



4 *RealSim*

Finite Element Analysis for the
Automotive 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 automotive industry

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"

Aircraft Modification Partner



Solutions for the Automotive Industry

Belgium, Netherlands, Luxembourg,
Spain and Portugal

- Abaqus
- Isight
- Fe-safe
- Tosca
- Simpack

Europe

- HyperSizer

Worldwide

- Engineering Services
- VCollab

Disclaimer

The following slides contain examples of our expertise

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



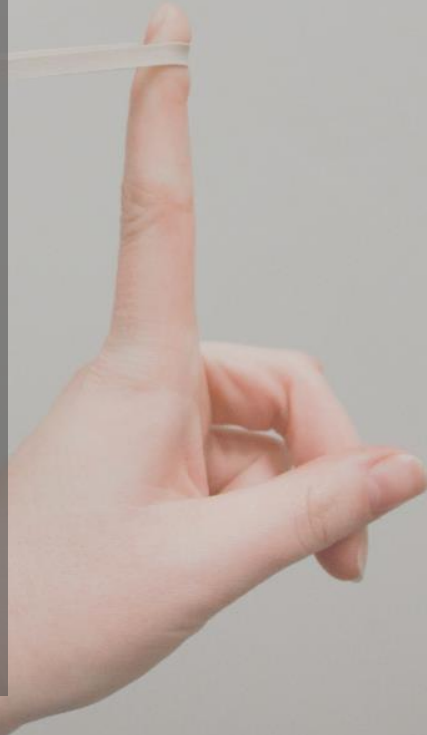


Examples

- Large deformation of rubber bushings
- Noise and vibration analysis
- Fracture and failure to assess crack growth [structural simulation]
- Crash analysis [impact simulation]
- Fuel tank sloshing [FSI simulation]
- Hydroplaning of treaded tires
- Fatigue of powertrain components
- Material characterization (Plastic Omnium)
- Powertrain multibody simulation
- Bracket topology optimization
- User-subroutine development
- Python script and plug-in development

Large deformation of rubber bushings

- **Software**
 - Abaqus
- **Goal**
 - Assess the structural behavior of highly nonlinear rubber bushings
- **Material**
 - Rubber
- **4RealSim involvement**
 - Setup of the simulation model
 - Monitor testing of material samples
 - Define viscoelastic material model
 - Simulation of the rubber bushing
 - Solve mesh issues related to large deformations
 - Comparison simulation results with the measured test data



Noise and vibration analysis

- **Software**

- Abaqus

- **Goal**

- Predict vibrations of engine while taking into account preload effects, contact and advanced material models

- **Material**

- Metallic
 - Frequency-dependent viscoelasticity
 - Acoustic material models (baffles/trim)

- **4RealSim involvement**

- Setup of simulation models
 - Comparing simulation results with test data

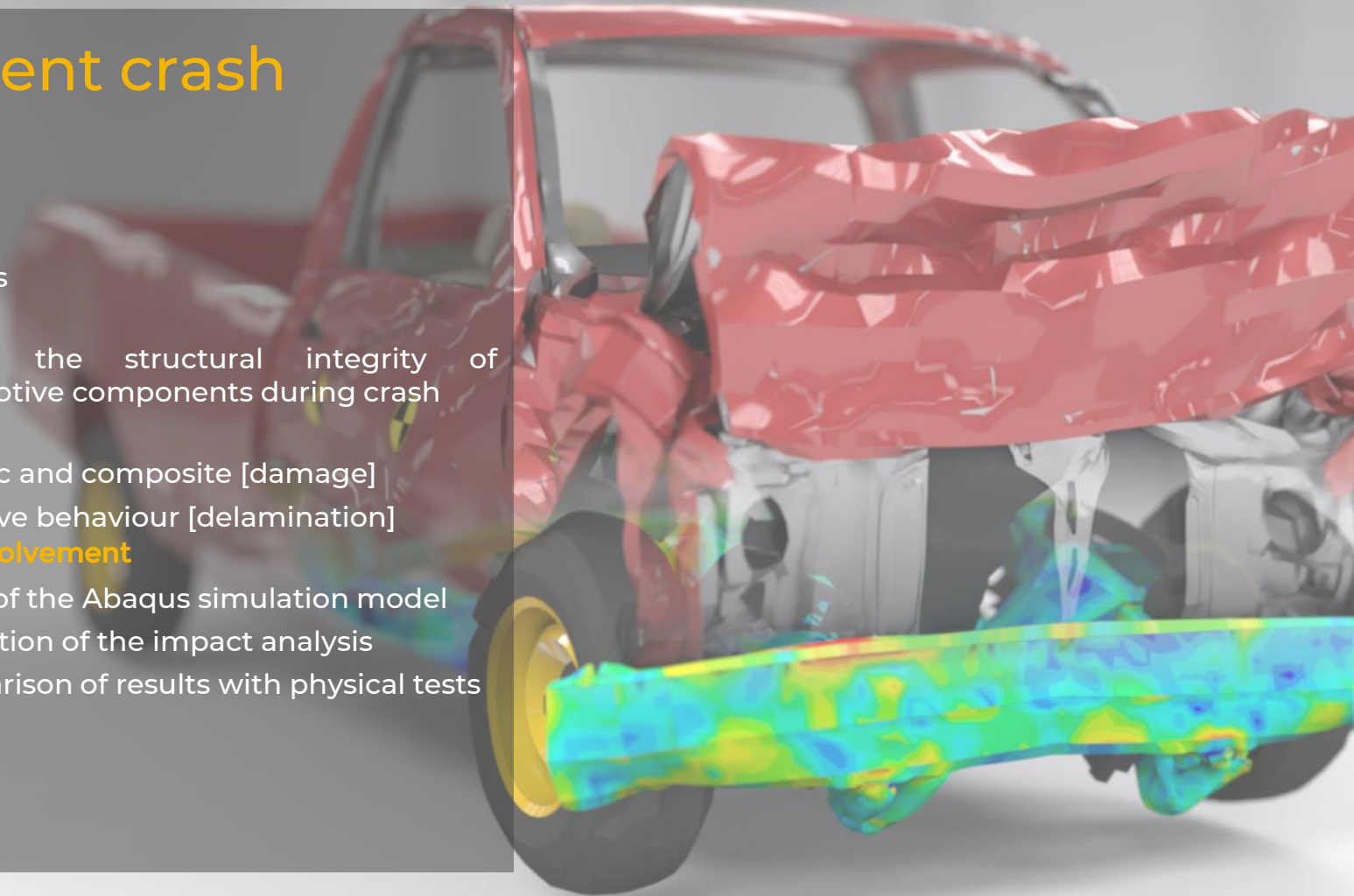


Fracture and failure to assess crack growth

- **Software**
 - Abaqus
- **Goal**
 - Predict speed and path of crack growth
- **Material**
 - Metallic and composite
 - Cohesive behaviour to include delamination
- **4RealSim involvement**
 - Setup of the simulation model in Abaqus
 - Simulation of the crack growth
 - Comparison of results with physical tests

Component crash analysis

- **Software**
 - Abaqus
- **Goal**
 - Assess the structural integrity of automotive components during crash
- **Material**
 - Metallic and composite [damage]
 - Cohesive behaviour [delamination]
- **4RealSim involvement**
 - Setup of the Abaqus simulation model
 - Simulation of the impact analysis
 - Comparison of results with physical tests



Fuel tank sloshing (fluid-structure interaction)

- **Software**

- Abaqus

- **Goal**

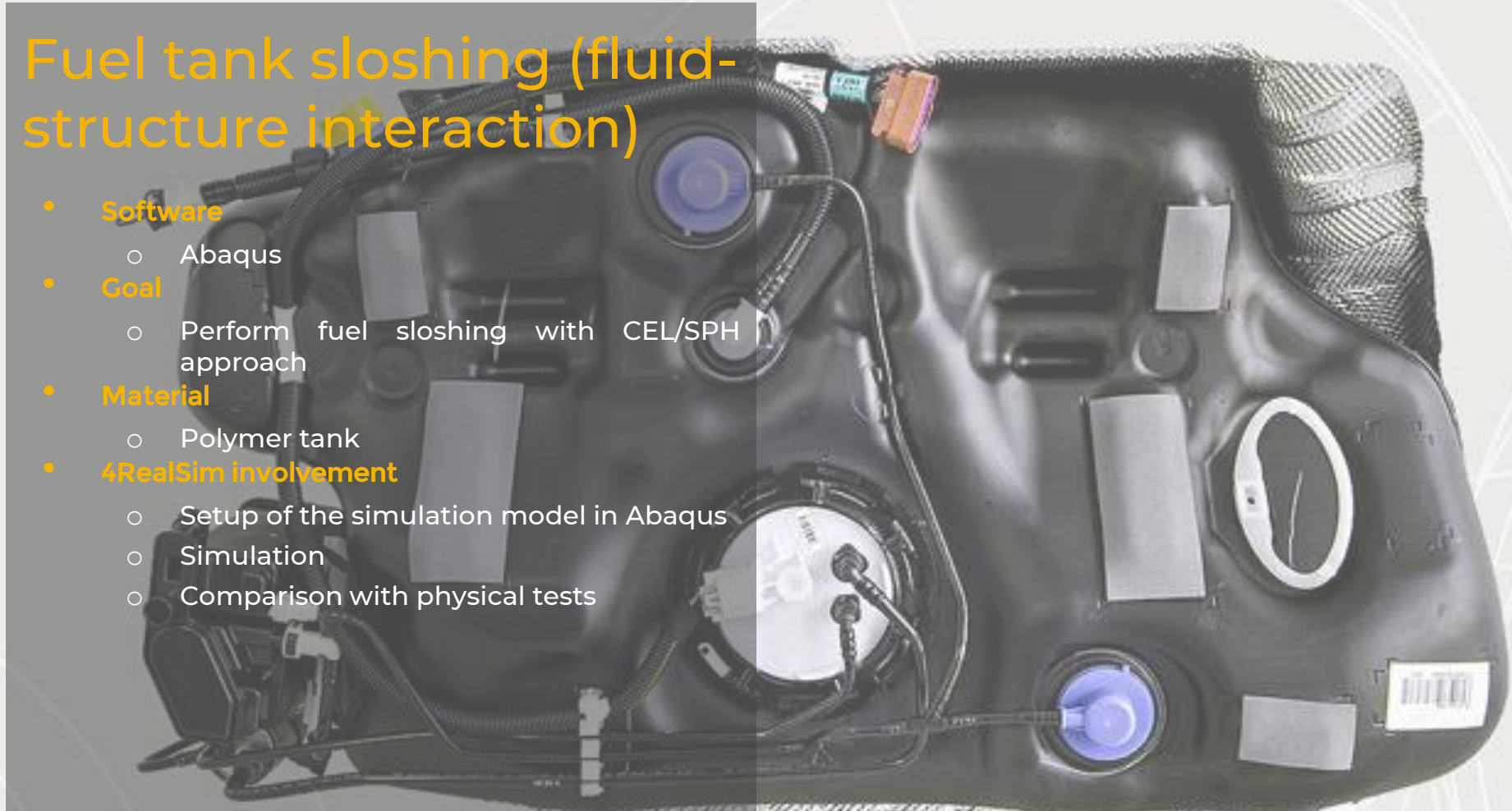
- Perform fuel sloshing with CEL/SPH approach

- **Material**

- Polymer tank

- **4RealSim involvement**

- Setup of the simulation model in Abaqus
- Simulation
- Comparison with physical tests



Hydroplaning of treaded tires (aquaplaning)

- **Software**
 - Abaqus
- **Goal**
 - Assess the loss of contact between the tire and the road due to water buildup underneath the tire
- **Material**
 - Rubber tire
- **4RealSim involvement**
 - Efficient tire-modeling including steady-state transport procedure and import from Abaqus/Standard to Abaqus/Explicit
 - Comparison with physical test



Fatigue of Powertrain Components

- **Software**
 - Abaqus, fe-safe
- **Goal**
 - Assess the durability of powertrain components
- **Material**
 - Steel
- **4RealSim involvement**
 - Define methodology
 - Perform the static simulations
 - Identify the fatigue parameters
 - Fatigue simulations

Material characterization

- **Software**

- Abaqus, Isight

- **Goal**

- Calibrate material models with test data

- **Material**

- Polymer

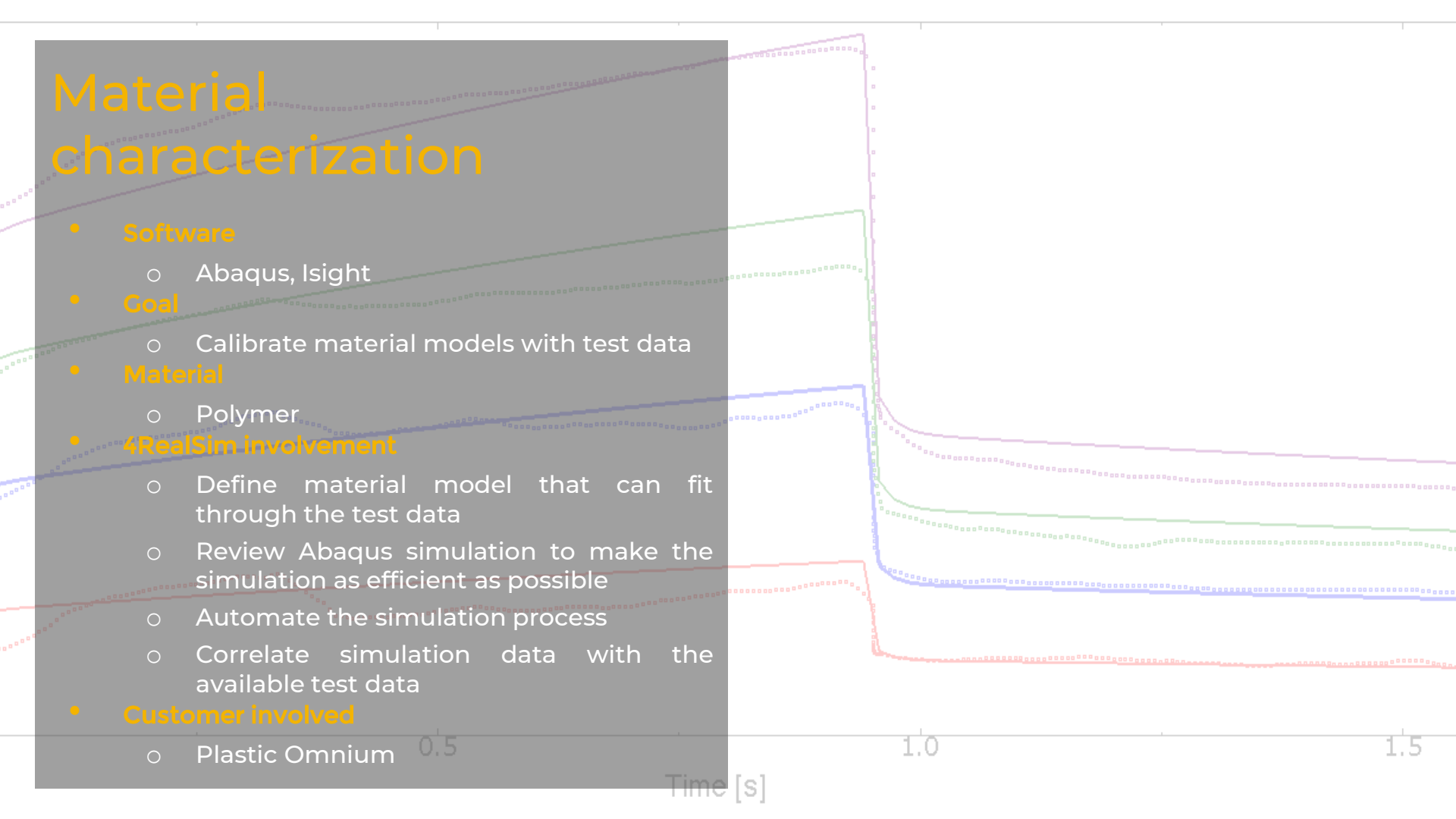
- **4RealSim involvement**

- Define material model that can fit through the test data
 - Review Abaqus simulation to make the simulation as efficient as possible
 - Automate the simulation process
 - Correlate simulation data with the available test data

- **Customer involved**

- Plastic Omnium

Time [s]



Powertrain multibody simulation



- **Software**

- Simpack

- **Goal**

- Multibody simulation model to assess gear whine

- **4RealSim involvement**

- Setup of the simulation model in Simpack
- Evaluate the influence of transmission errors, tooth parameters, finishing quality, gear loading, ...
- Setup of flexible bodies to be included in the MBS
- Compare with test data

Bracket topology optimization

- **Software**
 - Abaqus, Tosca
- **Goal**
 - Nonlinear topology optimization of a bracket component
- **Material**
 - Metallic
- **4RealSim involvement**
 - Define finite element model
 - Correlation to test data
 - Topology optimization on FEA model
 - Reconstruction of the optimal solution



User-subroutine development

- **Software**

- Abaqus

- **Goal**

- Develop user-subroutine to enhance the existing damage evolution models

- **Material**

- Fibre-reinforced composite

- **4RealSim involvement**

- Develop the user-subroutine

```
function(scope, element, attr, ngSwitchController) {  
  scope.$watch(attr.ngSwitch, function() {  
    selectedTranscludes = [];  
    selectedElements = [];  
    previousElements = [];  
    selectedScopes = [];  
  
    scope.$watch(watchExpr, function ngSwitchWatchAction(value) {  
      var i, ii;  
      for (i = 0, ii = previousElements.length; i < ii; ++i) {  
        previousElements[i].remove();  
      }  
      previousElements.length = 0;  
      for (i = 0, ii = selectedScopes.length; i < ii; ++i) {  
        var selected = selectedElements[i];  
        selectedScopes[i].$destroy();  
        previousElements[i] = selected;  
        $animate.leave(selected, function() {  
          previousElements.splice(i, 1);  
        });  
      }  
      selectedElements.length = 0;  
      selectedScopes.length = 0;  
      if ((selectedTranscludes = ngSwitchController.cases['!' + value] || ngSwitchController.defaultCase)) {  
        scope.$eval(attr.change);  
        forEach(selectedTranscludes, function(selectedTransclude) {  
          var selectedScope = scope.$new();  
          selectedScopes.push(selectedScope);  
          selectedTransclude(selectedScope, element);  
        });  
      }  
    });  
  });  
}
```

Python script and plug-in development

- **Software**
 - Abaqus
- **Goal**
 - Develop Abaqus/CAE and 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

```
    "close", "click .collapsed"
    (var b,d=this,e=this
    AssetUrl(c.router.themePath+
    "ready"),a(document.body)
    "ready"),c.router.selected
    this.undelegateEvents
    "collapsed").toggleClass
    togglePreviewDeviceButton
    keyEvent:function
    maybeRequestFileSystem
    Backbone.View.extend
    listenTo(c.collection,
    length),c.announceSearch
    function(){c.overlay
    (b))}})),render:function
    this.renderTheme
```




Contact us
for more information

info@4realsim.com

NL: +31 30 260 0593

ES: +34 96 000 1823

Summary

- Company specialized 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 automotive industry