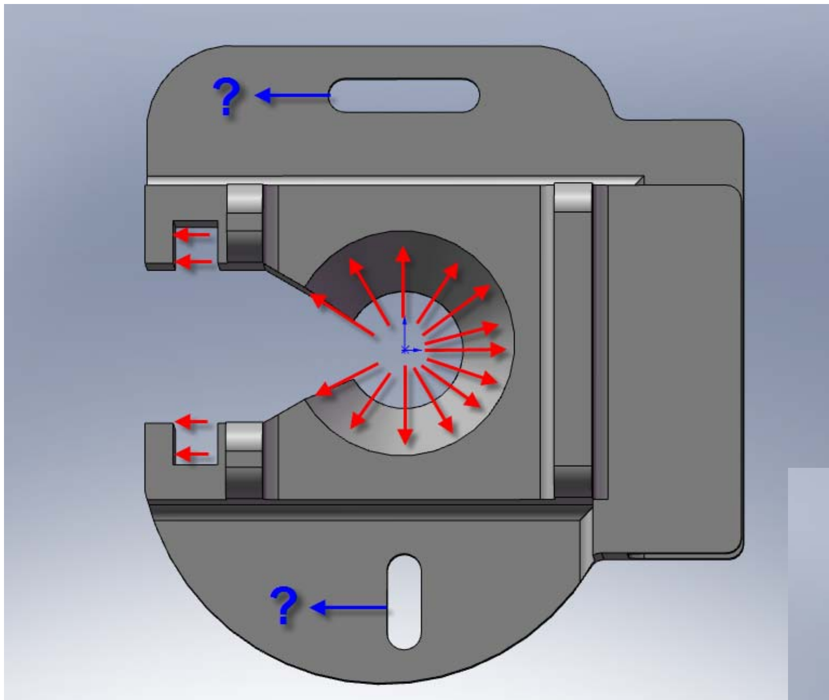




Wedge part – free body diagram

All 6 Degrees of Freedom must balance
Variation of vertical reaction throughout base
Assumptions?

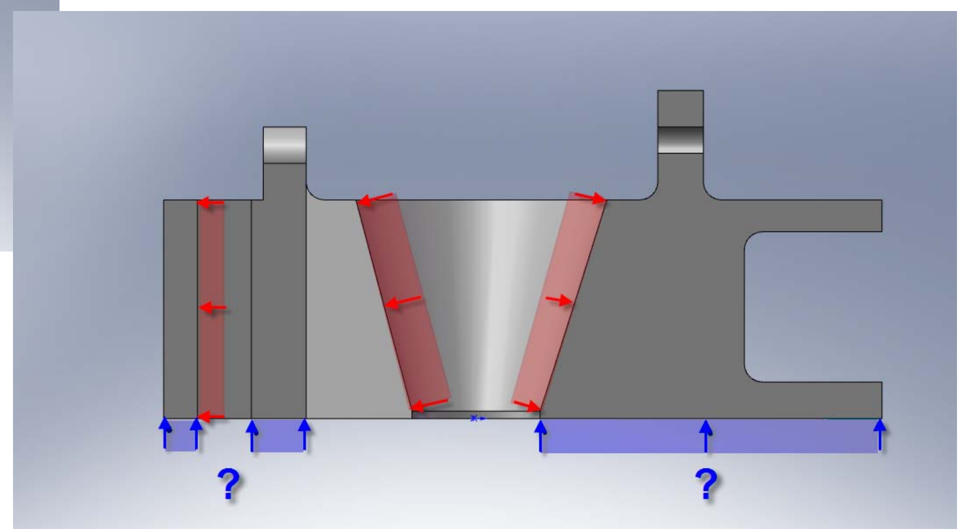
Anticipate the Load Paths



Housing Part – free body diagram

All 6 Degrees of Freedom must balance

Assumptions?



Lateral balance of Housing very sensitive to bolt locations

Variation of vertical reaction throughout base

Anticipate the Load Paths

Class Topics taking this further

- Principals of equilibrium
- Free body diagrams
- Load path evaluation
- Case studies

Practical Advice for FEA of Your Design

Agenda (continued)

- Why not use 20 million elements
  – let the computer take the strain?
- Real world boundary conditions and loading
  – some examples of good and bad modeling
- Anticipate the load paths
  – examples of how to produce free body diagrams to use as a sanity check on your models
- Checking the results
  – FEA is guilty until proven innocent

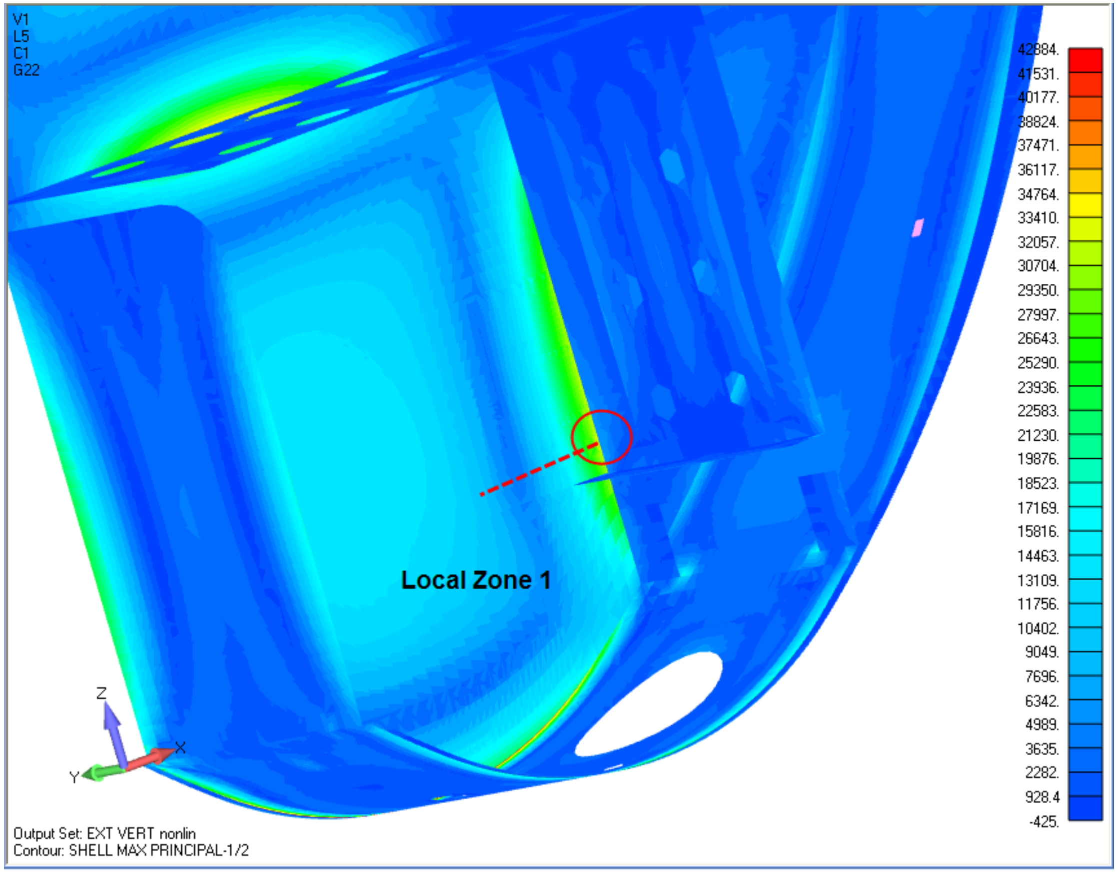
Checking the Results

Results:

- Check you have loaded correct results into the post processor
- Check reactions – balance loads?
- Check peak displacement and stresses – sensible?
- Use local coordinate systems for hoop, radial, directional
- Compare with hand calculations
- Check stress distributions – hot spots where you expect them – or surprises?
- Use Von Mises, Max and Min Principal and directional stresses
- Check animated displacements – any unusually stiff or flexible regions?
- Check stress gradients – switch off averaging (remember this is a displacement method)
- Check Epsilon for numerical stability, or whatever your solver uses
- Check against test, or previous analysis

Guilty until proven innocent!

Checking the Results



Area of high stress – note mesh refinement

Max principal used as primarily bending, opposite face also checked

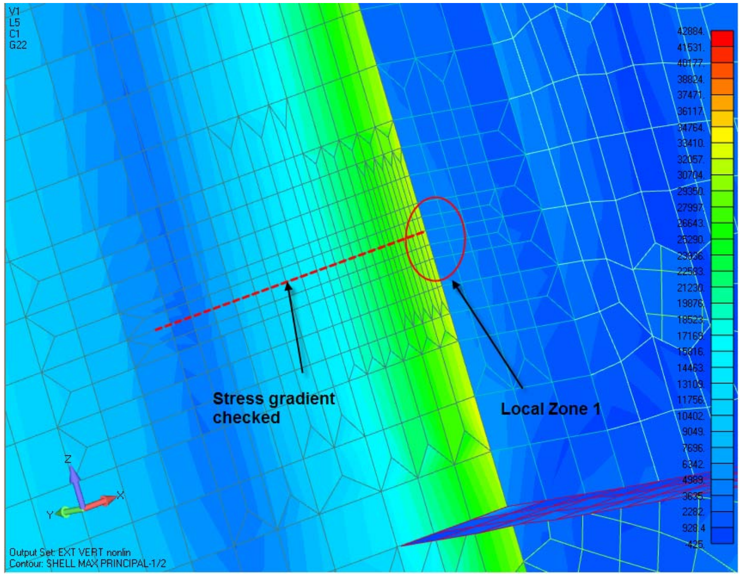
Checking the Results

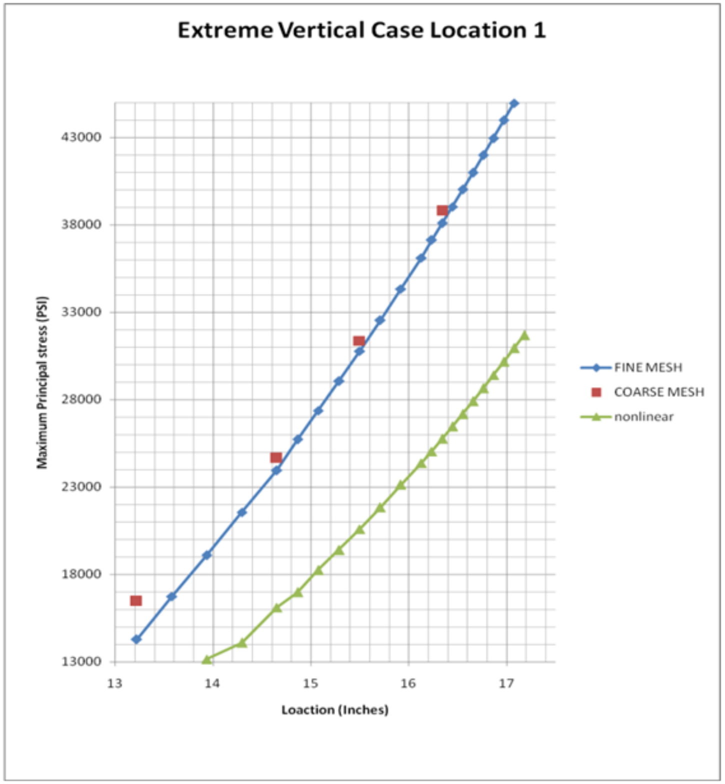
RECTANGULAR PLATE		**UNDER EXTREME VERTICAL CASE**					
				INERTIA			
TABLE 26. CASE 8A UNIFORM PRESSURE				Q AT TOP	0.9201	CONSTANT	0.9201
CONSTANT Q	3.1	PSI		Q AT BOTTOM	2.101	MAX	1.1809
				FACTOR	0.7		
PATE WIDTH	34.5	INCHES					
PLATE LENGTH	36.82	INCHES					
T	0.187	INCHES		**TABLE 26. CASE 8D UNIFORM DECREASING PRESSURE**			
Q AT MID HT	3.7441	(ADDS CONSTANT INERTIA TERM)					
BETA1	0.3078	FROM TABLE		BETA5	0.1686	FROM TABLE	
BETA2	0.1386	FROM TABLE		BETA3	0.0762	FROM TABLE	
MAX STRESS	-39225	PSI	AT 0.45 B	MAX STRESS	-4743.8		
CENTER STRESS	17663	PSI	AT 0.4 B	CENTER STRESS	9710.78		
TOTAL STRESS	**-43969**	PSI					

Roark equations plugged into Excel is a good sanity check

Takes some experience to judge ‘what fits’

Checking the Results





Using xy stress plotting to check stress gradient
Coarse mesh checked against fine mesh – convergence
Large Displacement analysis gives some alleviation (transition to membrane effects)

Checking the Results

Class Topics taking this further

- Review of Stress Types – direct, shear, bending
- Stress Components – Principals, Von Mises
- Typical FE Software Stress smoothing methods
- Gauss Point Stresses
- Limitations of linear analysis stresses

Conclusions

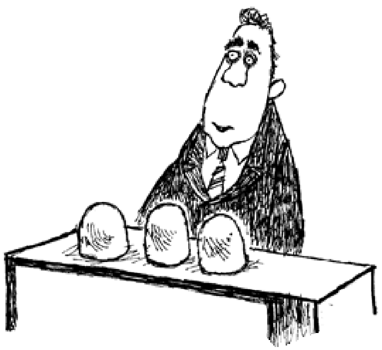
FE Analysis can be an important part of your design to manufacture route

- Increase understanding of performance
- Qualify designs
- Avoid extensive testing
- Right first time

But: It must be treated with caution

- Establish the analysis objectives
- Define your QA process
- Consider the physics involved
- Examine Load and Boundary Condition assumptions
- Simplify where possible
- Check resources available
- Verify results at least to an order of magnitude

Check and recheck



Q and A