Multibody Dynamics Simulations with Abaqus from SIMULIA

8.5.2008

Martin Kuessner

Martin.KUESSNER@3ds.com

Abaqus Deutschland GmbH

SIMULIA

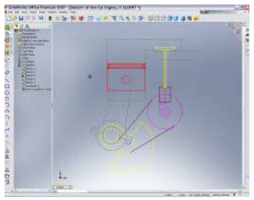
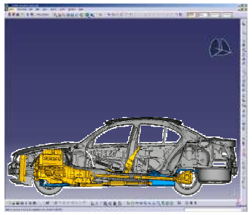
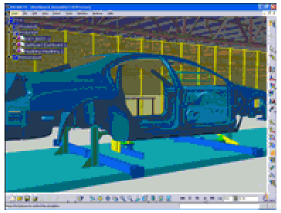
Copyright 2008 Dassault Systèmes

©2001 DASSAULT SYSTEMES-R2

SIMULIA

One Company, First Class Brands

DASSAULT SYSTEMES

3D MCAD	Virtual Product	Virtual Testing	Virtual Production	PLM Collaboration	User Experience
SolidWorks	CATIA	SIMULIA	DELMIA	ENOVIA	3D For ALL
					
Productive, easy to use 3D Design	Product Excellence	Engineering Quality	Production Performance	Collaborative Intelligence	Life Experience

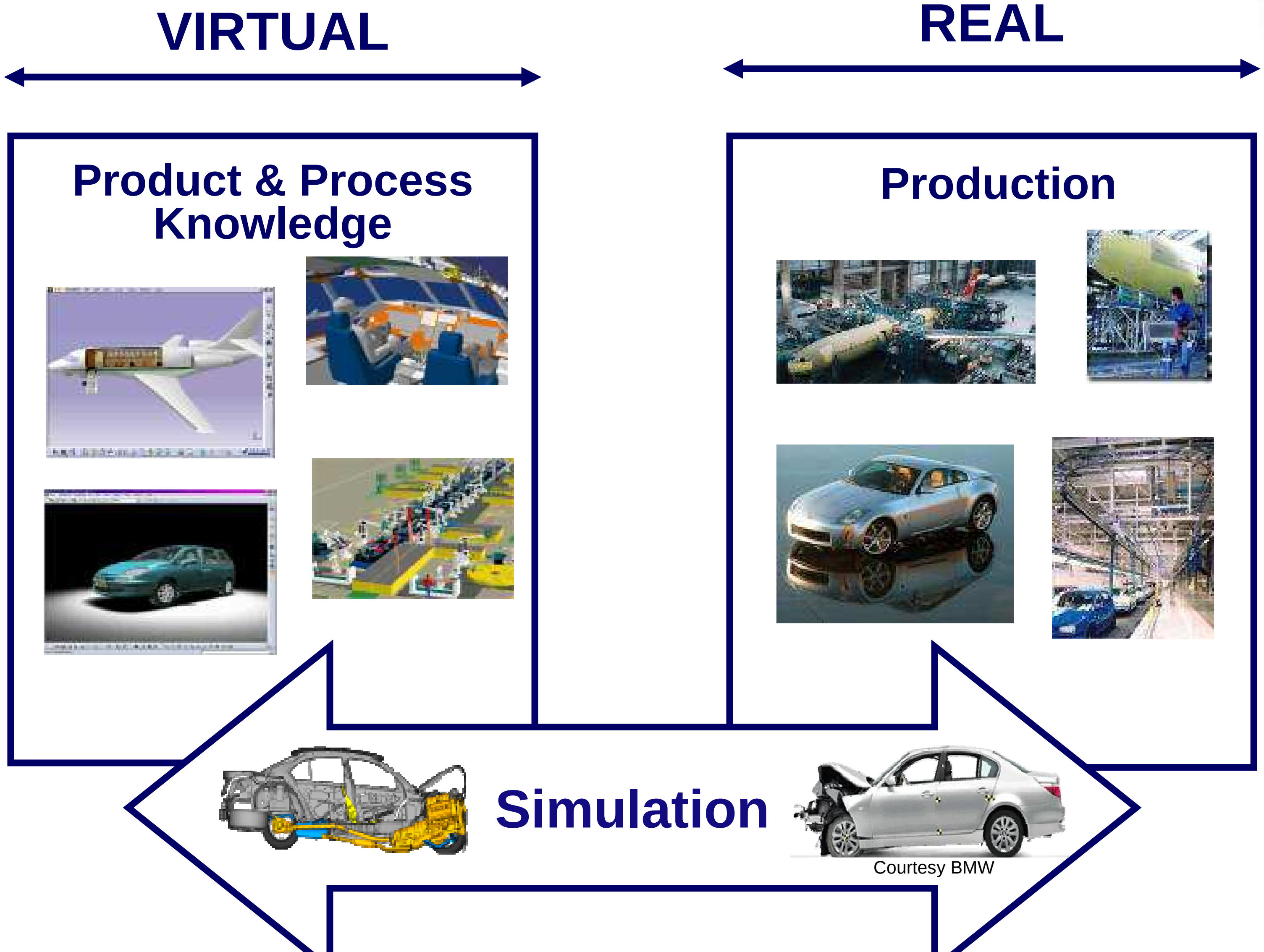
DS Brands: delivering customer benefits through the whole value chain

Copyright 2008 Dassault Systèmes

Realistic Simulation

"Define, Monitor and Control the Physical World"

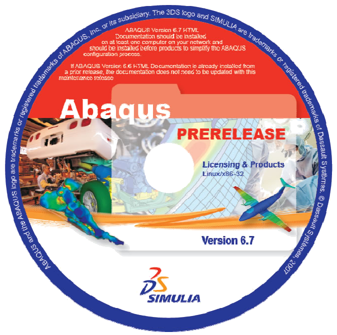
SIMULIA



Copyright 2008 Dassault Systèmes

SIMULIA – our identity

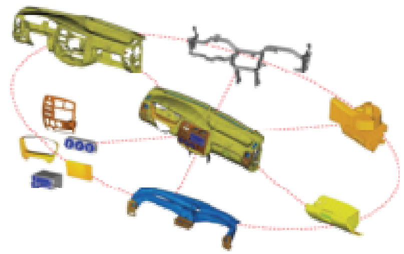
- **What is SIMULIA?**
  - SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including:
    - Abaqus products for Unified Finite Element Analysis
    - Multiphysics solutions - New!
    - Simulation Lifecycle Management solutions - New!
  - SIMULIA is not a company, product, or technology itself but a brand which is responsible for multiple products and technologies in the simulation domain.
  - Abaqus lives on as the name of our Unified FEA products



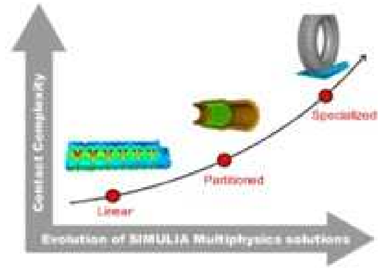
Copyright 2008 Dassault Systèmes

Product Strategy

- **Unified FEA solutions for all users**
  - Scalable Unified FEA products and services with Abaqus quality



- **Multiphysics for the most challenging customer applications**
  - Open Multiphysics platform and complete solutions for key problems

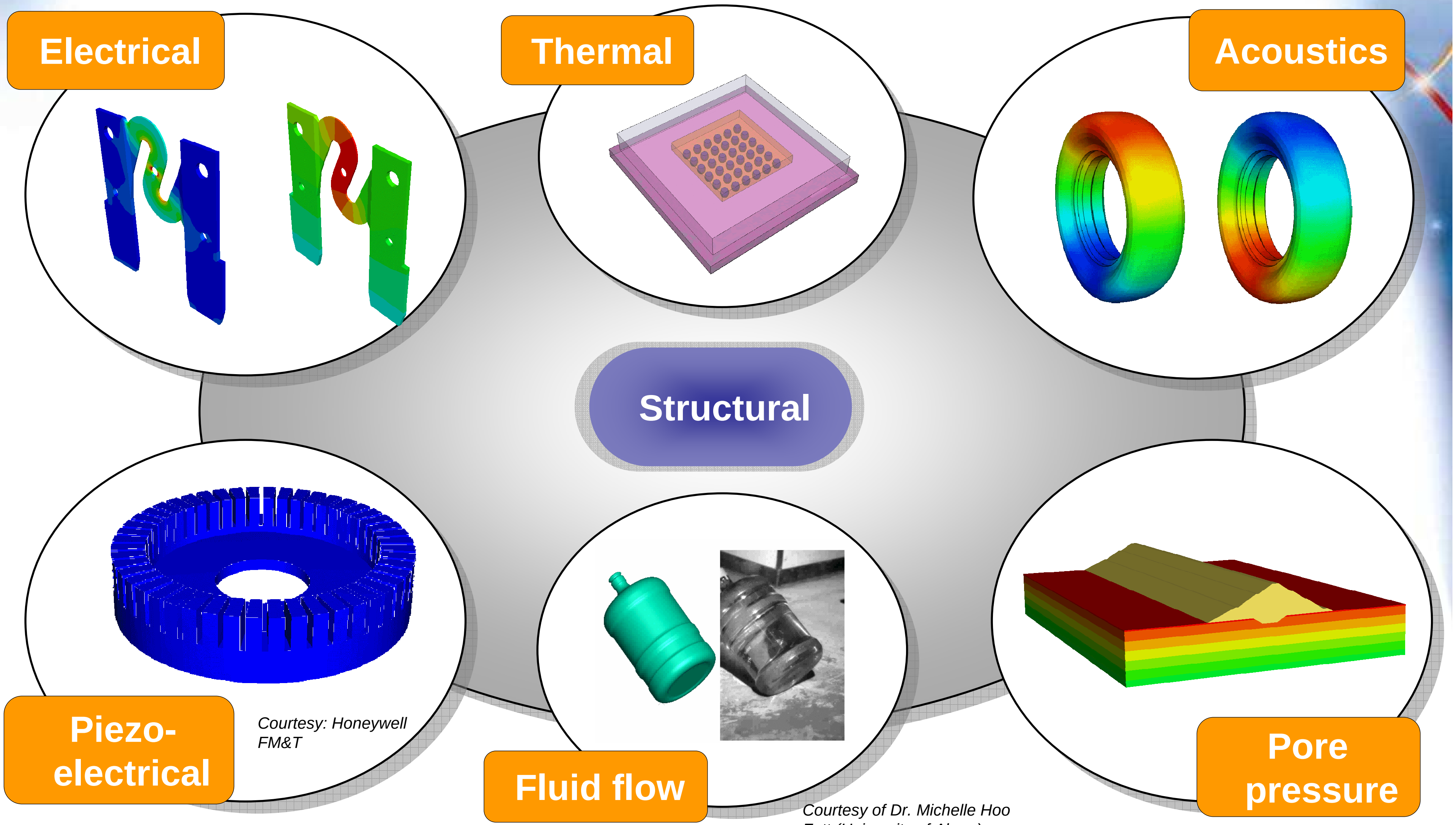


- **Full Simulation Lifecycle Management**
  - Collaborative environment to manage simulation processes, data and IP

Copyright 2008 Dassault Systèmes

Multiphysics in Abaqus Unified FEA

- **Abaqus enables coupling of multiple fields**



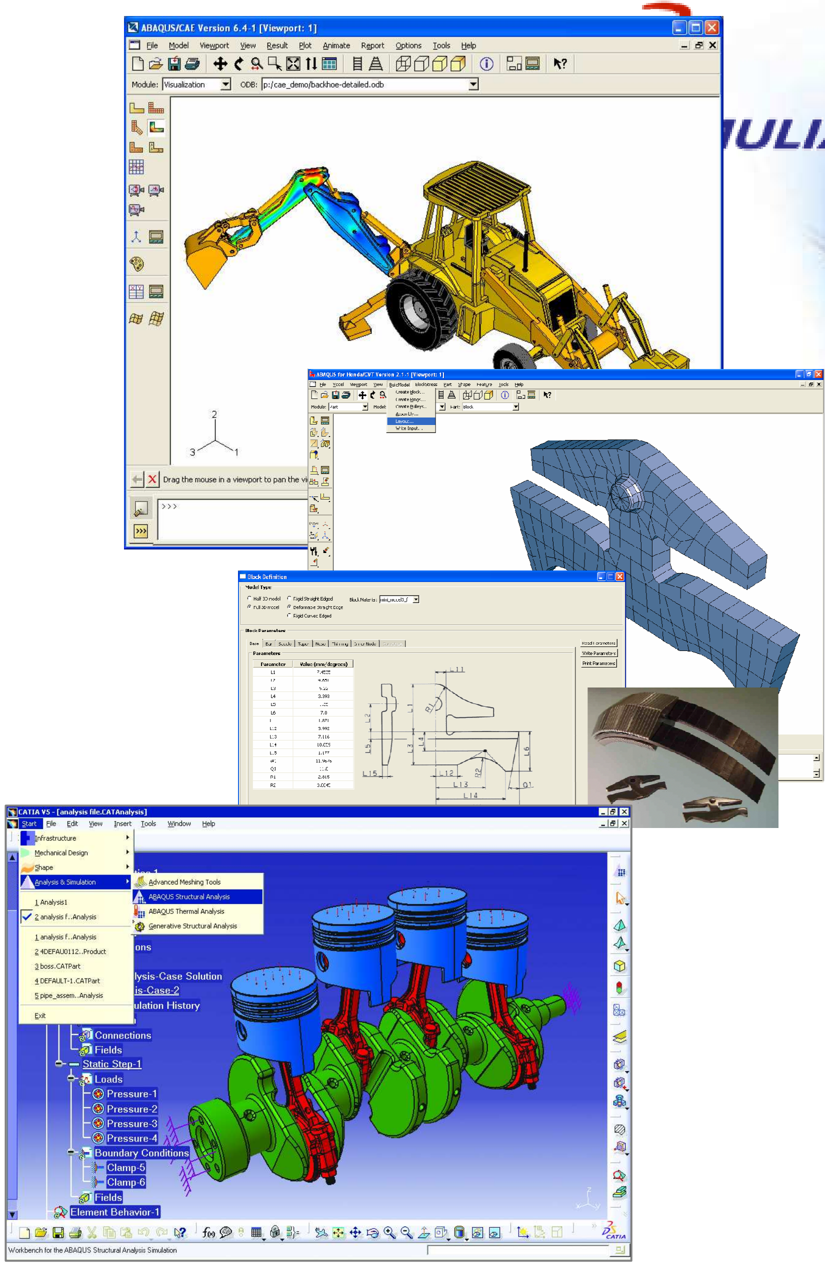
Courtesy of Dr. Michelle Hoo Fatt (University of Akron)

Copyright 2008 Dassault Systèmes

Abaqus FEA Products

- **Interactive Products (modeling and visualization)**
  - Abaqus/CAE
    - Process Automation Toolkit
  - Abaqus for CATIA V5

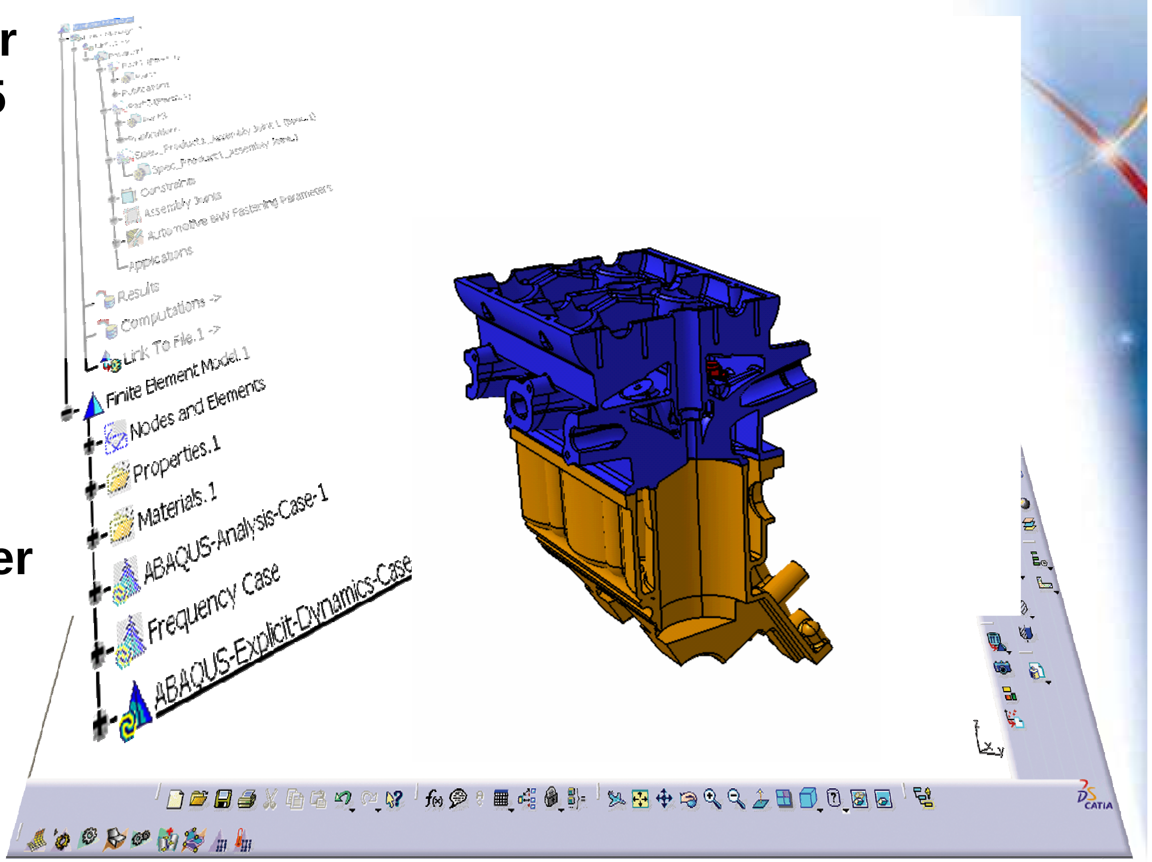
- **Analysis Products (finite element solution)**
  - Abaqus/Standard
  - Abaqus/Explicit



Copyright 2008 Dassault Systèmes

What is Abaqus for CATIA V5?

- **End-to-end Abaqus modeler fully integrated in CATIA V5**
  - Preprocessing
  - Job submission and monitoring
  - Postprocessing



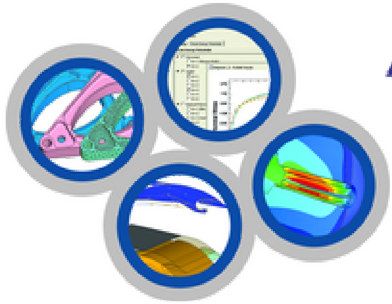
- **Take advantage of the power of Abaqus and CATIA V5**
  - CATIA V5:

    Intuitive modeling of complex assemblies
  - Abaqus:

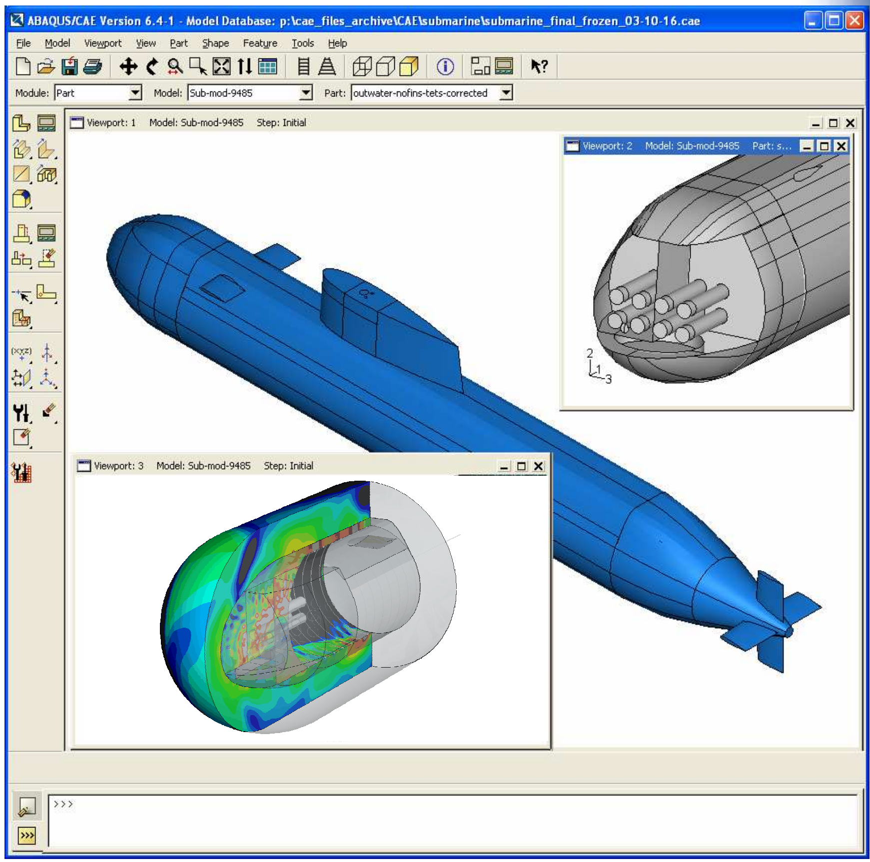
    Powerful analysis tools for linear and nonlinear FEA

Copyright 2008 Dassault Systèmes

Abaqus/CAE



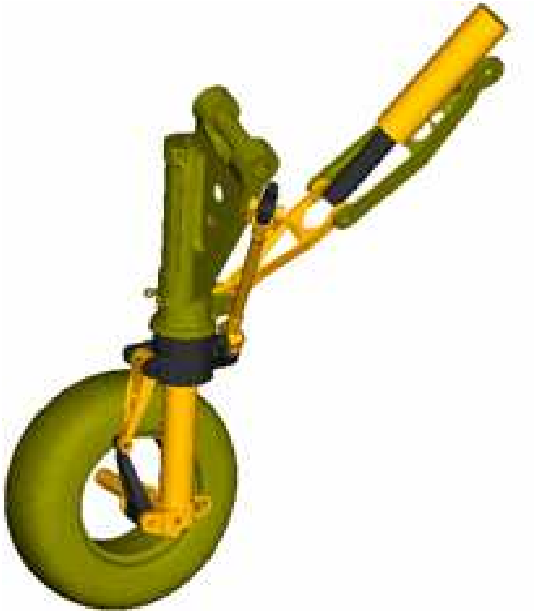
- **Interactive, parametric, finite element modeler and results viewer**
- **Hybrid (geometry & mesh based) FEA environment for maximum flexibility**
- **The most complete and fastest graphical user interface for the dedicated Abaqus user**
- **Direct interface to CATIA V4, Parasolid, IDEAS**
- **Associative import from Pro/ENGINEER, CATIA V5 and SolidWorks**



Copyright 2008 Dassault Systèmes

Abaqus/Standard

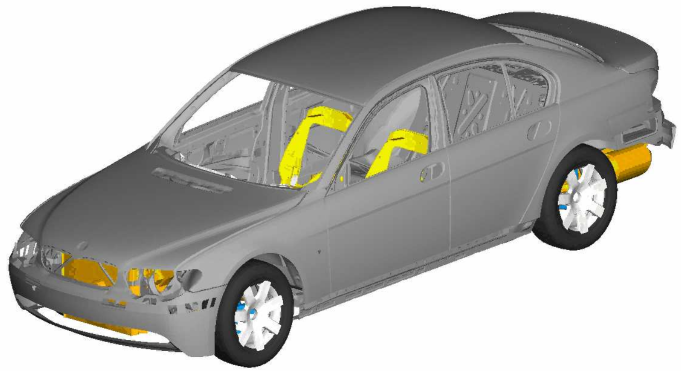
- Comprehensive linear & nonlinear implicit general purpose finite element analysis of structural, thermal, acoustic & mechanism simulations
- Integration with Abaqus/Explicit provides maximum flexibility for multi-physics simulation ("Unified FEA")
- Sophisticated contact, failure, material modeling & other advanced nonlinear capabilities
- High-performance direct and iterative solvers with support for shared and distributed memory configurations
- Powerful interfaces for user customization



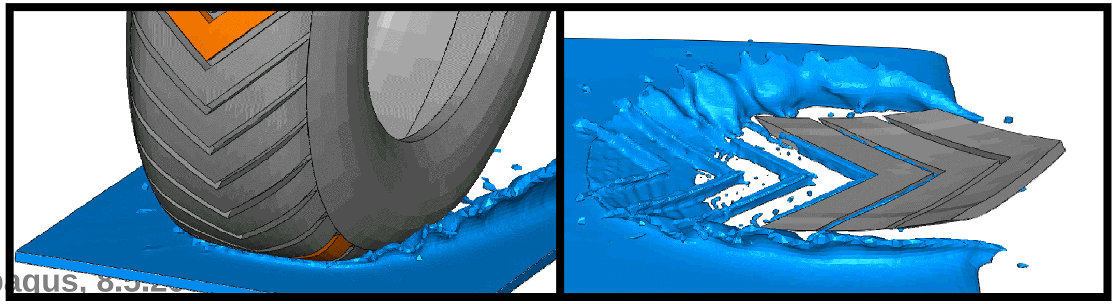
Copyright 2008 Dassault Systèmes

Abaqus/Explicit

- **Comprehensive explicit finite element analysis of structural, thermal, acoustic & mechanism simulations**
- **Integration with Abaqus/Standard provides maximum flexibility for multi-physics simulation ("Unified FEA")**
- **High-performance solver with support for shared and distributed memory configurations**
- **Powerful interfaces for user customization**



Courtesy BMW



Copyright 2008 Dassault Systèmes

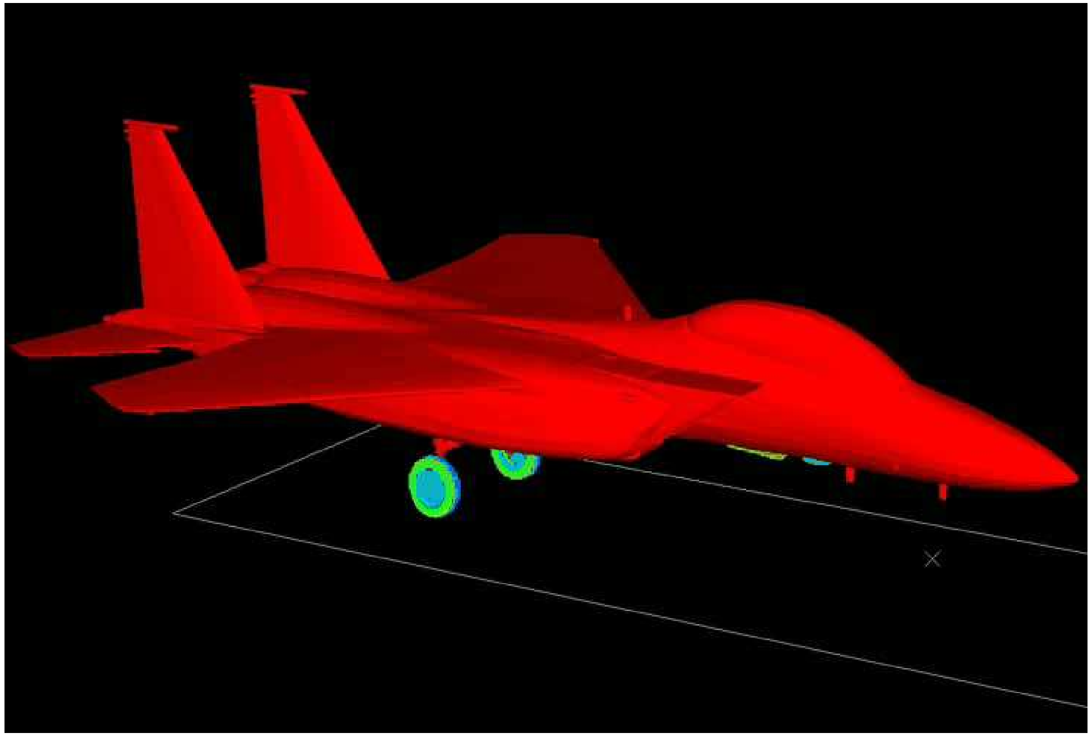
Finite element based multibody dynamics

©2001 DASSAULT SYSTEMES-R2

Copyright 2008 Dassault Systèmes

Mechanism analysis

- Mechanism analysis in Abaqus covers the broad class of analyses that include moving parts



Copyright 2008 Dassault Systèmes

Connectors in Mechanism Analysis

- **Connectors are a fundamental ingredient in mechanism analyses**
  - Enforce kinematic constraints
  - Allow joint flexibility
  - Model sophisticated interactions: joint stops (contact), locking mechanisms, driven motion, internal force/moment controls, friction, etc.

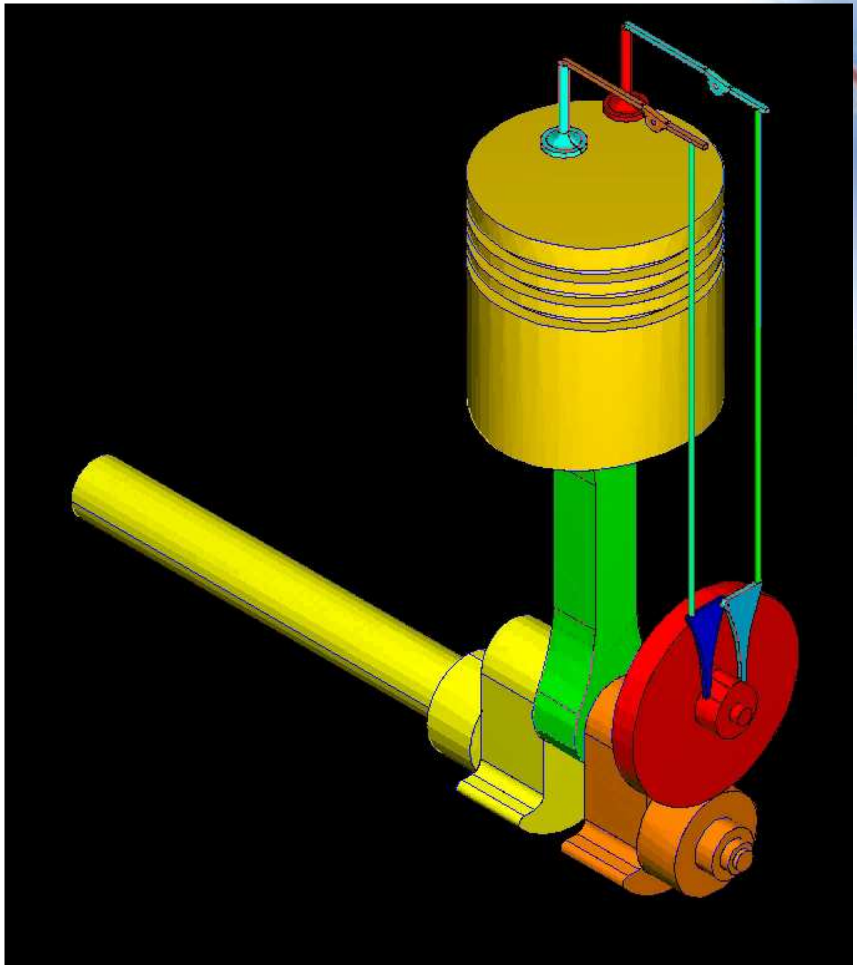
Copyright 2008 Dassault Systèmes

Mechanism Overview

- **Mechanism analysis comes in three categories:**

1 Rigid components connected by joints:

  - Kinematic analysis focusing on constraint forces
  - Multi-body dynamic analysis focusing on inertia forces

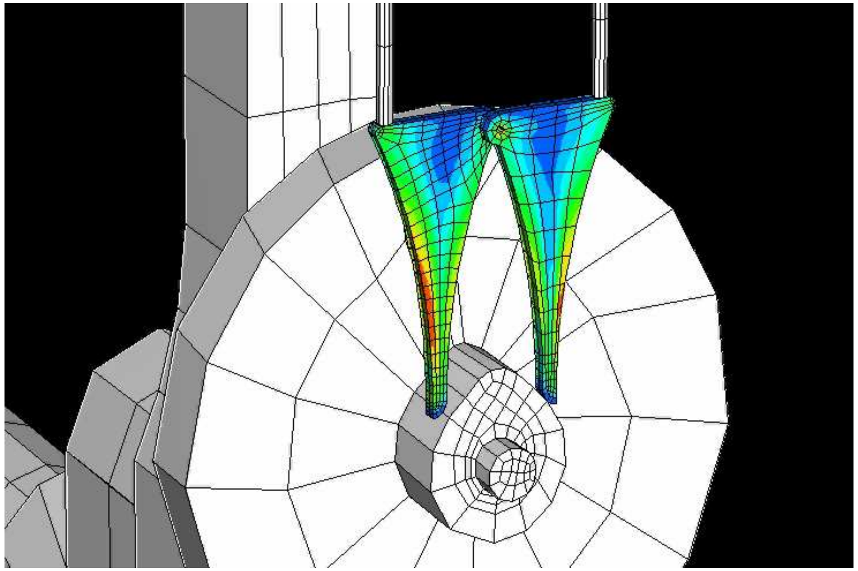


Copyright 2008 Dassault Systèmes

Mechanism Overview

2 Large (nonlinear) deformation of bodies connected by joints:

- Component-level stress analysis



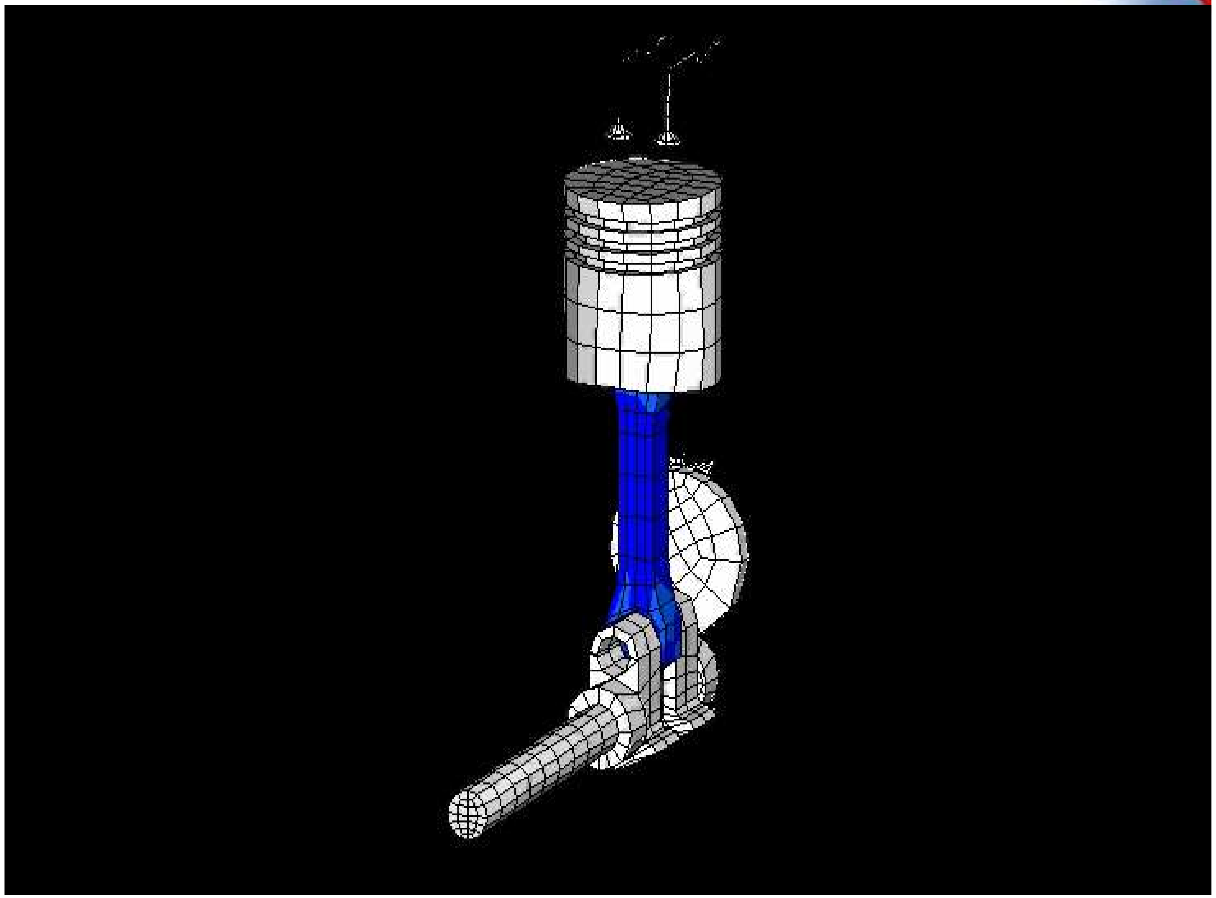
Copyright 2008 Dassault Systèmes

Mechanism Overview

3

Small deformation bodies in geometrically nonlinear analysis connected by joints:

- Linear elastic flexibility in kinematic or multi-body dynamic analysis combined with rigid components or large deformation components



Copyright 2008 Dassault Systèmes

©2001 DASSAULT SYSTEMES-R2

Connectors

SIMULIA

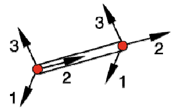
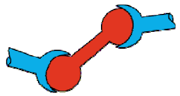
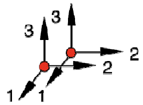
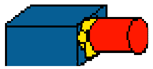
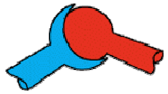
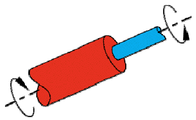
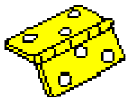
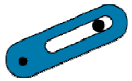
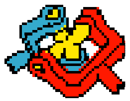
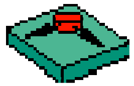
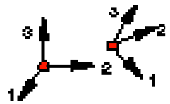
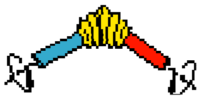
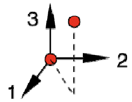
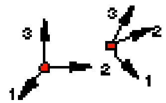
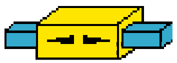
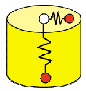
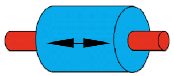
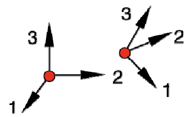
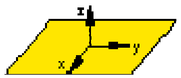
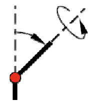
Copyright 2008 Dassault Systèmes

Connectors

- **Connectors model discrete, physical connections between bodies**
- **Features include:**
  - Kinematic constraints
  - Oriented spring, dashpot, and frictional behavior
  - Joint stops
  - Constraint forces and moment output
  - Joint failure
  - No degree of freedom elimination (connectors can be used between rigid bodies)
  - Relative motion actuation
  - Internal load actuation

Copyright 2008 Dassault Systèmes

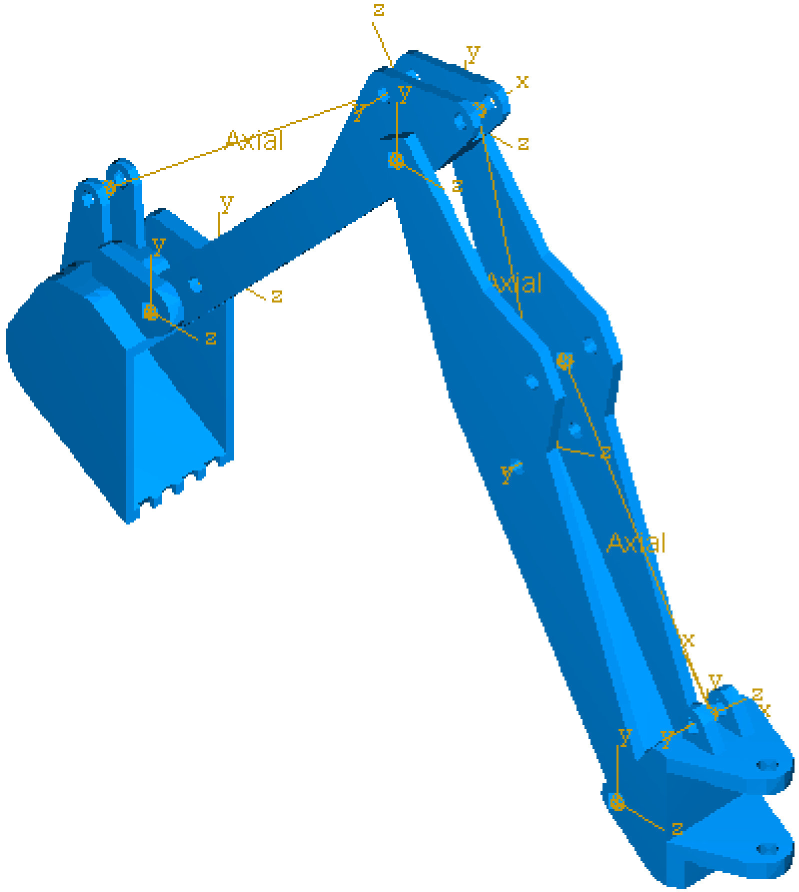
Connection Library

Assembled		Basic translational		Basic rotational	
BEAM		LINK		ALIGN	
WELD		JOIN		REVOLUTE	
HINGE		SLOT		UNIVERSAL	
UJOINT		SLIDE-PLANE		CARDAN	
CVJOINT		CARTESIAN		EULER	
TRANSLATOR		RADIAL-THRUST		CONSTANT VELOCITY	
CYLINDRICAL		AXIAL		ROTATION	
PLANAR				FLEXION-TORSION	

Copyright 2008 Dassault Systèmes

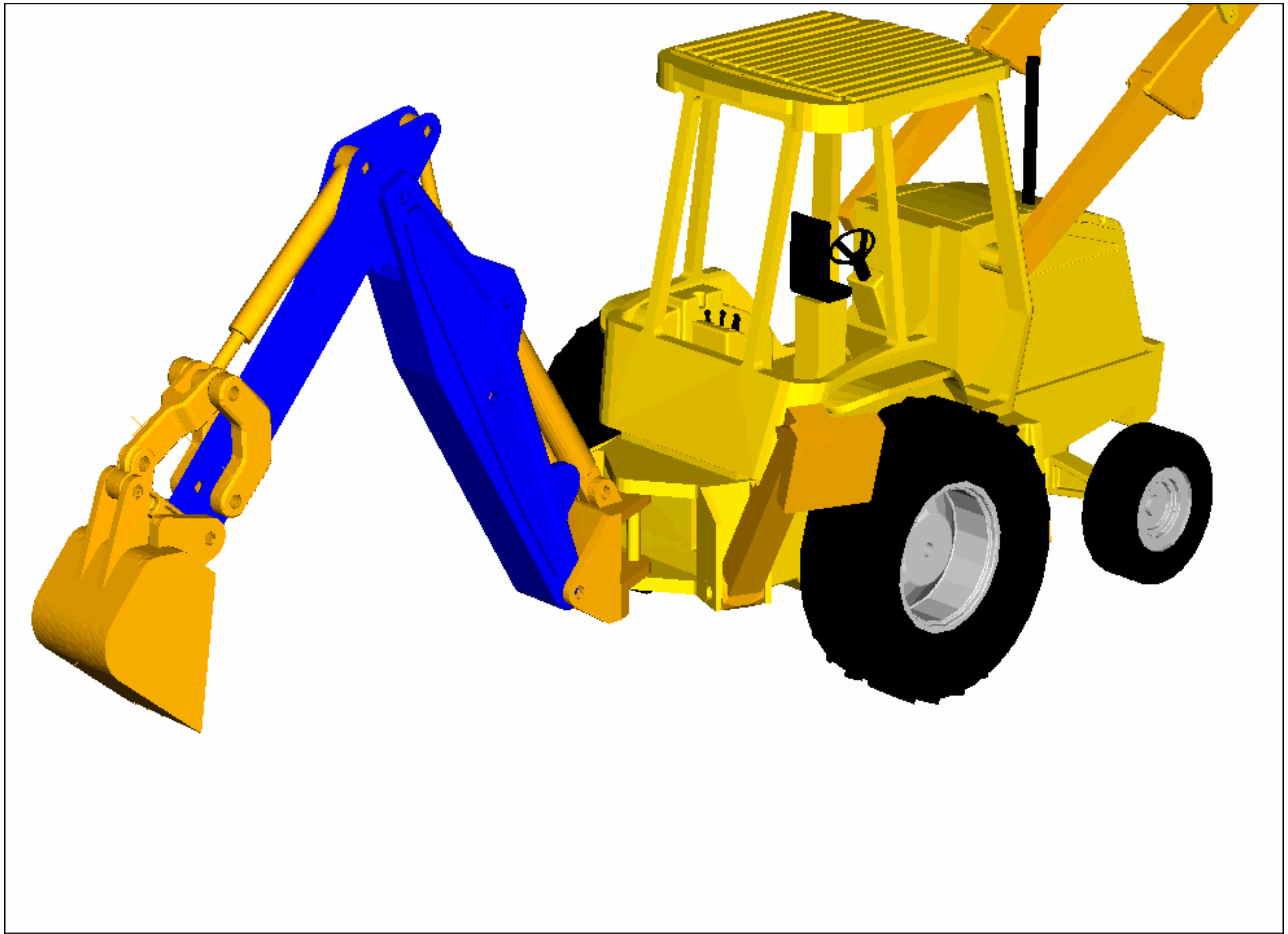
Connector Support in Abaqus/CAE and Abaqus/Viewer

- **Full support of analysis functionality**
  - Definition of all connection types, behavior options, actuations, and output variables supported in Abaqus/CAE
    - Display of the connector orientation systems, important for debugging connection definitions
  - Full display and animation of results in Abaqus/Viewer



Copyright 2008 Dassault Systèmes

Connector Support in Abaqus/CAE and Abaqus/Viewer



backhoe-3.avi

Copyright 2008 Dassault Systèmes

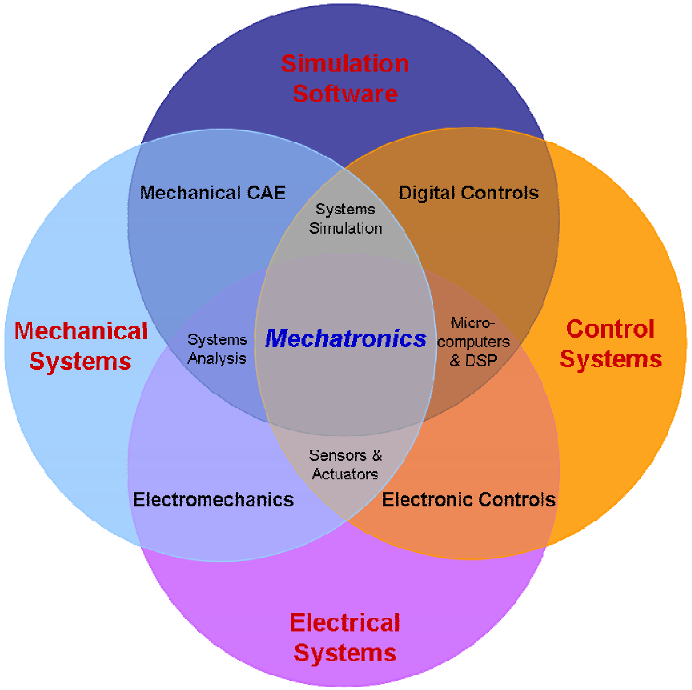
Logical-Physical co-simulation

©2001 DASSAULT SYSTEMES-R2

SIMULIA

Copyright 2008 Dassault Systèmes

Mechatronics



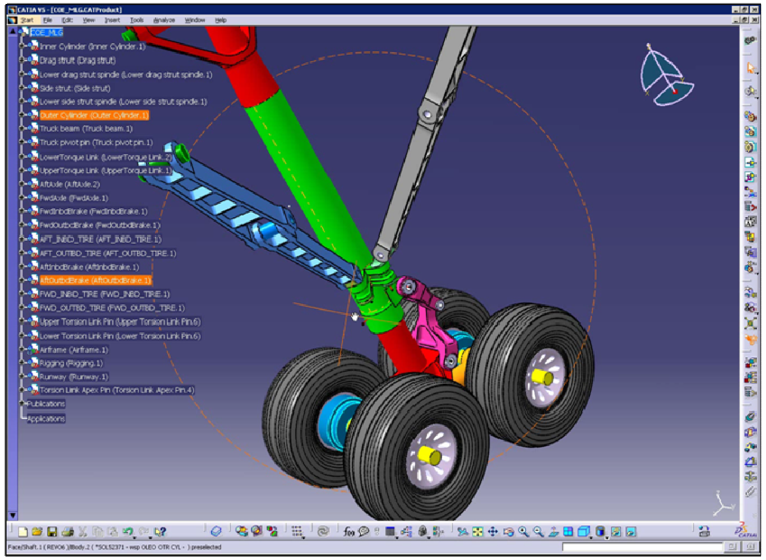
In many cases:

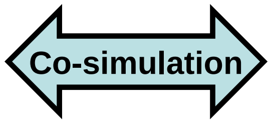
- ***Sensors:*** **collect data regarding the state of the mechanical system**
- ***Control modules:*** **using the sensor information compute the necessary actuation to advance the mechanical system in the desired state**
- ***Actuators:*** **apply the (mechanical) loading (e.g., electric motors, hydraulic pistons) computed by the control modules.**

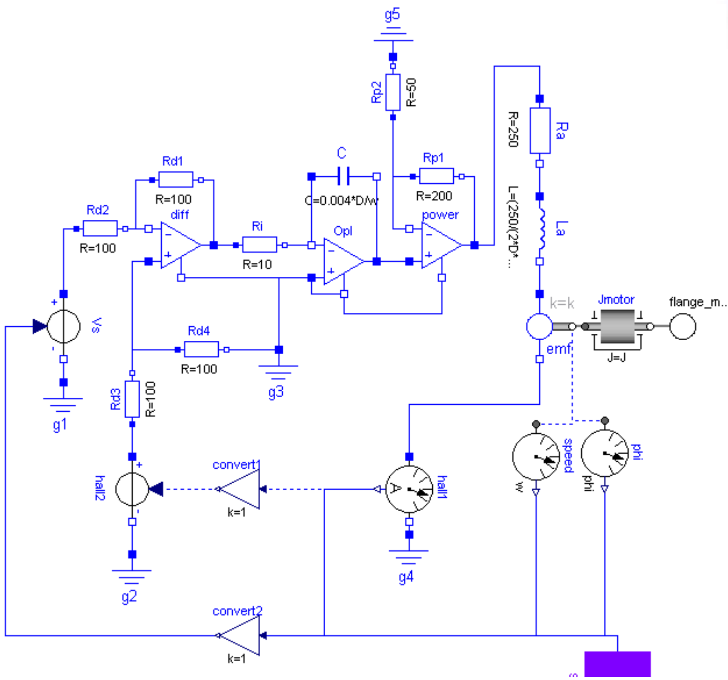
Copyright 2008 Dassault Systèmes

Logical-Physical co-simulation

- **Goal: provide a comprehensive, easy to use capability of modeling system level behavior**
- **How: co-simulation between Abaqus and Dymola, both DS owned products.**
  - Abaqus: models the physics in the system such as mechanical, thermal, acoustics, etc. Provides sensors and actuators.
  - Dymola: models electronics, hydraulics







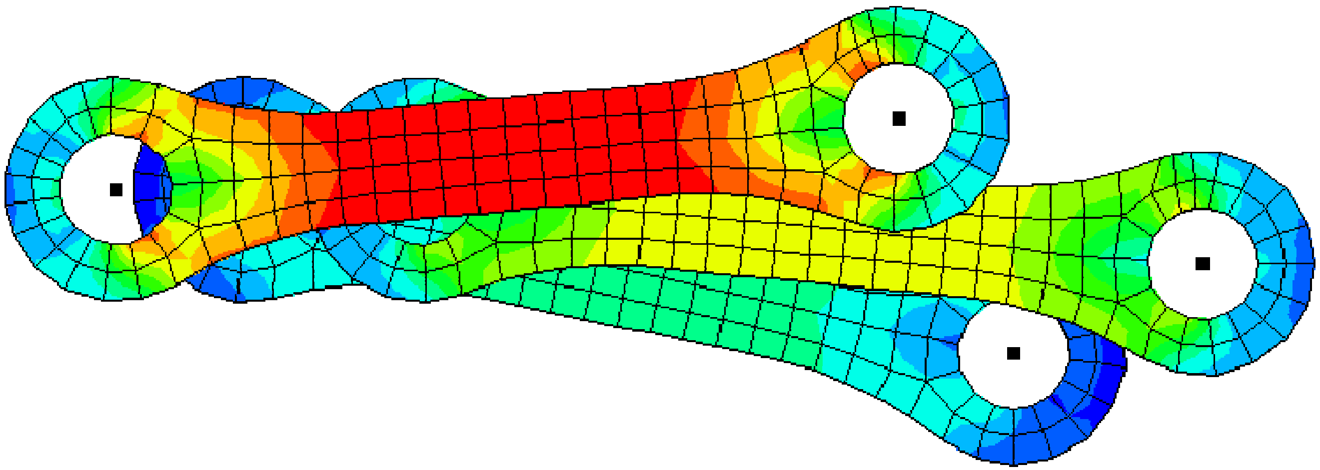
Copyright 2008 Dassault Systèmes

Substructures

Copyright 2008 Dassault Systèmes

Rotating Substructures

- **Geometrically nonlinear analysis:**
  - Substructures are valid for large rotations and large translations
  - Stress recovery occurs in the current configuration
- **Strains are assumed small**

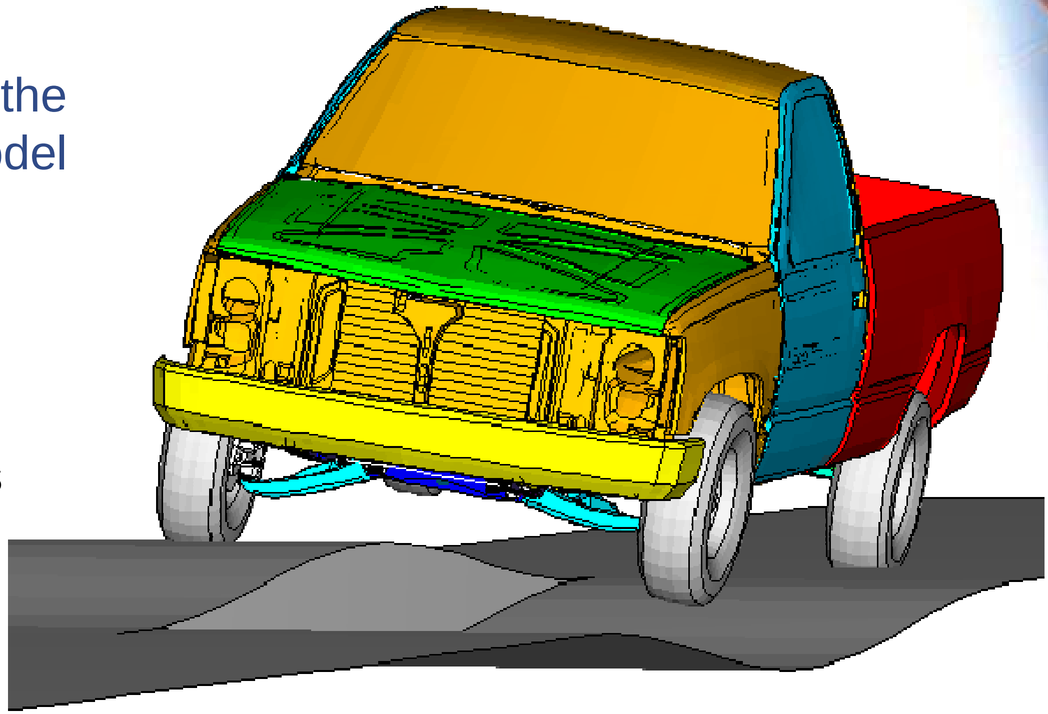


Tie rod stress during crank rotation

Copyright 2008 Dassault Systèmes

Substructure Model of a Pickup Truck

- **1994 Chevrolet C1500**
  - Example problem:
    - 55,000 elements in the fully-deformable model
    - 20 substructures
    - 2 rigid bodies
    - 81 connectors
  - Analysis speed up by as much as **120** times



The FEM model is obtained from the Public Finite Element Model Archive of the National Crash Analysis Center at George Washington University

(**`http://www.ncac.gwu.edu/archives/model/index.html`**)

Copyright 2008 Dassault Systèmes

1994 Chevrolet C1500

- **In-phase bump**

Copyright 2008 Dassault Systèmes

Summary

©2001 DASSAULT SYSTEMES-R2

SIMULIA

Copyright 2008 Dassault Systèmes

Summary

- Finite Element based multibody dynamics
- Implicit and explicit time integration
- Allows for all kinds of nonlinearities
- Coupling with Controls for extended application

Copyright 2008 Dassault Systèmes

Multibody Dynamics Simulations with Abaqus from SIMULIA

Thank you for your attention

8.5.2008

Martin Kuessner

Martin.KUESSNER@3ds.com

Abaqus Deutschland GmbH

SIMULIA

©2001 DASSAULT SYSTEMES-R2

Copyright 2008 Dassault Systèmes