3DEXPERIENCE®

VIRTUAL PREDICTION AND VALIDATION OF STRUCTURAL COMPONENT BEHAVIOR UNDER REALISTIC OPERATING CONDITIONS

Jan Granlund
SIMULIA EuroNorth Industry Process Consultant Expert
Jan.Granlund@3ds.com

DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

TABLE OF CONTENTS

TABLE OF CONTENTS

INTRODUCTION

Our company

A purpose-driven company

Combining Art, Science & Technology
for a more sustainable world

20,000 passionate people

From 133 countries
188 sites
One global R&D / 69 labs

Long-term driven

Majority shareholder control
Revenue: €4,5 billions*
Operating margin: 30,2%*

*Figures as of FY 2020 / Non-IFRS

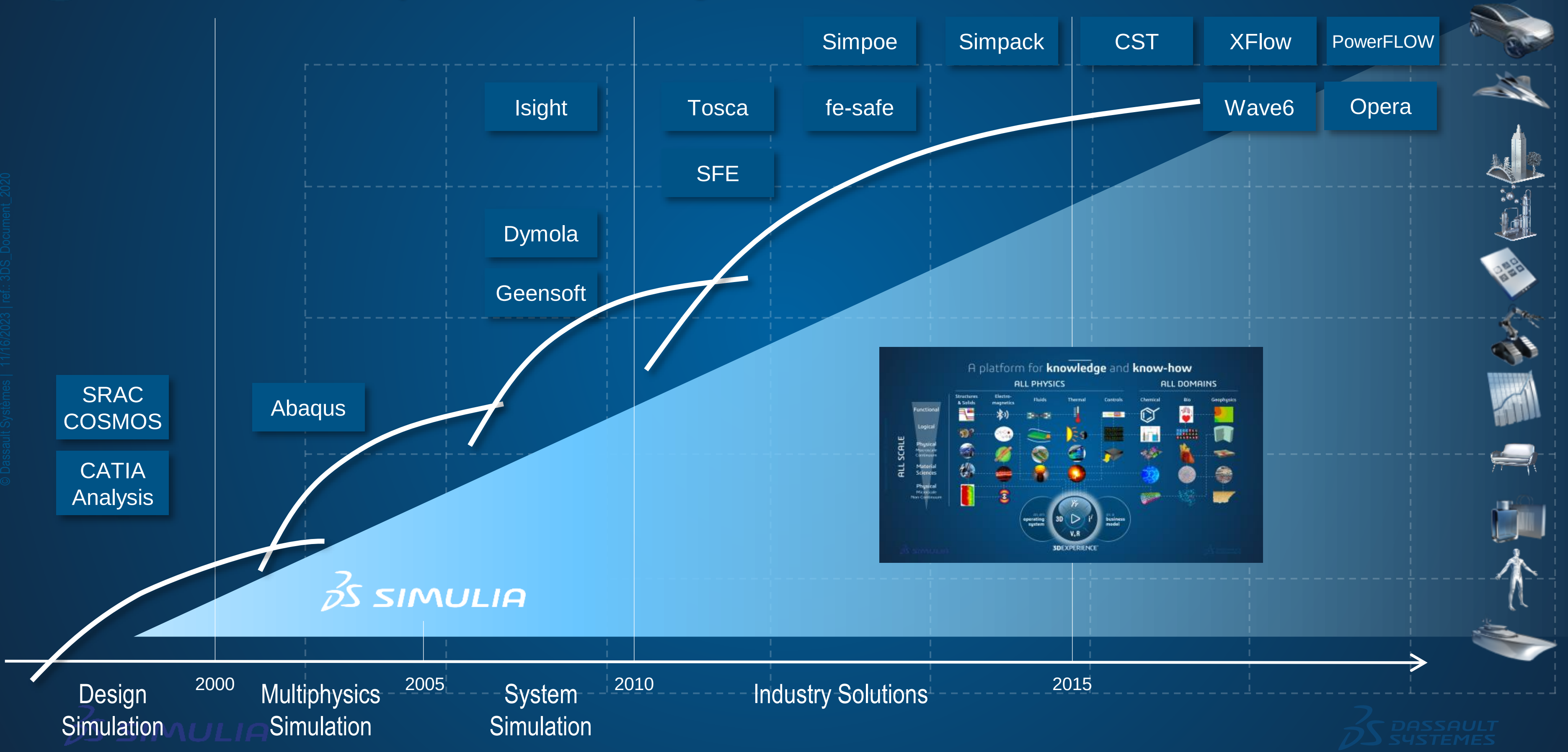
12,260 partners

Software, Technology & Architecture
Content & Online services
Sales
Consulting & System Integrators (C&SI)
Education
Research

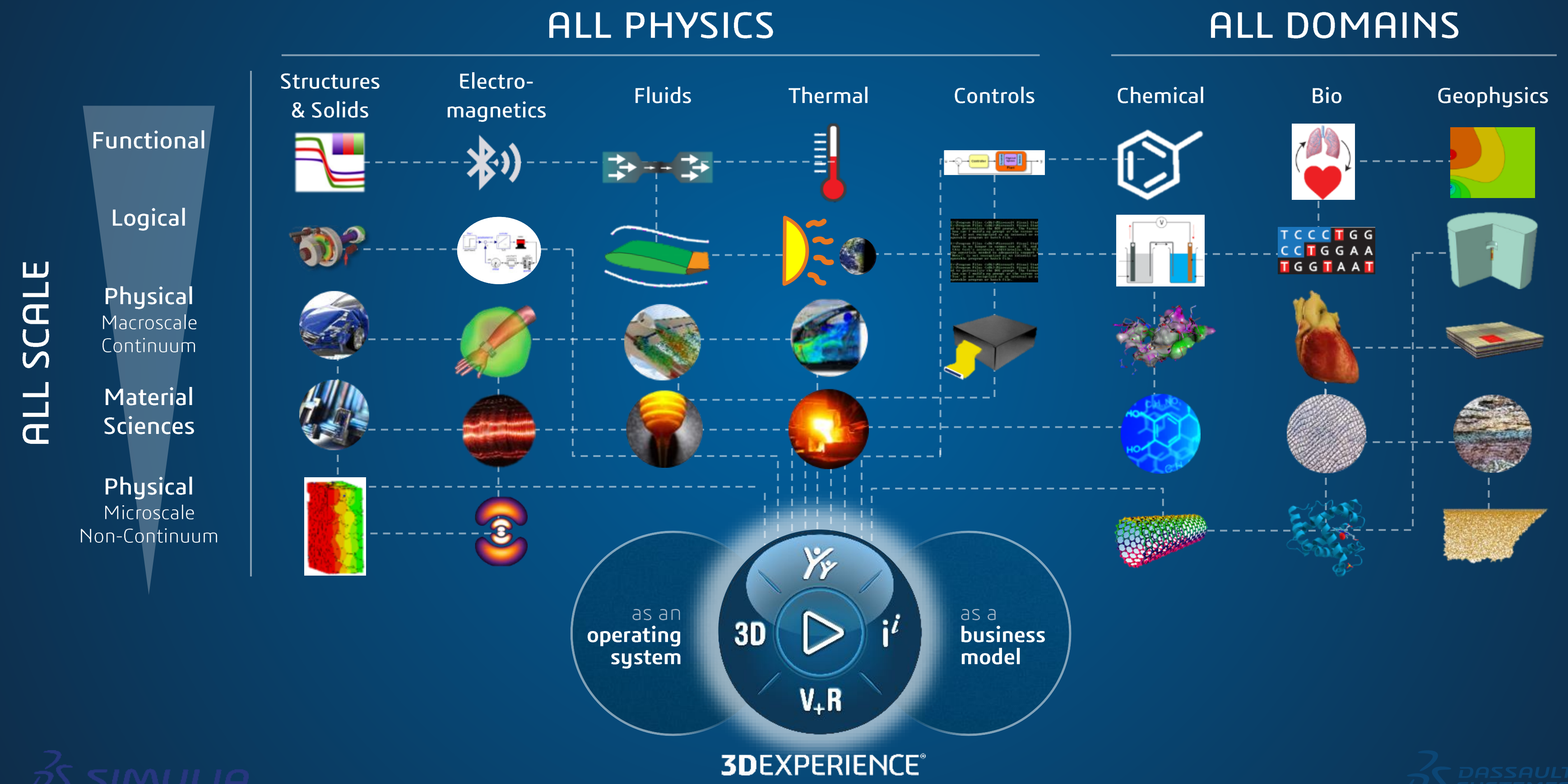
290,000 customers

11 industries in 140 countries
26 million users
Game-changing
3DEXPERIENCE platform

Dassault Systèmes Long-term Commitment to Simulation



A platform for **knowledge** and **know-how**

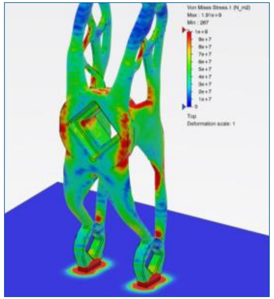


SIMULIA

DASSAULT SYSTEMES

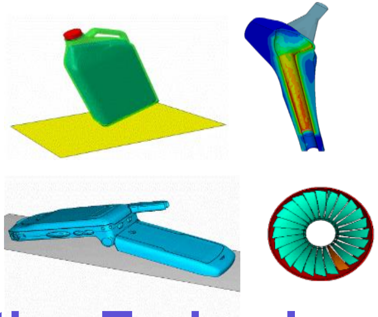
TABLE OF CONTENTS

Structures Simulation Technology



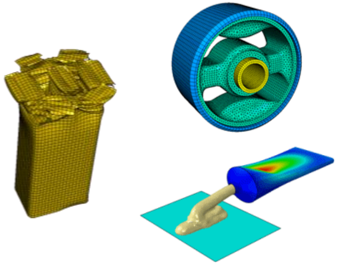
Realistic simulation with FEA & Multiphysics

Complete solutions for a vast spectrum of industrial applications



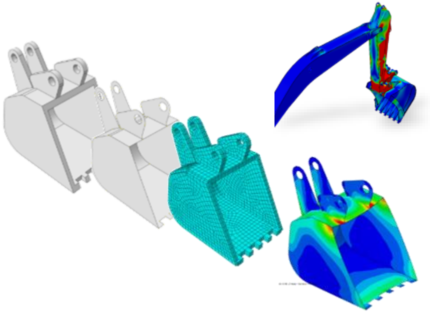
Abaqus FEA, fe-safe Durability & Tosca Optimization Technology

World’s technology-leading suite of finite element analysis software



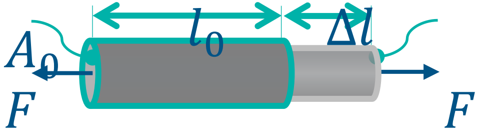
Complex materials, assemblies, contact, fracture, failure & durability

Rapid turnaround with High-Performance Computing (HPC)

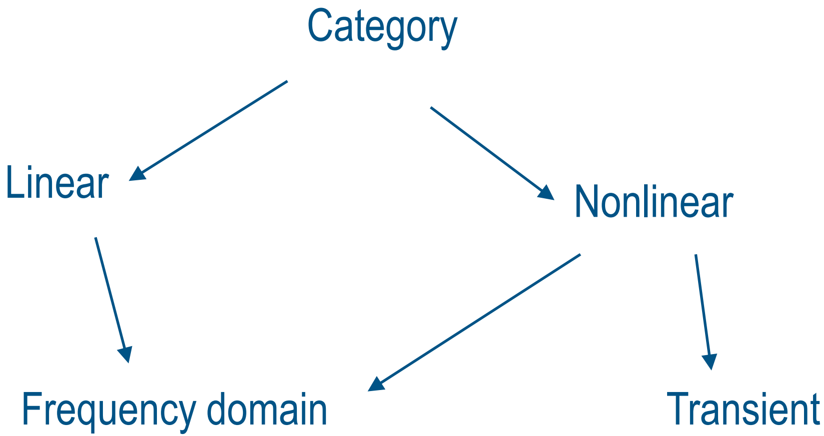


STRUCTURAL SIMULATIONS

- In the following it is assumed that the audience is not necessary experts in the simulation discipline
- This presentation focues on simulations based on the Finite Element Method – FEM
  - There are other abstraction levels that also may fall into this category
- FEM is usually based on the theory of continuum mechanics
  - Stress is force divided by the area
- Natural for fine grained metals
- Not necessarily logical for some composites (e.g concrete..)
- Requirers mathematical material models
  - May need tests to calibrate

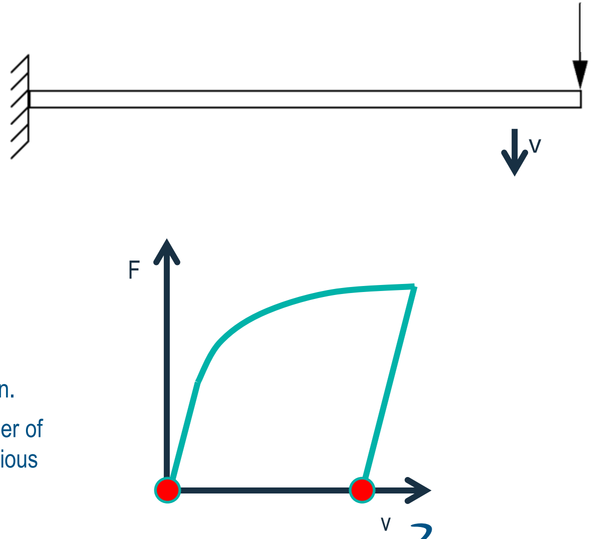


STRUCTURAL SIMULATIONS



In the elastic regime, only one solution.

In the plastic regime, an infinite number of solutions exist depending on the previous load history.



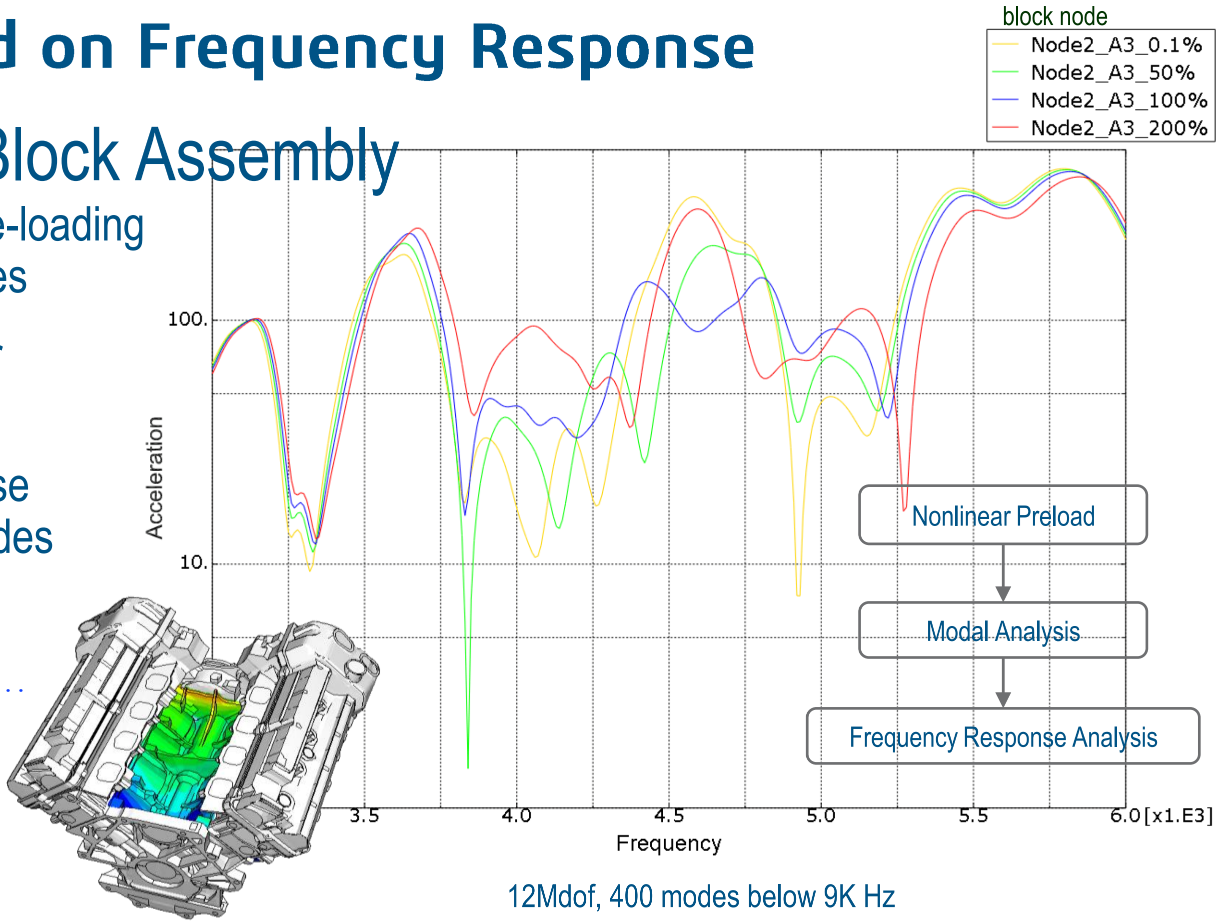
STRUCTURAL SIMULATIONS

- Vibration analysis are often used for qualifying various performance aspects of products today
  - Can be for fatigue evaluations
  - Can be for prediction of noise
- Often times these simulations are done using modal methods
  - The system is reduced by extracting the eigenmodes of the system
  - Simulations in the frequency domain can be ‘cheaper’ to perform
  - Time domain simulations can also be used
- This type of simulations are generally and widely accepted in most industries
- Several challenges exists though:
  - Inherent nonlinearites may require nonlinear pre-loading
  - The user will have to make sure that suffient number of eigemodes are extracted
  - Damping mechanisms can be difficult to include and may require exprimental data for calibration

Nonlinear Preload on Frequency Response

Engine Head and Block Assembly

- Effects of nonlinear bolt pre-loading on engine system responses
- "Stress Stiffening" at higher frequencies
- Distinct changes in response correlate to changes in modes



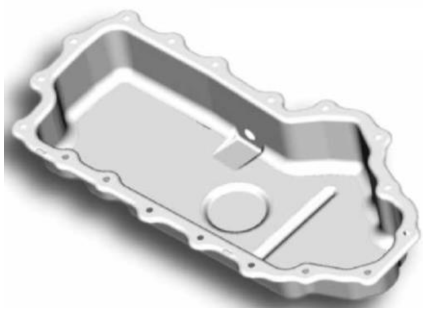
12Mdof, 400 modes below 9K Hz

```
*GASKET SECTION, ELSET=GSK1, BEHAVIOR=MGSK1
*GASKET BEHAVIOR, NAME=MGSK1
*GASKET THICKNESS BEHAVIOR, DIRECTION=LOADING, ...

*PRE-TENSION SECTION,SURFACE=SPT1IN,NODE=9000001
*PRE-TENSION SECTION,SURFACE=SPT1EX,NODE=9000002

*CLOAD,OP=NEW
 9000001, 1, 70000.0
 9000002, 1, 70000.0
```

"Understanding the Effects of Nonlinear Preloads on Engine System Dynamic Response using Abaqus/Standard" (TB-13-NLP-1)

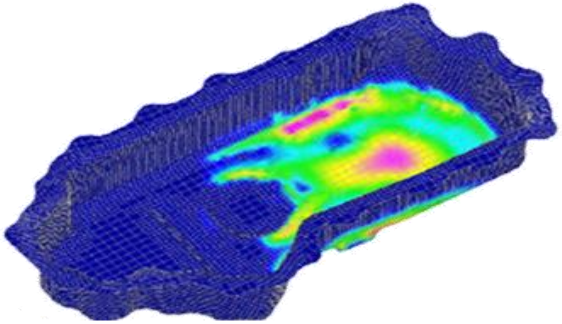
Modal Transient Analysis

- Oil pan vibration: accounting for sheet metal forming
  - Thickness reduction from stamping
  - Residual stresses from springback and trimming
  - Modal dynamic analysis results are imported into *fe-safe* to predict fatigue life
  - Inclusion of the formed state reduces predicted life

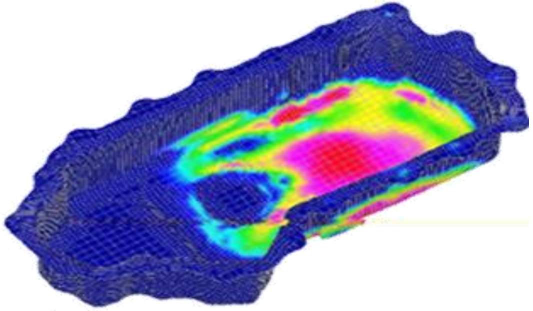


Formed part

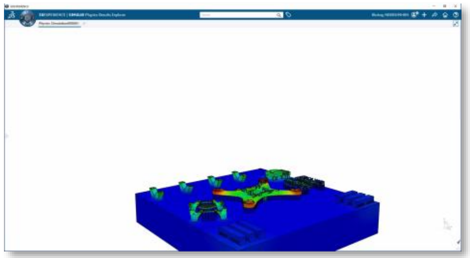
Calculated first mode:
with forming effects: 168 Hz
without forming effects: 185 Hz

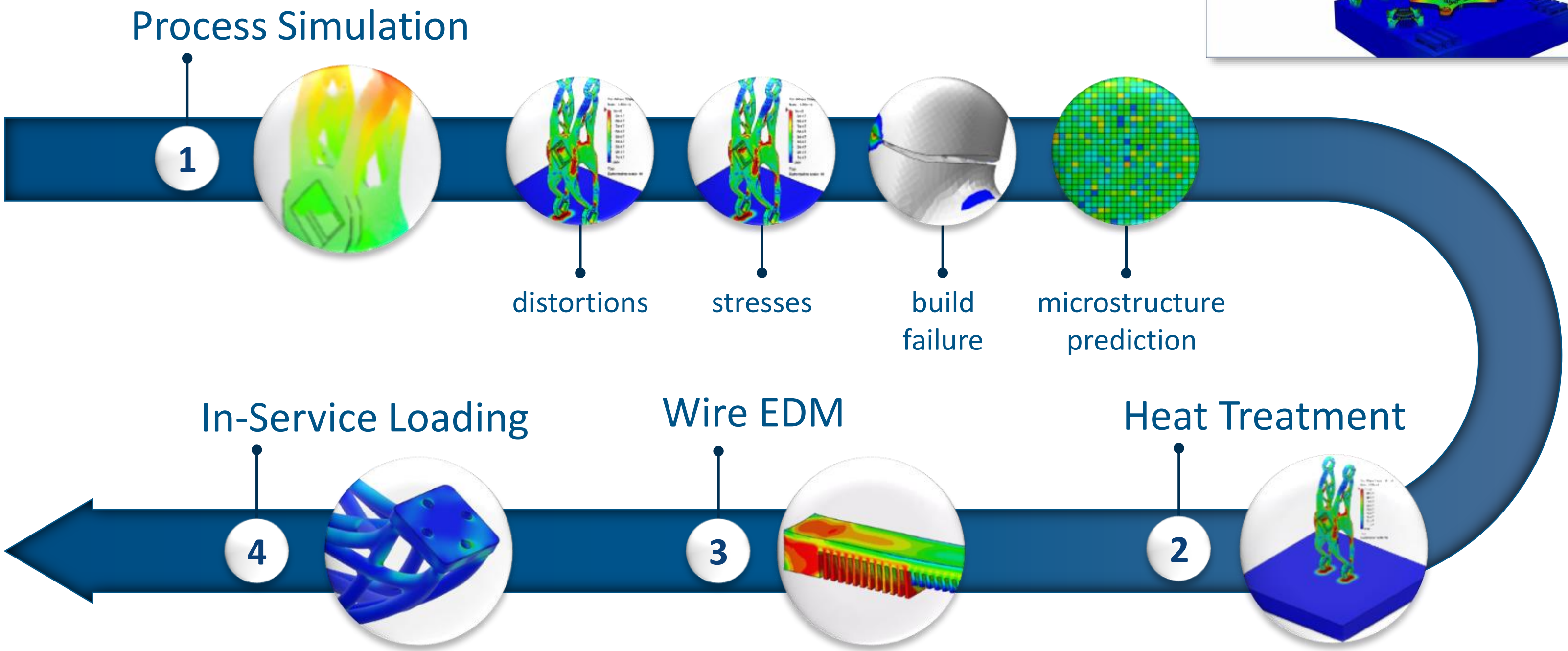


Life excluding forming effects



Life including forming effects





Incorporate residual stress effects and manufacturing history to investigate in-service part performance; e.g. NVH

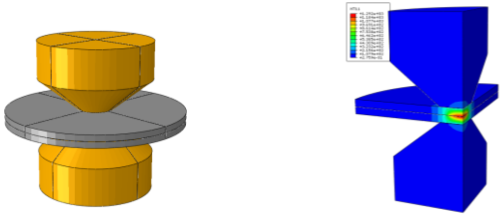
TABLE OF CONTENTS

MULTIPHYSICS

- Multiphysics means that more than one physical field is considered
- Examples might be:
  - Temperature-displacement
  - Structural-acoustics
  - Pore-pressure-displacement
  - Electromagnetic-thernal
  - Coupled Temperature-Displacement, Electrochemical, Pore Pressure (battery cells)
  - ..
- In some cases the coupling is weak
  - Allows for sequential coupling
- In some cases the coupling is strong
  - Requiring a fully coupled procedure

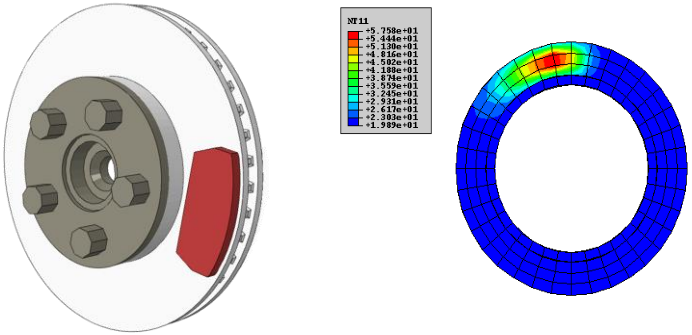
Multiphysics Analysis

Coupled Thermal-Electrical-Stress Analysis

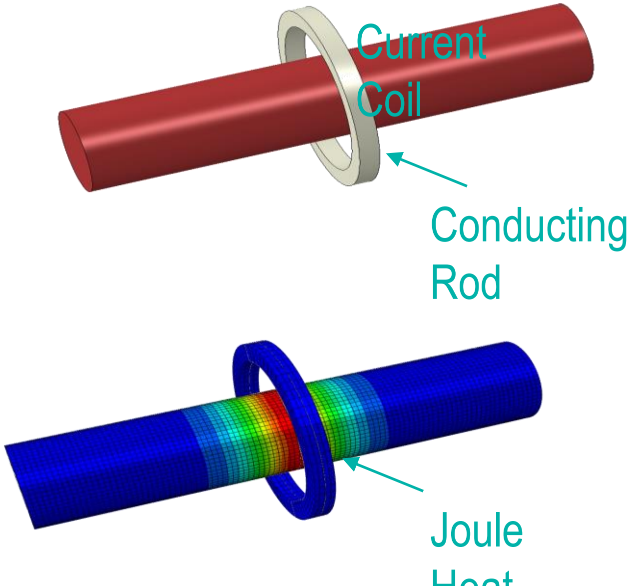


Resistance spot welding

Thermal Stress Analysis



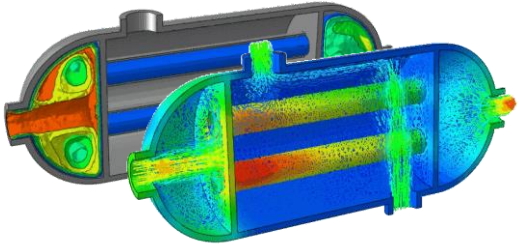
Disk brake



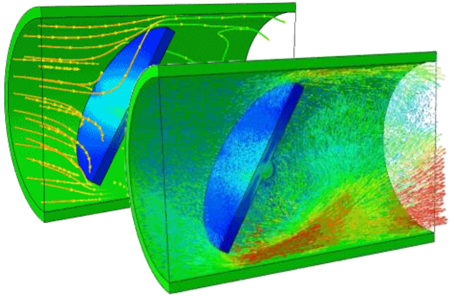
Multiphysics Analysis

FSI

Water can drop test

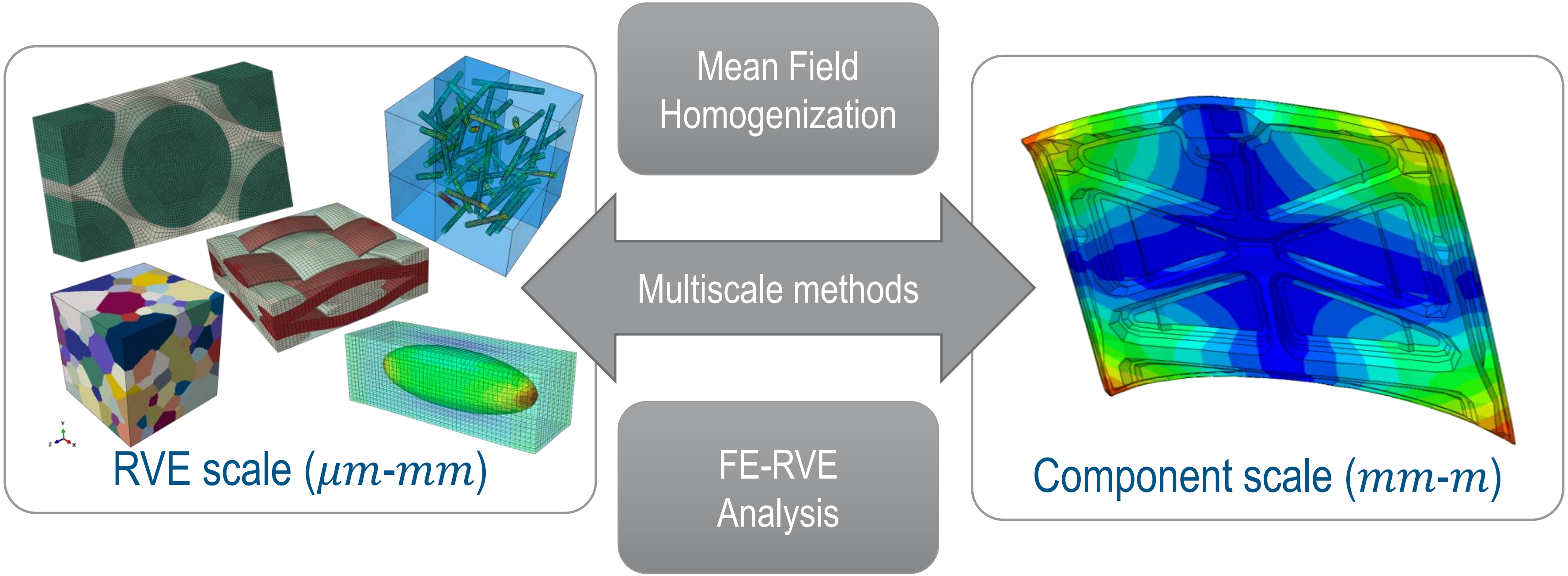


Heat Exchanger



Butterfly valve

TABLE OF CONTENTS

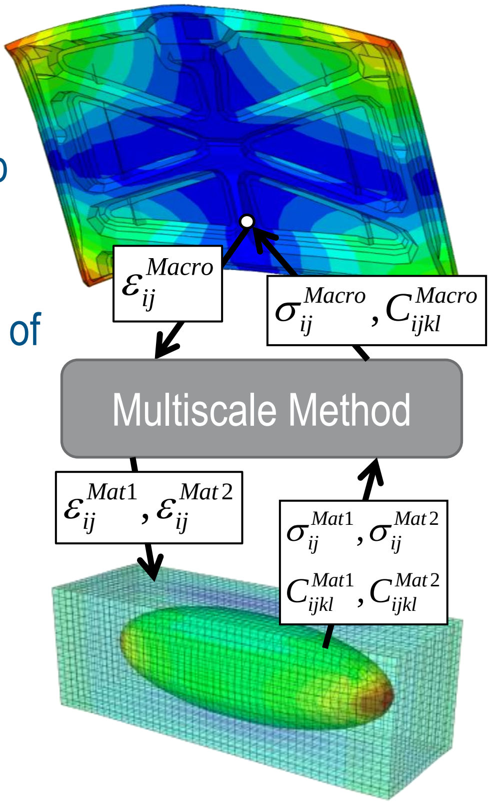

Mean Field
Homogenization
Multiscale methods
FE-RVE
Analysis
RVE scale (μm-mm)
Component scale (mm-m)

Mean-Field

- Analytically calculates relationship between macro/micro strains/stress/stiffness
- Fast – can be run in FEA analysis of macro model to capture nonlinear constituent behavior
- Constituent behavior based on *constituent-averaged* strain

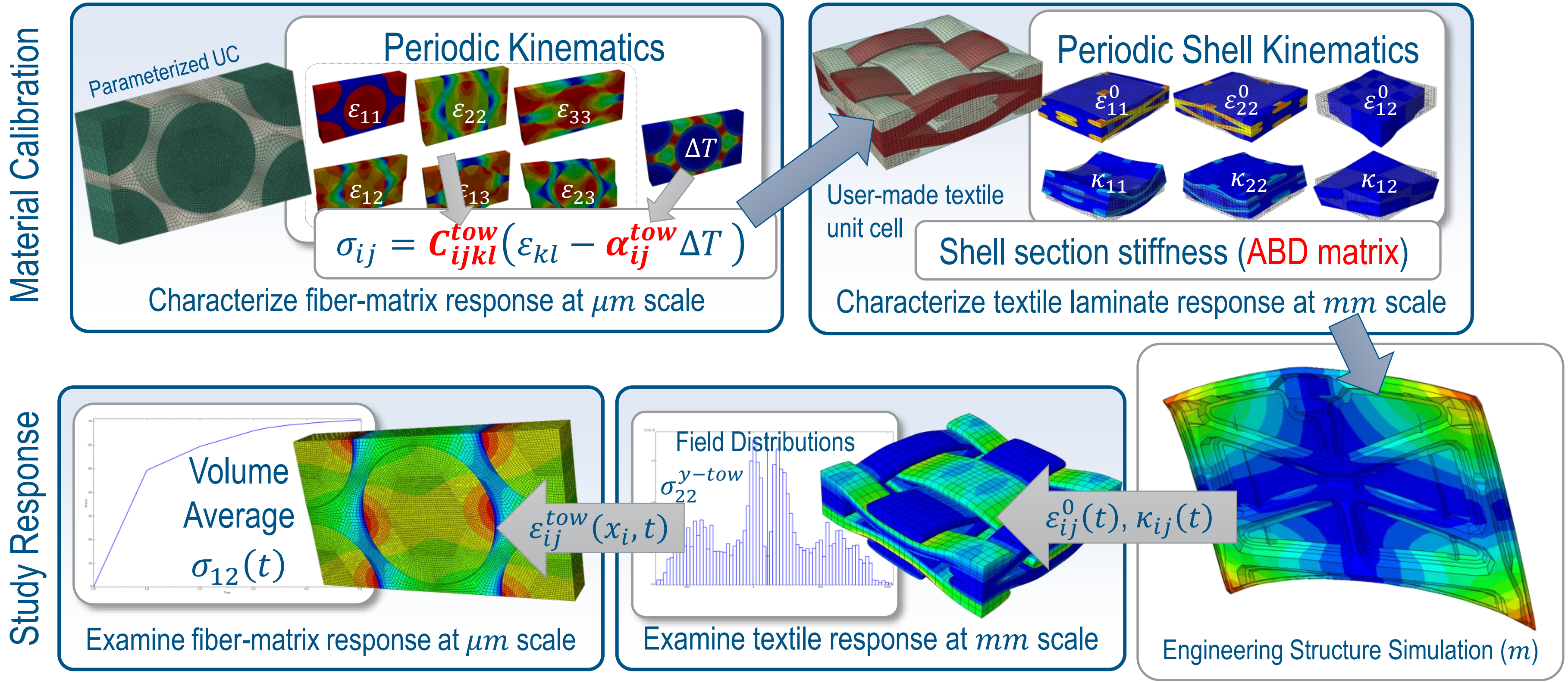
$$\left\langle \varepsilon_{ij}^{Mat1} \right\rangle = A_{ijkl}^{Mat1} \varepsilon_{kl}^{Macro}$$

- Analytical relationships based on many simplifying assumptions



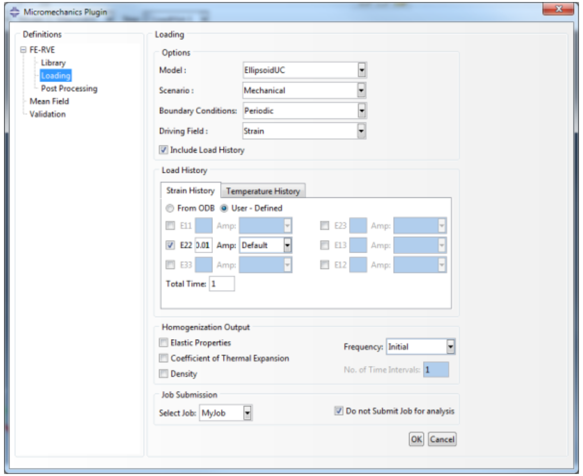
FE-RVE

- Calculates micro fields with FEA
- Slow – performance depends on micro model size
- Not set up to run concurrently with macro-scale model
- Accurate – predicts full solution field in micro model
- Assumes far-field strain doesn’t vary significantly between adjacent micro-scale models



Material Calibration
Parameterized UC
Periodic Kinematics
ε_{11} ε_{22} ε_{33} ΔT
ε_{12} ε_{13} ε_{23}
$\sigma_{ij} = C^{tow}_{ijkl}(\varepsilon_{kl} - \alpha^{tow}_{ij}\Delta T)$
Characterize fiber-matrix response at μm scale
User-made textile unit cell
Periodic Shell Kinematics
ε^0_{11} ε^0_{22} ε^0_{12}
κ_{11} κ_{22} κ_{12}
Shell section stiffness (ABD matrix)
Characterize textile laminate response at mm scale
Study Response
Volume Average $\sigma_{12}(t)$
$\varepsilon^{tow}_{ij}(x_i, t)$
Examine fiber-matrix response at μm scale
Field Distributions
σ^{y-tow}_{22}
$\varepsilon^0_{ij}(t), \kappa_{ij}(t)$
Examine textile response at mm scale
Engineering Structure Simulation (m)

- CAE plugin to facilitate FE-RVE analysis
- What is an FE-RVE?
  - Built-in and user-defined RVE’s
  - Automatic setup of boundary conditions, loads, homogenization steps
  - Solid-solid & shell-solid
  - Generation of elastic & thermal properties
  - Results visualization
- FE-RVE is loaded based on far-field solution, local solution field obtained from finite elements



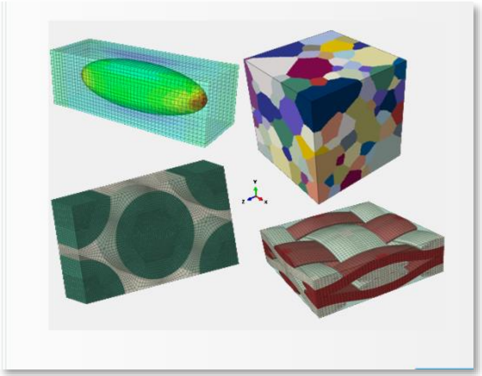


TABLE OF CONTENTS

EXAMPLE – HYDROGEN EMBRITTLEMENT

- Mass diffusion analysis can be used to simulate diffusion of one material through another – such as hydrogen in a metal
- The gouvering equation is:

$$\mathbf{J} = -s\mathbf{D} \cdot \left[\frac{\partial \phi}{\partial \mathbf{x}} + \kappa_s \frac{\partial}{\partial \mathbf{x}} \left(\ln(\theta - \theta^Z) \right) + \kappa_p \frac{\partial p}{\partial \mathbf{x}} \right]$$

where D(c,θ,f) is the diffusivity; s(θ,f) is the solubility; κs(c,θ,f) is the “Soret effect and κp(c,θ,f) is the pressure stress factor

EXAMPLE – HYDROGEN EMBRITTLEMENT

- The physical problem considered here is that of a pressure vessel shell wall fabricated from 2 1/4 Cr–1 Mo steel alloy base metal with an internal weld overlay of Type 347 stainless steel
- The problem is run in two parts. The first part consists of a step in which a single increment of steady-state uncoupled mass diffusion analysis is performed with an arbitrary time step to establish the initial steady-state hydrogen concentration distribution corresponding to the initial temperature
- The hydrogen diffusion during cooling is then analyzed in a mass diffusion transient analysis step. Results given by Fujii et al. (1982), with which we compare the Abaqus results, are presented at four specific times during the transient: 2.7 h (673.15 K, 400.0°C), 5.2 h (623.15 K, 350.0°C), 10.2 h (523.15 K, 250.0°C), and 21.5 h (298.15 K, 25.0°C)

Figure 1. Pressure vessel shell wall section.

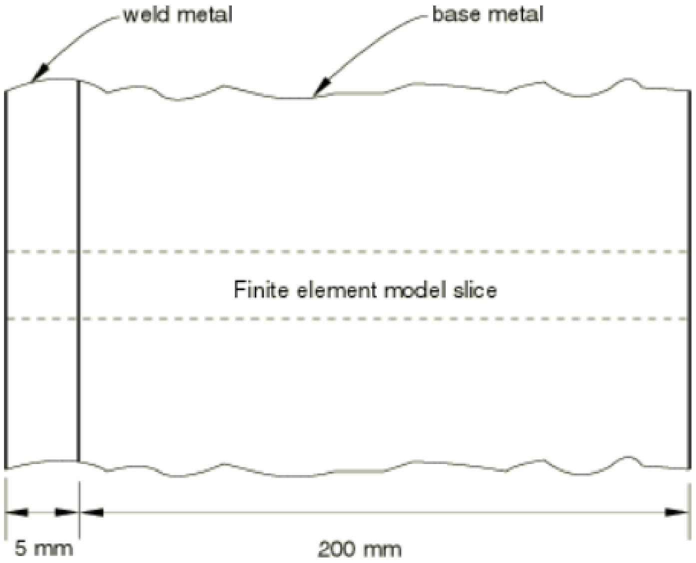


Figure 2. Finite element model of shell wall.

References

Fujii, T., T. Nazama, H. Makajima, and R. Horita, "A Safety Analysis on Overlay Disbonding of Pressure Vessels for Hydrogen Service," Journal of the American Society for Metals, pp. 361–368, 1982

EXAMPLE – HYDROGEN EMBRITTLEMENT

Figure 3. Hydrogen concentration distribution in weld metal.

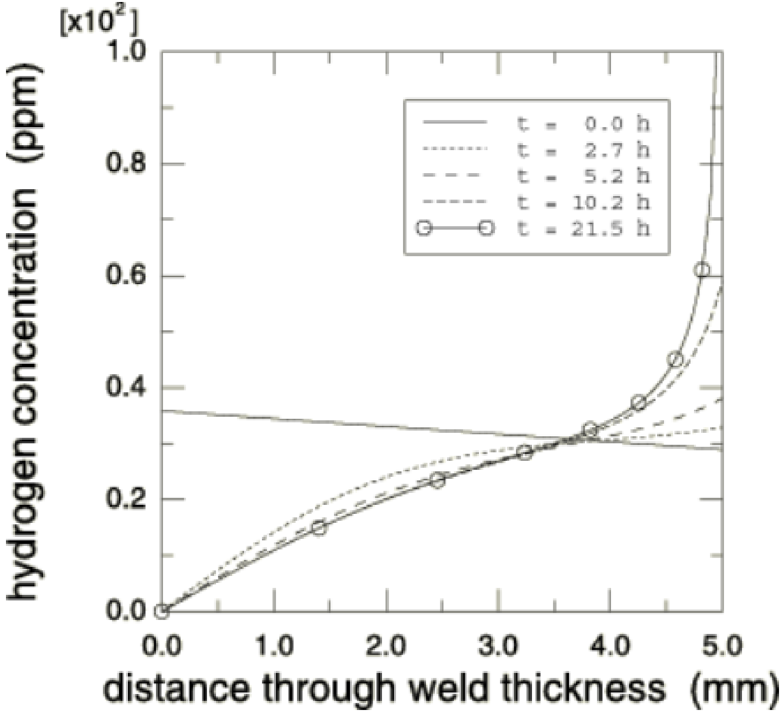
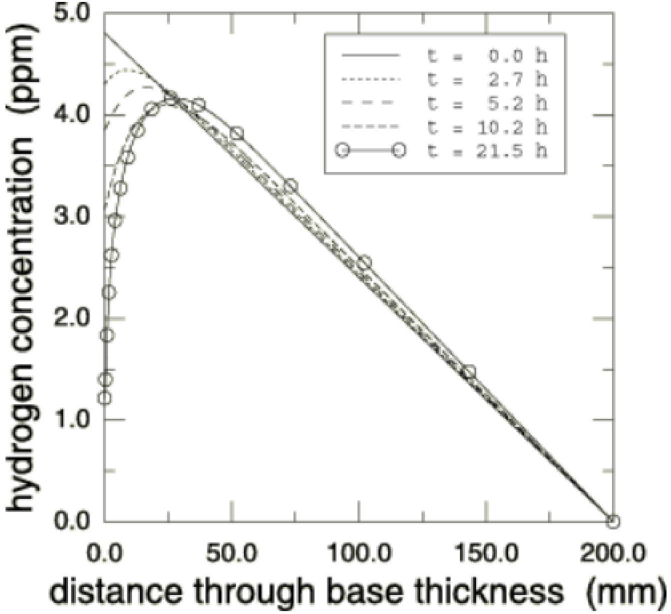
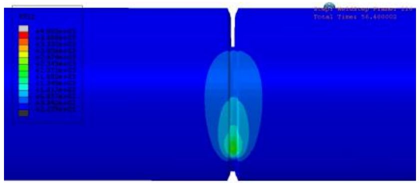


Figure 4. Hydrogen concentration distribution in base metal.

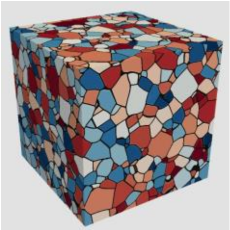


Hydrogen Embrittlement Workflow – under validation



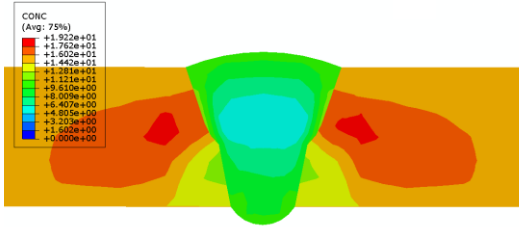
Welding Simulation

Residual stresses and microstructure composition in/near weld
(from: Abaqus welding simulation)



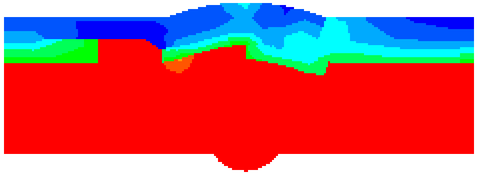
Molecular Simulation

Diffusivity and Solubility in/near weld
(from: molecular scale simulation – BIOVIA)



Hydrogen Diffusion

H2 concentration in/near weld
(from: Abaqus H2 mass diffusion simulation)



Fracture Simulation

Material degradation and failure from fracture mechanics and fatigue analysis
(from: Abaqus XFEM and/or cohesive zone)

TABLE OF CONTENTS

SUMMARY

- Simulation can be used to predict many product performance indicators
- Simulation can provide useful insights
- Simulation can be used to plan test set up and thus improve efficiency
- Simulation set usually requires
  - Understanding the physics involved
  - Good engineering judgement
- Verification of reference cases are recommended
  - Both for simulation and tests

IF WE ™