FLUID CODES

Introduction to ANSYS Mechanical

Fluid Codes FZ LLE

Channel Partner for Middle East

Fluid Codes Technical Team

Presentation Overview

- What is FEA?
- Real Application cases
- Conclusions

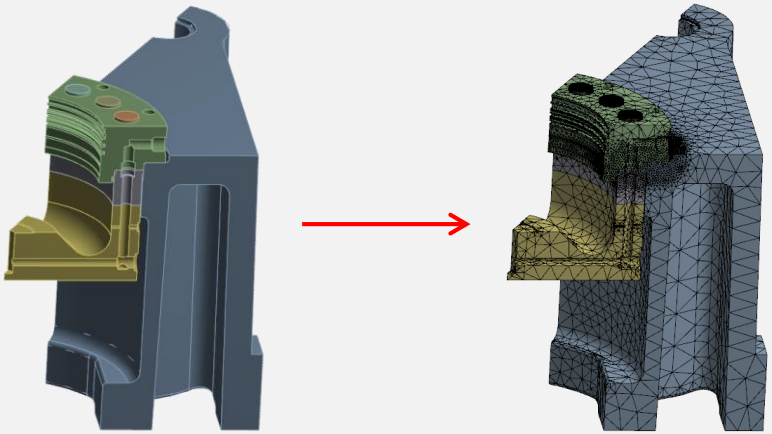
Presentation Overview

- **What is FEA?**
- Real Application cases
- Conclusions

What is FEA?

Finite Element Analysis **is a way to simulate loading conditions on a design and determine the design's response to those conditions.**

The design is modeled using discrete building blocks called *elements*.



Historical Note

- ***The finite element method of structural analysis was created by academic and industrial researchers during the 1950s and 1960s.***
- ***The underlying theory is over 100 years old, and was the basis for pen-and-paper calculations in the evaluation of suspension bridges and steam boilers.***

Loads and constraints are defined on the model

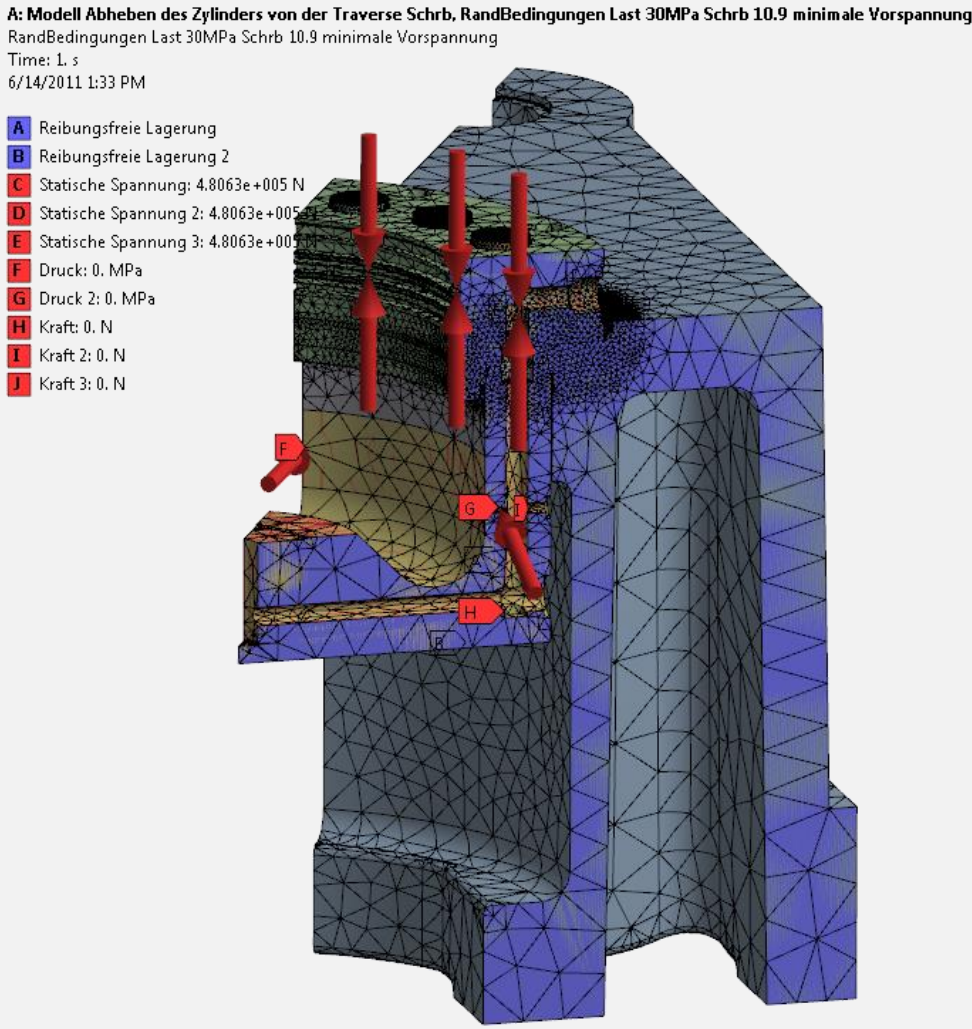
Equation solved for a static analysis

$$[K]\{U\} = \{F\}$$

External forces

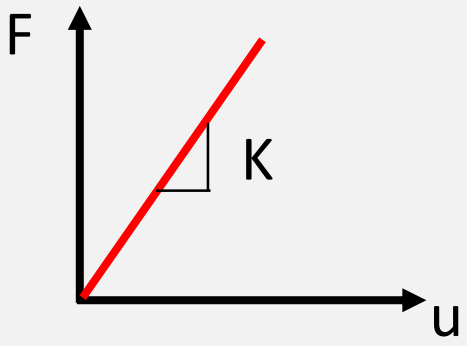
Displacement vector

Stiffness matrix



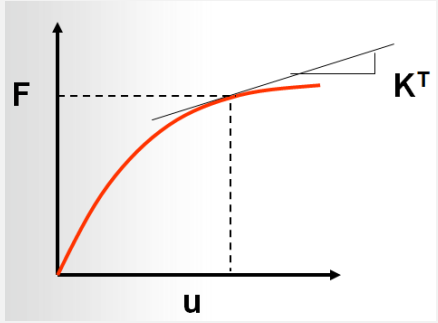
Static Structural: Linear Analysis

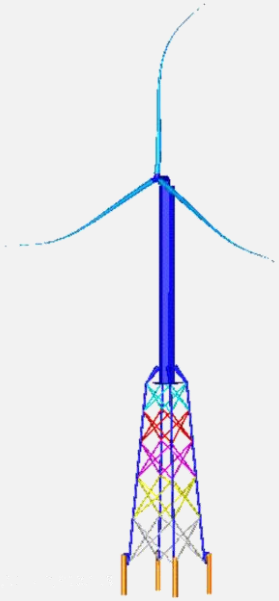
$$[K]\{u\} = \{F\}$$



Static Structural: Non Linear Analysis

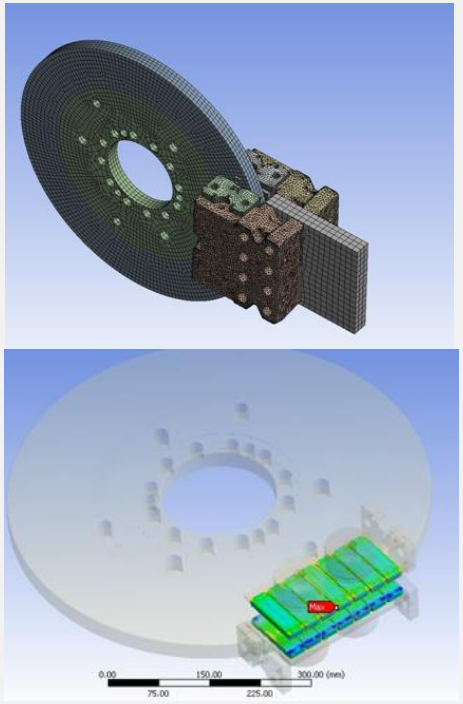
$$[K(u)]\{u\} = \{F(u)\}$$





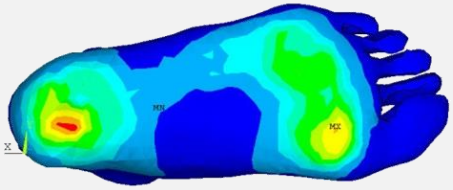
Courtesy of REpower Systems AG

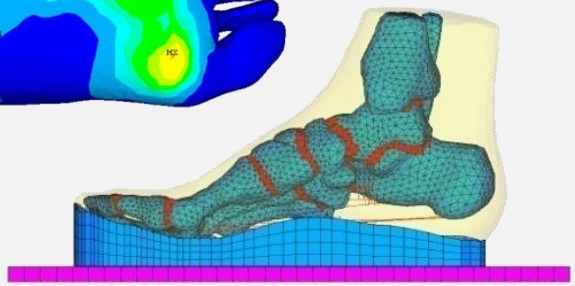
Large Displacements



Contacts

von-Mises Stress distributions on the plantar surface of the foot in utilization of flat insole



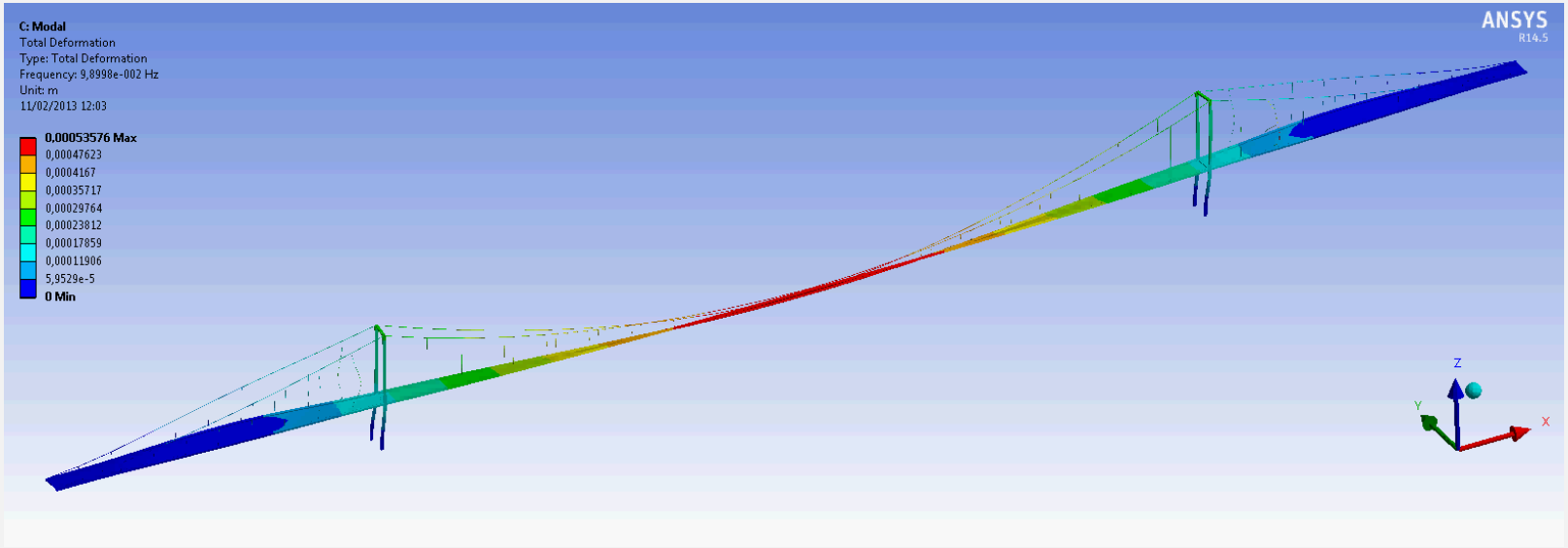


Bones, ligaments and cartilages were included in the mechanical model.

Non Linear Materials

Dynamic Structural: Modal

Modal analysis **is used to determine a structure's vibration characteristics, i.e., natural frequencies and mode shapes.**

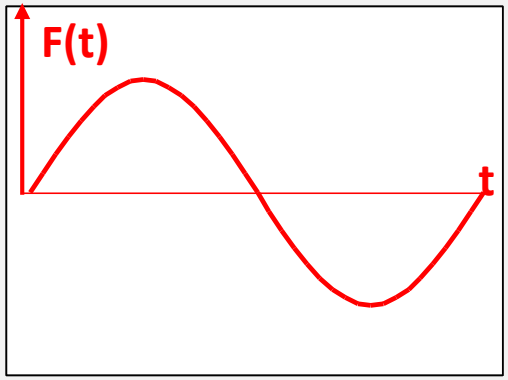


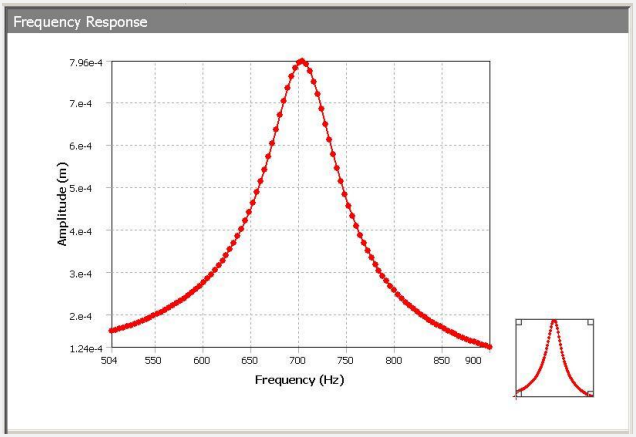
ANSYS Dynamic Structural: Harmonic

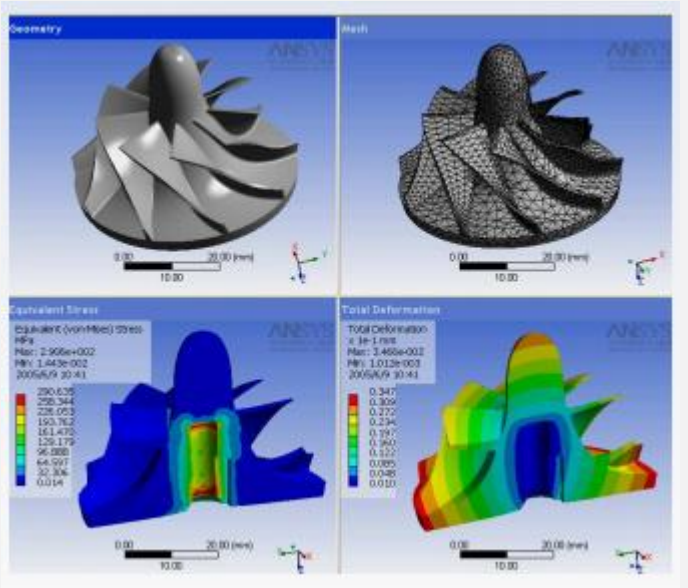
FLUID CODES

The harmonic-response **analysis is used to determine a structure's response to steady, harmonic (sinusoidally varying) loads.**

Rotating machines exert steady, alternating forces on bearings and support structures. These forces cause different deflections and stresses depending on the speed of rotation.



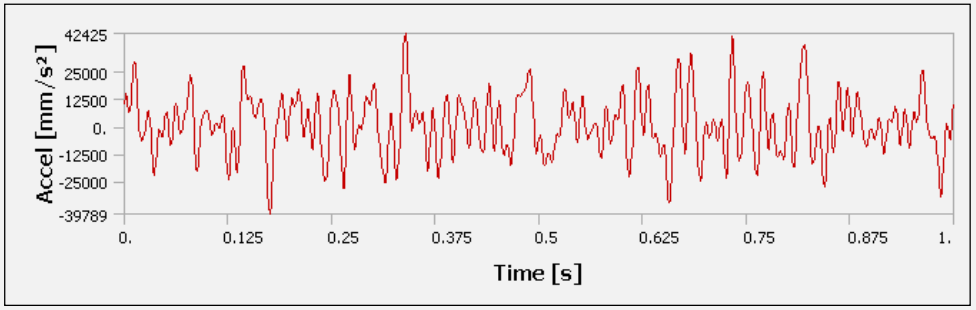


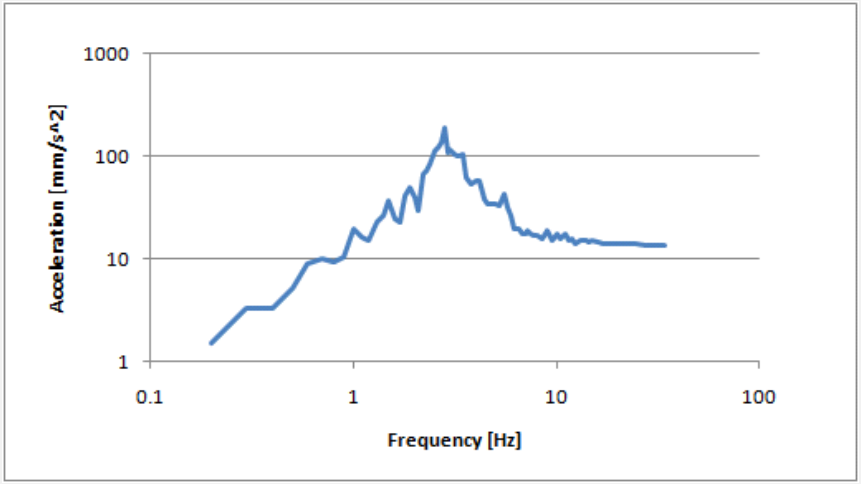


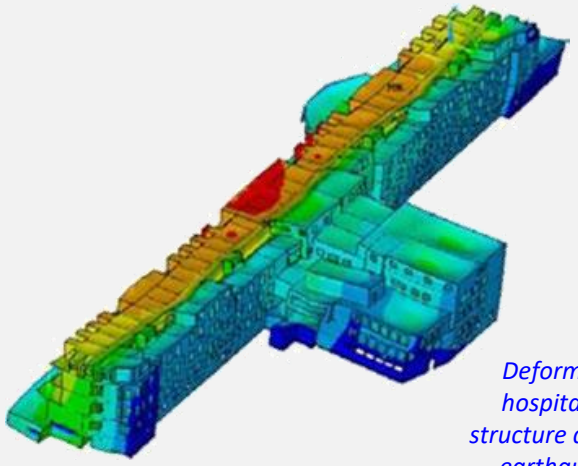
Dynamic Structural: Response Spectrum

A response-spectrum **analysis can be used to determine how a component responds to earthquakes.**

Skyscrapers, bridges and other structures must withstand multiple short-duration transient *shock/impact loadings*, common in seismic events.







Deformation of a hospital masonry structure due to a code earthquake event

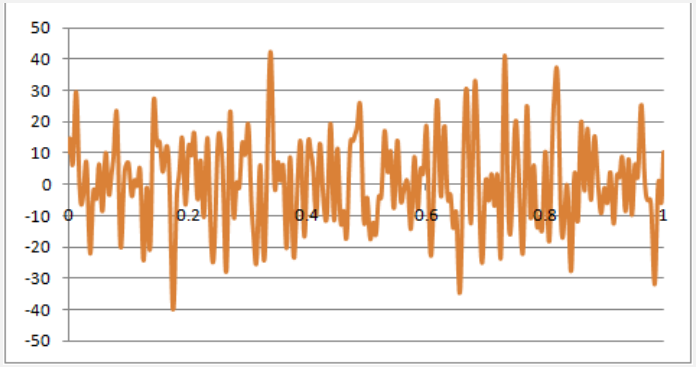
Results

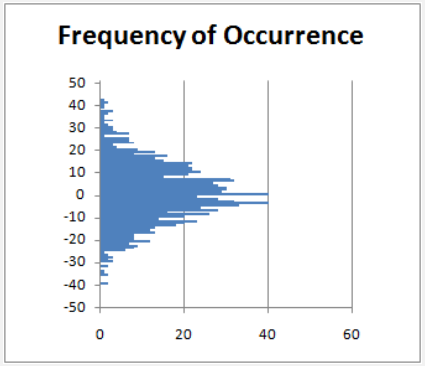
- Deformation
- Velocity
- Acceleration
- Stresses
- Strains

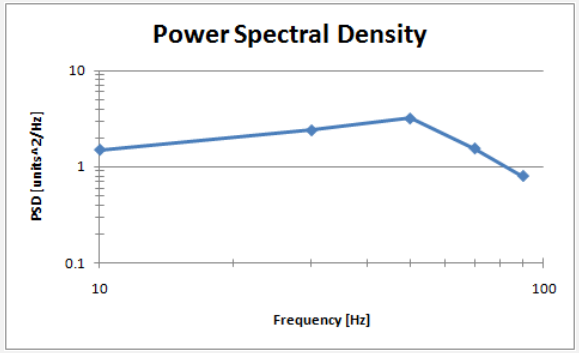
Dynamic Structural: Random Vibration

A random-vibration **analysis can be used to determine how a component responds to random vibrations.**

Spacecraft and aircraft components must withstand random loading of varying frequencies for a sustained time period.

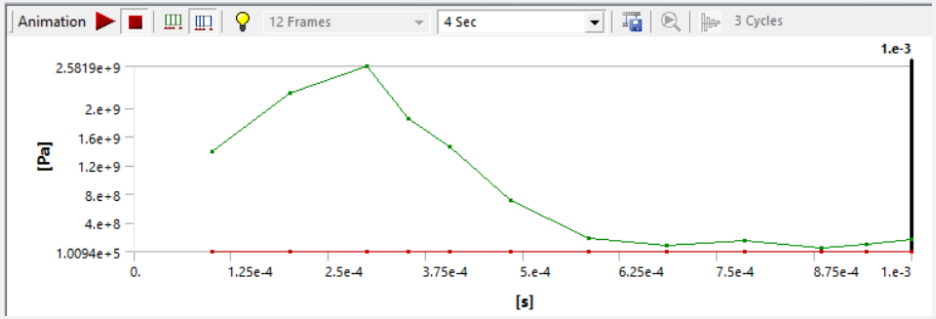


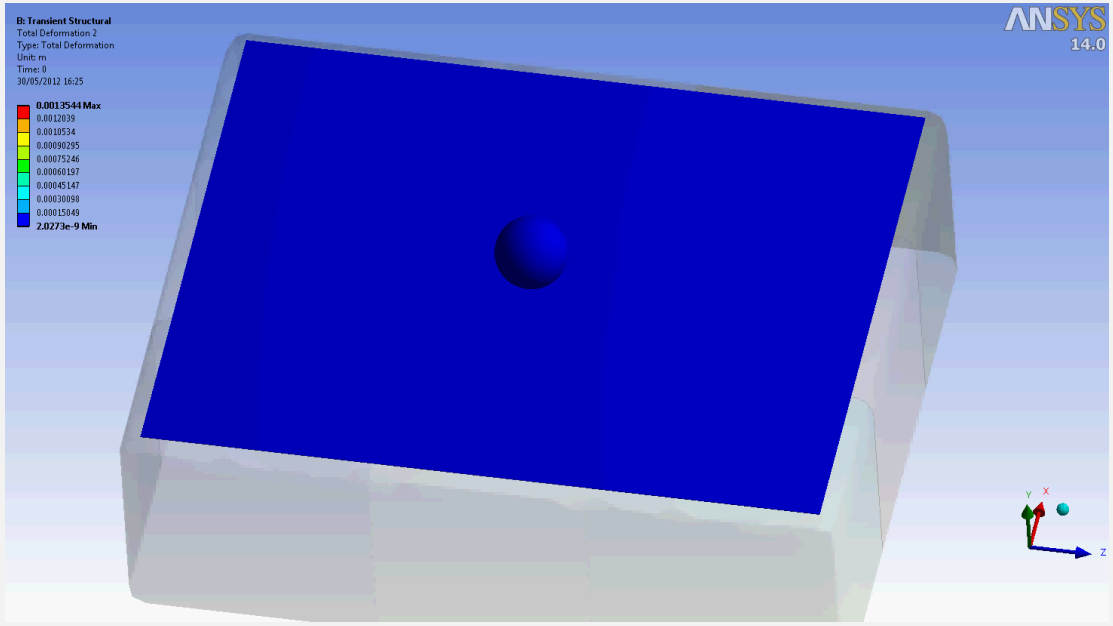




Dynamic Structural: Transient

A transient analysis **can be used to calculate a structure's response to time varying loads.**





Dynamic Structural: Explicit analysis

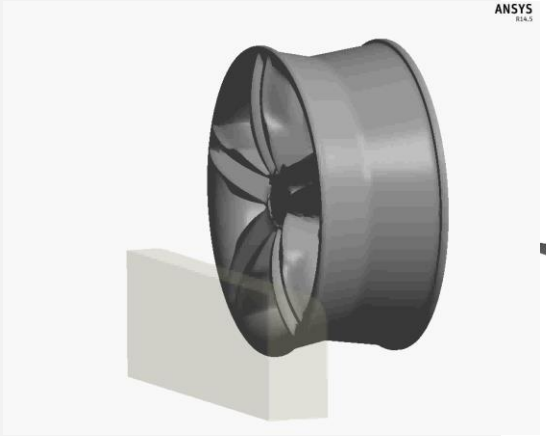
FLUID CODES

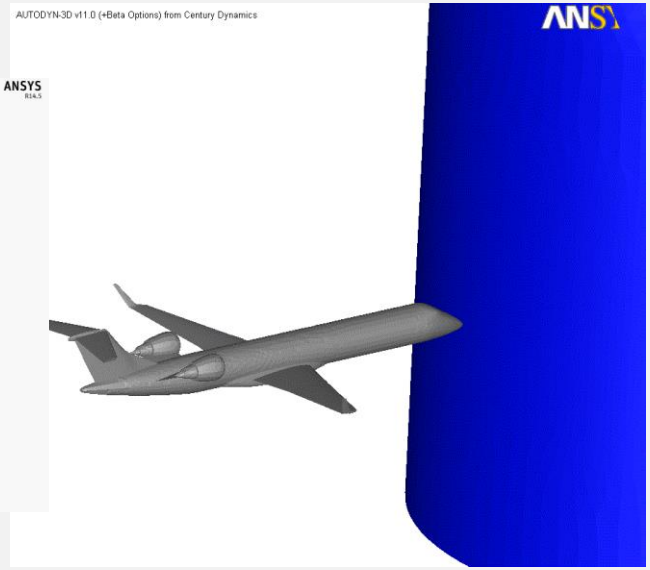
Explicit time integration is more accurate and efficient for simulations involving

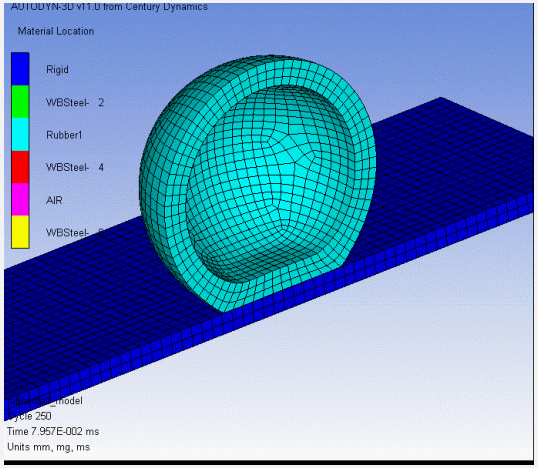
- **Shock wave propagation**
- **Large deformations and strains**
- **Non-linear material behavior**
- **Complex contact**
- **Fragmentation**
- **Non-linear buckling**

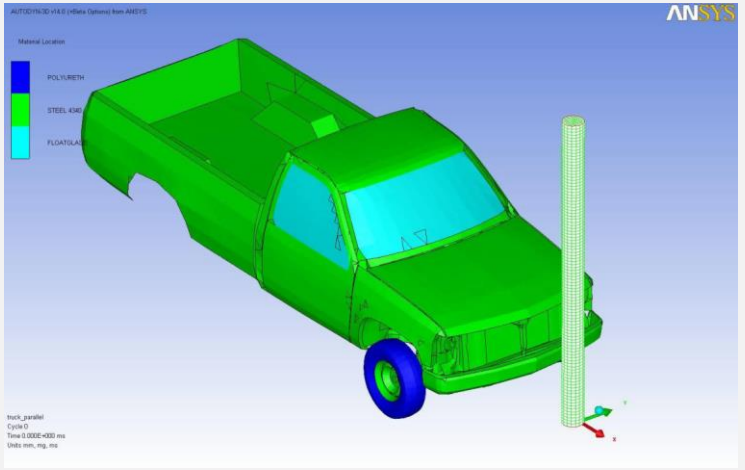
Typical applications

- **Drop tests**
- **Impact and Penetration**

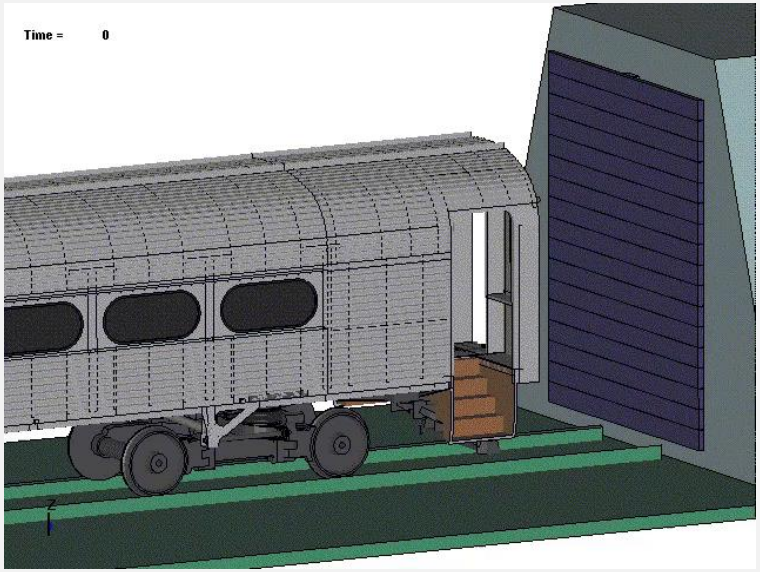


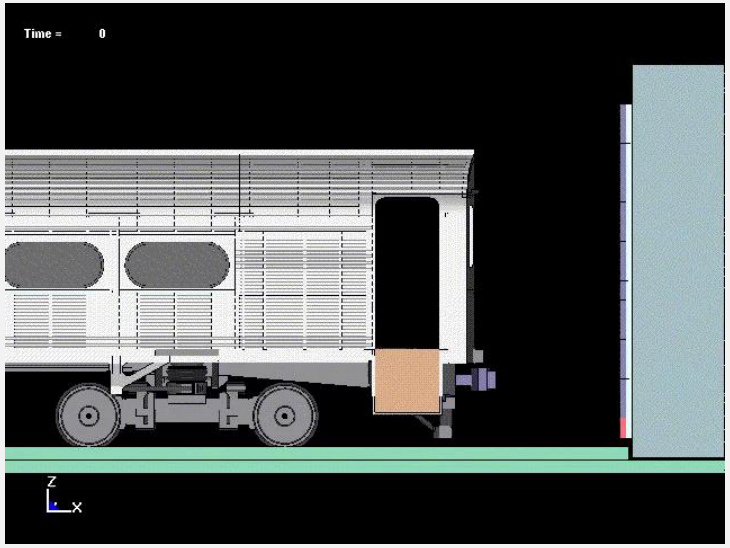






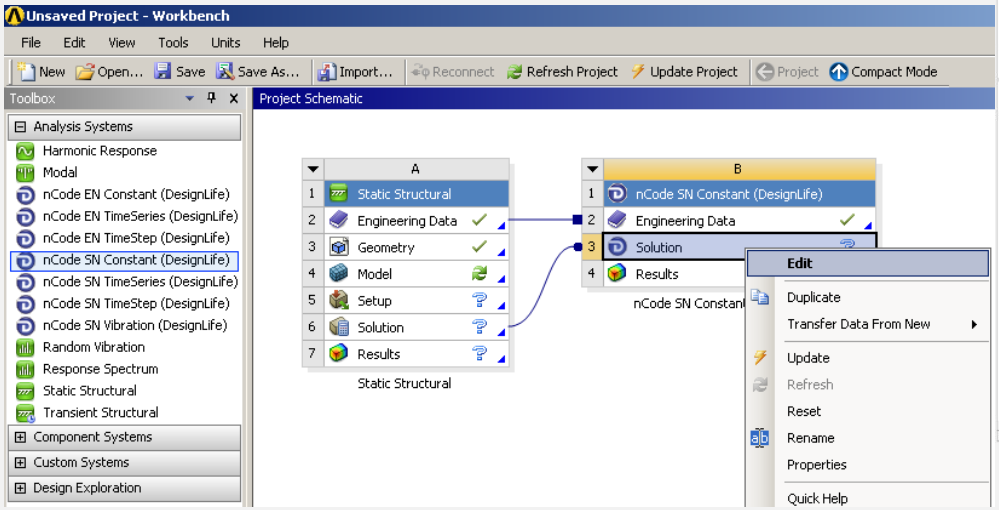
Crash Test

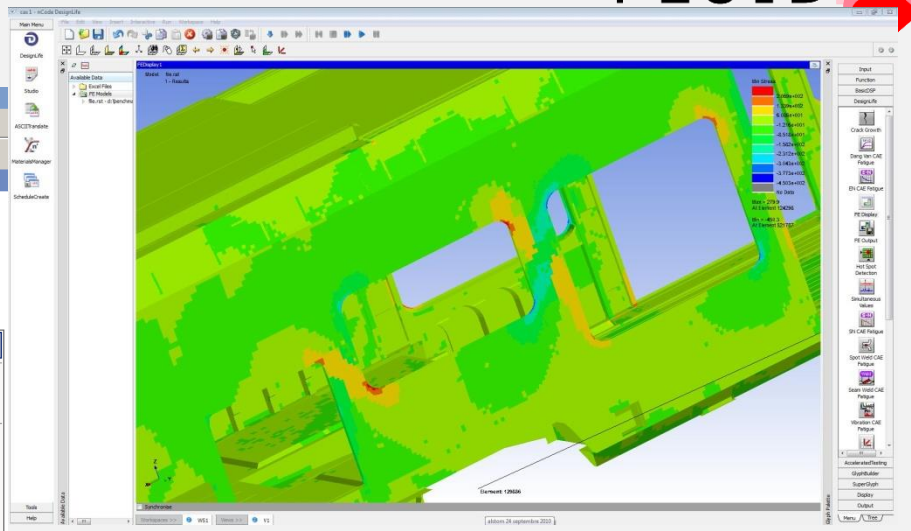


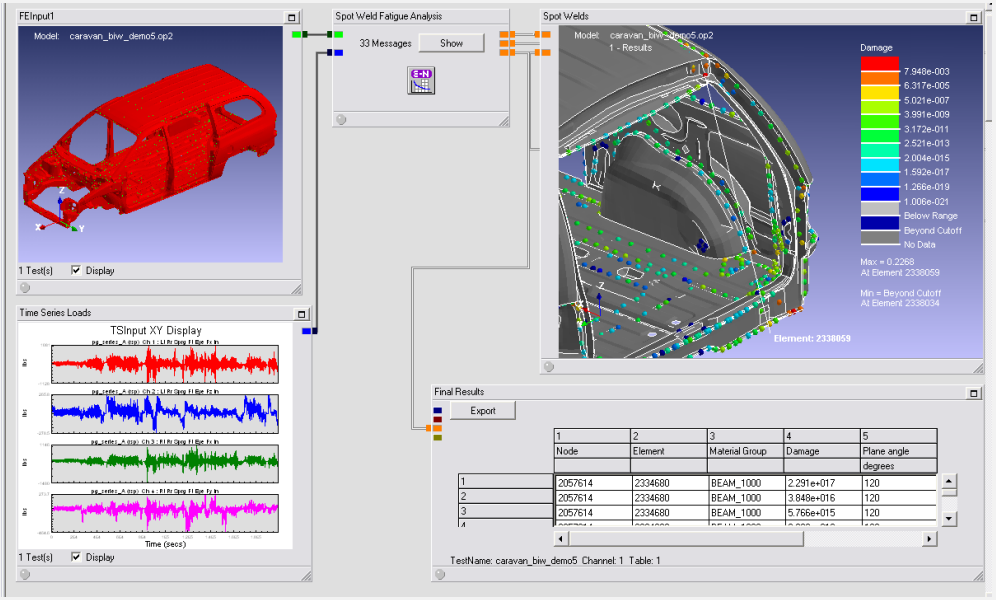


Fatigue Analysis

FLUID



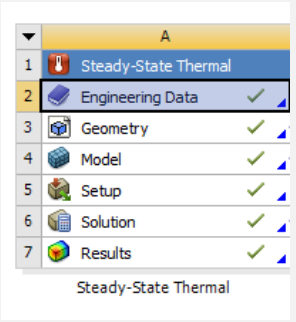


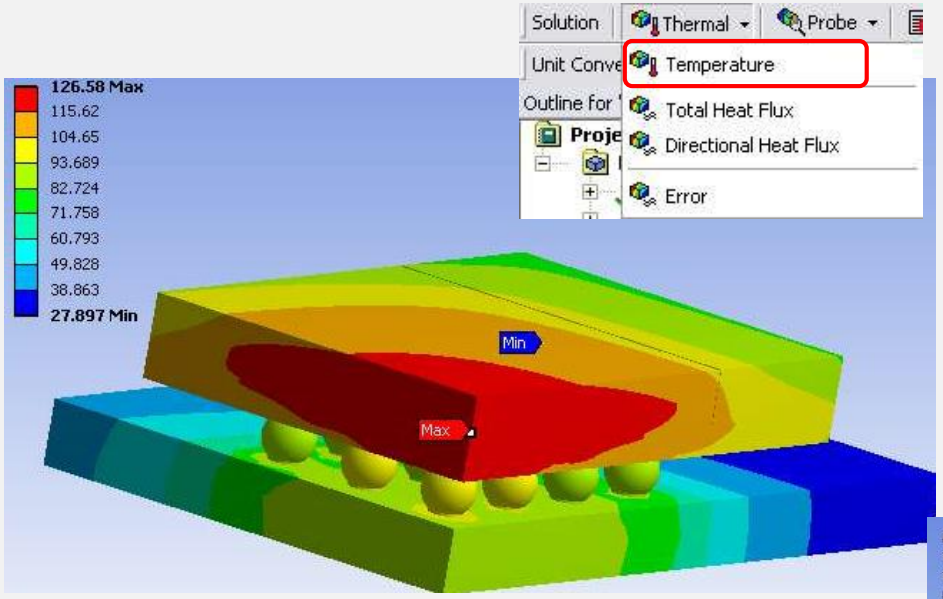


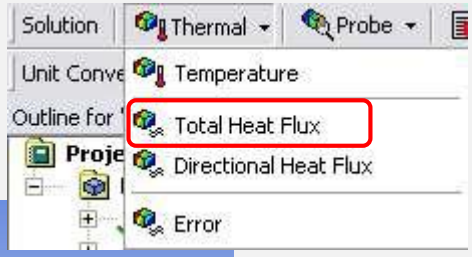
Thermal Analysis

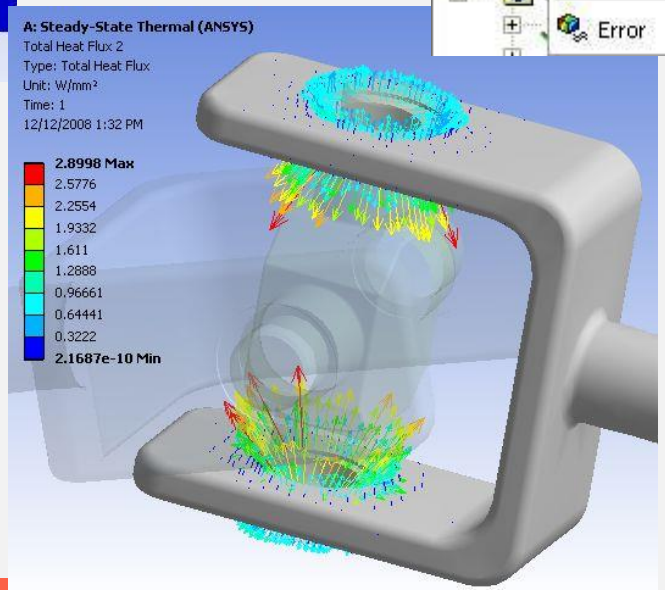
- **Steady State and Transient**

$$[K(T)]\{T\} = \{Q(T)\}$$



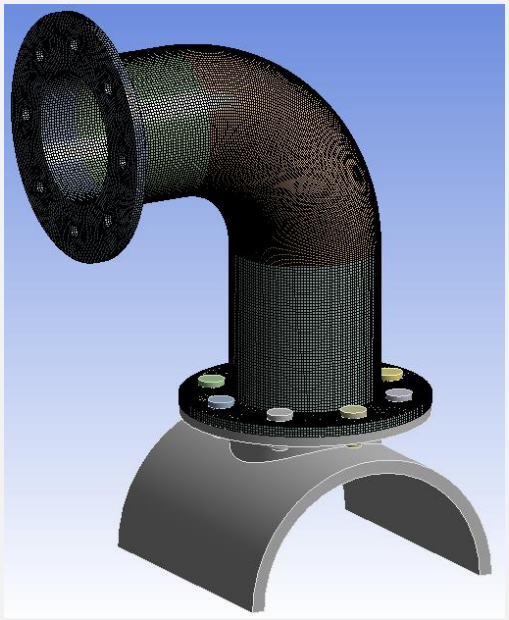


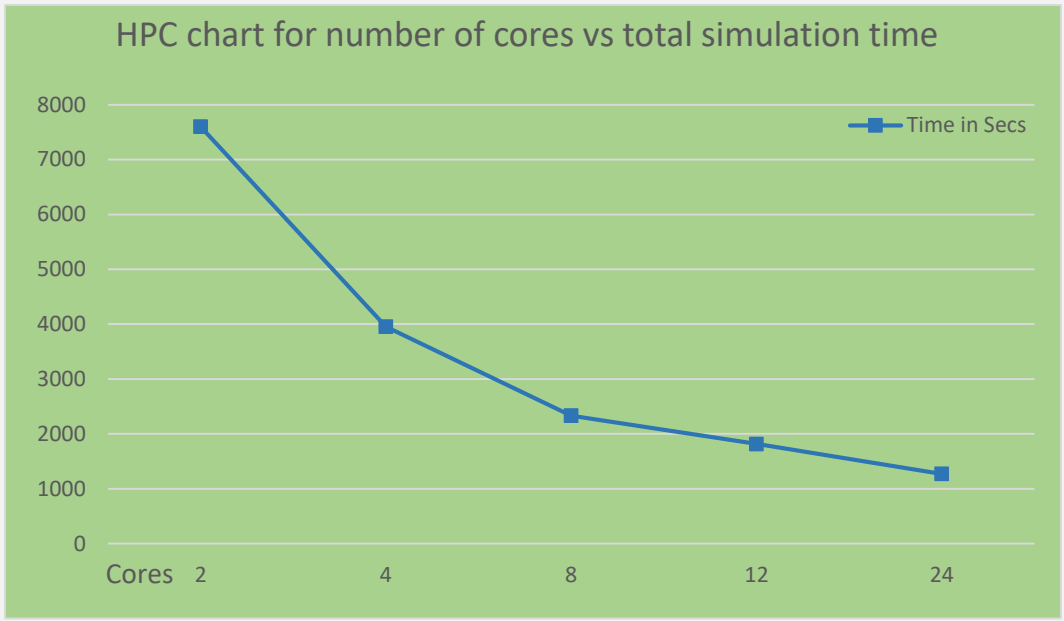




What about getting the solution FASTER?

Simulation of a NOZZLE joined to a pipe by pre-stressed screws





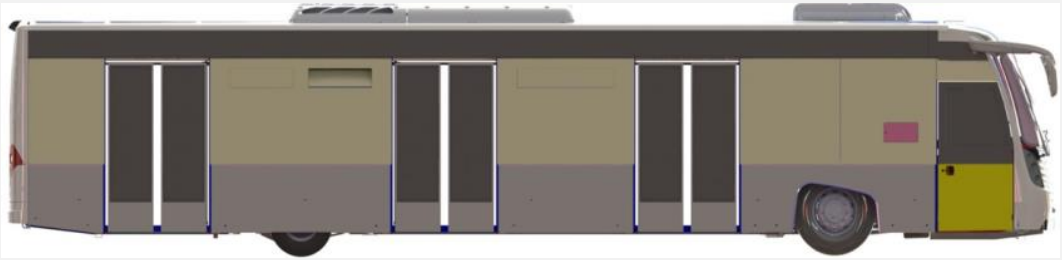
Time with 2 cores: 2.11 h
Time with 24 cores: 20 mins
Speed up: 6x

Presentation Overview

- What is FEA?
- **Real Application cases**
- Conclusions

Bus design optimization

- **Description**
  - Decrease bus weight and fuel consumption
  - Standardize chassis platform, materials and components
  - Keep or improve the bus features
  - Decrease the development time
  - Identify the essential studies to be performed in simulation and increase “Know how” for future projects

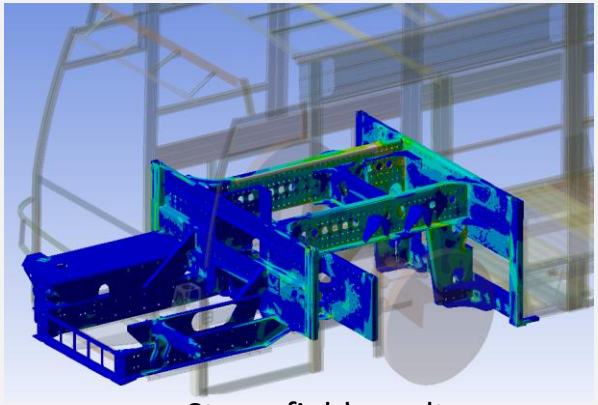


CAETANOBUS
GRUPO SALVADOR CAETANO

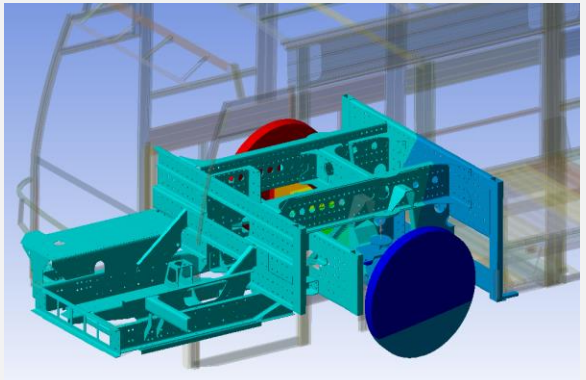
Courtesy of Caetanobus

Bus design optimization

- **Result**
  - Development **time** and cost is reduced (62%)
  - Weight reduction: 387 kg (only optimizing the chassis)
  - ANSYS model is **validated with experimental data**
  - Ability to predict and analyze the whole vehicle solicitations with ANSYS allows for a better product knowledge.
  - Existing parts are improved according to simulation results.
  - In a very complex model, with important simplifications, **less than 10% fidelity error is achieved**



Stress field result



Displacement field result

Courtesy of Caetanobus

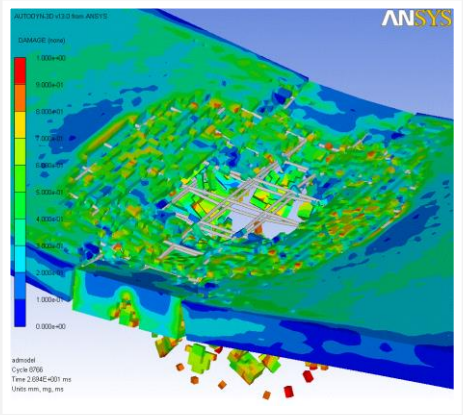
Structural behavior in explosions

- Material parameters selection metodology
- Several models validation by comparing simulation and physical testing
- Failure modes and cracks representation
- Boundary conditions analysis

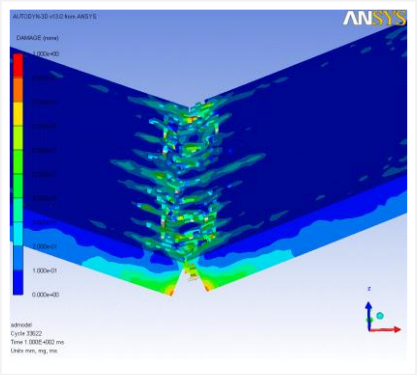
Courtesy of euroestudios

Structural behavior in explosions

- Results for a high TNT load and H=0.5m



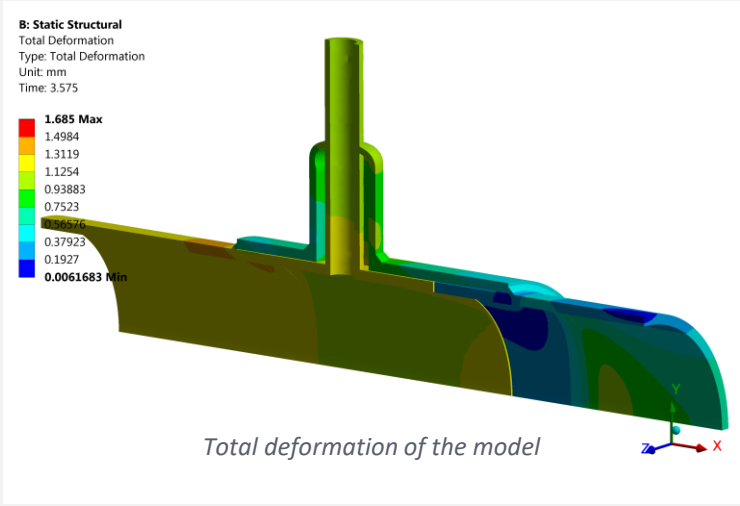
- Results for a high TNT load and H=1m



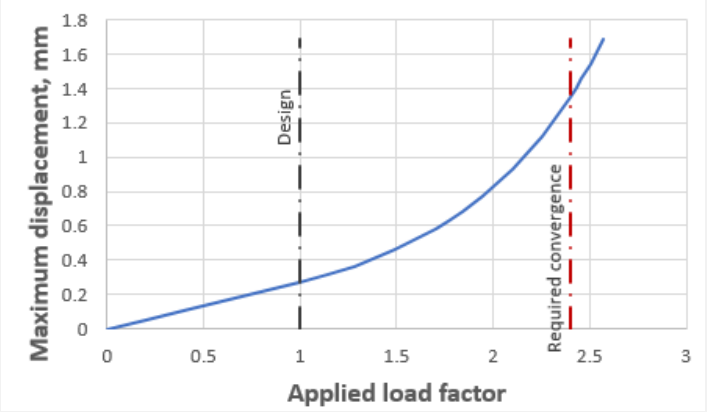
Courtesy of euroestudios

ASME local failure assessment

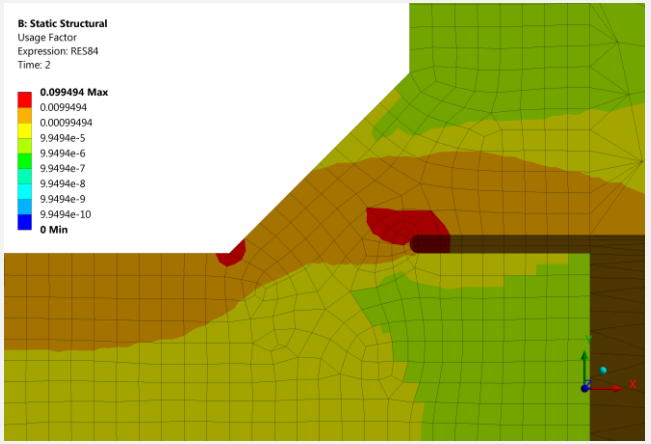
- **Objective:**
  - ✓ To perform an ASME local failure assessment in order to verify the seal box cover
- **Challenges:**
  - ✓ Circumferential leaking crack developed in steam line. In order to avoid unplanned shutdown, it was proposed to install a welded leak seal box over the cracked line
- **Solution:**
  - ✓ Finite element analysis was used to evaluate the design of the box.
  - ✓ Convergence was achieved to more than the required load factor of 2.4.



Total deformation of the model



Model Convergence



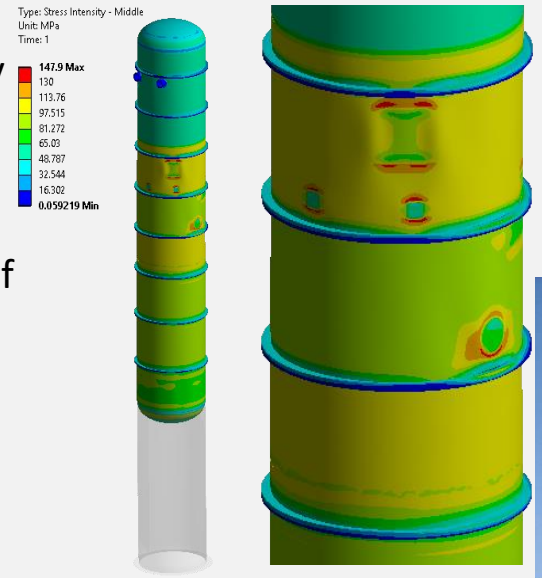
Local Failure Evaluation as per ASME Code

Bulged Pressure Vessel

- **Objective:**
  - ✓ To verify the heavily corroded & renovated column by its structural integrity for various working conditions.

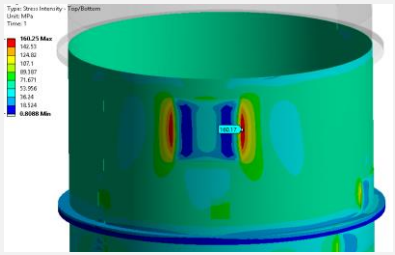
- **Challenges:**
  - ✓ Existing column built in 1978. Critical analysis phase of design, as a failure prediction would indicate shutdown of the plant.

- **Solution:**
  - ✓ Various input were considered: internal pressure, nozzle, pipe support, insulation, operating liquid and self-weight. The critical locations in the column and the maximum pressure at which the column can safely operate were predicted.



Column-Stress contour plot

Actual structure



Stress contour plot near the bulge

Industrial Cases

Problem

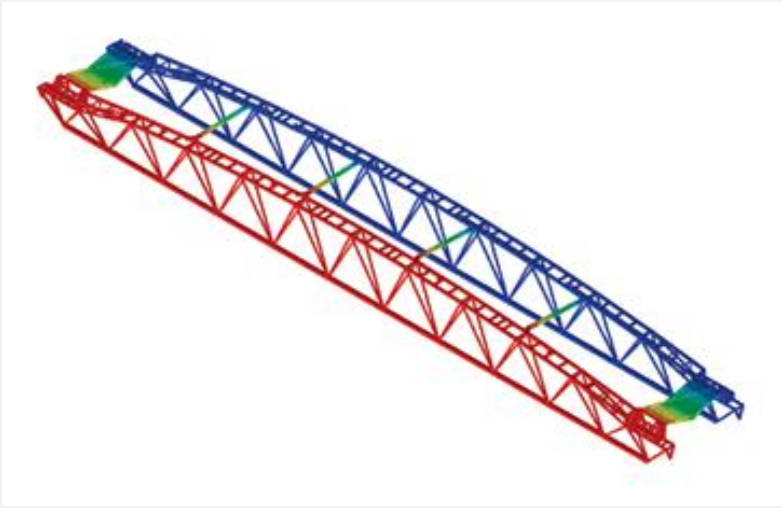
Verify the design of the new retractable roof at Wimbledon

Solution

Conduct both static and dynamic analyses to ensure that the roof would perform properly under real-world loads and stresses.

Result

The retractable roof made its Championships debut in mid-2009, when rain would have otherwise interrupted a match. As the two roof sections came together, the capacity crowd rose in a standing ovation.



Industrial Cases

Problem

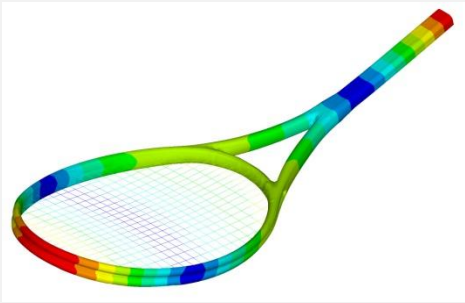
The tennis equipment market is categorized by a process of continual innovation and the need to produce advanced equipment with very short product cycles.

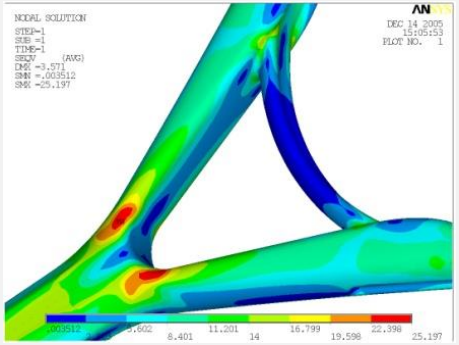
Solution

HEAD developers used ANSYS Mechanical to evaluate **1 million designs in about a week** to improve the structure of the racket and used ANSYS Parametric Design Language to automatically run each test on new designs.

Result

Racket technology continues to improve at a rapid pace and HEAD rackets have helped top tennis players secure tournament victories, including three Grand Slams in 2011.

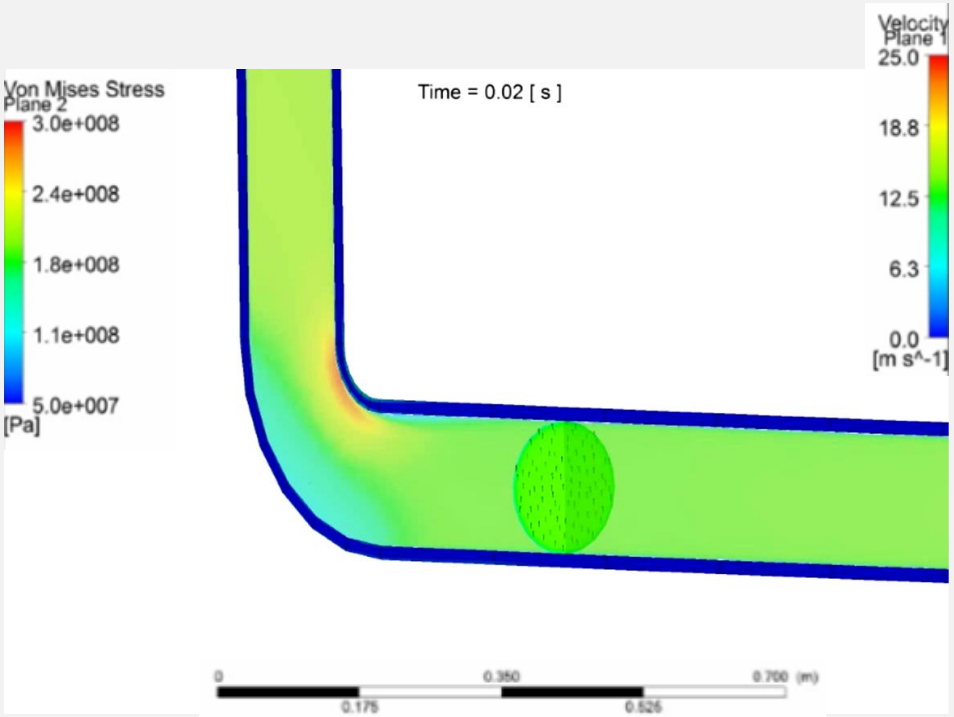




HEAD Sport is supported in this work by CADFEM GmbH.

Application Examples

- 2-way FSI analyses of a Jumper Pipe
- Goal: Simulate the varying internal flow due to deformation of the pipe. Mixing properties between oil and gas will be affected by the deforming boundaries
- ANSYS products used
  - Fluent
  - Mechanical
- Other Industrial Applications
  - Process equipment, slug flow
  - Nuclear piping



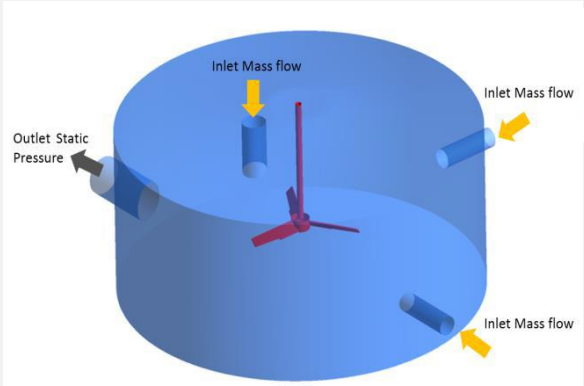
Transferred quantities:

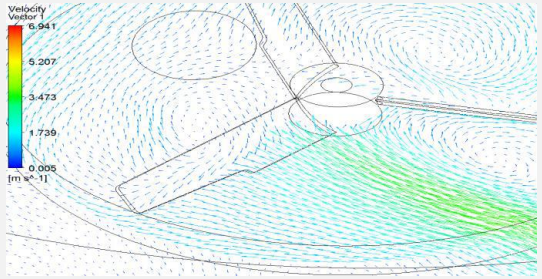
- *Force & Displacements*

Reactor tank with agitator

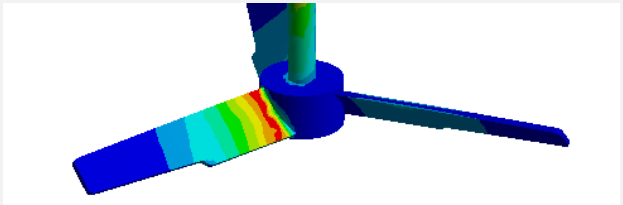
- **Objective:**
  - ✓ To predict the failure of Agitator blades by mapping the pressure loads from CFD and to do the FEA Analysis by the mapped pressure.

- **Challenges:**
  - ✓ The complex Fluid structure interaction (FSI), data has to be transferred to different physics.
- **Solution:**
  - ✓ CFD & FEA analysis was performed only on the shaft and agitator. Due to higher velocity jet hitting the blade at 0.125 span we can notice that higher pressure are noted and the same results in higher blade loading. FEA stress distribution matches with the actual failure location on the blades.





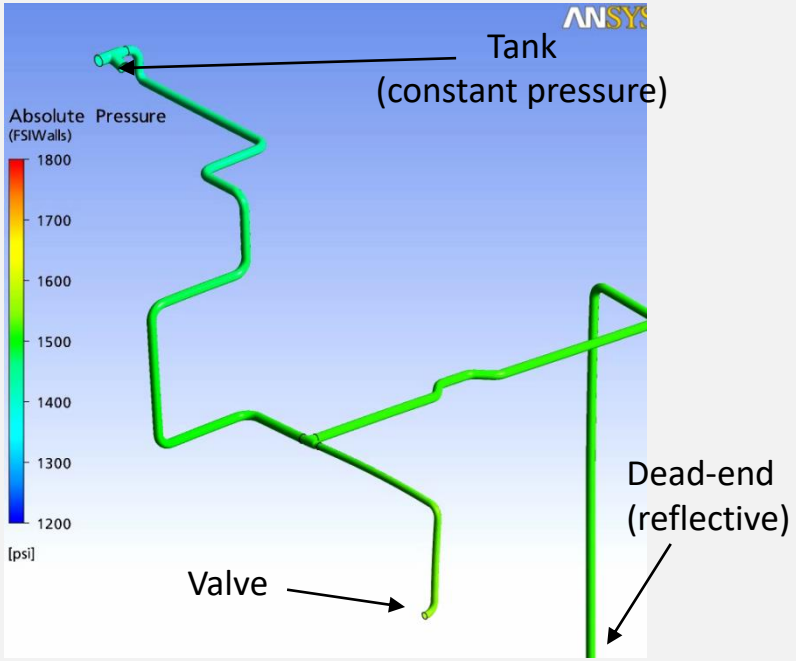
Velocity vectors showing the fluid behavior during agitator in motion

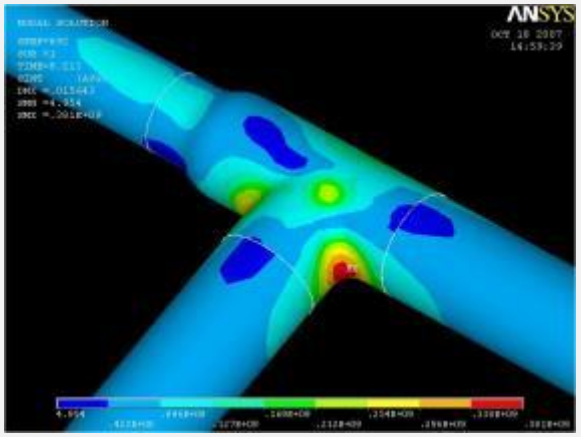


Contour plot indicating the stress on agitator blades.

Examples - Water Hammer Effect

- 2-way FSI analyses of water hammer effect in a nuclear reactor cooling line during shut-down
- Goal: Simulate the system characteristics WRT stresses, deformations & damping during a pressure pulse.
- ANSYS products used
  - CFX
  - Mechanical
- Other Industrial Applications
  - Oil & Gas pipelines





Transferred quantities:

- *Force*
- *Displacements*

Presentation Overview

- What is FEA?
- Real Application cases
- **Conclusions**

A summary of ANSYS Strengths

- **Finite Element Analysis (FEA)** is a way to simulate loading conditions on a design and determine the design's response to those conditions.
- ANSYS has enhanced capabilities in **meshing, contacts, physics interaction**, **solver performance** and **ease of use**
- **HPC technology is needed for getting the results faster**
- ANSYS offers solutions which amplify engineering **productivity for a wide range of industries and applications**