
4RealSim
Finite Element Analysis for the
Aerospace Industry

4RealSim

- Company specialised in Finite Element Software and Services
- Based in the Netherlands and Spain
- 8 Finite Element specialists with 40+ years of experience
- Working for aerospace industry
  - OEM
  - Tier 1 suppliers
  - Tier 2 suppliers

Customer quotes

"We repeatedly use the Finite Element Analysis expertise of 4RealSim to help us with our challenging problems.

Recently we were unable to solve a fuel tank simulation, as we were running against the limits of our hardware.

With the help of 4RealSim, we build a workaround to tackle this issue. 4RealSim solution not only made it possible to run the simulation on our hardware. Their solution also reduced the required hardware resources tremendously, making the simulation much more practical for our production sites."

Plastic Omnium

Very good reactivity, availability and flexibility of the team

All milestones of the project were respected

Technical expertise on the domain

Sabena Technics

Solutions for the Aerospace Industry

- Engineering Services (Worldwide)

- Abaqus (Benelux and Spain)
- Isight (Benelux and Spain)
- Fe-safe (Benelux and Spain)
- Tosca (Benelux and Spain)
- Simpack (Benelux and Spain)
- VCollab (Worldwide)
- HyperSizer (Europe)

Disclaimer

The following slides contain examples of our expertise

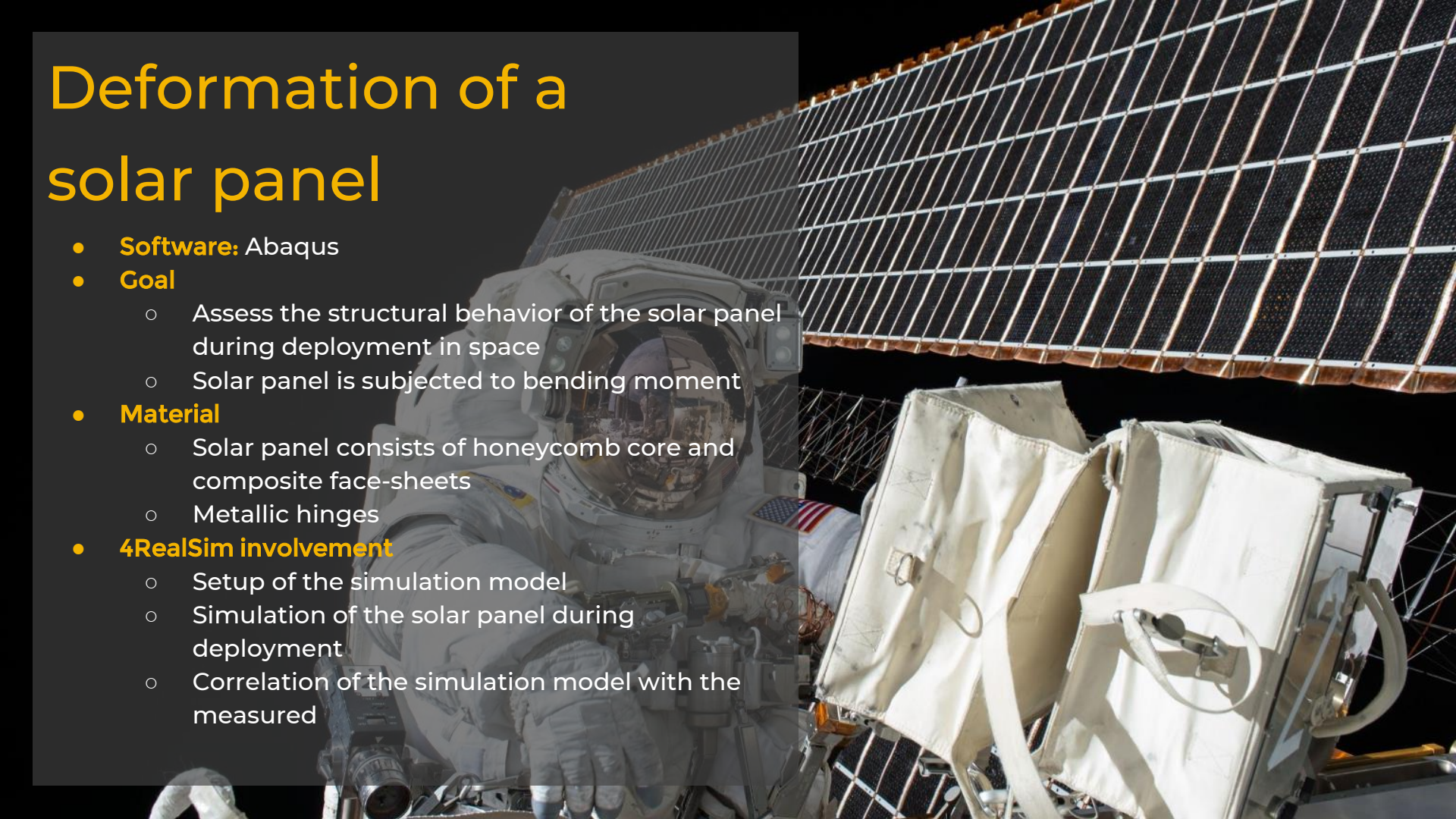
Due to confidentiality, original pictures are not shown in this presentation

Topics

- Deformation of a solar panel (structural simulation)
- Boom deployment (multibody simulation)
- Bird strike (impact simulation)
- Wing Leading Edge Laminate Optimization for bird impact (impact simulation + optimization)
- Crack growth
- User-subroutine development
- Python script and plug-in development
- Fuel tank sloshing
- Simulating Spacecraft Launch and Re-entry
- FEA driven optimization of a stiffened panel construction
- Topology optimization
- Topology and beam optimization

Deformation of a solar panel

- **Software:** Abaqus
- **Goal**
  - Assess the structural behavior of the solar panel during deployment in space
  - Solar panel is subjected to bending moment
- **Material**
  - Solar panel consists of honeycomb core and composite face-sheets
  - Metallic hinges
- **4RealSim involvement**
  - Setup of the simulation model
  - Simulation of the solar panel during deployment
  - Correlation of the simulation model with the measured

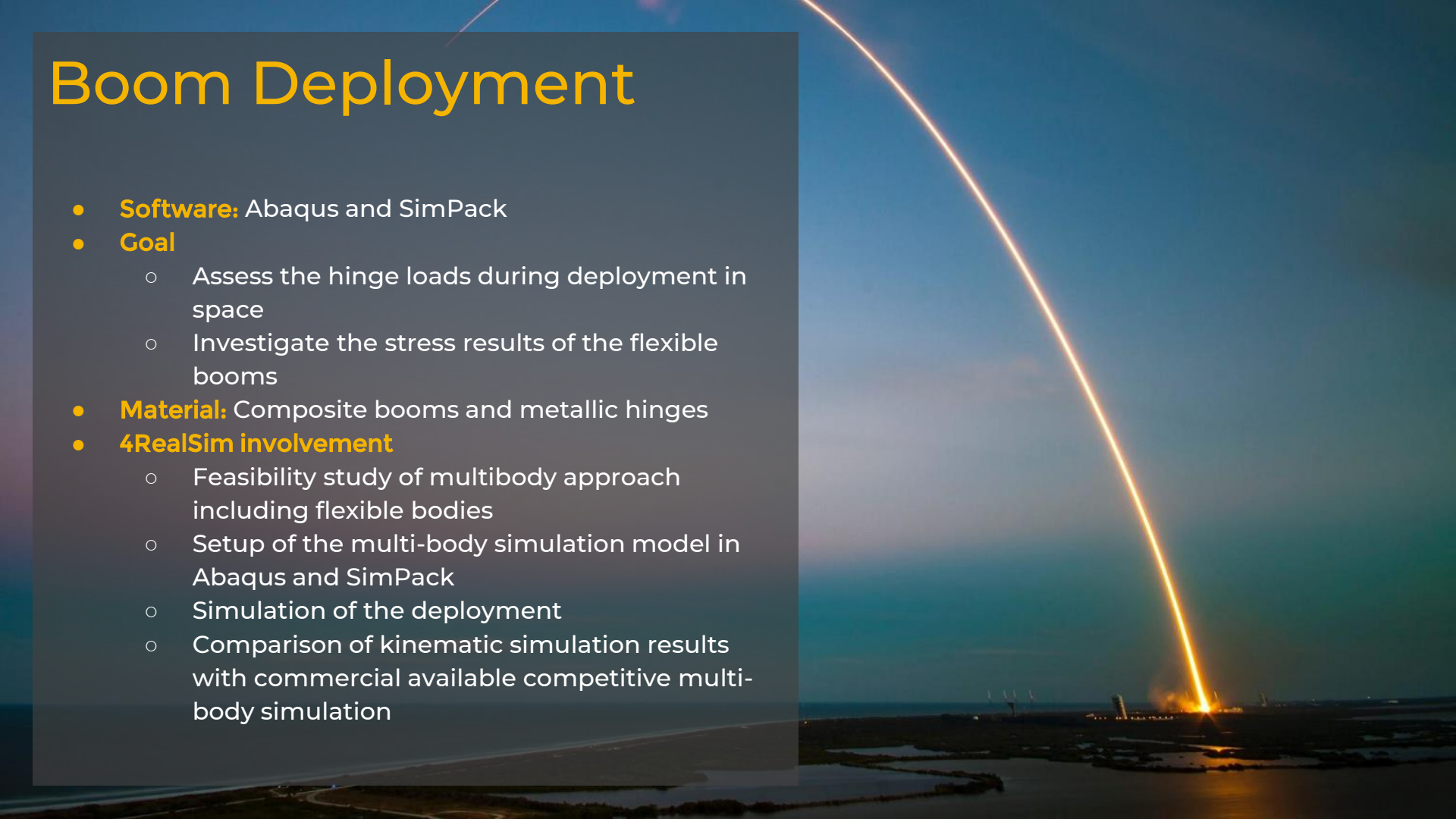


Wing Leading Edge Laminate Optimization

- **Software:** Abaqus
- **Goal**
  - Optimize leading edge of a composite wing to resist bird impact
- **Material**
  - Composite
- **4RealSim involvement**
  - Define methodology
  - Couple HyperSizer/Abaqus
- **Customer**
  - Fokker

Boom Deployment

- **Software:** Abaqus and SimPack
- **Goal**
  - Assess the hinge loads during deployment in space
  - Investigate the stress results of the flexible booms
- **Material:** Composite booms and metallic hinges
- **4RealSim involvement**
  - Feasibility study of multibody approach including flexible bodies
  - Setup of the multi-body simulation model in Abaqus and SimPack
  - Simulation of the deployment
  - Comparison of kinematic simulation results with commercial available competitive multi-body simulation



Bird strike analysis

- **Software:** Abaqus
- **Goal**
  - Assess the structural integrity of aerostructures after bird impact
- **Type of structures**
  - Wings, tails, fuselage, pods, domes
- **Material**
  - Metallic and composite including damage
  - Bird material
  - Cohesive behavior to include delamination
- **4RealSim involvement**
  - Setup of the simulation model in Abaqus
  - Simulation of the bird strike
  - Comparison of results with physical tests

Crack growth with XFEM

- **Software:** Abaqus
- **Goal**
  - Predict speed and path of crack growth
- **Type of structures**
  - Wings, fuselage, etc.
- **Material**
  - Metallic and composite
  - Cohesive behavior to include delamination
- **4RealSim involvement**
  - Setup of the simulation model in Abaqus
  - Simulation of the crack growth
  - Comparison of results with physical tests

Python script and plug-in development

- **Software:** Abaqus
- **Goal**
  - Develop Abaqus/CAE and Abaqus/Viewer python scripts to automate simulation tasks
- **4RealSim plug-ins examples**
  - QA Report Generator
  - Failure envelope script
  - Fastener generation script
  - Virtual strain gauge plug-in

Topology optimization

- **Software:** Abaqus, Tosca
- **Goal**
  - Weight reduction of a landing gear structure
- **Material**
  - Metallic
- **4RealSim involvement**
  - Setup of the simulation model in Abaqus
  - Setup of the topology optimization in Tosca
  - Optimization
  - Result conversion to CAD
- **Customer**
  - Fokker Landing Gear

Fuel tank sloghing

- **Software:** Abaqus
- **Goal**
  - Perform fuel sloshing with CEL/SPH approach
- **Material**
  - Water, Metallic
- **4RealSim involvement**
  - Setup of the simulation model in Abaqus
  - Simulation
  - Comparison with physical tests

User-subroutine development

- **Software:** Abaqus
- **Goal**
  - Develop user-subroutine to enhance the existing damage evolution models
- **Material**
  - Fiber reinforced composite
- **4RealSim involvement**
  - Develop the user-subroutine

Simulating Spacecraft Launch and Re-entry

- **Software:** Abaqus
- **Goal**
  - Simulate the thermal and structural behavior of a new metallic thermal protection system for hypersonic space vehicles
- **Material**
  - Metallic
- **4RealSim involvement**
  - Support of the customer to develop the structural and thermal models
  - Support of the customer to map the CFD results on the FEA model
- **Customer**
  - Dutch Space (now Airbus Defence and Space Netherlands)

Topology and beam optimization

- **Software:** Abaqus, Tosca, HyperSizer Pro
- **Goal**
  - Optimization of a ring structure
- **Material**
  - Metallic
- **4RealSim involvement**
  - Ideal topology of inner ring identified via topology optimization
  - Ideal beam family for each of the beams
  - Found optimal configuration satisfying all margins of safety

Fea driven optimization of a stiffened panel construction

- **Software:** Abaqus, Isight, HyperSizer Pro
- **Goal**
  - Optimize an interstage model in the early design phase
  - Proposed concept is based on stiffened shell construction with ring-frames
- **Material**
  - Metallic
- **4RealSim involvement**
  - Isight
    - Automate the simulation process
      - Make parametric model (Nr stiffeners, height & width stiffeners, thickness of different components
      - Extract output results (Frequency, Stress, Buckling mode)
    - Run the optimization to optimize the weight and satisfy all constraints
  - HyperSizer
    - Preliminary sizing with smeared model approach
      - Identify spacing of stiffeners
      - Define failure analyses
    - Detailed sizing with a discrete model approach
      - Identify the optimal stiffener shape for each panel
      - Define failure analyses

4RealSim

Contact us
for more information:

info@4realsim.com

NL: +31 30 260 0593
ES: +34 96 000 1823

- Company specialised in Finite Element Software and Services
- Based in the Netherlands and Spain
- 8 Finite Element specialists with 40+ years of experience
- Experience with finite element analysis and optimization in the aerospace industry