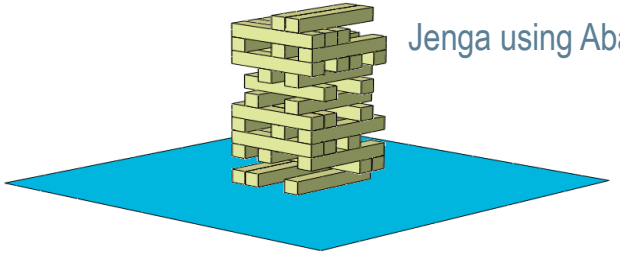

3DEXPERIENCE

SIMULIA

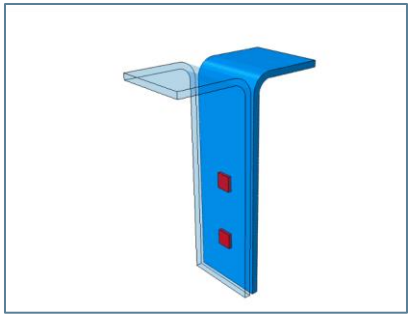
Realistic Simulation made easy

# IF WE have a robust nonlinear simulation tool, will we be able stop making assumption and perform realistic simulations?

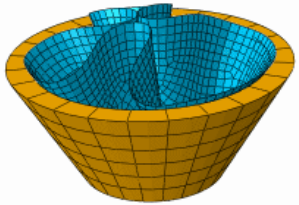


Jenga using Abaqus

Awesome contact functionality



Advanced joining techniques to simulate adhesives and delamination



Wipe dispensing

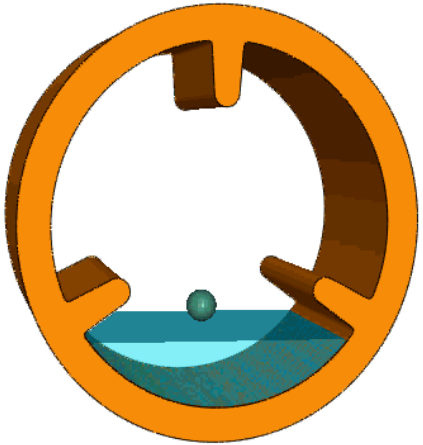
Large deformation with changing contact conditions

*Stent deployment*

Large Material model database from linear to highly non linear

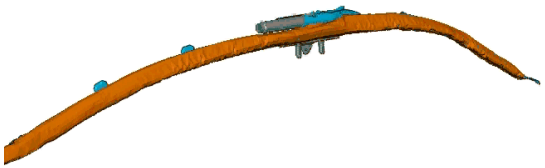
# IF WE had one tool to model all the physics in a simulation, wouldn't that help us make better products?

*Tire rolling on a road with water buildup*



*Front-load washing machine*

Courtesy of TAKATA



*Side curtain airbag deployment*


3DEXPERIENCE

SIMULIA

**Jan Granlund**

Technical Sales Expert, SIMULIA EuroNordics
December 2014
Jan.Granlund@3ds.com

# SIMULIA - The Dassault Systemes brand dedicated to making…

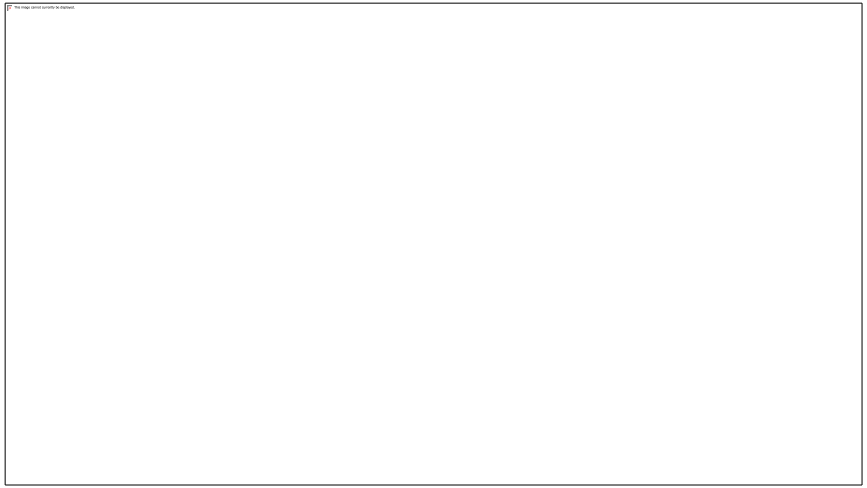
**Realistic Simulation**

*an* ***integral business practice***

to Explore, Discover, Understand, Improve

***product, life, & nature***

# …to Reduce Physical Testing, Save Time and Money, Improve Quality….



***Courtesy Mechanical Design and Analysis Corporation, 2010 SCC***

# SIMULIA

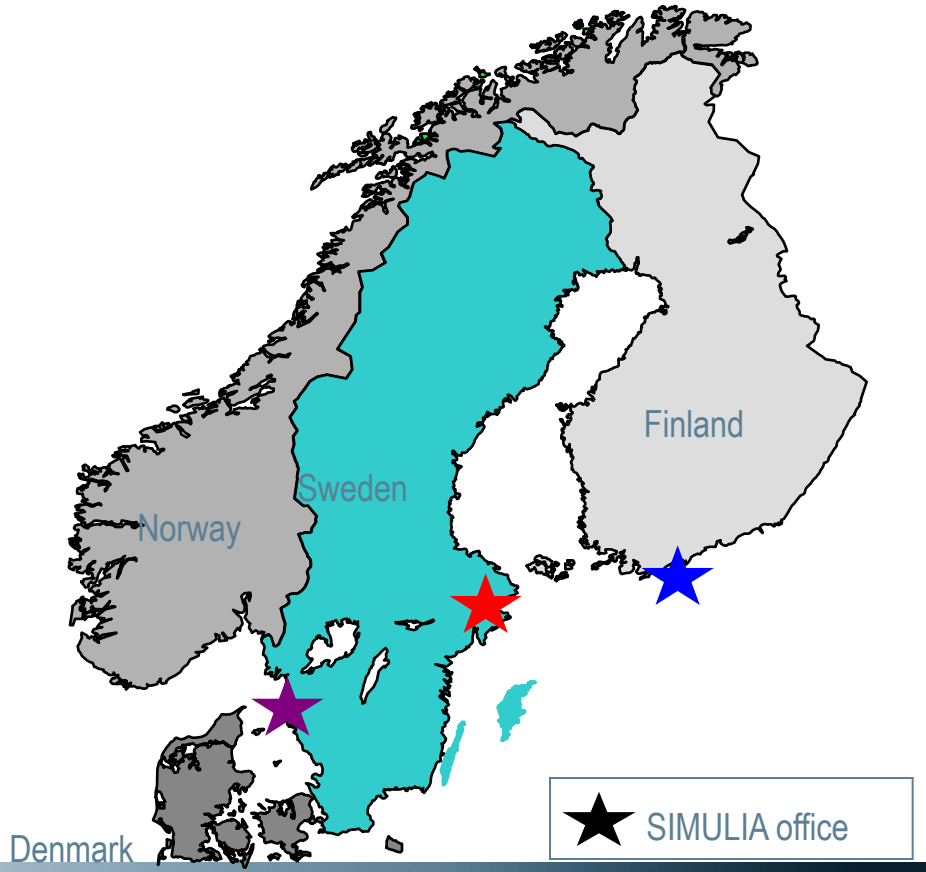
- SIMULIA is not a product, or a company. It as a brand populated by products:
  - Abaqus Analysis Products
  - Abaqus/CAE – Interactive pre- and postprocessor
  - Isight for Process Management and Optimization,  Tosca for Shape and Topology Optimization  and FE-Safe for Durability and Fatigue
  - ….and some more…

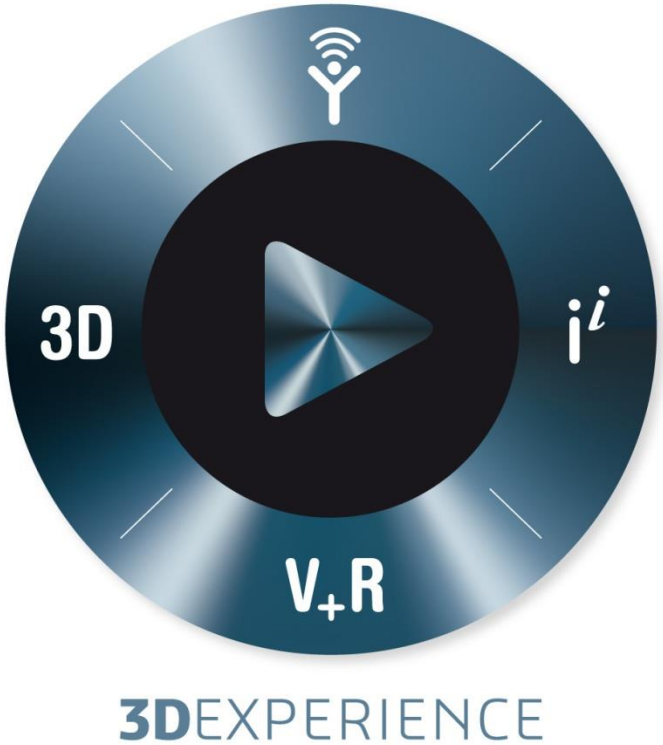
# Global Presence – Local Support

- Brand headquarters in Providence, RI
  - More than 1000 employees Worldwide
  - 30+ Centers for Simulation Excellence
- Strong focus on R&D
  - 12 R&D labs
  - Technical experts focusing on each industry – example Life Sciences, Automotive, Energy, etc.
- Dedicated to:
  - High-quality products and support
  - Innovative technology
  - Customer satisfaction


# SIMULIA EuroNordics

- Sweden
  - Göteborg
    - Jonas Dyberg. Manager Nordic, sales
    - Anton Jurinic. Engineering Service Manager, consulting
    - Joakim Asklund. Consulting, support, training
    - Ulf Karlsson. Key Account Manager, sales
    - Jan Rydin. Consulting, support, training
    - Martin Roswall, Consulting, support, training
  - Stockholm
    - Jan Granlund. Business Development, consulting, training
    - Håkan Lind. Support, training, consulting
    - Anders Winkler. FE-Safe
- Finland
  - Vantaa
    - Kari Saarinen. Manager Finland, sales
    - Mikko Ollila. Consulting, training, support
    - Reijo Lindgren. Consulting, training, support

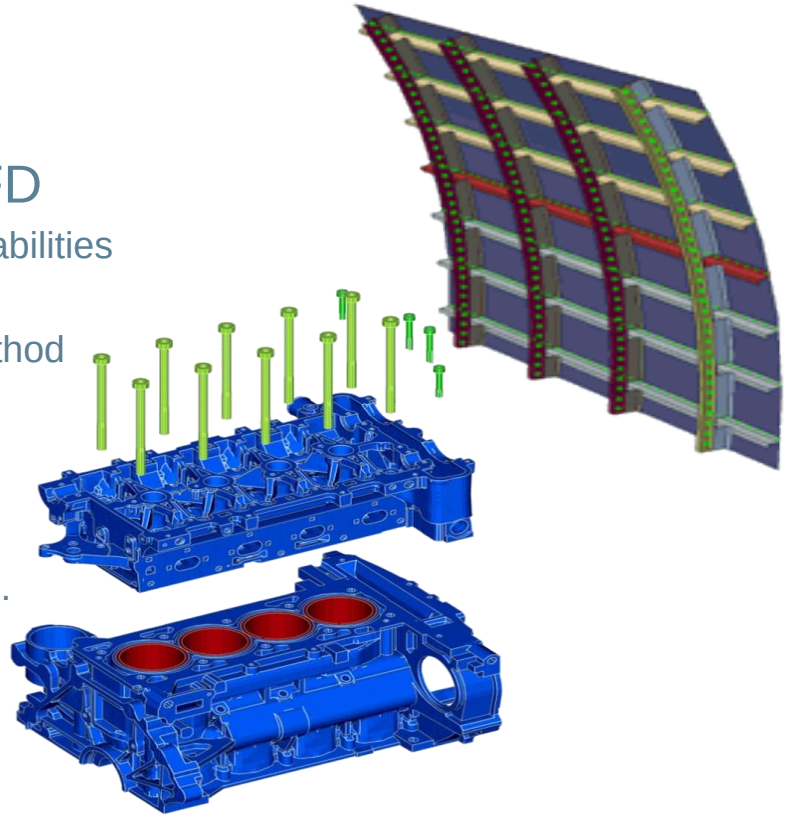





# Abaqus Overview

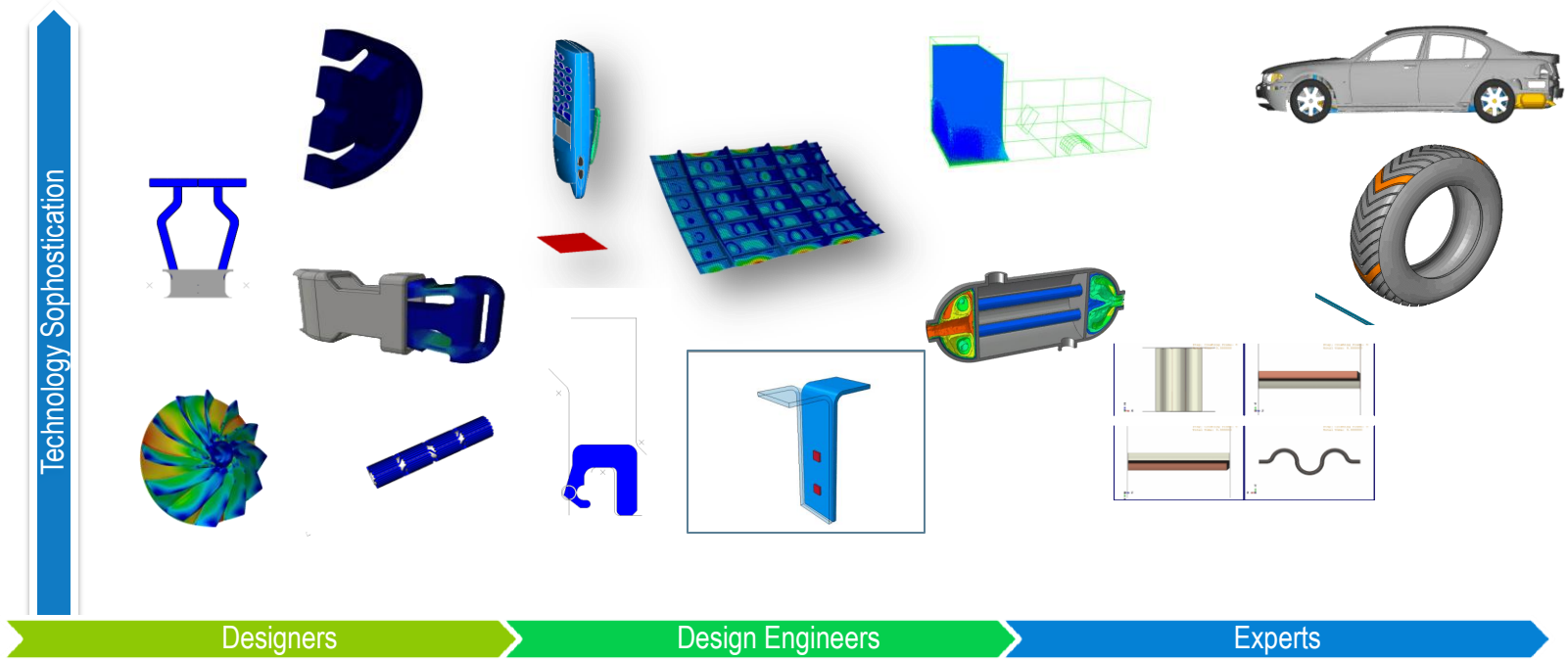
# Abaqus products

- Abaqus/Standard, /Explicit & /CFD
  - Family of mostly nonlinear engineering capabilities
  - Designed for production use
  - Mathematical basis is the finite element method
- Abaqus/CAE:
  - Complete Abaqus Environment
  - For Pre/Post processing of Abaqus analysis.



# One tool for all

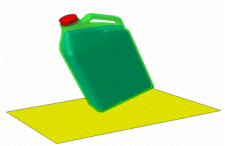
## Simulation solution covering Designers to Simulation Experts



# Solutions for the entire range of industries

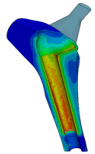
## Consumer Packaged Goods

Plastic and Glass Forming
Conveyor Systems
Container Drop
Pressure Analysis
Thermal Analysis
Bottle Sealing
Adhesives



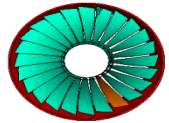
## Life Sciences

Tissue Modeling
Surgical Equipment
Stents
Drug Delivery
Orthopedics
Medical Packaging



## Aerospace & Defense

Avionics
Landing Gear
Aerostructures
Aeroengines
Composites
Defense Systems
Space Systems

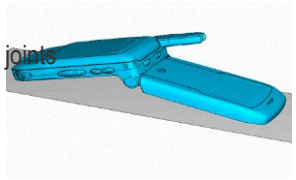


## Automotive & Transportation

Chassis
Body
Tires
Interiors
Crashworthiness
Brake Systems
Powertrain
Electronics

## High Tech

Thermal cycling of solder joints
Drop Testing
Vibration Analysis
Semiconductors
Circuit Boards
Hand-held Devices
Computers & Peripherals

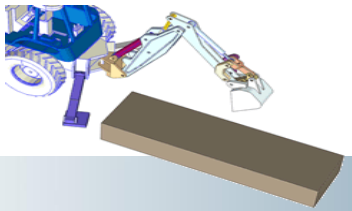


## Architecture & Construction

Earthquake loading
Structural integrity due to fire
Concrete analysis
Soil-pore interaction
Failure limits

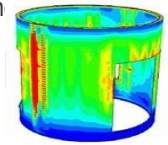
## Industrial Equipment

Nonlinear Stress Analysis
Thermal Analysis
Cyclic Loading
Flexible Multibody Dynamics
Soil-Structure Interaction



## Energy

Wave loading on rigs and pipelin
Piping and pressure vessels
Thermal analysis
Blast loading
Drop or Impact



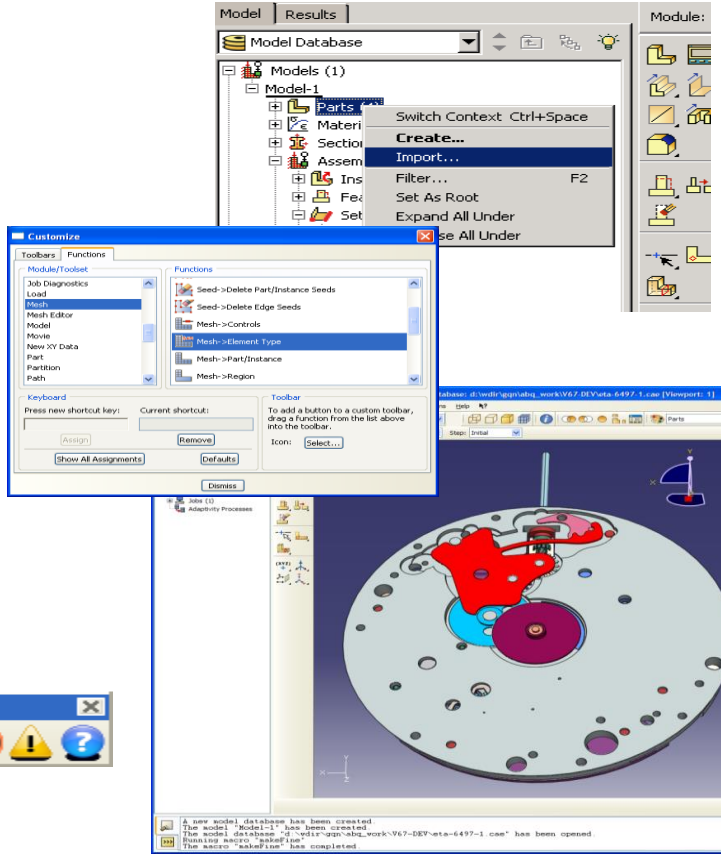

3DEXPERIENCE

# Abaqus/CAE

# Abaqus/CAE

## Modern graphical interface

- Modular & well organized
- Easy to learn & use
- Object-action paradigm
- Model Tree
- Customizable toolbars

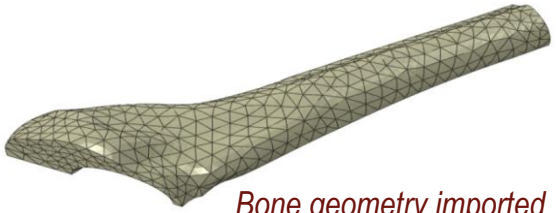



- A short history
  - Birth: 1999
  - Evolution: 2000-2003
  - Windows native: 2004
  - Aggressive development: 2005+

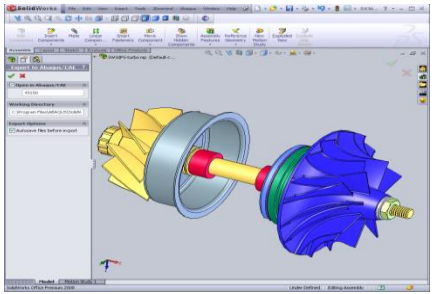
# Abaqus/CAE

## Geometry & model import

- Create moderately complex geometry
- Associative CAD interfaces preserve analysis attributes when geometry changes
- Complete set of geometry repair tools

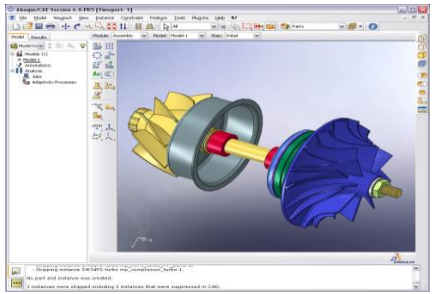


*Bone geometry imported from STL file*

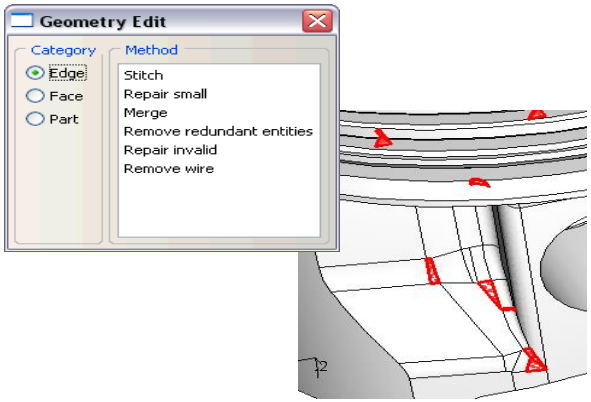


*Solidworks*

*Solidworks AI*



*Abaqus/CAE*

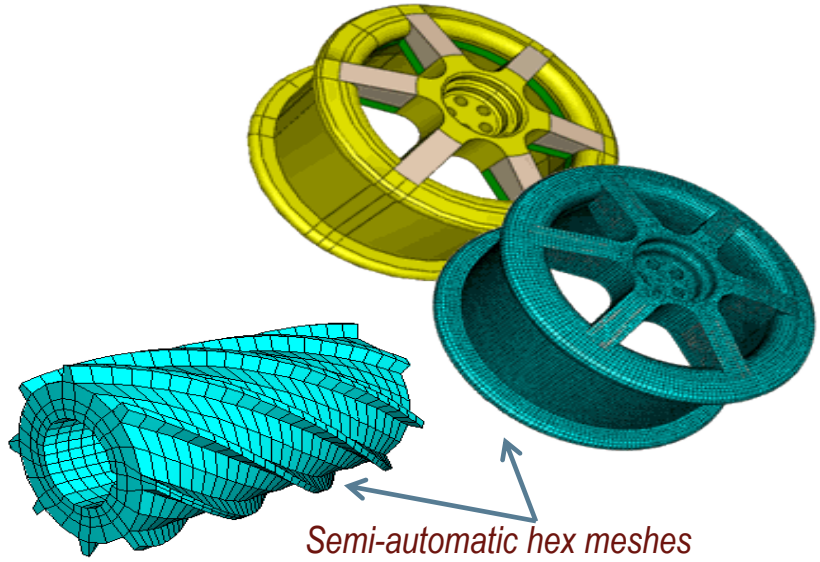



*Geometry repair toolset*

# Abaqus/CAE

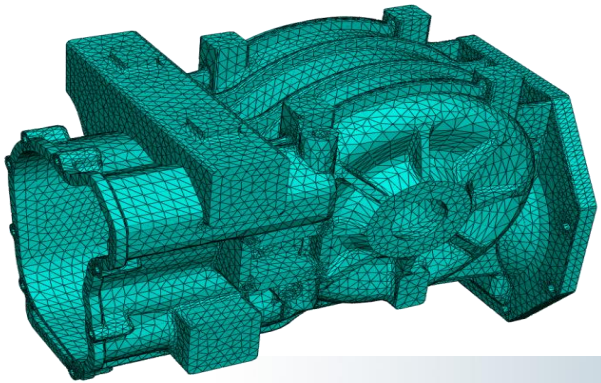
## Powerful & flexible meshing

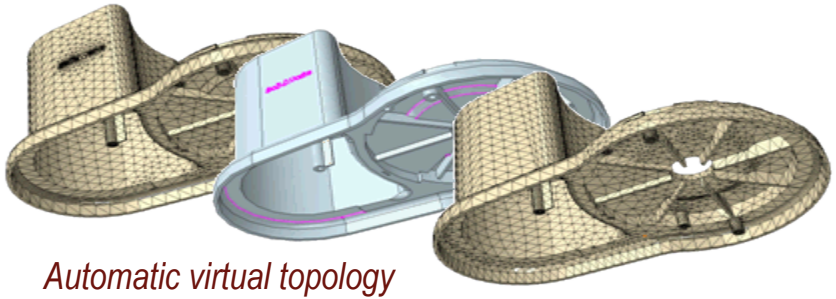
- Fast, automatic free meshing
- Semi-automatic hex meshing
- Abstract away unimportant features using virtual topology



*Semi-automatic hex meshes*

*Automatic free mesh*



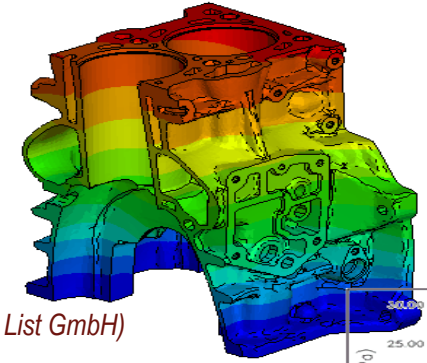


*Automatic virtual topology removes step-like feature*

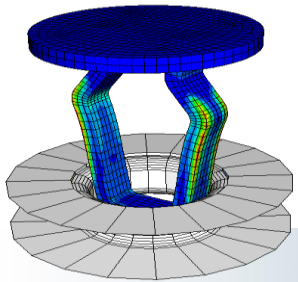
# Abaqus/CAE

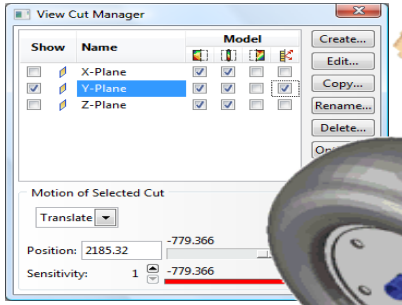
## Preferred tool for visualizing Abaqus results

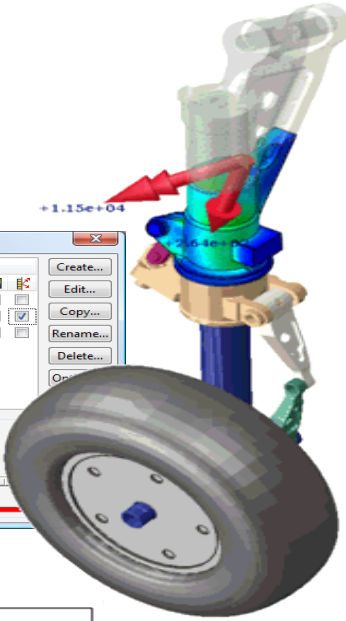
- High performance for large models
- Transparency
- View cuts
- Probing
- XY plotting
- Much more . . .

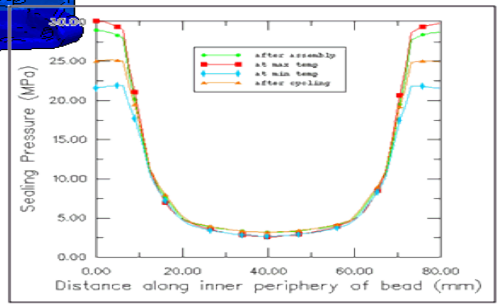


*(Courtesy of AVL List GmbH)*





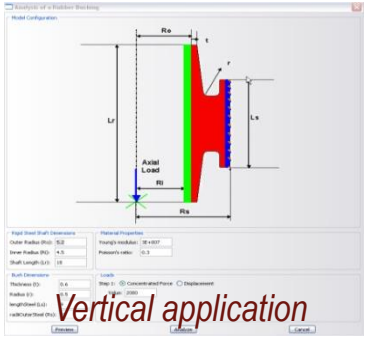



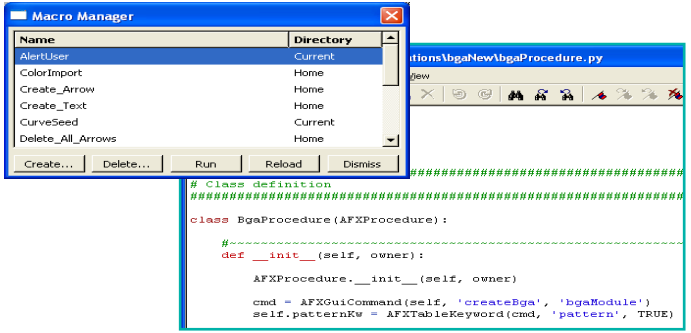
# Abaqus/CAE

## Powerful extensibility & customization

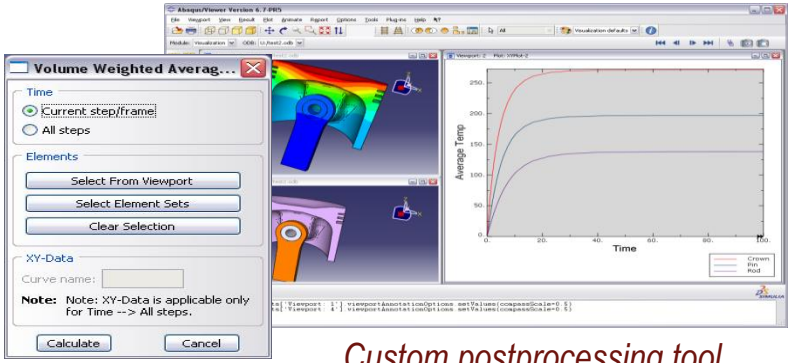
- Automate repetitive tasks with macros
- Create vertical applications for deployment of sophisticated workflows
- Open-standard Python scripting language
- Drag-and-drop GUI builder




*Vertical application*



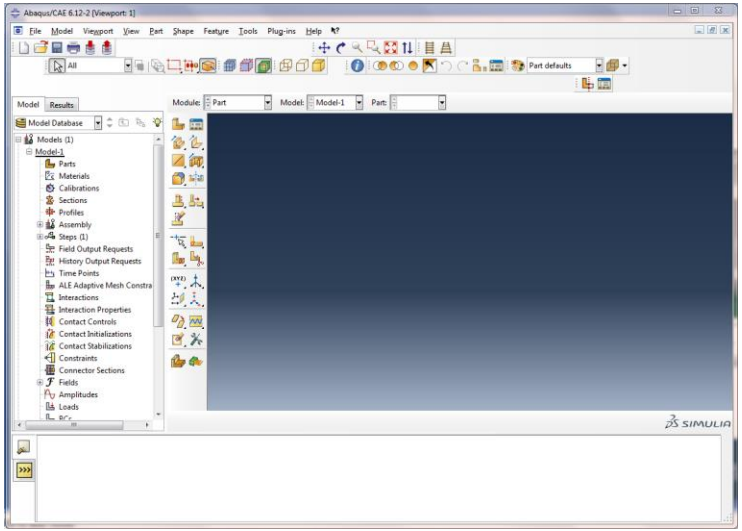

*Macro-manager and scripting language*



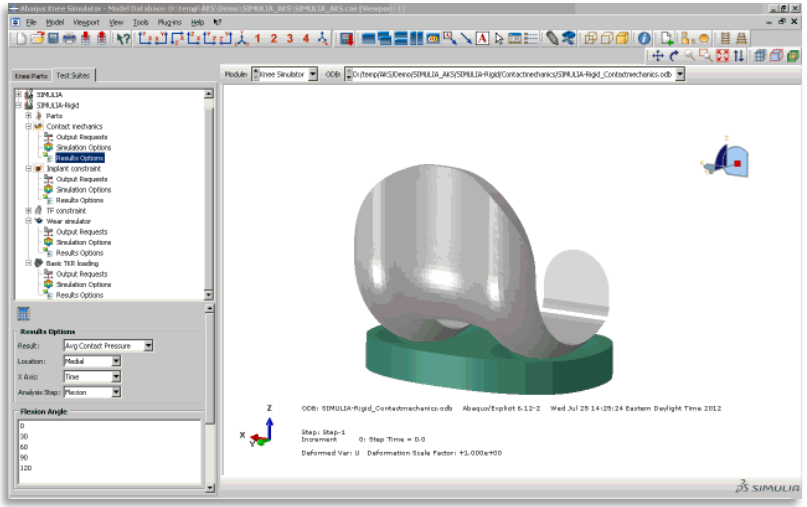

*Custom postprocessing tool*

# Abaqus/CAE

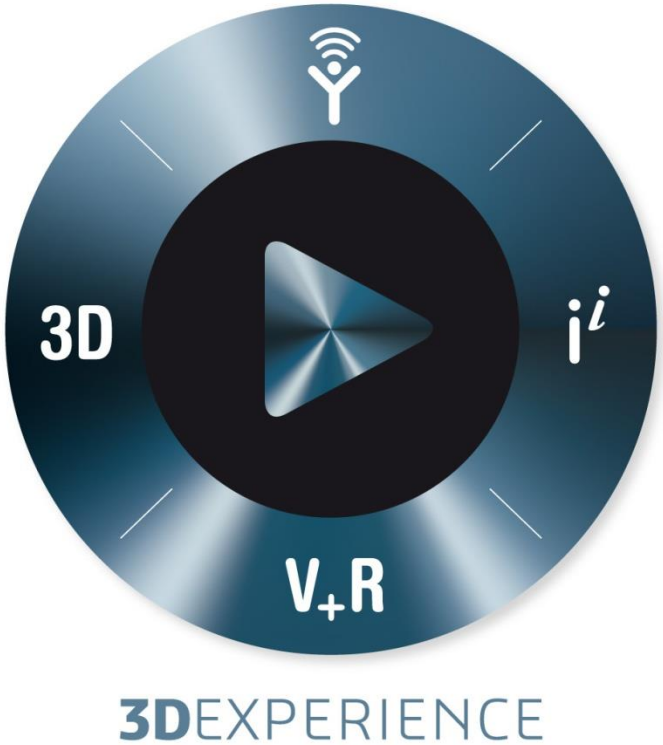
## Customizable graphical interface



Out-of-the-box Abaqus/CAE



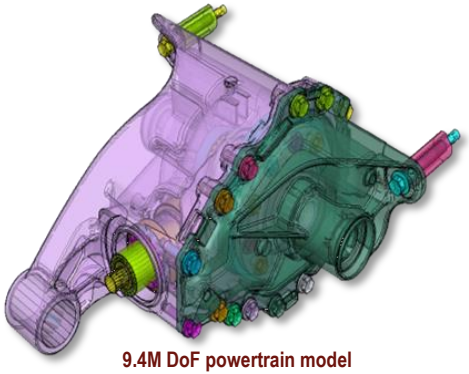
Abaqus Knee Simulator



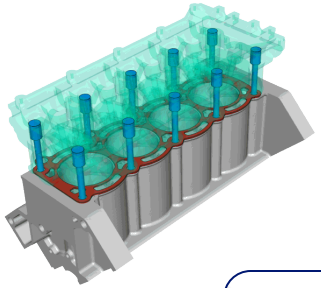

# Abaqus Analysis Products

# Robust Accurate Nonlinear Solver

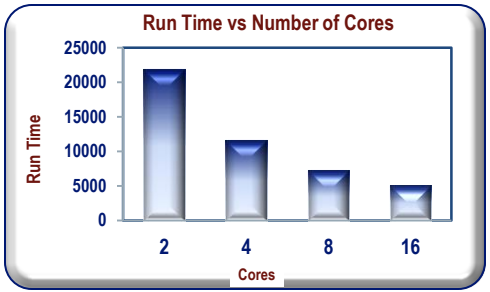
- Robust solver
  - Designed to handle model of any size.
- Fast solving
  - Marches to solution faster
  - Scales efficiently when using multiple cores
- Reliable & Accurate



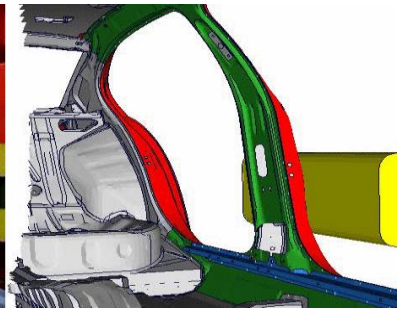
9.4M DoF powertrain model
(Courtesy Dana Holding Corp.)



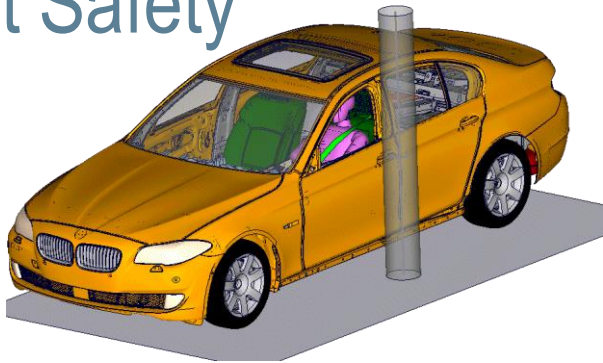
4M DoF
Engine Block Model




- A short history
  - Birth: 1978
  - Explicit: 1992
  - Abaqus 6.1: 2000
  - CFD 2010

# BMW Crashworthiness and Occupant Safety



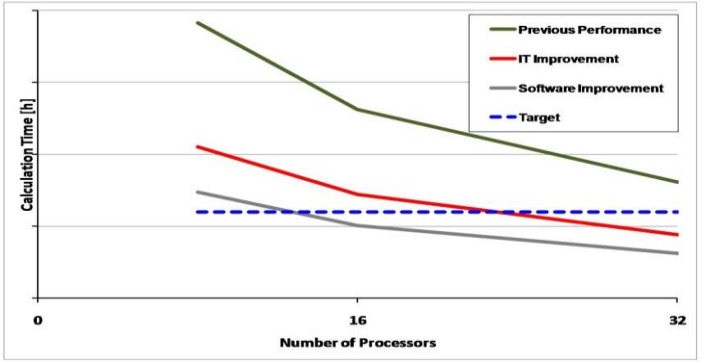
## State of the Art – HPC Performance

*"…the goal at BMW to completely eliminate prototype hardware and testing, such issues can only be subsequently evaluated through simulation."*

Volume of crashworthiness simulations
to be carried out continues to increase

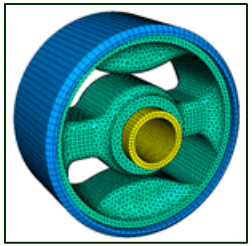
- Crash models continue to grow in size and complexity
- Target "24-hr turnaround time" will be achieved in a short timeframe

**Performance** (New BMW Method)

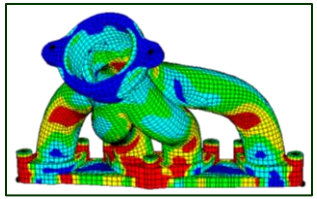


# Extensive Material models

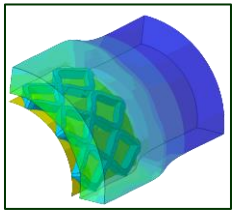
- Metals, rubbers, and composites
  - Linear/nonlinear elasticity and plasticity
  - Isotropic or anisotropic
  - Rate and temperature dependence
- Additional materials include:
  - Soils and rocks
  - Concrete and ceramics
  - Pastes and polymers
- Damage & failure modeling



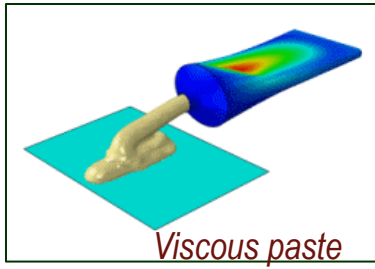
Viscoelastic rubber



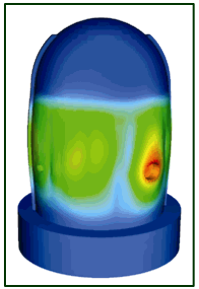
Cast iron plasticity



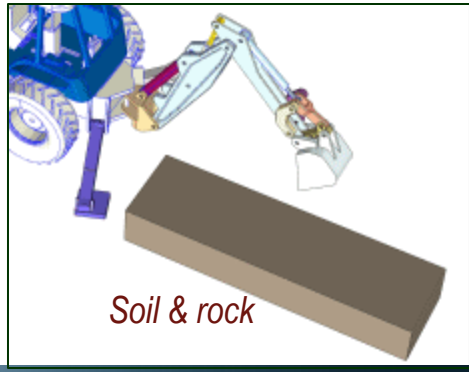
Human tissue



Viscous paste



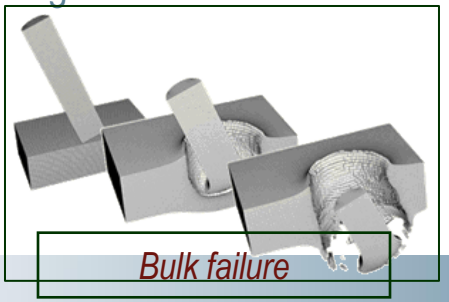
Reinforced concrete



Soil & rock

Test

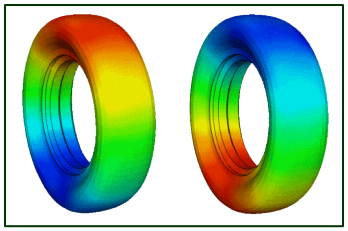
Simulation



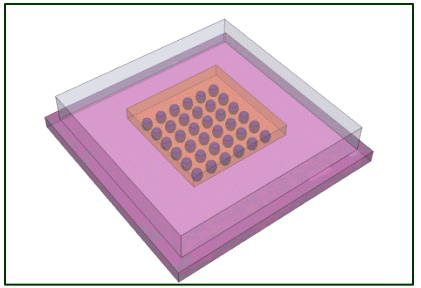
Bulk failure

# Abaqus /Standard

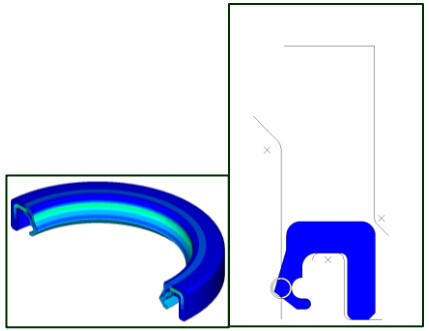
- Based on implicit solution techniques
  - Intended for static and structural dynamic events
  - Linear or nonlinear behavior



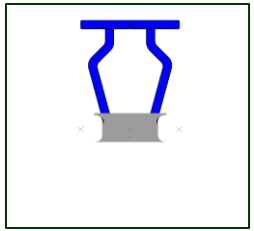
Tire noise



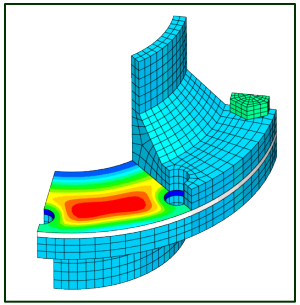
Solder joint fatigue



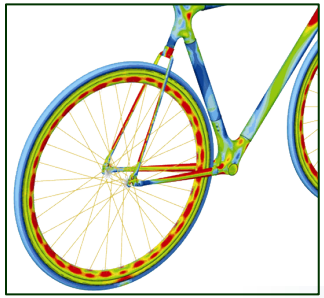
Shaft seal insertion



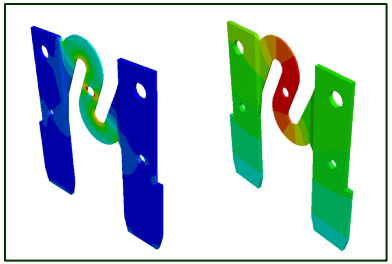
Clip insertion



Gasket sealing



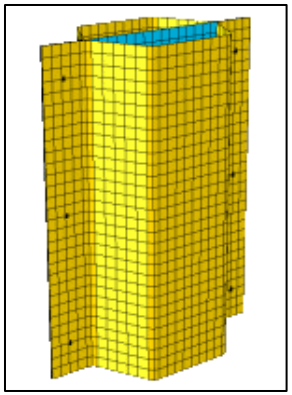
Bicycle frame stress



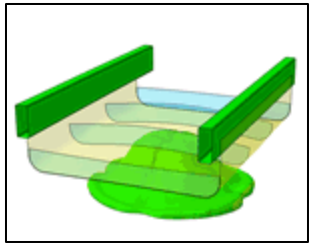
Electrical fuse heating

# Abaqus /Explicit

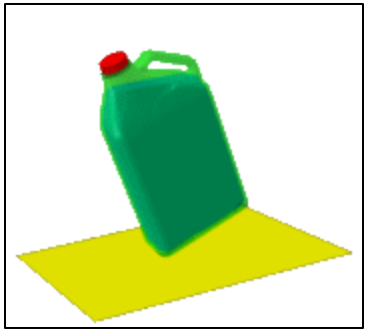
- Based on explicit time integration
  - Intended for high-speed transient dynamic events
  - Also suited for highly nonlinear quasi-static events



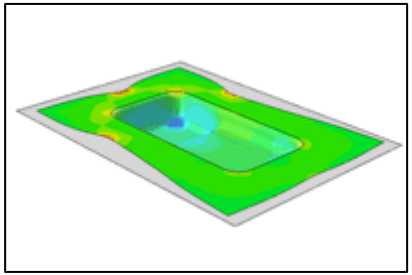
*Energy absorption*



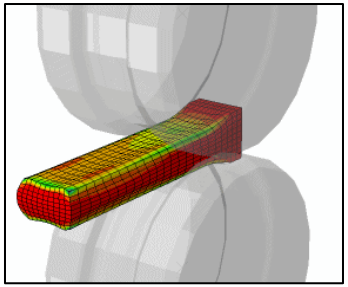
*Blast loading*



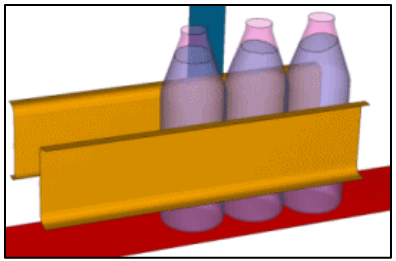
*Bottle drop*



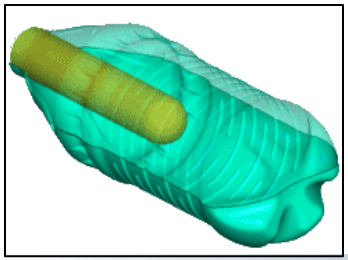
*Deep drawing*



*Metal rolling*



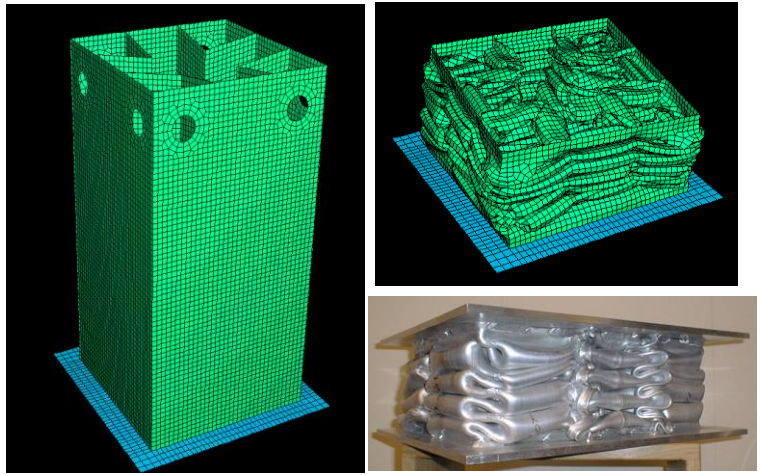
*Bottle conveying system*



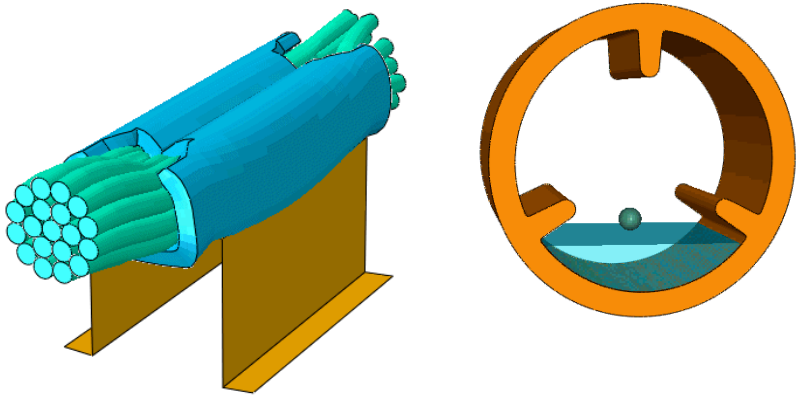
*Blow molding*

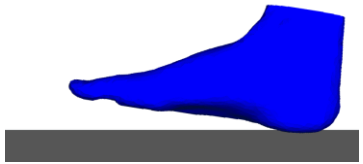
# Contact

- Extremely robust
- Accurate
- General contact capability
  - Extremely simple to setup



Courtesy of *Alcan Mass Transportation Systems*, Zürich



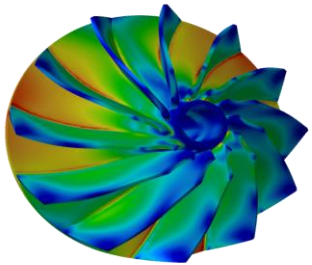


*Courtesy of BMW**

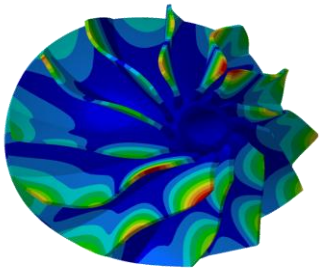
* Gholami, T., J. Lescheticky, and R. Paßmann, "Crashworthiness Simulation of Automobiles with Abaqus/Explicit," ABAQUS Users' Conference, Munich, 2003

# Advanced Analysis

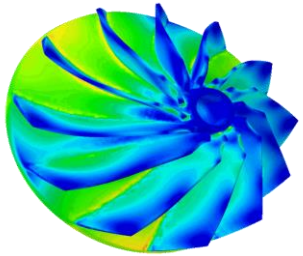

Natural frequency including preloading effects



Step1:Stress at 10000 RPM



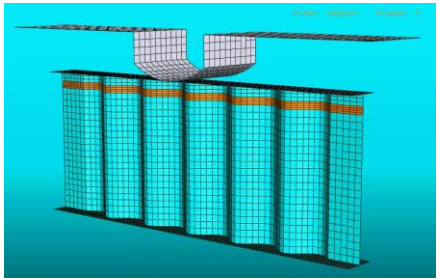
Step2:Mode shape at 9728 Hz



Final results

# Advanced Analysis

Damage & Fracture



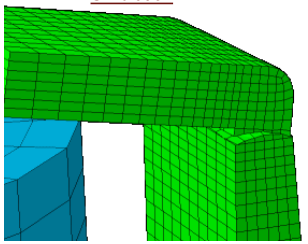
Progressive damage of fiber reinforced composite

Delamination modeling

*Test*

Crack propagation

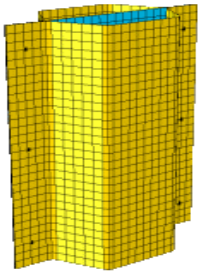
*Simulation*



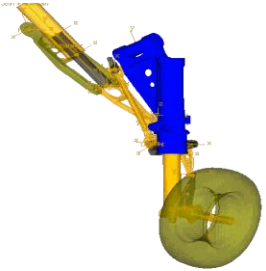
*Pulling tool*

# Advanced Analysis

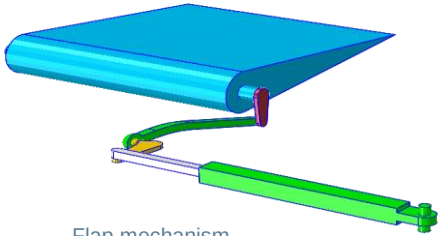

Connectors define rigid or deformable mechanism (i.e Ridig body motions, etc)



Spotwelds w/ failure



Landing gear mechanism w/ deformable parts

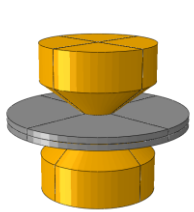


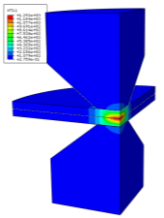
Flap mechanism

# Multiphysics Analysis

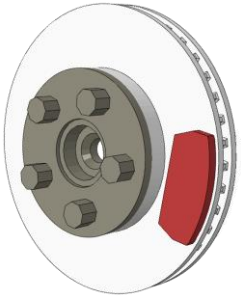
Thermal Stress Analysis

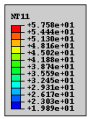
Coupled Thermal-Electrical-Stress Analysis

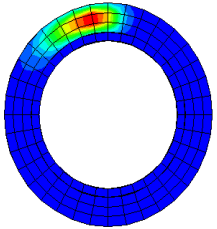




Resistance spot welding



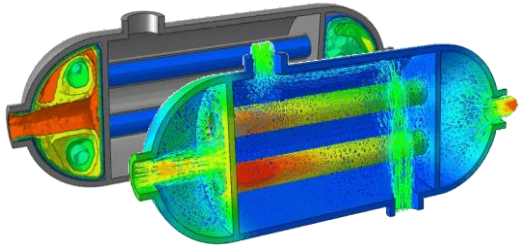




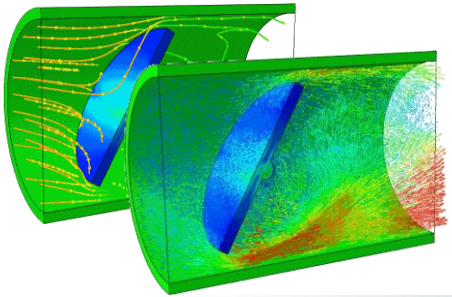
Disk brake

# Multiphysics Analysis


Water can drop test



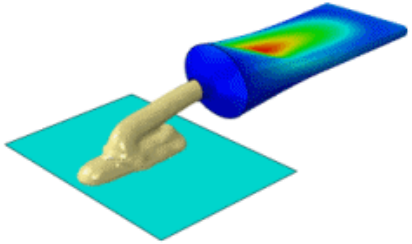
Heat Exchanger



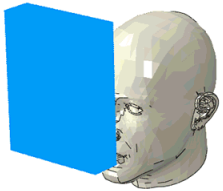
Butterfly valve

# Multiphysics Analysis

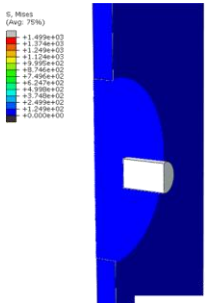

CEL -Tire hydroplaning



CEL- Paste squeeze

SPH- Water Splashing of a Figurehead

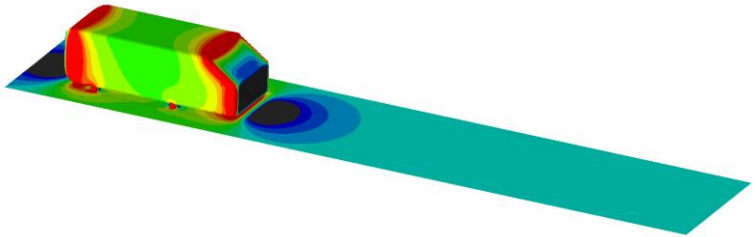


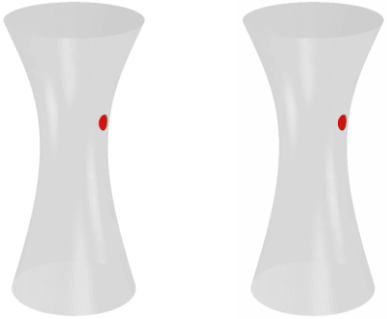
SPH- Bullet impact on a plate

# Multiphysics Analysis


- Incompressible pressure-based flow solver
  - Both Transient & Steady State
- Laminar and turbulent flow.
- Material properties support
  - Newtonian
  - Non-Newtonian
  - Temperature - dependent properties

parallel and Scalable solver





Porous copper Diffuser (85 %porosity)

Free inflow

*high voltage insulator*

# Some Industrial Applications with Abaqus


- Air Spring Buckling Investigation

  Courtesy Firestone Industrial Products

- Abaqus FEA for Oil & Gas Industry
- Abaqus for Wind Turbines

# Airsprings from Firestone Industrial Products

- An airspring is a pressurized rubber envelope that allows heavy loads to supported at very low spring rates
  - First patented by Firestone in 1938

- Airspring components are:
  - An end cap
  - A cord rubber sleeve
  - An internal piston sleeve

- As the airspring lengthens and shortens, the rubber sleeve rolls over on the piston and changes the length of the airspring with very little change in the internal force

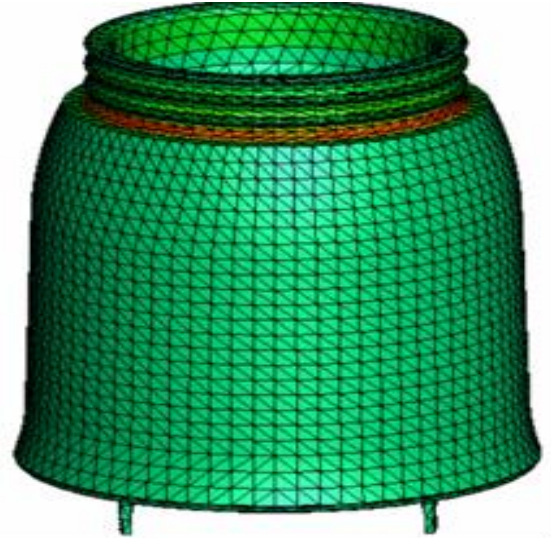
Images and animation courtesy of Firestone Industrial Products

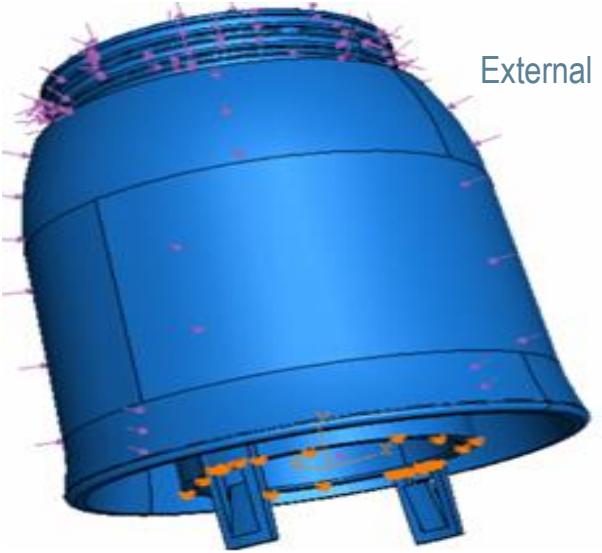
# Improved Piston Design

- To meet new customer requirements, Firestone engineers proposed a new piston design
- In testing, the prototype buckled and cracked

# Piston Buckling

- To simulate buckling, a three-dimensional piston model was created and loaded with pressure to simulate contact loads from the rolling rubber sleeve

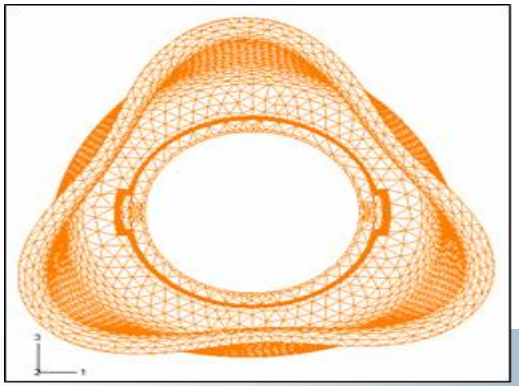


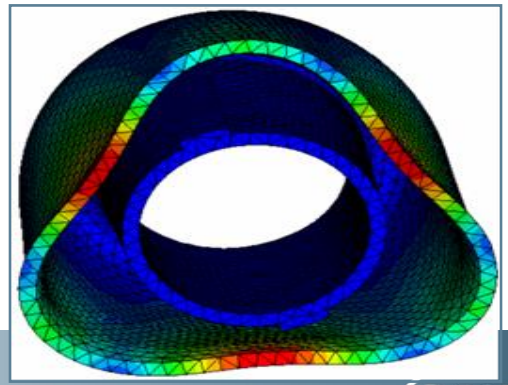


# Buckling Prediction

- Procedure to simulate buckling:
  - Extract the “buckling modes” in a buckling step
  - Apply these modes to the structure as imperfections in a new analysis
  - Use a Riks step to simulate the transition to the post-buckled state

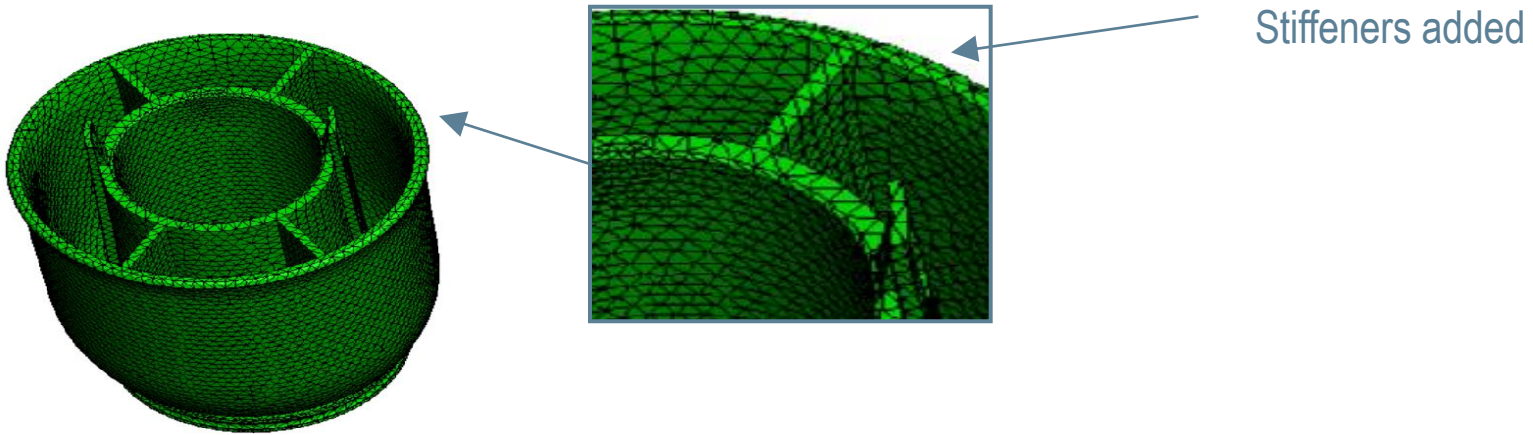
- With this procedure, Firestone engineers determined that the critical buckling pressure for the new design was *too low*





# The Redesigned Piston

- With the information from the simulation, Firestone engineers stiffened the piston with internal ribs to eliminate the buckling response




- The resulting buckling simulation showed that the critical buckling pressure was now *well above* the operating range

# Summary

- Using Abaqus, Firestone engineers were able to diagnose and eliminate an airspring piston failure found during prototype test
- The buckling simulation relied on the traditional robust nonlinear buckling functionality in Abaqus/Standard
- Buckling simulation can now be a part of the upfront airspring design procedure to reduce prototype testing and speed design of future airspring products

# Abaqus FEA for Oil & Gas Industry

- Sample Offshore Applications

Platforms

RAOs
Wave loading
Fixed, TLP, Spar

Risers

Reeling
Umbilicals
Pipe-in-pipe
Riser touch-down
VIV

Connections & Equipment

Threaded connections
Tensioners
Welds

Pipelines

Pipeline buckling
Pipeline upheaval
Pipeline walking

Foundations

Suction piles
Spud cans
Ice gouging
Anchor gouging
Trenching

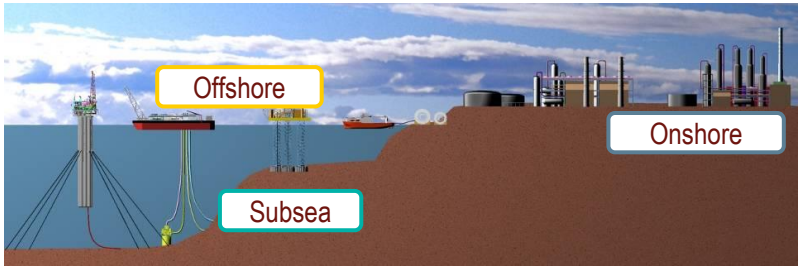
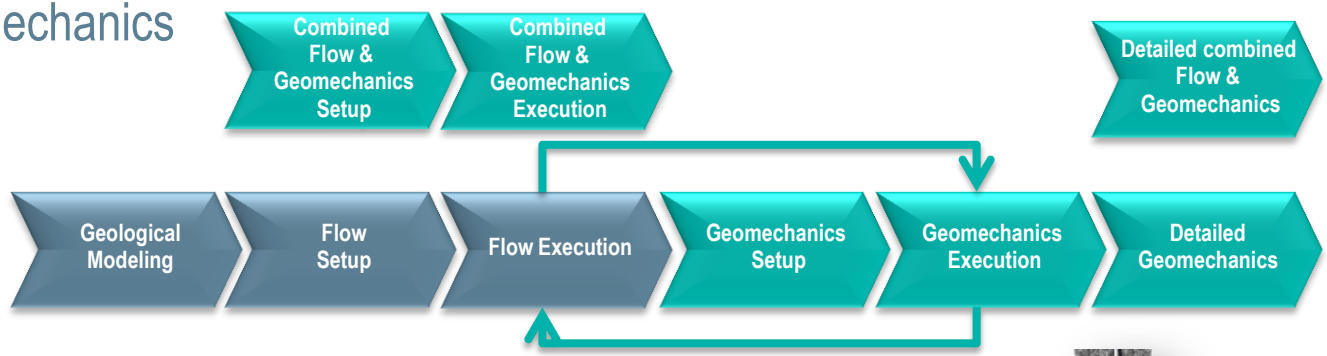
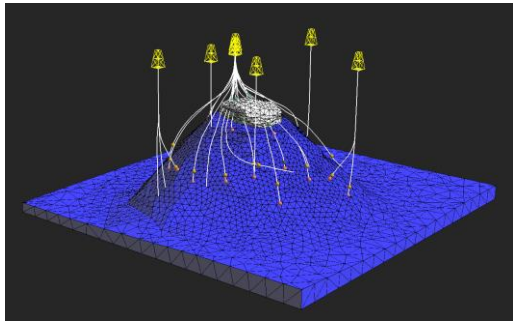



Image Courtesy: Technip

# Abaqus FEA for Oil & Gas Industry

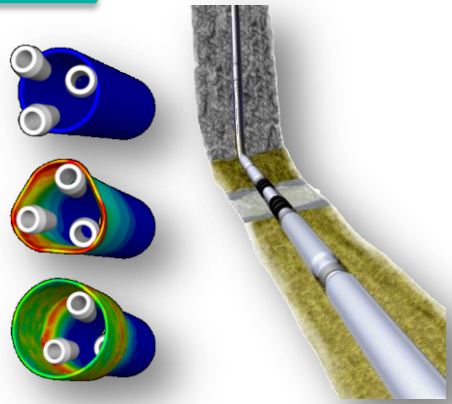
- Geomechanics



Courtesy: GMI

Courtesy: Weatherford



# Ice Scouring

## Application

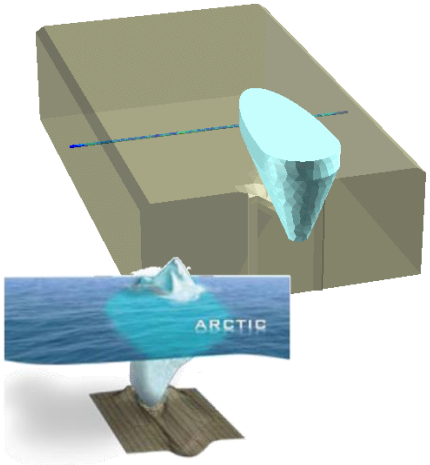
- Iceberg gouging of sea floor near buried arctic oil pipelines

## Why Abaqus Unified FEA?

- Coupled Eulerian Lagrangian capability to capture ice-soil-structure interactions accurately
- Extensive nonlinear material modeling for different types of soil.
- General contact to easily setup all contact interactions.
- Scalable parallel performance on many cores.

## Benefits

- Optimize required pipeline burial depth for safe operations in the arctic ecosystem

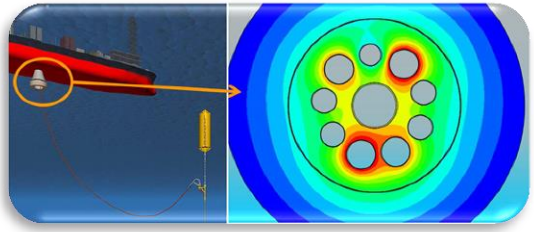



Courtesy: JP Kenny

# Thermal Management

## Applications

- Estimate peak temperatures of umbilicals within a riser due to higher temperatures in production flowlines
- Estimate temperature drop in deepwater pipelines during no-flow conditions



## Why Abaqus Unified FEA?

- 2D, Axisymmetric, and 3D heat transfer capabilities, including thermal contact
- Radiation heat transfer
- FSI for coupling heat transfer between fluid flow and structure

## Benefits

- Develop cost-effective cooling solution for umbilicals
- Develop insulation to maintain pipeline flow assurance under extreme operational conditions & upsets

Courtesy: Technip

# Composite Pipelines

## Application

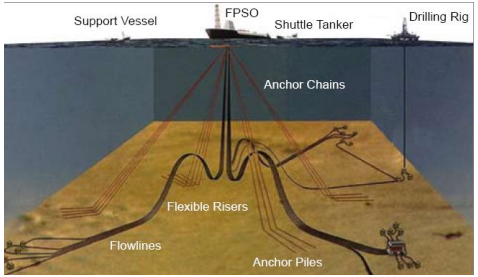
- Assess failure modes and progression of failure in composite pipelines for various loads

## Why Abaqus Unified FEA?

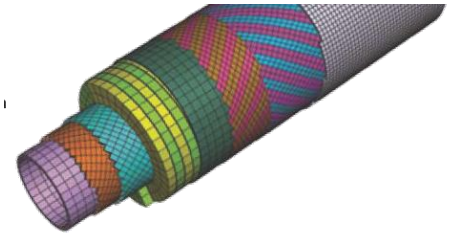
- Comprehensive composites simulation capabilities
- Combination of Abaqus/Standard and Abaqus/Explicit for different loading and operating conditions.
- Model change to simulate different conditions of the pipe
- High performance parallel solutions to minimize run times

## Benefits

- Develop an all-composite pipe that can withstand the greater external hydrostatic pressures, higher internal wellhead pressures, and temperature extremes that accompany deepwater work




Courtesy: DeepFlex



# Offshore Platforms

## Application

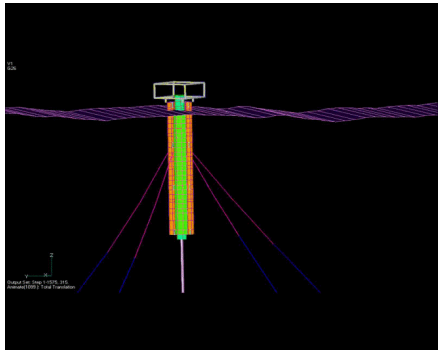
- Assess response of offshore platforms under different loading conditions

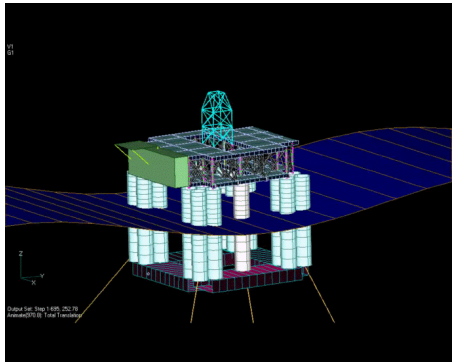
## Why Abaqus Unified FEA?

- Abaqus/Aqua for wave and wind loads
- Beams, shells, and constraints to accurately represent the structures
- Implicit and explicit dynamics for different loading conditions
- Thermal stress assessments to evaluate fire accidents

## Benefits

- Assess and improve strength, stability, and safety of offshore structures for different loading conditions, including accidental impacts





Courtesy: Horton Deepwater

# Offshore Platforms

## Application

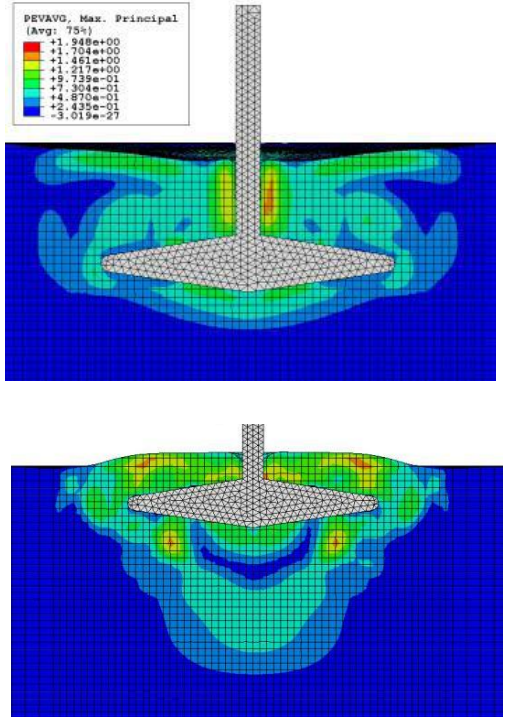
- Assess the integrity of spudcan foundations taking into account installation procedure and operational loads

## Why Abaqus Unified FEA?

- Coupled Eulerian Lagrangian capability to capture ice-soil-structure interactions accurately.
- Extensive nonlinear material modeling for different types of soil.
- General contact to easily setup all contact interactions.
- Scalable parallel performance on many cores.

## Benefits

- Improve stability of offshore structures



# Subsea Pipelines

## Application

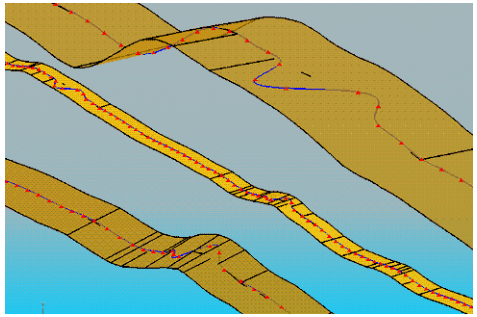
- Assess if the expansion from thermal and internal pressure loads will overload the PLETs or jumpers and/or will cause pipeline to buckle.

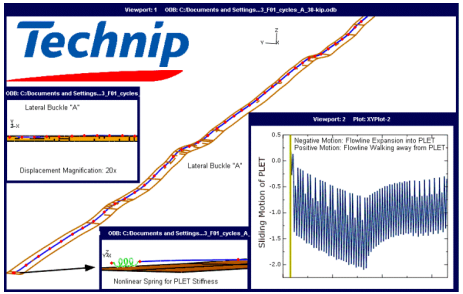
## Why Abaqus Unified FEA?

- Include complex 3D geometry of seabed profiles in nonlinear contact analysis
- Nonlinear pipe-soil interaction behavior with friction models, including option of user defined friction models
- Element technology including pipes and connector elements
- Option of static analysis with stabilization or dynamic analysis to capture buckling behavior
- Thermal effects

## Benefits

- Improved pipeline integrity taking into account shutdown/startup cycles
- Plan for buckle control measures to control fatigue damage






Courtesy: Technip

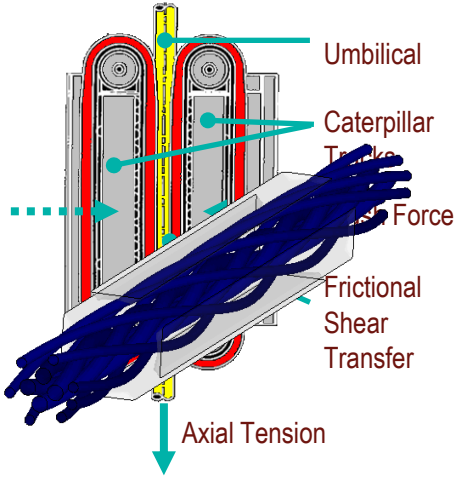
# Umbilicals

## Application

- Assess effect of deepwater umbilical installation process on umbilical behavior


## Why Abaqus Unified FEA?

- Full 3D modeling capability, including scripting for automating model generation.
- Easy-to-use and yet very sophisticated “general” contact capability
- Combination of Abaqus/Explicit and Abaqus/Standard
- Element library with shells, solids, and rigid elements
- High performance parallel computing



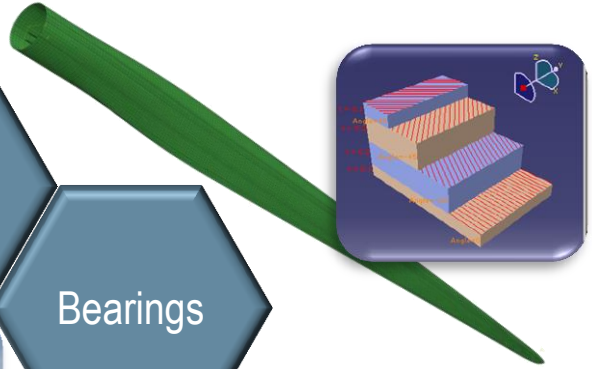

Courtesy: Technip

## Benefits

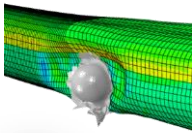
- Improve durability of deepsea umbilicals with greater upfront confidence in designs

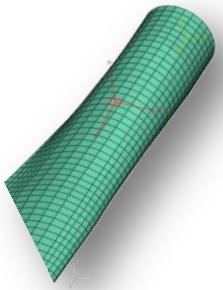
# Wind Turbines

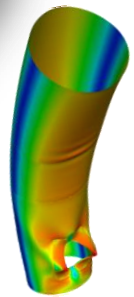
Simulation, Optimization, and Lifecycle Management provide business advantage











Courtesy: DNV Wind Energy

Composite Blades

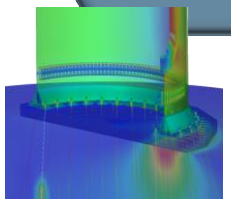
Nacelle

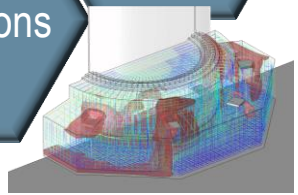
Bearings

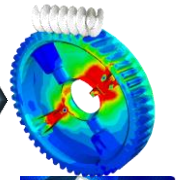
Towers

Gear Trains

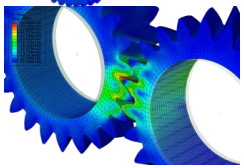
Foundations







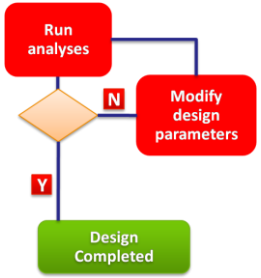
Courtesy: LM Glasfiber



Courtesy: DNV Wind Energy

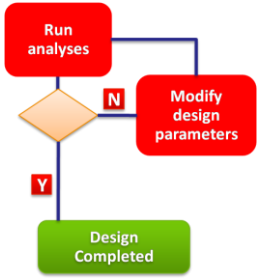
# Blade Structure

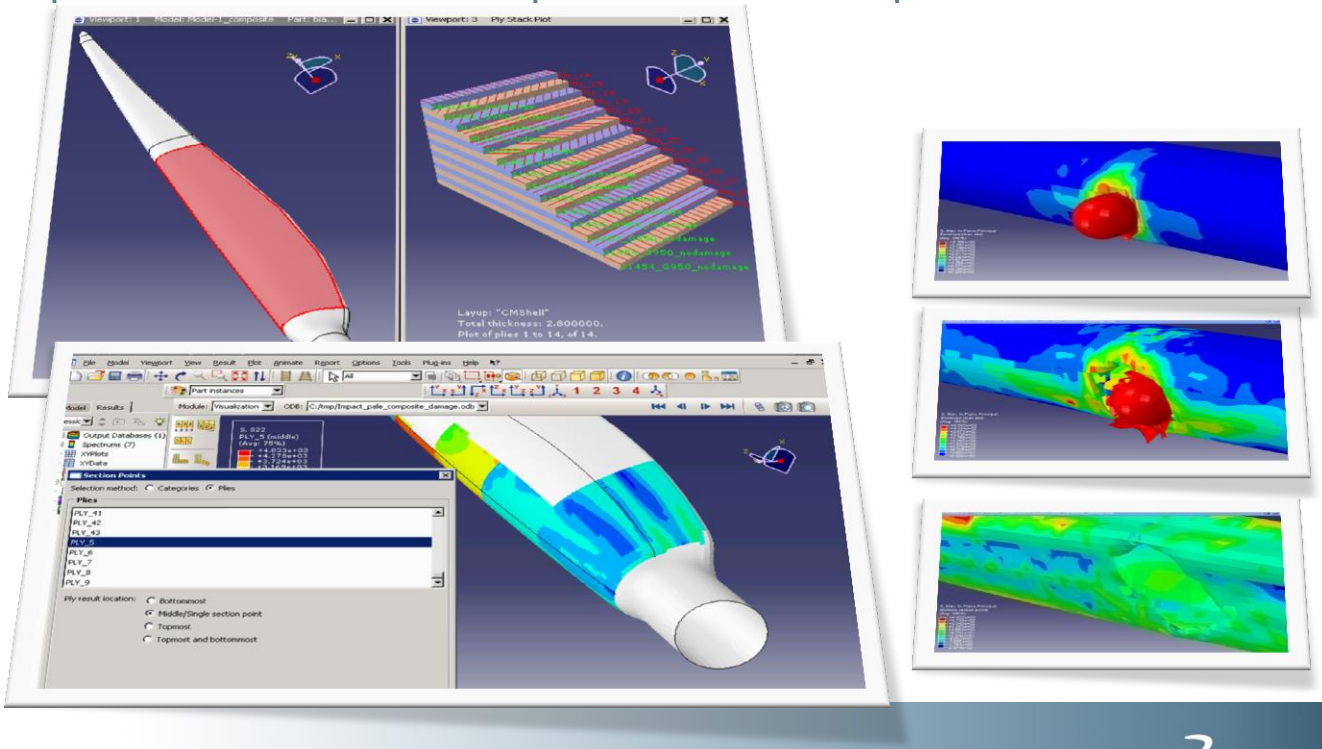
## Comprehensive composites capabilities with Abaqus FEA




# Blade Structure

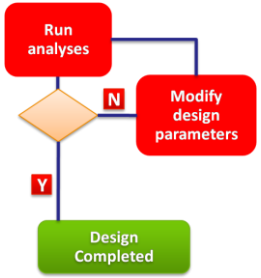
## Comprehensive simulation capabilities with Abaqus FEA

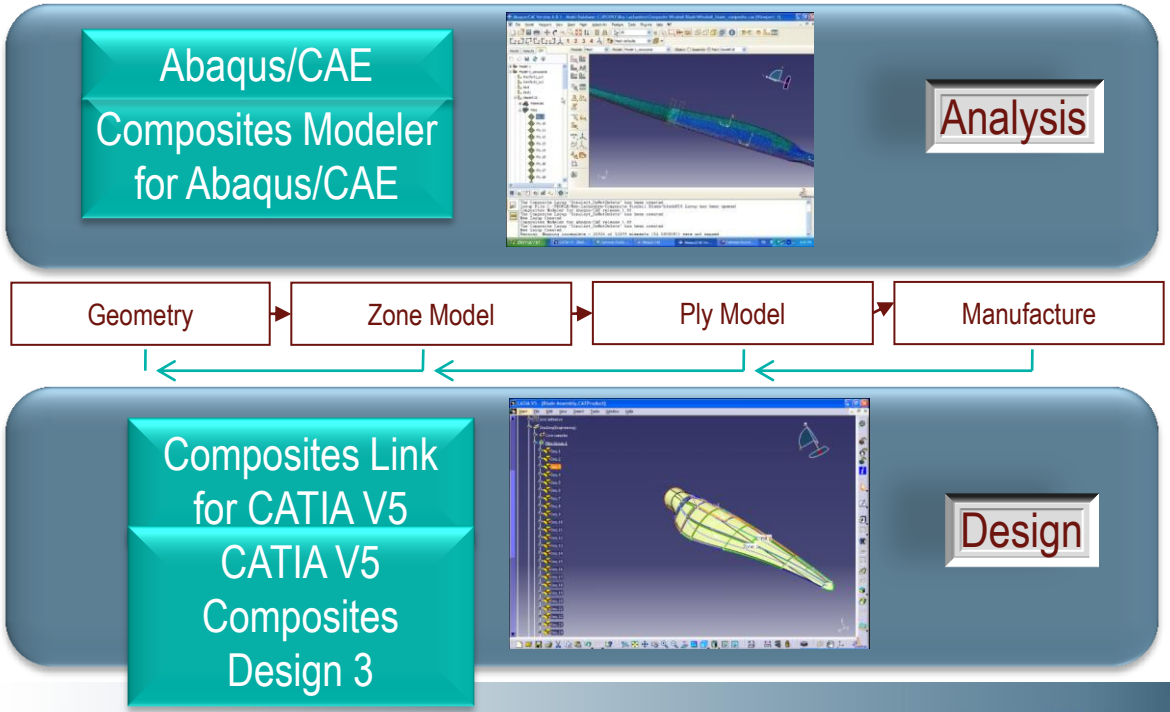


# Blade Structure

## Integration of composites design, analysis, and manufacturing



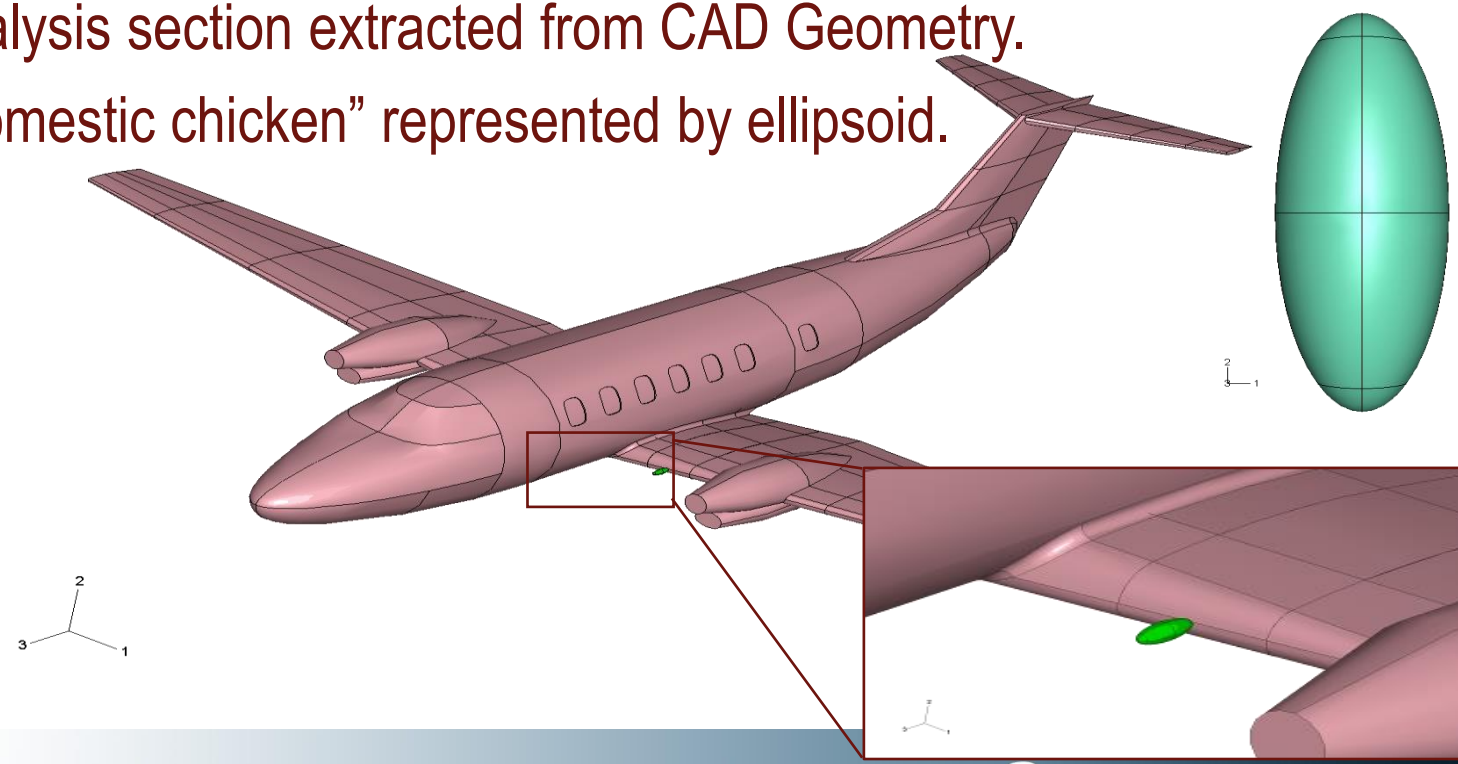
# Bird Strike Motivation

- FAR 25.631
  - Empennage designed for safe flight after bird strike.
  - “Compliance is shown using analysis, tests, or both”.
  - FAR 25.571, 25.575, 33.76 also outline performance after a bird strike.
- International Bird Strike Committee
  - The domestic chicken often used for certification.
  - Artificial birds substituted for real birds becoming more attractive*.
    - Ellipsoid
    - Straight Ended Cylinder
    - Hemispherical ended cylinder

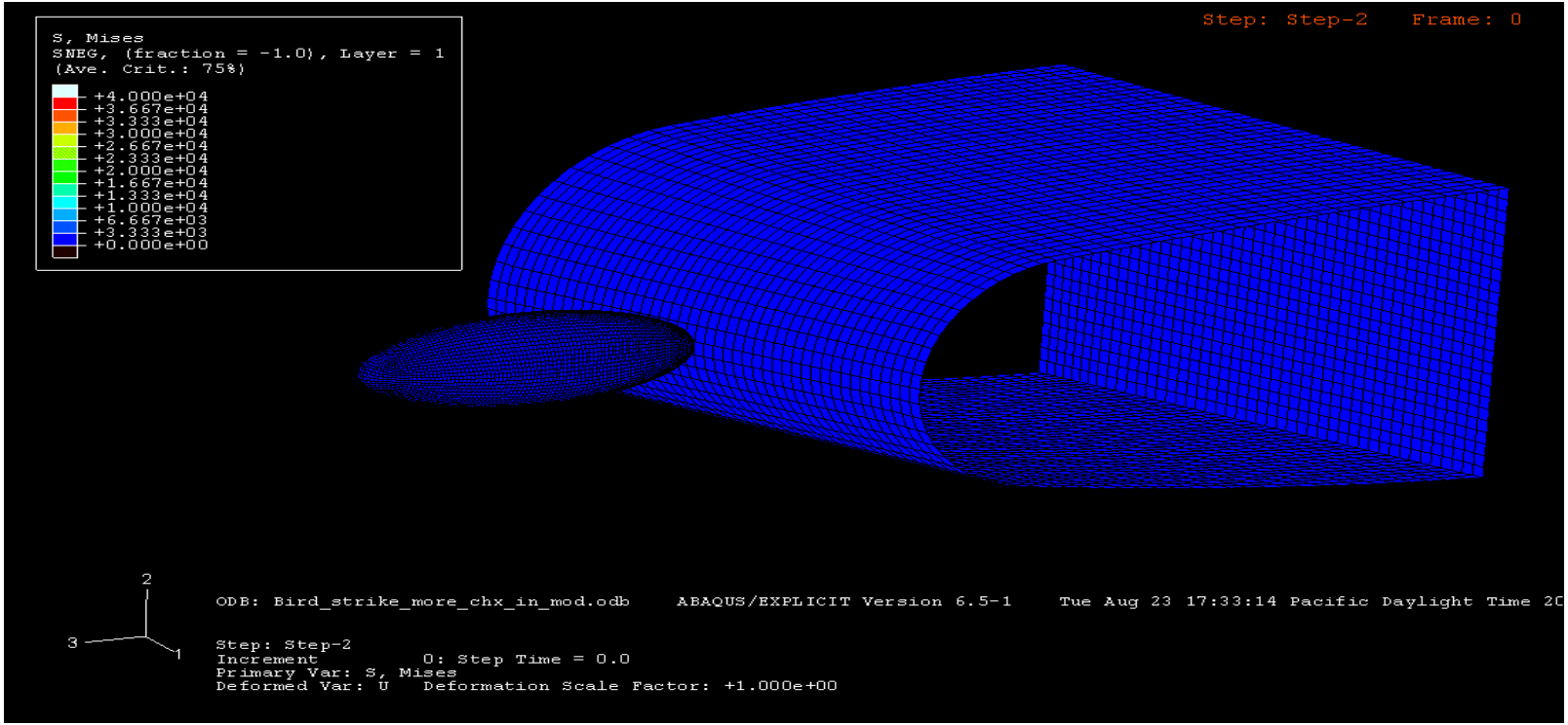

*
IBSC25/WP-IE3, Amsterdam, April 17-21, 2000

# Structural FEM was Created Using Abaqus/CAE

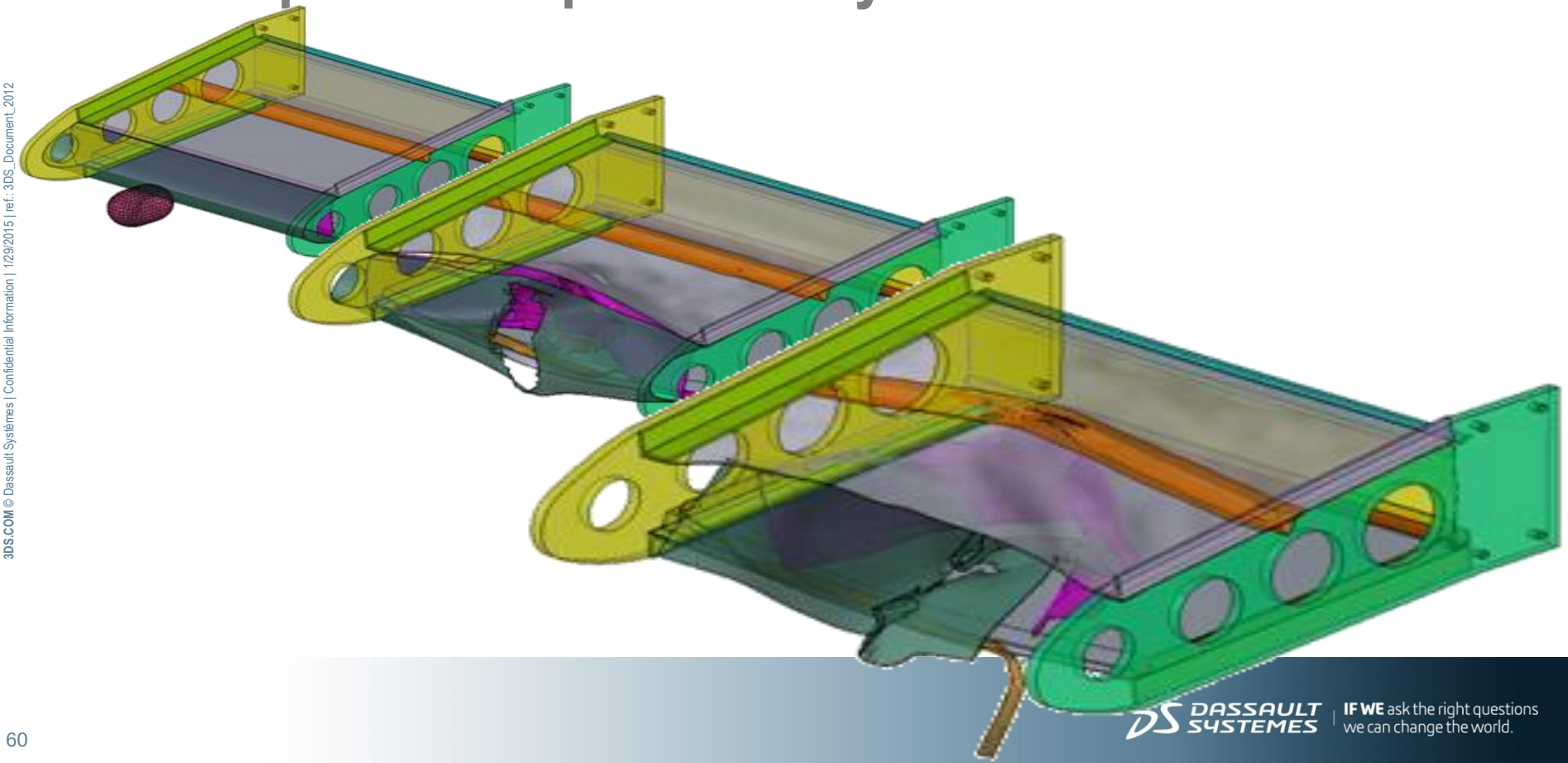
- Analysis section extracted from CAD Geometry.
- "Domestic chicken" represented by ellipsoid.



# Case Study



# Example from Spirit AeroSystems

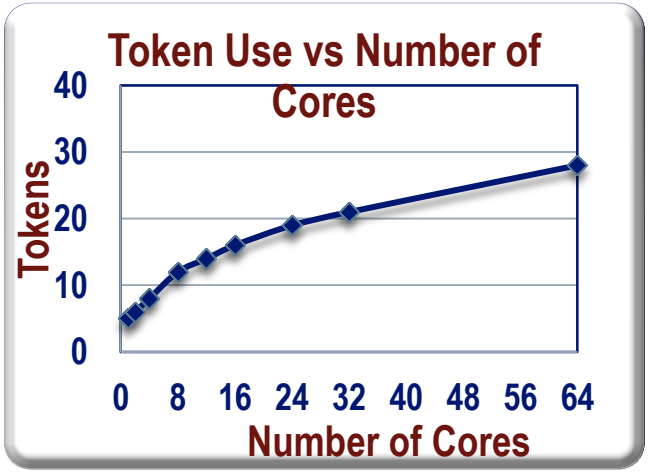


# Licensing Model

# Licensing Model

## Token-based licensing

- What are tokens?
  - Sharable pool of floating network licenses
  - Provides high degree of flexibility
- Token usage
  - Analysis products require 5 tokens to execute on 1 core
  - Additional tokens required for parallel execution
  - Abaqus/CAE uses separate tokens



# Summary

- Abaqus Analysis Products
  - Realistic simulation for all industries
  - Linear & nonlinear behavior
  - Sophisticated contact & material modeling
  - High-performance solvers
- Abaqus/CAE
  - Modern, easy-to-use interface
  - Integration with popular CAD packages
  - Comprehensive support for Abaqus functionality
  - Extensive customization capabilities
- Single solution for all analysis needs

# Thank you!