FIDESYS
strength analysis system

CAE Fidesys overview and examples of its industrial applications

Anatoly Vershinin
CTO at Fidesys LLC

Introduction

Fidesys LLC is an engineering software company with offices in Moscow, Russia and NJ, USA, founded in 2009 as a research group of **Lomonosov Moscow State University.**
Fidesys LLC develops the next-gen universal software suite for high-end structural analysis (CAE, computer-aided engineering) using a new generation of FEA methods.

CAE Fidesys is used in mechanical engineering, mining, oil and gas industries.

The company is a resident of the **Skolkovo Innovation Center** and a member of **NAFEMS**, the International Association for the Engineering Modelling, Analysis and Simulation Community.

CAE Fidesys is verified according to the NAFEMS standards.

Our Team

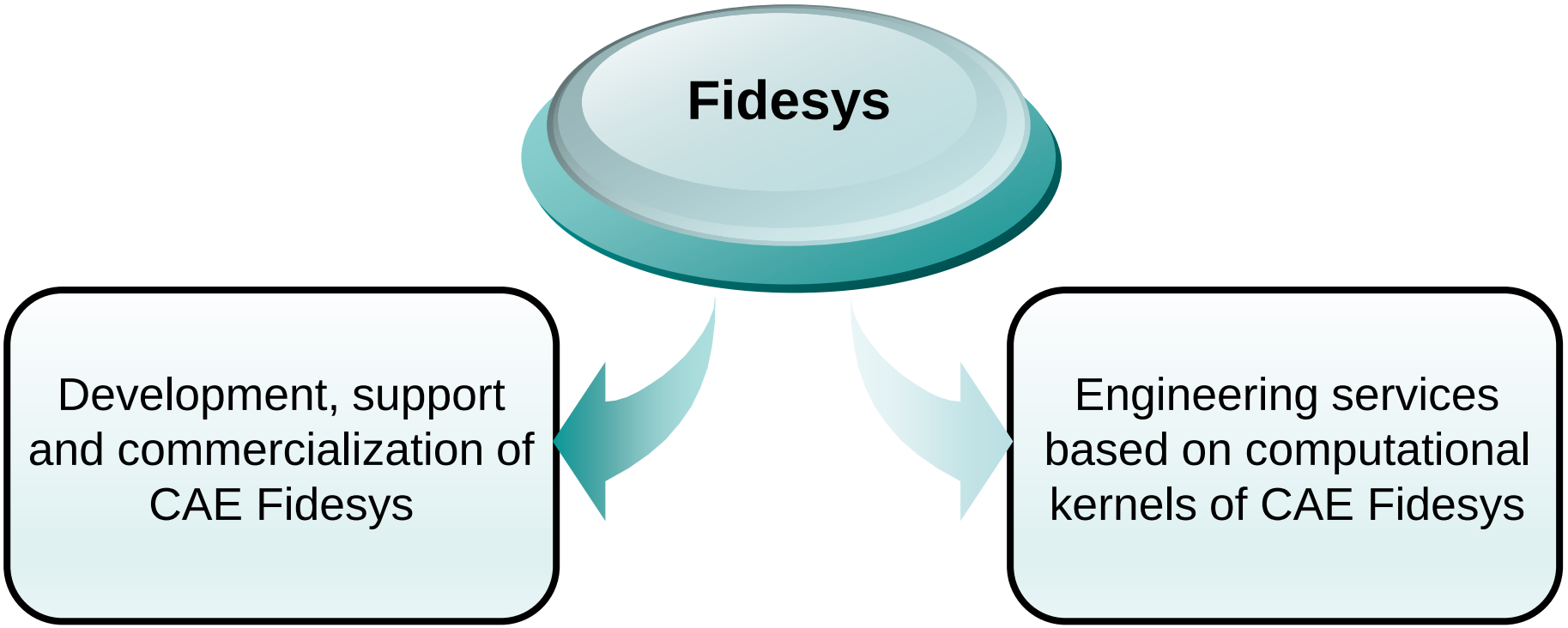
- **20 programmers–mathematicians** (postgraduates, PhDs and PhD candidates from the top Russian universities*)
- **11 consulting professors**, working in the **Russian Academy of Sciences**, **Columbia University**, **University of New Hampshire** and **Iowa State University**
- Management, sales and marketing specialists with extensive experience in various technical fields and international background

Board of directors incl. top-mangers of IT industry (former VPs of PTC CIS, Autodesk CIS)

Key expertise: precise math modeling for structural analysis and related engineering fields

*incl. Lomonosov Moscow State University, Bauman Moscow State Technical University, Moscow Institute of Physics and Technology, and other leading regional universities

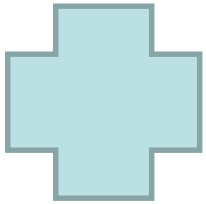
Business directions



CAE Fidesys - Desktop bundles

CAE Fidesys Standard 1.7

- Linear problems of elasticity
- Plane stress and plane strains problems
- 3D static and dynamic (transient) analysis
- Modal and buckling analysis
- Beam/Shell/Solid elements and their combinations



CAE Fidesys Professional 1.7 =

CAE Fidesys Standard 1.7 +

- Accurate estimation of the nonlinear effects: physical, geometrical and contact nonlinearities
- Problems for weakly compressible and completely incompressible materials
- Strength analysis for elastoplastic materials: Mises, Drucker-Prager
- Bonded/sliding/friction contacts
- Hyperelastic materials (Murnaghan, Mooney-Rivlin)
- Thermal conductivity and thermoelastic problems

✓ Fidesys HPC

- Parallelization of major stages of computational process
- Speed-up calculations by up to 30 times
- OpenMP technology: parallelization on all computational cores of a workstation
- MPI technology: parallelization on several workstations inside a network or on nodes of a supercomputer

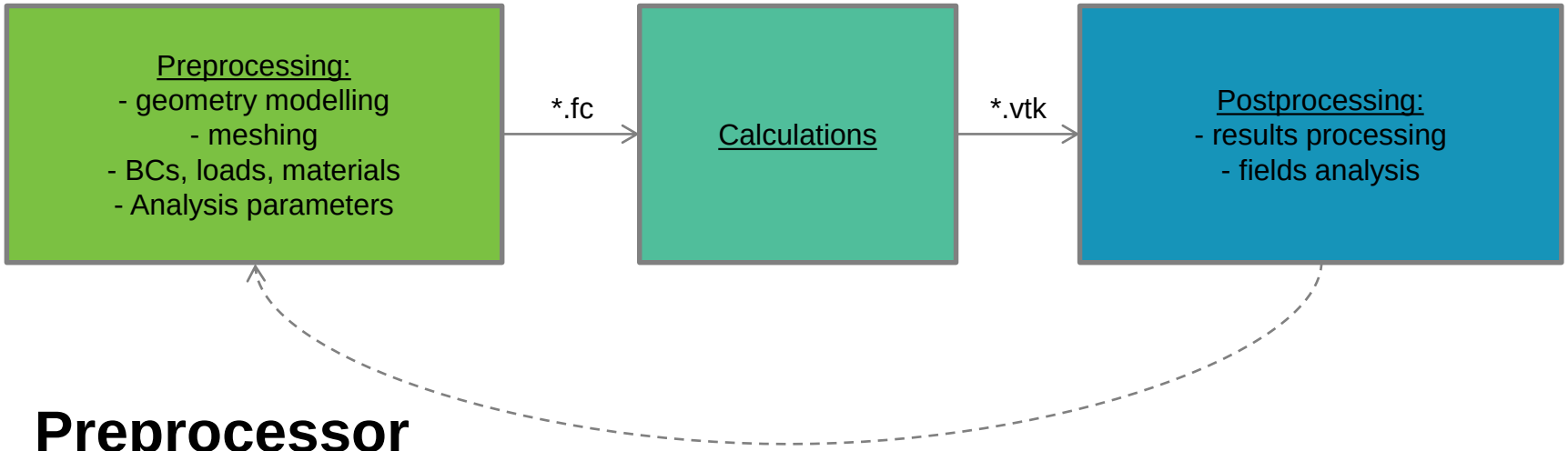
✓ Fidesys Dynamics

- Application of spectral element method for linear and nonlinear problems
- Non-stationary problems
- Full waveform modeling
- Seismic modeling
- Non-destructive control modeling

✓ Fidesys Composite

- Analysis of composites' effective properties
- Construction of the realistic composite's microstructure
- Strength analysis of tools and parts made of composite materials (including porous, fiber-laminated and woven composites)
- Estimation of effective properties of a monolayer
- Rubber-cord materials modelling

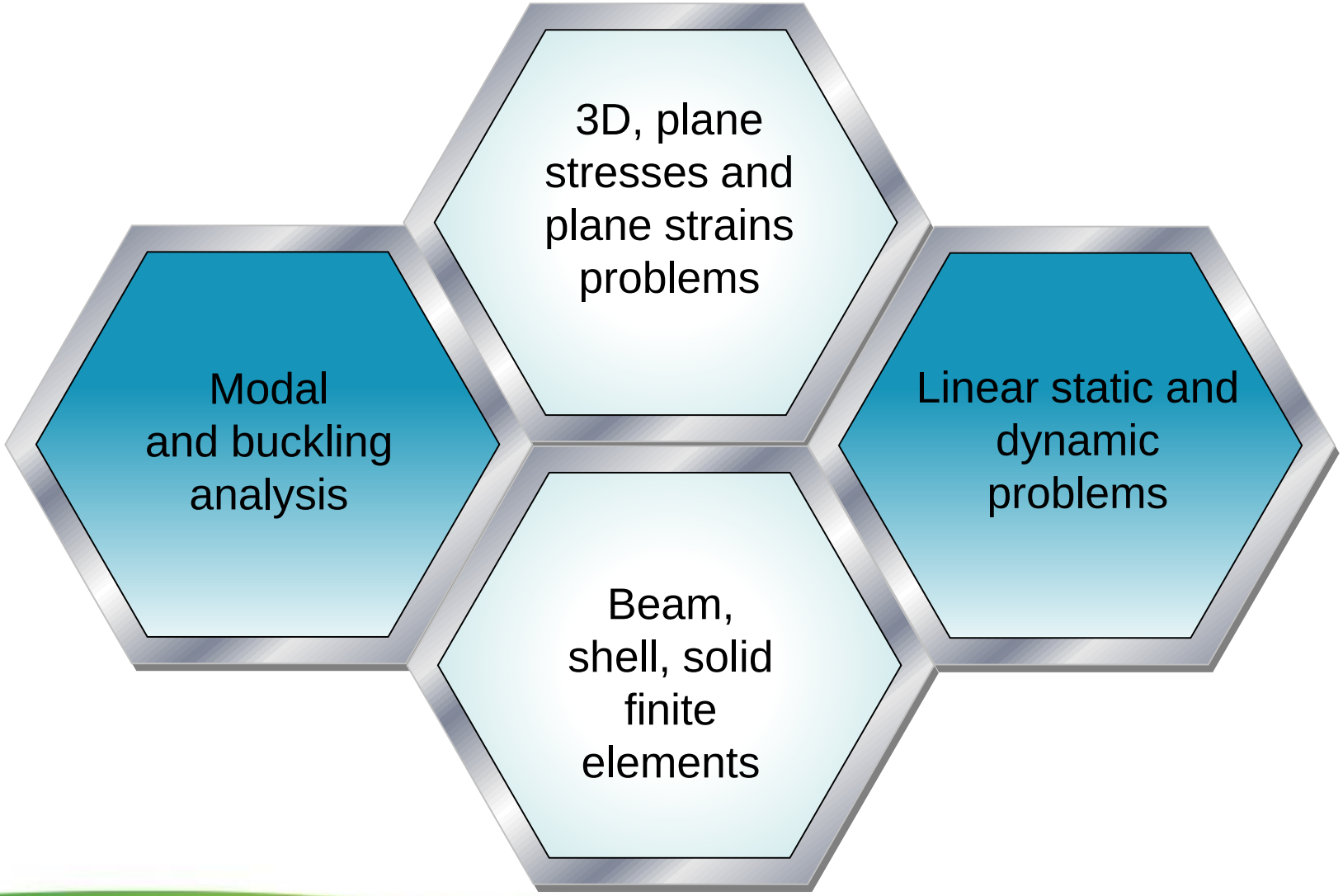
Analysis process



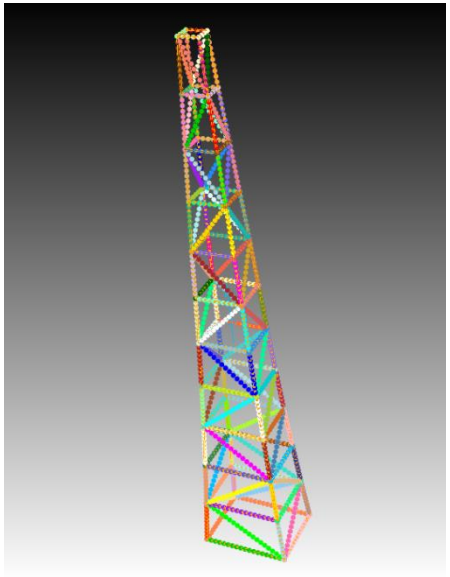
- **Preprocessor**
  - GUI
  - Console interface with an integrated automation mechanism
- **Calculation**
  - Console application with an external Python based API
- **Postprocessor** (Fidesys Viewer)
  - GUI
  - Console interface with an integrated automation mechanism

CAE Fidesys Standard

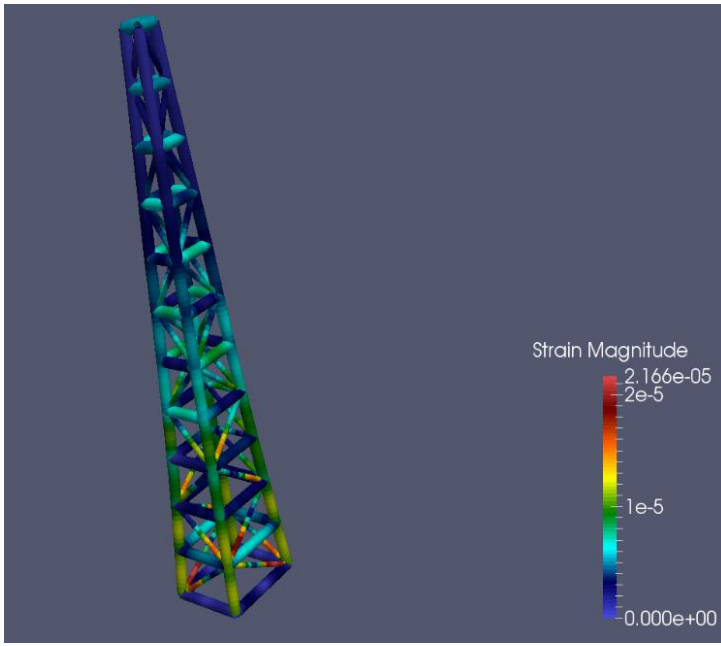
CAE Fidesys Standard

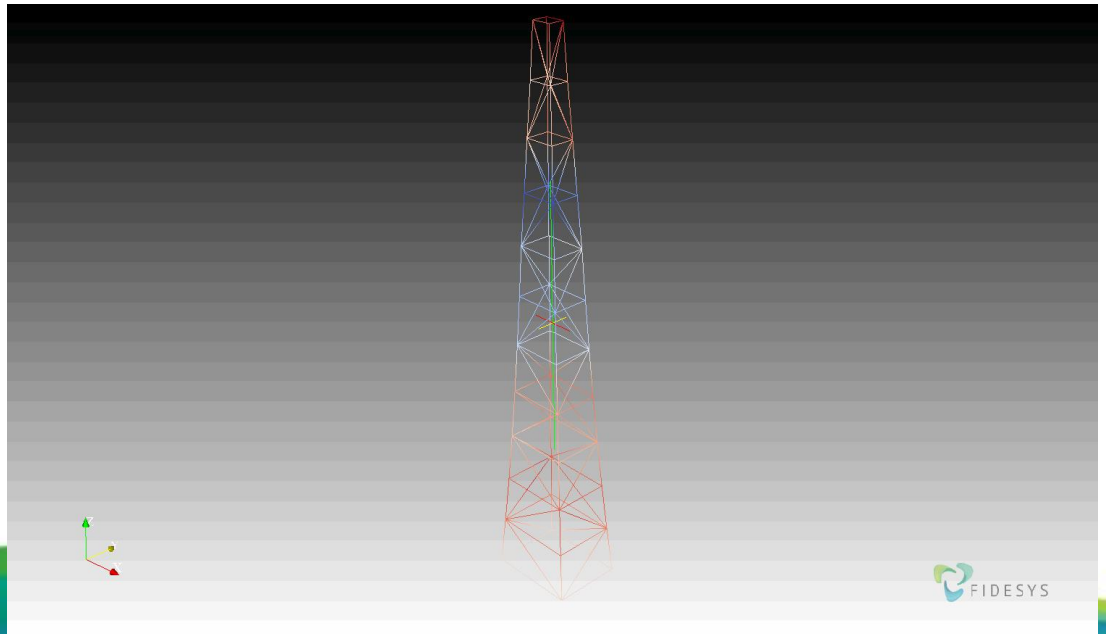


Truss structure analysis

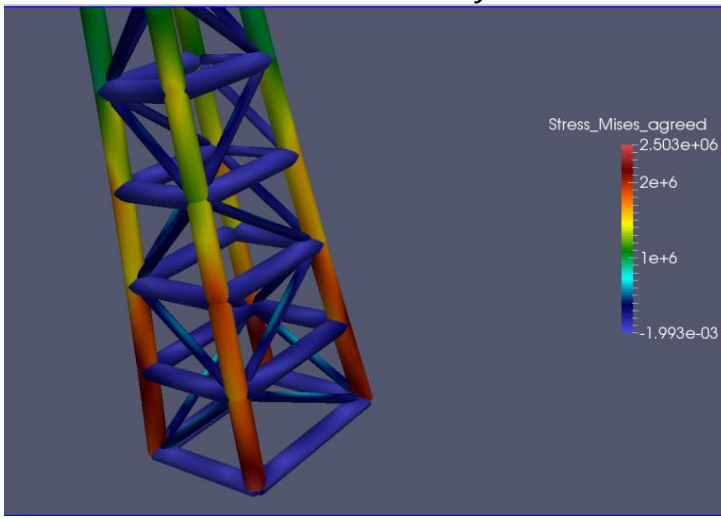


Strain distribution

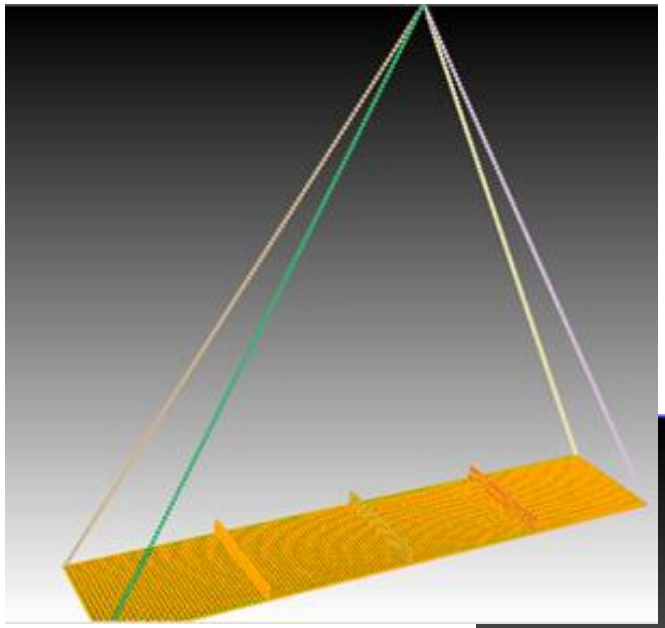


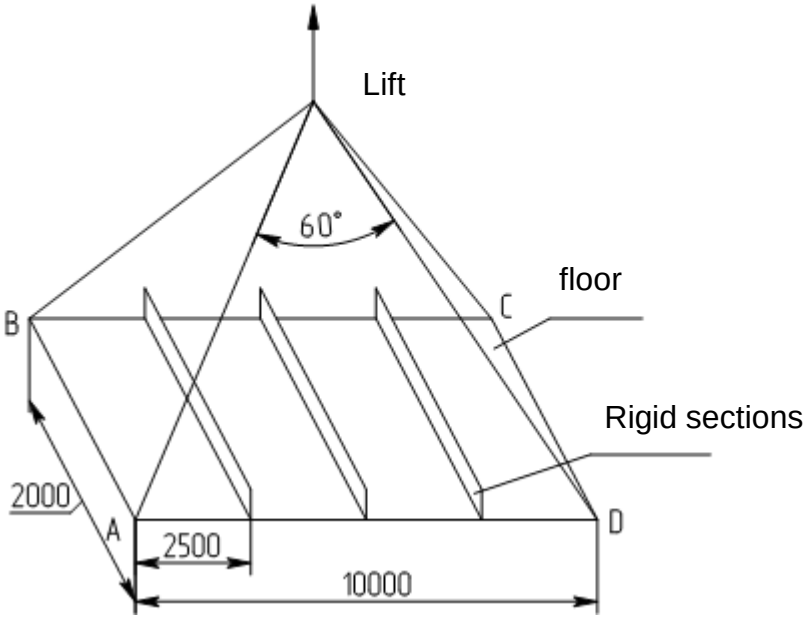


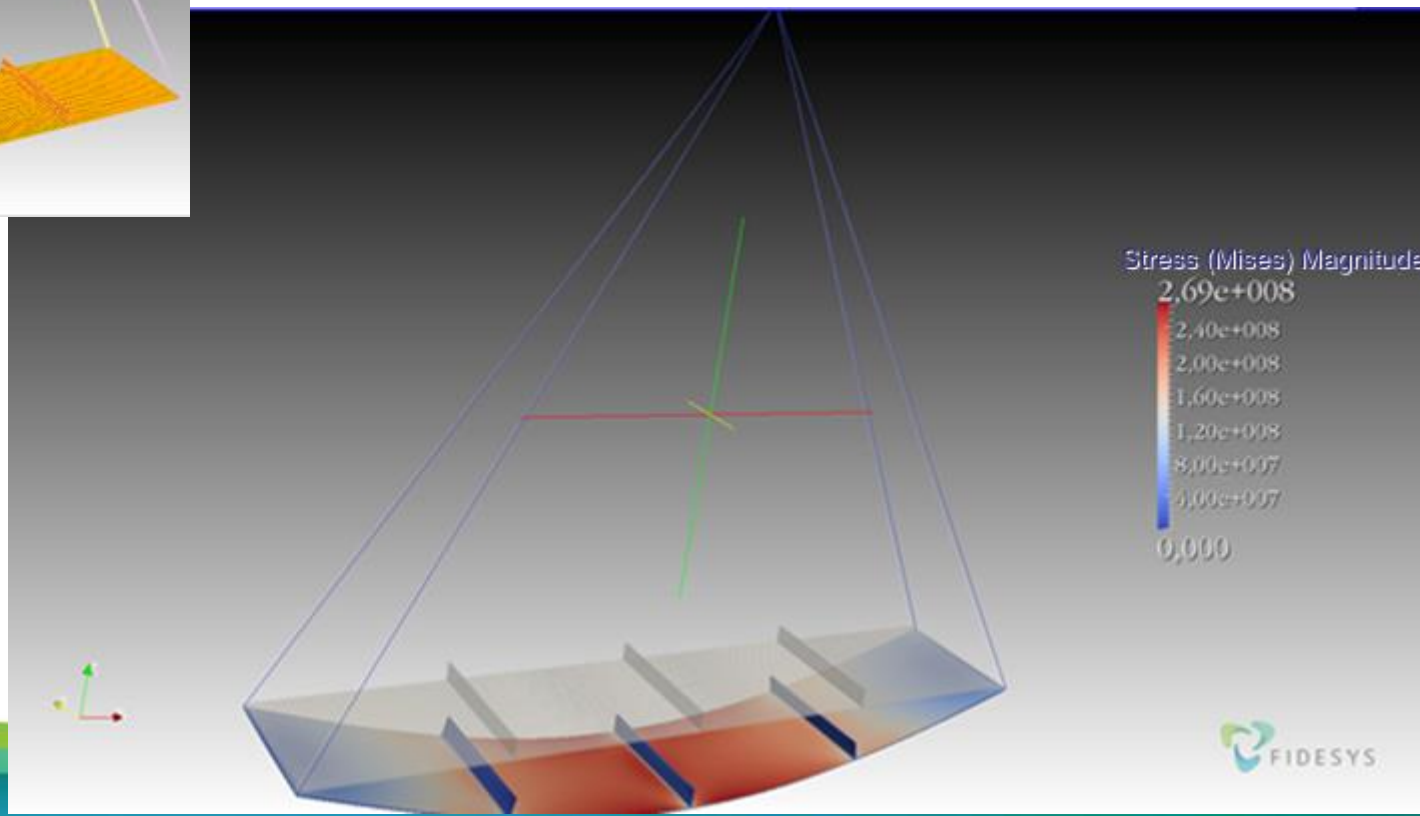
Stress intensity



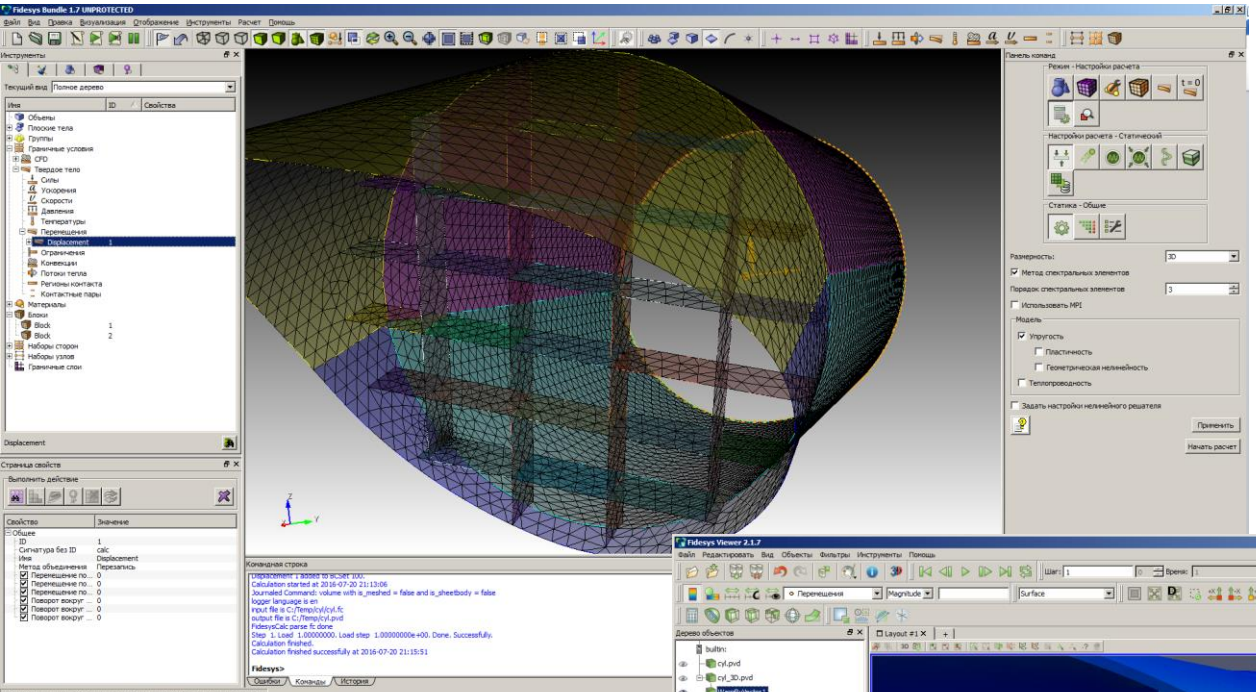
Section lifting

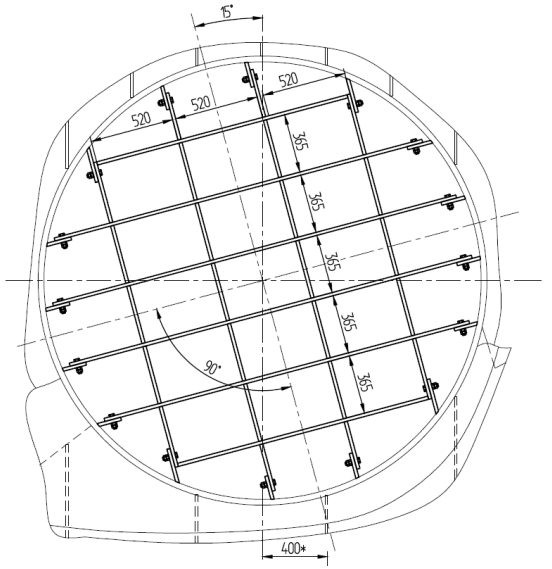


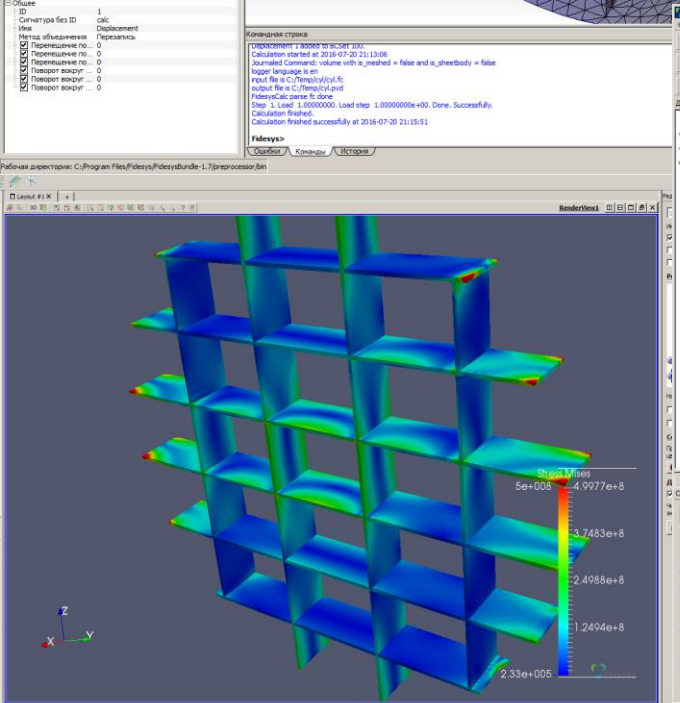


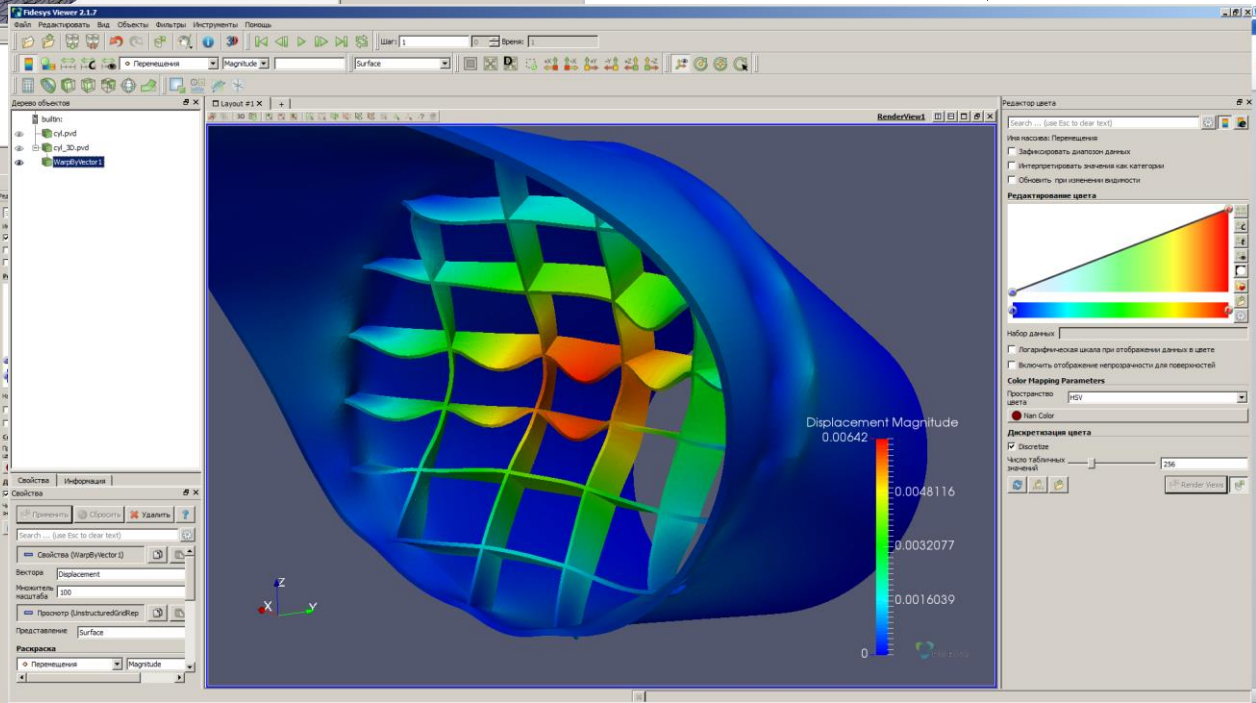


Analysis of the ice load on the thruster

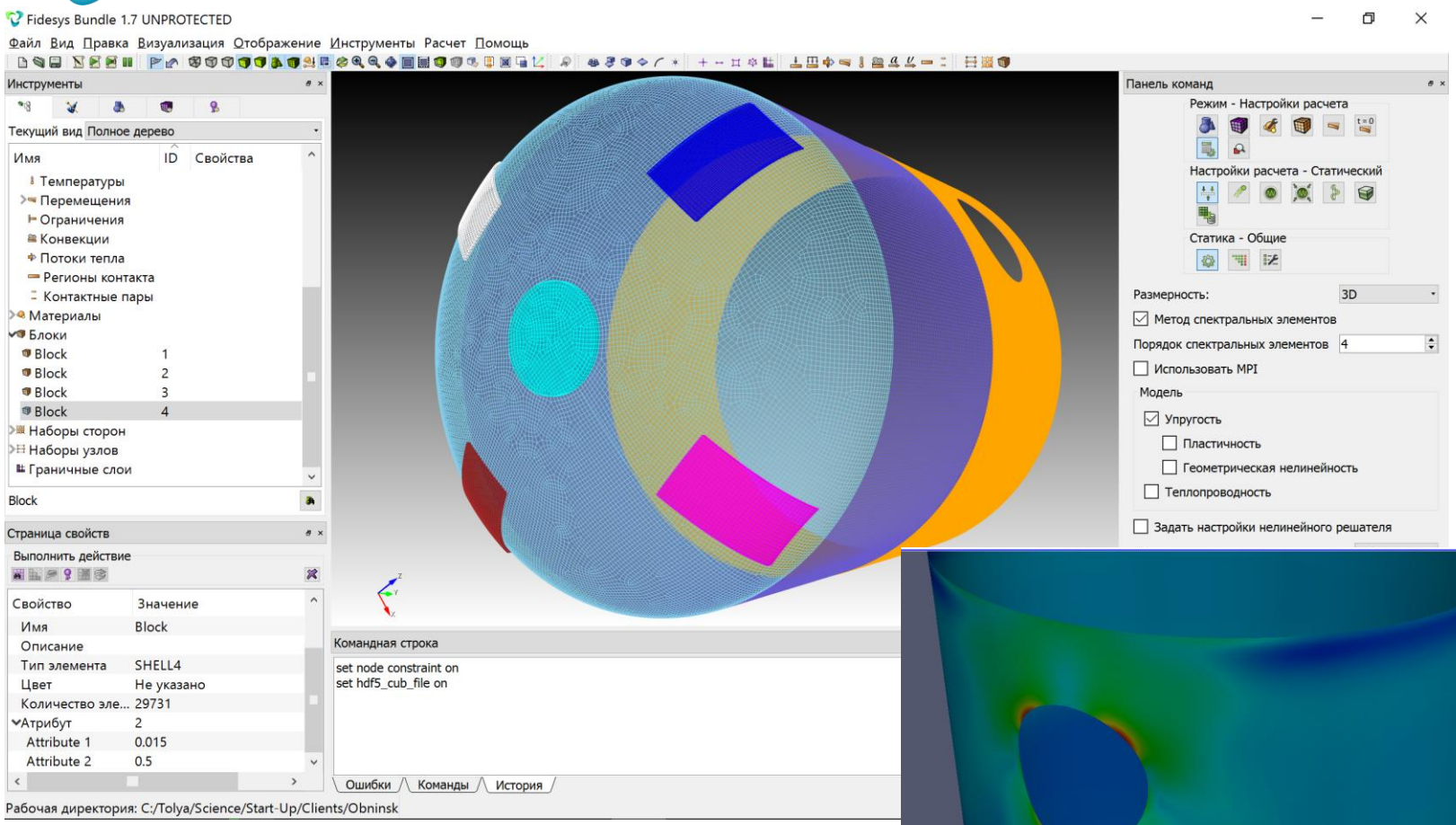


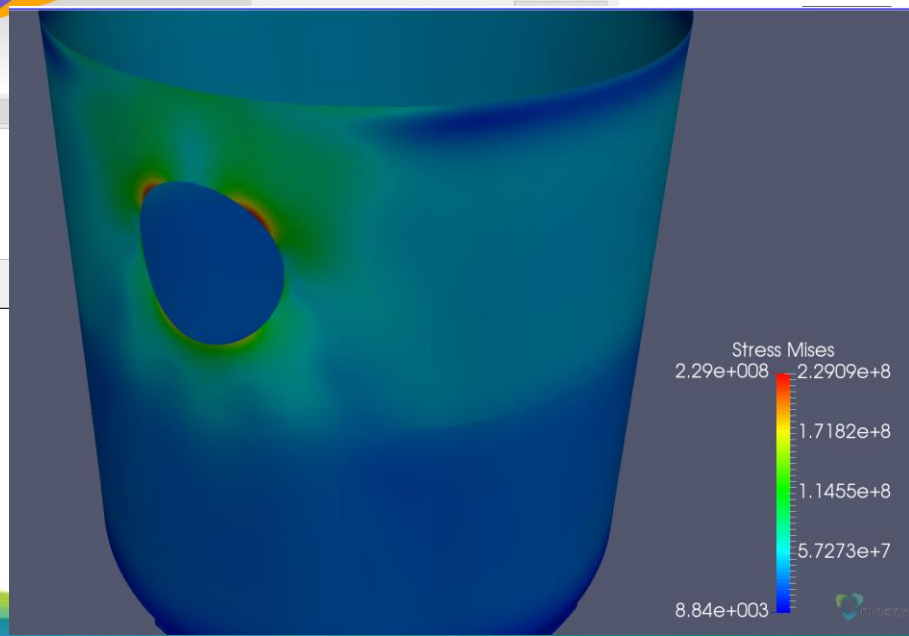




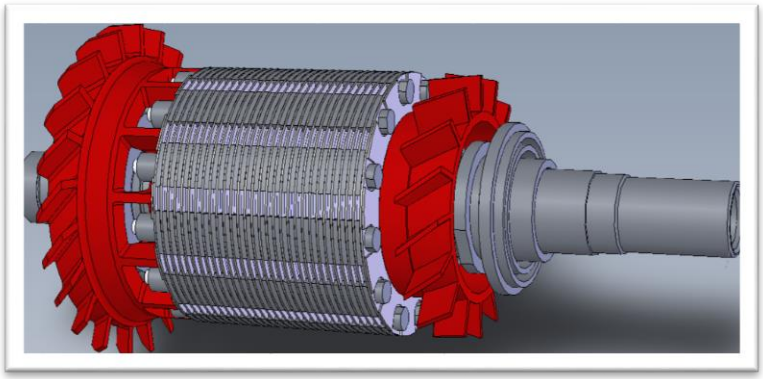


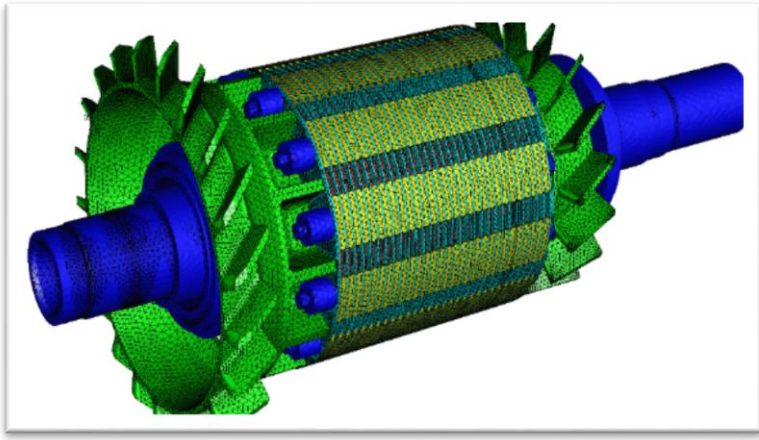
Analysis of the pressure tank

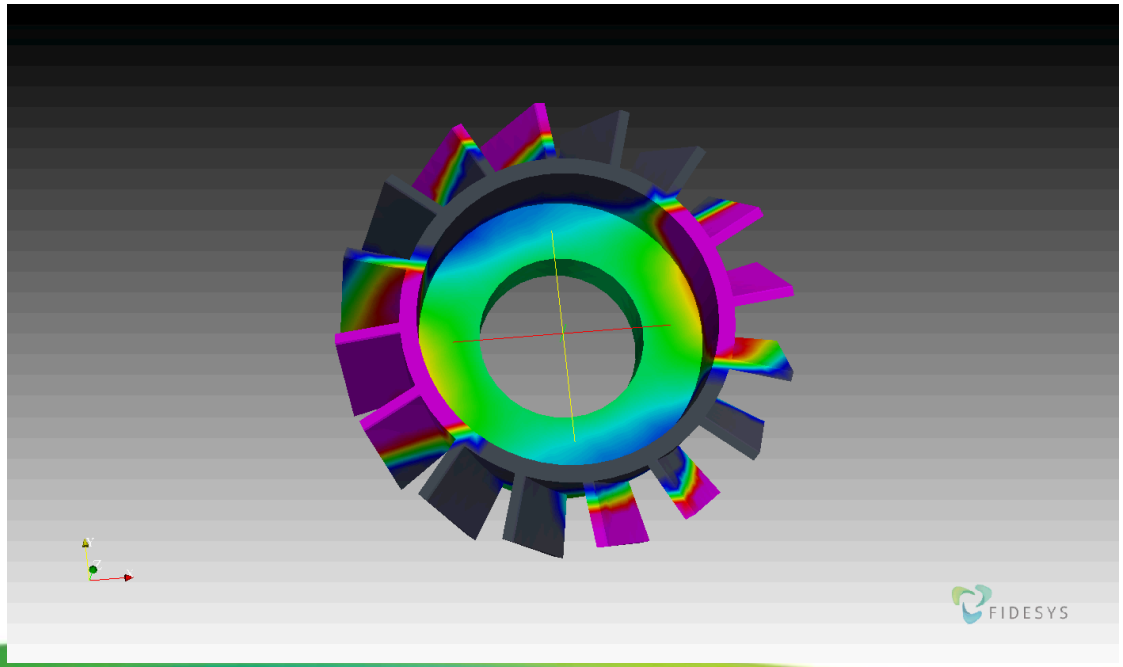


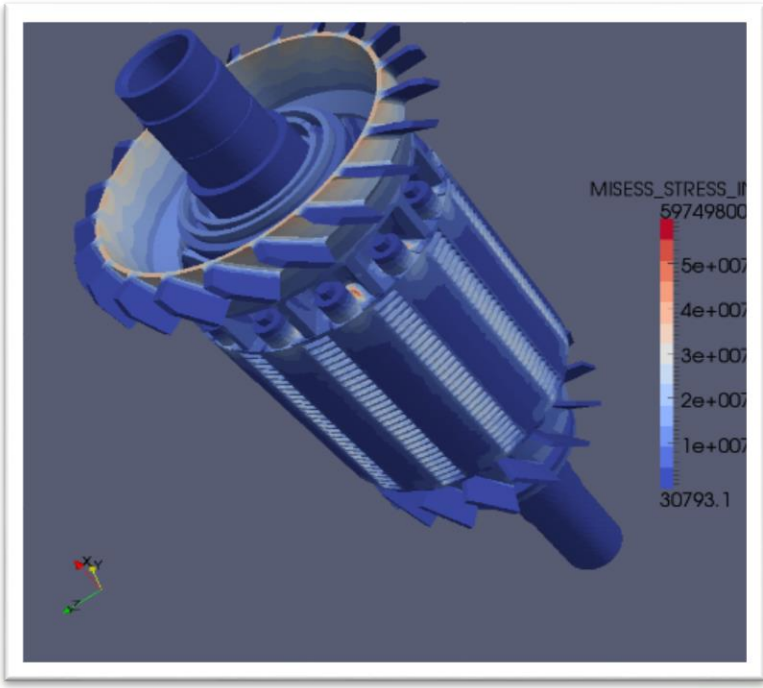


Analysis of the of the electrical rotor









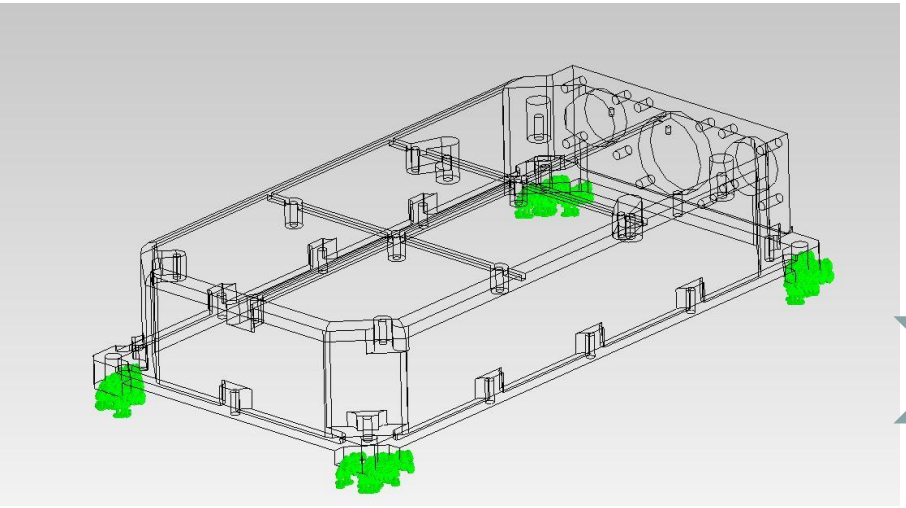
Compatibility and support

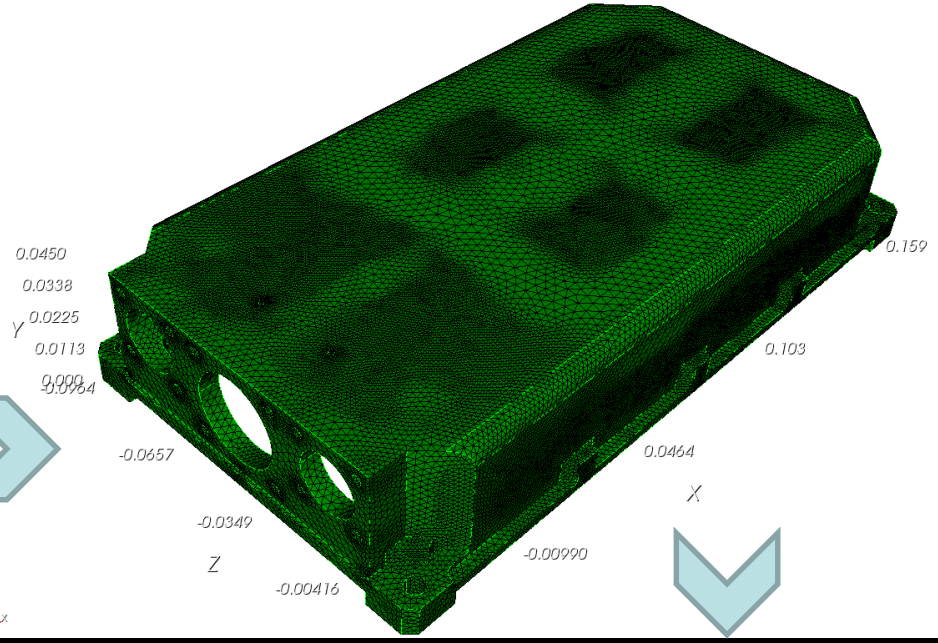
- Support for the most CAD/CAE formats

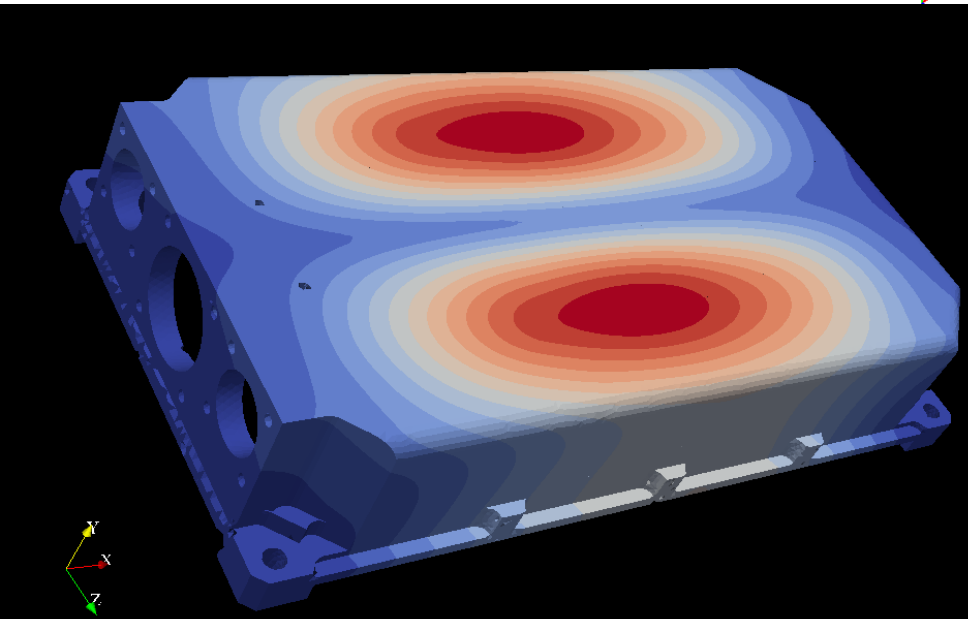
- Support of general data types

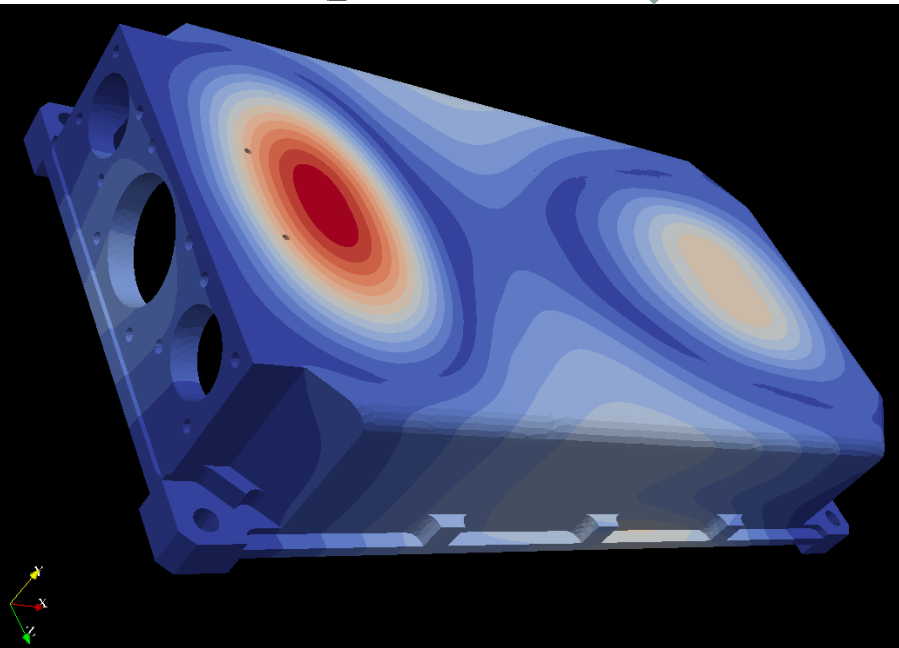
  - ACIS
  - IGES
  - STEP
  - AVS
  - Genesis/Exodus
  - Facets
  - STL
  - Ideas
  - Cubit

Modal analysis of the cover

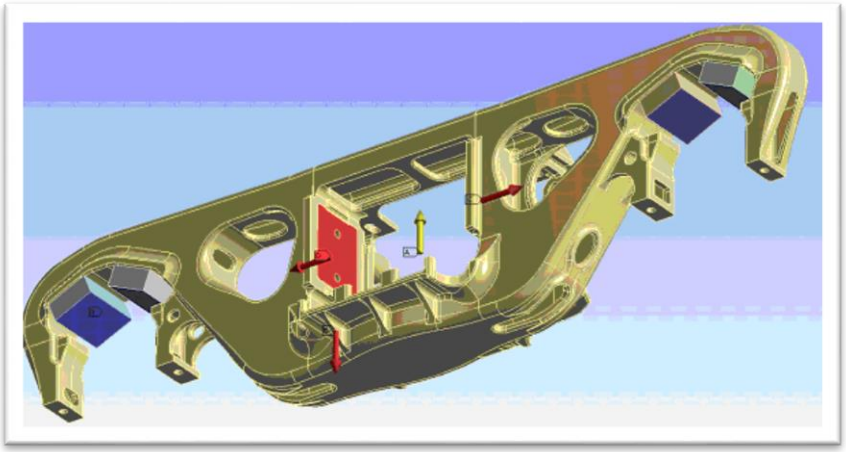


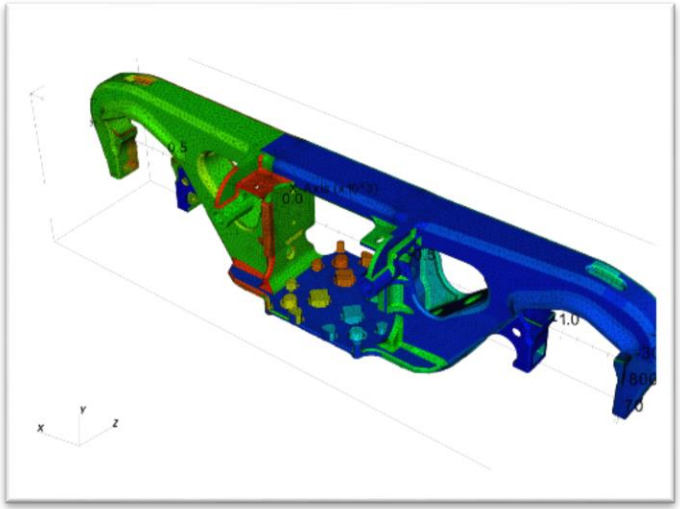


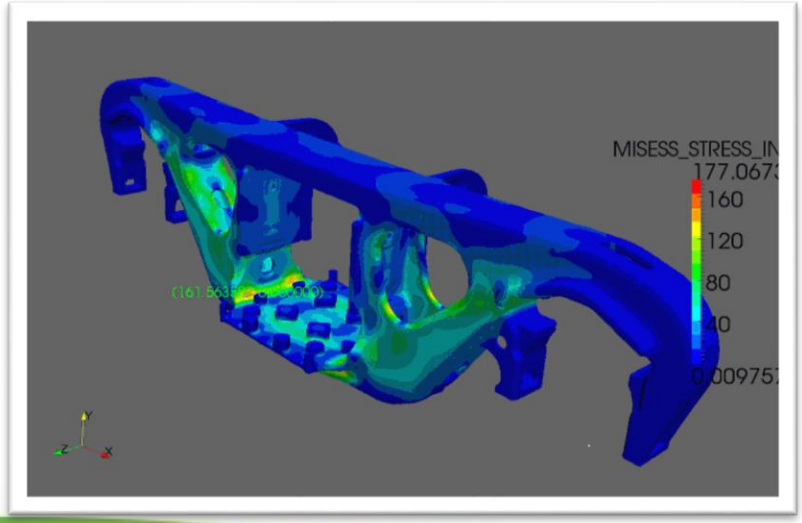




Railway wheel side analysis





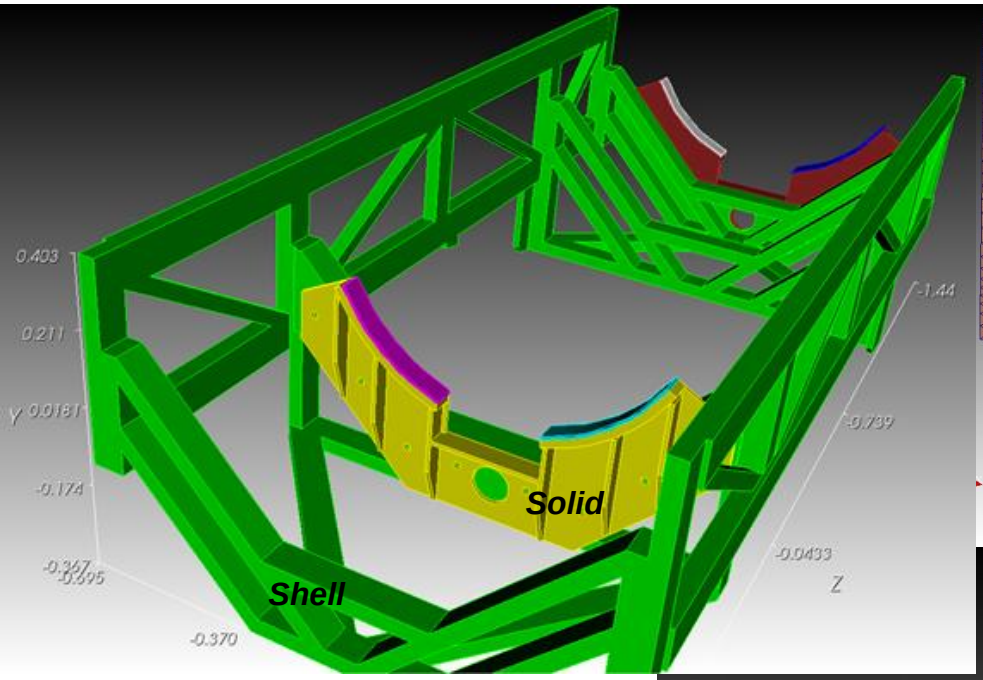


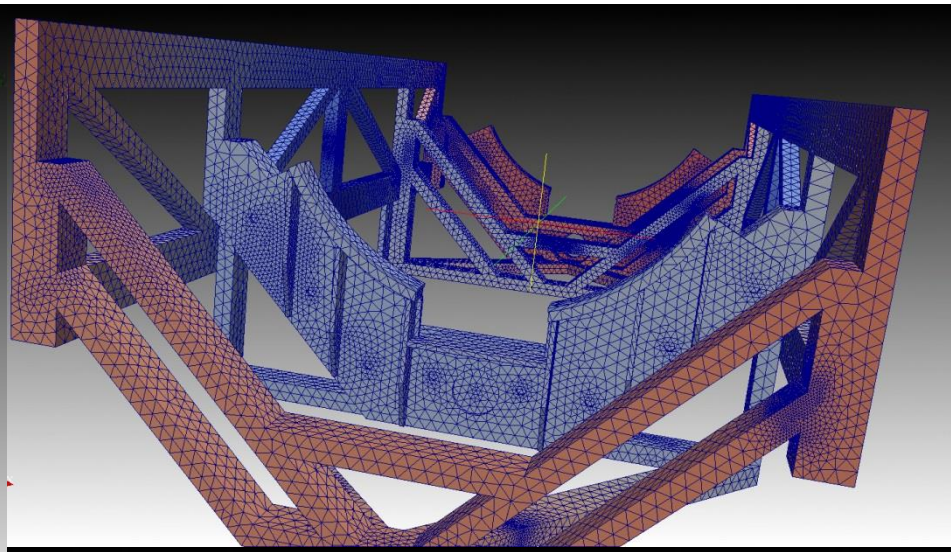
Data formats, finite elements

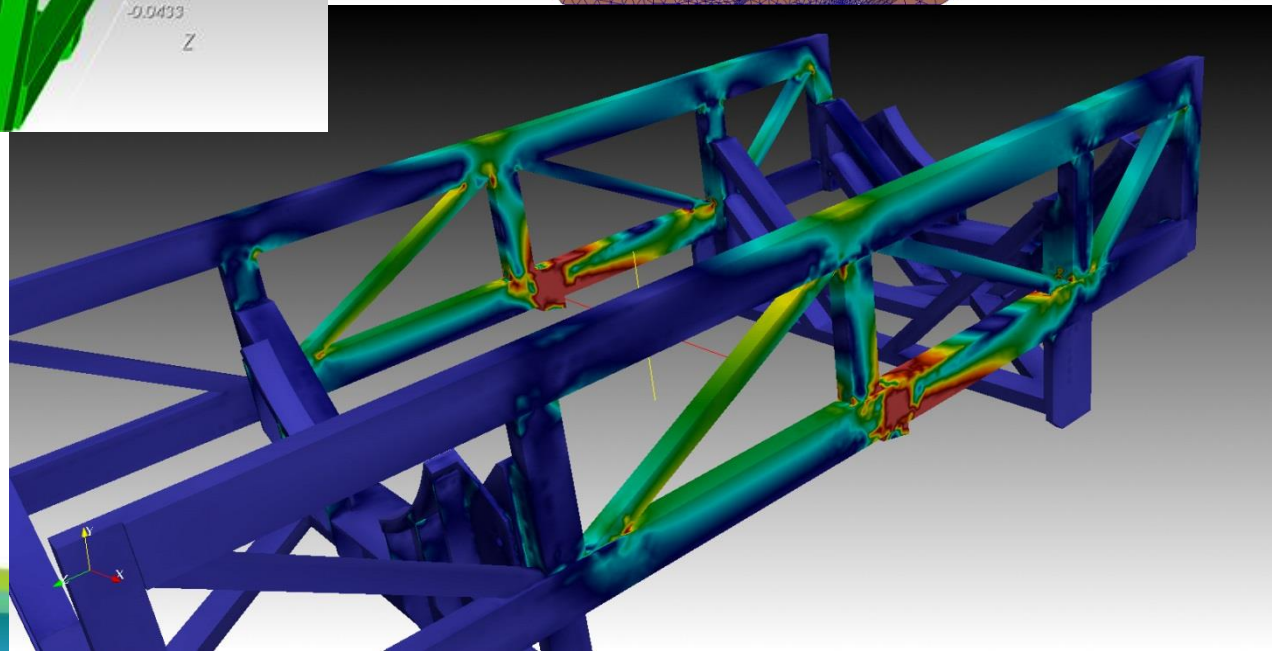
- ***ACIS (*.sat, *.sab);***
- ***IGES (*.igs, *.iges);***
- ***STEP (*.stp, *.step);***
- ***AVS (*.avs);***
- ***Genesis/Exodus (*.g, *.gen, *.e, *.exo);***
- ***Facets (*.fac);***
- ***GAMBIT Real Geometry (*.dbs)***
- ***Catia (*.CATPart, *.CATProduct, *.ncgm);***
- ***Parasolid (*.x_t, *.x_b)***
- ***SolidWorks (*.sldprt, *.sldasm)***
- ***Pro/E (*.prt, *.asm)***
- ***STL Files (*.stl);***
- ***Patran (*.pat, *.neu, *.out);***
- ***Ideas (*.unv);***
- ***Abaqus (*.inp);***
- ***Fluent (*.msh);***
- ***Nastran (*.bdf);***
- ***LS-Dyna (*.k*)***
- ***Ansys (*.cdb)***

- BEAM2 (beam)
- BEAM3 (curved beam)
- QUAD4 (4-noded quadrilateral)
- QUAD8 (8-noded quadrilateral)
- QUAD9 (9-noded quadrilateral)
- SHELL4 (4-noded quadrilateral)
- SHELL8 (8-noded quadrilateral)
- SHELL9 (9-noded quadrilateral)
- TRI3 (3-noded triangle)
- TRI6 (6-noded triangle)
- TRISHELL3 (3-noded triangle)
- TRISHELL6 (6-noded triangle)
- HEX8 (8-noded hexahedron)
- HEX20 (20-noded hexahedron)
- HEX27 (27-noded hexahedron)
- TETRA4 (4-noded tetrahedron)
- TETRA10 (10-noded tetrahedron)
- WEDGE6 (6-noded wedge)
- WEDGE15 (15-noded wedge)
- PYRAMID5 (5-noded pyramid)
- PYRAMID13 (13-noded pyramid)
- SPRING
- LUMPMASS
- CONSTRAINT
- **SEM *N (spectral element of Nth order)***

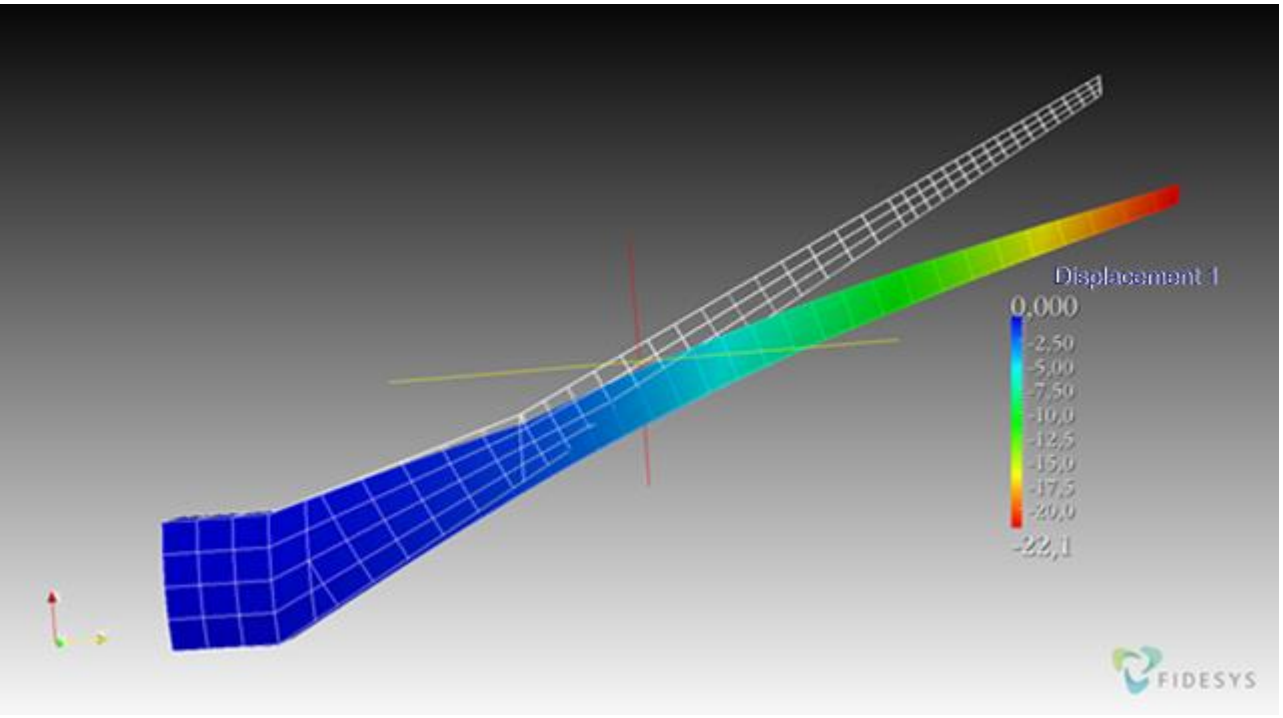
Solid-Shell structural analysis

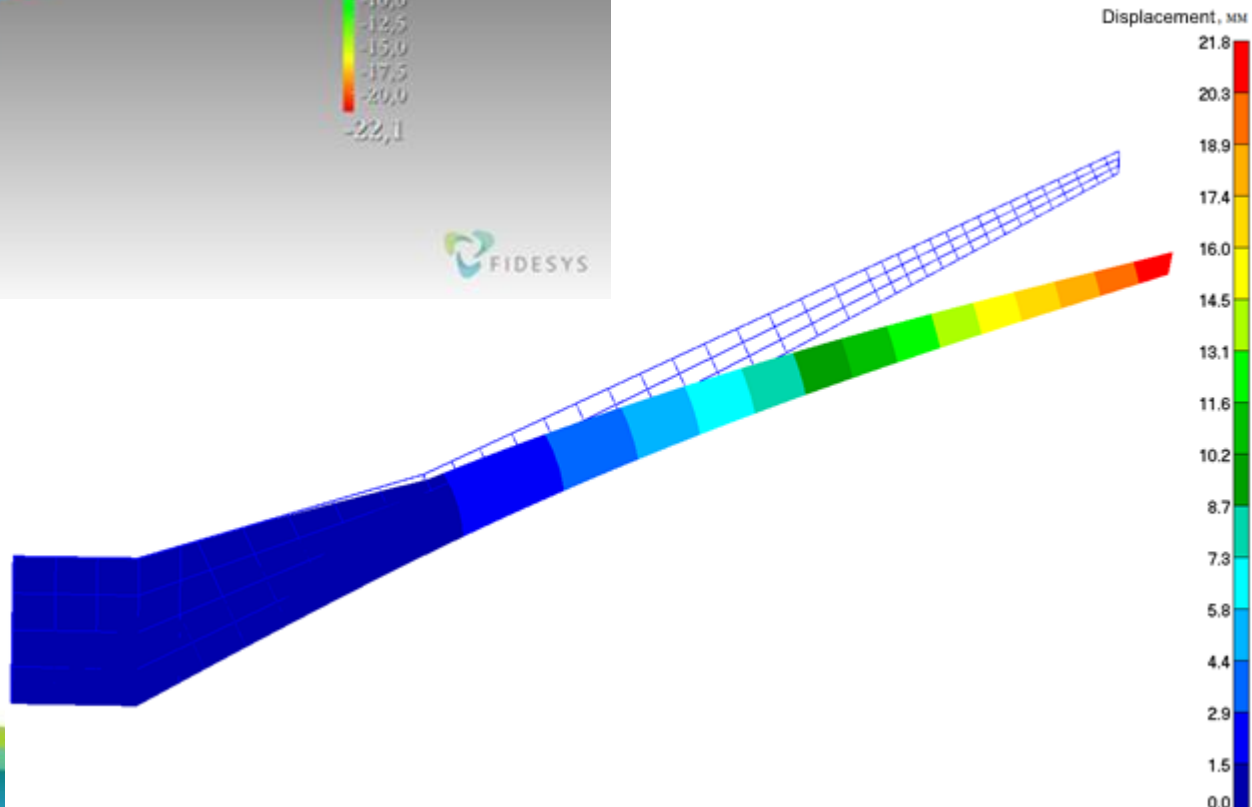




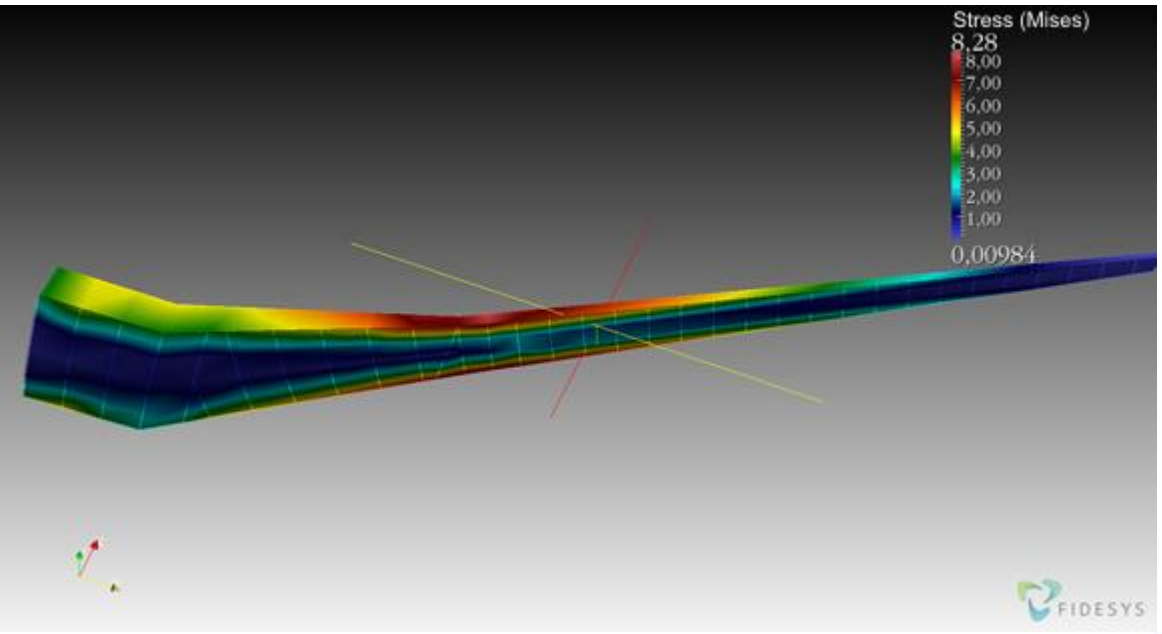


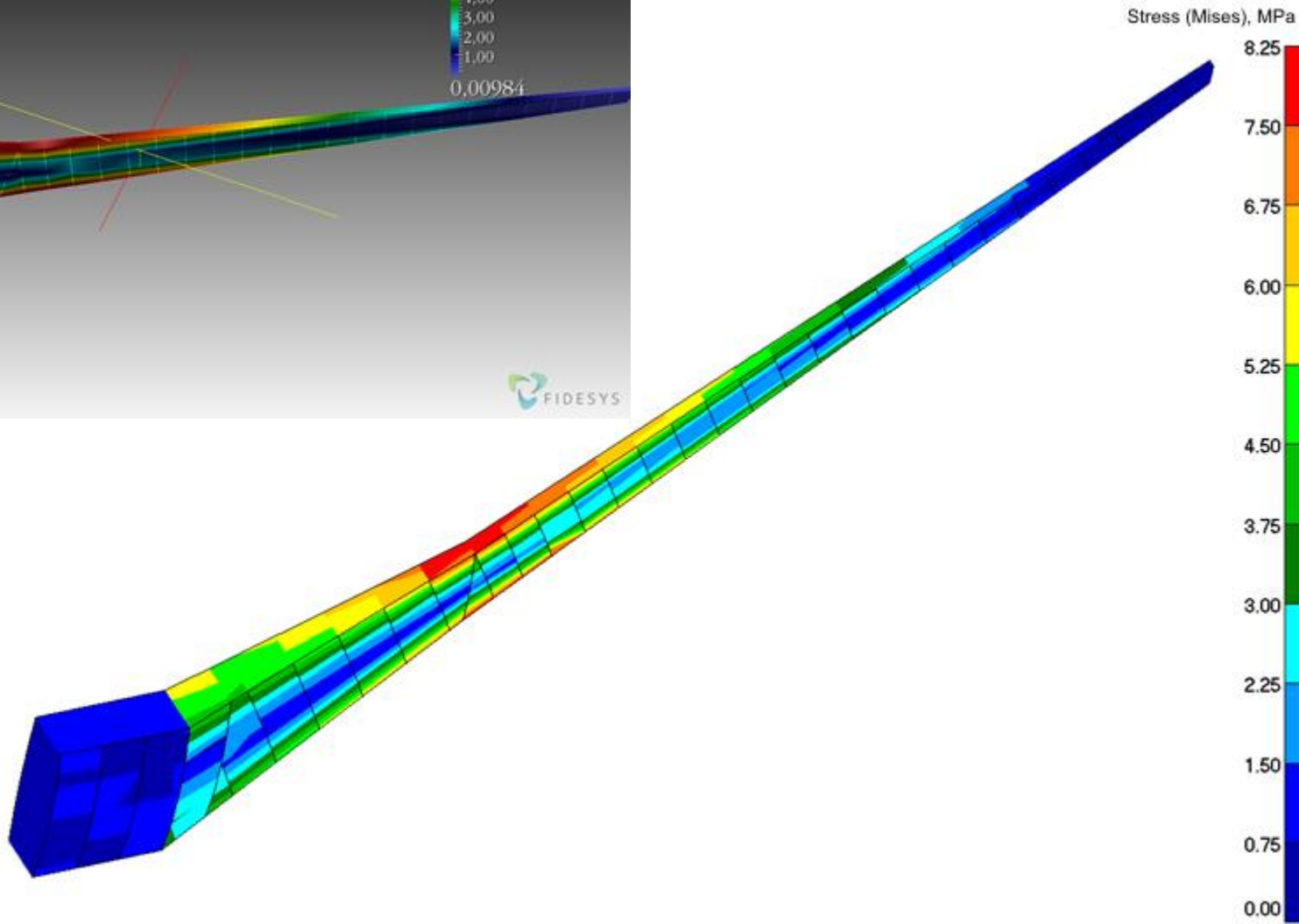
Analysis of the wing box



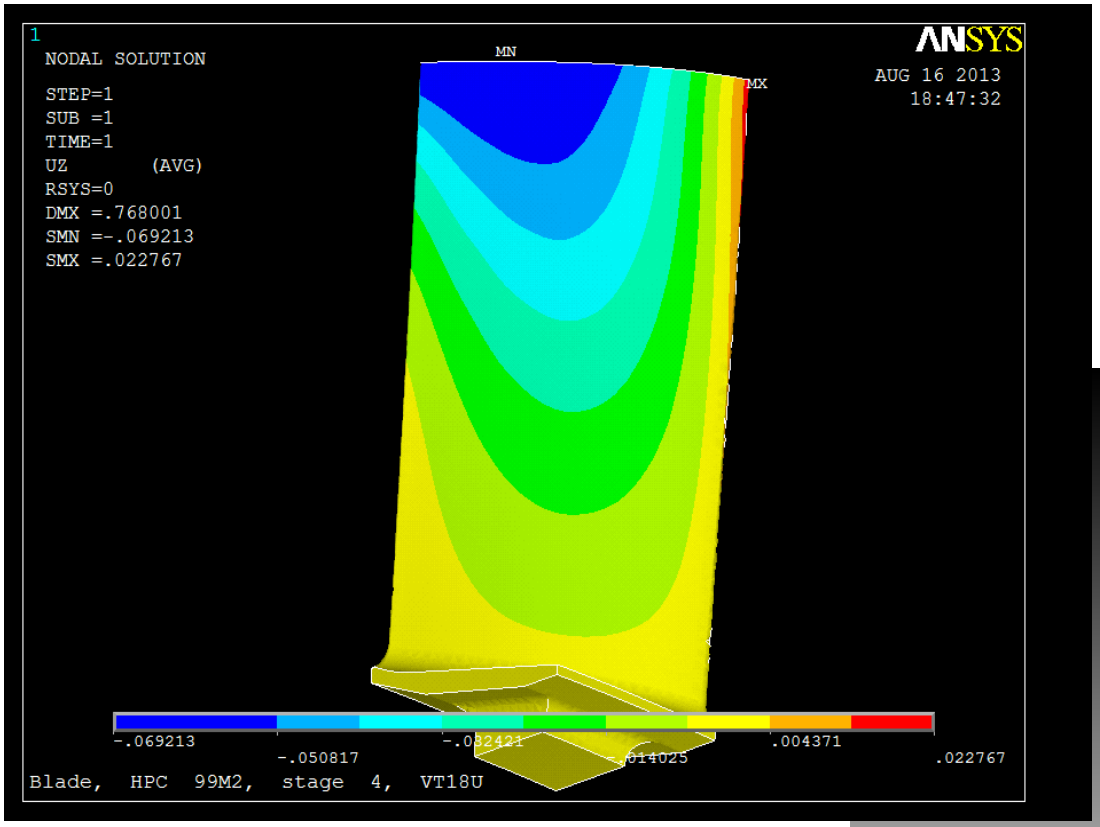


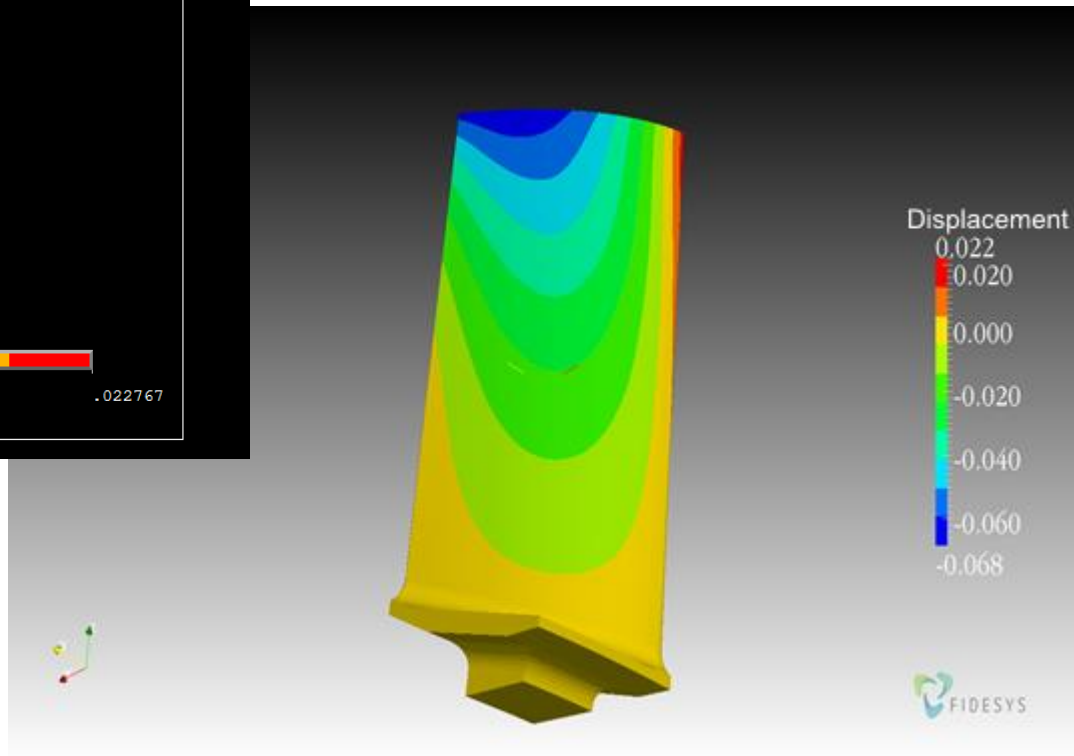
Analysis of the wing box



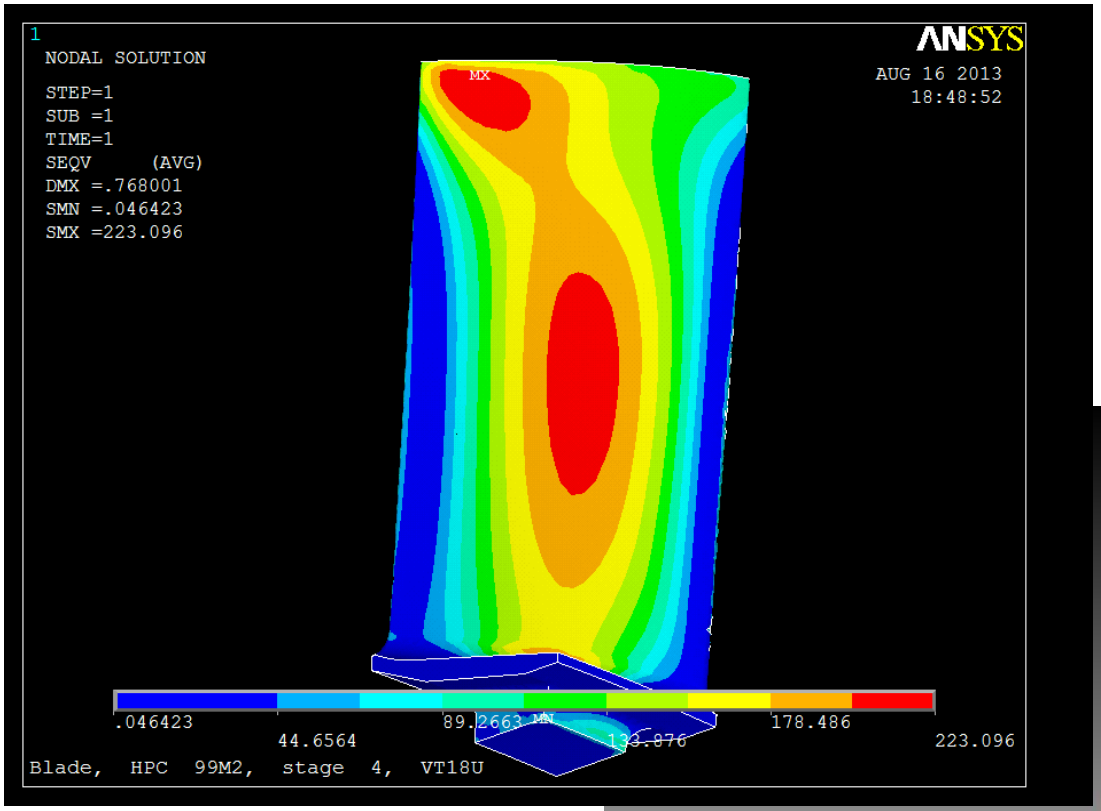


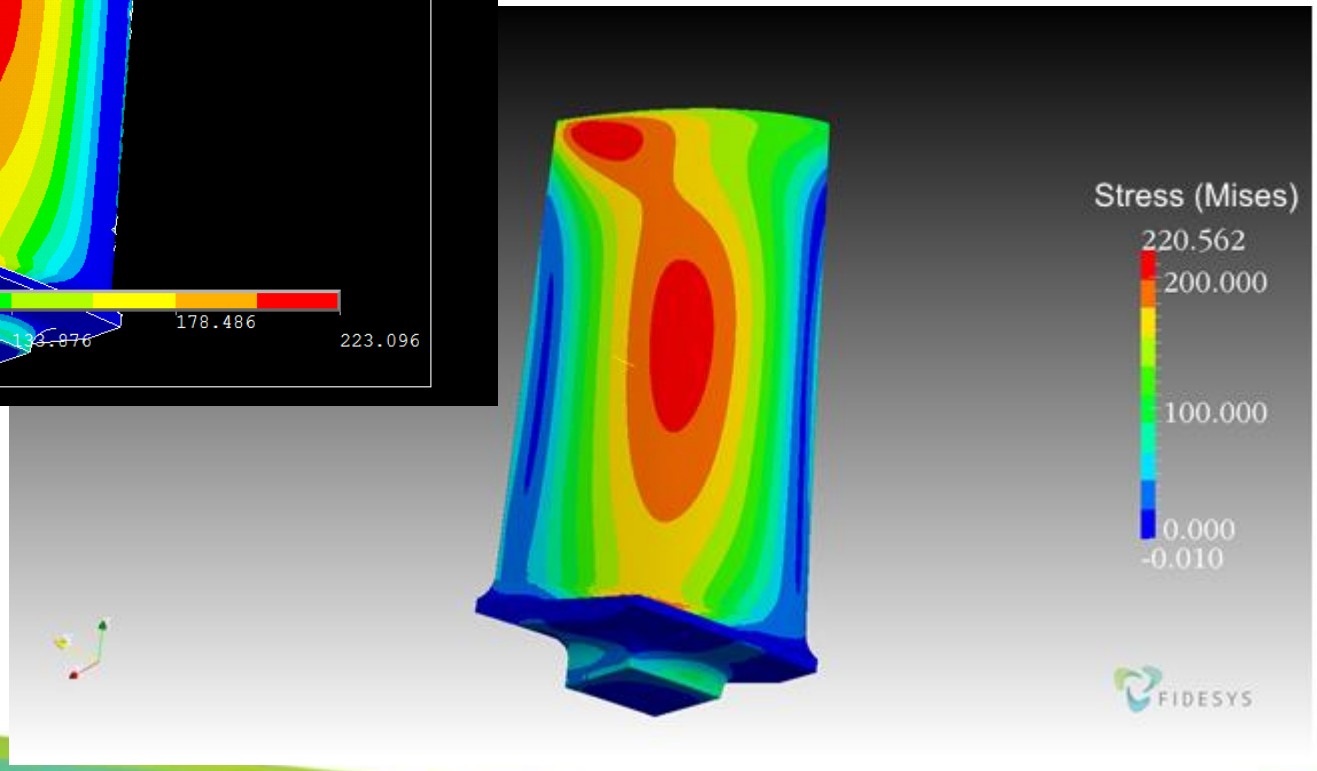
Analysis of engine blades





Analysis of engine blades

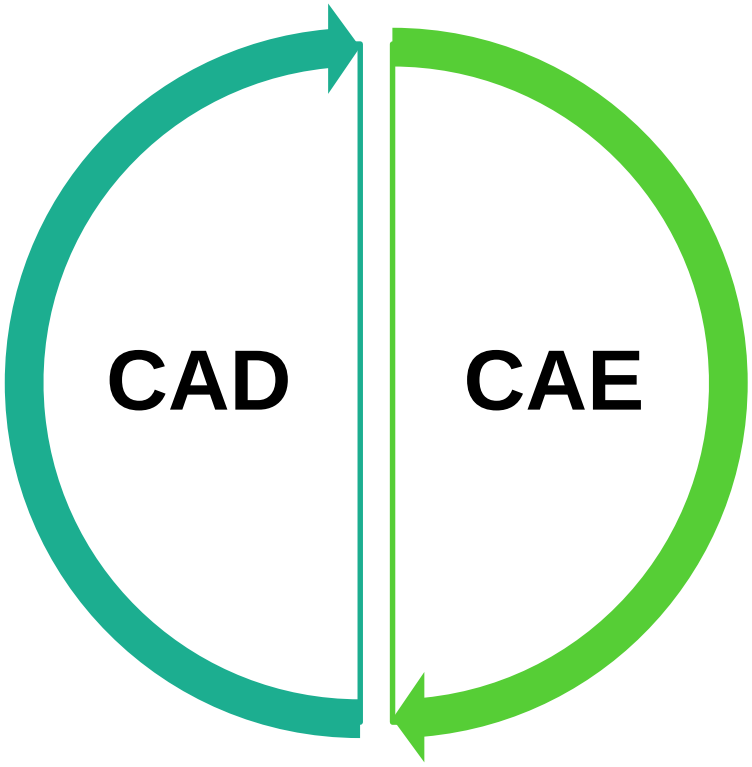




CAE Fidesys features

Integration with CAD software

- A push-button export of CAD-model into CAE Fidesys
- Automatic meshing
- Simulation and analysis at the early design stages
- Evaluation of structural performance for design alternatives



BRICSCAD

NANOCAD
УМНОЕ ПРОЕКТИРОВАНИЕ

КОМПАС-3D

INVENTOR

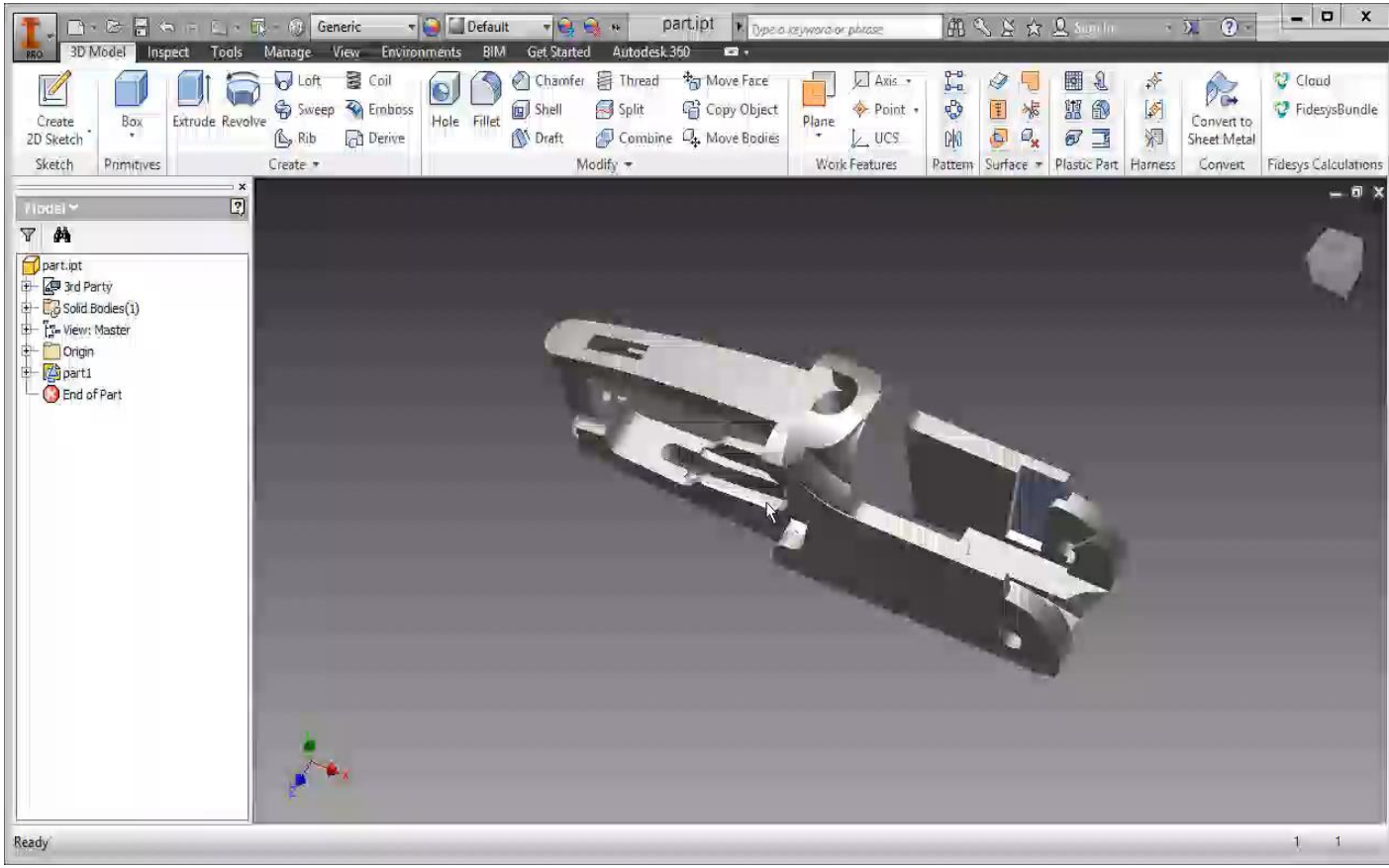
DATADVANCE

CAE Fidesys features

AUTODESK. Authorized Developer

Integration with Autodesk Inventor

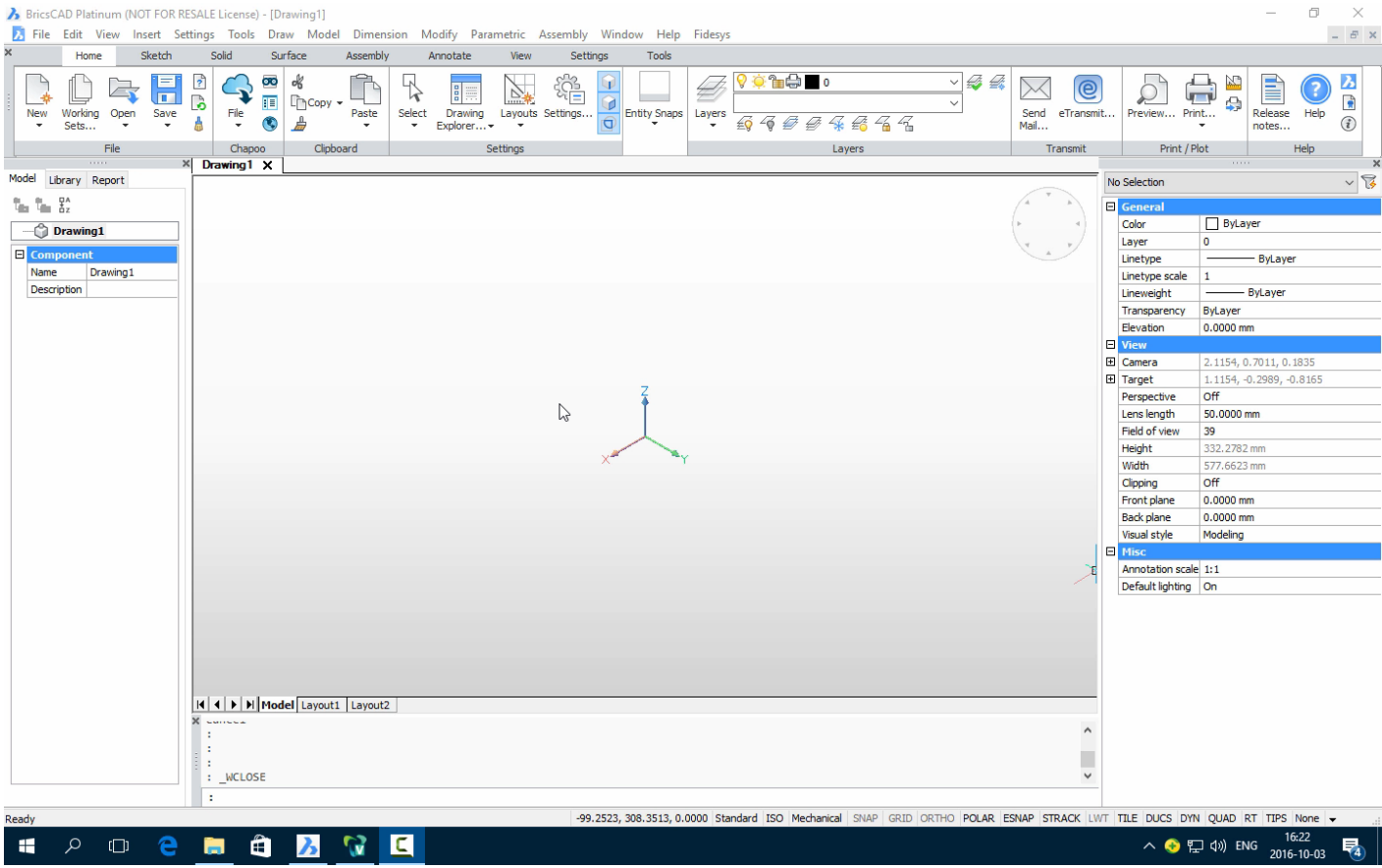
- One-button export of CAD model from Inventor into CAE Fidesys;
- No need to re-build FEA model if the CAD-model parameters were changed in Inventor;
- Export to Sim4Design for analysis in the cloud



CAE Fidesys features

Integration with BricsCAD

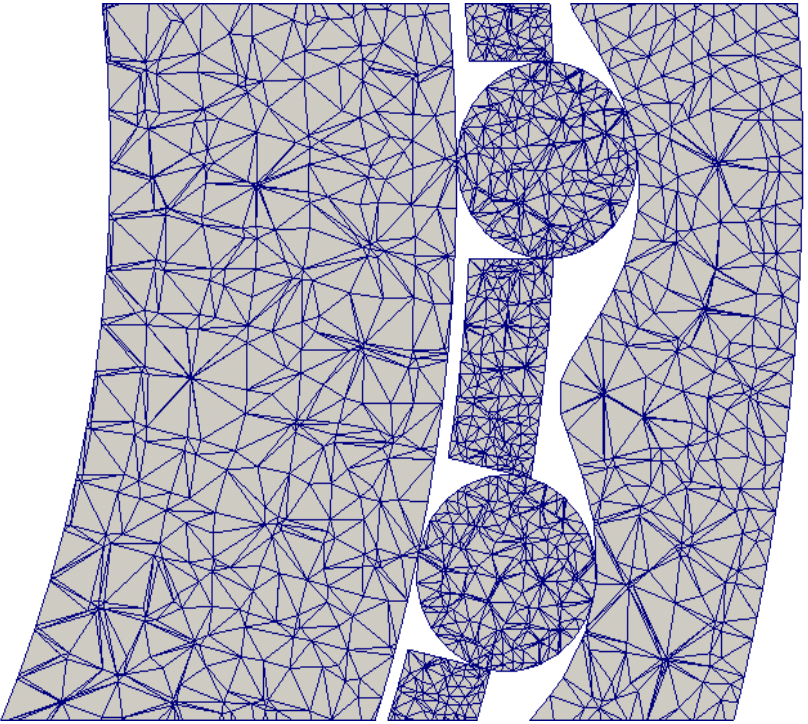
- One-button export of CAD model from BricsCAD into CAE Fidesys;
- Automatization of engineering analysis of the model in BricsCAD;
- A possibility to perform model optimization and tuning

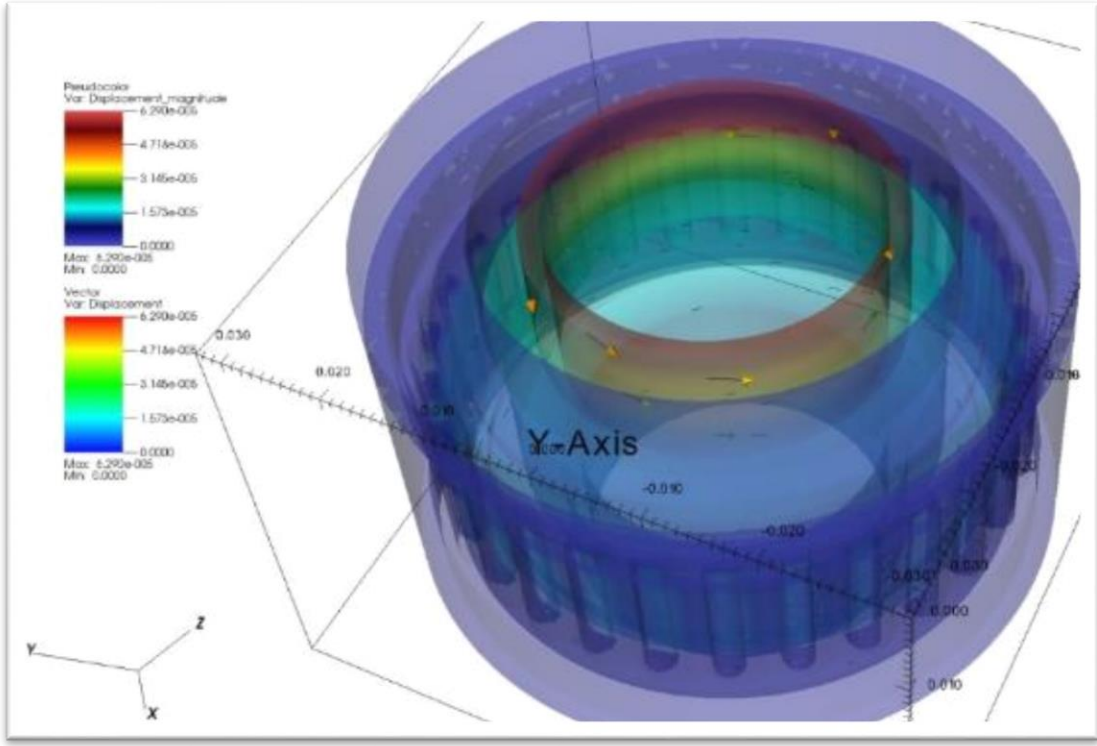


CAE Fidesys Professional

Contact mechanics of elastic bodies

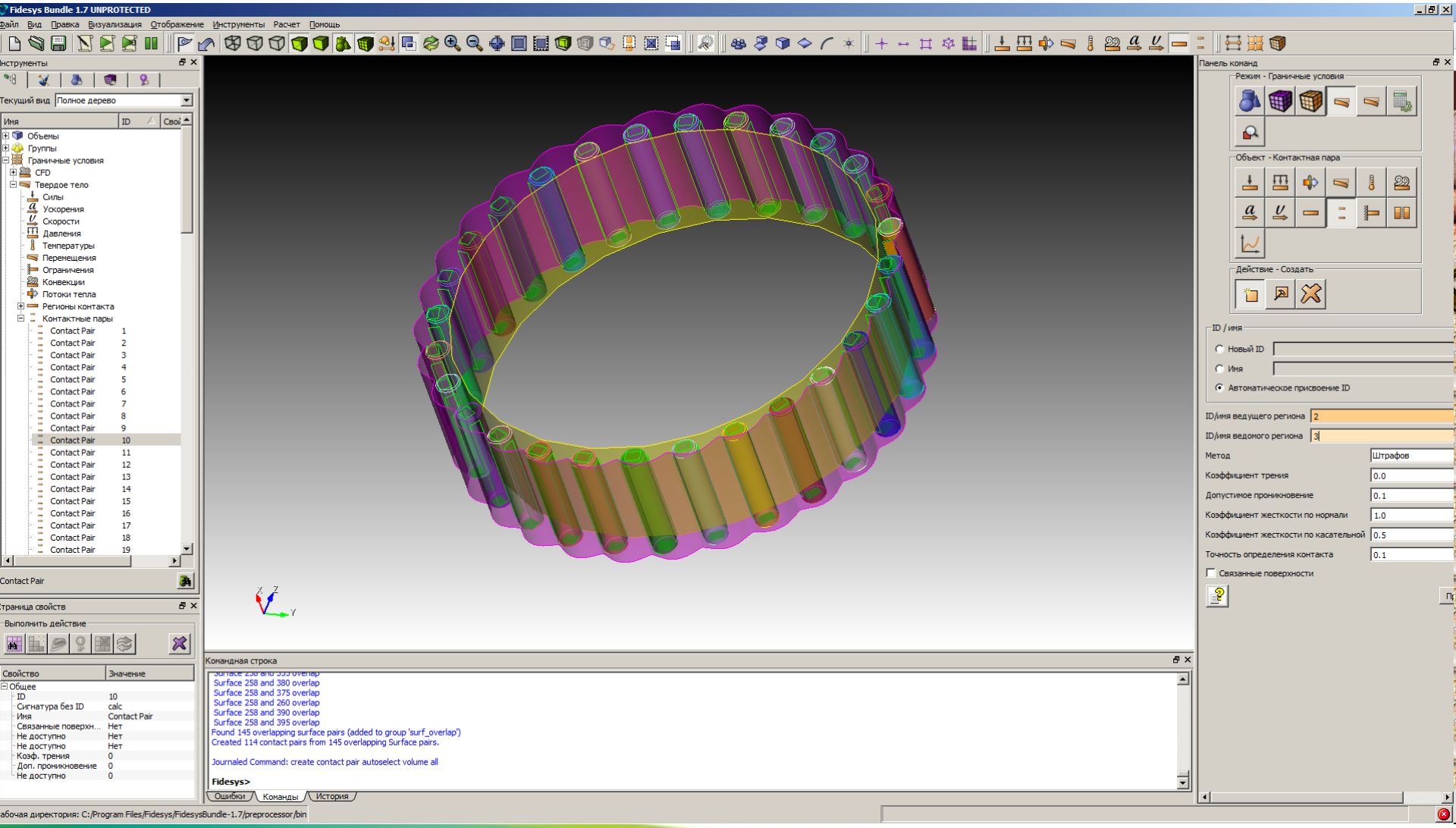
- Non-linear contact interactions between solids
- Static or dynamic (with/without friction) contact problems
- Bonded contact
- Internal penalty method, Lagrange multipliers method



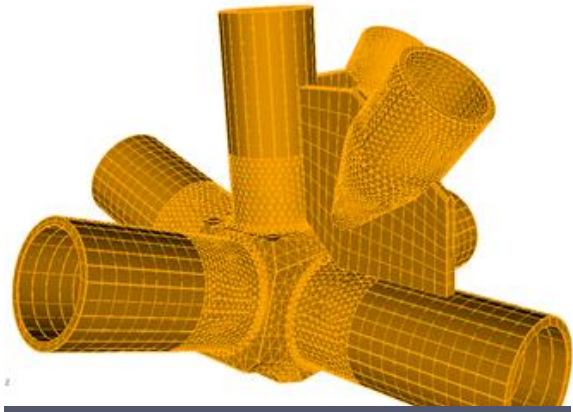


Contact interactions between rollers in the mechanical reductor

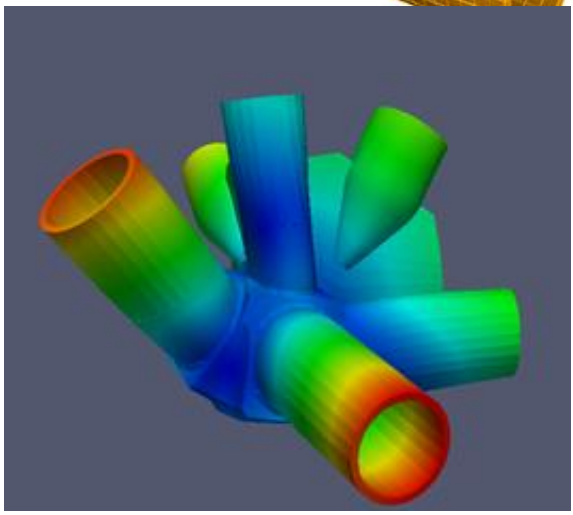
Automatic contact pairs detection



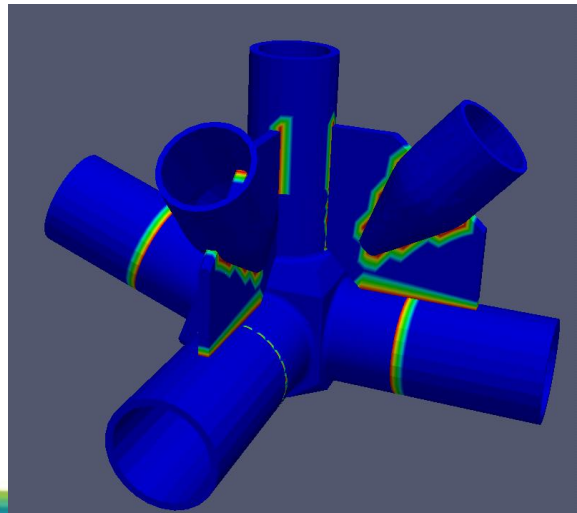
Bonded contact



- Support for nonconformal meshes with gaps/overlaps between contacting bodies => no need for simplification/healing of an input CAD assembly

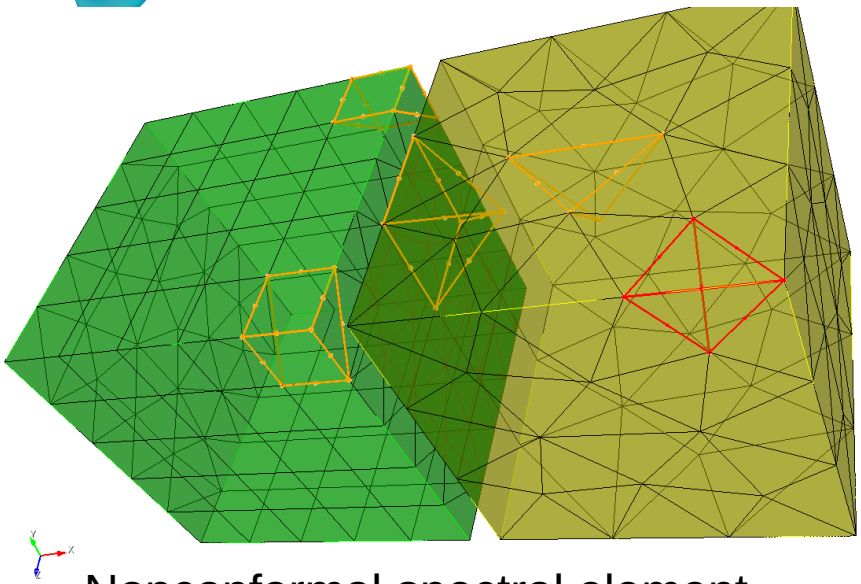


- Continuous displacements and stresses even in case of gaps/overlaps!

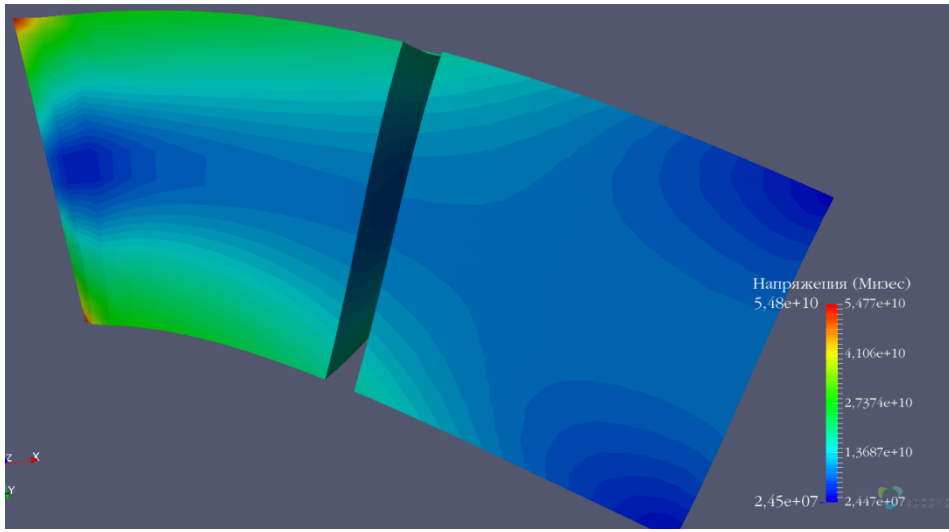


- Automatic contact zones detection based on a specified geometrical parameter

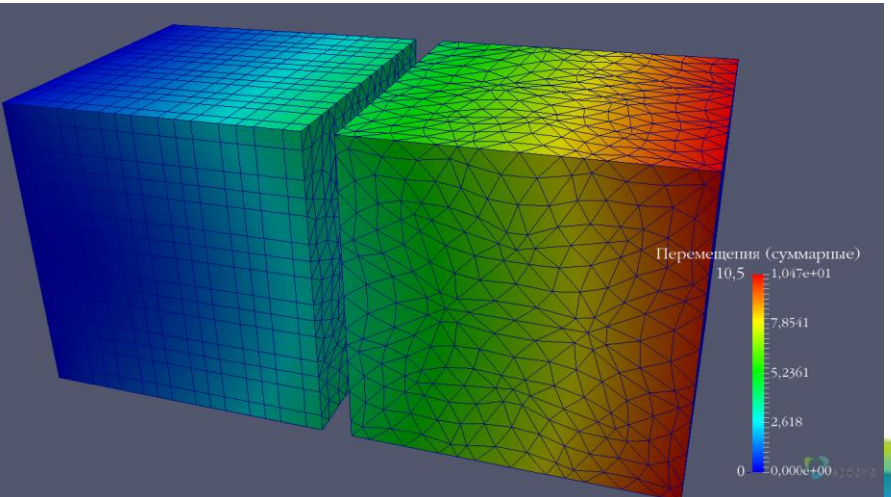
Bonded contact: example

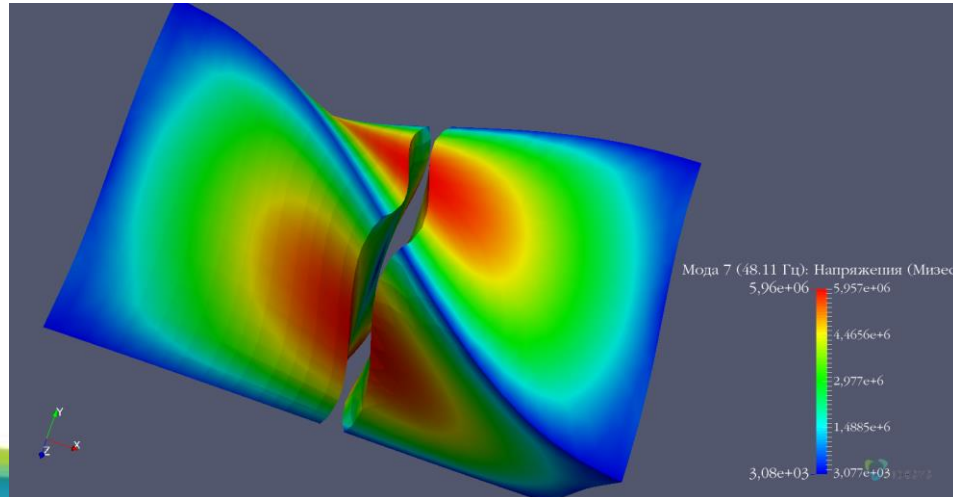


Nonconformal spectral element mesh of different orders and mixed types of elements



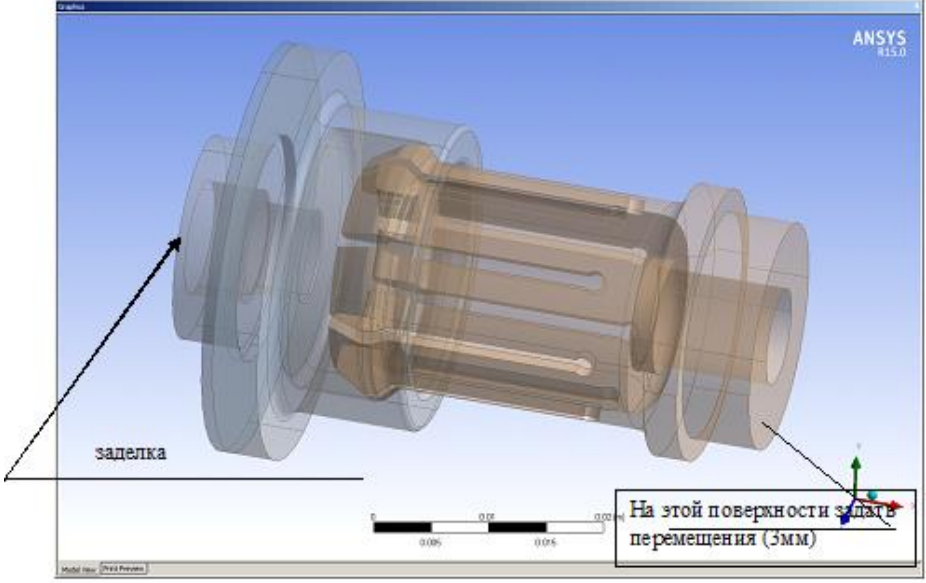
Continuous solution even in the presence of the gap between solids

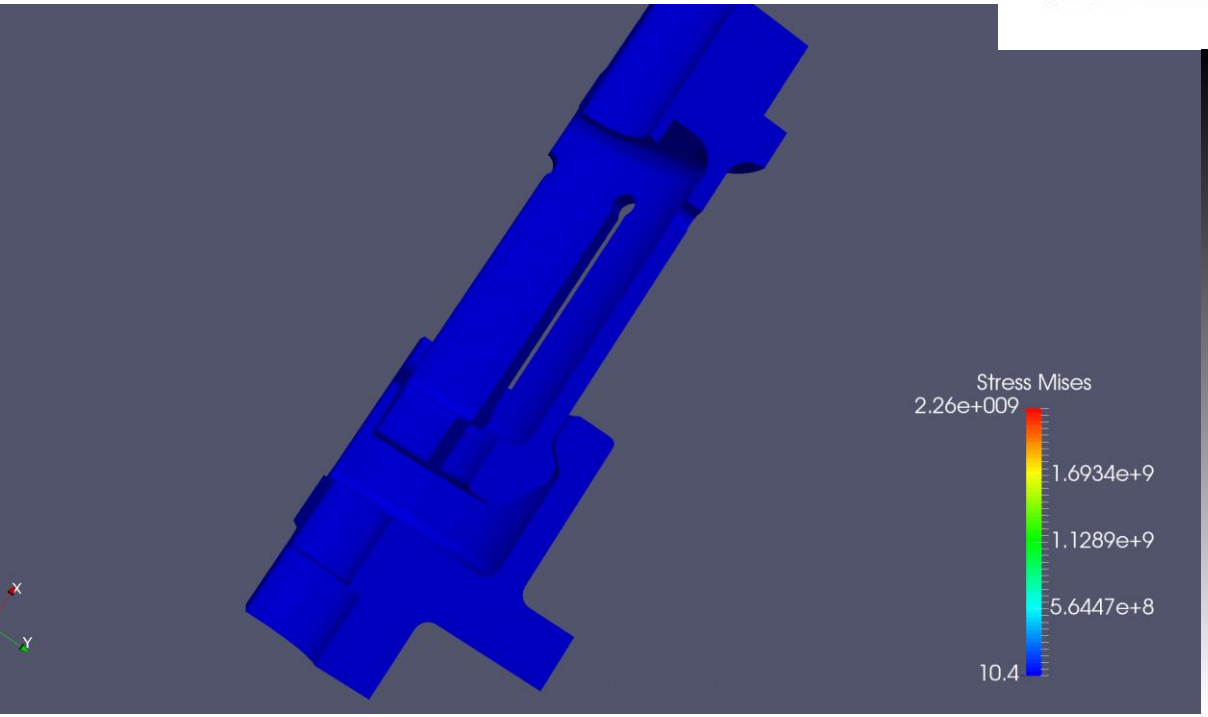


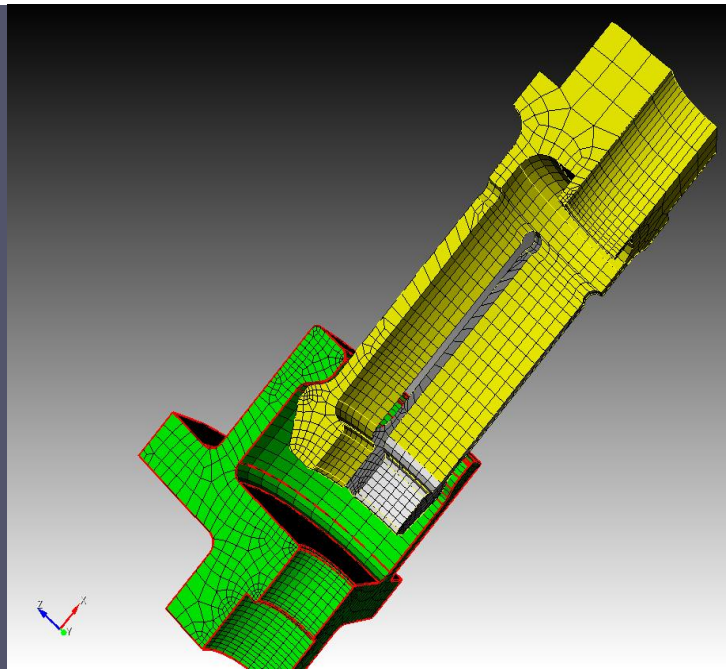


Collet lock

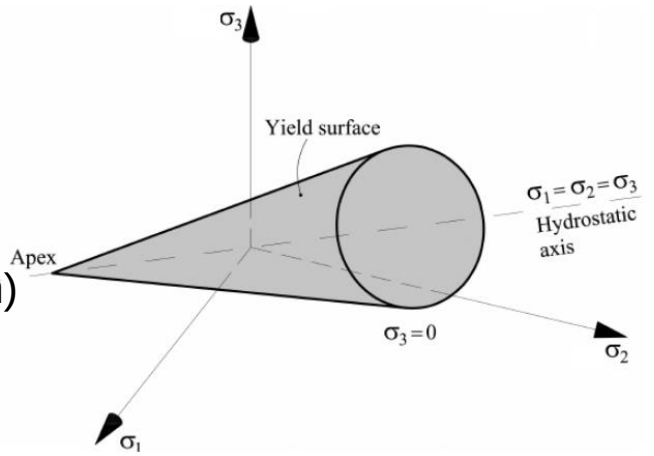
- Kinematic movement of the collet until its complete exit from the bushing
- Variable contact zone at different loading steps







Physical nonlinearity

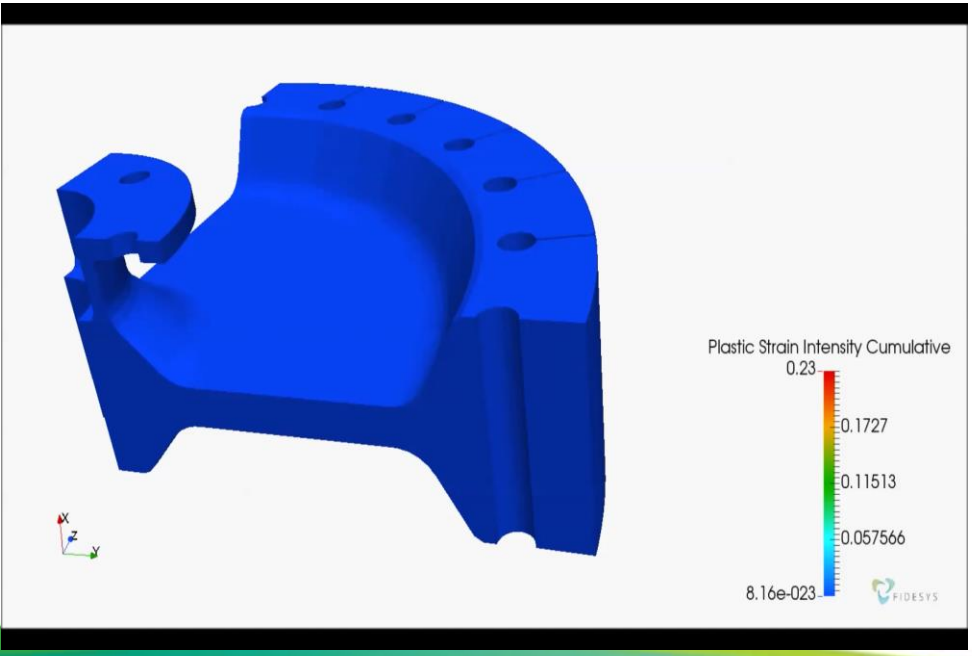


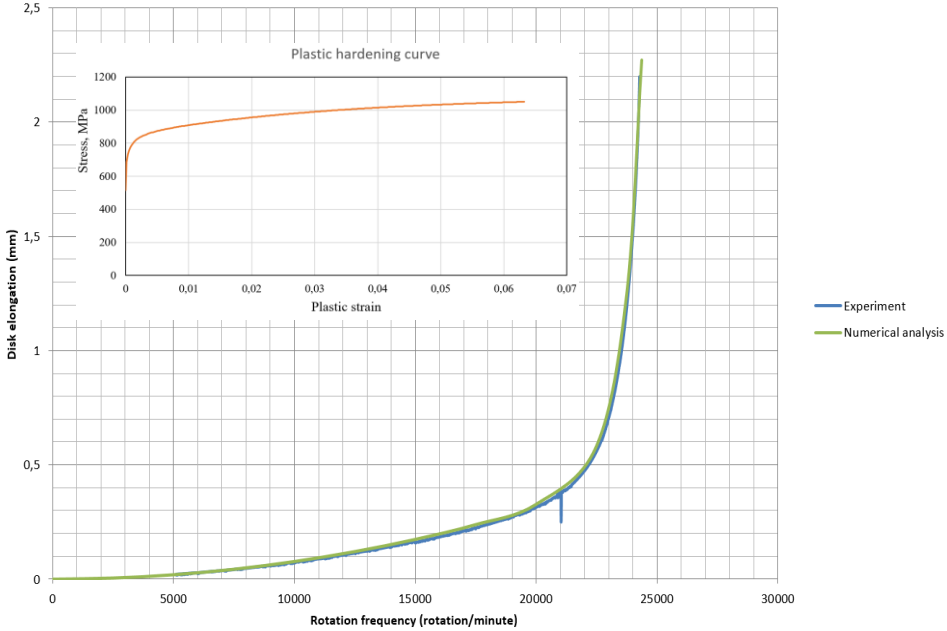
- Nonlinear elastic material models (Murnaghan, Mooney–Rivlin)
- Elastoplastic models (Mises, Drucker-Prager)
- Non-associated plastic flow rule
- Linear/Polylinear/Power hardening

$$f(\sigma_{ij}, \varepsilon_{ij}^{p}) = 0, \quad f(\sigma,\tau) = \tau - \alpha\sigma - Y$$

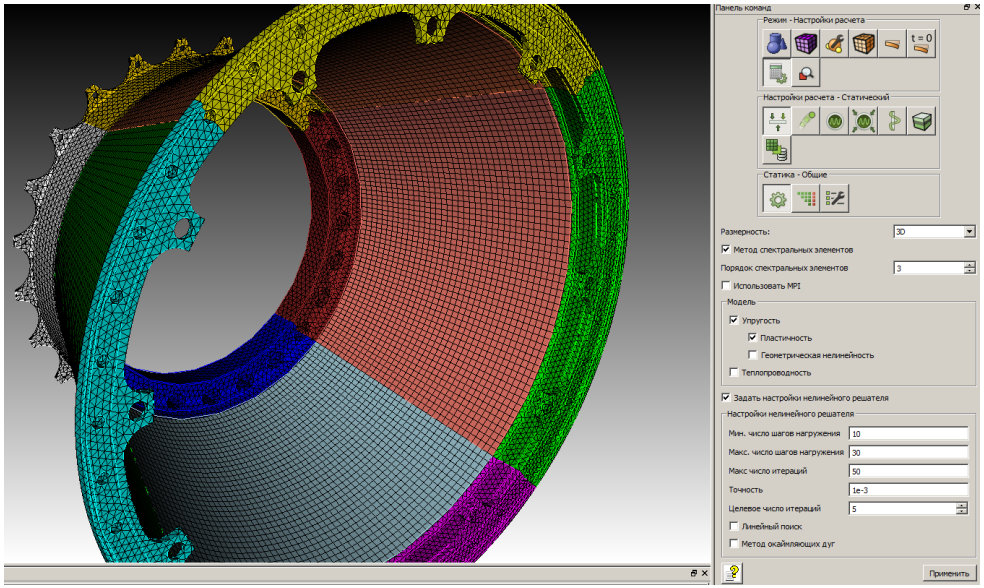
$$g(\sigma_{ij}, \varepsilon_{ij}^{p}) = 0, \quad g(\sigma,\tau) = \tau - \Lambda\sigma$$

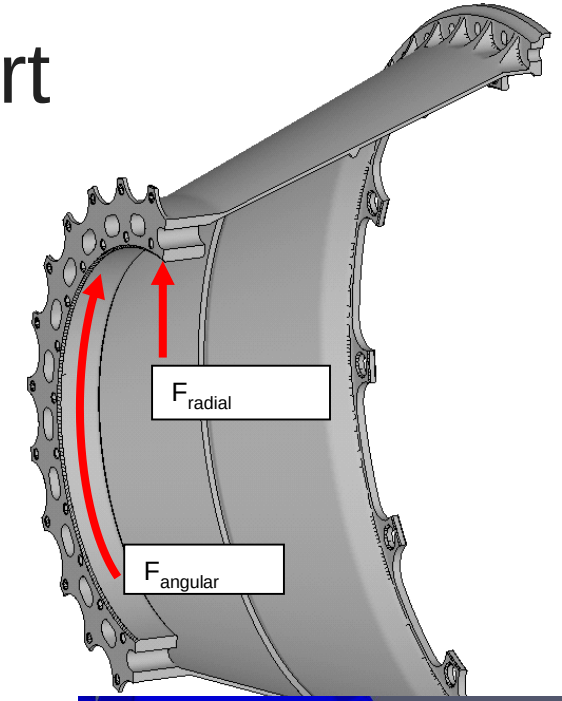
$$\mathrm{d}\varepsilon_{ij}^{p} = \mathrm{d}\lambda \frac{\partial g}{\partial \sigma_{ij}},$$

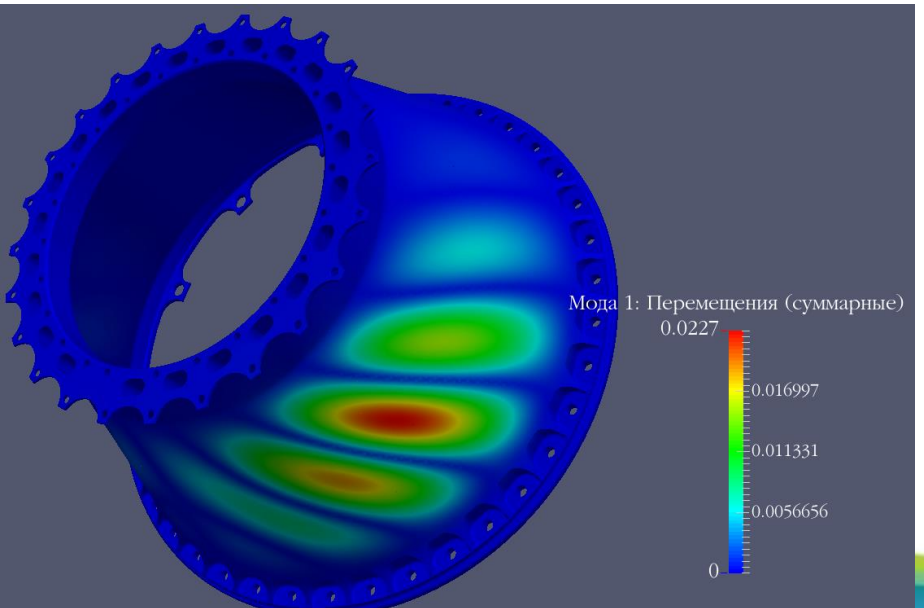


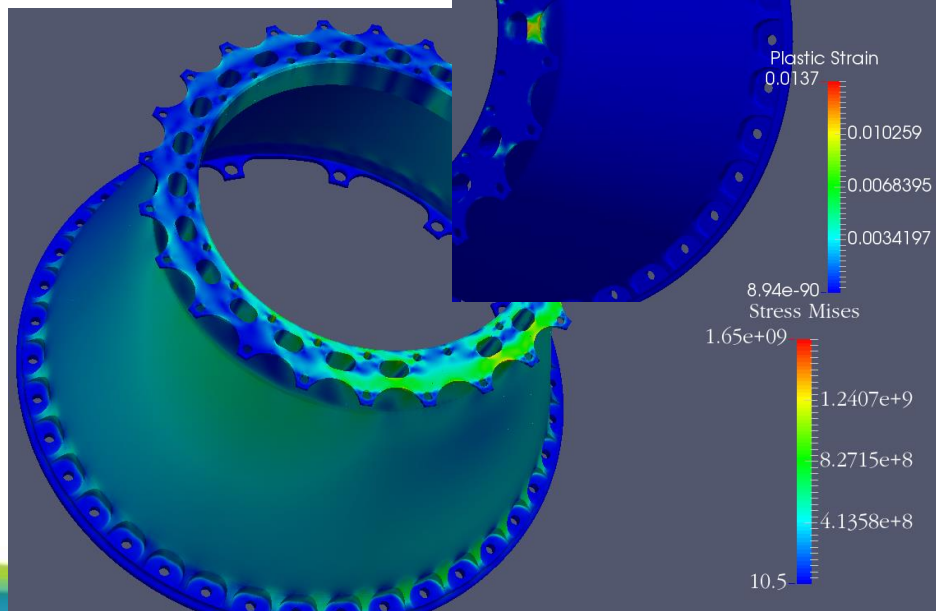


Stresses in the bearing support due to the fan blade breakage

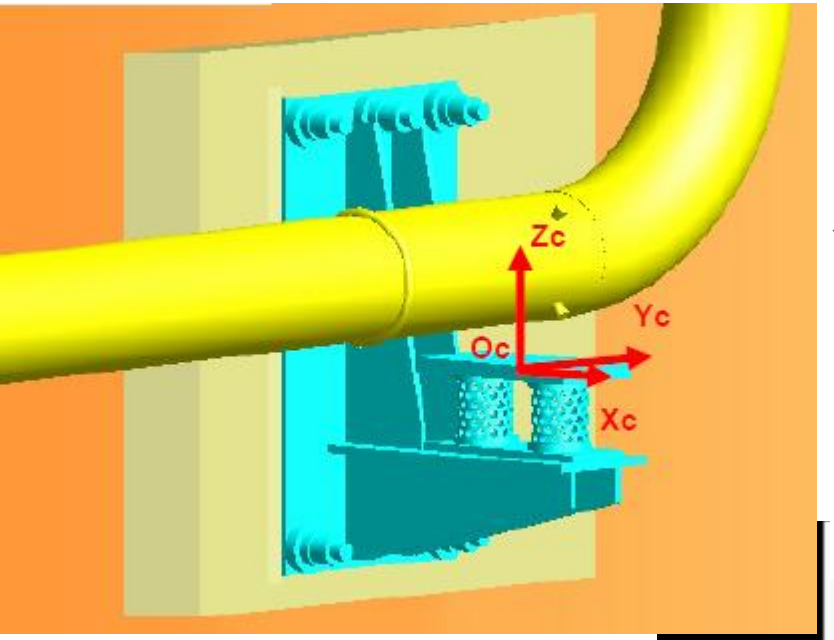


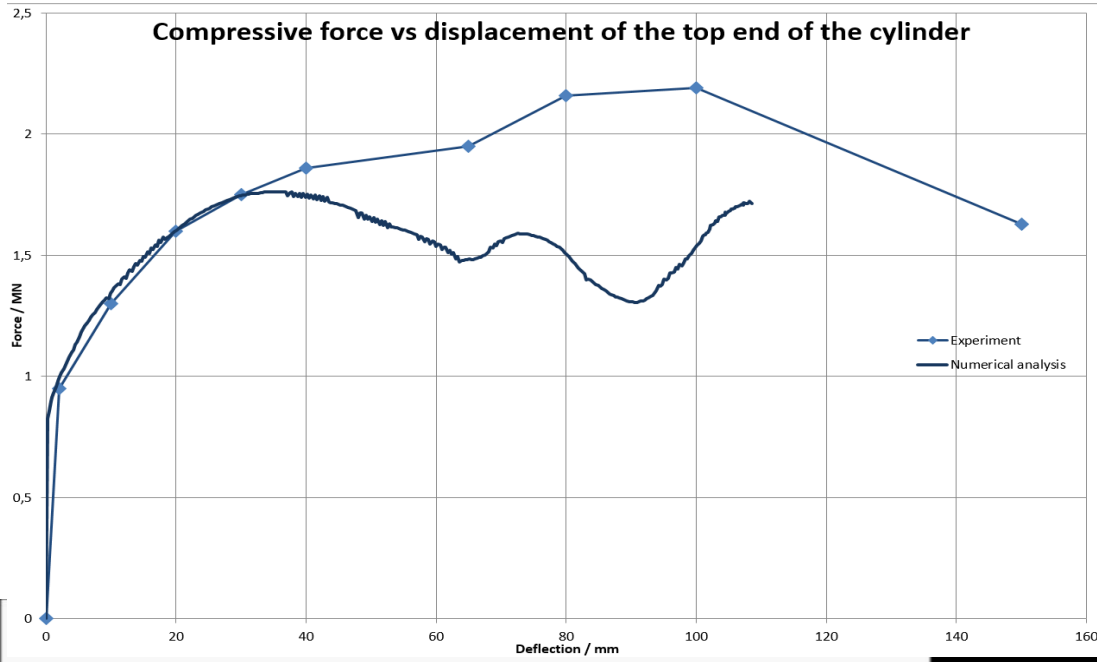




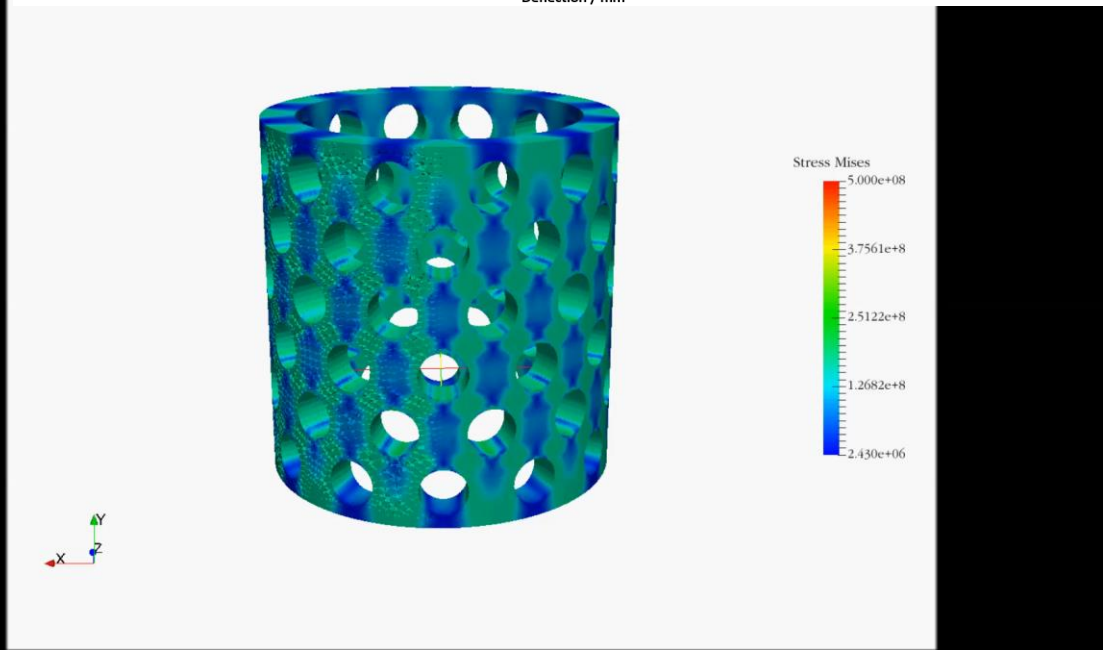


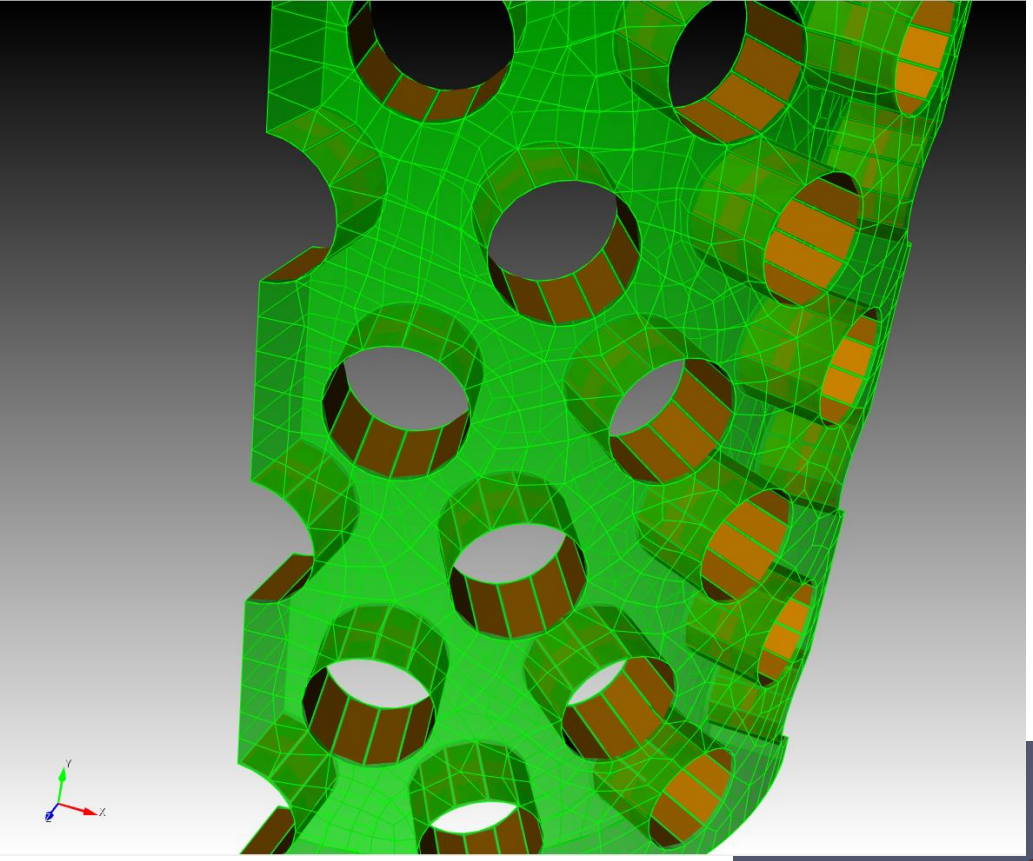
Mathematical modeling of whiplash limiters for pipeline ruptures



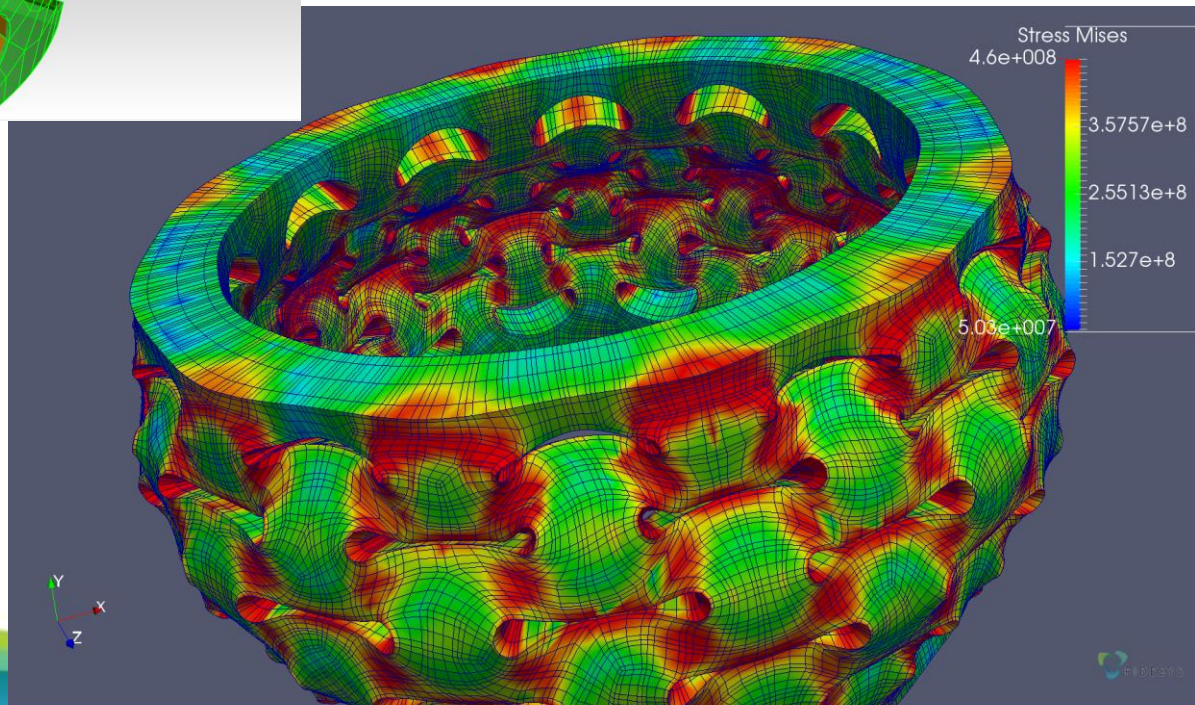


Elastoplastic nonlinear hardening under finite strains and multiple self contacts





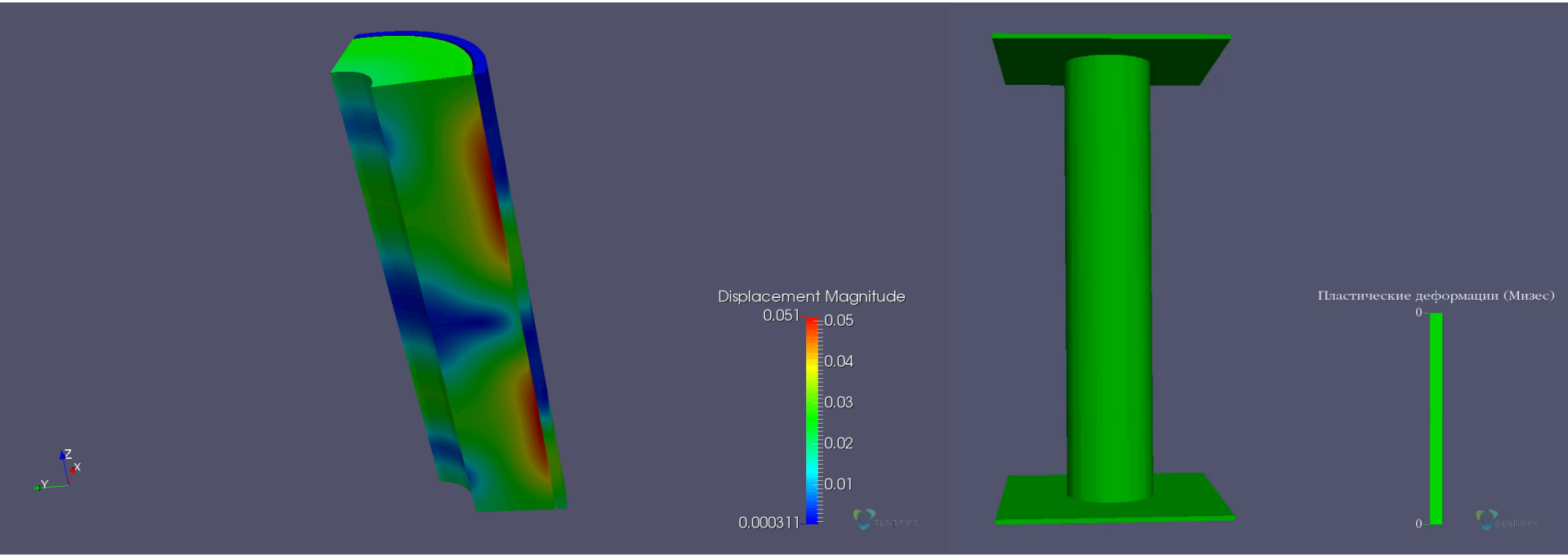
Curvilinear coarse mesh used for simulation



Numerical results at high order spectral element mesh

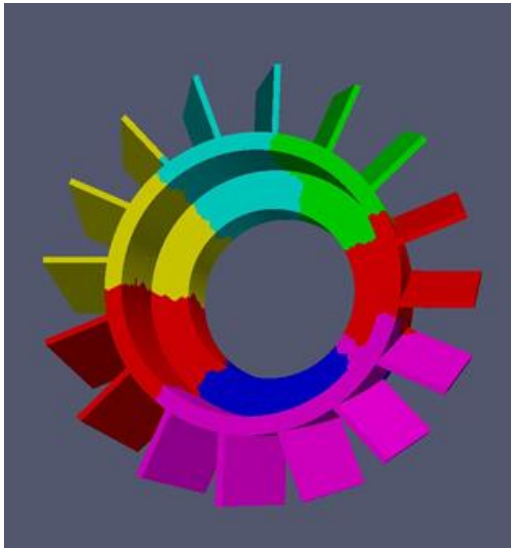
Complex nonlinear problems

- Steady-state and transient thermal conduction
- Different kinds of BCs: temperature, heat flux, convective heat transfer and radiative heat transfer
- Thermo(Poro)elasticity, Thermo(Poro)elastoplasticity
- Self contact problems under large plastic strains



Additional modules

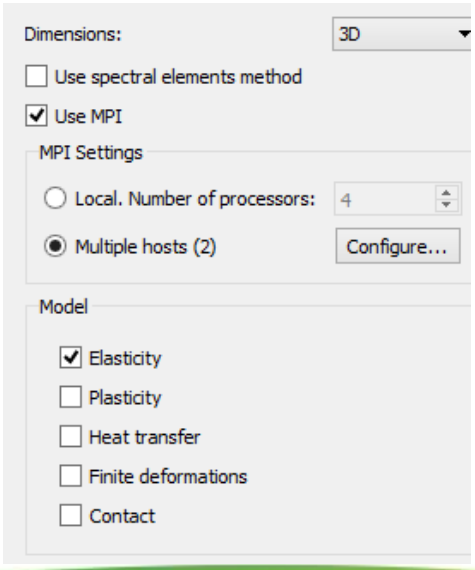
CAE Fidesys structure

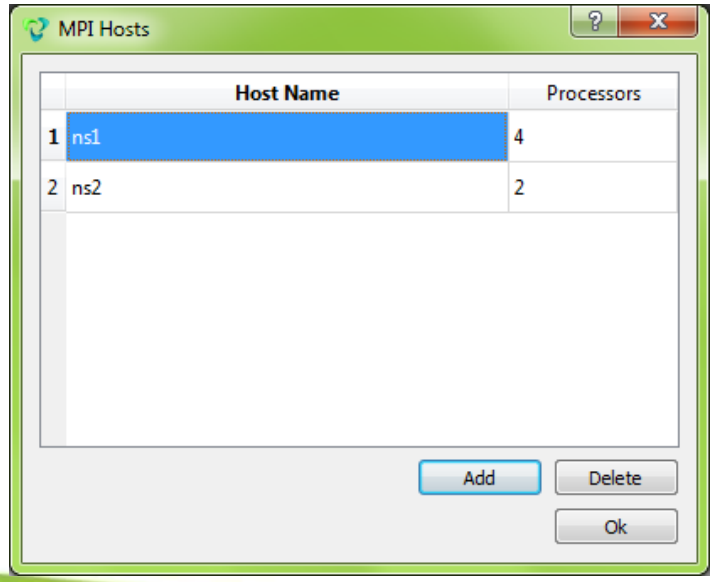


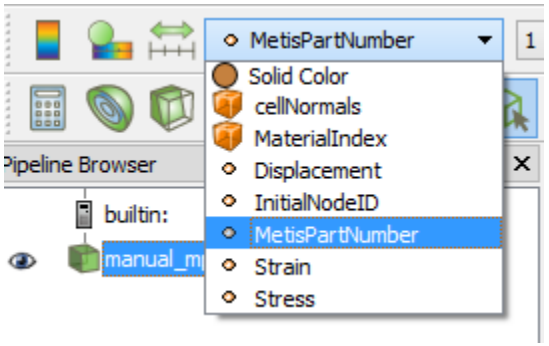
Fidesys HPC
high-performance computing

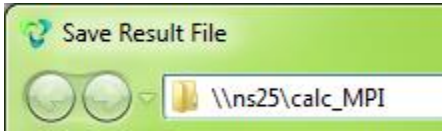
Acceleration of calculations

- All major analysis stages are parallelized by
- OpenMP & MPI technologies at
- Multi-core systems and
- Cluster/network systems which gives
- A drastic reduction of the total analysis time

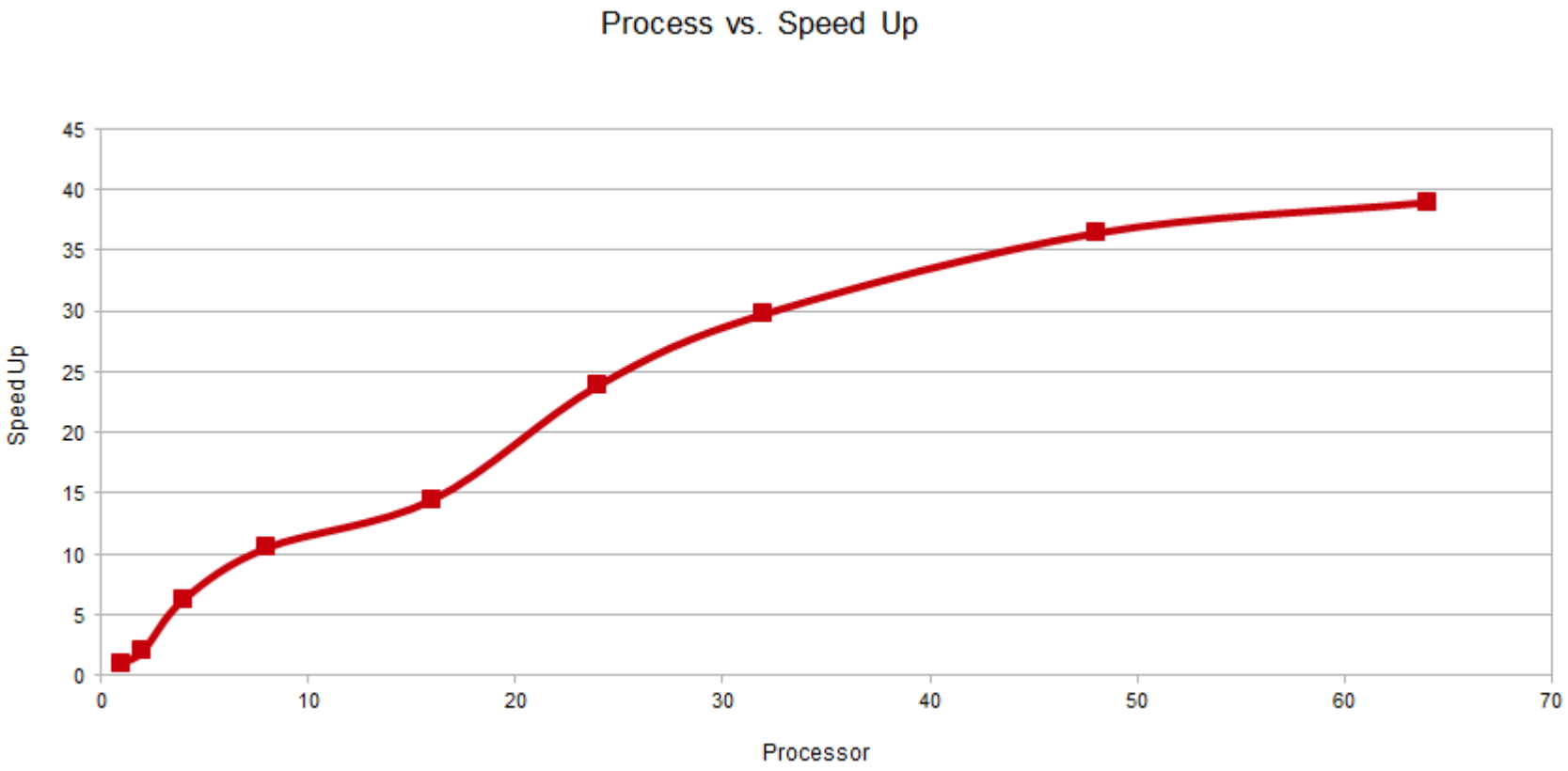




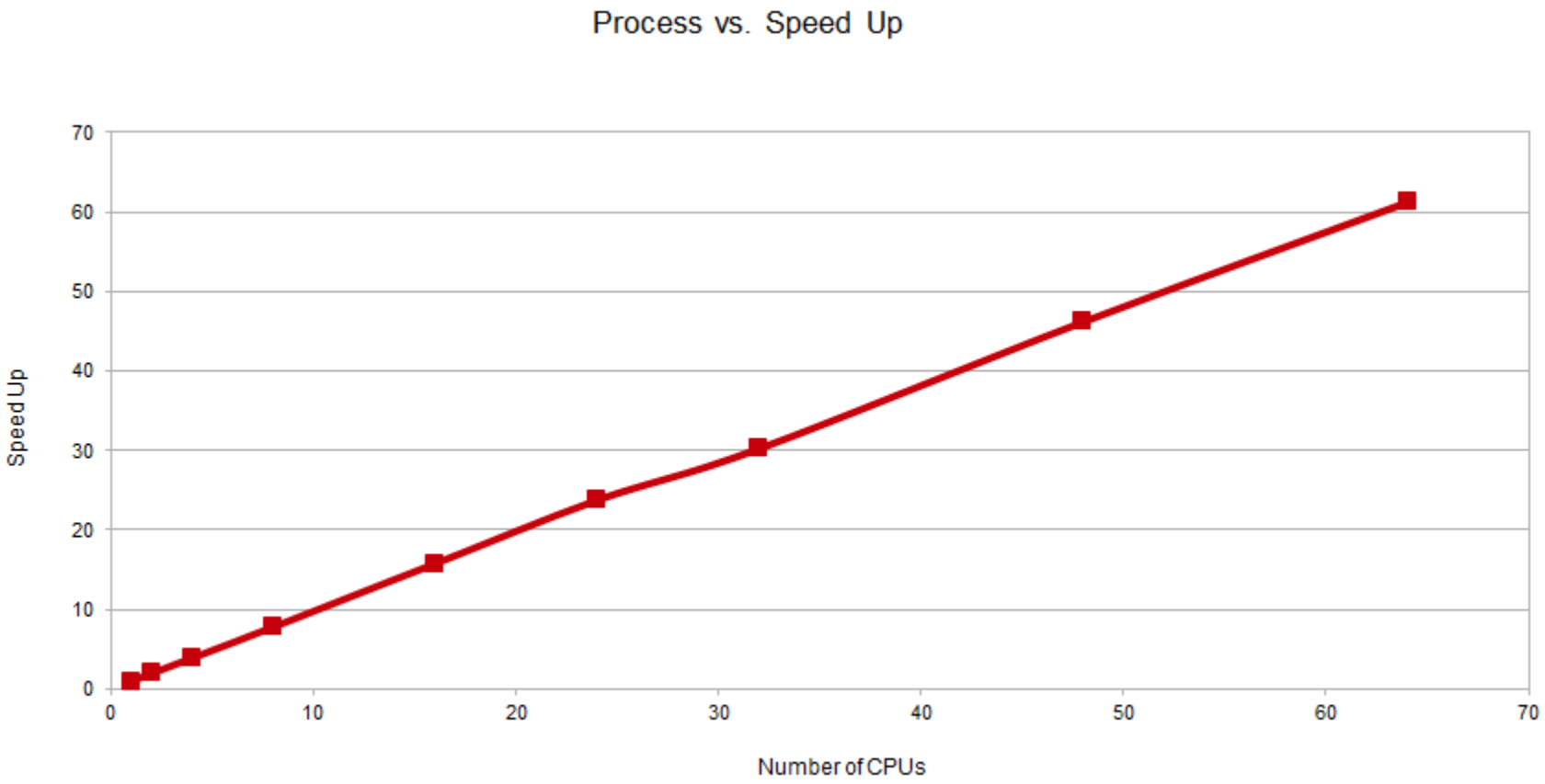




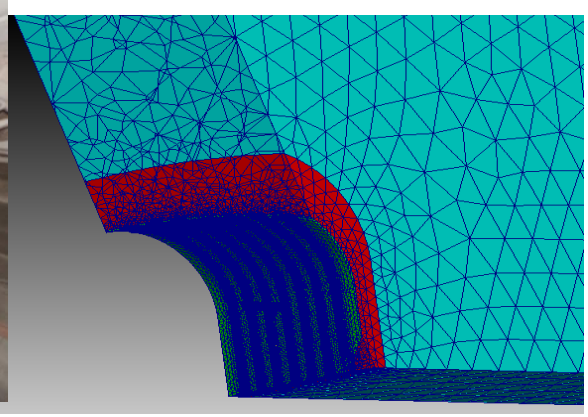
Scalability – 500 thousands of DOFs



Scalability – 4 millions of DOFs

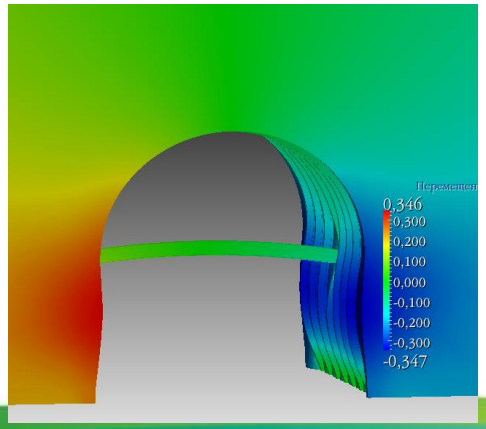


HPC example: geomechanical analysis of mine workings (> 30 mln. elements)

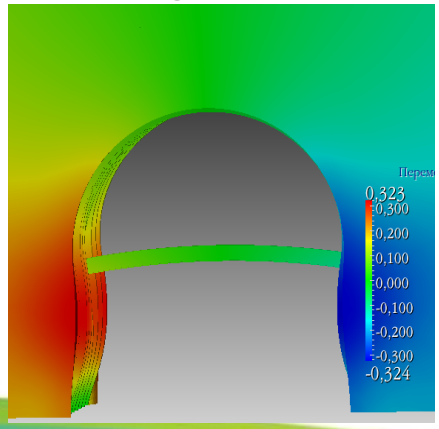


The current geomechanical situation in the mine is reproduced and the result of the possible strengthening of the rock mass after puffing (tamponage) or the replacement of the linings.

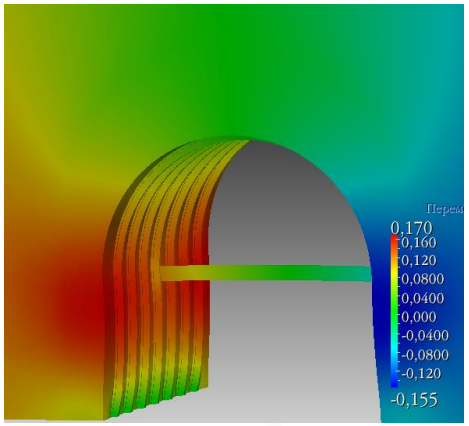
Current situation horizontal convergence 70cm



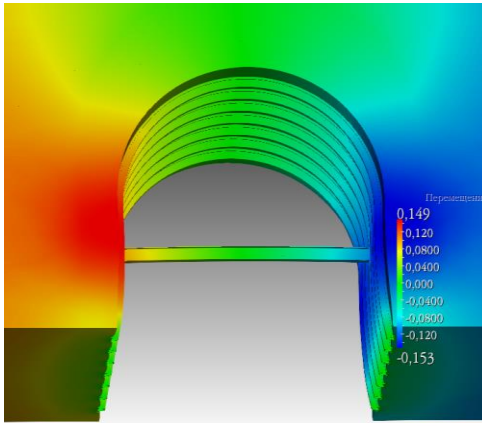
Replacement of SVP 27 for SVP 33 Horizontal convergence 64 cm



Reinforcement (tamponage) Horizontal convergence 32 cm



Reinforcement (tamponage) + Replacement of SVP 27 at SVP 33 Horizontal convergence 30 cm

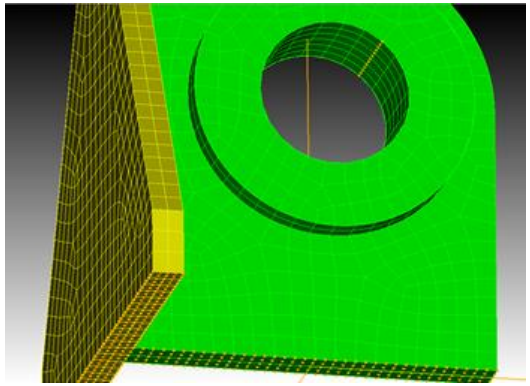


CAE Fidesys structure

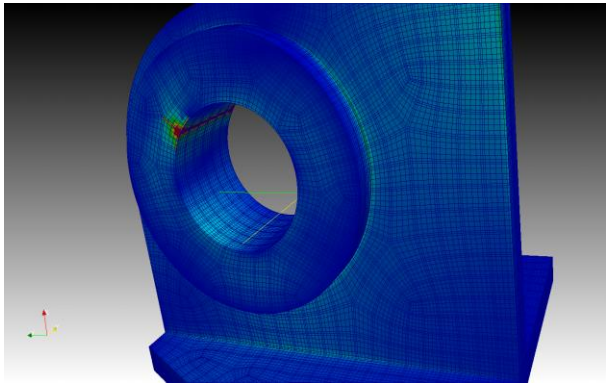
Fidesys Dynamics
Spectral element method.

High order space discretization (SEM)

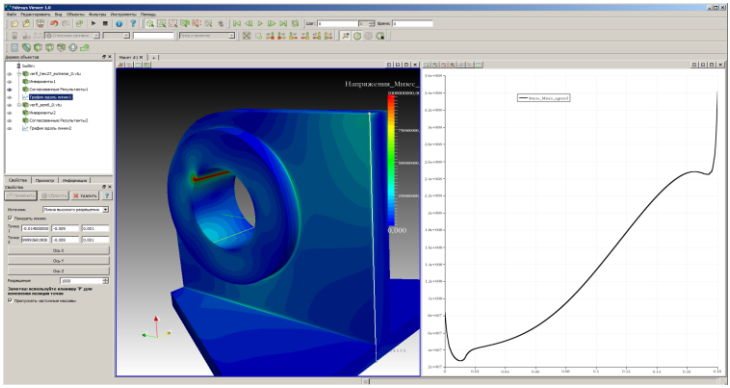
- High accuracy and stability of the numerical algorithm
- Automatization of grid convergence analysis
- Lack of locking issues for high orders of the numerical scheme



A 3D discrete model of the deck eye



Spectral element mesh of the model

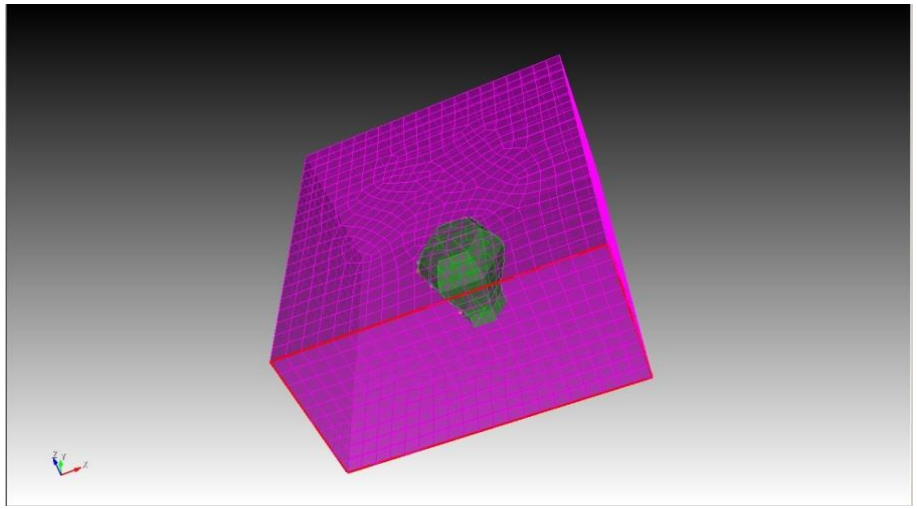


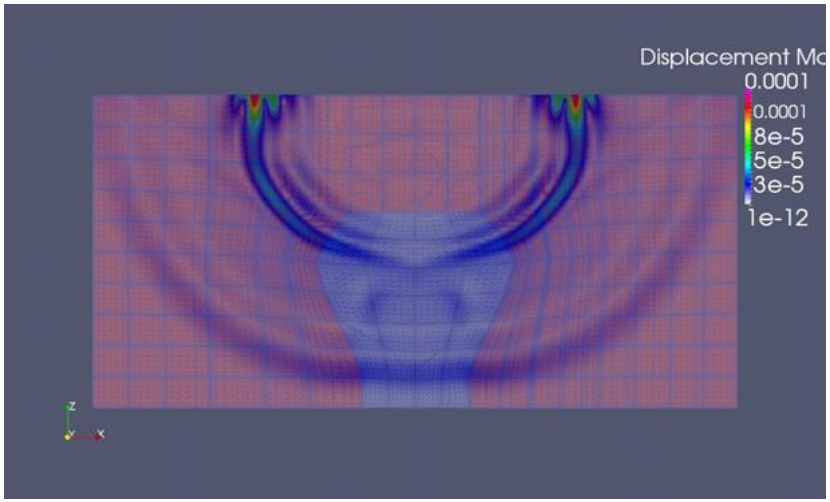
Analysis of stress concentrations

Additional modules

Fidesys Dynamics
Spectral element method.
High-precision non-stationary problems

- Spectral element method – a modern FEM modification
- Most effective for dynamical analysis
- Non-stationary problems with fast changing processes
- Increased speed and accuracy of calculations
- Effective parallelization available with Fidesys HPC



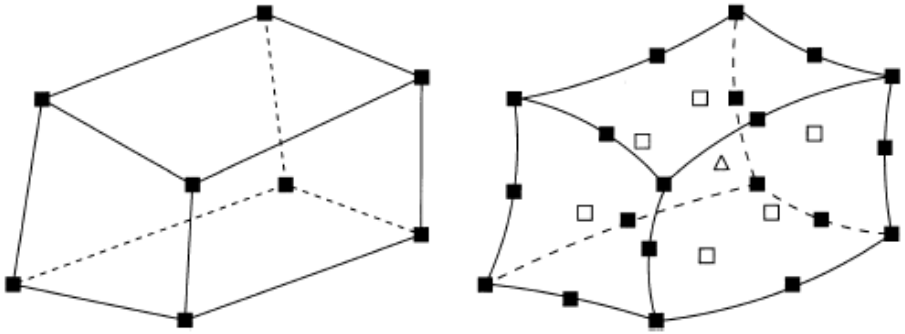


3D seismic full wave modelling

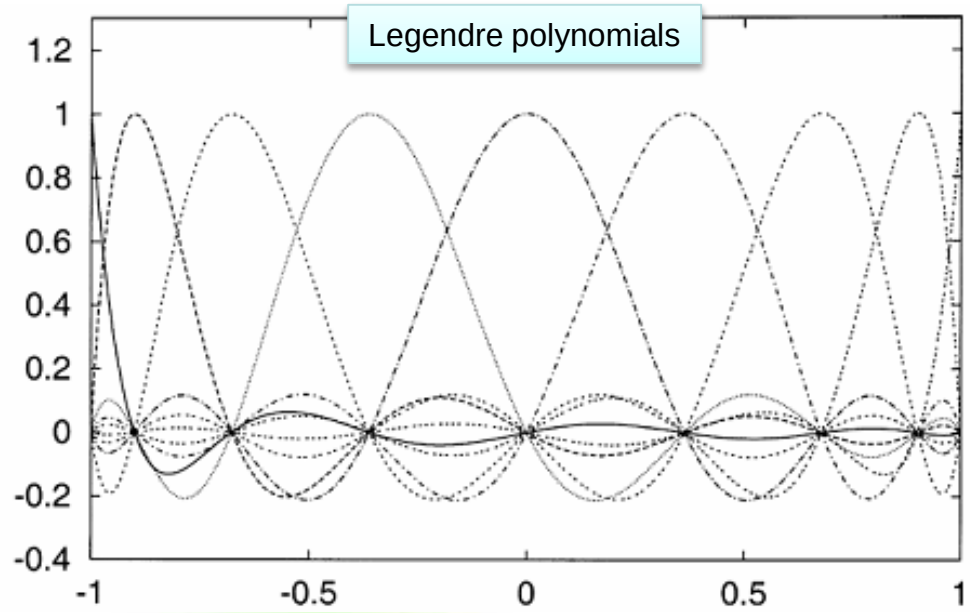
Spectral Element Method

Basis functions — Legendre polynomials, providing a spectral-order approximation in space

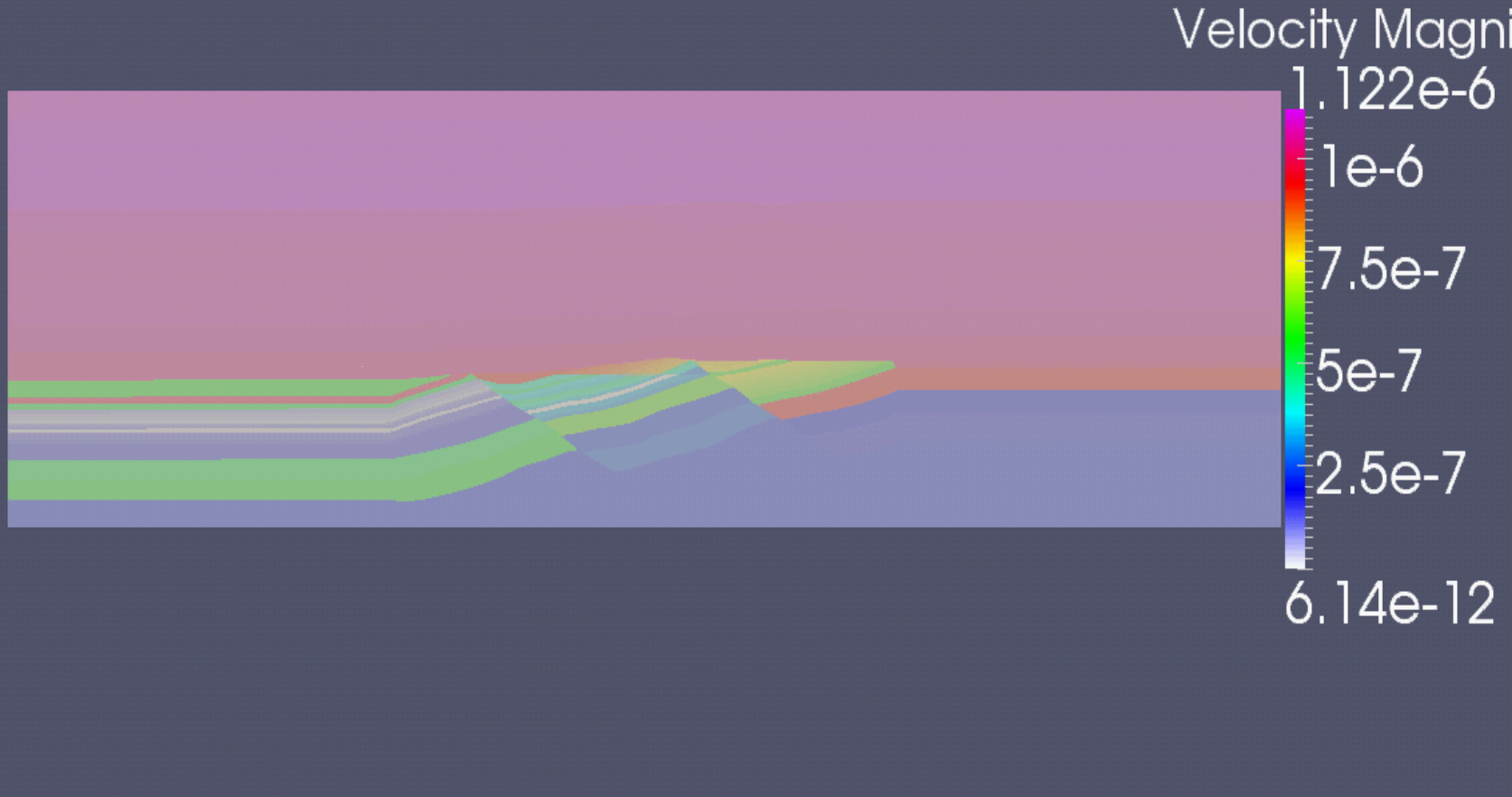
Elements of the grid



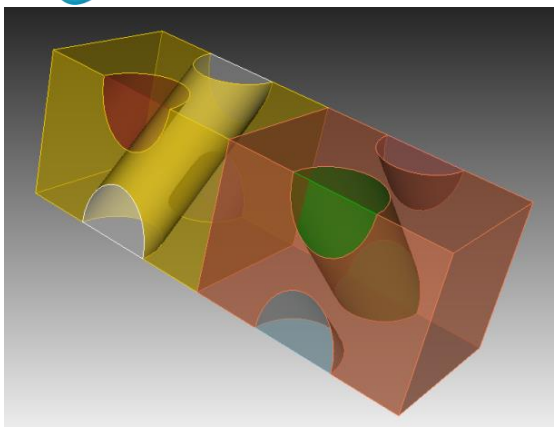
n = 1 n = 2

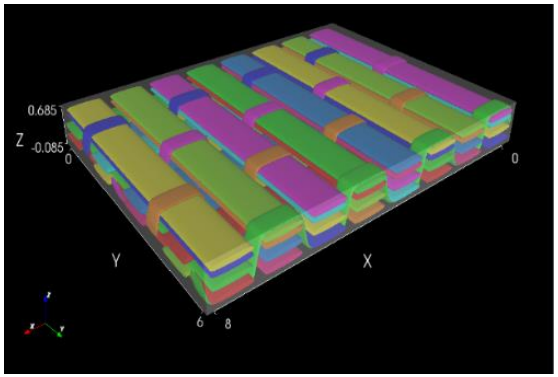


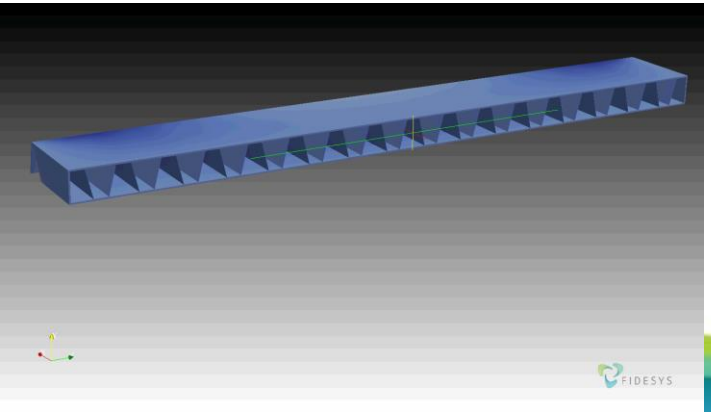
High accuracy in approximating the curvilinear geometry of the body



CAE Fidesys structure





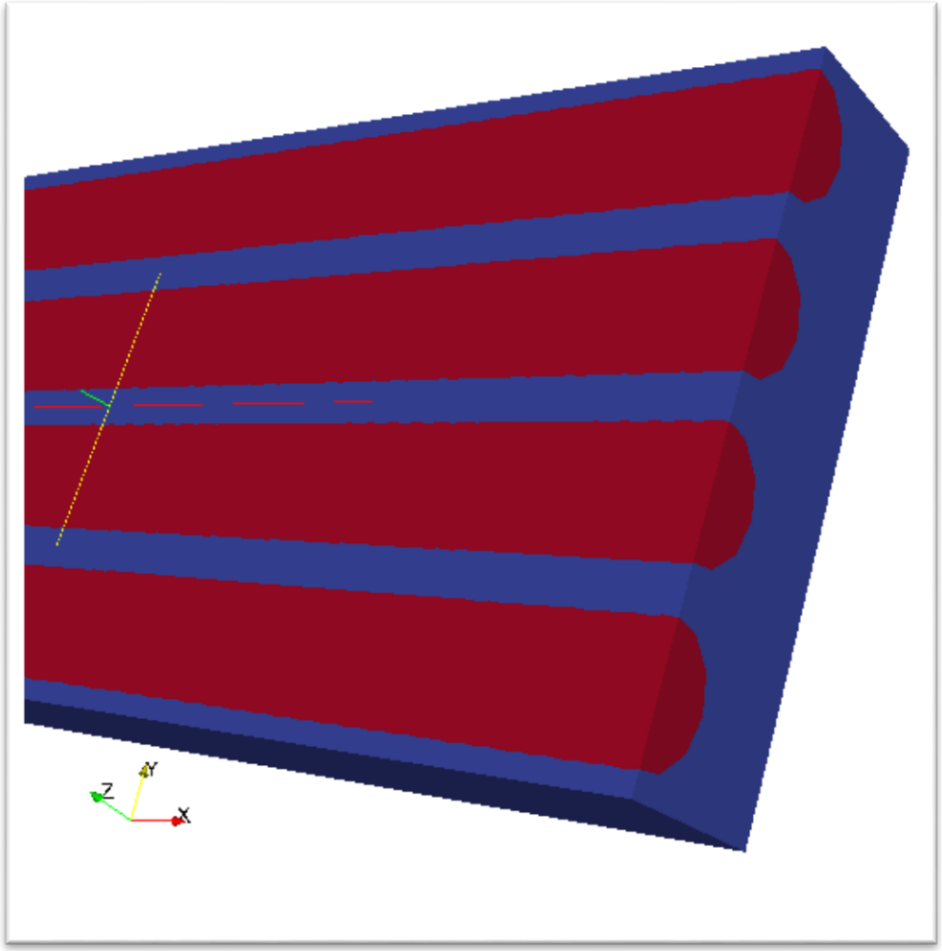


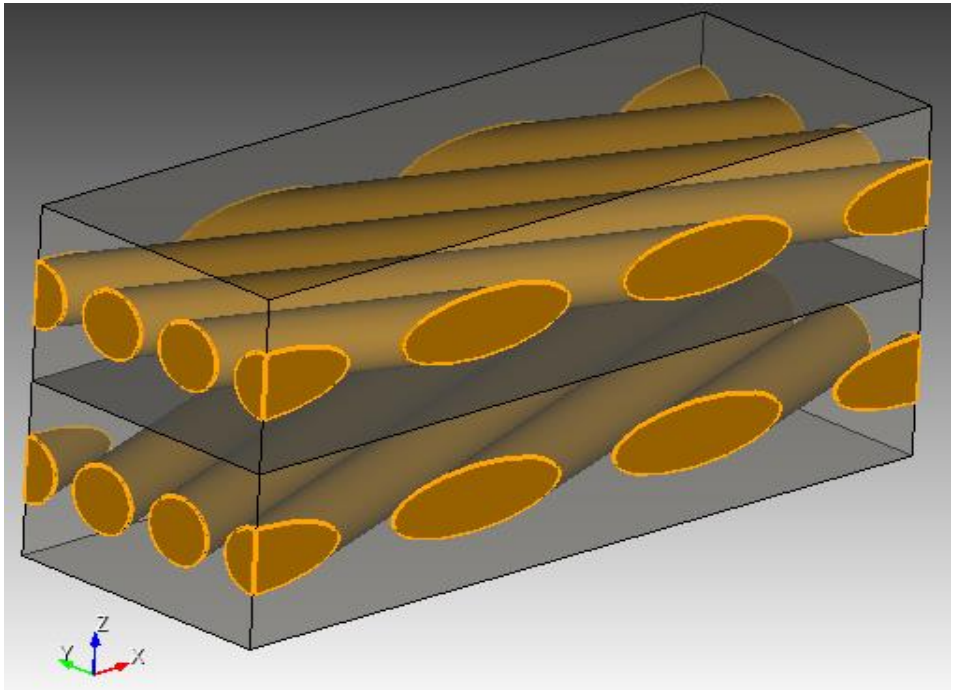
Fidesys Composite
Multiscale simulation

Estimation of the composite effective properties

- Engineering analysis of composites and structural elements made of them
- Multiscale simulations
  - Properties of reinforcing fibers
  - Composite matrix modeling
  - Effective properties of the composite are estimated based on the representative volumetric element (RVE) analysis
- Progressive fracture growth in the composite
  - Determination of the critical stress value of the structure
  - Simulation of a composite behavior after fracture initiation
- Nonlinear effective properties, prestressed materials

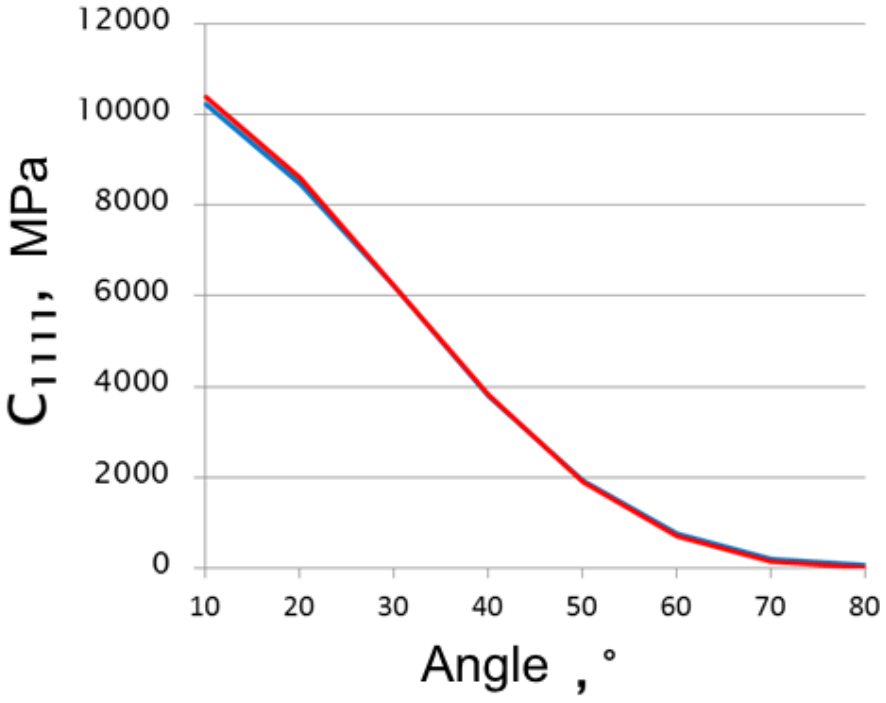
Evaluation of rubber cord effective properties

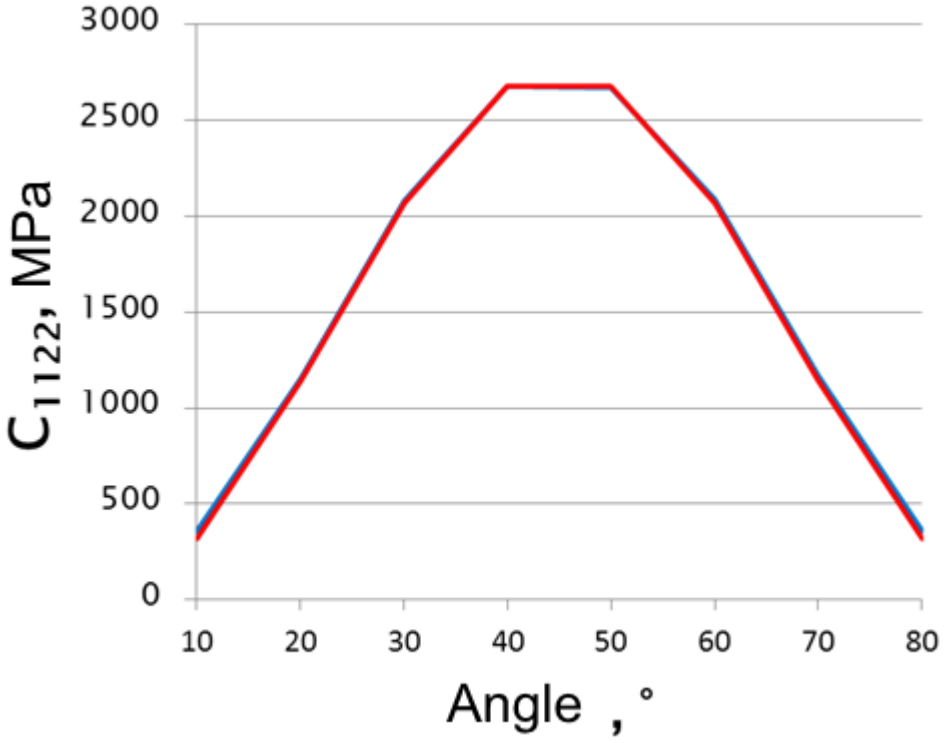




Numerical vs Analytical results

- Linear problem, dual-layer rubber cord
- Young's modulus of the cord and the rubber differ in 4 orders





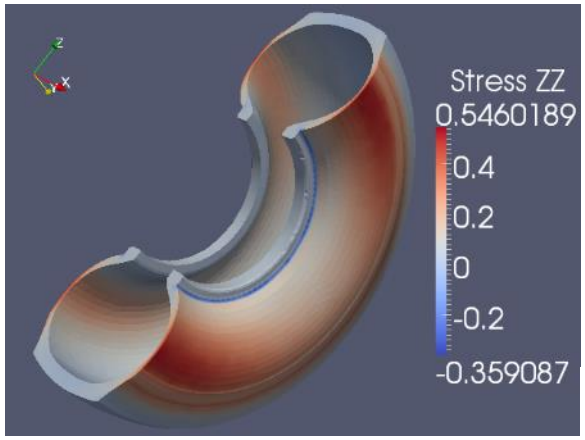
----- Numerical solution

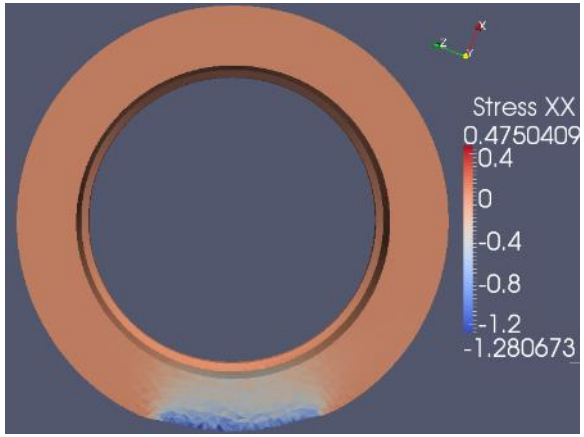
----- Analytical solution (Bidermann ,1963)

CAE Fidesys structure

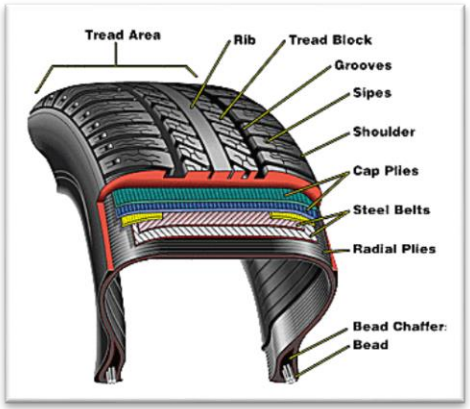
Fidesys Composite
Multiscale simulation

Effective properties of composite materials

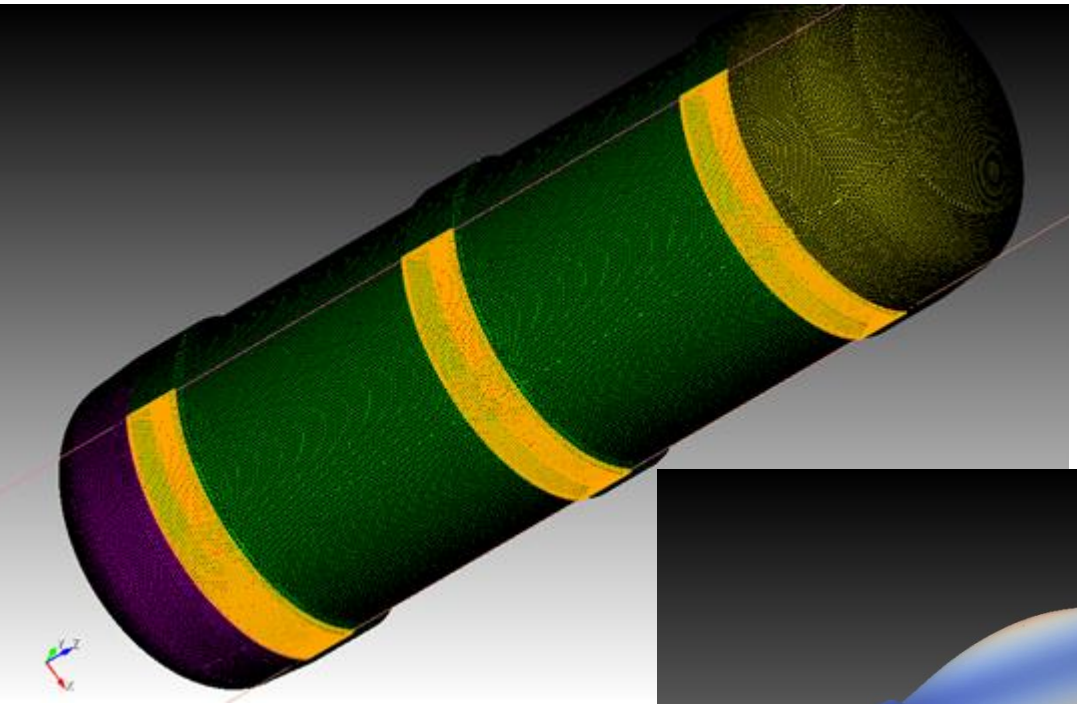


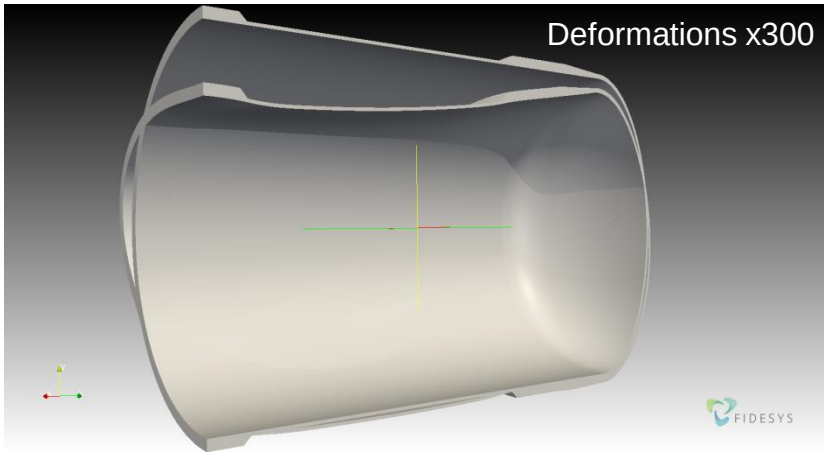


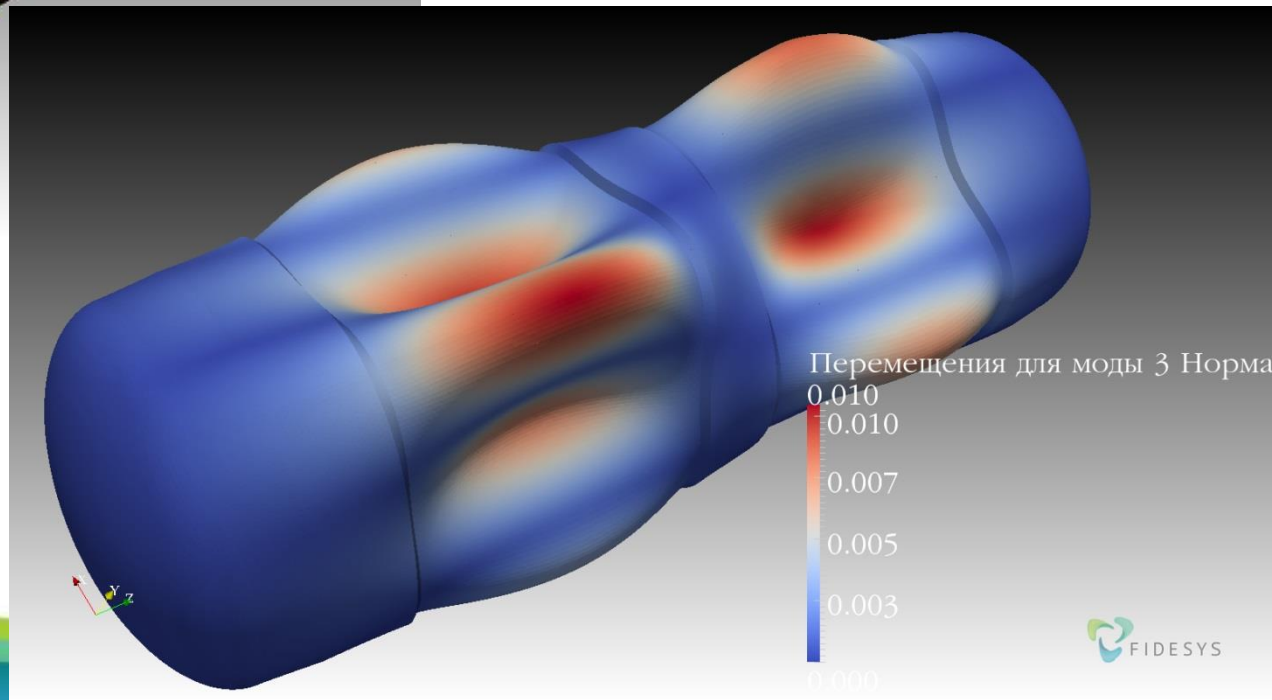
- Modeling a realistic composite microstructure
- Simulation of products made of a rubber cord



Analysis of the composite tank under internal hydrostatic pressure





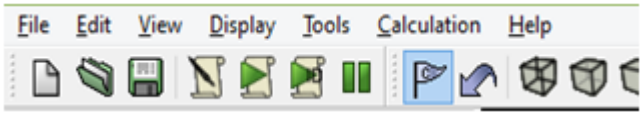


3rd Buckling mode

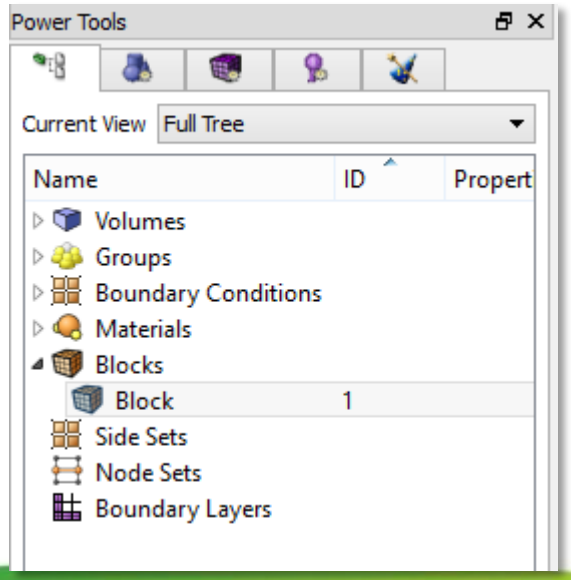
CAE Fidesys interfaces

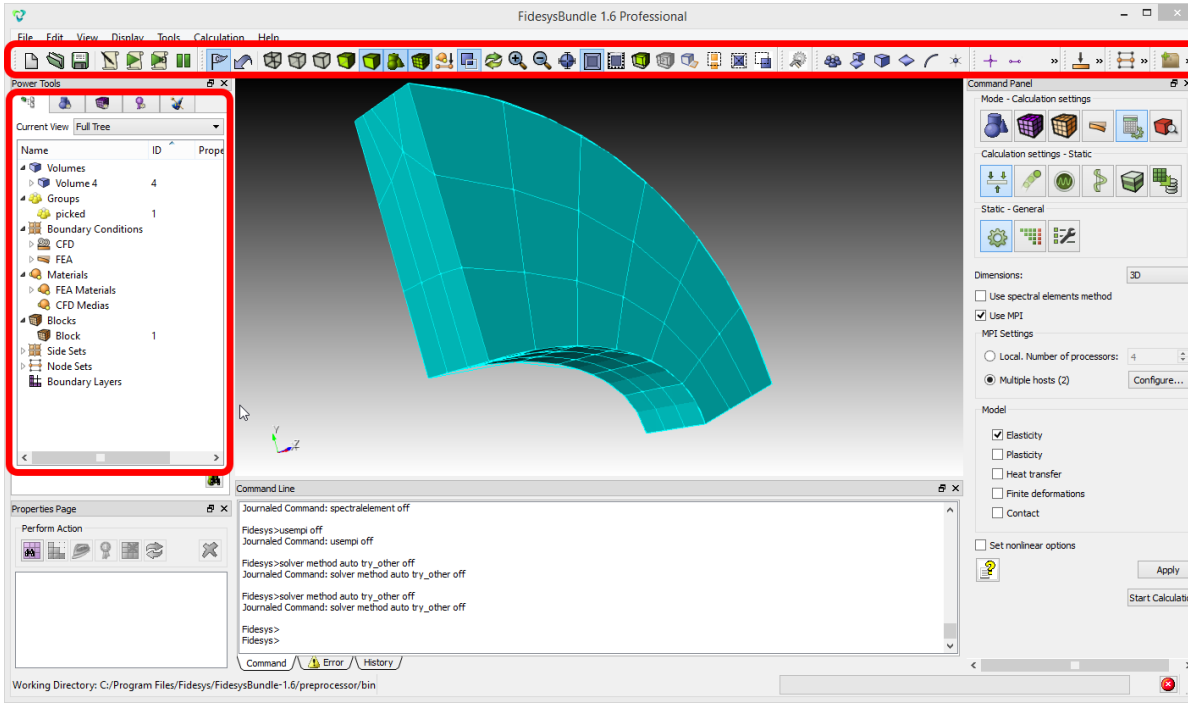
CAE Fidesys interface

- Standard operations on files and projects



- The Model Tree

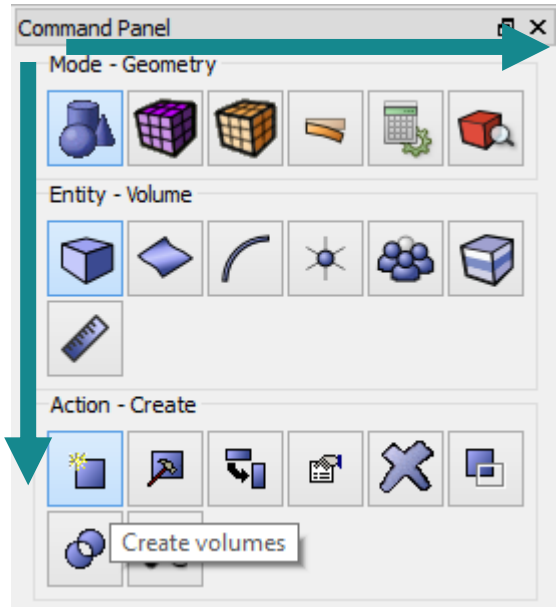


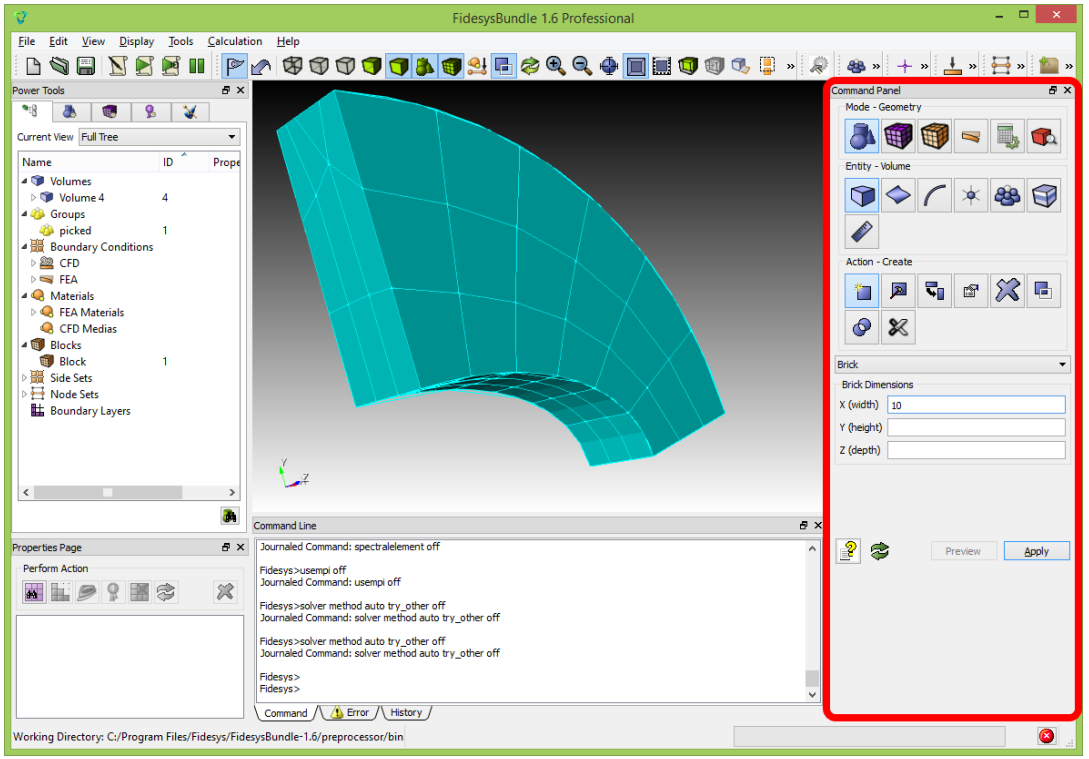


Main window

CAE Fidesys interface

- Logically arranged Command Panel

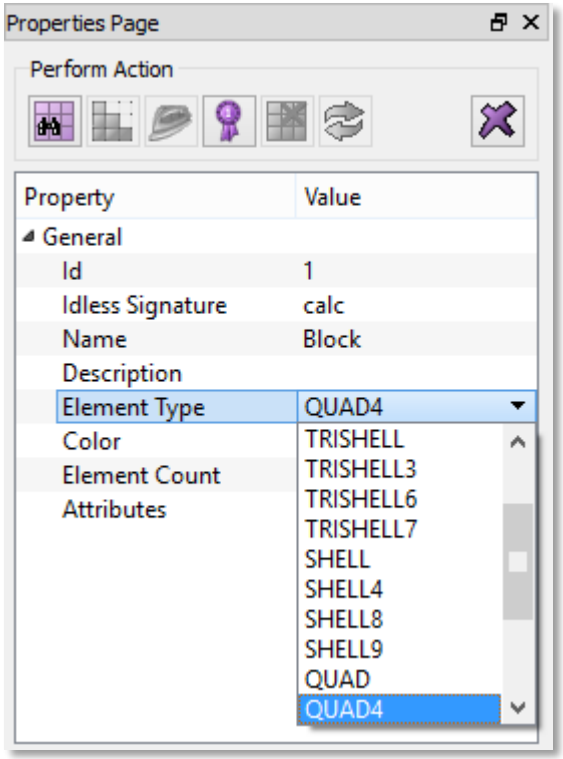


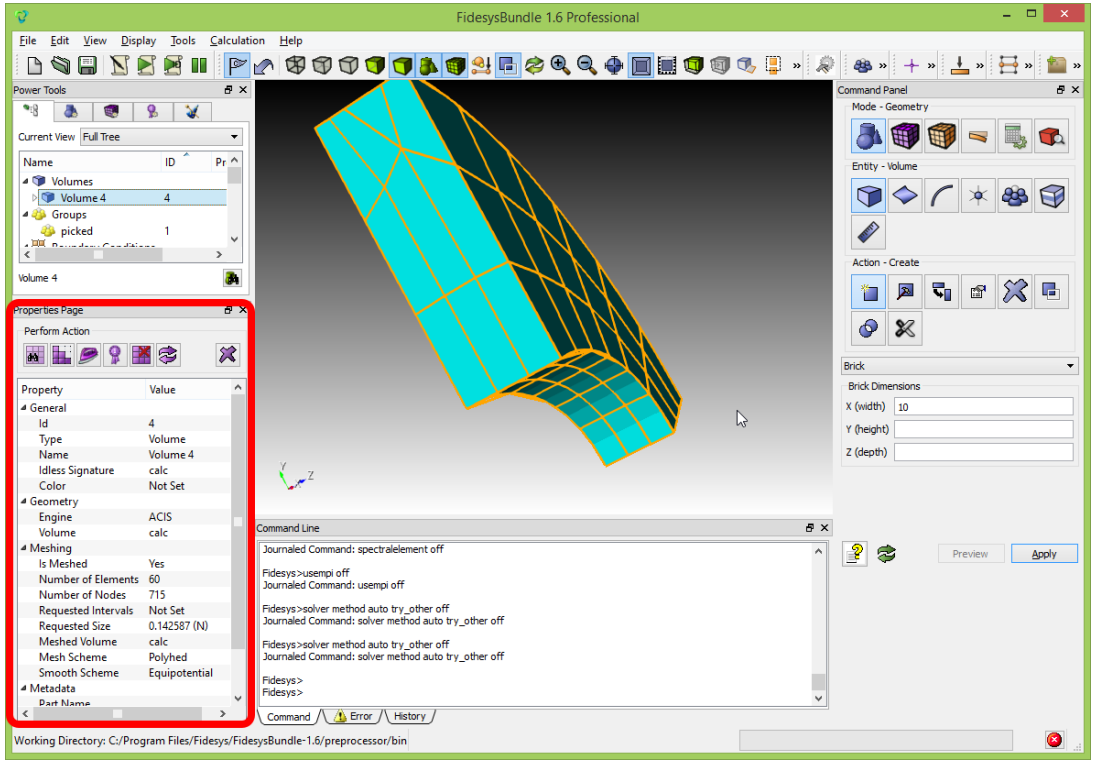


Main window

CAE Fidesys interface

- Properties of the selected object (Properties Page)

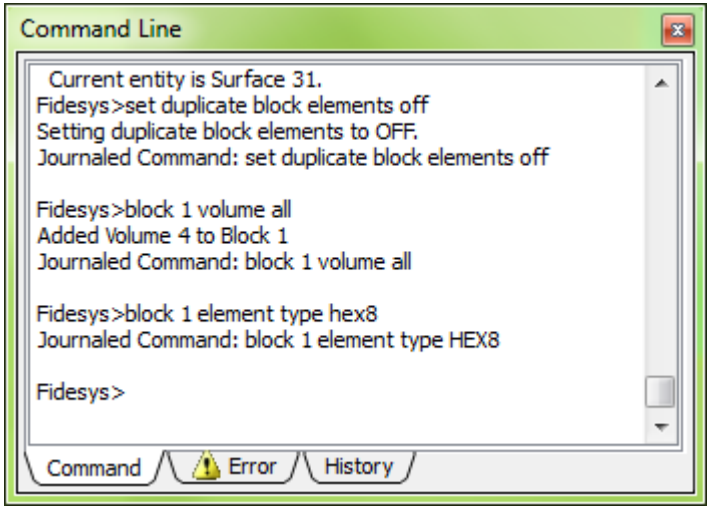




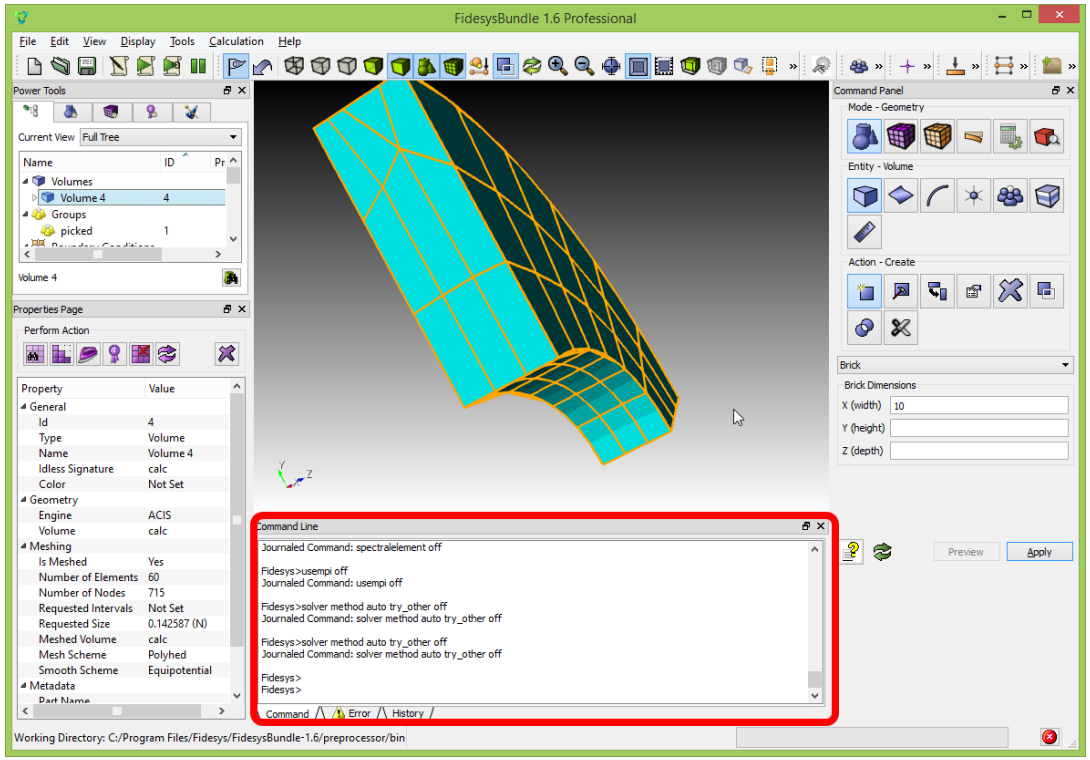
Main window

CAE Fidesys interface

- Console



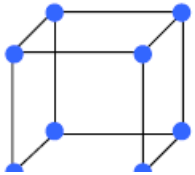
- ✓ Entering Commands
- ✓ Displaying messages
- ✓ History
- ✓ Running scripts



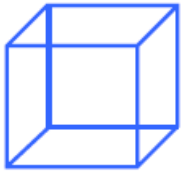
Main window

CAE Fidesys interface

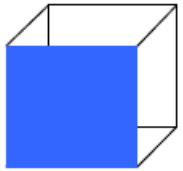
Geometry creation



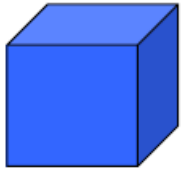
Vertex



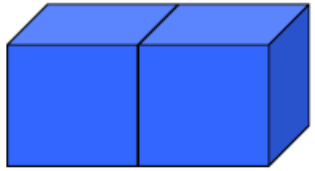
Curve



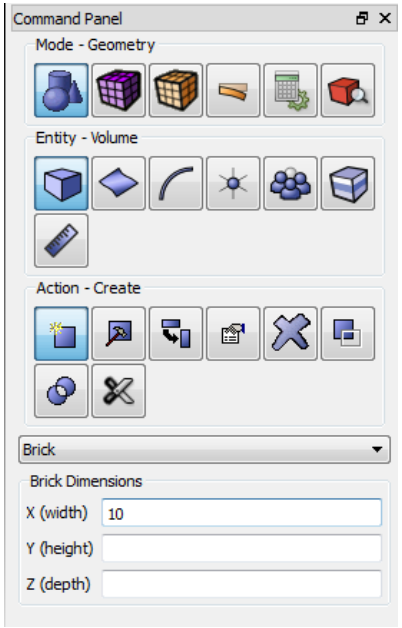
Surface

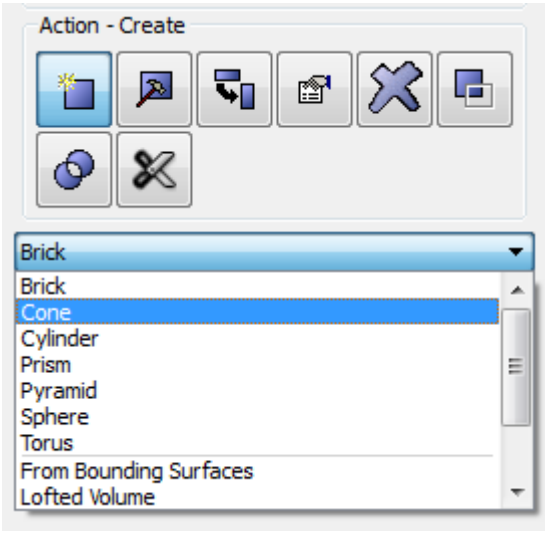


Volume



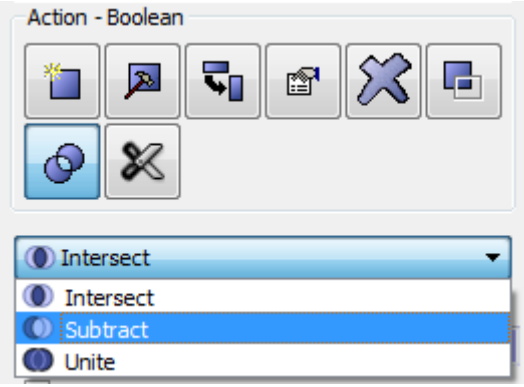
Body

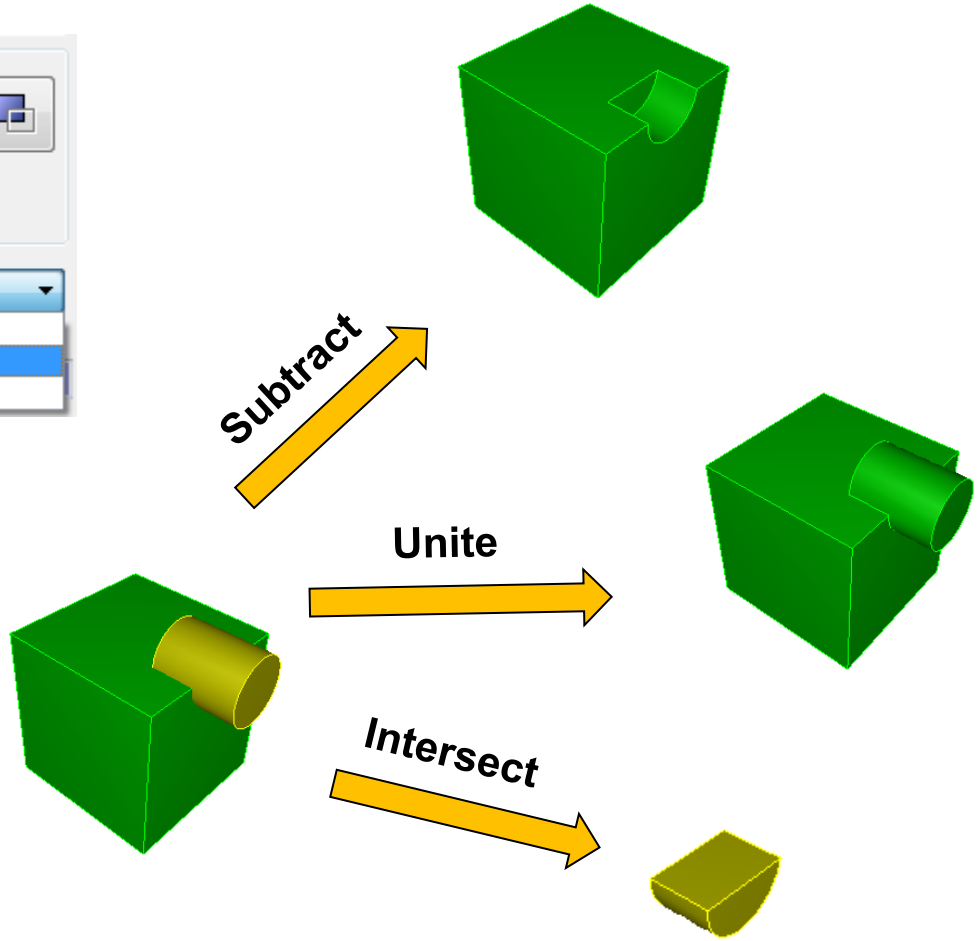




CAE Fidesys interface

Geometry editing





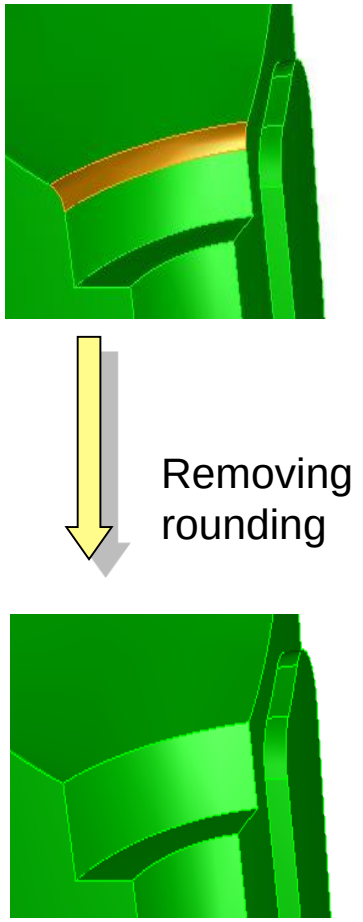
CAD vs CAE

- CAD-model
  - In most cases is 3D
  - Acknowledges all the geometrical tiny details
  - Possible intersections and overlappings

- CAE-model
  - Strength characteristics are of major importance
  - Some three-dimensional parts should be replaced by beams and shells

Model preparation

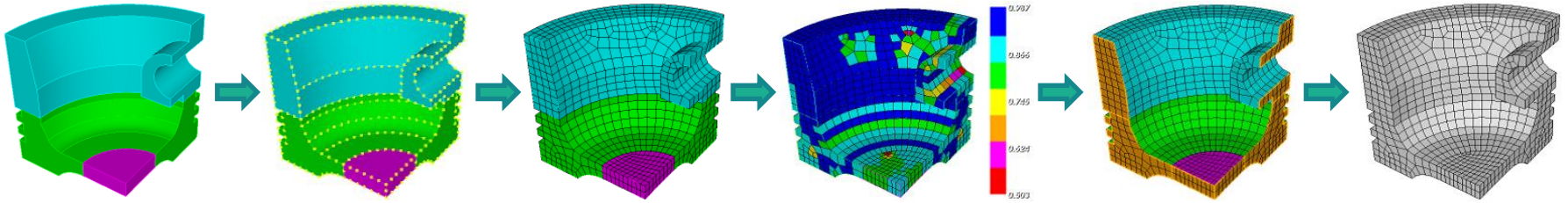
- 3D → 2D, 1D
  - Beam structure generation
  - Shell structure generation
- 3D geometry editing
  - Chamfers and roundings elimination
  - Virtual combining of surfaces
  - Conformity of volumes



CAE Fidesys interface

Meshing

1. Geometry import
2. Specifying meshing intervals
3. Mesh generation
4. Mesh quality checking
5. Creating BCs
6. Calculation parameters



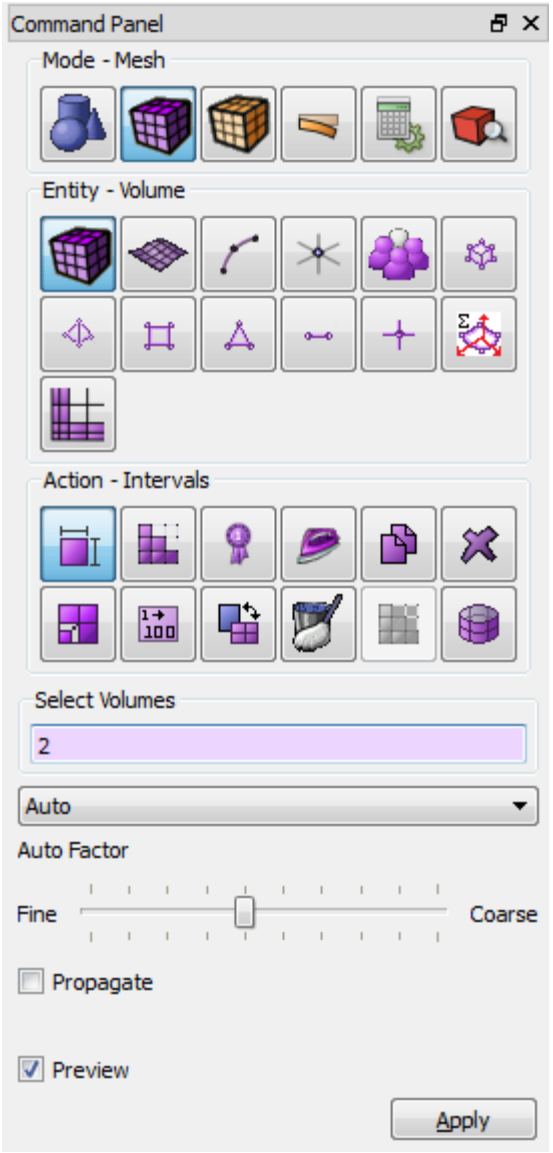
- Supported types of the finite elements for meshes:

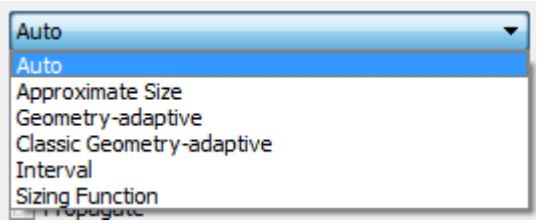
3D	2D	Shells	Beams
4-noded tetrahedron (TETRA/TETRA4), 10-noded tetrahedron (TETRA10)	3-noded triangle (TRI/TRI3), 6-noded triangle (TRI6)	3-noded triangle (TRISHELL/TRISHELL3), 4-noded quadrilateral (SHELL/SHELL4)	2-noded beam (BEAM/BEAM2)
8-noded hexahedron (HEX/HEX8), 20-noded hexahedron (HEX20), 27-noded hexahedron (HEX27)	4-noded quadrilateral (QUAD/QUAD4), 8-noded quadrilateral (QUAD8), 9-noded quadrilateral (QUAD9)	6-noded triangle (TRISHELL6), 8-noded quadrilateral (SHELL8), 9-noded quadrilateral (SHELL9)	3-noded beam (BEAM3)
5-noded pyramid (PYRAMID/PYRAMID5), 13-noded pyramid (PYRAMID13)			
6-noded wedge (WEDGE/WEDGE6), 15-noded wedge (WEDGE15)			

CAE Fidesys interface

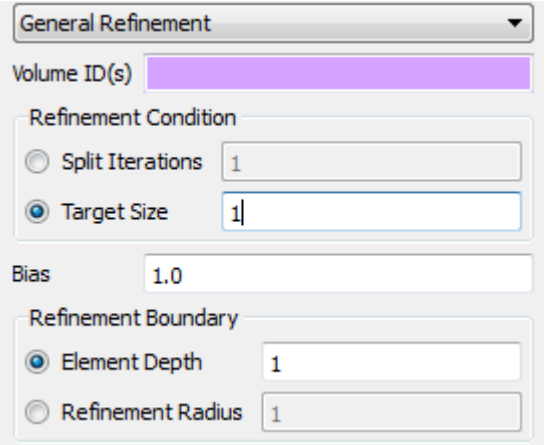
Meshing

- Adjustable degree of mesh refinement (interval);
- Several meshing algorithms:
  - Automatic
  - Specifying approximate element size
  - Geometry-adaptive
  - Interval meshing
- Meshing schemes and attributes;
- Mesh refinement;
- Mesh optimization and smoothing.





Specify meshing schemes and attributes

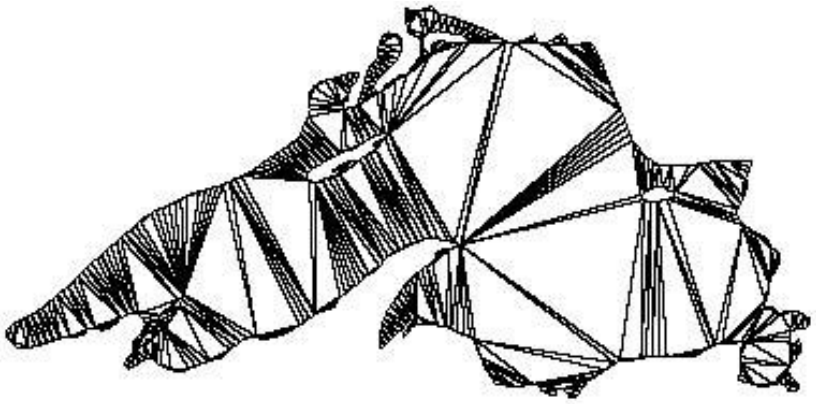


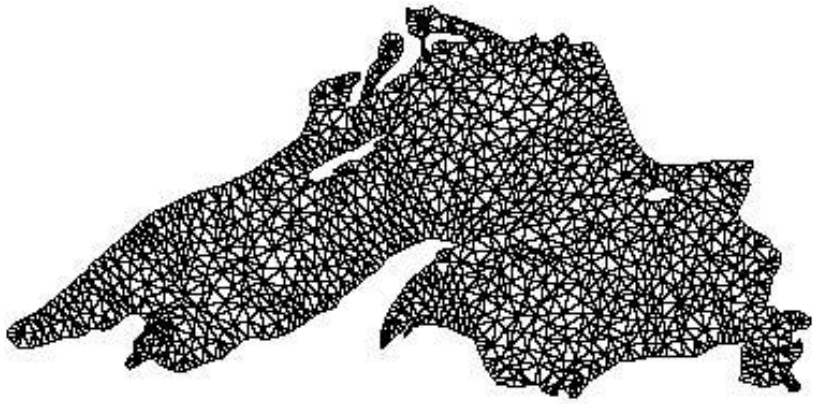
Mesh refinement

CAE Fidesys interface

Generation of irregular mesh and quality control

- Mesh refinement in accordance to user-defined metric

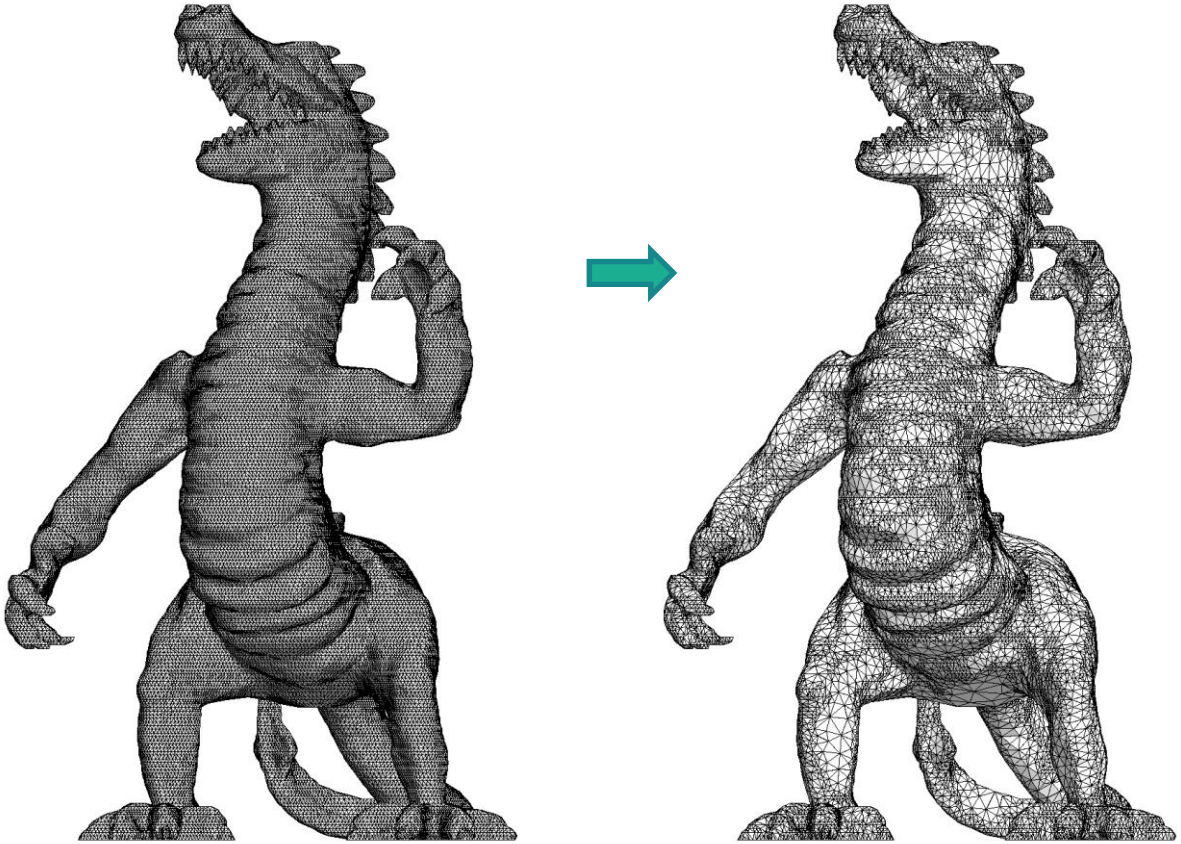




CAE Fidesys interface

Generation of irregular mesh and quality control

- 3D-mesh refinement

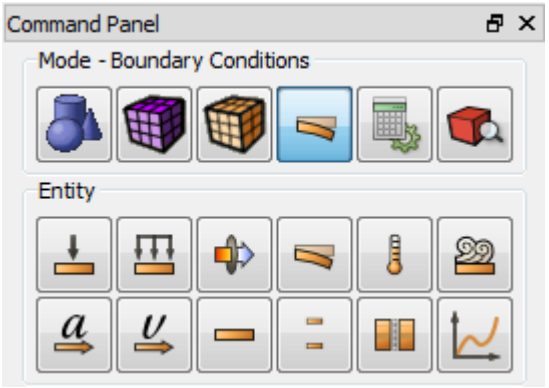


CAE Fidesys interface

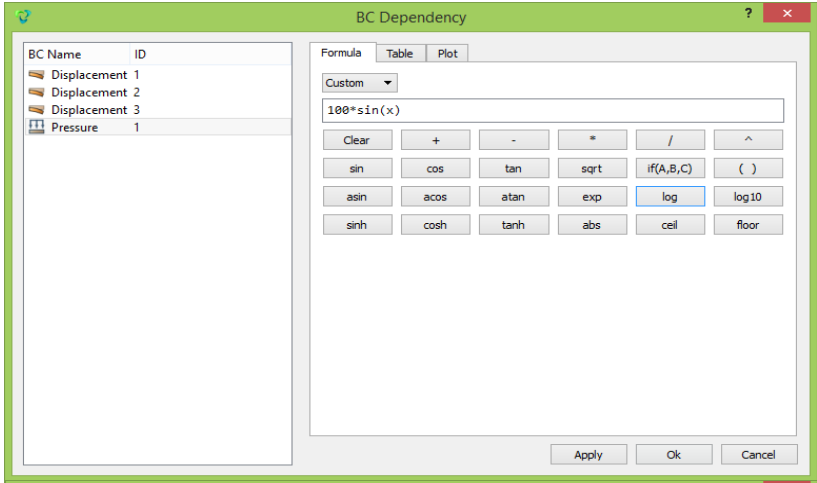
Boundary Conditions

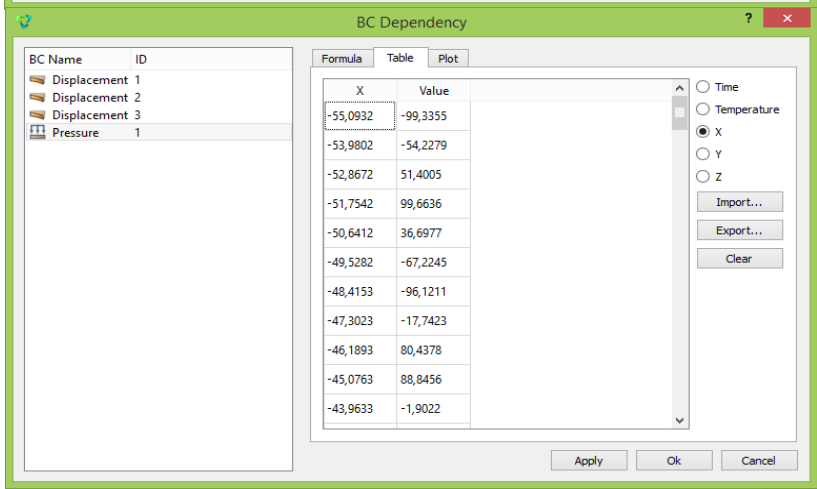
Supporting different types of BCs:

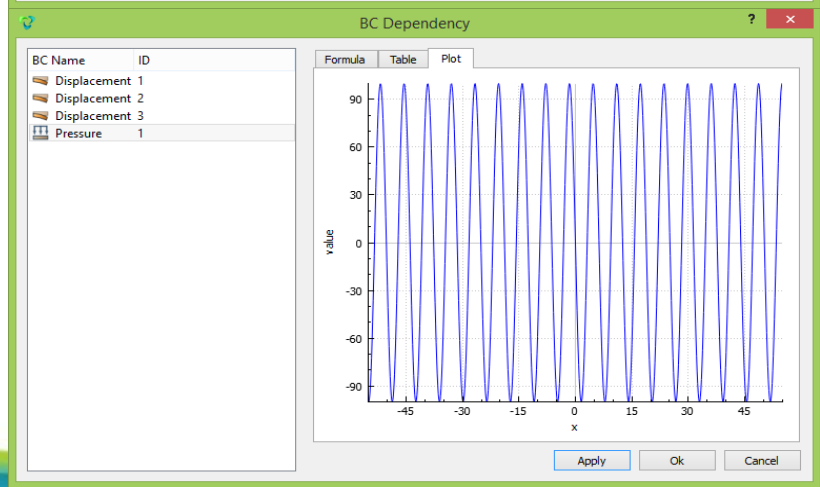
- Force;
- Pressure;
- Heat flux;
- Displacement;
- Temperature;
- Convection;
- Acceleration;
- Velocity;
- Contact pair;
- Rigid connection.



Mode – Boundary Conditions



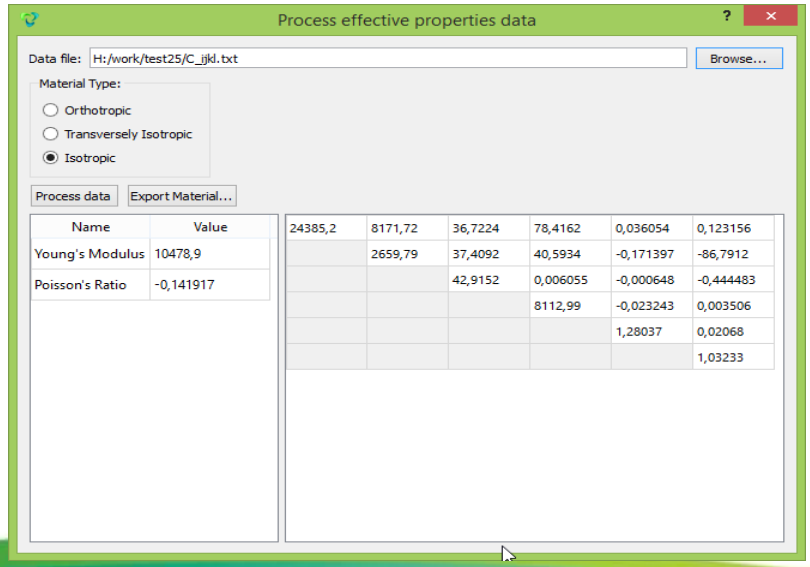


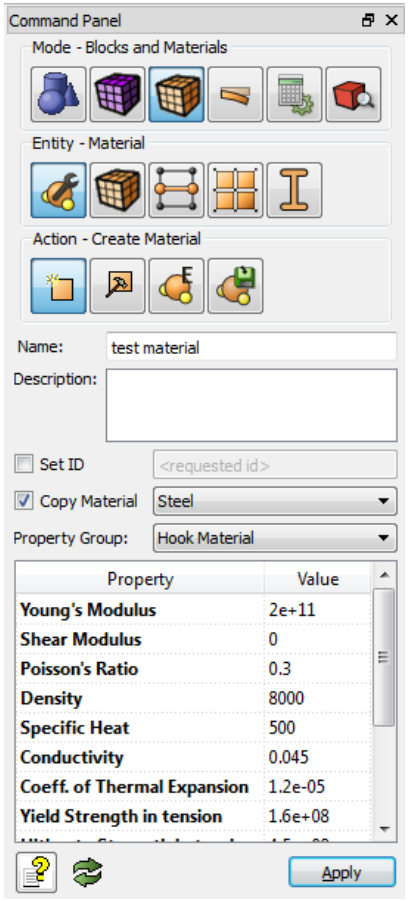


CAE Fidesys interface

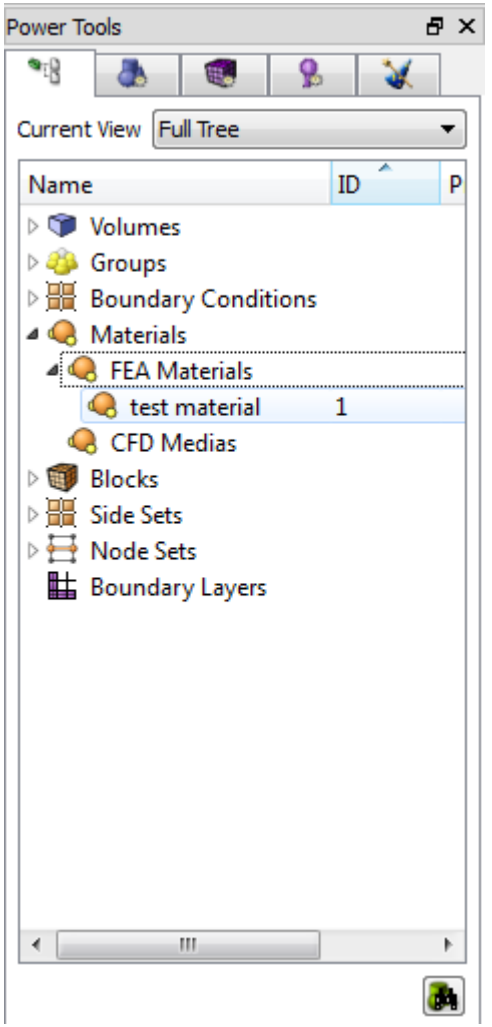
Materials

- Physical models:
  - Linear isotropic material (Hooke's material);
  - Composite materials, concrete, geomechanical samples;
  - Rubber materials;
  - Steel, plexiglass, copper, etc.
- Materials library;





Material generation window

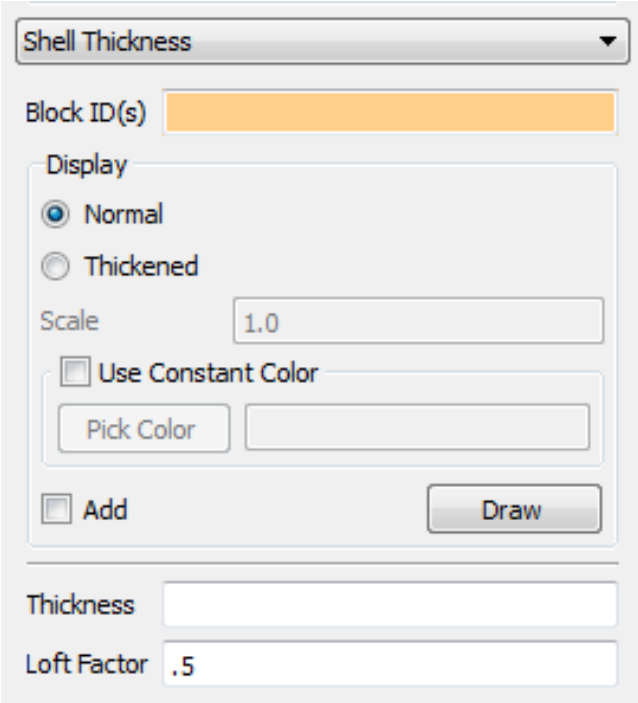


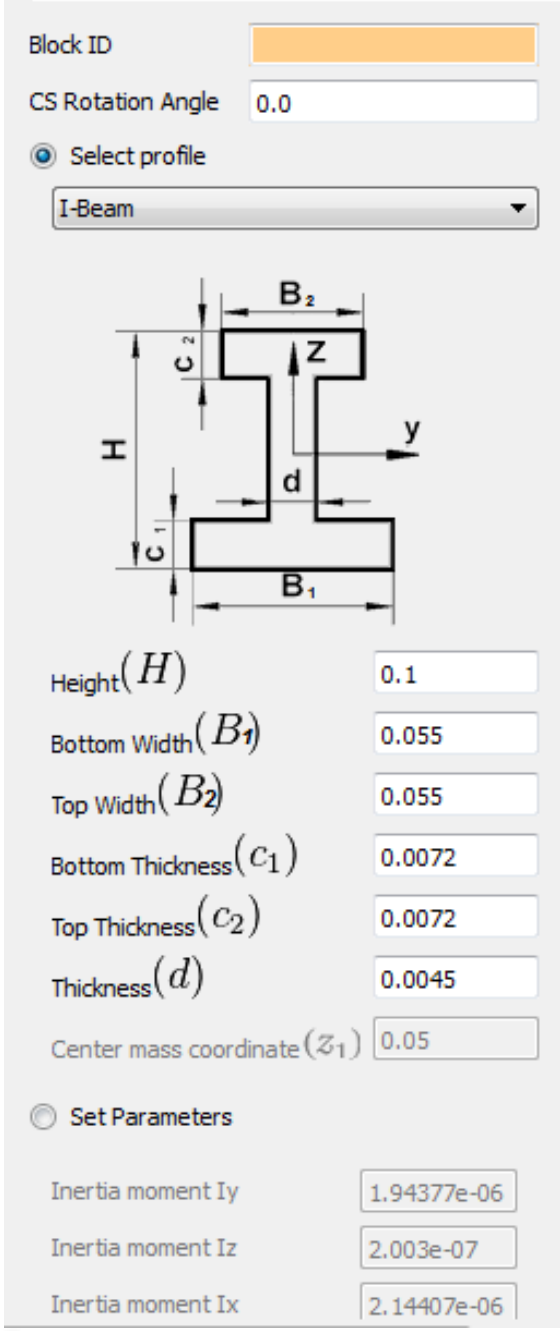
Materials and blocks in the Object Tree

CAE Fidesys interface

Beams and shells

- Thickness and Loft Factor for shells;
- Beam cross section properties;

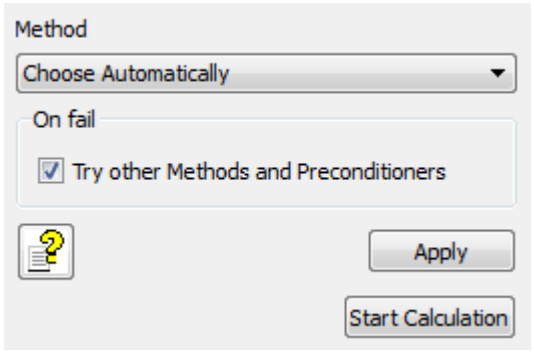


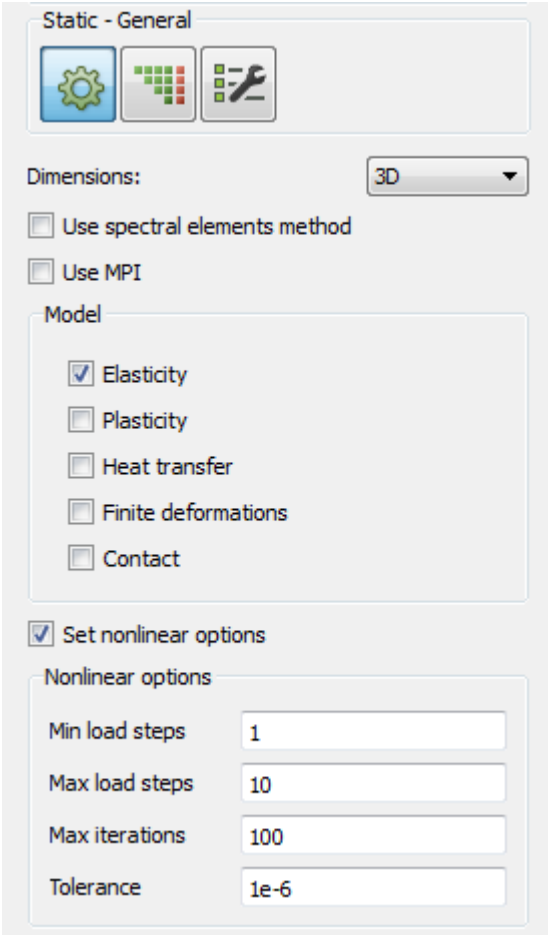


CAE Fidesys interface

Calculation parameters

- Analysis types:
  - Static;
  - Dynamic (transient);
  - Modal;
  - Harmonic;
  - Buckling;
  - Effective properties of composites.
- Solvers:
  - Finite element method (by default);
  - Spectral element method.
- Solution progress in the console.

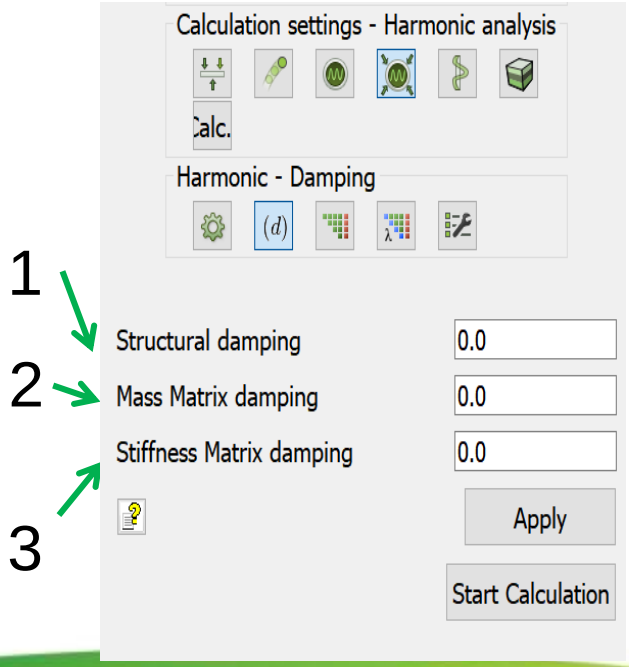


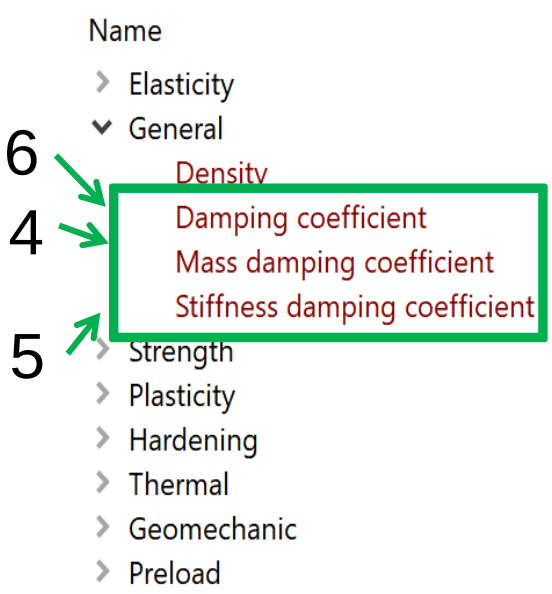


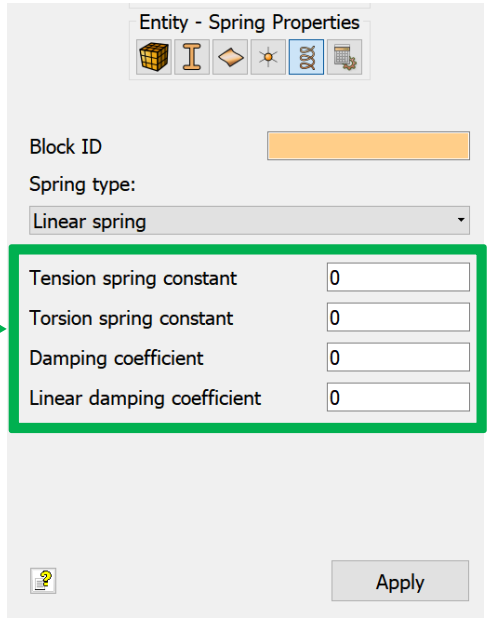
Harmonic analysis: damping parameters

$$[C] = \alpha[M] + \sum_{j=1}^{N_m} \alpha_j^m [M_j] + (\beta + \beta_c)[K] + \sum_{j=1}^{N_m} \left[\left(\beta_j^m + \frac{2}{\Omega} \beta_j^\zeta \right) [K_j] \right] + \sum_{k=1}^{N_m} [C_k]$$

2 4 3 1 5 6 7



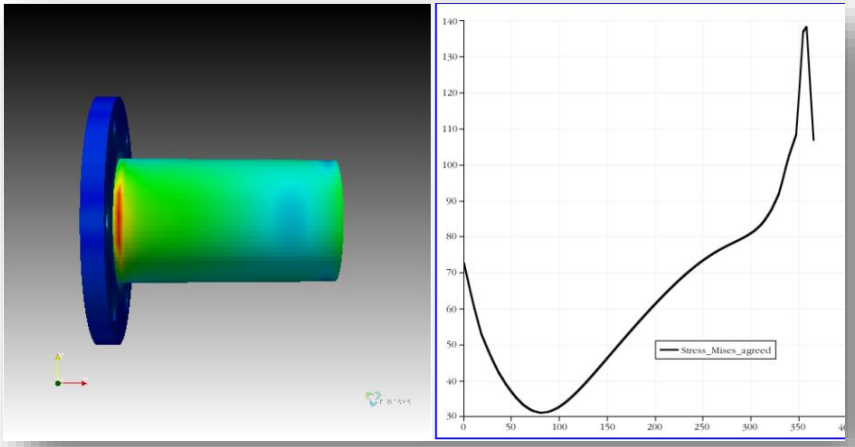


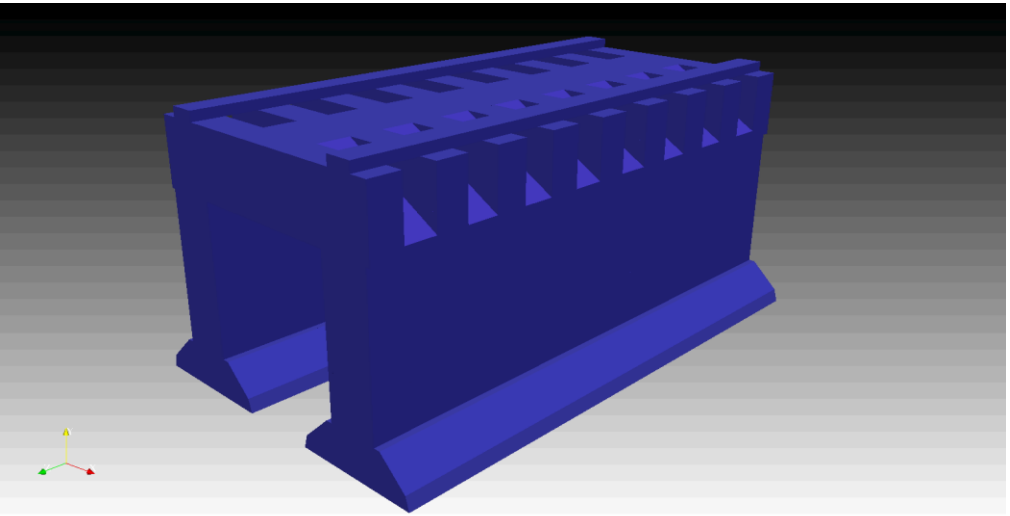


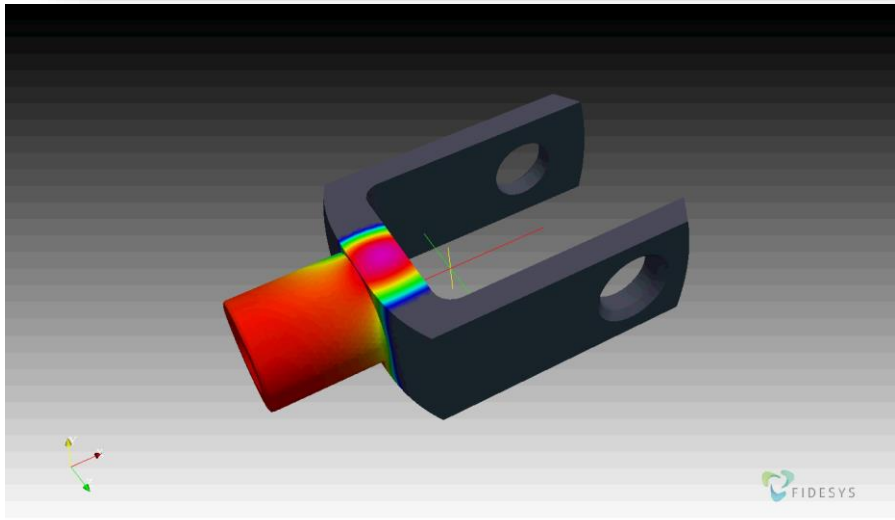
CAE Fidesys interface

Results Visualization and Postprocessing

- Visualization and analysis of the obtained results:
  - Visualization of vector and tensor fields;
  - 1D plots, export to CSV;
  - Time dependency analysis.
- Evaluation of the mesh quality
- Slice and cross section views
- Data export

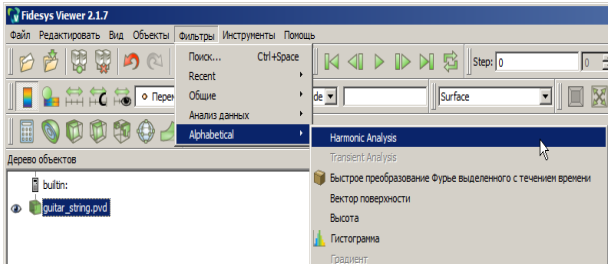


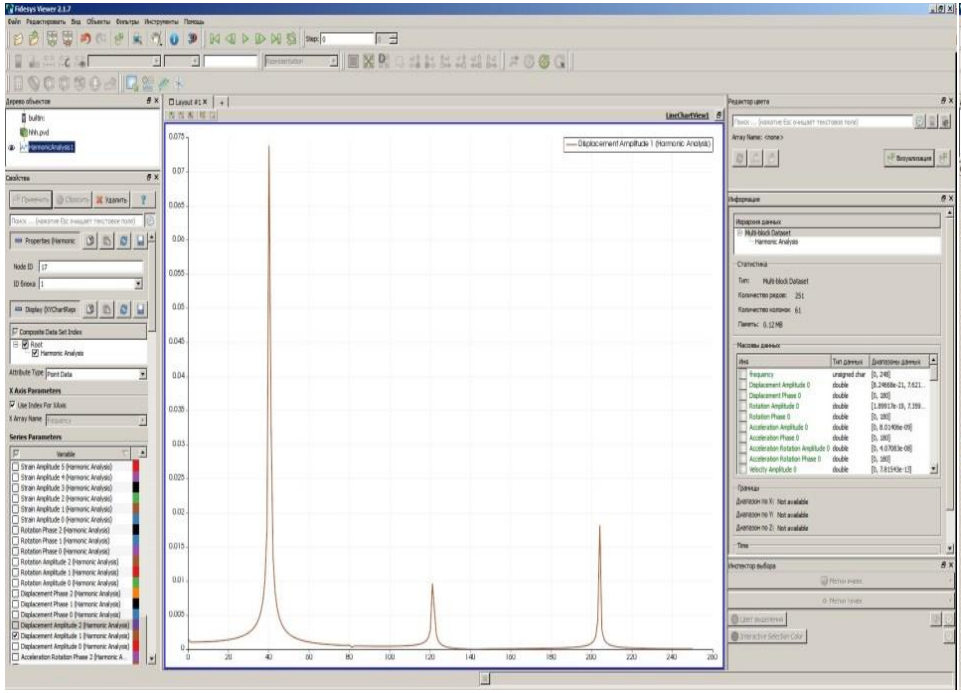


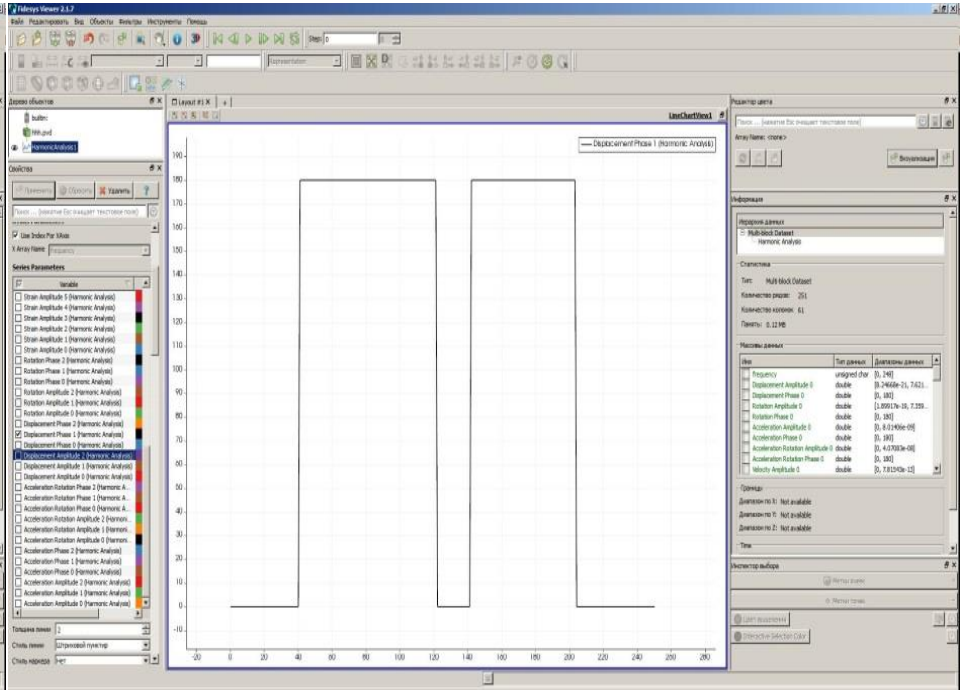


Harmonic analysis

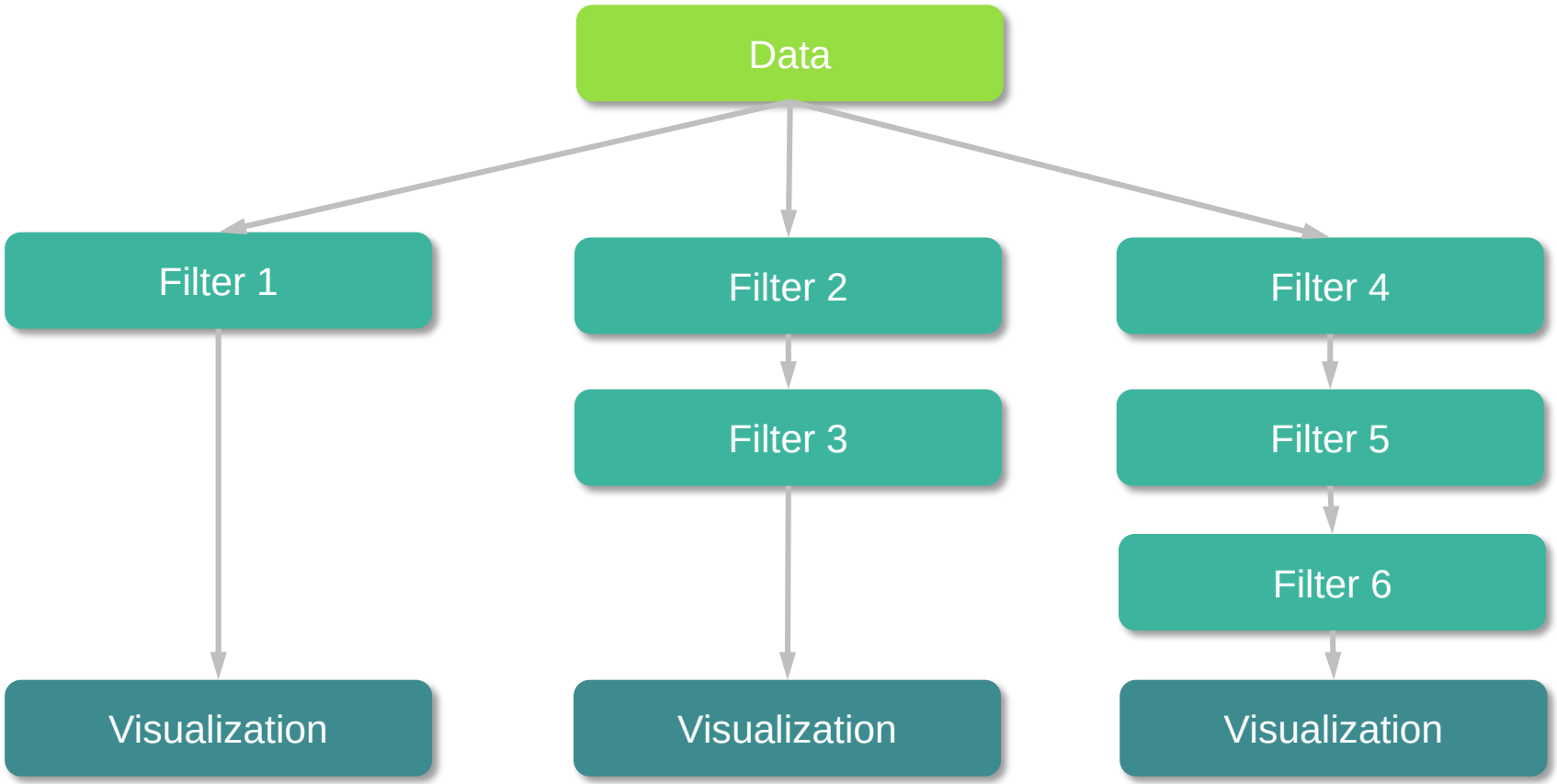
Visualization of amplitudes and phases for displacements, velocities, stresses etc.





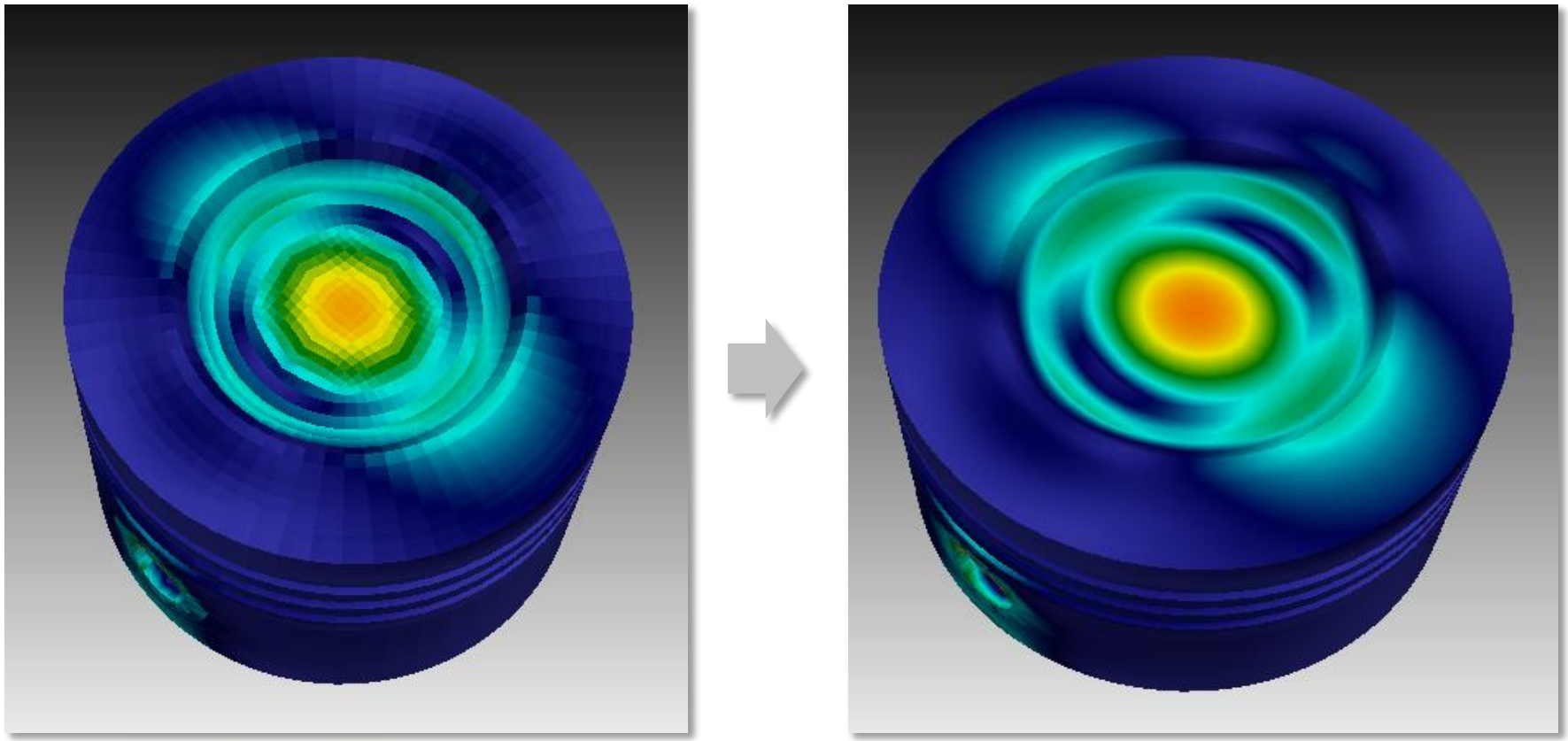


Visual filters



Smoothing the obtained results

Physically consistent stresses and strains



Safety margins

- The first theory of strength $n = \frac{\sigma^+}{\sigma_1}$
- Energy theory $n = \frac{\sigma_T}{\sigma_i}$
- Pisarenko-Lebedev theory

$$n = \frac{\sigma^+}{\chi\sigma_i + (1-\chi)\sigma_1}, \quad \text{where} \quad \chi = \frac{\sigma^+}{\sigma^-}$$

- Mohr's theory $n = \frac{\sigma^+}{\sigma_1 - \chi\sigma_3}$, where $\chi = \frac{\sigma^+}{\sigma^-}$
- Tresca theory $n = \frac{\sigma^+}{\sigma_1 - \sigma_3}$

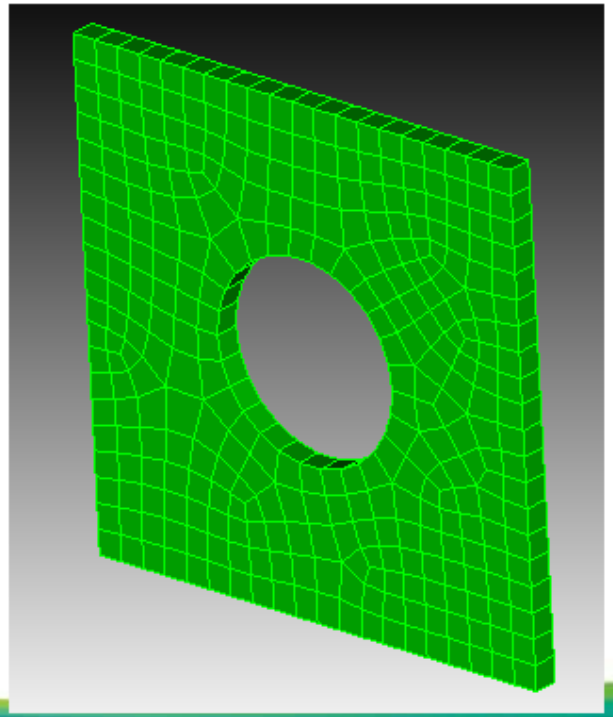
σ_1 , σ_2 , σ_3 — principal stresses
σ_i — stress intensity (von Mises)
σ^+ , σ^- — tensile and compression strength
σ^T — yield strength
n — safety margin

Fidesys scripting language

Fidesys can be controlled using the commands:

- Commands in special language are generated in GUI and then transferred to the command handler
- Command history is written to a ***.jou** file
- Commands are displayed in the command console window
- One can program using this command language

```
bri x 10
cyl radius 2 z 12
subtract vol 2 from vol 1
vol 1 size .5
vol 1 scheme auto
mesh vol 1
draw surf in vert 1
draw hex in node in surf 1
```



Aprepro

- Aprepro = Algebraic Pre-Processor
- Integrated programming language:
  - Parametrization of the journal files
  - Error control
  - Logical control

```
#{i=1}
#{size=10}
#{loop(50)}
brick x {size}
move volume {i} location {size*i} 0 0
#{i++}
#{endloop}
```

Python - widely used scripting language.

Lots of software use it for automation, eg.:

- *Abaqus*
- *Paraview*
- *PyTrilinos*

Installer and full documentation are available on the official website www.python.org.

Example

```
# Results file open

from vtk import *

reader = vtkXMLUnstructuredGridReader()
filename = os.path.abspath(os.path.join(os.getcwd(), “C:/result/result.vtu”))
reader.SetFileName(filename)
reader.Update()
grid = vtkUnstructuredGrid()
grid = reader.GetOutput()

#  Stress intensity calculation (based on Mises)

stress = grid.GetPointData().GetArray("Stress")
maxmises = -1.0

for i in xrange(stress.GetNumberOfTuples()):
    XX = stress.GetTuple(i)[0]
    YY = stress.GetTuple(i)[1]
    ZZ = stress.GetTuple(i)[2]
    XY = stress.GetTuple(i)[3]
    YZ = stress.GetTuple(i)[4]
    ZX = stress.GetTuple(i)[5]
    mises = math.sqrt( ( SQUARE(XX - YY) + SQUARE(YY - ZZ) + SQUARE(ZZ - XX) +
                    6 * ( SQUARE(XY) + SQUARE(YZ) + SQUARE(ZX) ) ) * 0.5)
    if maxmises < mises:
        maxmises = mises
print maxmises
```

Advantages of CAE Fidesys

- High speed and accuracy of calculations
- Flexible & adaptive geometry-mesh generator
- Wide range of supported CAD-formats
- Cross-platform (OS Windows, Linux)
- Low system requirements
- Low price compared to other solutions
- Cloud version - Fidesys online (SaaS)

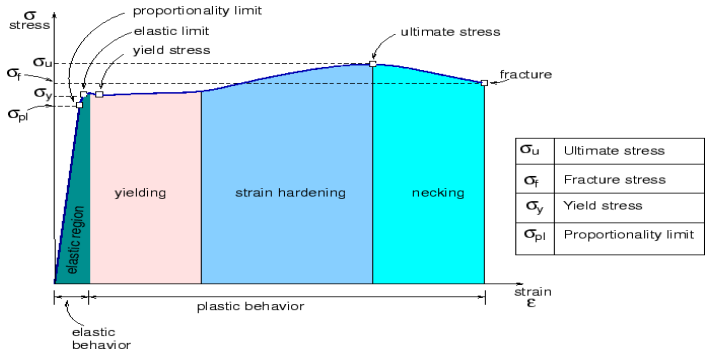
Custom software development

- On the basis of CAE Fidesys's software modules, a custom corporate or industry-specific software is developed (e.g. Fidesys Geomechanics for NTC Gazpromneft).
- The customized package will be more functional and easy-to-use than the general purpose CAE. It is focused on specific problems of the client.
- Development cycle of a custom software takes about 6-18 months depending on the customer's specifications with the involvement of leading industry consultants.
- As a result, the customer obtains a dedicated corporate product.
- **The package can be used as a simulation software which is capable for fine tuning and extensively applied during R&D stage.**

Simulation Designed for Design

Why aren't more companies using Simulation as an integral part of Design?

- Simulation is often considered too much to be used for influencing design decision
  - Too difficult - Too complex - Too much expertise required - Too expensive - Too compute intensive
- Just too much?



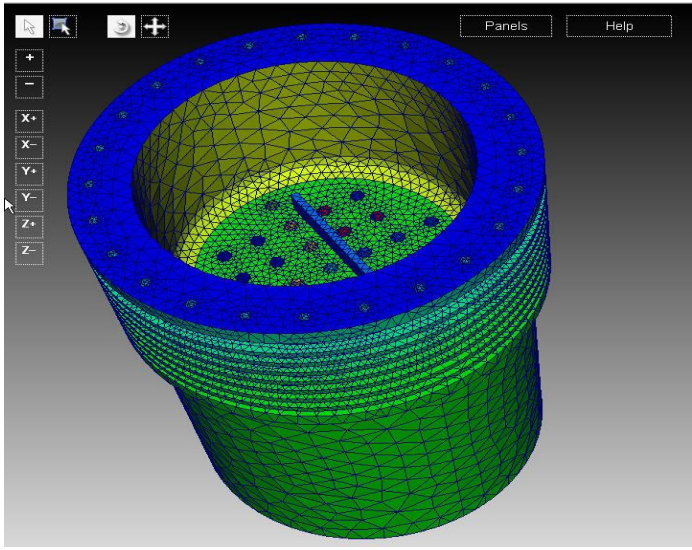
Simulation Designed for Design is now available!

- Simulation is no longer too much

SimForDesign - Released in January 2015

Streamlined simulation for design

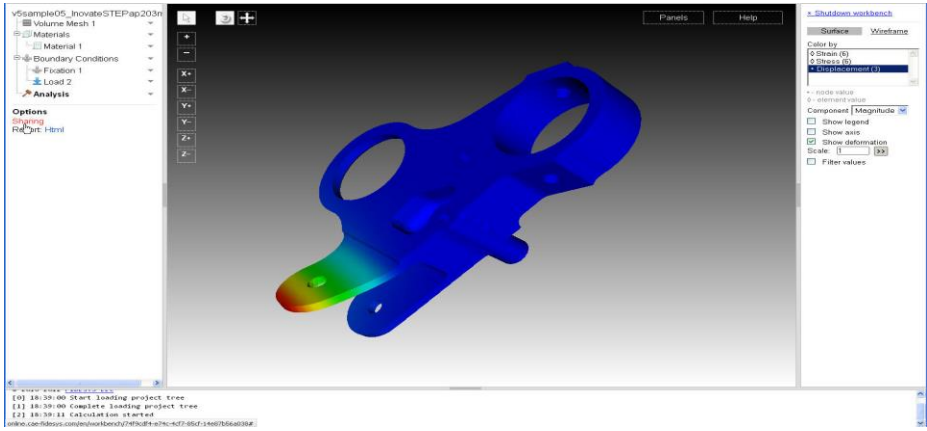
- Making simulation accessible to designers and other non-simulation specialists
- Assess parts and simple assemblies
  - Single material and linear behavior
- Web browser based analysis
  - No software to license or install
- Pre-Paid Usage pricing



Simulation Designed for Design

Streamlined simulation process

- Upload a STEP or IGES file
  - CAD plug-ins coming
- Mesh created automatically
- Define Material
- Assign Boundary Conditions
- Run Calculation
- Visualize Results in Web browser
- Generate a report

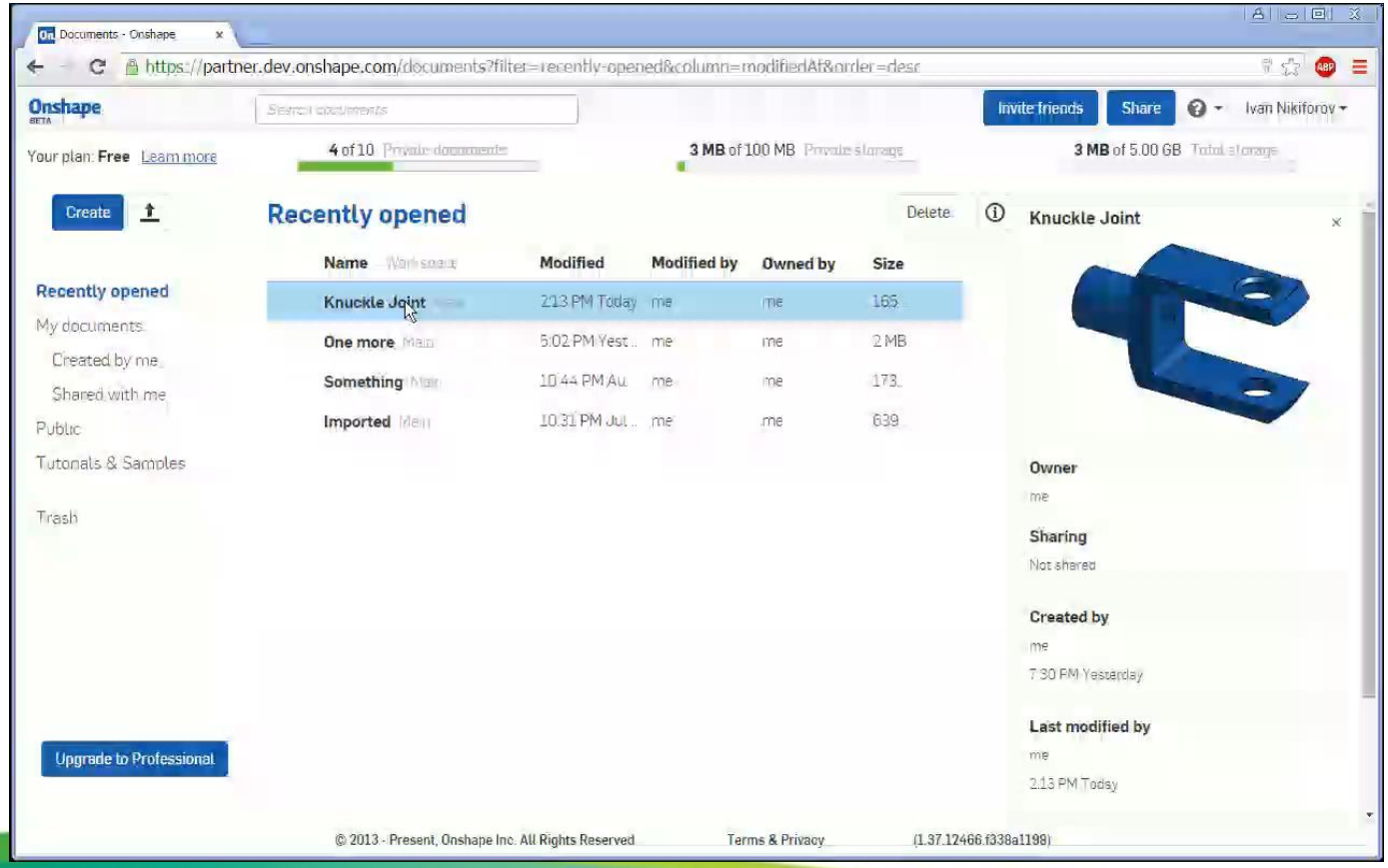


Pay-as-you-go pricing model

Free	Standard	Premium	Professional	Corporate
$0.00	$10.00	$50.00	$250.00	$500.00
1 User	1 User	1 User	1 User	Multiple Users
0 CPU Hours	1 CPU Hours	10 CPU Hours	60 CPU Hours	150 CPU Hours
0 GB Storage	1 GB Storage	10 GB Storage	100 GB Storage	1000 GB Storage
Free Examples Project	5 Projects (+ Free Examples)	10 Projects (+ Free Examples)	Unlimited Projects (+ Free Examples)	Unlimited Projects (+ Free Examples)
Sign Up Now!	Sign Up Now!	Sign Up Now!	Sign Up Now!	Sign Up Now!

Fidesys Online – cloud CAE

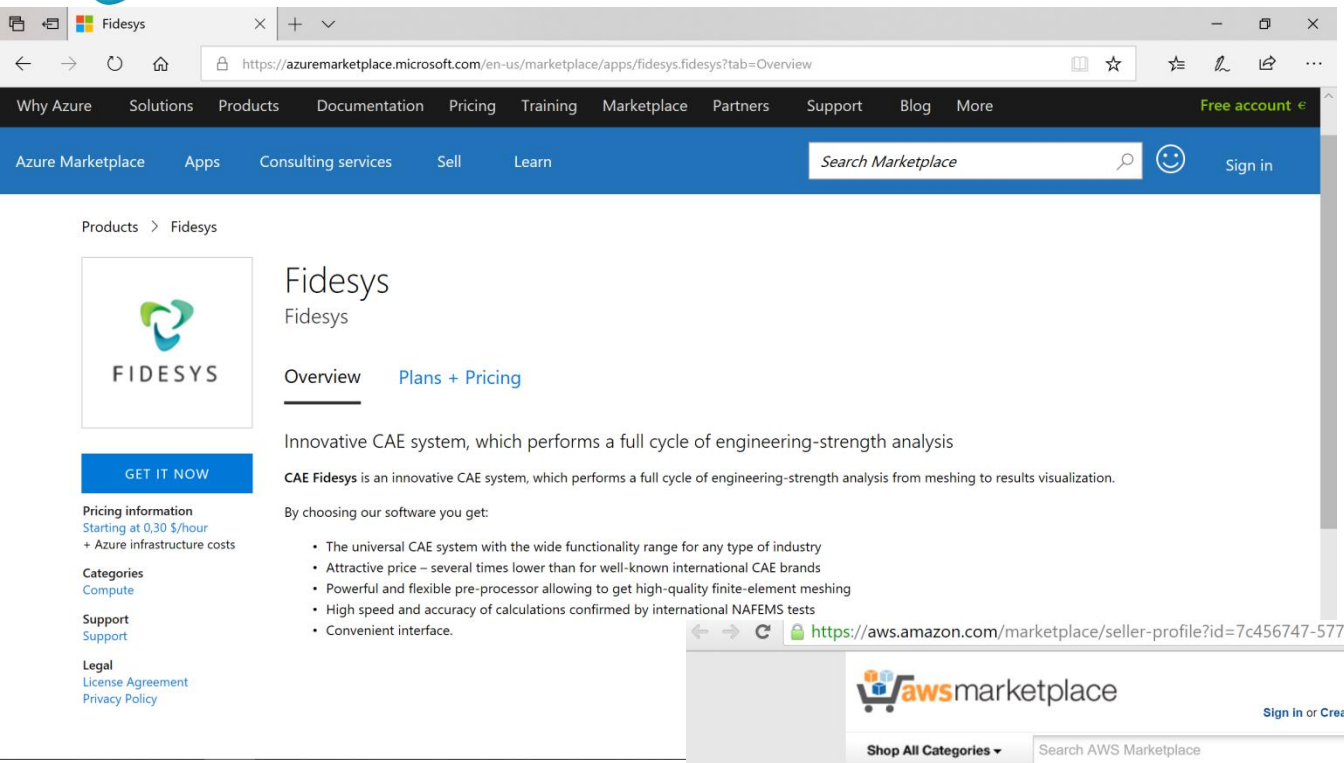
- Available 24/7 in your browser at *http://sim4design.com*
- All calculations, renderings etc are done by the cloud
- Collaboration and joint project analysis



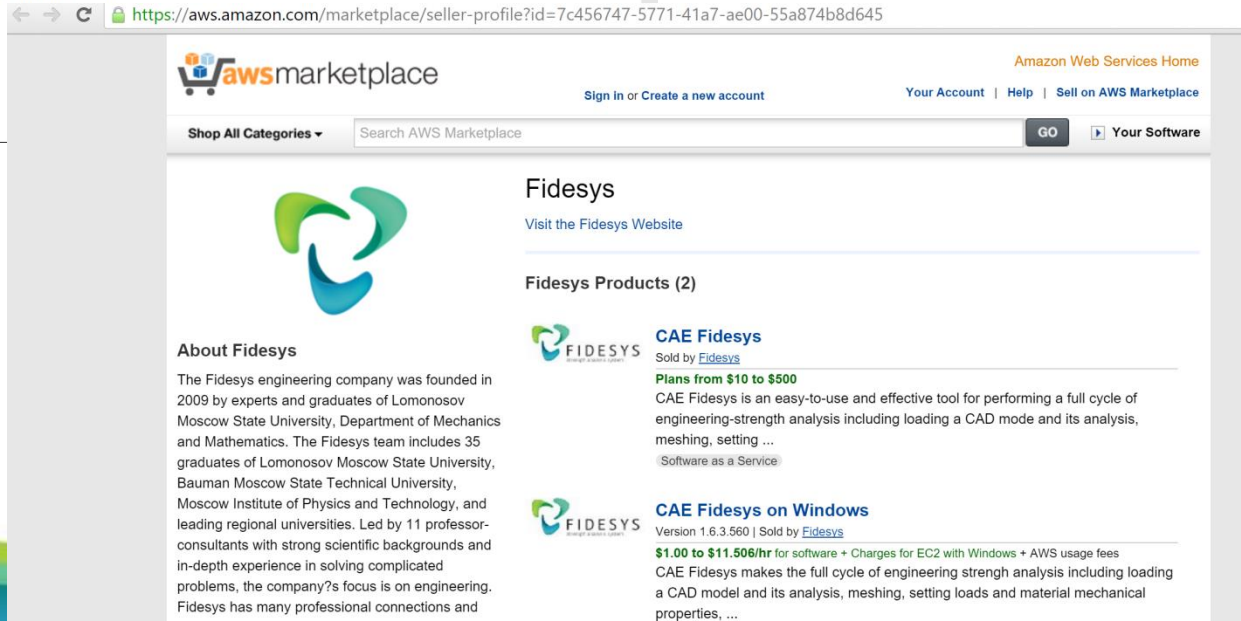
Corporate CAE-platform

- Private cloud solution for the client's internal usage
- Available from any device connected to internal Ethernet/VPN
- Cost reduction: cost of ownership, support, security

Fidesys in Azure/Amazon marketplaces



Preprocessing, simulation, analysis, rendering etc. are done in the cloud. 24/7 access in the browser.

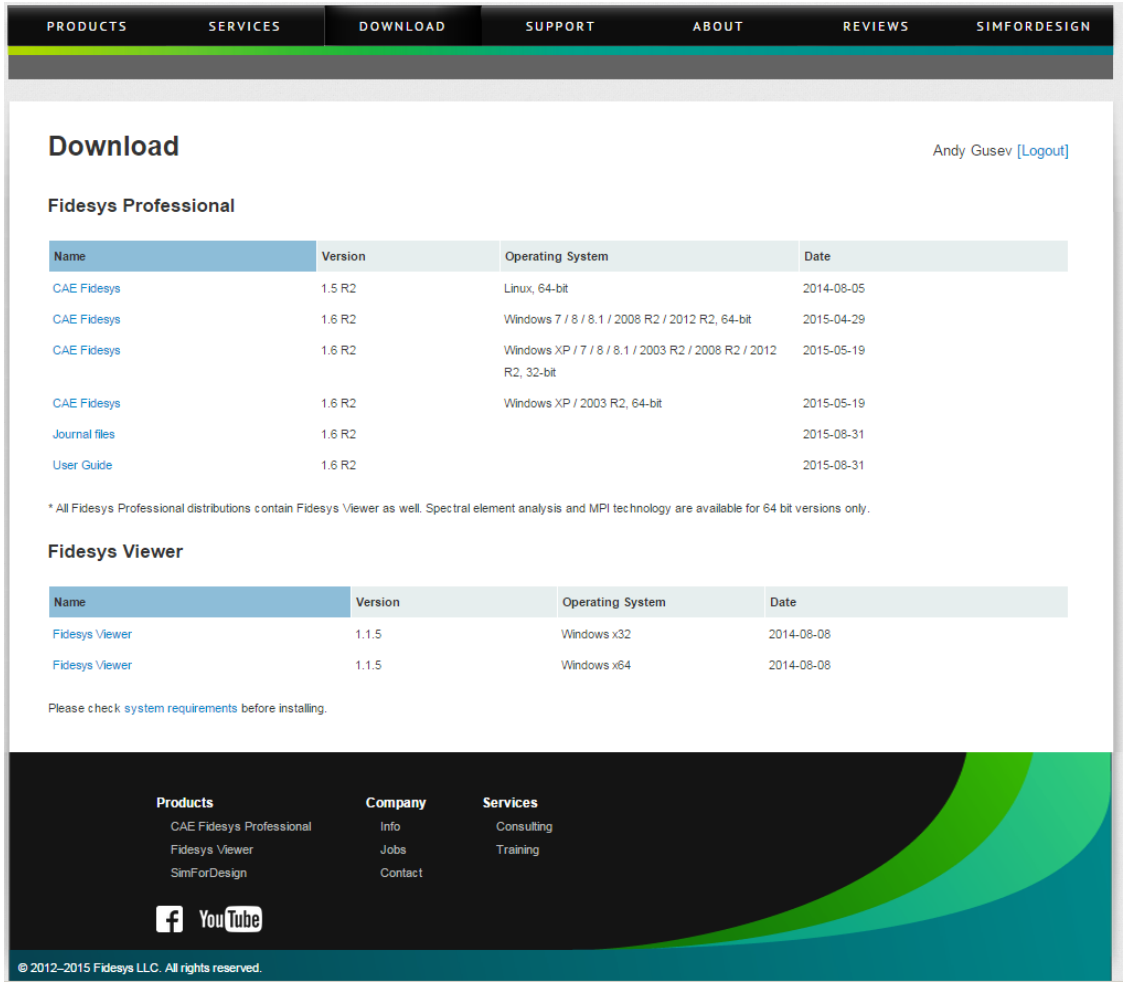


Pay-as-you go and subscription based licensing models flexible for different problem sizes.

CAE Fidesys trial version

http://www.cae-fidesys.com/

- 30 days
- Fully functional
- Step by step examples (+scripts)
- Test report
- Windows/Linux 32/64
- Fidesys Viewer (free of charge!)



Thank you!

Anatoly Vershinin, CTO
Email: a.v.vershinin@cae-fidesys.com
Phone: +7 (916) 184-49-28
WWW: www.cae-fidesys.com