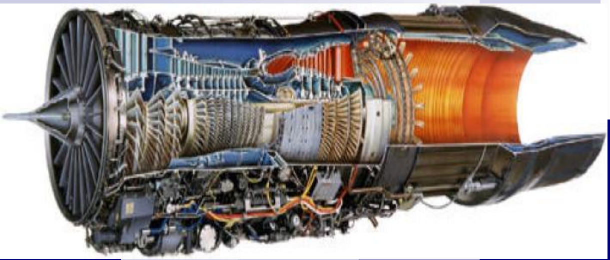
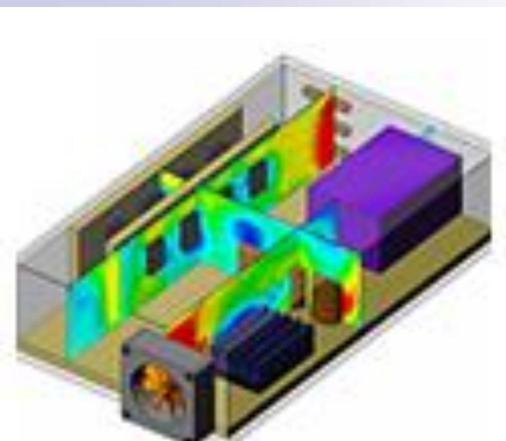
ADVANCED ENGINEERING SOLUTIONS Inc.

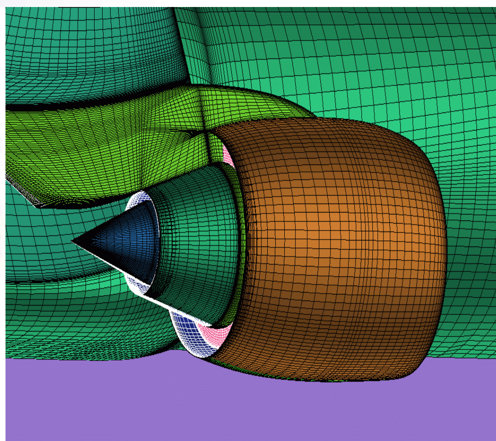
Innovative Steps to Excellence…



Engineering Solutions
Through CAD / CAE / CFD

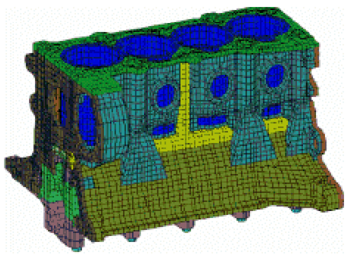
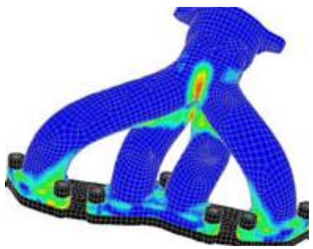
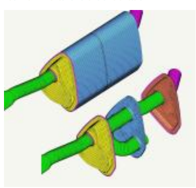
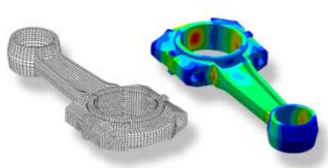
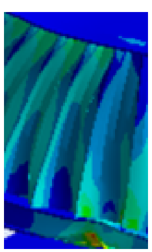
AES FEA Capabilities & Case Studies





AES FEA Competencies

- ANSYS for FE analysis
- ABAQUS for Non-linear Structural FEA
- Nastran
- LS-DYNA for Impact analysis
- HYPERMESH & ICEM CFD for meshing
- PRO/E, AUTOCAD, CATIA and UNIGRAPHICS for Design and Model

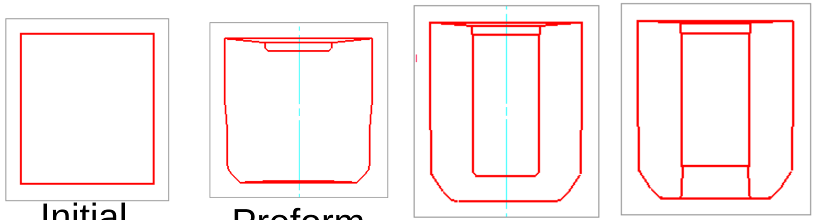

FEA Capability

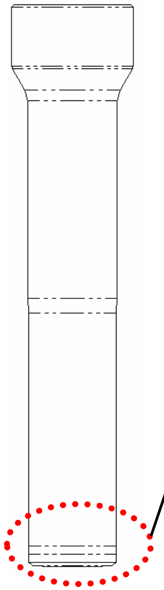
- Finite Element Analysis for Static & Dynamic
- Linear and Non-linear Analysis
- Steady state/ Transient Analysis
- Stress Analysis, Structural and Thermal analysis
- Crash and Impact analysis
- Fatigue and Durability analysis
- Plastic Deformation
- Creep Analysis
- Rotor Dynamics
- Vibration
- Fluid Flow Impact & Large Deflection
- Electro-Magnetic
- Impact Analysis
- Harmonic, Spectrum & Random analysis
- Transient Dynamic
- Coupled-physics involving acoustic, piezoelectric, thermo-structural
- Sub-model analysis and
- Design Optimization studies

Punch Design for Cold Forming Process

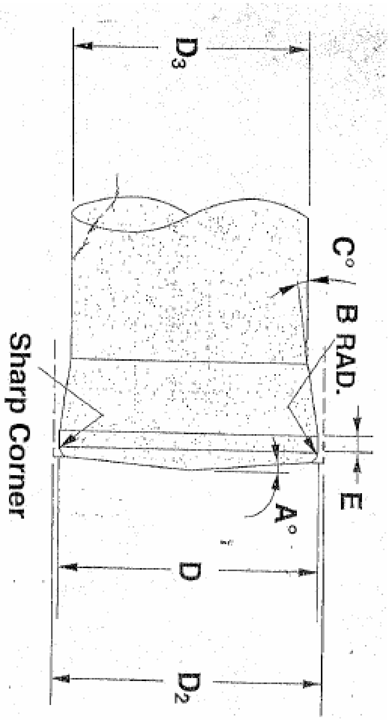
- Project goal
  - Simulate forward and backward non-linear metal forming simulation
  - Design extrusion punch – Nose Design

Hole making process





Back-extrude punch



Generic Nose Design

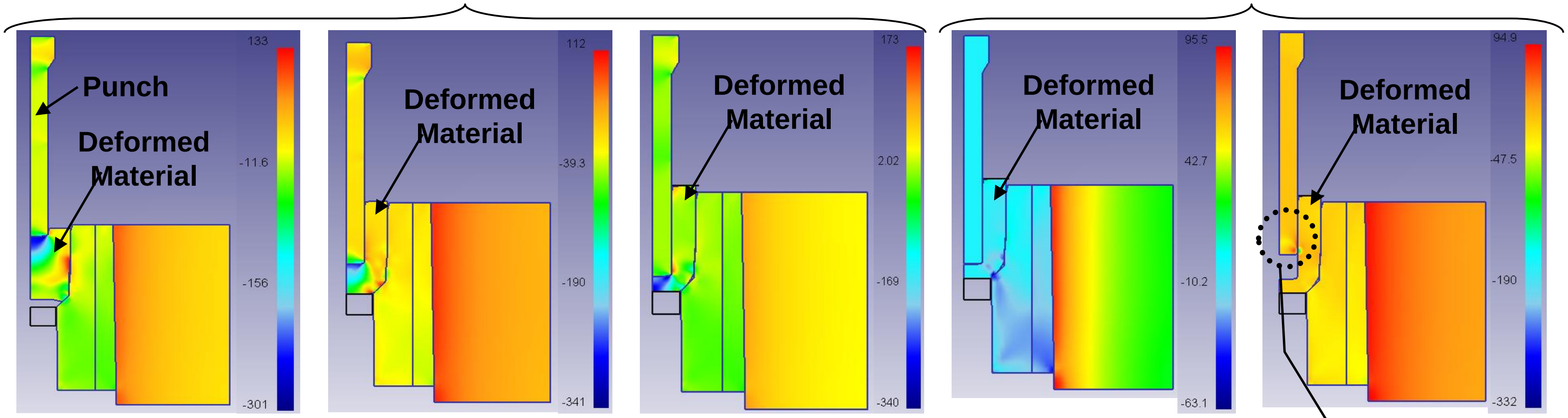
Challenges

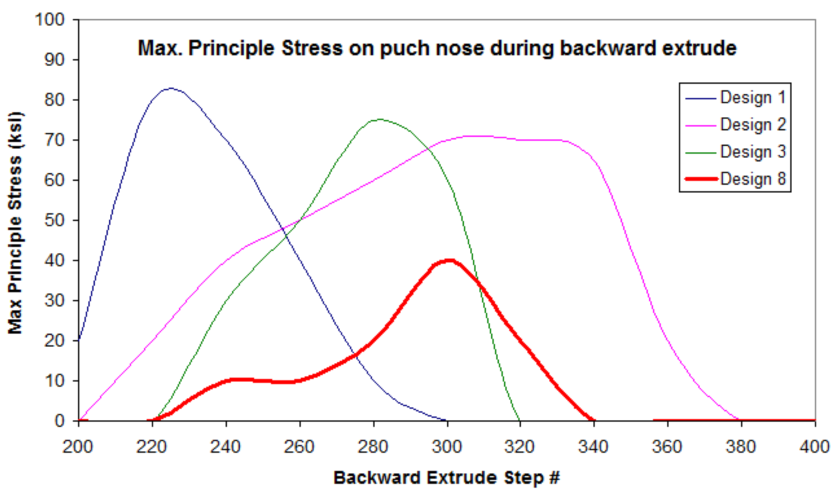
- Carbide punches can withstand very high compressive stress but are very poor under tensile stresses
- Punch nose design criteria's
  - Less wall friction to reduce forward compressive stress
  - Avoid material build up behind nose to reduce tensile stress while punch backing up

Results – Punch Nose Design

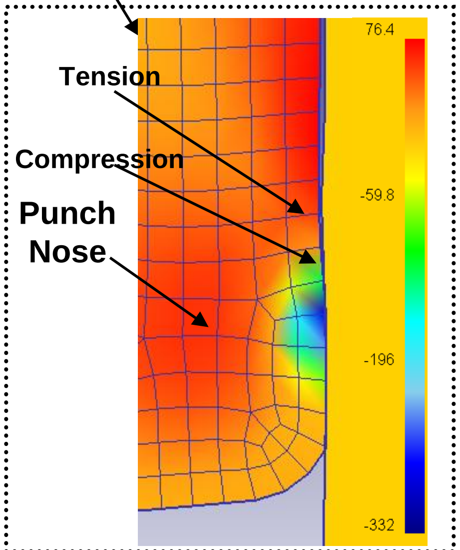
Forward Extrude Simulation

Backward Extrude Simulation



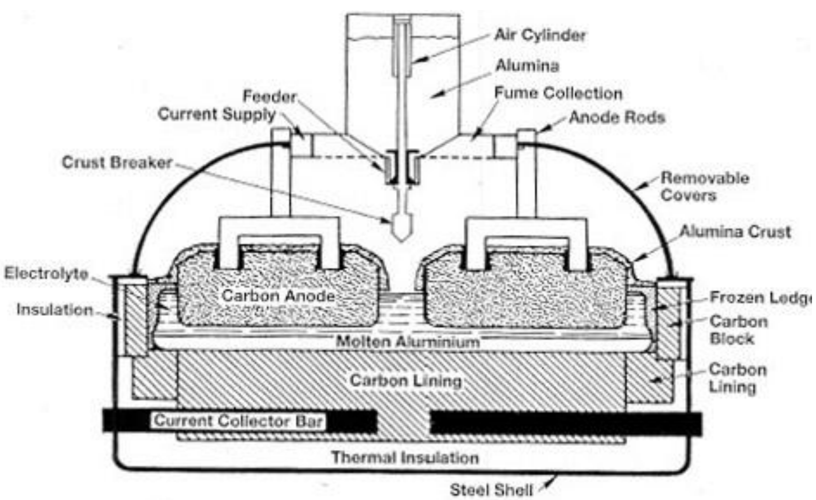


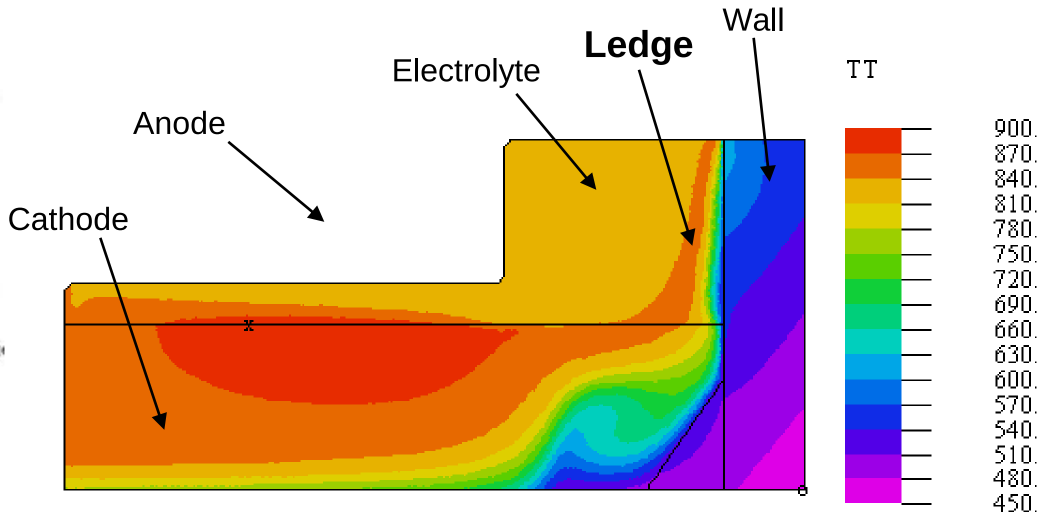
Successfully designed punch nose that has lower tensile stress during backward extrude



Multi-Physics Electro Thermal Analysis

- Project goal
  - Simulate Smelting Process – Electrolysis process for Aluminum Production
  - Find best process variables that produce desired temperature distribution

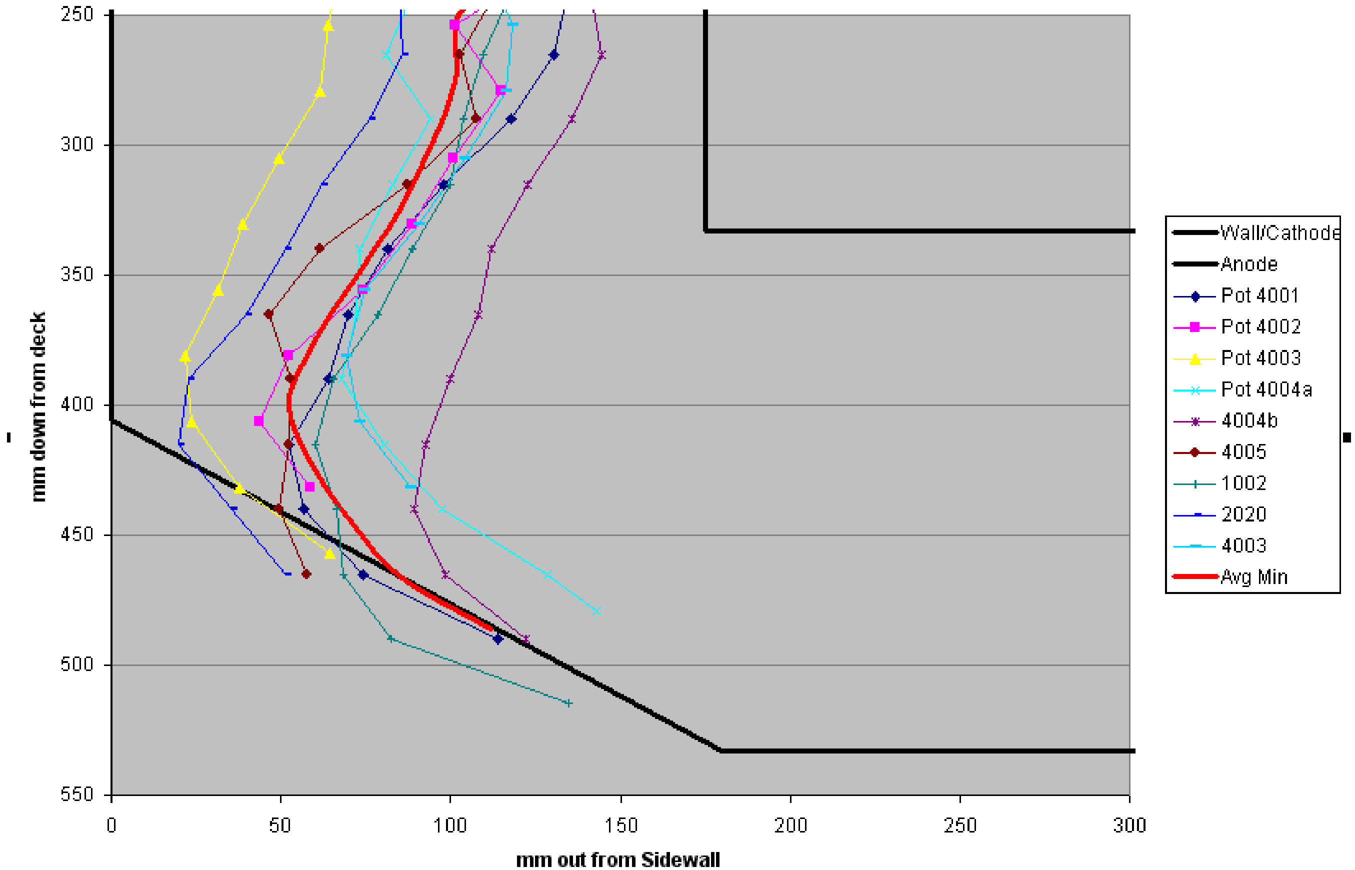




Temperature profiles showing temp. isotherms

- Heat produced = Elec. Resistance x I^2 x time

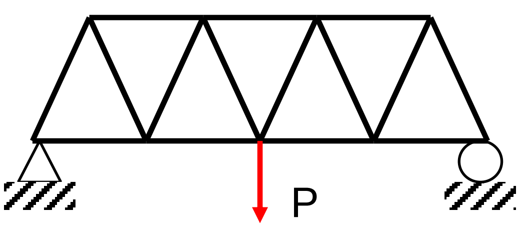
Results – Ledge Design

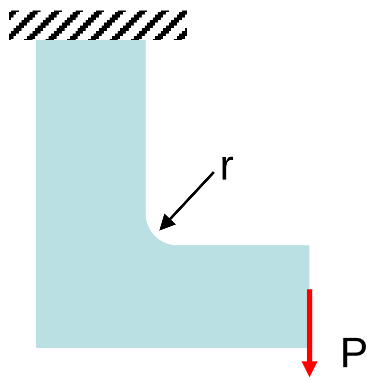


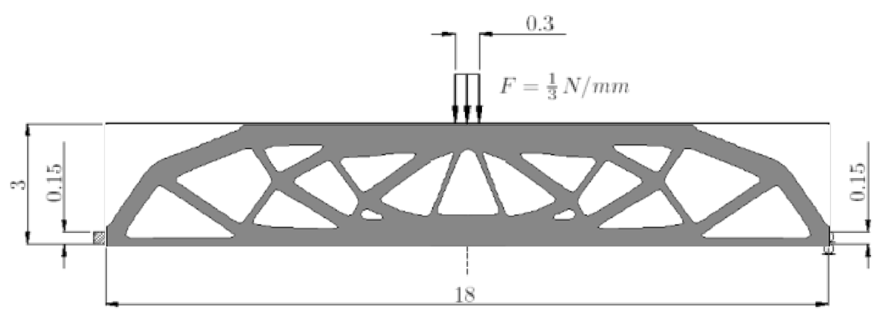
- Electrolyte below 840° C remains in a solid state protecting the walls of the “POT” from the highly corrosive electrolyte that is in liquid state

Design Optimization Capabilities

Type	Size	Shape	Number of holes	Design Variables	Complexity	Potential Savings
Size	Changes	Fixed	Fixed	Element thickness; cross-sectional area	Lower	Lower
Shape	Fixed	Changes	Fixed	Non-parametric: nodal coordinates Parametric: geometric parameters	Medium	Medium
Topology	Fixed	Changes	Changes	Element densities	Higher	Higher







Engine Front Mount Casting Design

Background

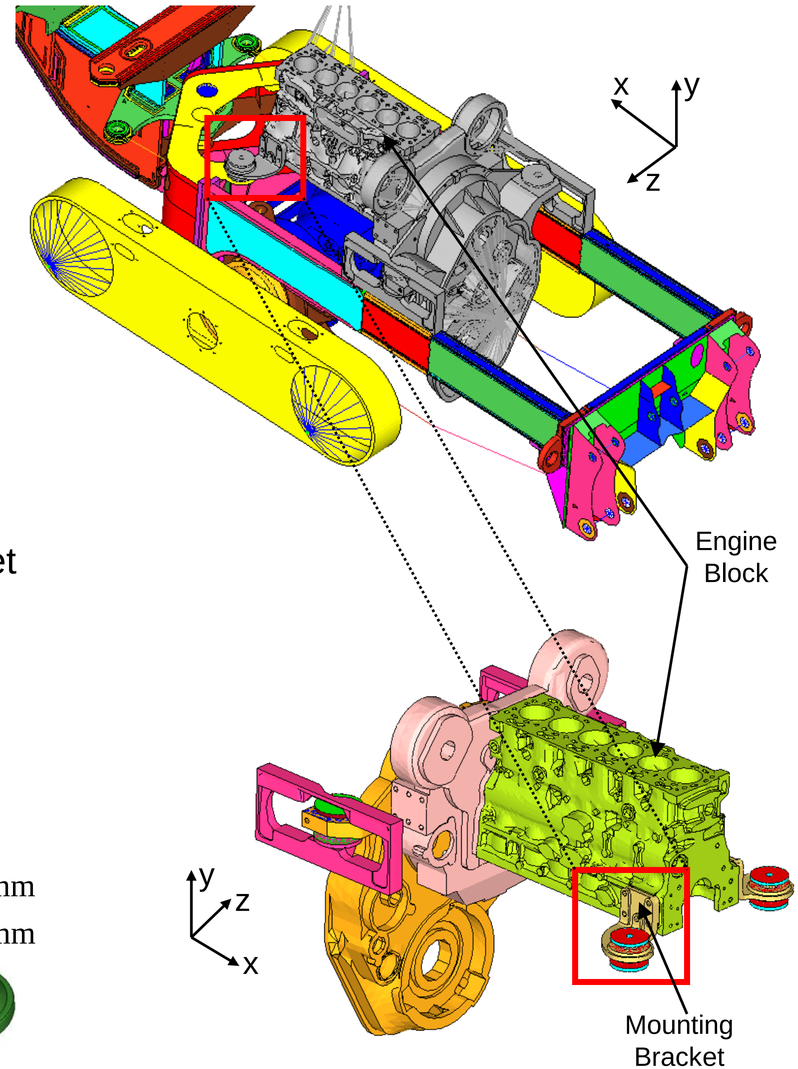
- Tier-IV Engine - additional 4” of overhang
- Modify bracket or Change frame

Objective

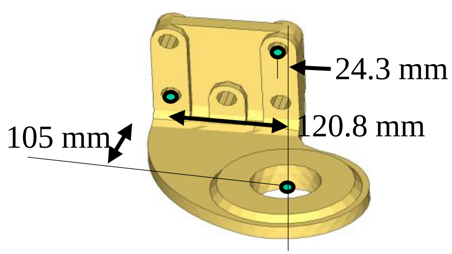
- Optimize the Tier-IV Engine bracket
- Minimize mass while improving performance

Constraints

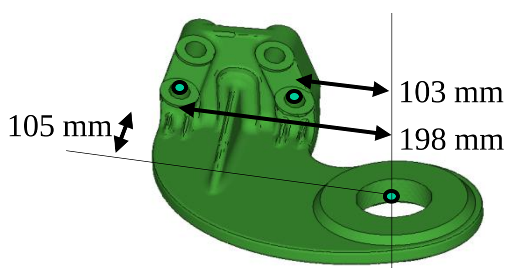
- Stress in Tier-IV bracket <= Tier-III bracket
- Stiffness of Tier-IV bracket >= Tier-III bracket
- Bolted Joint Performance of Tier-IV bracket >= Tier-III bracket



Tier 3 – Bracket

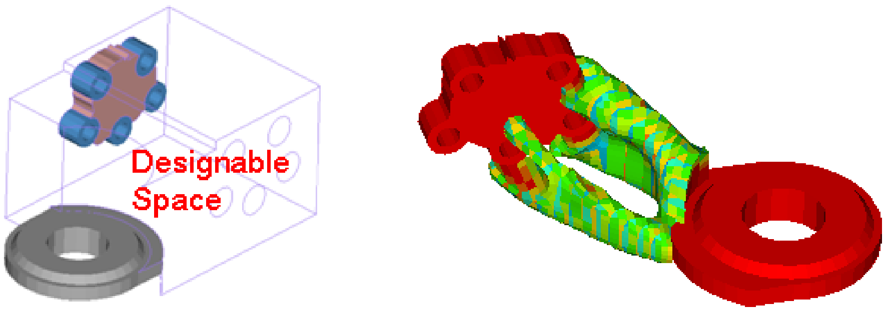


Tier 4 – Concept Bracket

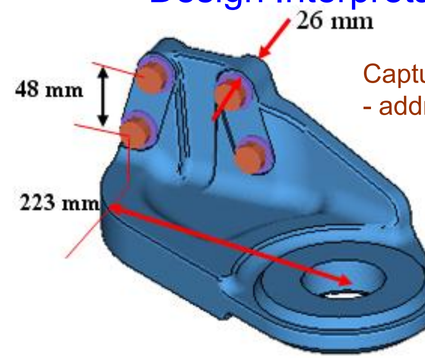


Design Process

Topology Optimization



Design Interpretation

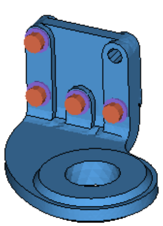


Captured Major Load paths
- address the 4” overhang

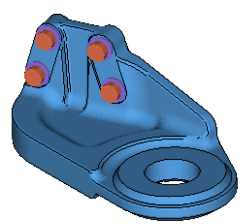
Fine Tune Design for Performance

Inertial loads in X, Y and Z directions – Stress Comparison
Stiffness of the bracket to meet Iso-Mounting requirements – Stiffness Ratio
Bolted Joint Analysis to evaluate integrity with 4” Overhang – Shear Capacity

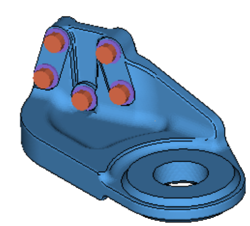
Tier III



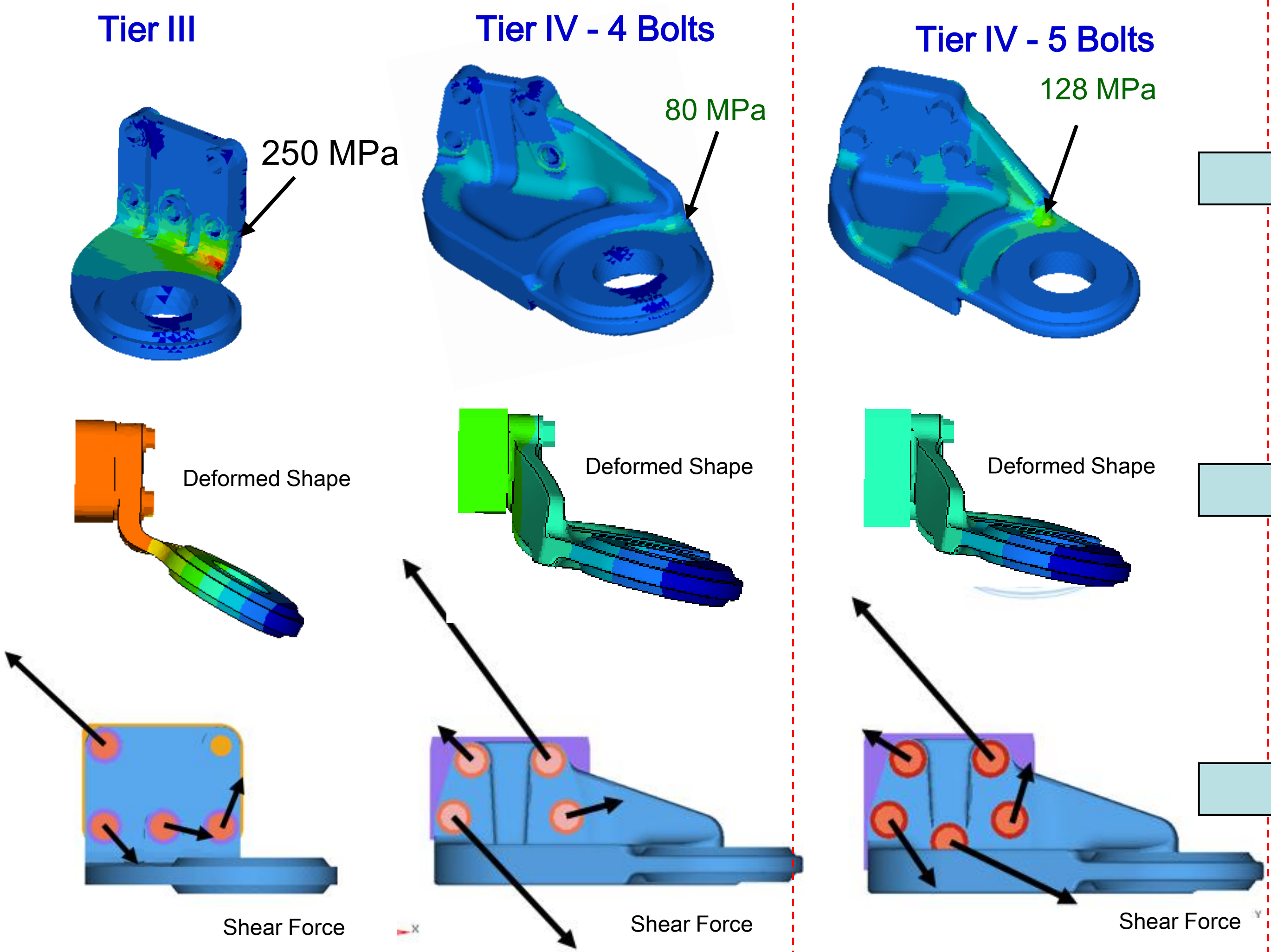
Tier IV - 4 Bolts



Tier IV - 5 Bolts



Structural Performance of Optimized Bracket



Prime Path

Stress ✔
Tier-IV Bracket <= Tier-III

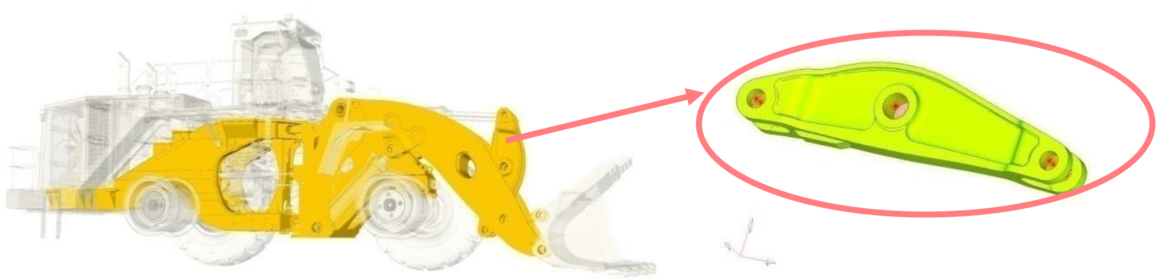
Stiffness ✔
Tier-IV Bracket <= Tier-III

Bracket	Bracket / Iso Mount Stiffness Ratio			Mass (Kg)
	X	Y	Z	
Baseline - Tier 3	170	3.7	262	5.9
Tier IV - 4 Bolt	279	7.2	121	9.2
Tier IV - 5 Bolt	180	6.3	99	9.5

Bolted Joint ✔
Tier-IV Bracket <= Tier-III

5th bolt is sharing some of the shear force with the 4th bolt and increasing the overall shear capacity of the joint

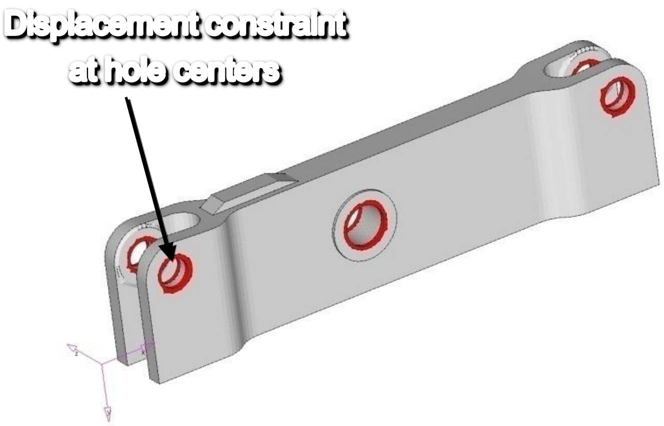
Tilt Lever Topology Optimization

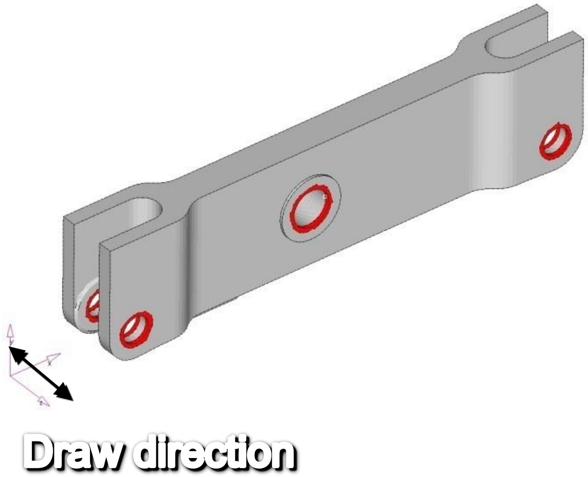


- Problem Statement: Complete an Optimization process to achieve 10,000 hours of life with minimum mass of the Tilt Lever.
  - Maximum Design Space for the Tilt Lever is provided in PRO/E part file format and NASTRAN input file.
  - Nine load cases and loading sequences for fatigue calculation is provided.
  - Symmetry is required with respect to Base_XY plane.
  - Fatigue curve for material is provided.
  - Material property is provided in NASTRAN/Optistruct input file.
  - Use 40% or less weight.

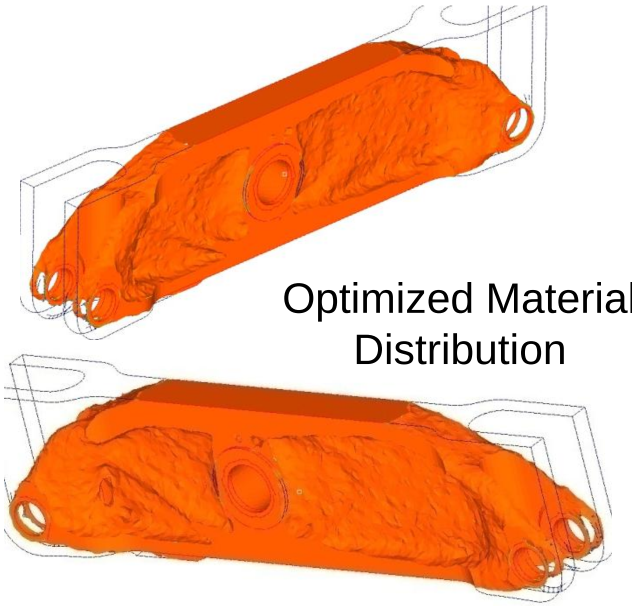
Topology Optimization Set-Up

- Define the maximum design space for the tilt lever as design variable in Optistruct.
- Define optimization responses:
  - Weighted Compliance, strain energy type of response for multiple load cases
  - Volumefrac, regional volume fraction response
- Define optimization constraint.
  - Volumefrac < 0.4
- Define optimization objective.
  - Minimize weighted compliance
- Define manufacturing constraint: Split Draw constraint in z-direction.

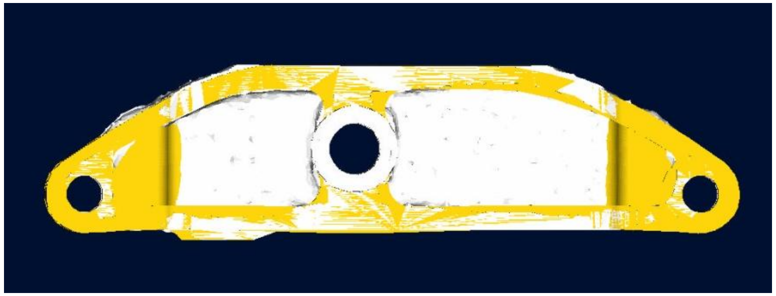


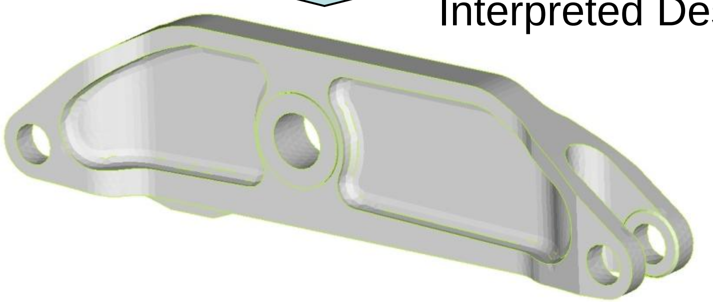


Topology Optimization Results

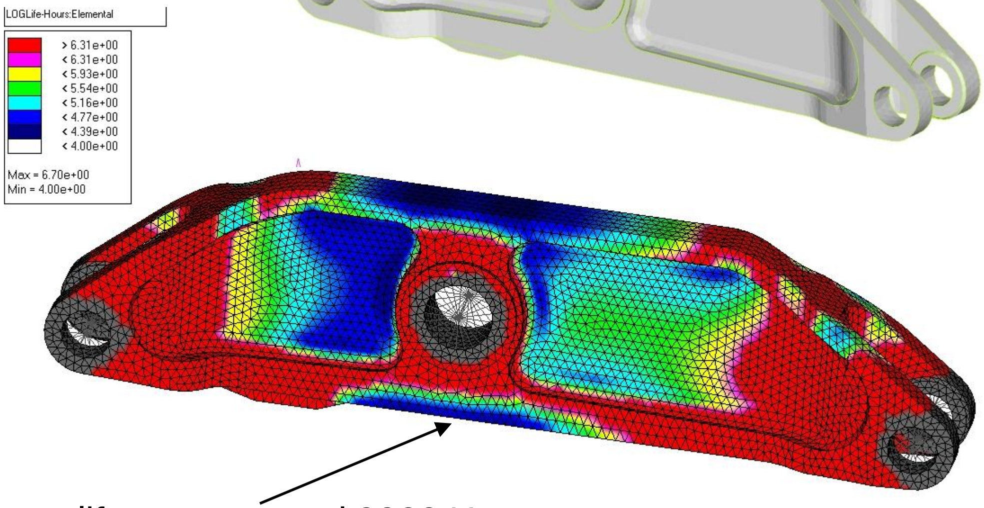


Optimized Material Distribution





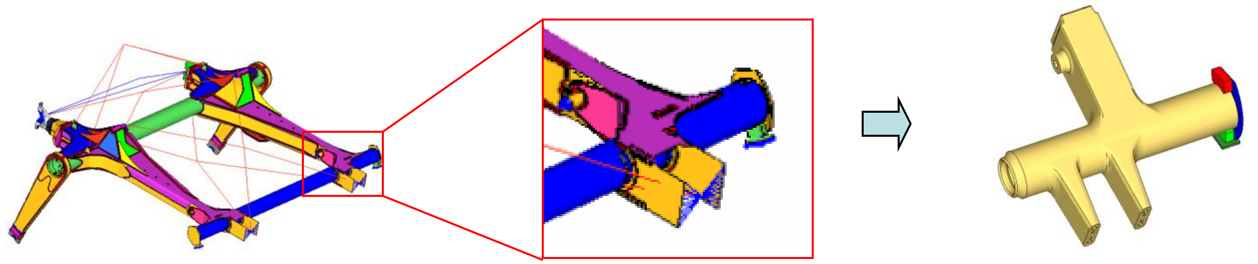
Interpreted Design



New Minimum life area, around 9900 Hrs

Casting Design – Size and Shape Optimization

- **Project goal**
  - Replace multiple overlaying welded sheet metal with a single casting
  - Reduce parent material stresses and weld stresses
  - Minimize weight of the casting

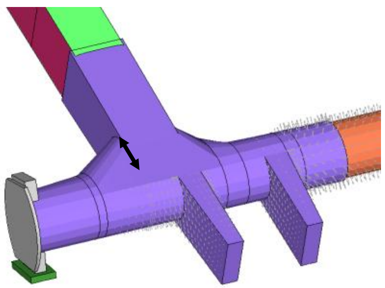


Multiple sheet metals

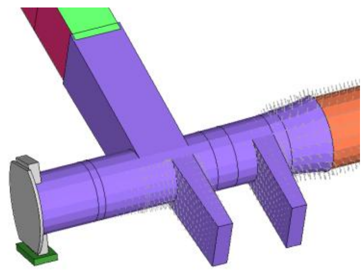
Single casting

- Model was provided in Pro-E format.
- Meshing was done in Hypermesh and Abaqus was used for the dynamic impact analysis
- Automation, Design trade off studies and Optimization was done in I-Sight

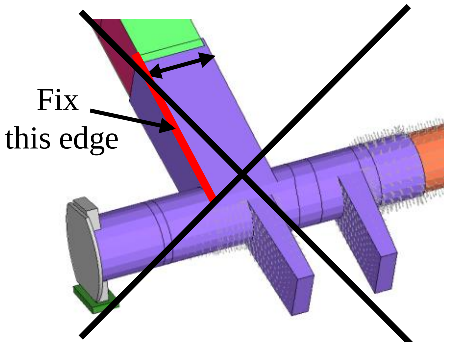
Shape and Size Design Variables



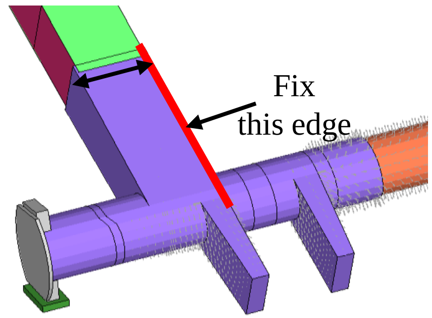
Variable 1: Radius Shape (Fore Aft-Cross tube)



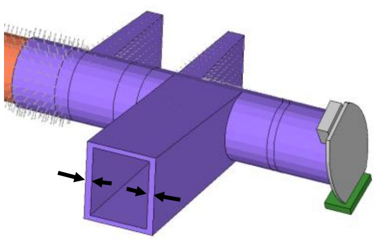
Variable 2: Lower Cross Tube Diameter



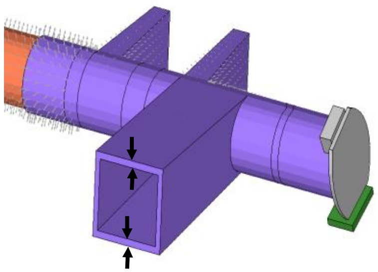
Variable 3: Fore Aft width



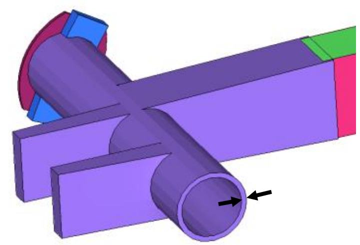
Variable 4: Fore Aft width



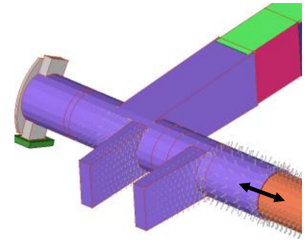
Variable 5: Casting Side Wall Thickness



Variable 6: Casting top/bottom Wall Thickness



Variable 7: Casting Lower Cross Tube Thickness

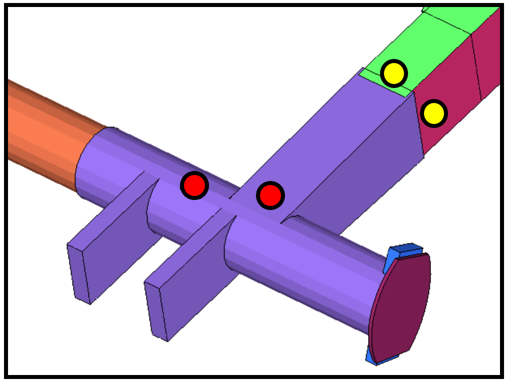


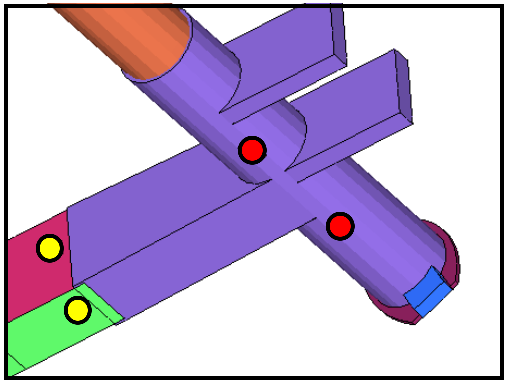
Variable 8: Casting Lower Cross Tube Length

- Variable 9 - Fore Aft Top Bottom Cover Plates
- Variable 10 - Fore Aft Side Cover Plates
- Variable 11 - Lower Cross Tube Thickness

Optimization Setup

- Objective
  - Minimize Weight of entire frame
- Subject to
  - Stress at weld location < 200 Mpa
  - Stress on Casting < 500 MPa

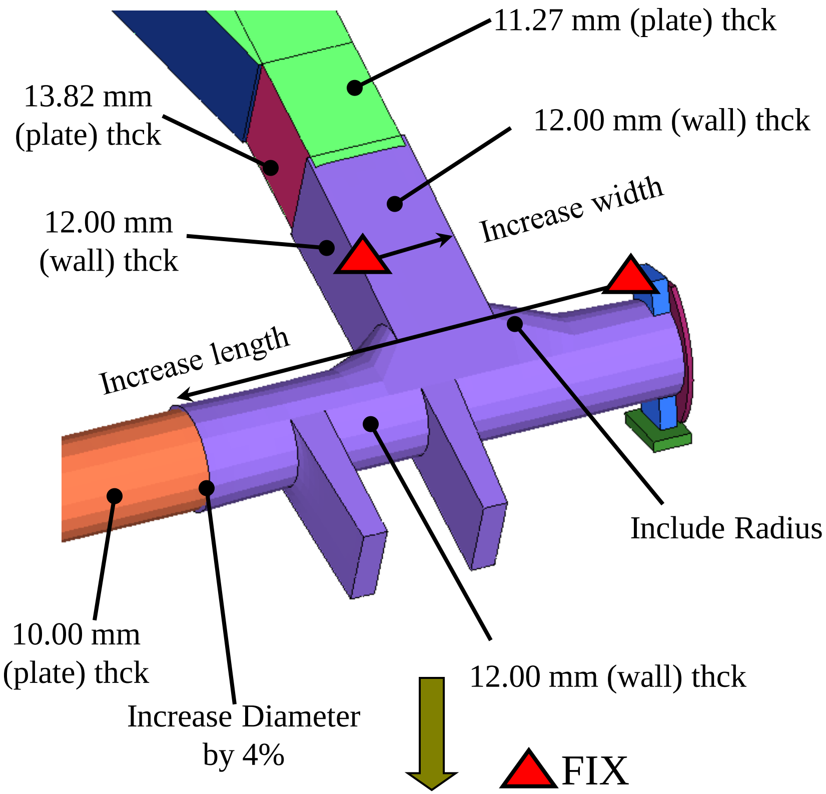




Optimized Results

Variables	Concept 1 Baseline(mm)	Optimized (mm)
Casting Side Wall Thickness	*12.70*	12.00
Casting Top-Bottom Wall Thickness	*12.70*	12.00
ForeAft-Crosstube Radius (shape)	*0.00*	9.09
Casting Cross tube thickness	*12.70*	12.00
Castin Lower Cross tube Length	*887.00*	910.16
Casting ForeAft width	*151.00*	159.18
ForeAft Side Plate Thickness	*12.00*	13.82
ForeAft Top-Bottom Plate Thickness	*5.00*	11.27
Lower Cross Tube Thickness	*12.70*	10.00
Lower Cross Tube Diameter	*163.50*	171.74

Response	Original	Optimized
Mass (of frame) (Tons)	*3.737*	**4.01957**



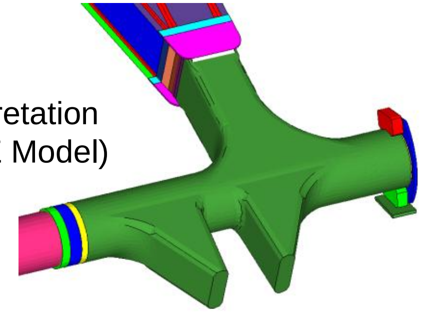
Stress on Cast part

Response	Original	Optimized
element 257 (Mpa)	*417*	**273**
element 266 (Mpa)	*408*	**424**
element 267 (Mpa)	*447*	**396**
element 468 (Mpa)	*473*	**385**

Stress at Weld Locations

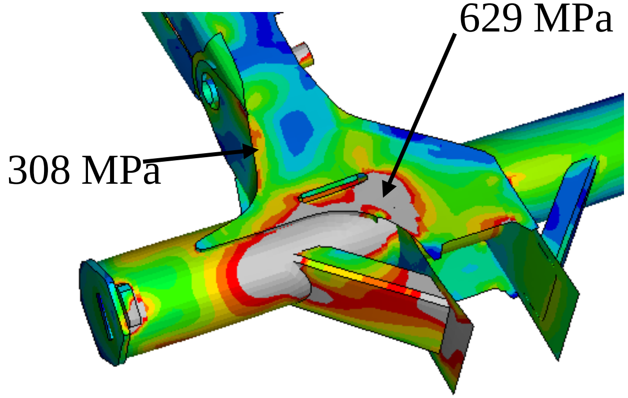
Response	Original	Optimized
element 211 (Mpa)	*205*	**204**
element 212 (Mpa)	*361*	**188**
element 213 (Mpa)	*87*	**73**
element 214 (Mpa)	*205*	**123**

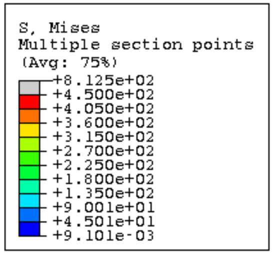
Interpretation (Pro-E Model)



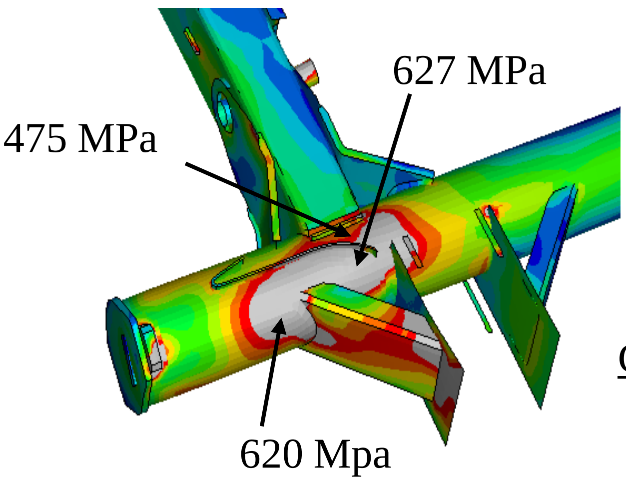
Modified Concept Performance Evaluation

With Wrapper Plate

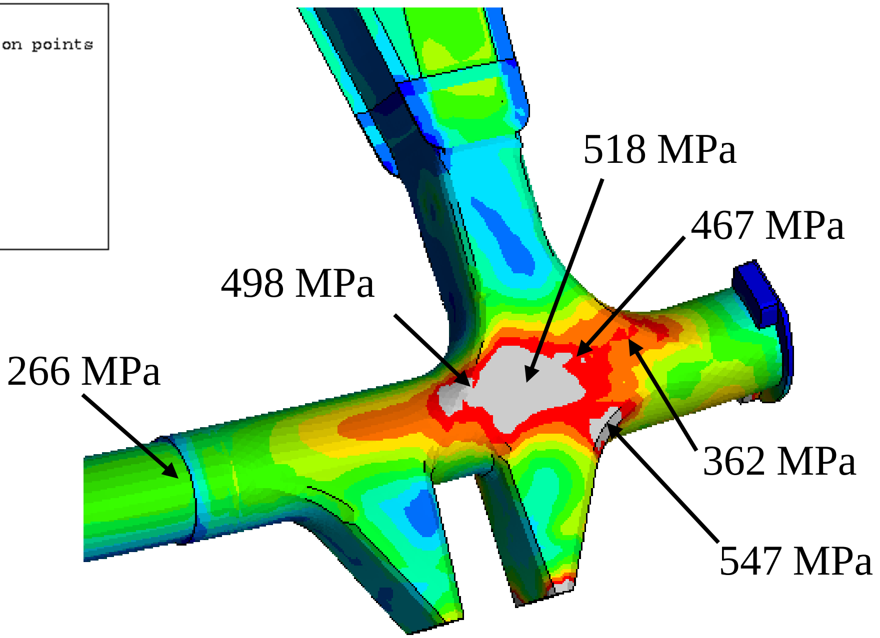




Without Wrapper Plate



Current Production



Concept 2

- All stresses are the max stress during the entire time history

AES - Finite Element Analysis - Capabilities on Oil and Gas structures

Static Structural FEA Simulations

* Oil pipe Pressure application analysis.
* Oil and Gas, Offshore structures Nonlinear Contact Simulation.
* Fatigue life estimation analysis.
* Fracture Simulations of Offshore structure base and pipes.
* Offshore structures Thermal and Thermo structural simulations.

Dynamic FEA Simulations

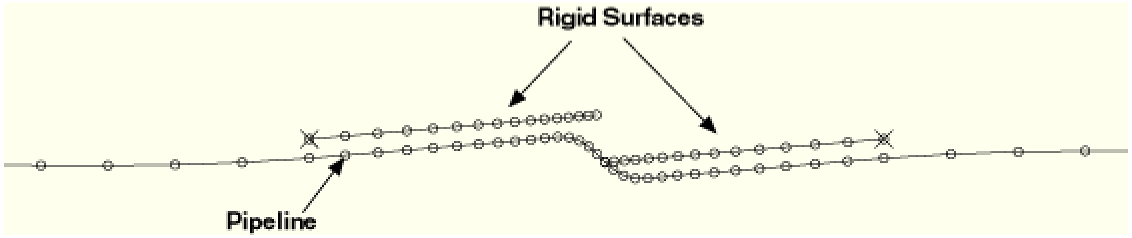
* Oil pipes and Offshore structures frequency analysis.
* Random Vibration analysis of Oil and Gas structures.
* Dynamic Explicit simulations of oil and gas structures.
* Impact simulations.
* Seismic analysis.

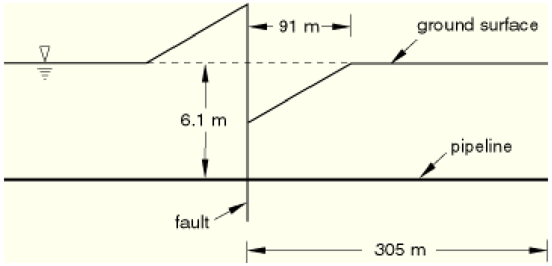
Few Case Studies on problems related to oil and gas structures FEA is presented in oncoming slides.

Finite Element Analysis of Pipeline buried in Soil

Problem Description:

Oil and gas pipelines are usually buried in the ground to provide protection and support. Buried pipelines may experience significant loading as a result of relative displacements of the ground along their length. Such large ground movement can be caused by faulting, landslides, slope failures, and seismic activity.

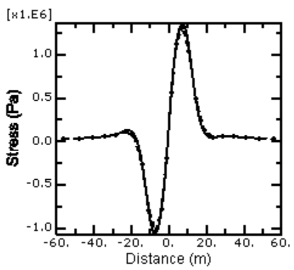


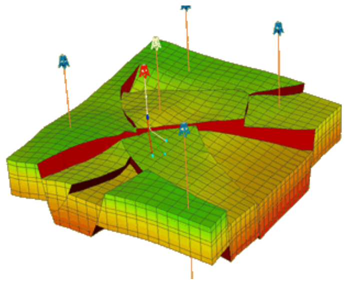


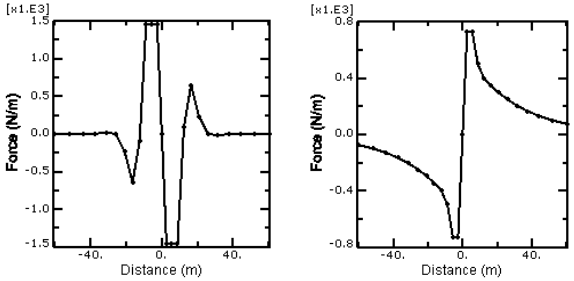
How the problem is modeled ?

Pipe-soil interaction (PSI) elements to model the interaction between a buried pipeline and the surrounding soil. The pipeline itself is modeled with any of the beam, pipe, or elbow elements.

What outputs are Extracted ?







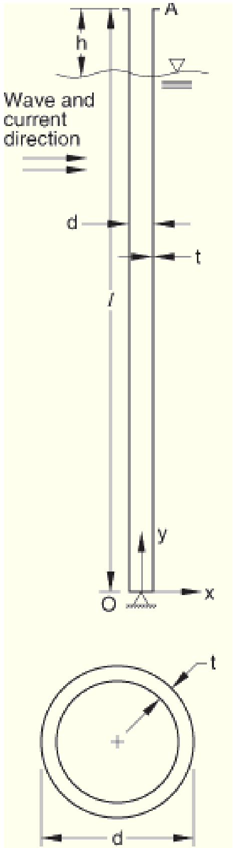
Axial stress along the bottom of the pipeline

Axial and vertical force per unit length of the pipeline

Dynamic analysis of a offshore deep water Riser

Problem Description

Pipelines extending from the sea floor to the ocean surface (risers) are subject to many types of load: self-weight, buoyancy, internal and external pressure, tensile forces arising from surface moorings, current drag, and oscillatory loads resulting from wave motion. The response of a riser to these loads is complex, and the difficulty of such analysis is heightened by the relative length of such pipelines (deep water risers).

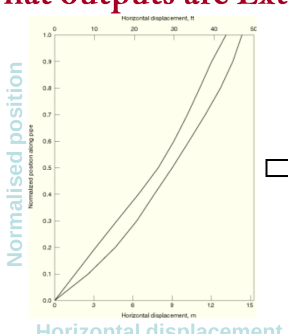


Offshore Riser Pipe specifications

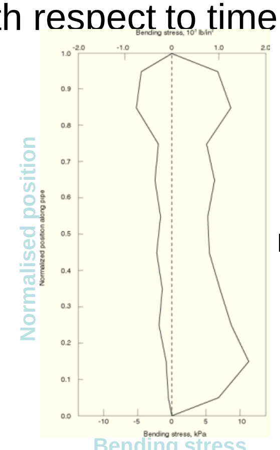
How the problem is modeled ?

The riser has a weight and is loaded by a top tension . Drag loading is applied by a steady current flowing by the riser with a velocity distribution varying linearly. Wave motion effect modeled with the Airy wave theory. The analysis is done in two steps. The first is the static step, in which the top tension is applied and the riser is moved from the vertical to its offset position by specifying the necessary horizontal displacement at the top of the pipeline. The second step, is a dynamic step, where deformation is applied with respect to time.

What outputs are Extracted ?



Displacement plot Vs riser position

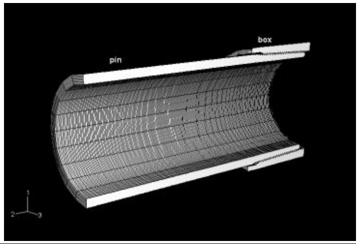


Bending stress
Vs
Riser Normalised position

Axisymmetric analysis of offshore threaded connection

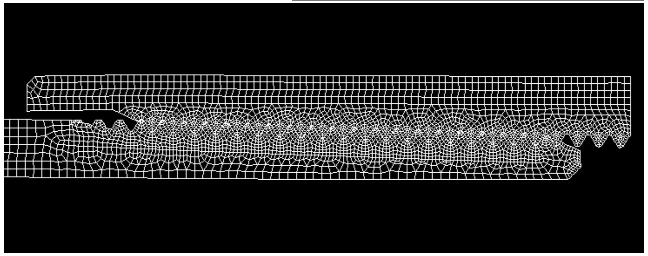
Problem Description:

Threaded connectors are commonly used components in the piping and offshore industry. They must withstand a variety of loading conditions: thread engagement, torque, bending, axial pullout, internal pressure under operating and overload conditions, and potential fluid leakage through threaded connections.

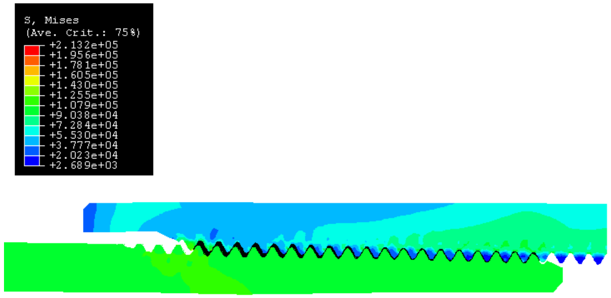


How the problem is modeled ?

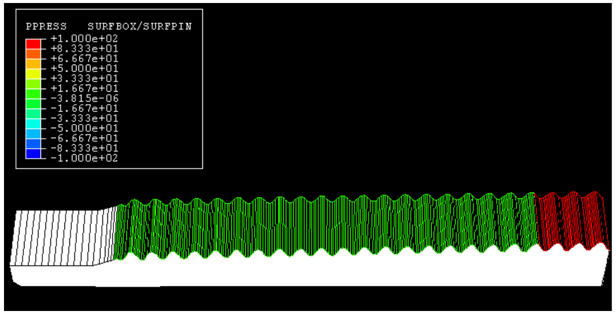
A three-dimensional cut-away view of the threaded connection assembly analyzed. Although the actual threads are helical, they are represented with an axis symmetric geometry. Three steps adopted for problem solving. Step-1: Resolving Interference fit between threads of pin and box, Step-2: the assembly is held fixed while the friction coefficient is changed from 0 to 0.1. Step-3: An internal gauge pressure of 0.689 MPa is applied.



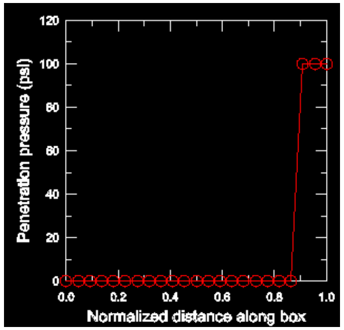
What outputs are Extracted ?



Stress Plots after Interference fit



Pressure Plots



Penetration pressure Vs distance along box

Finite Element Analysis of kingpin failure in heavy cranes

Problem Description:

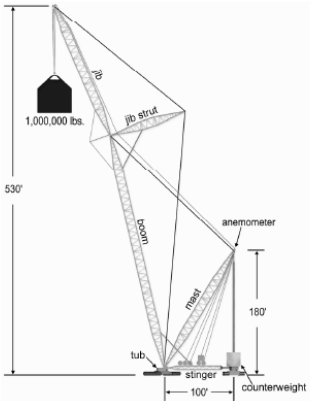
While lifting a one million pound section of space frame roof structure at a construction site, a crane failed and overturned. During the ensuing failure investigation, the strength of the crane's king pin, a 12″ diameter, 11′ long steel shaft about which the front crawler body rotates, was a key issue along with any changes in king pin strength that may occur with various king pin connection details.

How the problem is modeled ?

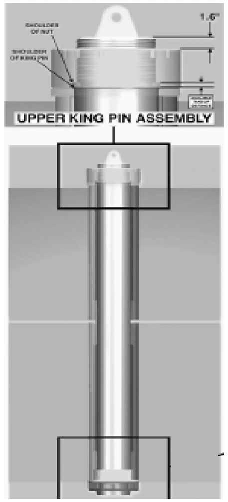
A quarter symmetry solid element model of the king pin is assumed for analysis. The model included multiple 3D deformable body-to-deformable body contact surfaces and material nonlinearities.

What outputs are Extracted ?

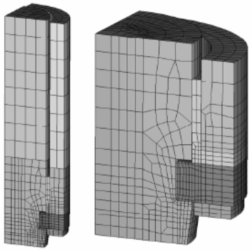
The analysis showed the variation of king pin strength with differing king pin connection details and this information was used as part of the multi-disciplinary investigation to determine a probable cause for the failure of the crane.



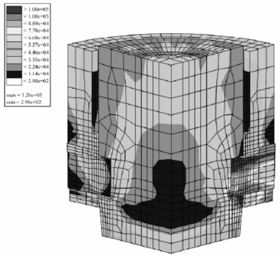
Heavy lift crane



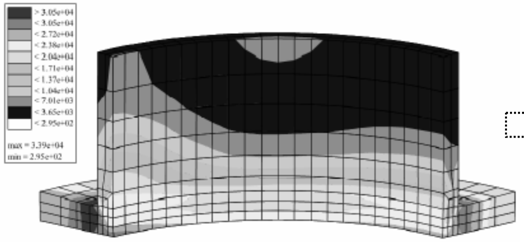
Kingpin connection details



Finite element model of Kingpin



King pin stress plots

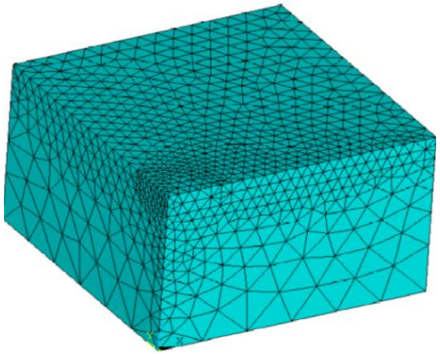


Thrust bearing stress plots

Finite Elements Analysis On Weak Foundation Of Oil Storage Tank

Problem Description:

Methods on how to deal with foundation and minimum compound modulus of elasticity and Poisson's ratio that are required in order to control the settlement in some magnitude are concluded by using finite element analysis to simulate the extra stress distribution of the single oil storage tank.



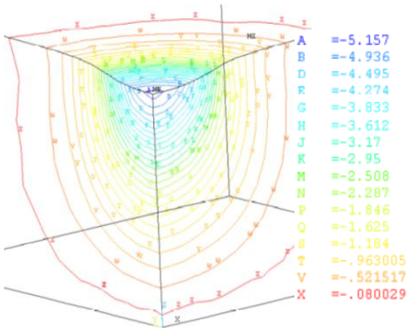
Finite element model of the foundation

How the problem is modeled ?

When the oil storage tank is filled with oil, the maximal additional pressure on the foundation is assumed up to 250 KN/m2. In fact, the contact status of soil and tank foundation is between viscous contact and slick contact, but if the tank foundation only supports vertical force and the soil of foundation is cohesive soil, the contact status of soil and tank foundation tends to viscous contact. Also, dynamic effect and horizontal lateral force of tank foundation are ignored in this work. A one-quarter model of the foundation is assumed for this study.

What outputs are Extracted ?

The maximal stress does not appear at the center of tank foundation, but appears at the areas between 15 and 25 meters away from the center, and it is also higher than the pressure that the foundation is loaded when the tank is filled with oil. For the areas that appear to have a concentration of stress, the soil moves into plastic status from elastic status. Moreover, the soil mass suffers from upward force if the soil is very far from the center of tank foundation.

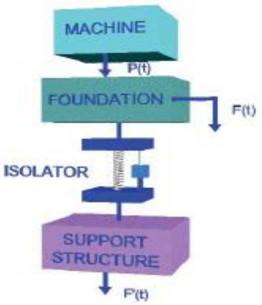


Settlement contour line of homogenous soil

Finite Elements Analysis of structural foundation

Problem Description:

Vibrating, rotating, reciprocating and impacting equipment create machine-induced vibration and/or shock, which is transmitted into their support systems. Rotating machines and equipment that are not properly balanced produce centrifugal forces creating steady state and random vibration. A finite element study conducted on the foundation with different combinations including soil and other dampening effects will reduce this problem.



Free body diagram of the loads taken by the support structure

How the problem is modeled ?

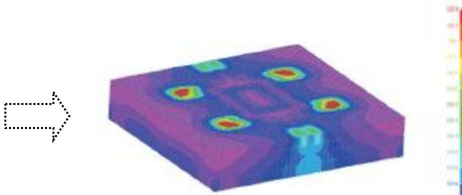
A foundation or any support structure must provide a reliable structural configuration that also meets the static and dynamic criteria for the structure. Deflections in the foundation caused by static loads or by dynamic forces/inputs should be within acceptable limits. This design approach sometimes requires modeling of the foundation as seen in the above figure, so that the real structure behavior is predetermined and errors are minimized.

What outputs are Extracted ?

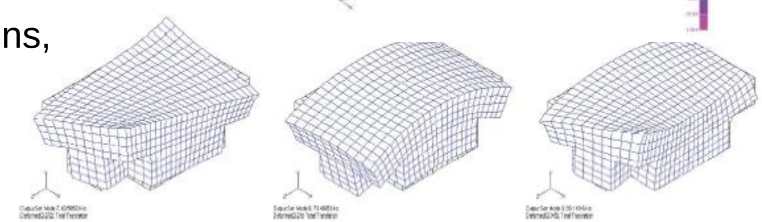
The calculations for the stiffness of a foundation yield the static and dynamic behavior and stress concentration points that occur.

Mode shapes (stiffness of a structure in each axis) identify the physical direction of each frequency mode and any deformations, such as bending or twisting.

Examining mode shapes in a vibrating structure is a valuable step in adjusting vibration amplitudes at critical points by varying the stiffness, mass and damping in a structure.



Foundation stress plots



Mode shapes of the structure

Case Study – FE analysis of Decking structure

Objective :

- To perform a 3D static, and seismic load analysis of the decking structure.
- To Predict reaction loads and reaction moments at all Anchor bolts.

FEA Modeling :

- Material properties, gravity and seismic loads, and CAD model as input. Several load-cases analyzed.
- Lumped mass formulation used for heavy equipments and assemblies supported by decking structure.
- Anchor bolts were modeled with appropriate contact s with the columns fixed to the ground.

Task Executed:

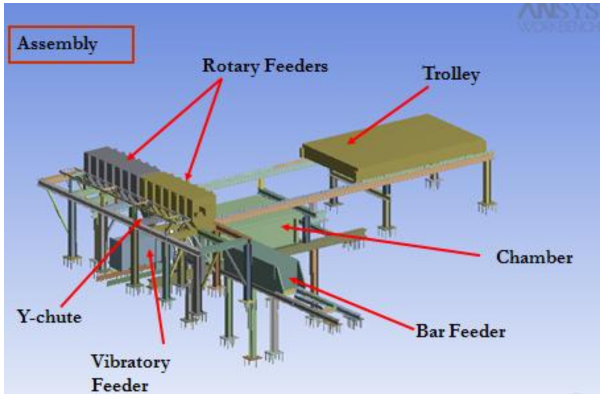
- The model cleaned using Unigraphics. Hex-dominant meshing carried out using Ansys Workbench. Model set-up and Analysis carried out in Ansys.
- Stress and displacements plots were plotted along with detailed tabular results for reaction loads and moments at all bolts.

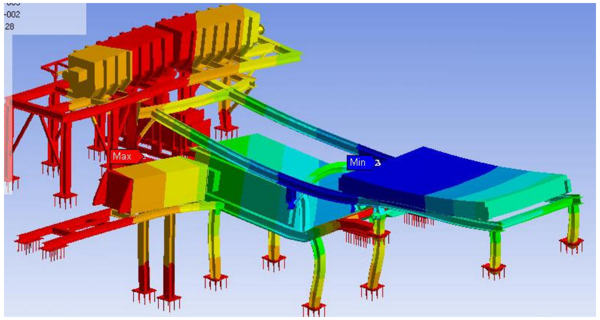
Conclusion & Reliability :

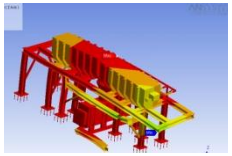
- CAE analysis was carried out to make required changes in the decking structure design and bolts' design.

Cost Savings :

- Installation was done in IBC zone-4 seismic regions. Decking structure design was iteratively modified.







Name	Total Reaction Force	Reaction Force Vector	Total Reaction Moment	Reaction Moment Vector
B.S.25-1	2,103.43 lbf	[-629.98 lbf x, 1,238.09 lbf y, 1,579.46 lbf z]	2,565.03 lbf·in	[-1,733.22 lbf·in x, 1,880.67 lbf·in y, -196.01 lbf·in z]
B.S.25-2	13,206.01 lbf	[-766.27 lbf x, 3,679.42 lbf y, 12,659.91 lbf z]	12,336.08 lbf·in	[11,679.78 lbf·in x, -3,967.63 lbf·in y, -140.06 lbf·in z]
B.S.25-3	3,732.37 lbf	[745.14 lbf x, 2,258.23 lbf y, 2,876.76 lbf z]	8,840.61 lbf·in	[-7,760.97 lbf·in x, -4,220.51 lbf·in y, 333.26 lbf·in z]
B.S.25-4	14,794.48 lbf	[-239.33 lbf x, -1,137.21 lbf y, 14,748.77 lbf z]	42,884.86 lbf·in	[10,007.95 lbf·in x, 41,698.08 lbf·in y, 471.8 lbf·in z]
B.S.25-5	20,186.04 lbf	[1,315.59 lbf x, 346.72 lbf y, 20,140.14 lbf z]	47,023.4 lbf·in	[-5,258.04 lbf·in x, -46,728.25 lbf·in y, -154.53 lbf·in z]
B.S.25-6	2,721.4 lbf	[-297.41 lbf x, -1,911.13 lbf y, 1,914.46 lbf z]	7,096.27 lbf·in	[5,630.13 lbf·in x, 4,293.94 lbf·in y, 469.9 lbf·in z]
B.S.25-7	13,255.56 lbf	[1,503.16 lbf x, -2,953.25 lbf y, 12,834.67 lbf z]	20,365.13 lbf·in	[-16,058.27 lbf·in x, 12,516.94 lbf·in y, 443.46 lbf·in z]
B.S.25-8	3,372.34 lbf	[1,199.66 lbf x, -2,088.17 lbf y, 2,360.73 lbf z]	7,150.04 lbf·in	[7,130.52 lbf·in x, -10.76 lbf·in y, -527.94 lbf·in z]

Case Study – NVH on PC Board

Objective

- To perform Dynamic Analysis on a PC Board under bolt pretension load.
- Modal analysis, Harmonic analysis and Random analysis on PC Board assembly.

FEA Modeling

- 3D FE model created using Hypermesh. Model set up done using Ansys Workbench
- Bolt pretension analysis followed by Modal, harmonic and random analysis. Input PSD available from customer for Random analysis'
- Detailed response charts post-processed.

Task Executed

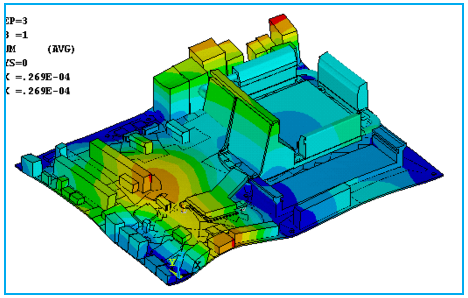
- The model is created in Unigraphics. Hexahedral meshing is carried out using Hypermesh. Model set-up done in Ansys Workbench and Analysis is carried out in Ansys classic.

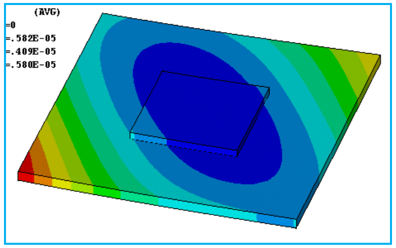
Conclusion & Reliability

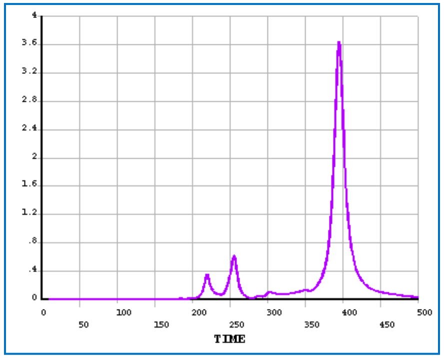
- Components with high resonance identified.
- Required vibration isolation and correct bolt pretension was determined later in the design cycle.

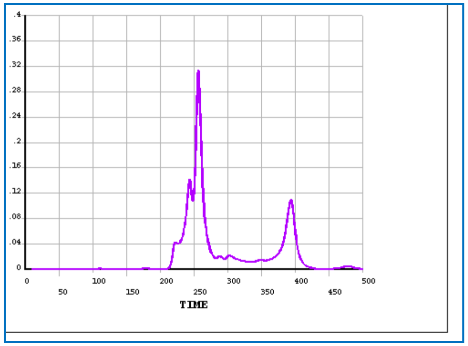
Cost Savings

- Simulation results validated with test data used for further design changes.









Case Study – Creep

Objective

- To predict creep deformation in the backing plate (of process chamber) after 10,000 hrs of operation.
- Repeat the analysis with the steel support structure, and predict reaction forces (at different locations).

FEA Modeling

- Quarter symmetric model analyzed. T=Creep constants for time hardening curve fit, available from the customer.
- Different sizes of backing plate with different thicknesses analyzed.
- Different sub-models, such as steel bolt fitment to the Aluminum plate, welds between transfer chamber and process chamber, etc were studied.

Task Executed

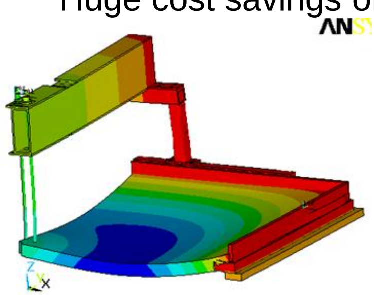
- 3D CAD model was created in Unigraphics. Meshing is carried out using Ansys Workbench. Model set-up was done in Ansys Workbench.
- Analysis and Post-processing was carried out in Ansys classics. Many other important factors, such as low-thickness BP lift at the corners, stresses in the lid-welds, etc, were studied.

Conclusion & Reliability

- Creep constants were validated by comparing results on 40 K without support structure with the real life field data.
- Steel support structure with modified fitment became standard solution for creep for all future projects.

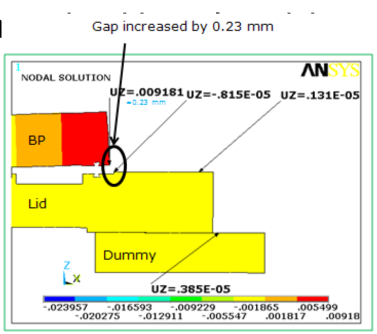
Cost Savings

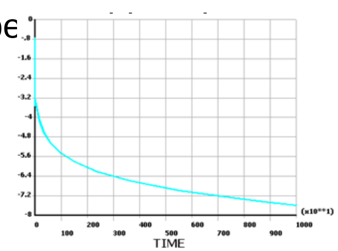
- Huge cost savings on pa … ng thin … be



Comparison chart

Cases/ Loadsteps		BP lift (corner) mm	BP sag (center/max) mm	Total gap (increased) between lid and BP mm	Von-mises (psi)	Von-mises @ weld (psi)	Reaction force –half (symmetry) bolt (lbf)	Reaction force –full bolt (lbf)	Creep strain
3.5"	Structural	0.0	0.0	0.0	00000	00000	00000	00000	
	Thermo-structural	0.0	0.0	0.0	00000	00000	00000	00000	
	Creep	0.0	0.0	0.0	00000	00000	00000	00000	0.00%
4"	Structural	0.0	0.0	0.0	00000	00000	00000	00000	
	Thermo-structural	0.0	0.0	0.0	00000	00000	00000	00000	
	Creep	0.0	0.0	0.0	00000	00000	00000	00000	0.00%
4.5"	Structural	0.0	0.0	0.0	00000	00000	00000	00000	-
	Thermo-structural	0.0	0.0	0.0	00000	00000	00000	00000	-
	Creep	0.0	0.0	0.0	00000	00000	00000	00000	0.00 %
5.5"	Structural	0.0	0.0	0.0	00000	00000	00000	00000	-
	Thermo-structural	0.0	0.0	0.0	00000	00000	00000	00000	-
	Creep	0.0	0.0	0.0	00000	00000	00000	00000	0.00 %





Case Study – TSSL chamber crack propagation

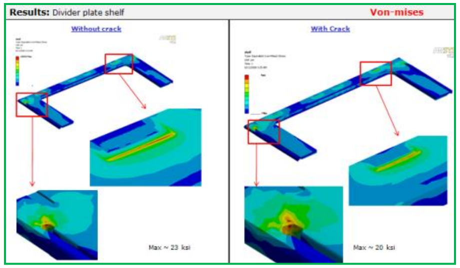
Objective :
- To perform a 3D weld propagation in the TSSL chambers. Existing cracks in the field.
- To check vacuum leak as worst case scenario.
- Design retrofit.

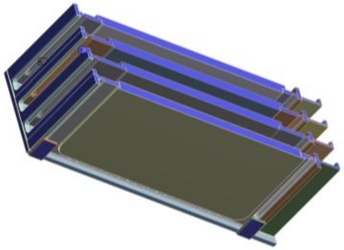
FEA Modeling :
- 3D FE model created using Ansys Workbench. All welds modeled as parts. Appropriate bonded and standard contacts assigned. Number of parts is very high.
- Linear static analysis to identify regions above fatigue yield limit.
- Crack created later to study crack propagation characteristics.

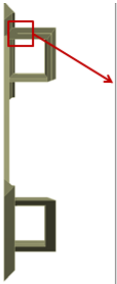
Task Executed:
- The model is created in Unigraphics. Meshing is carried out using Ansys Workbench. Model set-up and Analysis carried out in Ansys Workbench.
- Crack propagation parameters calculated and analyzed. Suitable design changes analyzed to check vacuum leak.

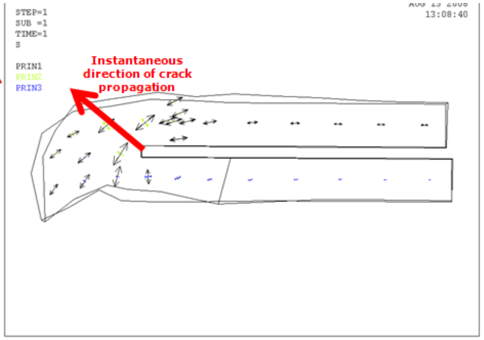
Conclusion & Reliability :
- Crack direction of propagation, growth rate identified.
- All equipments installed in field were later worked on over with suggested retrofit.

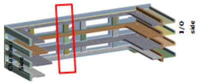
Cost Savings :
- Cracks appearing in field identified and retrofit made well in time before vacuum leaks.






Weld throat plane perpendicular to view





Stress in ksi		Column 2			
		VAV		AVA	
		VonMises	Max Principal	VonMises	Max Principal
Upper Load Lock	TOP				
	BOTTOM				
Middle Load Lock	TOP				
	BOTTOM				
Lower Load lock	TOP				
	BOTTOM				

* -ve Sign Indicates Compression

Case Study – Hoist-bracket structural analysis

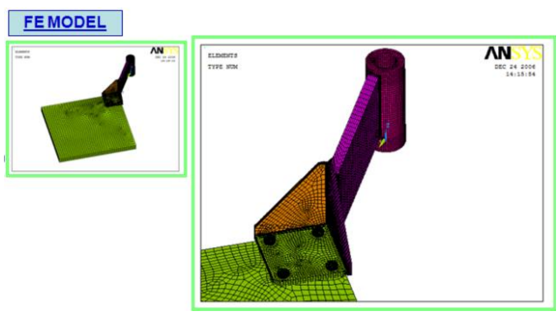
Objective :
- To perform a 3D stress analysis hoist bracket assembly.
- Identify regions of low safety factor or failure as the case may be.

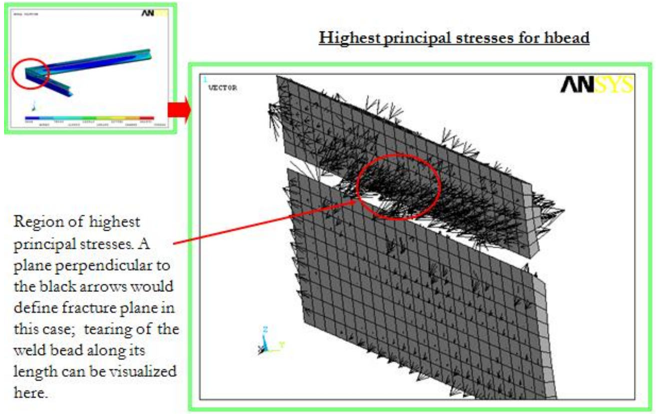
FEA Modeling :
- 3D FE model created using Ansys classic. Welds analyzed as parts with appropriate contacts.
- Post-yield Non-linear material properties were used.

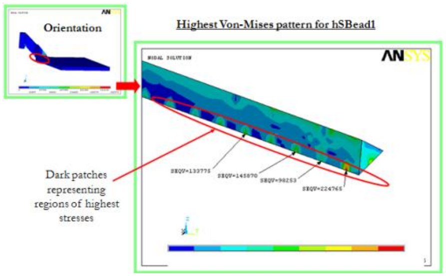
Task Executed:
- The model is created in Unigraphics. Hexahedral meshing carried out using Hypermesh. Model set-up and Analysis carried out in Ansys Classic.
- Welds were particularly meshed with a good quality, being the main focus area.

Conclusion & Reliability :
- Some of the welds were actually failing in some of the load-cases. This was consistent with the testing.

Cost Savings :
- The problem was identified and rectified before entire product was shipped.







Case Study – FE analysis of PSU spacer panel

Objective :

- To perform a 3D stress analysis on 6” spacer panels used in aircraft interior.
- Analyze model against different gravity load-cases, and against rail motion envelope as specified by the aircraft manufacturer.

FEA Modeling :

- Linear material properties used initially, to avoid added complexity due to highly non-linear contacts. A total of 21 cases analyzed including inboard rail motion envelope.
- Worst case scenarios re-analyzed with proposed design changes.

Task Executed:

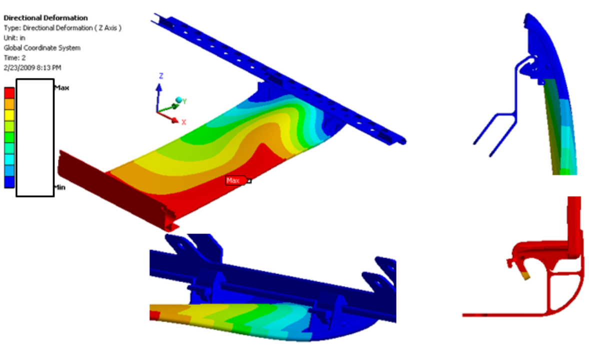
- The model is created in Unigraphics. Hex-dominant meshing carried out using Workbench. Model set-up and Analysis carried out in Ansys Workbench
- Design changes made based on high stress gradients and high strains.

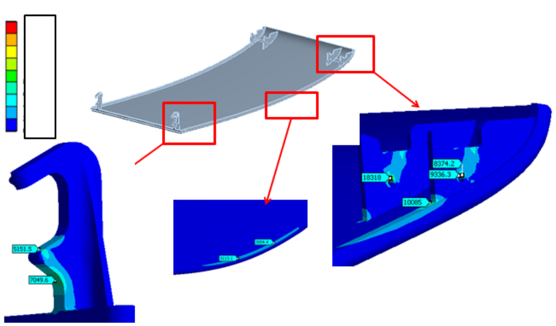
Conclusion & Reliability :

- Project is currently on, and results will be validated by testing.

Cost Savings :

- Client is a tier-1 supplier to Boeing. New designs ‘ conformation to the specifications will open up new opportunities in PSU market.





	Case-1	Case-2	Case-3	Case-4	Case-5	Case-6	Case-7	Case-8	Case-9	Case-10	Case-11	Case-12
	0.25” up	0.75” down	0.25”” out-board	0.25” in-board	1” Forward	0.3” aft	3⁰ Pitch up	3⁰ Pitch down	3⁰ Toe in	0.83⁰ tow-out	Roll-out 10⁰	Roll-in 10⁰
Max Von-mises stress (psi)	00000	00000	00000	00000	00000	00000	00000	00000	00000	00000	00000	00000
Max Von-mises strain (%)	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Case Study – FE analysis of O-ring compression

Objective :

- To perform a 3D stress analysis on O-ring used in the antenna assembly.
- To analyze the performance at various temperatures.
- To predict the deformation of plastic housing due to O-ring compression

FEA Modeling :

- Input all material non-linearities, including the hyper-relastic O-ring and non-linear stress strain curve for the plastic material, at different temperatures.
- Appropriate contact control parameters and time-step control used..

Task Executed:

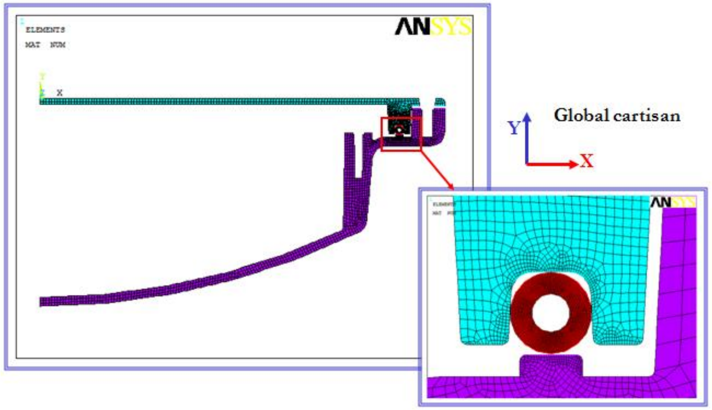
- The model is created in Unigraphics. Hex-dominant meshing carried out using Hypermesh. Model set-up and Analysis carried out in Ansys.
- Stress and displacements plots were plotted along with a detailed analysis of performance.

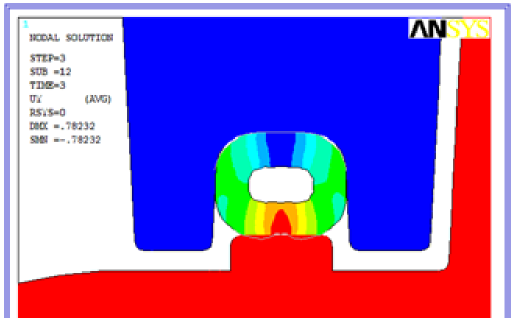
Conclusion & Reliability :

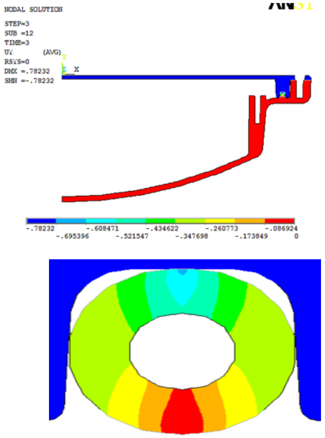
- Results were validated with testing at room temperature.

Cost Savings :

- Antenna is installed in severe weather conditions. These conditions were simulated using the FE A tool.







AES Oil & Gas Case Studies

CFD Analysis of

1. Catalyst (Monolith Modeling)
2. Mixing Tank
3. Reservoir
4. Continuous stirred-tank reactor
5. Porous Modeling with Fluid Injection
6. Nozzle Analysis

Case Study

on

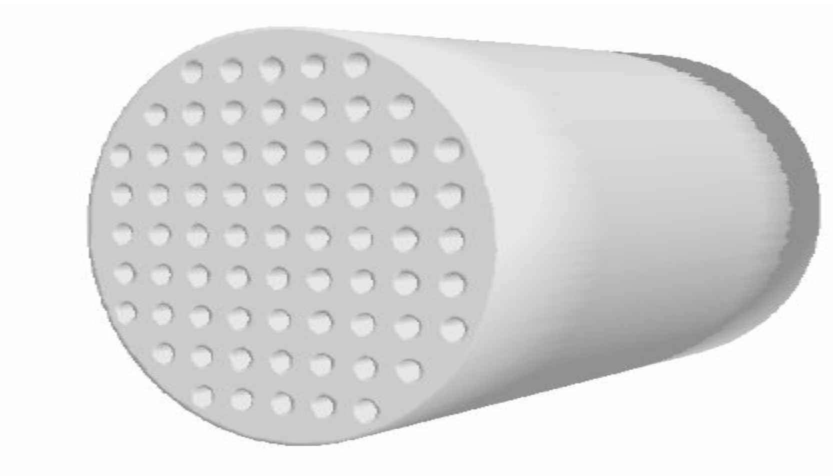
Catalyst (Monolith Modeling)

CFD Analysis of Catalyst

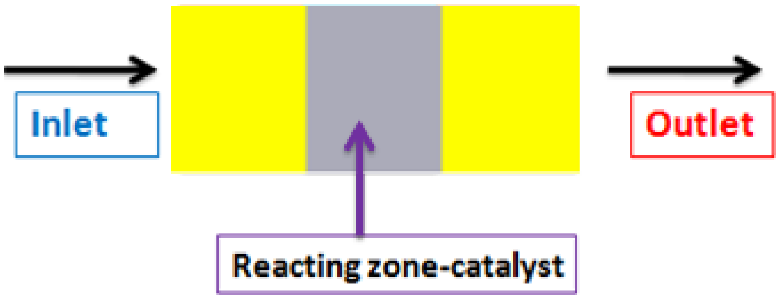
Objectives

➢CFD analysis to predict the flow pattern in Catalyst.
➢To predict the Surface reactions in porous media, Pressure drop, static pressure & total pressure distribution, Accounts for fluid acceleration effects, temperature and mass fraction in the Catalyst.

Geometry Modeling & Meshing



Honeycomb



Honeycomb Model as porous domain

Analysis Methodology

Model:

- Turbulent and unsteady steady incompressible flow in 2D Axisymmetric Domain is solved in Ansys FLUENT parallel solver.

Boundary conditions

- Inlet velocity: 0.8 m/s
- Inlet temperature: 300K
- Adiabatic walls
- Inlet hydrogen volumetric fraction 6%
- Inlet methane concentration is 1.5%
- The operating reference pressure for the mixing tank is 1 atm.

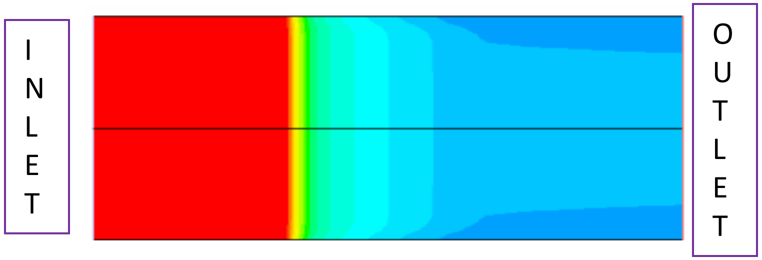
Monitoring

- Residuals to keep track of convergence
- User defined expressions for tracking the Reaction.

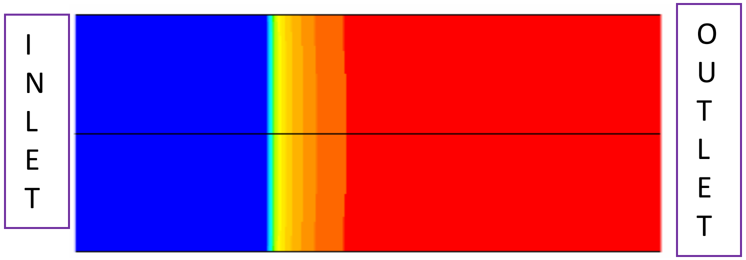
Result & Discussion

Task Executed

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in FLUENT after applying the material properties and different boundary conditions. Different plots are provided in the technical report using FLUENT- Post-processing.



Methane concentration contours



Temperature contours

Typical contour plots

Final Results

From the Analysis, Pressure drop , static pressure, total pressure, static temperature, mass fractions of fluids at different location and velocity components are plotted. Based on the analysis, performance are optimized for the given design as per the SOW.

Conclusion

1. Reacting zone is a catalyst and modeled as porous domain.
2. Methane concentration & temperature contour and were plotted.

Case Study

on

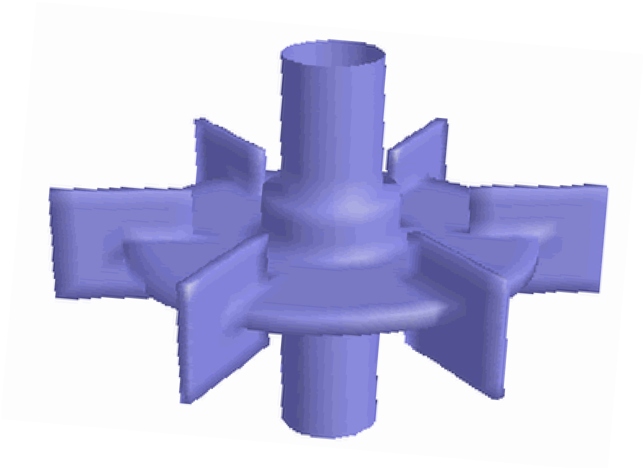
Mixing Tank

CFD Analysis of Mixing Tank

Objectives

- CFD analysis to predict the flow pattern in the mixing tank.
- To predict the static pressure & total pressure distribution , velocity and mass fraction in the mixing tank.
- To optimise the design base on series of runs for different flow rates of jet.

Geometry Modeling & Meshing



Impeller Model

Analysis Methodology

Model

- Turbulent and unsteady steady incompressible flow in 3D Domain is solved in Ansys CFX 11 parallel solver.

Boundary conditions

- The jet was placed theoretically next to the impeller and its effect on reducing the mixing time was investigated
- Liquid (surface tension 0.02 N/m & density 900 kg/m^3) is inside the mixing tank of 19,000 m3 Outlet is section is maintained at gauge pressure of 0 Pa .
- The operating reference pressure for the mixing tank is 1 atm.

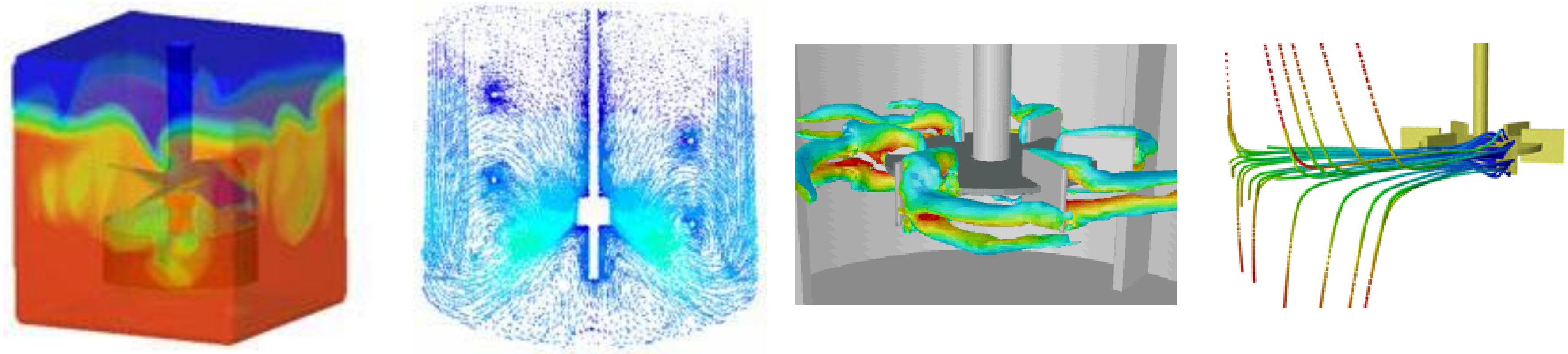
Monitoring

- Residuals to keep track of convergence
- User defined expressions for tracking the flow stability.

Result & Discussion

Task Executed

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in CFX after applying the material properties and different boundary conditions. Different plots are provided in the technical report using CFX- Post-processing.



Typical contour plots

Final Results

From the Analysis, the static pressure, total pressure, mach number, static temperature, mass fractions of fluids at different location and velocity components are plotted. Based on the analysis, performance are optimized for the given design as per the SOW.

Conclusion

Various jet outflow rates including: 33, 66, 132 and 264 m3 h−1 were examined and the effect of the angle between the jet and the impeller on the mixing time for four setups was investigated.

Case Study

on

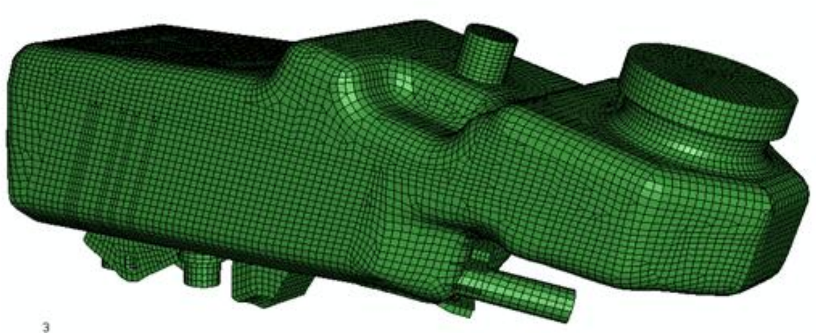
Reservoir

CFD Analysis of Reservoir

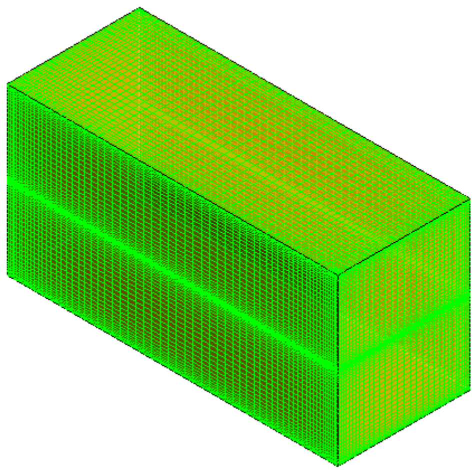
Objectives

- CFD analysis to predict the flow pattern and distribution in reservoir.
- To predict the static pressure & total pressure distribution , velocity and mass flow distribution of liquid at different accelerations of the vehicle.
- To optimise the baffle system for reducing the sloshing problem in the reservoir

Geometry Modeling & Meshing



Customer Model



Base model for Test Case

Analysis Methodology

Model

- Isothermal, Turbulent, Multiphase and unsteady incompressible flow in 3D Domain is solved in Ansys CFX 11 parallel solver.

Boundary conditions

- Completely closed geometry filled with brake fluid and the above is atmospheric air.
- Fluid (surface tension 0.04 N/m & density 1700 kg/m^3) free surface is separating the both fluids.
- The operating reference pressure for reservoir chamber is 1 atm.

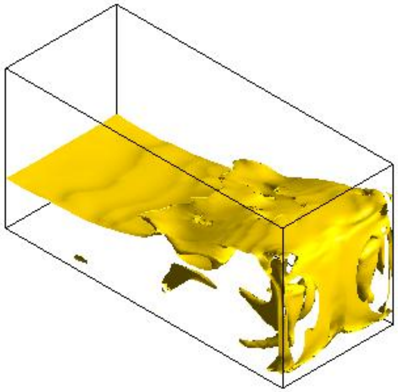
Monitoring

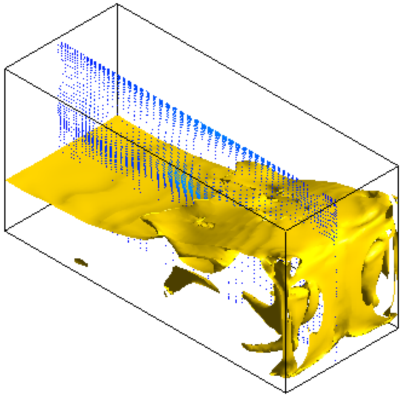
- Residuals to keep track of convergence
- User defined expressions for tracking the flow stability.
- Free surface and pressure data has been captured for every millisecond.

Result & Discussion

Task Executed

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in CFX after applying the material properties and different boundary conditions. Different plots are provided in the technical report using CFX- Post-processing.





Typical contour plots

Final Results

From the Analysis, the static pressure, total pressure, fluid level at different accelerations of the vehicle. Based on the analysis, good baffle design has achieved through few simulations.

Conclusion

Best insight of fluid behavior for each design of baffles made us to judge the design of few baffle systems without any simulations and client could finish the design in less number of simulations than expected.

Case Study

on

Continuous stirred-tank reactor

CFD Analysis of Continuous stirred-tank reactor

Objectives

- CFD analysis to predict the flow pattern in the tank.
- To predict the static pressure & total pressure distribution , velocity and mass fraction in the mixing tank.
- To optimise the design base on series of runs for different flow rates of jet.

Reactor Tank

Stirred Model

Analysis Methodology

Model

- Turbulent and unsteady incompressible flow in 3D Domain is solved in Ansys CFX 11 parallel solver.

Boundary conditions

- The jets were placed theoretically on top of the tank and its effect on reducing the mixing time was investigated
- Outlet is section is maintained at gauge pressure of 0 Pa .
- The operating reference pressure for the tank is 48.3 bar and operating temperature is 300 deg C.

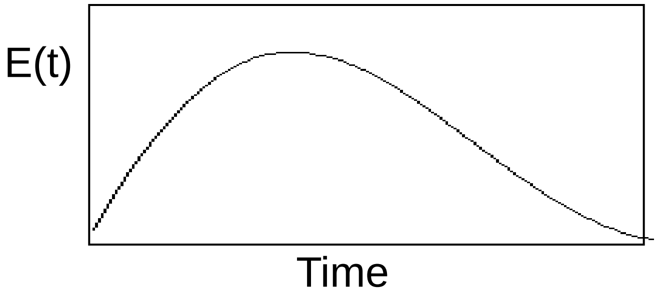
Monitoring

- Residuals to keep track of convergence
- User defined expressions for tracking the flow stability.

Result & Discussion

Task Executed

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in CFX after applying the material properties and different boundary conditions. Different plots are provided in the technical report using CFX- Post-processing.

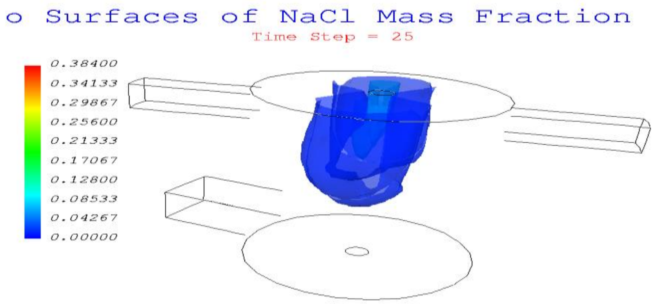


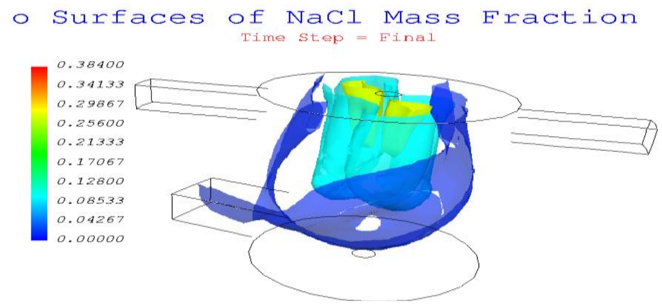
Typical Transient data plot

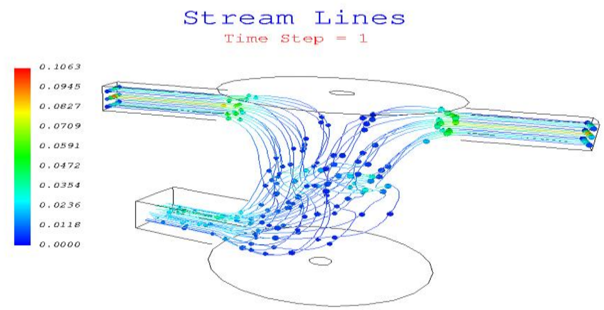
Final Results

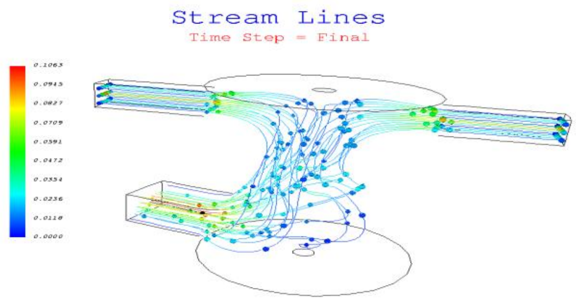
From the Analysis, the static pressure, total pressure static temperature, mass fractions of fluids at different location and velocity components are plotted. Based on the analysis, performance are optimized for the given design as per the SOW.

Typical contour plots









Conclusion

Various jet outflow rates including: 3, 7, 14 and 21 m3 h−1 were examined. The visual contours and tracking along with the transient reported data for parametric flows has added value to the design changes. Residence time is 7.28 times the mixing time.

Case Study

on

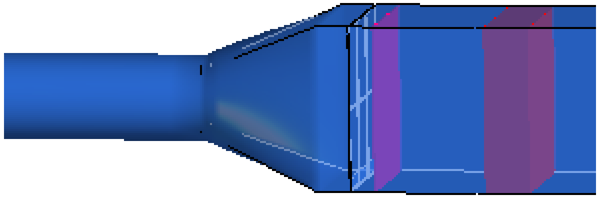
Porous Modeling with Fluid Injection

CFD Analysis of Porous Modeling with Fluid Injection

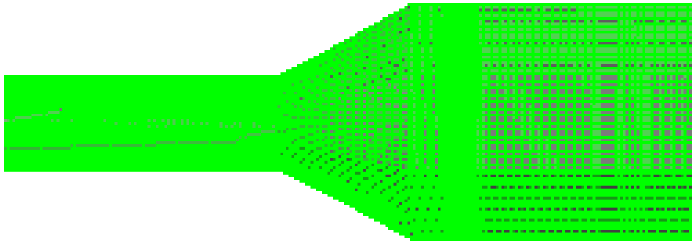
Objectives

- CFD analysis to predict injected fluid concentration
- To predict the velocity, injected fluid concentration, static pressure & total pressure distribution at different location.
- To optimise the design.

Geometry Modelling & Meshing



Model



Mesh

Analysis Methodology

a. Boundary Condition

Inlet : Mass Flow Rate
Outlet : Static Pressure
Porosity Model
Ammonia Injection rate : Defined
Volume Porosity = 0.5

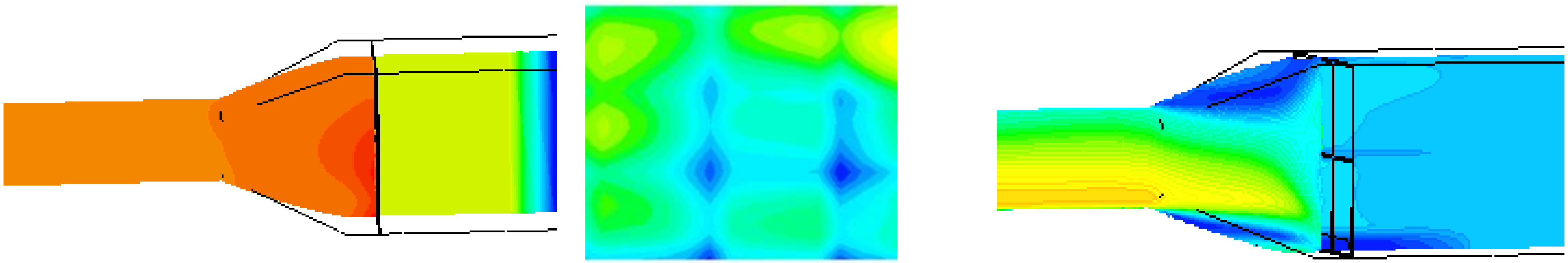
b. Methodology & Convergence

- The flow is Incompressible
- Gases as defined in RFQ are used as the Fluid Medium, Density of individual gas is considered.
- Solver used in the analysis is CFX.
- k-ε Turbulence Model is considered with Standard log law wall function near wall treatment with Inlet turbulence Intensity= 5%.
- The convergence criteria from CFX is taken where the scaled residuals decrease to 10e-4 for all equations.

Result & Discussion

Task Executed

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in CFX after applying the material properties and different boundary conditions. Different plots are provided in the technical report using CFX- Post-processing.



Typical contour plots

Final Results

From the Analysis, the static pressure, total pressure, mach number, static temperature, NH3 injection concentration at different location and velocity components are plotted. Based on the analysis, the perforated plate and fluid injection rate is optimized for the required design.

Conclusion

Initially Customer verified the result with the experimental results and optimized other designs based on CFD analysis.

Case Study

on

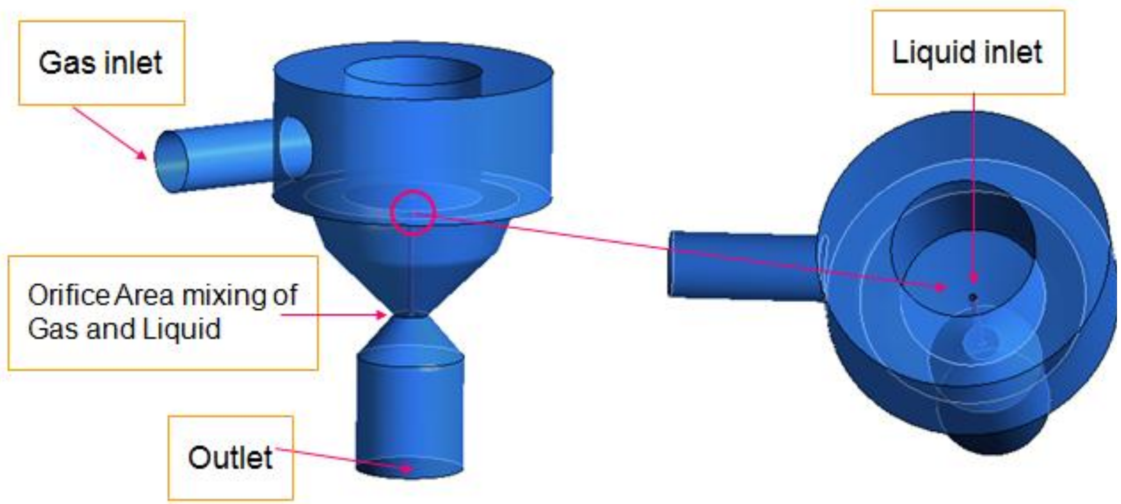
Nozzle Analysis

CFD Analysis of Nozzle

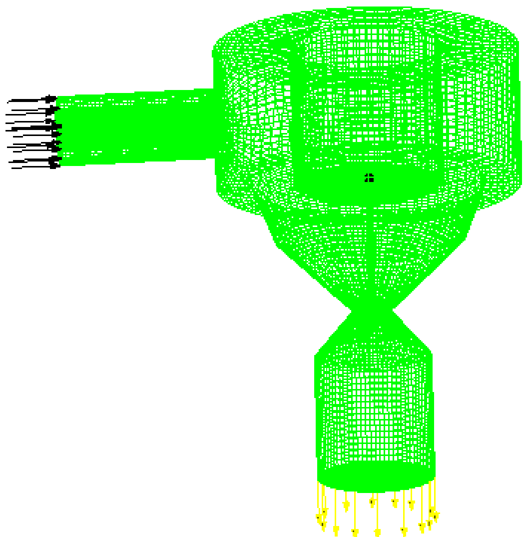
Objectives

- CFD analysis to predict the flow pattern in the nozzle system.
- To predict the static pressure & total pressure distribution , velocity and mass flow distribution of liquid in the nozzle system.
- To optimise the design.

Geometry Modeling & Meshing



Model



Mesh

Analysis Methodology

Model

- Isothermal, Turbulent and steady incompressible flow in 3D Domain is solved in Ansys CFX 11 parallel solver.

Boundary conditions

- Gas is entering through the gas inlet section with mass flow rate of 10 Liter /minute
- Liquid (surface tension 0.02 N/m & density 900 kg/m^3) is entering through the liquid inlet section with a flow rate of 10 milliliters /hour
- Outlet is section is maintained at gauge pressure of 6 psi .
- The operating reference pressure for the nozzle chamber is 1 atm.

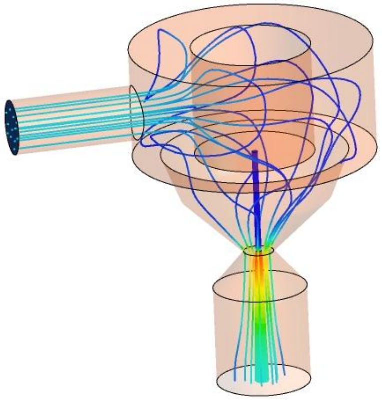
Monitoring

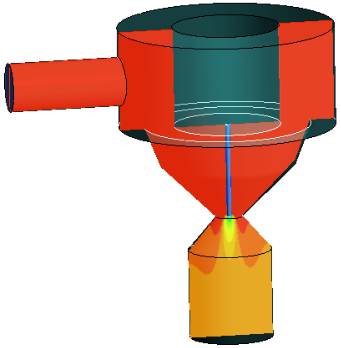
- Residuals to keep track of convergence
- User defined expressions for tracking the flow stability.

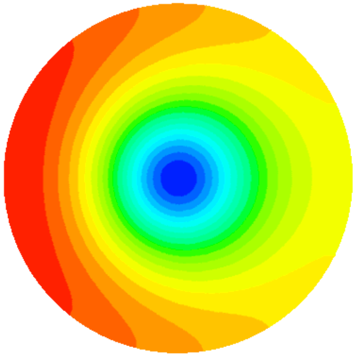
Result & Discussion

Task Executed

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in CFX after applying the material properties and different boundary conditions. Different plots are provided in the technical report using CFX- Post-processing.







Typical contour plots

Final Results

From the Analysis, the static pressure, total pressure, mach number, static temperature, mass fractions of fluids at different location and velocity components are plotted. Based on the analysis, the liquid gas ratio and performance are optimized for the given design as per the SOW.

Conclusion

Customer verified the initial results as well as optimized results obtained by CFD analysis with the experimental data.