Introduction to ANSYS Mechanical
Presentation Overview

- What is FEA?
- Real Application cases
- Conclusions
Presentation Overview

- What is FEA?
- Real Application cases
- Conclusions
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

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

- **Equation solved for a static analysis**
- **Stiffness matrix**
- **Displacement vector**
- **External forces**
\[
\begin{bmatrix}
K
\end{bmatrix}
\begin{bmatrix}
u
\end{bmatrix}
=
\begin{bmatrix}
F
\end{bmatrix}
\]
\[
\left[ K(u) \right] \{u\} = \{F(u)\}
\]
Modal analysis is used to determine a structure’s vibration characteristics, i.e., natural frequencies and mode shapes.
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.
**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.

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

- Steady State and Transient

\[ [K(T)][T] = \{Q(T)\} \]
Simulation of a NOZZLE joined to a pipe by pre-stressed screws

What about getting the solution FASTER?

HPC chart for number of cores vs total simulation time

- 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
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](Stress_field_result.png)

![Displacement field result](Displacement_field_result.png)

*Courtesy of Caetanobus*
Structural behavior in explosions

- Material parameters selection methodology
- Several models validation by comparing simulation and physical testing
- Failure modes and cracks representation
- Boundary conditions analysis
Structural behavior in explosions

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

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

Transfered 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**