FIDESYS

strength analysis system

Version 4.1

Verification manual

Contents

Introduction

About the software

CAE Fidesys is a software package for strength analysis. The package comprises the following types of analysis:

- Static loading;
- dynamic (transient) loading;
- buckling ;
- analysis of natural frequencies;
- harmonic analysis;
- calculation of effective material properties;
- mod combination based on spectral analysis;
- топологической оптимизации моделей;
- Automechanic analysis.

The package also includes a program ***Fidesys Viewer*** for visualization and analysis of the obtained results:

- Visualization of scalar and vector fields;
- SEG-Y files visualization;
- building graphs and charts;
- building frequency dependencies ;
- time dependency analysis.

General

CAE Fidesys is an innovative CAE-system that performs a full cycle of engineering calculations, from the construction of the computational grid to the visualization of the calculation results.

CAE Fidesys is is continuously being verified by the developers as new features are added. These verification are performed in accordance with procedures that are part of ***CAE Fidesys***' overall quality assurance program. This ***CAE Fidesys 4.1*** test verification manual presents a small subset of QA test cases that are used to test new features. Test cases are comparisons of ***CAE Fidesys*** solutions with analitical solutions and other independently calculated solutions.

The presented test cases are selected in such a way as to validate different problem areas, types of loads, boundary conditions corresponding to the new featches and the statements of work of ***CAE Fidesys 4.1***.

Result Comparison

Each test case verifies a specific set of parameters. Also, for each test case, the expected result is given,which is considered as target. The test case is considered to be successful if the relative error of the calculation results compared to the reference does not exceed 5%. The relative error is calculated by theformula:

$$\Delta = \left| \frac{P - P_0}{P_0} \right| \cdot 100\% ,$$

Where Δ is the value of the relative error of the parameter; P is the calculated in ***CAE Fidesys*** value of the parameter; P_0 is the expected value of the parameter.

System requirements

CAE Fidesys has low system requirements for the package. It can be run on an ordinary personal computer. If the computer has one or more multi-core processors, calculations are automatically parallelized on all cores. Starting with version 1.5, calculation parallelization to several nodes connected to a local network or a cluster is available in the 64-bit version of the program package.

CAE Fidesys software package has following minimal requirements for software and hardware:

Hardware requirements

CPU: Dual-core 1,7 GHz minimum
RAM: 4GB minimum
Free hard drive space: 5 GB
Video card NVIDIA GeForce GTX 460 or faster
Screen resolution: 1024x768 or higher

Operating system

Following operating systems are supported. (for the 64-bit versions)

- Windows 7 Service Pack 1;
- Windows 8;
- Windows 8.1;
- Windows Server 2008 R2 SP1;
- Windows Server 2008 Service Pack 2;
- Windows Server 2012;
- Windows Server 2012 R2;
- Windows 10;
- Ubuntu 18.04;
- CentOS 6;
- CentOS 7;
- Debian 9;
- RedHat 6;
- RedHat 7;
- Open SUSE Leap 15;
- Alt Linux 7;
- Alt Linux 8.

1. Test cases with analytical solutions

1.1. Test Case №1.1

Problem Description

Determination of effective mechanical characteristics for an orthogonally reinforced composite.

Input Values

Material Properties:

Matrix material

- Isotropic;
- Young's modulus = 1 Pa;
- Poisson ratio = 0.25;
- Thermal conductivity coefficient = 2 $\frac{W}{m*K}$.

Thread material:

- Isotropic;
- Young's modulus = 1 Pa;
- Poisson ratio = 0.25;
- Thermal conductivity coefficient = 10 $\frac{W}{m*K}$.

Geometric model:

- Two cubes 16 x 16 x 16, adjacent to each other along the Z axis;
- Thread of length 16 and radius of 2.85459861019 (selected so the thread volume concentration in composite is equal to 10%) runs through center line parallel to the X axis for the first cube and through center line parallel to the Y axis for the second cube;
- Thread: λ = 10;
- Matrix: λ = 2.

Boundary conditions:

- Periodic.

<u>Mesh:</u>

- First order tetrahedrons.

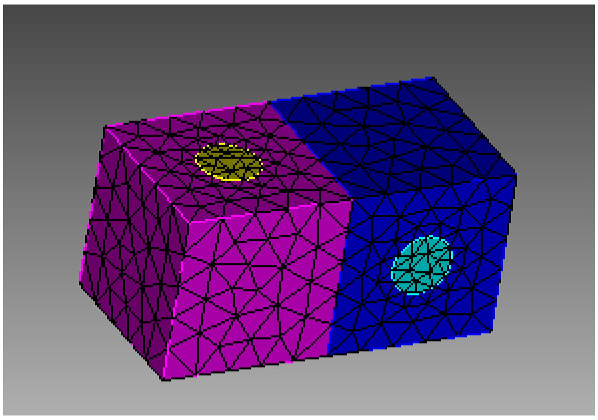

Fig 1 – Tetrahedral mesh

Target results

№	Value	Description	Unit	Target
1	Effective thermal conductivity coefficients	λ_11	$\frac{W}{m*K}$.	2.54285
2	Effective thermal conductivity coefficients	λ_22	$\frac{W}{m*K}$.	2.54285
3	Effective thermal conductivity coefficients	λ_33	$\frac{W}{m*K}$.	2.17647

Analytical solution description

Orthogonally reinforced composite is a composite that for one fiber along Y axis has k fibers along X axis. Analytical solution taken from 1.6.2 paragraph of Karpinos's "Composite materials". Thermal conductivity coefficient along X axis for such composites determined by following formula:

$$\lambda_x^{ort} = \lambda_x \frac{k}{k+1} + \frac{\lambda_y}{k+1} = \frac{1}{k+1}(\lambda_x k + \lambda_y)$$

along Y axis – by formula

$$\lambda_y^{ort} = \frac{\lambda_x}{k+1} + \lambda_y \frac{k}{k+1} = \frac{1}{k+1}(\lambda_x + \lambda_y k)$$

Here λ_x, λ_y determined by formulas for fibrous material.

Taking same fiber count along X and Y axis

$$\lambda_x^{ort} = \lambda_y^{ort} = \frac{\lambda_x + \lambda_y}{2}$$

Boundary conditions - only periodic.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficients	λ_11	$\frac{W}{m*K}$.	2.54285	2.524	-0.73%
2	Effective thermal conductivity coefficients	λ_22	$\frac{W}{m*K}$.	2.54285	2.525	-0.69%
3	Effective thermal conductivity coefficients	λ_33	$\frac{W}{m*K}$.	2.17647	2.2705	4.14%

CAE Fidesys script :

```
reset
#{length = 16.0}
#{pitch = 16.0}
#{thick = 16.0} # thickness
#{conc = 10} # cord concentration, percents
#{rad = sqrt( 0.01 * pitch * thick * conc / 3.1415926 )}
#{size = 3.0}
create brick width {length} depth {pitch} height {thick}
create cylinder height {length} radius {rad}
volume 2 rotate 90.0 about y
subtract volume 2 from volume 1 keep
delete volume 1
move volume all z {-thick/2.0} include_merged
volume all move z {thick} copy
rotate volume 2 3 angle 90 about z include_merged
imprint volume all
merge volume all
volume all scheme Tetmesh
volume all size {size}
mesh volume all
create material 1 name 'fiber'
modify material 1 set property 'MODULUS' value 1
```

```
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'ISO_CONDUCTIVITY' value 10
create material 2 name 'matrix'
modify material 2 set property 'MODULUS' value 1
modify material 2 set property 'POISSON' value 0.25
modify material 2 set property 'ISO_CONDUCTIVITY' value 2
block 1 volume 2 4
block 2 volume 3 5
block 1 material 'fiber'
block 2 material 'matrix'
block 1 2 element solid order 1
analysis type effectiveprops heattrans dim3
periodicbc on
```

Reference

[1] Вишняков Л.Р., Грудина Т.В., Кадыров В.Х., Карпинос Д.М., Олейник В.И., Сапожникова А.Б., Тучинский Л.И. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

[2] Кристенсен Р. Введение в механику композитов. – М.: «Мир», 1982. – 334 с.

[3] Победря Б.Е. Механика композиционных материалов. - М.: Издательство Московского университета, 1984. - 335 с.

1.2. Test Case №1.2

Problem Description

Determination of effective mechanical characteristics for a single layer fibrous composite.

Input Values

Material Properties:

Matrix material

- Isotropic;
- Young's modulus = 1 Pa;
- Poisson ratio = 0.25;
- Thermal conductivity coefficient = 2 $\frac{W}{m*K}$.

Thread material:

- Isotropic;
- Young's modulus = 1 Pa;
- Poisson ratio = 0.25;
- Thermal conductivity coefficient = 10 $\frac{W}{m*K}$.

Geometric model:

- Parallelepiped 4 x 16 x 16;
- Thread of length 25 and radius of 2.85459861019 (selected so the thread volume concentration in composite is equal to 10%) runs through center line parallel to the X axis;
- Thread: λ = 10;
- Matrix: λ = 2.

Boundary conditions:

- Periodic.

Mesh:

- First order tetrahedrons.

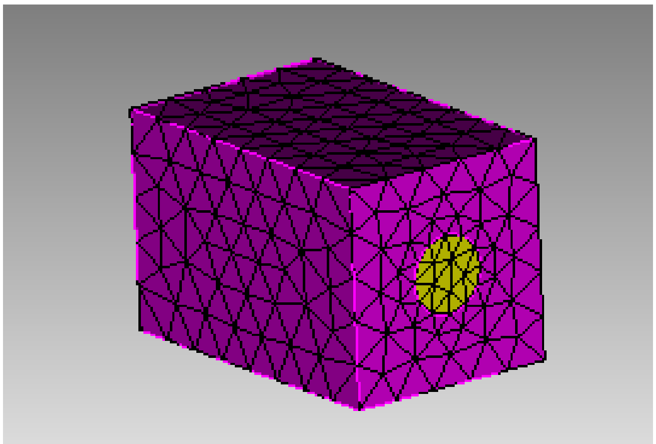

Fig 2 – Tetrahedral mesh

Target results

№	Value	Description	Unit	Target
1	Effective thermal conductivity coefficients	λ _11	$\frac{W}{m*K}$.	2.8
2	Effective thermal conductivity coefficients	λ _22	$\frac{W}{m*K}$.	2.28571
3	Effective thermal conductivity coefficients	λ _33	$\frac{W}{m*K}$.	2.28571

Analytical solution description

Analytical solution taken from 1.6.2 paragraph of Karpinos's "Composite materials". Effective thermal conductivity coefficients determined by following formula:

$$\lambda_x = \gamma_f \lambda_f + \gamma_m \lambda_m$$

$$\lambda_y = \lambda_z \approx \lambda_m \frac{1 + \gamma_f + \gamma_m \dfrac{\lambda_m}{\lambda_f}}{\gamma_m + (1 + \gamma_f)\dfrac{\lambda_m}{\lambda_f}}$$

Fibers here directed along X axis, λ_f, λ_m - thermal conductivity coefficients of thread and matrix, γ_f, γ_m - volume concentration of thread and matrix (in total they are equal to 1).

Boundary conditions - only periodic..

Results

First order tetrahedral mesh

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficients	λ _11	$\frac{W}{m*K}$.	2.8	2.773	-0.97%
2	Effective thermal conductivity coefficients	λ _22	$\frac{W}{m*K}$.	2.28571	2.2829	-0.12%
3	Effective thermal conductivity coefficients	λ _33	$\frac{W}{m*K}$.	2.28571	2.2917	0.26%

CAE Fidesys script:

```
reset
#{length = 25.0}
#{pitch = 16.0}
#{thick = 16.0} # thickness
#{conc = 10} # cord concentration, percents
#{rad = sqrt( 0.01 * pitch * thick * conc / 3.1415926)}
#{size = 3.0}
create brick width {length} depth {pitch} height {thick}
create cylinder height {length} radius {rad}
volume 2 rotate 90.0 about y
subtract volume 2 from volume 1 keep
delete volume 1
imprint volume all
merge volume all
volume all scheme Tetmesh
volume all size {size}
mesh volume all
create material 1 name 'fiber'
modify material 1 set property 'MODULUS' value 1
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'ISO_CONDUCTIVITY' value 10
```

create material 2 name 'matrix'

modify material 2 set property 'MODULUS' value 1

modify material 2 set property 'POISSON' value 0.25

modify material 2 set property 'ISO_CONDUCTIVITY' value 2

block 1 volume 2

block 2 volume 3

block 1 material 'fiber'

block 2 material 'matrix'

block 1 2 element solid order 1

analysis type effectiveprops heattrans dim3

periodicbc on

Reference

[1] Вишняков Л.Р., Грудина Т.В., Кадыров В.Х., Карпинос Д.М., Олейник В.И., Сапожникова А.Б., Тучинский Л.И. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

[2] Кристенсен Р. Введение в механику композитов. – М.: «Мир», 1982. – 334 с.

[3] Победря Б.Е. Механика композиционных материалов. - М.: Издательство Московского университета, 1984. - 335 с.

1.3. Test Case №1.3

Problem Description

Determination of effective mechanical characteristics for a single layer fibrous composite.

Input Values

Material Properties:

Matrix material

- Isotropic;
- Young's modulus = 2 Pa;
- Poisson ratio = 0.3;
- Thermal conductivity coefficient = $7.7 * 10^{-5} \frac{W}{m*K}$.

Thread material:

- Isotropic;
- Young's modulus = 2000 Pa;
- Poisson ratio = 0.2;
- Thermal conductivity coefficient = $1.3 * 10^{-5} \frac{W}{m*K}$.

Geometric model:

- Parallelepiped 25 x 16 x 16;
- Thread of length 25 and radius of 2.85459861019 (selected so the thread volume concentration in composite is equal to 10%) runs through center line parallel to the X axis;
- Thread: λ = 10;
- Matrix: λ = 2.

Boundary conditions:

- Periodic.

<u>Mesh:</u>

- Second order hexahedrons.

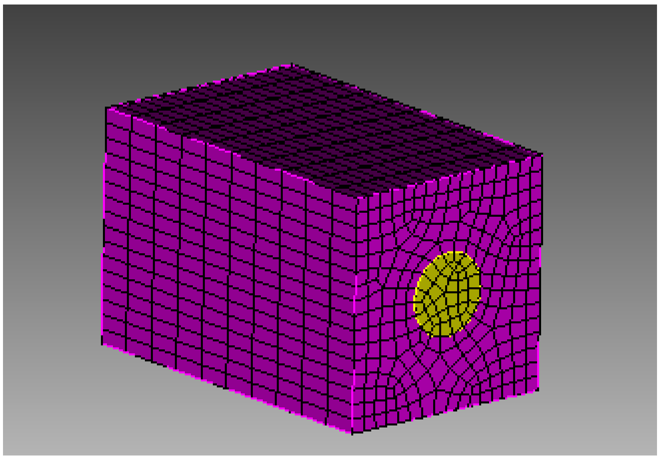

Fig 3 – hexahedral mesh

Target results

№	Value	Description	Unit	Target
1	Effective thermal conductivity coefficients	λ_11	$\frac{W}{m*K}$.	$1.35709 * 10^{-5}$
2	Effective thermal conductivity coefficients	λ_22	$\frac{W}{m*K}$.	$8.58878 * 10^{-5}$
3	Effective thermal conductivity coefficients	λ_33	$\frac{W}{m*K}$.	$8.58878 * 10^{-5}$

Analytical solution description

Analytical solution taken from 1.6.2 paragraph of Karpinos's "Composite materials". Effective thermal conductivity coefficients determined by following formula:

$$\lambda_x = \gamma_f \lambda_f + \gamma_m \lambda_m$$

$$\lambda_y = \lambda_z \approx \lambda_m \frac{1 + \gamma_f + \gamma_m \frac{\lambda_m}{\lambda_f}}{\gamma_m + (1 + \gamma_f)\frac{\lambda_m}{\lambda_f}}$$

Fibers here directed along X axis, λ_f, λ_m - thermal conductivity coefficients of thread and matrix, γ_f, γ_m - volume concentration of thread and matrix (in total they are equal to 1).

Boundary conditions - only periodic.

Results

Second order hexahedral mesh

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficients	λ _11	$\frac{W}{m*K}$,	$1.35709 * 10^{-5}$	$1.358 * 10^{-5}$	0.05%
2	Effective thermal conductivity coefficients	λ _22	$\frac{W}{m*K}$,	$8.58878 * 10^{-5}$	$8.308 * 10^{-5}$	3.27%
3	Effective thermal conductivity coefficients	λ _33	$\frac{W}{m*K}$,	$8.58878 * 10^{-5}$	$8.477 * 10^{-5}$	1.31%

CAE Fidesys script:

```
reset
set default element hex
#{length = 25.0}
#{pitch = 16.0}
#{thick = 16.0} # thickness
#{conc = 10} # cord concentration, percents
#{rad = sqrt( 0.01*pitch*thick*conc/3.1415926)}
#{size = 1.0}
create brick width {length} depth {pitch} height {thick}
create cylinder height {length} radius {rad}
volume 2 rotate 90.0 about y
subtract volume 2 from volume 1 keep
delete volume 1
imprint volume all
merge volume all
volume all size {size}
curve 18 20 22 24 interval 10
mesh volume all
create material 1 name 'fiber'
modify material 1 set property 'MODULUS' value 2000
modify material 1 set property 'POISSON' value 0.2
modify material 1 set property 'ISO_THERMAL_EXPANSION' value 1.3e-5
create material 2 name 'matrix'
```

```
modify material 2 set property 'MODULUS' value 2
modify material 2 set property 'POISSON' value 0.3
modify material 2 set property 'ISO_THERMAL_EXPANSION' value 7.7e-5
block 1 volume 2
block 2 volume 3
block 1 material 'fiber'
block 2 material 'matrix'
block all element solid order 2
analysis type effectiveprops heatexpansion dim3
periodicbc on
```

Reference:

[1] Вишняков Л.Р., Грудина Т.В., Кадыров В.Х., Карпинос Д.М., Олейник В.И., Сапожникова А.Б., Тучинский Л.И. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

[2] Кристенсен Р. Введение в механику композитов. – М.: «Мир», 1982. – 334 с.

[3] Победря Б.Е. Механика композиционных материалов. - М.: Издательство Московского университета, 1984. - 335 с.

1.4. Test Case №1.4

Problem Description

Infinite space filled with homogeneous isotropic elastic medium affected by concentrated force applied to point and acted according to Berlage law is considered as a problem (Stokes problem [1]). It is considered that source is point, i.e. it is small compared to characteristic dimensions of space. The problem has an analytical solution.

Input Values

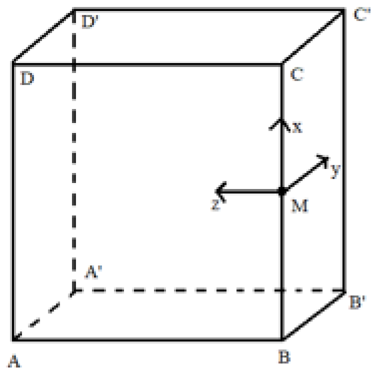

Fig 4 – Geometric model for Stokes problem

Material Properties:

- Isotropic
- Young's modulus E = 2e8 Pa;
- Poisson ratio ν = 0.3.
- Density = 1900 kg/m^3.

Geometric model:

- Cube 100 x 100 x 100 m;
- Cube moved to coordinates (0, 50, 50) so M = (0, 0, 0)

Boundary conditions:

- Displacement along Y axis for ABCD face equals 0.
- Displacement along Z axis for BB`C`C face equals 0.
- Displacement along X axis for A`D` edge equals 0.
- At point M = (0, 0, 0) applied 100 kN force acted along X axis
- Dependence of force on time according to the Berlage formula with an amplitude of 25e6 m and a cyclic frequency of 10 Hz. Note: in CAE Fidesys considered a quarter of the real model, so the amplitude used to implement the analytical solution in the mathematical package should be divided by 4;

- Non-reflective BC in planes AA`D`D, A`B`C`D`, DCC`D`, ABB`A`;
- Along the line of action of the force, receivers are assigned to the nodes in all directions for each field from the drop-down list (displacement, speed, principal stresses, pressure).Сетка:

Mesh:

- Hexahedron (order 1, order 2);
- Element height of the first block h = 10 m;
- Element height of the second block h = 9 m;
- Spectral seventh order hexahedrals.

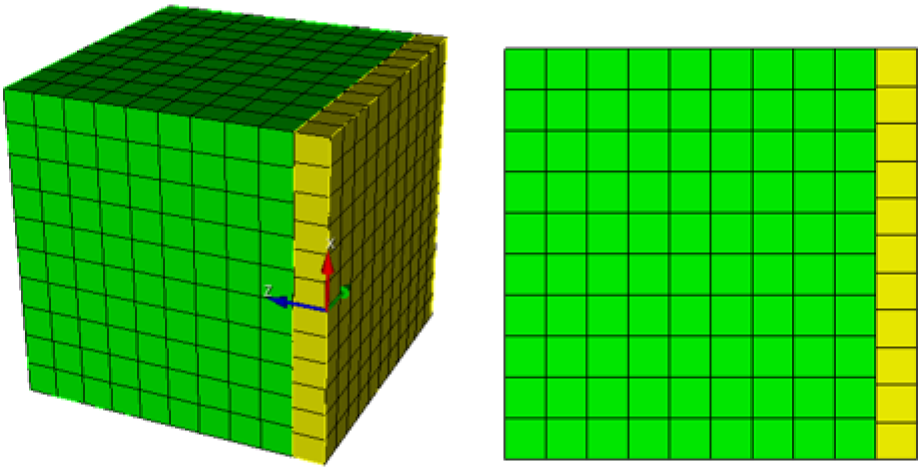

Fig 5 – Non-conformal finite element mesh for the Stokes problem

Target results

The displacement values are checked at point (20, 10, 20).

№	Value	Description	Unit	Target
1	X component of displacement vector for mesh nodes at step 0.13	Displacement X	m	5.308e-06
2	Y component of displacement vector for mesh nodes at step 0.144	Displacement Y	m	4.79e-06
3	Z component of displacement vector for mesh nodes at step 0.144	Displacement Z	m	9.581e-06

№	Value	Description	Unit	Target
4	X component of displacement vector for mesh nodes at step 0.199	Displacement X	m	1.843e-05
5	Y component of displacement vector for mesh nodes at step 0.206	Displacement Y	m	-7.416e-06
6	Z component of displacement vector for mesh nodes at step 0.2033	Displacement Z	m	-1.5e-05
7	X component of displacement vector for mesh nodes at step 0.249	Displacement X	m	-1.027e-05
8	Y component of displacement vector for mesh nodes at step 0.2532	Displacement Y	m	3.563e-06
9	Z component of displacement vector for mesh nodes at step 0.2532	Displacement Z	m	7.125e-06
10	X component of displacement vector for mesh nodes at step 0.299	Displacement X	m	3.536e-06
11	Y component of displacement vector for mesh nodes at step 0.3	Displacement Y	m	-1.1e-06
12	Z component of displacement vector for mesh nodes at step 0.303	Displacement Z	m	-2.328e-06

Analytical solution description

Let a concentrated force applied at a point $(x_0,\ y_0,\ z_0)$ and directed along a certain x_j axis act on an infinite space filled with a homogeneous isotropic elastic. Let this force be equal to zero in magnitude at $t<0$ and $X_0(t)$ at $t>0$. The vector of elastic displacements $u_i\ (x,\ t)$ corresponding to such a force is determined by the following Stokes formulas [1]:

$$u_i(x,\ t) \ =\ \frac{1}{4\pi\rho}\,(3\gamma_i\gamma_j-\delta_{ij})\frac{1}{r^3}\int_{\frac{r}{\alpha}}^{\frac{r}{\beta}}\tau X_0(t-\tau)d\tau+\frac{1}{4\pi\rho\alpha^2}\,\gamma_i\gamma_j\,\frac{1}{r}\,X_0(t-\frac{r}{\alpha})-$$

$$-\frac{1}{4\pi\rho\beta^2}\,(\gamma_i\gamma_j-\delta_{ij})\,\frac{1}{r}\,X_0(t-\frac{r}{\beta}),$$

where $r=\sqrt{(x-x_0)^2+(y-y_0)^2+(z-z_0)^2}$, $\gamma_i=\frac{x_i}{r}$ – direction cosines, $\alpha=\sqrt{\frac{\lambda+2\mu}{\rho}}$ – longitudinal wave velocity, $\beta=\sqrt{\frac{\mu}{\rho}}$ – shear wave velocity, $\mu=\frac{E}{2(1+\nu)}$, $\lambda=\frac{\nu E}{(1+\nu)(1-2\nu)}$ – Lame constants, ρ – density of the medium in which the waves propagate.

The Kronecker symbol δ_{ij} is interpreted as follows:

$$\delta_{ij}=0\ \ with\ i\neq j,$$
$$\delta_{ij}=1\ \ with\ i=j.$$

The force is applied along the x axis and propagates according to the Berlage law. It has been experimentally established that the propagation of elastic waves in the earth's crust is qualitatively described when the load is specified by the Berlage law [2]:

$$X_0(t)=A\cdot\omega_1^2e^{-\omega_1 t}\cdot\left(\sin(\omega_0 t)\left(\frac{-t^2}{\omega_1}+\frac{t}{\omega_1^{\ 2}}+\frac{1}{\omega_1^{\ 3}}\right)-\cos(\omega_0 t)\sqrt{3}\left(\frac{t^2}{\omega_1}+\frac{t}{\omega_1^{\ 2}}\right)\right),$$

$$\omega_0=2\pi\omega\ \ ,\ \ \omega_1=\frac{\omega_0}{\sqrt{3}}\,,$$

where A – vibration amplitude, ω – cyclic vibration frequency.

After analyzing all the coefficients in the Stokes formula, we will rewrite it more specifically for our setting:

$$u_x(x,\ t)=\frac{1}{4\pi\rho}\,(3\gamma_x\gamma_x-1)\frac{1}{r^3}\int_{\frac{r}{\alpha}}^{\frac{r}{\beta}}\tau X_0(t-\tau)d\tau+\frac{1}{4\pi\rho\alpha^2}\,\gamma_x\gamma_x\,\frac{1}{r}\,X_0(t-\frac{r}{\alpha})-$$

$$-\frac{1}{4\pi\rho\beta^2}\,(\gamma_x\gamma_x-1)\,\frac{1}{r}\,X_0(t-\frac{r}{\beta}),$$

$$u_y(x,t) = \frac{1}{4\pi\rho}(3\gamma_y\gamma_x - 0)\frac{1}{r^3}\int_{\frac{r}{\alpha}}^{\frac{r}{\beta}} \tau X_0(t-\tau)d\tau + \frac{1}{4\pi\rho\alpha^2}\gamma_y\gamma_x \frac{1}{r} X_0(t-\frac{r}{\alpha}) -$$

$$-\frac{1}{4\pi\rho\beta^2}(\gamma_y\gamma_x - 0)\frac{1}{r} X_0(t-\frac{r}{\beta}),$$

$$u_z(x,t) = \frac{1}{4\pi\rho}(3\gamma_z\gamma_x - 0)\frac{1}{r^3}\int_{\frac{r}{\alpha}}^{\frac{r}{\beta}} \tau X_0(t-\tau)d\tau + \frac{1}{4\pi\rho\alpha^2}\gamma_z\gamma_x \frac{1}{r} X_0(t-\frac{r}{\alpha}) -$$

$$-\frac{1}{4\pi\rho\beta^2}(\gamma_z\gamma_x - 0)\frac{1}{r} X_0(t-\frac{r}{\beta}).$$

Thus, the input data for the implementation of the analytical solution of the Stokes problem in mathematical packages are: $A,\ \omega,\ E,\ \nu,\ \rho$

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	X component of displacement vector for mesh nodes at step 0.136	Displacement X	m	5.328e-06	5.54992e-06	3.08
2	Y component of displacement vector for mesh nodes at step 0.144	Displacement Y	m	4.79e-06	4.85984e-06	1.56
3	Z component of displacement vector for mesh nodes at step 0.144	Displacement Z	m	9.58e-06	9.43758e-06	1.39
4	X component of displacement vector for mesh nodes at step 0.2	Displacement X	m	1.841e-05	1.87276e-05	1.67
5	Y component of displacement vector for mesh nodes at step 0.2	Displacement Y	m	-7.33e-06	-7.20336e-06	1.73
6	Z component of displacement vector for mesh nodes at step 0.2	Displacement Z	m	-1.466e-05	-1.52926e-05	4.32
7	X component of displacement vector for mesh nodes at step 0.248	Displacement X	m	-1.025e-05	-1.05004e-05	2.54
8	Y component of displacement vector for mesh nodes at step 0.256	Displacement Y	m	3.51e-06	3.28308e-06	0.77
9	Z component of displacement vector for mesh nodes at step 0.256	Displacement Z	m	7.021e-06	6.99676e-06	0.63

CAE Fidesys script:

```
reset
```

```
set default element hex
brick x 100 y 100 z 100
move Volume 1  x 0 y 50 z 50 include_merged
webcut volume 1 with plane zplane offset 10
move Volume 2 x 0 y 0 z -0.1 include_merged
partition create curve 6  position 0 0 0
volume 1 size 10
mesh volume 1
volume 2 size 9
mesh volume 2
create material 1
modify material 1 set property 'MODULUS' value 2e8
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 1900
set duplicate block elements off
block 1 add volume 1
block 1 material 1 cs 1 element solid order 4 #fixed
block 2 add volume 2
block 2 material 1 cs 1 element solid order 4 #fixed
create displacement  on curve 2  dof 1 fix 0
create displacement  on surface 10 14  dof 2 fix 0
create displacement  on surface 2  dof 3 fix 0
create absorption on surface 1 8 9 11 13 15 16
create force  on vertex 17  force value 1 direction 1 0 0
bcdep force 1 value 'berlage(25e6, 10, time)'
create contact master surface 7  slave surface 12  tolerance 0.11 type tied method auto
analysis type dynamic elasticity dim3 preload off
dynamic method full_solution scheme explicit maxtime 0.4 maxsteps 50000
output nodalforce off energy off record3d on log on vtu on material off results everystep 10
```

Reference

[1] Аки К. Количественная сейсмология/ Ричардс П. — М.: Мир, т. 1, 1983. — 880 с.

[2] Geophysics, vol. 55, no. 11, november 1990. — P. 1508-1511, 2 figs.

1.5. 1.6. Test case №1.5

Problem Description

A two-dimensional problem of the all-round tension of a flat unbounded plate with a circular cut is considered. The problem has an analytical solution. For the case of staged loading, it is taken into account that in the linear case the result does not depend on the loading path. Thus, the load is divided into 6 loading steps. In the case, the correctness of setting the boundary pressure condition for stage-by-stage loading is checked.

Input Values

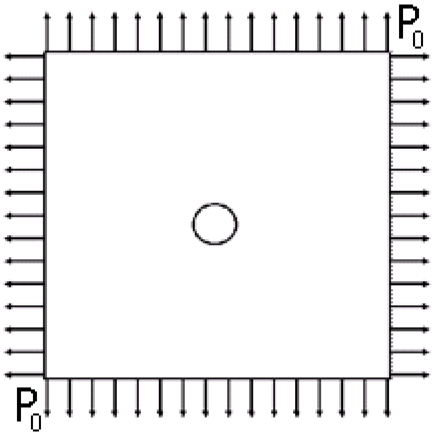

Figure 6 – Geometric model for a plate with ful all-round tension

Material prorerties:

- Young's modulus E = 200 GPa;
- Poisson ratio ν = 0.3 ;

Geometric model:

- Due to the symmetry of the problem, 1/4 of the plate is considered;
- Side of the plate 10 m;
- Hole diameter 0.5 m;
- Polar coordinates are used.

Bordery conditions:

- Zero displacements along the X axis on the line AB;
- Zero displacements along the Y axis on the line ED;
- P_0 = 0.1 MPa, 0.25 MPa, 0.5 MPa, 0.75 MPa, 0.9 MPa, 1 MPa.

Mesh:

- 2D third order quadrangular spectral elements

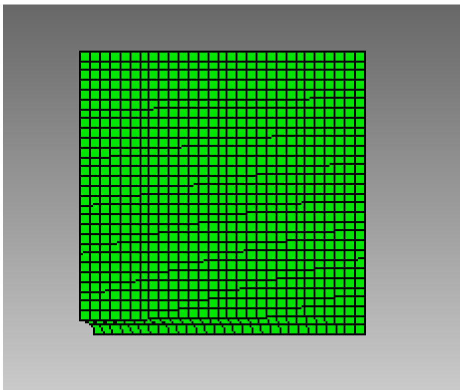

Fig 7 – 2D third order quadrangular spectral elements mesh

Target results

№	Value	Description	Unit	Target
1	Stress tensor components at mesh nodes in polar coordinates	Stress FF	MPa	2
2	Step number	step	-	6

Analitical solution

The values are calculated using the formula [1]:

$$\sigma_\theta = 2P_0.$$

Results

Quadrangular spectral elements

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Stress tensor components at mesh nodes in polar coordinates	Stress FF	MPa	2	2	0.00
2	Step number	step	-	6	6	-

CAE Fidesys Script:

```
reset
set default element hex
set node constraint on
create surface rectangle width 5 height 5 zplane
move surface 1 x 2.5 y 2.5
create surface circle radius 0.25 zplane
subtract body 2 from body 1
```

```
surface 3 size auto factor 2
surface 3 scheme auto
mesh surface 3
create material 1
modify material 1 name 'material 1'
modify material 1 set property 'MODULUS' value 2e+11
modify material 1 set property 'POISSON' value 0.3
set duplicate block elements off
block 1 add surface 3
block 1 material 1 cs 1 element plane order 3
create displacement on curve 7 dof 2 fix 0
create displacement on curve 8 dof 1 fix 0
create pressure on curve 1 4 magnitude 0
create table 1
modify table 1 dependency time
modify table 1 insert row 1
modify table 1 insert row 1
modify table 1 insert row 1
modify table 1 insert row 1
modify table 1 insert row 1
modify table 1 insert row 1
modify table 1 cell 1 1 value 1
modify table 1 cell 2 1 value 2
modify table 1 cell 3 1 value 3
modify table 1 cell 4 1 value 4
modify table 1 cell 5 1 value 5
modify table 1 cell 6 1 value 6
modify table 1 cell 1 2 value -100000
modify table 1 cell 2 2 value -250000
modify table 1 cell 3 2 value -500000
modify table 1 cell 4 2 value -750000
modify table 1 cell 5 2 value -950000
modify table 1 cell 6 2 value -1e+06
bcdep pressure 1 table 1
analysis type static elasticity dim2 planestrain
static steps 6
```

Reference

[1] Седов Л.И. “Механика сплошной среды, том 2”. М.: Наука, 1970г.

1.6. Test case №1.6

Problem Description

The problem of stress distribution in the vicinity of a vertical well of radius R_W drilled to depth h is considered. The reservoir is considered to be isotropic and homogeneous. The problem has an analytical solution [1]. The test task is designed to check the correctness:

- calculation of the pore pressure of the medium;

- taking into account the plastic properties of the material when calculating the stress-strain state of the medium;

- Drucker-Prager plasticity criterion with symmetric hardening;

- nonlinear model for calculating mechanical strength;

output fields of Displacements, Stresses, Elastic deformations, Plastic deformations taking into account the occurrence of plasticity.

Input values

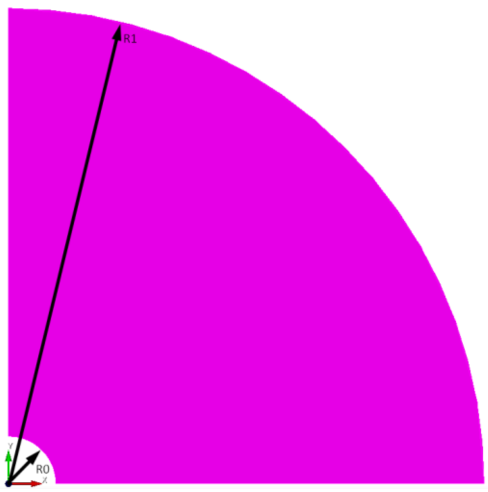

Fig 8 – Geometrical model

Geometrical model:

- Due to the symmetry of the problem, we consider 1/4 of the plate;
- R_1 = 10, R_2 = 1;
- Analytical solution uses polar coordinates

Bordery conditions:

- Well pressure p = 4e7;
- Pressure at a distance p = 8e7;
- Fastening based on symmetry conditions;
- Pore pressure p = 4e7.

Material parameters:

- Young's modulus E = 1e9 Pa;
- Poisson's ratio ν = 0.25;
- Cohesion K = 5.43712e + 6;
- Internal friction angle α = 21.43;
- Dilatancy angle β = 21.43;
- Porosity = 0.25;
- Permeability = 1e-12;
- Liquid viscosity = 0.005;
- Biot coefficient = 1;
- The liquid modulus of elasticity = 1e9.

Mesh:

- Second order hexahedrons.

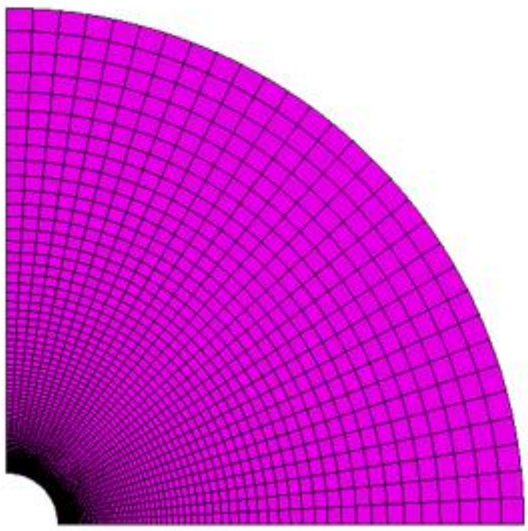

Fig 9 – 3rd order spectral elements for the Lamb problem

Calculation settings:

- Dynamic calculation;
- Maximum time – 3 s;
- Maximum number of steps – 2025;
- Output every 135 step to a .vtu file.

Target results

Target results are obtained from the analytical solution below and are presented with the calculated results.

Analytical solution

Verification of the numerical poroelastoplastic CAE Fidesys model is based on the analytical solution considered in part 1 of work [1].

The distribution of stresses in the vicinity of a vertical well of radius R_W drilled to depth h is studied. The reservoir is considered to be isotropic and homogeneous.

The problem is solved in a cylindrical coordinate system.

The initial stress state of the formation is considered as a state of all-round compression by rock pressure $Q = -\gamma h$, where γ is the average specific weight of the overlying rocks.

The paper assumes that the Biot coefficient is equal to 1, p_0 is the initial reservoir pressure of the filtering fluid. Then the initial effective stresses are determined by the expressions

$$S_r^0 = S_\theta^0 = S_z^0 = Q + p_0$$

and full stresses

$$\sigma_r = S_r - p_0, \sigma_\theta = S_\theta - p_0, \sigma_z = S_z - p_0$$

In the statement of part 1 [1], it is considered that there is no fluid filtration, therefore, the pore pressure p_W in the well coincides with p_0.

In [1], it is assumed that the Coulomb-Mohr criterion is used as a yield criterion with parameters τ_s - adhesion coefficient, ρ - angle of internal friction of the rock. CAE Fidesys uses the Drucker-Prager criterion. The Drucker-Prager surface is a smoothed Coulomb-Mohr surface (in CAE Fidesys, the Drucker-Prager surface is inscribed in the Coulomb-Mohr hex cone). Based on the study [2], we assume that the differences in the results for the Drucker-Prager and Coulomb-Mohr criteria should be insignificant.

Results

The displacement values are checked at the point (70.4225, 4.31214e-15, 0.0).

№	Value	Point coordinates	Description	Unit	Target	CAE Fidesys result	Error, %
1	Stress component σ_{yy}	(1,0,0)	Stress_YY	Pa	-6.81E+07	-6.771E+07	-0.58
2	Stress component σ_{yy}	(1.1102, 0,0)	Stress_YY	Pa	-7.75E+07	-7.758E+07	-0.10
3	Stress component σ_{yy}	(1.2063, 0,0)	Stress_YY	Pa	-8.57E+07	-8.643E+07	-0.88
4	Stress component σ_{yy}	(1.30623, 0,0)	Stress_YY	Pa	-9.31E+07	-9.400E+07	-0.98

№	Value	Point coordinates	Description	Unit	Target	CAE Fidesys result	Error, %
5	Stress component σ_{yy}	(1.38922, 0,0)	Stress_YY	Pa	-9.68E+07	-9.757E+07	-0.78
6	Stress component σ_{yy}	(1.49691, 0,0)	Stress_YY	Pa	-9.92E+07	-9.969E+07	-0.51
7	Stress component σ_{yy}	(1.655, 0,0)	Stress_YY	Pa	-1.00E+08	-1.003E+08	-0.02
8	Stress component σ_{yy}	(1.74951, 0,0)	Stress_YY	Pa	-9.92E+07	-9.901E+07	-0.14
9	Stress component σ_{yy}	(1.99968, 0,0)	Stress_YY	Pa	-9.48E+07	-9.469E+07	-0.11
10	Stress component σ_{yy}	(2.50458, 0,0)	Stress_YY	Pa	-8.96E+07	-8.956E+07	-0.08
11	Stress component σ_{yy}	(3.01979, 0,0)	Stress_YY	Pa	-8.68E+07	-8.676E+07	-0.06
12	Stress component σ_{yy}	(3.4908, 0,0)	Stress_YY	Pa	-8.52E+07	-8.520E+07	-0.05
13	Stress component σ_{yy}	(4.01398, 0,0)	Stress_YY	Pa	-8.41E+07	-8.407E+07	-0.04
14	Stress component σ_{yy}	(6.01916, 0,0)	Stress_YY	Pa	-8.21E+07	-8.212E+07	-0.02
15	Stress component σ_{yy}	(8.01412, 0,0)	Stress_YY	Pa	-8.15E+07	-8.144E+07	-0.01
16	Stress component σ_{yy}	(10, 0,0)	Stress_YY	Pa	-8.11E+07	-8.113E+07	-0.02
17	Stress component σ_{xx}	(1, 0,0)	Stress_XX	Pa	-4.00E+07	-4.000E+07	-0.01
18	Stress component σ_{xx}	(1.1102, 0,0)	Stress_XX	Pa	-4.32E+07	-4.329E+07	-0.17
19	Stress component σ_{xx}	(1.2063, 0,0)	Stress_XX	Pa	-4.63E+07	-4.634E+07	-0.07
20	Stress component σ_{xx}	(1.30623, 0,0)	Stress_XX	Pa	-4.98E+07	-4.971E+07	-0.15
21	Stress component σ_{xx}	(1.38922, 0,0)	Stress_XX	Pa	-5.29E+07	-5.245E+07	-0.82
22	Stress component σ_{xx}	(1.49691, 0,0)	Stress_XX	Pa	-5.67E+07	-5.578E+07	-1.53
23	Stress component σ_{xx}	(1.655, 0,0)	Stress_XX	Pa	-6.09E+07	-6.001E+07	-1.45
24	Stress component σ_{xx}	(1.74951, 0,0)	Stress_XX	Pa	-6.29E+07	-6.216E+07	-1.18
25	Stress component σ_{xx}	(1.99968, 0,0)	Stress_XX	Pa	- 6.69E+07	-6.648E+07	-0.64
26	Stress component σ_{xx}	(2.50458, 0,0)	Stress_XX	Pa	-7.17E+07	-7.158E+07	-0.11
27	Stress component σ_{xx}	(3.01979, 0,0)	Stress_XX	Pa	74260000	-7.439E+07	-0.17
28	Stress component σ_{xx}	(3.4908, 0,0)	Stress_XX	Pa	-7.57E+07	-7.594E+07	-0.31
29	Stress component σ_{xx}	(4.01398, 0,0)	Stress_XX	Pa	-7.68E+07	-7.707E+07	-0.42

№	Value	Point coordinates	Description	Unit	Target	CAE Fidesys result	Error, %
30	Stress component σ_{xx}	(6.01916, 0,0)	Stress_XX	Pa	-7.86E+07	-7.901E+07	-0.57
31	Stress component σ_{xx}	(8.01412, 0,0)	Stress_XX	Pa	-7.92E+07	-7.969E+07	-0.63
32	Stress component σ_{xx}	(10, 0,0)	Stress_XX	Pa	-7.95E+07	-8.000E+07	-0.66
33	Elastic strain component ε_{xx}	(1, 0,0)	Elastic_Strain_X	-	0.012107	0.0122	0.10
34	Elastic strain component ε_{xx}	(1.12917, 0,0)	Elastic_Strain_X	-	0.01336	0.01341	0.36
35	Elastic strain component ε_{xx}	(1.30623, 0,0)	Elastic_Strain_X	-	0.011978	0.01205	0.61
36	Elastic strain component ε_{xx}	(1.97385, 0,0)	Elastic_Strain_X	-	-0.00726	-7.271E-03	-0.16
37	Elastic strain component ε_{xx}	(2.69, 0,0)	Elastic_Strain_X	-	-0.01554	-1.562E-02	-0.52
38	Elastic strain component ε_{xx}	(3.685, 0,0)	Elastic_Strain_X	-	-0.02012	-2.017E-02	-0.20
39	Elastic strain component ε_{xx}	(6.137, 0,0)	Elastic_Strain_X	-	-0.02347	-2.348E-02	-0.05
40	Elastic strain component ε_{xx}	(10, 0,0)	Elastic_Strain_X	-	-0.02465	-2.465E-02	-0.00
41	Elastic strain component ε_{yy}	(1, 0,0)	Elastic_Strain_Y	-	-0.02285	-0.02251	-1.49
42	Elastic strain component ε_{yy}	(1.497, 0,0)	Elastic_Strain_Y	-	-0.0488	-0.04919	-0.81
43	Elastic strain component ε_{yy}	(1.609, 0,0)	Elastic_Strain_Y	-	-0.04991	-0.04998	-0.13
44	Elastic strain component ε_{yy}	(2.187, 0,0)	Elastic_Strain_Y	-	-0.04021	-4.010E-02	-0.28
45	Elastic strain component ε_{yy}	(3.054, 0,0)	Elastic_Strain_Y	-	-0.03297	-3.291E-02	-0.18
46	Elastic strain component ε_{yy}	(3.93, 0,0)	Elastic_Strain_Y	-	-0.02996	-2.992E-02	-0.13
47	Elastic strain component ε_{yy}	(5.455, 0,0)	Elastic_Strain_Y	-	-0.02774	-2.772E-02	-0.08
48	Elastic strain component ε_{yy}	(10, 0,0)	Elastic_Strain_Y	-	-0.02607	-2.606E-02	-0.04
49	Displacement component u_x	(1, 0,0)	Displacement X	m	-0.14374	-1.435E-01	-0.17

№	Value	Point coordinates	Description	Unit	Target	CAE Fidesys result	Error, %
50	Displacement component u_x	(1.1102, 0,0)	Displacement X	m	-0.11475	-1.146E-01	-0.16
51	Displacement component u_x	(1.2063, 0,0)	Displacement X	m	-0.10044	-1.003E-01	-0.14
52	Displacement component u_x	(1.30623, 0,0)	Displacement X	m	-0.09343	-9.218E-02	-1.34
53	Displacement component u_x	(1.47495, 0,0)	Displacement X	m	-0.0868	-8.621E-02	-0.69
54	Displacement component u_x	(1.70187, 0,0)	Displacement X	m	-0.08491	-8.453E-02	-0.44
55	Displacement component u_x	(2.10559, 0,0)	Displacement X	m	-0.08714	-8.683E-02	-0.35
56	Displacement component u_x	(2.44476, 0,0)	Displacement X	m	-0.09106	-9.079E-02	-0.29
57	Displacement component u_x	(3.1594, 0,0)	Displacement X	m	-0.1026	-1.024E-01	-0.20
58	Displacement component u_x	(7.5045, 0,0)	Displacement X	m	-0.19975	-1.996E-01	-0.05
59	Displacement component u_x	(10, 0,0)	Displacement X	m	-0.26066	-2.606E-01	-0.04
60	Plastic strain	(1, 0,0)	Plastic_Strain_XX	-	0.330137	0.3313	0.36
61	Plastic strain	(1.12917, 0,0)	Plastic_Strain_XX	-	0.161152	0.1604	0.47
62	Plastic strain	(1.38922, 0,0)	Plastic_Strain_XX	-	0.024793	0.02468	0.45
63	Plastic strain	(1.72569, 0,0)	Plastic_Strain_XX	-	0	1.509E-05	0.00
64	Plastic strain	(1, 0,0)	Plastic_Strain_YY	-	-0.12074	-0.1209	-0.17
65	Plastic strain	(1.12917, 0,0)	Plastic_Strain_YY	-	-0.06783	-0.06768	-0.22
66	Plastic strain	(1.38922, 0,0)	Plastic_Strain_YY	-	-0.01683	-16.85	-0.10
67	Plastic strain	(1.72569, 0,0)	Plastic_Strain_YY	-	0	-1.452E-05	0.00

CAE Fidesys script:

reset

```
set default element hex
create surface circle radius 10 zplane
create surface circle radius 1 zplane
subtract body 2 from body 1
webcut body 1  with plane yplane offset 0
webcut body 3  with plane xplane offset 0
delete Body 4
delete Body 1
merge all
create material 1
modify material 1 set property 'MODULUS' value 1e+09
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'COHESION' value 5.43712e+06
modify material 1 set property 'INT_FRICTION_ANGLE' value 21.43
modify material 1 set property 'DILATANCY_ANGLE' value 21.43
modify material 1 set property 'BIOT_ALPHA' value 1
modify material 1 set property 'POROSITY' value 0.25
modify material 1 set property 'PERMEABILITY' value 1e-12
modify material 1 set property 'FLUID_VISCOCITY' value 0.005
modify material 1 set property 'FLUID_BULK_MODULUS' value 1e9
curve 8 12 interval 90
curve 8 scheme bias factor 1.05 start vertex 7
curve 12 scheme bias factor 1.05 start vertex 11
curve 13 14 interval 30
mesh surface all
create displacement on curve 8 dof 2 fix 0
create displacement on curve 12 dof 1 fix 0
create porepressure on curve 13 14 value 4e7
create pressure on curve 13 magnitude 4e7
create pressure on curve 14 magnitude 8e7
block 1 surface all
block 1 material 1
block 1 element plane order 2
analysis type static elasticity plasticity porefluidtrans dim2 planestrain
nonlinearopts maxiters 100 minloadsteps 30 maxloadsteps 10000000 tolerance 1e-3
```

Reference

[1] Журавлев А.Б. Влияние фильтрации на напряженно-деформированное состояние породы в окрестности скважины / А.Б. Журавлев, В.И. Карев, Ю.Ф. Коваленко, К.Б. Устинов // Прикладная математика и механика, Т. 78, Вып. 1, 2014, стр. 86-97.

[2] Mountaka Souley, Alain Thoraval. Nonlinear mechanical and poromechanical analyses : comparison with analytical solutions. COMSOL Conference 2011, Oct 2011, Stuttgart, Germany. pp.NC. ffineris00973639

1.7. Test case №1.7

Problem Description

The proposed case simulates the Hertz problem for two hemispheres contacting at the origin. Test case aimed to check correctness of:

- setting a sliding contact without friction in the interface;

- static solution taking into account sliding contact without friction for 3D models;

the correctness of the output of the Stress field, taking into account the contact interaction.

Input Values

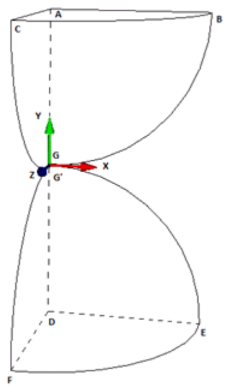

Fig 10 – Geometrical model

Geometrical model:

- Due to symmetry, one fourth of the hemispheres contacting at the origin is considered;
- Radius of hemispheres r = 50 mm.

Material Properties:

- Isotropic;
- Young's modulus = 2e4 MPa;
- Poisson ratio = 0.3;

Boundary conditions:

- Fixation normal to surfaces ABG и DEG': $u_z\big|_{z=0} = 0$;
- Fixation normal to surfaces ACG и DFG': $u_x\big|_{x=0} = 0$;
- Displacement on surface ABC: $u_y\big|_{y=r} = -2$ мм;

- Displacement on surface DEF: $u_y \big|_{y=-r} = 2$ мм;
- Common contact for surfaces ABCG and DEFG`.

Mesh:

- First order hexahedrons

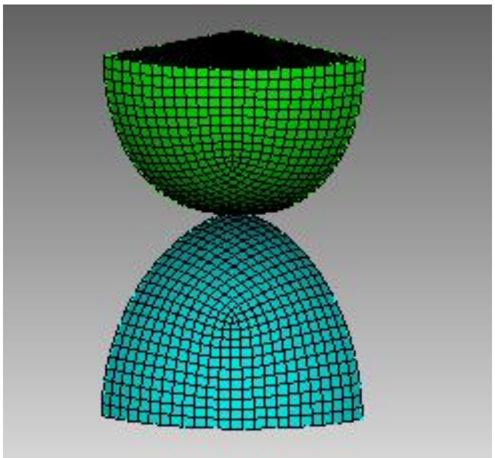

Fig 11 – Hexahedrons

Calculation settings:

- Static analysis;
- Elasticity;
- 3D.

Target results

№	Value	Description	Unit	Target
1	σ_{yy} component of stress tensor	Stress YY	MPa	-2798.3

Analytical solution

The reference value is obtained using the formula [1]:

$$\sigma_{yy}\Big|_G = -\frac{E}{\pi}\frac{1}{1-\nu^2}\sqrt{\frac{4u_y\big|_{y=-r}}{r}} \quad .$$

Results

First order hexahedral mesh

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	σ_{yy} component of stress tensor	Stress YY	MPa	-2798.3	-2.935E+03	4.87

CAE Fidesys script:

```
reset
create sphere radius 50
move Volume 1 y 50 include_merged
create sphere radius 50
move Volume 2 y -50 include_merged
webcut volume 1  with plane yplane offset 50
webcut volume 2  with plane yplane offset -50
delete volume 3 2
webcut volume all with plane xplane offset 0
webcut volume all with plane zplane offset 0
delete volume 5 6 7 8 9 10
volume all scheme polyhedron
volume all size auto factor 4
mesh volume all
create material 1
modify material 1 name 'Material 1'
modify material 1 set property 'MODULUS' value 2e4
modify material 1 set property 'POISSON' value 0.3
set duplicate block elements off
block 1 add volume all
block 1 material 1 cs 1 element solid order 1
create displacement  on surface 25 33 dof 1 fix 0
create displacement  on surface 23 31 dof 3 fix 0
create displacement  on surface 24 dof 2 fix -2
create displacement  on surface 34 dof 2 fix 2
create contact master surface 32 slave surface 26 tolerance 0.0005 friction 0.0 preload 0.0 offset 0.0 ignore_overlap off type general method auto
analysis type static plasticity elasticity dim3
nonlinearopts maxiters 50 minloadsteps 10 maxloadsteps 30 tolerance 1e-3 targetiter 5
```

Reference

[1] G. DUMONT: "Method of the active stresses applied to the unilateral contact" Note HI-75/93/016.

1.8. Test case №1.8

Problem Description

In the proposed problem, a steel cylinder is pressed into an aluminum block. Both materials are assumed to be linearly elastic. In this case, a point force F acts on the cylinder in the negative direction of the Y axis. The problem has an analytical solution for the case when the coefficient of friction $\mu = 0$. The test case is designed to check the correctness of:

- setting the parameters of sliding contact without friction in the interface;

- static solution taking into account sliding contact without friction for the case of 2D;

- the correctness of the output of the voltage field in the contact.

Input values

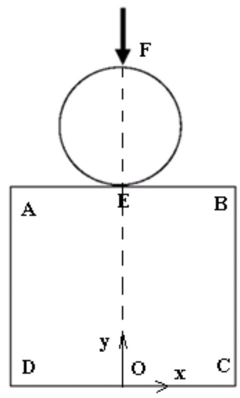

Fig 12 - Geometrical model

Geometrical model:

- Circle with diameter d = 100 mm;
- Sqare plate 200×200 mm.

Material Properties:

- Isotropic;
- Circle Young's modulus E $_{circle}$ = 210 KPa;
- Plate Young's modulus E $_{plate}$ = 70 KPa;
- Poisson ratio ν = 0,3.

Boundary conditions:

- Due to symmetry, ½ part of the model is considered;
- For edge OC $u_x = u_y = 0$;

- For edge OE, EF $u_x = 0$;
- At point F, a force of 35 kN is applied, directed along the negative Y-axis;.
- Sliding contact without friction (common) for surfaces EF and ABCD.

Mesh:

- 8-node sqare elements.

Calculation settings:

- Static analysis;
- 2D;
- Plain strain.

Target results

№	Value	Description	Point	Unit	Target
1	Stress tensor components in the contact zone	Contact Stress Node X	(6.22562, -49.6109, 0)	Pa	0

Analytical solution

An analytical solution to this contact problem can be obtained from the contact formulas of Hertz [1] for two cylinders. The maximum contact pressure is determined by the formula:

$$p_{\max} = \sqrt{\frac{F_n E^*}{2\pi B R^*}},$$

where F_n is the applied normal force, E^* is the combined modulus of elasticity, B is the length of the cylinder, and R^* is the combined radius.

Contact width 2a is defined as:

$$a = \sqrt{\frac{8 F_n R^*}{\pi B E^*}} .$$

Using a normalized coordinate with a Cartesian coordinate system $\xi = x/a$ and coordinate x, the pressure distribution is determined as follows:

$$p = p_{\max}\sqrt{1-\xi^2} .$$

The combined modulus of elasticity is determined from the modulus of elasticity and Poisson's ratio of the cylinder E_1, ν_1 and block E_2, ν_2 as follows:

$$E^* = \frac{2E_1E_2}{E_2(1-\nu_1{}^2)+E_1(1-\nu_2{}^2)} .$$

The total radius of curvature is calculated from the radius of curvature of the cylinder R_1 and block R_2 as follows:

$$R^* = \frac{R_1 R_2}{R_1 + R_2}.$$

For the target solution, the block is approximated by an infinitely large radius. The combined radius is then evaluated as:

$$R^* = \lim_{R_2 \to \infty} \frac{R_1 R_2}{R_1 + R_2} = R_1 .$$

Results

№	Value	Description	Point	Unit	Target	CAE Fidesys result	Error, %
1	Stress tensor components in the contact zone	Contact Stress Node X	(6.22562, -49.6109, 0)	Pa	0	0.00	0.00

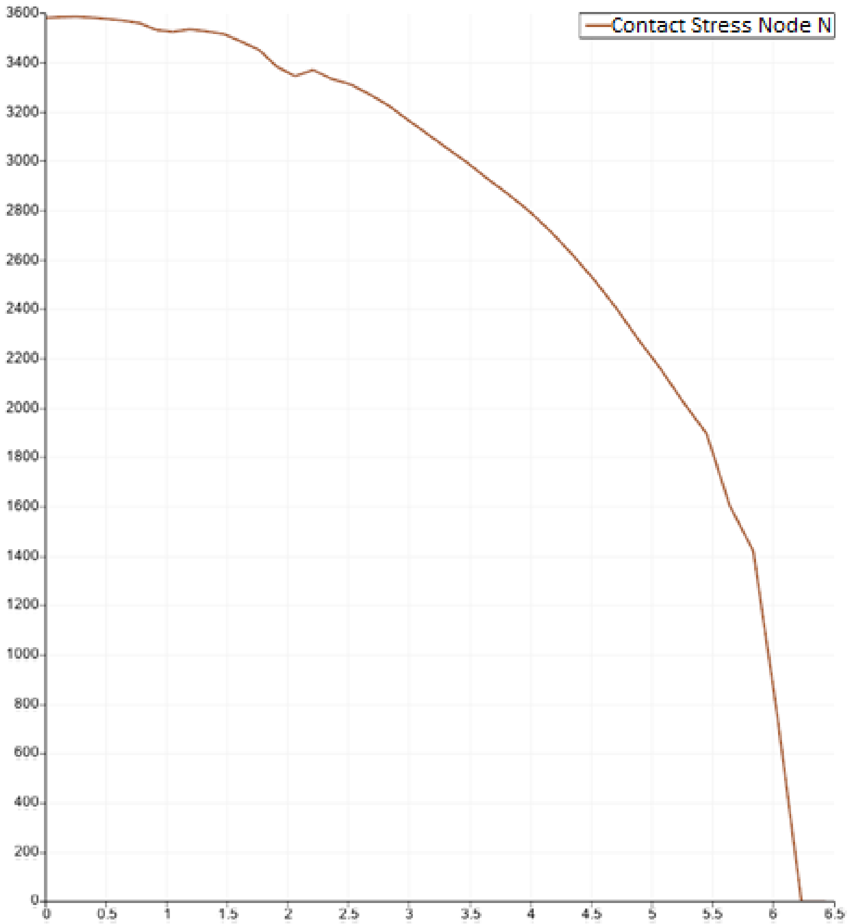

Fig. 13 - Graph of contact stress node stress N distribution with the contact zone 6.2 mm

CAE Fidesys script:

```
reset
set default element hex
create surface circle radius 50 zplane
create surface rectangle width 200 height 200 zplane
move Surface 2 y -150 include_merged
webcut body 1 2 with plane xplane offset 0
delete Surface 4 6
split surface 3 across location position 0 0 0 location position 50 0 0
create surface rectangle width 25 zplane
move Surface 9 y -62.5 include_merged
move Surface 9 x 12.5 include_merged
split surface 5 with surface 9
delete Body 5
split surface 11 across location position 0 -150 0 location position 100 -150 0
curve 18 17 scheme bias fine size 0.25 factor 1.025 start vertex 7
mesh curve 18 17
surface 7 size auto factor 3
mesh surface 7
surface 8 size auto factor 3
mesh surface 8
surface 10 size 1
mesh surface 10
surface 13 12 size auto factor 3
mesh surface 13 12
create material 1
modify material 1 name 'Mat_cube'
modify material 1 set property 'MODULUS' value 2.1e5
modify material 1 set property 'POISSON' value 0.3
create material 2
modify material 2 name 'Mat_cyl'
modify material 2 set property 'MODULUS' value 7e4
modify material 2 set property 'POISSON' value 0.3
```

set duplicate block elements off

block 1 surface 12 13 10

set duplicate block elements off

block 2 surface 8 7

block 1 material 'Mat_cube'

block 2 material 'Mat_cyl'

create displacement on curve 11 dof all fix

create displacement on curve 20 17 28 35 32 dof 1 dof 3 dof 4 dof 5 dof 6 fix

create force on vertex 6 force value 17500 direction ny

block 1 element plane order 2

block 2 element plane order 2

create contact master curve 27 slave curve 18 tolerance 0.0005 type general friction 0.0 preload 0.0 offset 0.0 ignore_overlap off method penalty normal_stiffness 1.0 tangent_stiffness 0.5

analysis type static elasticity dim2 planestrain

Reference

[1] Hertz, H., Über die Berührung fester elasticher Körper. J. Reine Angew. Mathm. 92, 156-171, 1881.

1.9. Test case №1.9

Problem Description

We consider the problem of finding the eigenfrequencies of a cantilever beam, which is divided into three parts, between which the condition of coupled contact acts. The beam is clamped at the left end and loaded with a tensile longitudinal force p at the right end. The test task is designed to check the correctness of the modal analysis calculation result, taking into account the rigid contact.

Input Values

Geometrical model:

- Length $L = 0.5$ m;
- Width $b = 0.05$ m;
- Height $h = 0.02$ m.

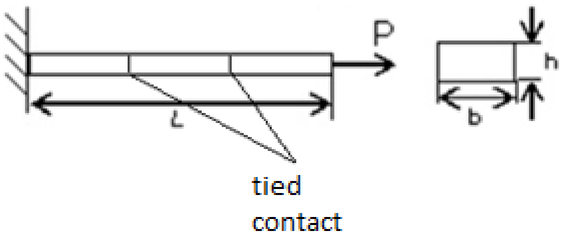

Fig 12 - Geometrical model

Boundary conditions:

- The left end of the beam is fixed along all axes ($u_x = u_y = u_z = r_x = r_y = r_z = 0$);
- A force applied at the right end of the beam $P = 50000$ N.

Material properties:

- Young's modulus $E = 2.1e+11$ Pa;
- Poisson ratio $\nu = 0.28$;
- Density $\rho = 7800$ kg/m³.

Mesh:

- second order tetrahedral mesh.

Contact cettings:

- Rigid;
- Method: auto.

Analysis settings:

- Modal analysis;
- Preloaded model;
- Search for the first lowest frequency.

Target results

№	Value	Description	Value
1	Natural frequency	Eigen Values 1, Hz	85.804

Analytical solution

The analytical solution is as follows [1]:

$$f_1^* = f_1 \cdot \sqrt{1 + \frac{5PL^2}{14EJ}}$$

$$f_1 = \frac{1}{2\pi}\left(\frac{k_1}{L}\right)^2 \sqrt{\frac{EJ}{\rho F}},$$

where f_1 is the first natural frequency of the cantilever beam, J is the moment of inertia, ρ is the density of the material, F is the cross-sectional area, $k_1 = 1.875$.

Results

The displacement values are checked at the point (20, 10, 20).

№	Value	Description	Unit	Value	CAE Fidesys result	Error, %
1	Natural frequency	Eigen Values 1	Hz	85.804	8.616E+01	0.42

CAE Fidesys script:

```
reset
brick x 0.5 y 0.02 z 0.05
webcut volume 1  with plane xplane offset 0.083333333
webcut volume 2  with plane xplane offset -0.083333333
merge all
volume all  size 0.01
volume all scheme Tetmesh
mesh volume all
create contact autoselect volume 1 2  tolerance 0.0005 type general friction 0.0 preload 0.0 offset 0.0 ignore_overlap off
method auto
```

create contact autoselect volume 3 2 tolerance 0.0005 type general friction 0.0 preload 0.0 offset 0.0 ignore_overlap off method auto
create material 1 name "mat1"
modify material 1 set property 'DENSITY' value 7800
modify material 1 set property 'POISSON' value 0.28
modify material 1 set property 'MODULUS' value 2.1e+11
set duplicate block elements off
block 1 volume all
block 1 material 'mat1'
create displacement on surface 4 dof all fix
list Surface 6 mesh
create force on vertex 2 5 6 1 force value 12500 direction x
block 1 element solid order 2
analysis type eigenfrequencies dim3 preload on
eigenvalue find 10 smallest

Reference

[1] AutoFem Analysis First Natural Frequency of the Cantilever Beam under the Stretching Lonqitudinal Force (https://autofem.com)

1.10. Test case №1.10

Problem description

The problem of the dependence of the critical force on the conditions for fixing the rod is considered. The rod is divided into two parts, between which the condition of common contact is valid. The rod is clamped at the left end and loaded with a tensile longitudinal force P at the right end. The control case is designed to check the correctness of the calculation for the analysis of buckling taking into account the common contact.

Input values

Geometrical model:

- Length $L = 2.54$ m;
- Width $b = 0.0508$ m;
- Height $h = 0.0508$ m.

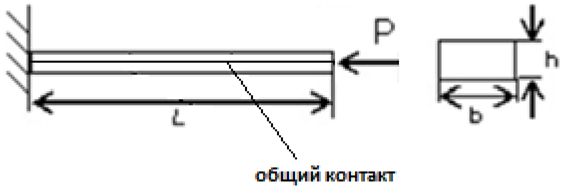

Fig 13 - Geometrical model

Boundary conditions:

- The left end of the beam is fixed along all axes ($u_x = u_y = u_z = r_x = r_y = r_z = 0$);
- A force applied at the right end of the beam $P = 0.1$ N.

Material properties:

- Young's modulus $E = 2.1e+11$ Pa;
- Poisson ratio $\nu = 0.3$;

Mesh:

- second order hexahedral mesh.

Contact cettings:

- Common;
- Method: auto.

Analysis settings:

- Buckling;
- Search for the first form of buckling;.

Target results

№	Value	Description	Target
1	Critical force	Critical Values 1	44527

Analytical solution

The analytical solution is as follows [1]:

$$P_{cr} = \frac{\pi^2 El}{(l/2)^2}.$$

Results

The displacement values are checked at the point (70.4225, 4.31214e-15, 0).

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Critical force	Critical Values 1	-	44527	4.458E+04	0.12

CAE Fidesys script:

```
reset
set default element hex
brick x 2.54 y 0.0508 z 0.0508
webcut volume 1 with plane yplane
webcut volume all with plane zplane
surface 19 26 33 31  scheme map
mesh surface 19 26 33 31
curve 2 4 6 8 interval 50
curve 2 4 6 8  scheme equal
mesh curve 2 4 6 8
volume all size auto factor 4
mesh volume all
create material 1
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 2.1e11
set duplicate block elements off
block 1 volume all
block 1 material 1
block 1 element solid order 2
create displacement on surface 23 35 29 21 dof all fix 0
```

```
create pressure on surface 19 26 33 31 magnitude 388
create contact autoselect tolerance 0.0005 type general method auto
analysis type stability elasticity dim3
eigenvalue find 1 smallest
```

Reference

[1] Феодосьев В.И. Сопротивление материалов: Учеб. для вузов. - 10-е издание, перераб. и доп. - М.: Изд-во МГТУ им. Н.Э.Баумана, 1999. - 592 с.

1.11. Test case №1.11

Problem description

Compression of an elastoplastic rectangular parallelepiped (multilinear isotropic hardening)

Input values

Geometrical model:

- Parallelepiped 5x1x1;

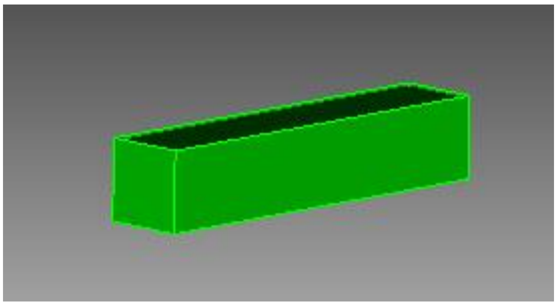

Fig 14 - Geometrical model

Bordery conditions:

- For face y = 0 u_y = 0;
- For face z = 0 u_z = 0;
- For whole model u_x = -2*x/5

Material properties:

- Young's modulus E = 5.1e+6;
- Poisson ratio ν = 0.25;
- Cohesion c = 15000;
- Internal friction angle ϕ = 0;
- Dilatation angle ψ = 0;

  The hardening given by the stress / plastic strain curve (tension) imported from the lider_hardening.csv file:

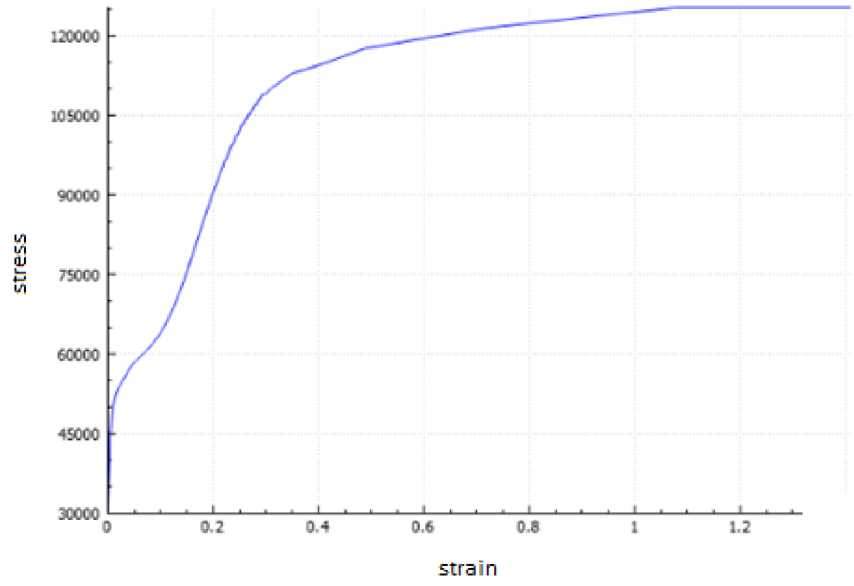

Fig 15 – Stress / plastic strain curve

Mesh:

- Second order hexahedrons.

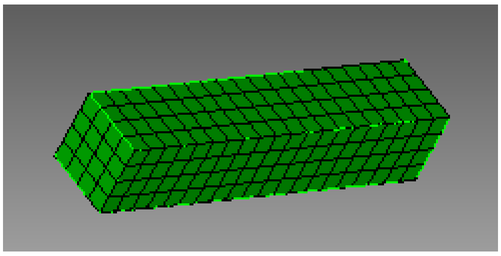

Fig 16 – Mesh

Target results

№	Value	Point	Description	Target
1	Stress tensor components for t=0.2	(5, 0, 1)	σ_{xx}, Pa	-60117.782
2	Stress tensor components for t=0.4	(5, 0, 1)	σ_{xx}, Pa	-74207.347
3	Stress tensor components for t=0.6	(5, 0, 1)	σ_{xx}, Pa	-96336.05
4	Stress tensor components for t=0.8	(5, 0, 1)	σ_{xx}, Pa	-108917.197
5	Stress tensor components for t=1	(5, 0, 1)	σ_{xx}, Pa	-113650.937

Analytical solution

From Hooke's law, elastic strain are related to stress as follows:

$$\varepsilon_{11}^{2} = \frac{1}{E}(\sigma_{11} - \nu(\sigma_{22} + \sigma_{33}))$$

$$\varepsilon_{22}^{2} = \frac{1}{E}(\sigma_{22} - \nu(\sigma_{11} + \sigma_{33}))$$

$$\varepsilon_{33}^{2} = \frac{1}{E}(\sigma_{33} - \nu(\sigma_{22} + \sigma_{11}))$$

Expressions for strain ε_{ij} are written as:

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_j}\right)$$

Based on the boundary conditions, $\sigma_{22} = \sigma_{33} = 0$, then Hooke's law and the expression for ε_{ij} can be written as follows:

$$\varepsilon_{11}(t) = t\frac{\partial u_1}{\partial x_1} = -0.4t$$

$$\varepsilon_{11}^{e} = \frac{\sigma_{11}}{E}$$

$$\varepsilon_{22}^{e} = -\frac{\nu\sigma_{11}}{E} = \varepsilon_{33}^{e}$$

For this case, the yield stress is reached when the strain ε_{ij} reaches the value:

$$\varepsilon_c = -\frac{\sigma_c}{E} = -\frac{2c}{E} = 0.00588$$

It is achieved at a time t equal to

$$t_c = \frac{\varepsilon_c}{\varepsilon_{11}(1)} = \frac{-0.00588}{-0.4} = 0.0147$$

After reaching the yield point, the material acts according to the Drucker-Prager plasticity criterion

$$F\left(\sigma, \varepsilon_{eq}^{p}\right) = \sigma_{eq} + \beta\sigma - R\left(\varepsilon_{eq}^{p}\right) = 0$$

where σ_{eq} - equivalent stress, H, β, σ_y – given constants, σ - the first invariant of the stress tensor, ε_{eq}^{p} – equivalent plastic strain

$$\sigma_{eq} = \sqrt{\frac{3}{2} S_{ij} \cdot S_{ij}}$$

$$\sigma = \sigma_{11} + \sigma_{22} + \sigma_{33}$$

$$\beta = \frac{2 \sin \phi}{3 - \sin \phi} = 0$$

$$\sigma_y = \frac{6c \cos \phi}{3 - \sin \phi} = 2c$$

$$\varepsilon^p_{eq} = \sqrt{\frac{2}{3} e^p_{ij} \cdot e^p_{ij}}$$

where S_{ij}- stress tensor deviator, e^p_{ij} - plastic strain tensor deviator, ε^p - the first invariant of the - plastic strain tensor

$$S_{ij} = \sigma_{ij} - \frac{\sigma}{3} \delta_{ij}$$

$$e^p_{ij} = \varepsilon^p_{ij} - \frac{\varepsilon^p}{3} \delta_{ij}$$

$$\varepsilon^p = \varepsilon^p_{11} + \varepsilon^p_{22} + \varepsilon^p_{33}$$

For the Drucker-Prager plasticity model, the relationship between stress and plastic strain has the following form

$$\varepsilon^p_{ij} = \varepsilon^p_{eq} \left(-\frac{3}{2} \frac{S_{ij}}{\sigma_{eq}} + \beta \delta_{ij} \right)$$

Full strain is a sum of elastic and plastic:

$$\varepsilon_{ij} = \varepsilon^p_{ij} + \varepsilon^e_{ij}$$

From boundary conditions, $\sigma_{22} = \sigma_{33} = 0$, then we can evaluate σ_{eq} and σ

$$\sigma = \sigma_{11}, \sigma_{eq} = |\sigma_{11}|$$

Since we consider uniaxial compression, $\sigma_{11} < 0$ и $\varepsilon^p_{11} < 0$, then the expressions for the criterion of plasticity and the relationship between stress and plastic strain take on a simpler form:

$$\sigma_{11} = -R\left(\varepsilon^p_{eq}\right)$$

$$\varepsilon^p_{11} = -\varepsilon^p_{eq}$$

$$\varepsilon^p_{22} = \frac{1}{2} \varepsilon^p_{eq} = \varepsilon^p_{33}$$

Then the final expression for σ_{11} will take the form:

$$\sigma_{11} = -R\left(-\varepsilon_{11}^{p}\right)$$

where $\varepsilon_{11}^{p} = \varepsilon_{11}^{p}(\varepsilon_{11})$:

$$\varepsilon_{11} = \frac{\sigma_{11}}{E} + \varepsilon_{11}^{p}$$

Results

№	Value	Point	Description	Target	CAE Fidesys result	Error, %
1	Stress tensor components for t=0.2	(5, 0, 1)	σxx, Pa	-60117.782	-6.188E+04	-2.93%
2	Stress tensor components for t=0.4	(5, 0, 1)	σxx, Pa	-74207.347	-7.262E+04	-2.14%
3	Stress tensor components for t=0.6	(5, 0, 1)	σxx, Pa	-96336.05	-9.657E+04	-0.24%
4	Stress tensor components for t=0.8	(5, 0, 1)	σxx, Pa	-108917.197	-1.041E+05	-4.39%
5	Stress tensor components for t=1	(5, 0, 1)	σxx, Pa	-113650.937	-1.137E+05	-0.0%

CAE Fidesys script:

```
reset
set default element hex
#{h=1}
brick x {5*h} y {h} z {h}
move volume 1 x {5*h/2} y {h/2} z {h/2}
create material 1
modify material 1 name "material"
modify material 1 set property 'MODULUS' value 5.1e6
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'COHESION' value 15000
modify material 1 set property 'INT_FRICTION_ANGLE' value 0
```

modify material 1 set property 'DILATANCY_ANGLE' value 0

create table 1 file "relative_path_to_model" #commandRelativePath:Models\lider_hardening.csv

modify table 1 dependency strain

modify material 1 set property 'SIGMA_CURVE' table 1

block 1 volume 1

block 1 material 'material'

block 1 element solid order 1

curve 2 4 6 8 interval 20

surface 4 6 size {h/4}

mesh volume 1

create displacement on surface 3 dof 2 fix 0

create displacement on surface 2 dof 3 fix 0

create displacement on volume 1 dof 1 fix 0

#compress

bcdep displacement 3 value '-2*x/5'

analysis type static elasticity plasticity dim3

nonlinearopts maxiters 50 minloadsteps 100 maxloadsteps 100 tolerance 1e-3 targetiter 5

Reference

[1] RICE JR: The localization of plastic deformations, in Theoretical and Applied Mechanics (1976)

1.12. Test case №1.12

Problem description

Tension of an elastoplastic rectangular parallelepiped (multilinear isotropic hardening)

Input values

Geometrical model:

- Parallelepiped 5x1x1;

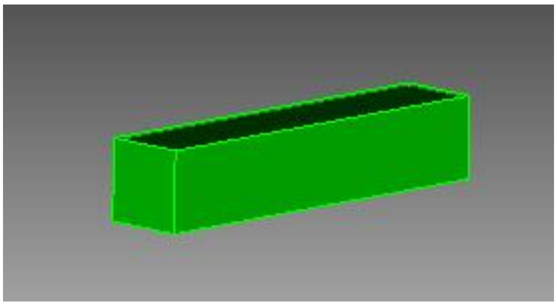

Fig 17 – Geometrical model

Bordery conditions:

- For face y = 0 u_y = 0;
- For face z = 0 u_z = 0;
- For whole model u_x = 2*x/5

Material properties:

- Young's modulus E = 5.1e+6;
- Poisson ratio ν = 0.25;
- Cohesion c = 15000;
- Internal friction angle ϕ = 0;
- Dilatation angle ψ = 0;

  The hardening given by the stress / plastic strain curve (tension) imported from the lider_hardening.csv file:

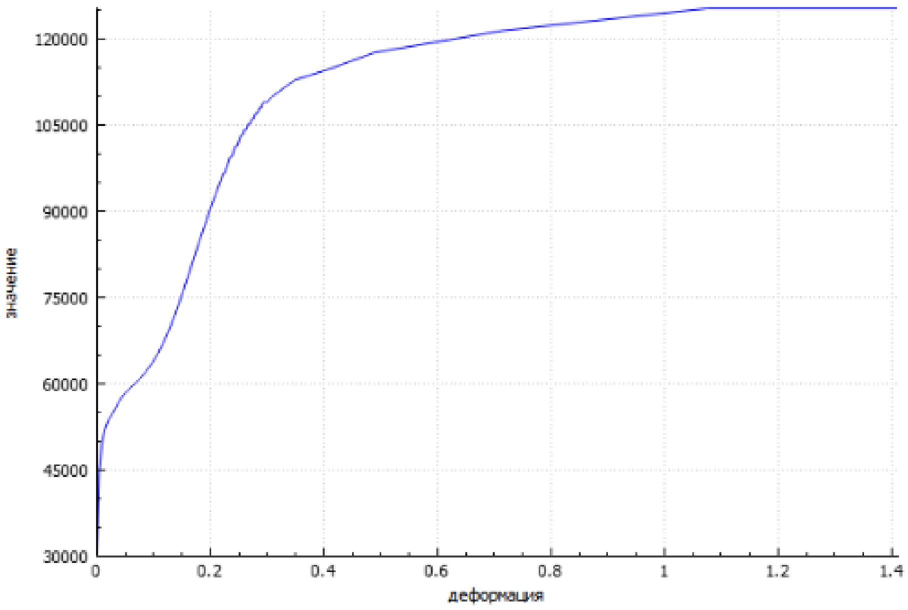

Fig 18 – Hardening curve

Mesh:

- Second order hexahedrons.

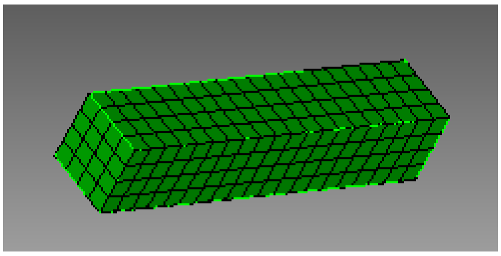

Fig 19 – Mesh

Target results

№	Value	Point	Description	Target
1	Stress tensor components for t=0.2	(5, 0, 1)	σ_{xx}, Pa	60117.782
2	Stress tensor components for t=0.4	(5, 0, 1)	σ_{xx}, Pa	74207.347
3	Stress tensor components for t=0.6	(5, 0, 1)	σ_{xx}, Pa	96336.05
4	Stress tensor components for t=0.8	(5, 0, 1)	σ_{xx}, Pa	108917.197

5	Stress tensor components for t=1	(5, 0, 1)	σxx, Pa	113650.937

Analytical solution

From Hooke's law, elastic strain are related to stress as follows:

$$\varepsilon_{11}^{2} = \frac{1}{E}(\sigma_{11} - \nu(\sigma_{22} + \sigma_{33}))$$

$$\varepsilon_{22}^{2} = \frac{1}{E}(\sigma_{22} - \nu(\sigma_{11} + \sigma_{33}))$$

$$\varepsilon_{33}^{2} = \frac{1}{E}(\sigma_{33} - \nu(\sigma_{22} + \sigma_{11}))$$

Expressions for strain ε_{ij} are written as:

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_j}\right)$$

Based on the boundary conditions, $\sigma_{22} = \sigma_{33} = 0$, then Hooke's law and the expression for ε_{ij} can be written as follows:

$$\varepsilon_{11}(t) = t\frac{\partial u_1}{\partial x_1} = -0.4t$$

$$\varepsilon_{11}^{e} = \frac{\sigma_{11}}{E}$$

$$\varepsilon_{22}^{e} = -\frac{\nu\sigma_{11}}{E} = \varepsilon_{33}^{e}$$

For this case, the yield stress is reached when the strain ε_{ij} reaches the value:

$$\varepsilon_c = -\frac{\sigma_c}{E} = -\frac{2c}{E} = 0.00588$$

It is achieved at a time t equal to

$$t_c = \frac{\varepsilon_c}{\varepsilon_{11}(1)} = \frac{-0.00588}{-0.4} = 0.0147$$

After reaching the yield point, the material acts according to the Drucker-Prager plasticity criterion

$$F\left(\sigma, \varepsilon_{eq}^{p}\right) = \sigma_{eq} + \beta\sigma - R\left(\varepsilon_{eq}^{p}\right) = 0$$

where σ_{eq} - equivalent stress, H, β, σ_y – given constants, σ - the first invariant of the stress tensor, ε_{eq}^{p} – equivalent plastic strain

$$\sigma_{eq} = \sqrt{\frac{3}{2} S_{ij} \cdot S_{ij}}$$

$$\sigma = \sigma_{11} + \sigma_{22} + \sigma_{33}$$

$$\beta = \frac{2 \sin\phi}{3 - \sin\phi} = 0$$

$$\sigma_y = \frac{6c \cos\phi}{3 - \sin\phi} = 2c$$

$$\varepsilon_{eq}^{p} = \sqrt{\frac{2}{3} e_{ij}^{p} \cdot e_{ij}^{p}}$$

where S_{ij} - stress tensor deviator, e_{ij}^{p} - plastic strain tensor deviator, ε^{p} - the first invariant of the - plastic strain tensor

$$S_{ij} = \sigma_{ij} - \frac{\sigma}{3} \delta_{ij}$$

$$e_{ij}^{p} = \varepsilon_{ij}^{p} - \frac{\varepsilon^{p}}{3} \delta_{ij}$$

$$\varepsilon^{p} = \varepsilon_{11}^{p} + \varepsilon_{22}^{p} + \varepsilon_{33}^{p}$$

For the Drucker-Prager plasticity model, the relationship between stress and plastic strain has the following form

$$\varepsilon_{ij}^{p} = \varepsilon_{eq}^{p} \left(-\frac{3}{2} \frac{S_{ij}}{\sigma_{eq}} + \beta\delta_{ij} \right)$$

Full strain is a sum of elastic and plastic:

$$\varepsilon_{ij} = \varepsilon_{ij}^{p} + \varepsilon_{ij}^{e}$$

From boundary conditions, $\sigma_{22} = \sigma_{33} = 0$, then we can evaluate σ_{eq} and σ

$$\sigma = \sigma_{11}, \sigma_{eq} = |\sigma_{11}|$$

Since we consider uniaxial compression, $\sigma_{11} < 0$ и $\varepsilon_{11}^{p} < 0$, then the expressions for the criterion of plasticity and the relationship between stress and plastic strain take on a simpler form:

$$\sigma_{11} = -R\left(\varepsilon^p_{eq}\right)$$

$$\varepsilon^p_{11} = -\varepsilon^p_{eq}$$

$$\varepsilon^p_{22} = \frac{1}{2}\varepsilon^p_{eq} = \varepsilon^p_{33}$$

Then the final expression for σ_{11} will take the form:

$$\sigma_{11} = -R\left(-\varepsilon^p_{11}\right)$$

where $\varepsilon^p_{11} = \varepsilon^p_{11}(\varepsilon_{11})$:

$$\varepsilon_{11} = \frac{\sigma_{11}}{E} + \varepsilon^p_{11}$$

Results

№	Value	Point	Description	Target	CAE Fidesys result	Error, %
1	Stress tensor components for t=0.2	(5, 0, 1)	σxx, Pa	60117.782	6.188E+04	2.93%
2	Stress tensor components for t=0.4	(5, 0, 1)	σxx, Pa	74207.347	7.262E+04	2.14%
3	Stress tensor components for t=0.6	(5, 0, 1)	σxx, Pa	96336.05	9.657E+04	0.24%
4	Stress tensor components for t=0.8	(5, 0, 1)	σxx, Pa	108917.197	1.041E+05	4.39%
5	Stress tensor components for t=1	(5, 0, 1)	σxx, Pa	113650.937	1.137E+05	0.0%

CAE Fidesys script:

```
reset
set default element hex
#{h=1}
brick x {5*h} y {h} z {h}
move volume 1 x {5*h/2} y {h/2} z {h/2}
create material 1
```

```
modify material 1 name "material"
modify material 1 set property 'MODULUS' value 5.1e6
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'COHESION' value 15000
modify material 1 set property 'INT_FRICTION_ANGLE' value 0
modify material 1 set property 'DILATANCY_ANGLE' value 0
create table 1 file "relative_path_to_model" #commandRelativePath:Models\lider_hardening.csv
modify table 1 dependency strain
modify material 1 set property 'SIGMA_CURVE' table 1
block 1 volume 1
block 1 material 'material'
block 1 element solid order 1
curve 2 4 6 8 interval 20
surface 4 6 size {h/4}
mesh volume 1
create displacement on surface 3 dof 2 fix 0
create displacement on surface 2 dof 3 fix 0
create displacement on volume 1 dof 1 fix 0
#compress
bcdep displacement 3 value '2*x/5'
analysis type static elasticity plasticity dim3
nonlinearopts maxiters 1000 minloadsteps 10 maxloadsteps 1000000 tolerance 1e-3 targetiter 5
```

Reference

[1] RICE JR: The localization of plastic deformations, in Theoretical and Applied Mechanics (1976)

2. Test cases with numerically approximate analytical solutions

2.1. Test case №2.1

Problem description

Determination of effective mechanical characteristics for a cube of a homogeneous isotropic material.

Input values

Material properties:

- Isotropic;
- Young's modulus E = 1 Pa;
- Poisson ratio ν = 0.25;
- Density ρ = 1 kg/m^3;
- Thermal conductivity coefficient ϰ = 1 W/(m·K);
- Thermal expansion coefficient α = 1 K^{-1}.

Geometrical model

- Cube with edge length of 1 m;
- Homogeneous material.

Boundary conditions:

- Non-periodic

Mesh:

- First order hexahedrons.

Target results

№	Value	Description	Unit	Target
1	Effective thermal expansion coefficients	α_{11}	K^{-1}	1
2	Effective thermal expansion coefficients	α_{22}	K^{-1}	1
3	Effective thermal expansion coefficients	α_{33}	K^{-1}	1

Numerically approximate analytical solution

Let us consider the representative volume V_0, allocated in the initial state, before deformation. At its boundary, we set the boundary conditions in the form of zero pressure

$$N \cdot \sigma|_{\Gamma_0} = 0$$

we change the temperature of the entire volume by ΔT and solve the boundary value problem of the elasticity theory on the representative volume

$$\nabla \cdot \sigma = 0$$

As a result of calculating the described problem, we obtain the distribution field of the strain tensor E on a representative volume. We average it by volume:

$$E^e = \frac{1}{V} \int_V E dV$$

As a result, we have that we set the same temperature change ΔT for the representative volume and no more boundary conditions, except for zero pressure at the boundary - and as a result of averaging we obtained the effective strain tensorE^e. We will seek effective thermoelastic characteristics in the form

$$E^e = \alpha_{ij} \Delta T$$

For a homogeneous material, a numerically approximate analytical solution is trivial: with averaging, we should obtain effective thermal expansion coefficients, equal to the thermal expansion coefficients of this homogeneous material. This works for isotropic, transversely isotropic, and orthotropic materials.

Results

First order hexahedral mesh

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal expansion coefficients	α_{11}	K^{-1}	1	1	0
2	Effective thermal expansion coefficients	α_{22}	K^{-1}	1	1	0
3	Effective thermal expansion coefficients	α_{33}	K^{-1}	1	1	0

CAE Fidesys script:

```
reset
brick x 1
volume 1 scheme Map
volume 1 size 0.5
mesh volume 1
create material 1 name 'Material1'
modify material 1 set property 'MODULUS' value 1
```

modify material 1 set property 'POISSON' value 0.25

modify material 1 set property 'ISO_THERMAL_EXPANSION' value 1

modify material 1 set property 'ISO_CONDUCTIVITY' value 1

block 1 volume 1

block 1 material 'Material1'

block 1 element solid order 2

analysis type effectiveprops heattrans dim3

periodicbc off

Reference

[1] Вишняков Л.Р., Грудина Т.В., Кадыров В.Х., Карпинос Д.М., Олейник В.И., Сапожникова А.Б., Тучинский Л.И. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

[2] Кристенсен Р. Введение в механику композитов. – М.: «Мир», 1982. – 334 с.

[3] Победря Б.Е. Механика композиционных материалов. - М.: Издательство Московского университета, 1984. - 335 с.

2.2. Test case №2.2

Problem description

Determination of effective mechanical properties for a cube of homogeneous orthotropic material.

Input values

Material Properties:

- Orthotropic;
- Young's modulus E_x = 12 Pa;
- Young's modulus E_y = 8 Pa;
- Young's modulus E_z = 4 Pa;
- Principal Poisson's ratio ν_{xy} XY = 0.375;
- Principal Poisson's ratio ν_{xz} = 0.75;
- Principal Poisson's ratio ν_{yz} = 0.5;
- Density ρ = 1 kg/m^3;
- Shear modulus G_{xy} =3 Pa;
- Shear modulus G_{xz} = 2 Pa;
- Shear modulus G_{yz} = 1 Pa;
- Thermal expansion coefficient $\alpha_x = 1\ K^{-1}$;
- Thermal expansion coefficient $\alpha_y = 2\ K^{-1}$;
- Thermal expansion coefficient $\alpha_z = 3\ K^{-1}$.

Geometrical model

- Cube with edge length of 1 m;
- Homogeneous material.

Boundary conditions:

- Non-periodic

Mesh:

- Second order hexahedrons.

Target results

№	Value	Description	Unit	Target
1	Effective thermal expansion coefficients	α_{11}	K^{-1}	1
2	Effective thermal expansion coefficients	α_{22}	K^{-1}	2
3	Effective thermal expansion coefficients	α_{33}	K^{-1}	3

Numerically approximate analytical solution

Numerically approximate analytical solution given in part 2.1.

Results

Second order hexahedral mesh

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal expansion coefficients	α_{11}	K^{-1}	1	1	0.00%
2	Effective thermal expansion coefficients	α_{22}	K^{-1}	2	2	0.00%
3	Effective thermal expansion coefficients	α_{33}	K^{-1}	3	3	0.00%

CAE Fidesys script:

```
reset
set default element hex
brick x 1.0
volume 1 size 0.5
mesh volume 1
create material 1
modify material 1 name 'Material 1'
modify material 1 set property 'ORTHOTROPIC_E_X' value 12
modify material 1 set property 'ORTHOTROPIC_E_Y' value 8
modify material 1 set property 'ORTHOTROPIC_E_Z' value 4
modify material 1 set property 'ORTHOTROPIC_PR_XY' value 0.375
modify material 1 set property 'ORTHOTROPIC_PR_XZ' value 0.75
modify material 1 set property 'ORTHOTROPIC_PR_YZ' value 0.5
modify material 1 set property 'ORTHOTROPIC_G_XY' value 3
modify material 1 set property 'ORTHOTROPIC_G_XZ' value 2
modify material 1 set property 'ORTHOTROPIC_G_YZ' value 1
modify material 1 set property 'ORTHO_THERMAL_EXPANSION_X' value 1
modify material 1 set property 'ORTHO_THERMAL_EXPANSION_Y' value 2
```

modify material 1 set property 'ORTHO_THERMAL_EXPANSION_Z' value 3

modify material 1 set property 'DENSITY' value 1

modify material 1 set property 'ORTHO_CONDUCTIVITY_X' value 1

modify material 1 set property 'ORTHO_CONDUCTIVITY_Y' value 2

modify material 1 set property 'ORTHO_CONDUCTIVITY_Z' value 3

block 1 volume 1

block 1 material 'Material 1'

block 1 element solid order 2

analysis type effectiveprops heattrans dim3

periodicbc off

Reference

[1] Вишняков Л.Р., Грудина Т.В., Кадыров В.Х., Карпинос Д.М., Олейник В.И., Сапожникова А.Б., Тучинский Л.И. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

[2] Кристенсен Р. Введение в механику композитов. – М.: «Мир», 1982. – 334 с.

[3] Победря Б.Е. Механика композиционных материалов. - М.: Издательство Московского университета, 1984. - 335 с.

2.3. Test case №2.3

Problem description

Determination of effective mechanical characteristics for a cube of a homogeneous transversely isotropic material.

Input values

Material Properties:

- Transversely isotropic;
- Young's modulus E_T = 3 Pa;
- Young's modulus E_L = 4 Pa;
- Principal Poisson's ratio ν_T = 0.25;
- Principal Poisson's ratio ν_{TL} = 0.5;
- Shear modulus G_{TL} = 1 Pa;
- Thermal expansion coefficient $\alpha_T = 1\ K^{-1}$;
- Thermal expansion coefficient $\alpha_L = 2\ K^{-1}$.

Geometrical model

- Cube with edge length of 1 m;
- Homogeneous material.

Boundary conditions:

- Non-periodic

Mesh:

- First order hexahedrons.

Target results

№	Value	Description	Unit	Target
1	Effective thermal expansion coefficients	α_{11}	K^{-1}	1
2	Effective thermal expansion coefficients	α_{22}	K^{-1}	1
3	Effective thermal expansion coefficients	α_{33}	K^{-1}	2

Numerically approximate analytical solution

Numerically approximate analytical solution given in part 2.1.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal expansion coefficients	α_{11}	K^{-1}	1	1	0.00%
2	Effective thermal expansion coefficients	α_{22}	K^{-1}	1	1	0.00%
3	Effective thermal expansion coefficients	α_{33}	K^{-1}	2	2	0.00%

CAE Fidesys script:

```
reset
brick x 1
volume 1 scheme Map
volume 1 size 0.5
mesh volume 1
create material 1
modify material 1 set property 'TR_ISO_CONDUCTIVITY_T' value 1
modify material 1 set property 'TR_ISO_CONDUCTIVITY_L' value 2
block 1 volume 1
block 1 material 1
block 1 element solid order 2
analysis type effectiveprops heattrans dim3
periodicbc off
```

Reference

[1] Вишняков Л.Р., Грудина Т.В., Кадыров В.Х., Карпинос Д.М., Олейник В.И., Сапожникова А.Б., Тучинский Л.И. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

[2] Кристенсен Р. Введение в механику композитов. – М.: «Мир», 1982. – 334 с.

[3] Победря Б.Е. Механика композиционных материалов. - М.: Издательство Московского университета, 1984. - 335 с.

2.4. Test case №2.4

Problem Description

Determination of effective mechanical properties for a single layer fibrous composite.

Input values

Material Properties:

- Matrix material:
  - Isotropic;
  - Young's modulus = 1 Pa;
  - Poisson ratio = 0.25;
  - Thermal conductivity coefficient = 2 $\frac{W}{m*K}$.
- Thread material:
  - Isotropic;
  - Young's modulus = 1 Pa;
  - Poisson ratio = 0.25;
  - Thermal conductivity coefficient = 10 $\frac{W}{m*K}$.

Geometrical model:

- Parallelepiped 4 x 16 x 16;
- Thread of length 25 and radius of 2.85459861019 (selected so the thread volume concentration in composite is equal to 10%) runs through center line parallel to the X axis;
- Thread: λ = 10;
- Matrix: λ = 2.

Boundary conditions:

- Periodic.

Mesh:

- First order tetrahedrons.

Target results

№	Value	Description	Unit	Target
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	2.8
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	2.28571
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	2.28571

Numerically approximate analytical solution

Numerically approximate analytical solutiontaken from 1.6.2 paragraph of Karpinos's "Composite materials". Effective thermal conductivity coefficients determined by following formula:

$$\lambda_x = \gamma_f \lambda_f + \gamma_m \lambda_m$$

$$\lambda_y = \lambda_z \approx \lambda_m \frac{1 + \gamma_f + \gamma_m \frac{\lambda_m}{\lambda_f}}{\gamma_m + (1 + \gamma_f)\frac{\lambda_m}{\lambda_f}}$$

Fibers here directed along X axis, λ_f, λ_m - thermal conductivity coefficients of thread and matrix, γ_f, γ_m - volume concentration of thread and matrix (in total they are equal to 1).

Boundary conditions - only periodic.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	2.8	2.800E+00	<0.00
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	2.28571	2.288E+00	0.10
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	2.28571	2.288E+00	0.10

CAE Fidesys script:

```
reset
#{length = 25.0}
#{pitch = 16.0}
#{thick = 16.0}   # thickness
#{conc = 10}    # cord concentration, percents
#{rad = sqrt( 0.01 * pitch * thick * conc / 3.1415926)}
#{size = 3.0}
create brick width {length} depth {pitch} height {thick}
create cylinder height {length} radius {rad}
volume 2 rotate 90.0 about y
subtract volume 2 from volume 1 keep
```

```
delete volume 1
imprint volume all
merge volume all
volume all scheme Tetmesh
volume all size {size}
mesh volume all
create material 1 name 'fiber'
modify material 1 set property 'MODULUS' value 1
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'ISO_CONDUCTIVITY' value 10
create material 2 name 'matrix'
modify material 2 set property 'MODULUS' value 1
modify material 2 set property 'POISSON' value 0.25
modify material 2 set property 'ISO_CONDUCTIVITY' value 2
block 1 volume 2
block 2 volume 3
block 1 material 'fiber'
block 2 material 'matrix'
block 1 2 element solid order 2
analysis type effectiveprops heattrans dim3
periodicbc on
```

Reference

[1] Карпинос Д. М. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

2.5. Test case №2.5

Problem Description

Determination of effective mechanical properties for a single layer fibrous composite.

Input values

Material Properties:

- Matrix material:
  - Isotropic;
  - Young's modulus = 2 Pa;
  - Poisson ratio = 0.3;
  - Thermal conductivity coefficient =$7.7 * 10^{-5} \frac{W}{m*K}$.
- Thread material:
  - Isotropic;
  - Young's modulus = 2000 Pa;
  - Poisson ratio = 0.2;
  - Thermal conductivity coefficient = $1.3 * 10^{-5} \frac{W}{m*K}$.

Geometrical model:

- Parallelepiped 25 x 16 x 16;
- Thread of length 25 and radius of 2.85459861019 (selected so the thread volume concentration in composite is equal to 10%) runs through center line parallel to the X axis;
- Thread: λ = 10;
- Matrix: λ = 2.

Boundary conditions:

- Periodic.

Mesh:

- First order tetrahedrons.

Target results

№	Value	Description	Unit	Target
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m * K}$	$1.35709 * 10^{-5}$
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m * K}$	$8.58878 * 10^{-5}$
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m * K}$	$8.58878 * 10^{-5}$

Numerically approximate analytical solution

Numerically approximate analytical solutiontaken from 1.6.2 paragraph of Karpinos's "Composite materials". Effective thermal conductivity coefficients determined by following formula:

$$\lambda_x = \gamma_f \lambda_f + \gamma_m \lambda_m$$

$$\lambda_y = \lambda_z \approx \lambda_m \frac{1 + \gamma_f + \gamma_m \frac{\lambda_m}{\lambda_f}}{\gamma_m + (1 + \gamma_f)\frac{\lambda_m}{\lambda_f}}$$

Fibers here directed along X axis, λ_f, λ_m - thermal conductivity coefficients of thread and matrix, γ_f, γ_m - volume concentration of thread and matrix (in total they are equal to 1).

Boundary conditions - only periodic.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	$1.35709 * 10^{-5}$	1.358E-05	0.05
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	$8.58878 * 10^{-5}$	8.308E-05	3.27
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	$8.58878 * 10^{-5}$	8.477E-05	1.31

CAE Fidesys script:

```
reset
set default element hex
#{length = 25.0}
#{pitch = 16.0}
#{thick = 16.0}   # thickness
#{conc = 10}    # cord concentration, percents
#{rad = sqrt( 0.01*pitch*thick*conc/3.1415926)}
#{size = 1.0}
create brick width {length} depth {pitch} height {thick}
create cylinder height {length} radius {rad}
volume 2 rotate 90.0 about y
subtract volume 2 from volume 1 keep
```

```
delete volume 1
imprint volume all
merge volume all
volume all size {size}
curve 18 20 22 24 interval 10
mesh volume all
create material 1 name 'fiber'
modify material 1 set property 'MODULUS' value 2000
modify material 1 set property 'POISSON' value 0.2
modify material 1 set property 'ISO_THERMAL_EXPANSION' value 1.3e-5
create material 2 name 'matrix'
modify material 2 set property 'MODULUS' value 2
modify material 2 set property 'POISSON' value 0.3
modify material 2 set property 'ISO_THERMAL_EXPANSION' value 7.7e-5
block 1 volume 2
block 2 volume 3
block 1 material 'fiber'
block 2 material 'matrix'
block all element solid order 2
analysis type effectiveprops heatexpansion dim3
periodicbc on
```

Reference

1. Карпинос Д. М. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592с.

2.6. Test case №2.6

Problem Description

Determination of effective mechanical properties for a laminated composite containing layers of two materials

Input values

Material Properties:

- Rubber:
  - Isotropic;
  - Young's modulus = 2 Pa;
  - Poisson ratio = 0.49;
  - Thermal conductivity coefficient =1 $\frac{W}{m*K}$.
- Steel:
  - Isotropic;
  - Young's modulus = 2e5 Pa;
  - Poisson ratio = 0.25;
  - Thermal conductivity coefficient = 40 $\frac{W}{m*K}$.

Geometrical model:

- Cube with edge length of 1.3;
- In the middle (perpendicular to the Z axis) of the cube there is a steel layer with thickness of 0.3;.

Boundary conditions:

- Periodic.

Mesh:

- Second order hexahedrons.

Target results

№	Value	Description	Unit	Target
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	10.0
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	10.0
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	1.29032

Numerically approximate analytical solution

A laminated composite consists of several layers of different materials glued together. In formulas [1], it is assumed that the layers lie in the XY plane.

$$\lambda_x = \lambda_y = \langle \lambda \rangle,$$

$$\lambda_z = \frac{1}{\langle 1/\lambda \rangle},$$

where the symbols ‹› mean the averaging of the value over the volume, that is, in fact, over the height.

The boundary conditions are strictly periodic.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	10.0	10	0.00
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	10.0	10	0.00
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	1.29032	1.291E+00	0.09

CAE Fidesys script:

```
cubit.cmd("reset")
 rub_thick = 1.0
steel_thick = 0.3
 rub_number = 1
 length = 1.3 # length
width = 1.3  # width
height = rub_number*(rub_thick + steel_thick)   # height
 def lambda_Calc_E_nu (E, nu): return E * nu / ((1+nu)*(1-2*nu))
 def G_Calc_E_nu(E, nu): return E / (2 + 2*nu)
 # steel constants
steel_E = 2.0e5
steel_nu = 0.25
steel_cond = 40.0
steel_lambda = lambda_Calc_E_nu(steel_E, steel_nu)
```

```
steel_G = G_Calc_E_nu(steel_E, steel_nu)

# rubber constants
rub_E = 2.0
rub_nu = 0.49
rub_cond = 1.0
rub_lambda =  lambda_Calc_E_nu(rub_E,rub_nu)
rub_G = G_Calc_E_nu(rub_E, rub_nu)
 mesh_size = 0.1
cubit.cmd("brick x " + str(length) + " y " + str(width) + " z " + str(height))
for i in range(0, rub_number): cubit.cmd( "webcut body all with plane zplane offset " + str(0.5*rub_thick + i*(rub_thick+steel_thick) - 0.5*height ) + " imprint merge")
 for i in range(0, rub_number): cubit.cmd( "webcut body all with plane zplane offset " + str( (i+1)*(rub_thick+steel_thick) - 0.5*height - 0.5*rub_thick) + " imprint merge" )
 # rubber block
command1 = "block 2 volume"
for i in range(1, rub_number+2): command1 = command1 + " " + str(i)
cubit.cmd(command1)
 # steel block
command2 = "block 1 volume"
for i in range(rub_number+2, 2*rub_number+2): command2 = command2 + " " + str(i)
cubit.cmd(command2)
 cubit.cmd("imprint volume all")
cubit.cmd("merge volume all")
 # materials
cubit.cmd("create material 1 name 'steel'")
cubit.cmd("create material 2 name 'rubber'")
cubit.cmd("modify material 1 set property 'MODULUS' value " + str(steel_E))
cubit.cmd("modify material 1 set property 'POISSON' value " + str(steel_nu))
cubit.cmd("modify material 1 set property 'ISO_CONDUCTIVITY' value " + str(steel_cond))
cubit.cmd("modify material 2 set property 'MODULUS' value " + str(rub_E))
cubit.cmd("modify material 2 set property 'POISSON' value " + str(rub_nu))
cubit.cmd("modify material 2 set property 'ISO_CONDUCTIVITY' value " + str(rub_cond))
# blocks
cubit.cmd("block 1 material 'steel'")
```

```
cubit.cmd("block 2 material 'rubber'")
cubit.cmd("block 1 2 element solid order 2")
 # meshing
cubit.cmd("volume all scheme Sweep")
cubit.cmd("volume all size " + str(mesh_size) )
cubit.cmd("mesh volume all")

# solution settings
cubit.cmd("analysis type effectiveprops heattrans dim3")
cubit.cmd("periodicbc on")
cubit.cmd("solver method direct use_uzawa auto try_other on")
```

Reference

[1] Победря Б.Е. Механика композиционных материалов. – М: Изд-во МГУ, 1984. – 335 с.

2.7. Test case №2.7

Problem Description

Determination of effective mechanical properties for a single layer fibrous composite.

Input values

Material Properties:

- Matrix material:
  - Isotropic;
  - Young's modulus = 1 Pa;
  - Poisson ratio = 0.25;
  - Thermal conductivity coefficient =2 $\frac{W}{m*K}$.
- Thread material:
  - Isotropic;
  - Young's modulus = 1 Pa;
  - Poisson ratio = 0.25;
  - Thermal conductivity coefficient = 10 $\frac{W}{m*K}$.

Geometrical model:

- 16 x 16 square;
- In the center there is a circle (thread) with a radius of 2.85459861019 (selected so the thread volume concentration in composite is equal to 10%);

Boundary conditions:

- Periodic.

Mesh:

- Second order flat triangular elements.

Target results

№	Value	Description	Unit	Target
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	2.28571
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	2.28571
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	2.8

Numerically approximate analytical solution

Numerically approximate analytical solutiontaken from 1.6.2 paragraph of Karpinos's "Composite materials". Effective thermal conductivity coefficients determined by following formula:

$$\lambda_x = \gamma_f \lambda_f + \gamma_m \lambda_m$$

$$\lambda_y = \lambda_z \approx \lambda_m \frac{1 + \gamma_f + \gamma_m \frac{\lambda_m}{\lambda_f}}{\gamma_m + (1 + \gamma_f)\frac{\lambda_m}{\lambda_f}}$$

Fibers here directed along X axis, λ_f, λ_m - thermal conductivity coefficients of thread and matrix, γ_f, γ_m - volume concentration of thread and matrix (in total they are equal to 1).

Boundary conditions - only periodic.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	2.28571	2.28571	0.00%
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	2.28571	2.28571	0.00%
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	2.8	2.8	0.00%

CAE FIdesys script:

```
reset
 #{pitch = 16.0}
#{thick = 16.0}    # thickness
#{conc = 10}    # cord concentration, percents
#{rad = sqrt( 0.01*pitch*thick*conc/3.1415926)}
#{size = 1.0}
 # geometry
create surface rectangle width {pitch} depth {thick} zplane
create surface circle radius {rad} zplane
subtract body 2 from body 1 keep
delete body 1
```

```
imprint body all
merge body all
 # meshing
surface all scheme trimesh
surface all size {size}
mesh surface all
 # materials
create material 1
modify material 1 name 'fiber'
modify material 1 set property 'MODULUS' value 1
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'ISO_CONDUCTIVITY' value 10
create material 2
modify material 2 name 'matrix'
modify material 2 set property 'MODULUS' value 1
modify material 2 set property 'POISSON' value 0.25
modify material 2 set property 'ISO_CONDUCTIVITY' value 2
# blocks
block 1 add surface 2
block 2 add surface 3
block 1 material 'fiber'
block 2 material 'matrix'
block 1 2 element plane order 2
 # solution options
analysis type effectiveprops heattrans dim2
periodicbc on
```

Reference

[1] Карпинос Д. М. Композиционные материалы. Справочник. – Киев: Наукова думка, 1985. – 592 с.

2.8. Test case№2.8

Problem description

We consider the problem of an elastic strip that moves with an initial velocity and crashes into a rigid wall. During interaction, the strip is in contact with the wall (sliding contact without friction). During the solution, the interaction and separation times, as well as the corresponding displacements and velocities on the contact surface, are determined and compared with the solution given in [1]. The test task checks the correctness of:

- support of contact interaction "sliding without friction";
- support for non-conformally matched grids from spectral elements.

Input values

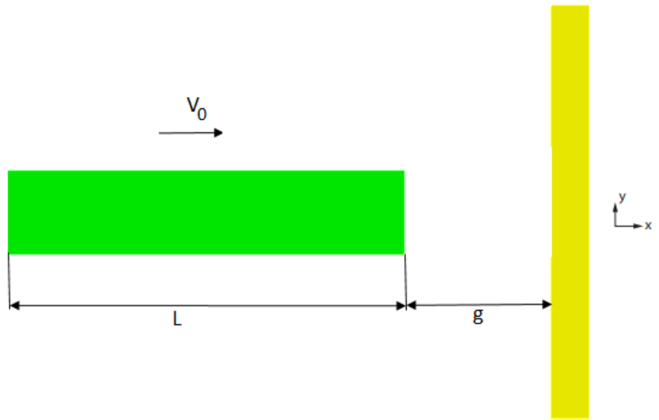

Fig 2.1.1. Geometrical model

Geometrical model

- Strip: rectangle (L=10 in, h=1 in);
- Wall: rectangle (L=5 in, h=1 in);
- Initial gap between strip and wall 0.01 in.

Material Properties:

- E_{strip}=3e7 psi, ν_{strip}=0.3;

Boundary conditions:

- The wall is fixed in all directions;
- The strip is fixed in the vertical direction;
- The strip is affected by the initial speed V_0=202.2 in/sec^2.

Mesh:

- 8-node elements.

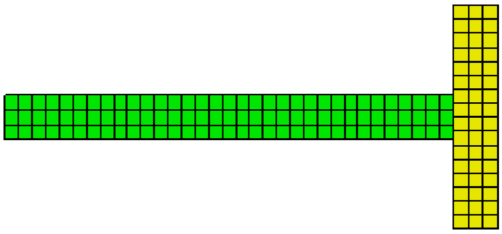

Fig 2.1.2. Mesh

Contact:

- General contact (master entity - curve 6, slave entity - curve 4);
- Friction 0;
- Accuracy 0.0005;
- Penalty method (normal contact stiffness 1, tangent contact stiffness 0.5).

Calculation settings:

- Dynamic anaysis;
- 2D;
- Plain strain;
- Full solution;
- Implicit;
- Max time 0.003 c;
- Step number 1000.

Target results

№	Value	Description	Unit	Target
1	Contact status in contact region at point (5,0,0) at t=0.00005 sec.	contact_status	-	2
2	Displacement vector component u_x at point (0,0,0) at t=0.00005 sec.	Displacement_XX	in	0.01
3	Velocity vector component v_x at point (0,0,0) at t=0.00005 sec.	Velocity_XX	In/c	202.2

№	Value	Description	Unit	Target
	Contact status in contact region at point (5,0,0) at t=0.00015 sec.	contact_status	-	0
	Displacement vector component u_x at point (0,0,0) at t=0.00015 sec.	Displacement_XX	in	0.01
	Velocity vector component v_x at point (0,0,0) at t=0.00015 sec.	Velocity_XX	In/c	-202.2

Table 2.3.1 Setting time dependency for force

Time	Force value, N
0	0
1	10^5
2	0
5	0

Numerically approximate analytical solution

Numerically approximate analytical solution given in [1].

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Contact status in contact region at point (5,0,0) at t=0.00005 sec.	contact_status	-	2	2	0.00
2	Displacement vector component u_x at point (0,0,0) at t=0.00005 sec.	Displacement_XX	in	0.01	1.011E-02	1.10
3	Velocity vector component v_x at point (0,0,0) at t=0.00005 sec.	Velocity_XX	In/c	202.2	2.022E+02	0.00

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
4	Contact status in contact region at point (5,0,0) at t=0.00015 sec.	contact_status	-	0	0	0.00

CAE Fidesys script:

```
reset
create surface rectangle width 10 height 1 zplane
create surface rectangle width 1 height 5 zplane
move Surface 2 x 5.51 include_merged
surface all size auto factor 5
undo group begin
surface all size auto factor 5
mesh surface all
undo group end
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'MODULUS' value 3e+07
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 0.73
set duplicate block elements off
block 1 add surface all
block 1 material 1 cs 1 element plane order 2
create displacement on surface 1 dof 2 dof 3 fix
create displacement on surface 2 dof all fix
create initial velocity on surface 1
modify initial velocity 1 dof 1 value 202.2
create contact master curve 6 slave curve 4 tolerance 0.0005 type general friction 0.0 preload 0.0 offset 0.0
ignore_overlap off method penalty normal_stiffness 1.0 tangent_stiffness 0.5
analysis type dynamic elasticity dim2 planestrain preload off
dynamic method full_solution scheme implicit maxtime 0.0003 steps 1000 newmark_gamma 0.005
calculation start path 'C:/fidesys01.pvd'
```

Reference:

1. N.J. Carpenter, R.L. Taylor and M.G. Katona, "Lafrange Constraints For Transient Finite Element Surface Contact", International Journal for Numerical Methods in Engineering, vol.32, 1991. pg 103-128.

2.9. Test case №2.9

Problem Description

We consider the plane static problem of material step by step changing. The goal of the assignment is to check the correctness of the material change in the solution steps. Sub-steps material properties are checked with the results in Fidesys Viewer. The test case checks the correctness:

- linear elastic mathematical model of the material;
- change of boundary conditions between loading steps;
- change of material properties between loading steps.

Input values

Geometrical model:

There is an inclusion in the plate. During the calculation, the material properties of the inclusion change.

- Plate length $a = 10$ m;
- Plate width $b = 5$ m;
- Circle with radius $R = 1$ m.

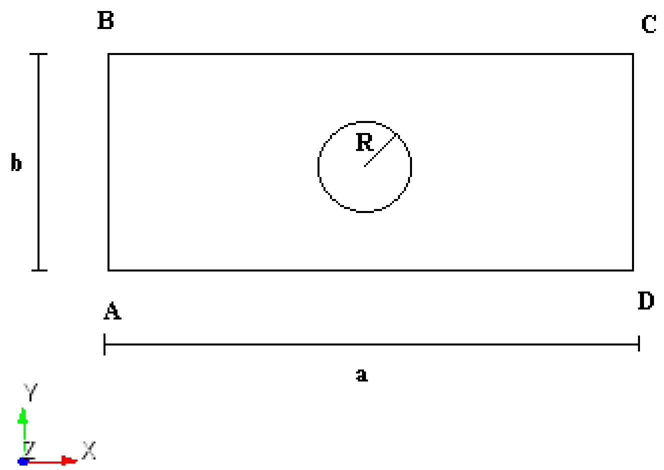

Fig 3.1.1 - Geometric model of the problem

Boundary conditions:

- The AB side is fixed on all axes and rotates;
- Sides AD and BC are fixed along the Y-axis;
- The pressure applied to the side CD with step by step load:
  - Step 1: - 1000 Pa;
  - Step 2: - 1000 Pa;
  - Step 3: 0 Па.

Material Properties:

- Material for the plate:
  - Elastic modulus $E = 2e+11$ Pa;

  - Poisson's ratio $\nu = 0.3$.
- Materials for the inclusion:
  - Material 2: $E = 0.7e+11$ Pa, $\nu = 0.34$;
  - Material 3: $E = 1e+11$ Pa, $\nu = 0.35$.

The material for the inclusion is entered in tabular form:

  - Step 1: Material 2;
  - Step 2: Material 2;
  - Step 3: Material 3.

Mesh:

- Conformal mesh;
- Quadrangular finite elements.

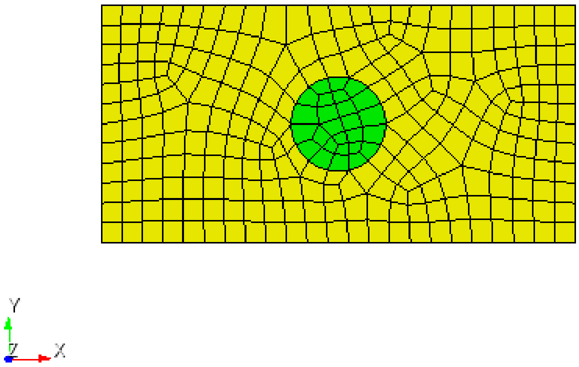

Fig 18 - Finite element mesh model

Calculation settings:

- static analisys;
- 2D plane strain state;
- elasticity;
- number of loading steps: 3.

Output Values

№	Loading steps	Value	Description	Unit	Target
1	Step 1	Young's modulus at a point (0, 0, 0)	Elasticity Modulus Young's modulus	Pa	7e10
2	Step 2	Young's modulus at a point (0, 0, 0)	Elasticity Modulus Young's modulus	Pa	7e10
3	Step 3	Young's modulus at a point (0, 0, 0)	Elasticity Modulus Young's modulus	Pa	1e11

Results

№	Loading steps	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Step 1	Young's modulus at a point (0, 0, 0)	Elasticity Modulus Young's modulus	Pa	7e10	7e10	0.00
2	Step 2	Young's modulus at a point (0, 0, 0)	Elasticity Modulus Young's modulus	Pa	7e10	7e10	0.00
3	Step 3	Young's modulus at a point (0, 0, 0)	Elasticity Modulus Young's modulus	Pa	1e11	1e11	0.00

CAE Fidesys script:

```
reset
set default element hex
create surface rectangle width 10 height 5 zplane
create surface circle radius 1 zplane
subtract surface 2 from surface 1 keep
delete surface 1
merge curve all
compress all
surface all size auto factor 4
mesh surface all
set duplicate block elements off
create material 1 from 'Steel'
create material 2
modify material 2 name ' 2'
modify material 2 set property 'POISSON' value 0.34
modify material 2 set property 'MODULUS' value 0.7e11
create material 3
modify material 3 name ' 3'
modify material 3 set property 'MODULUS' value 1e+11
modify material 3 set property 'POISSON' value 0.35
block 1 add surface 2
block 2 add surface 1
```

```
block 1 material 1
block 2 material 2
block all element plane order 2
create displacement on curve 3 dof all fix
create displacement on curve 2 4 dof 2 fix
create pressure on curve 5 magnitude 1
create table 1
modify table 1 dependency time
modify table 1 insert row 1
modify table 1 insert row 1
modify table 1 insert row 1
modify table 1 cell 1 1 value 1
modify table 1 cell 1 2 value -1000
modify table 1 cell 2 1 value 2
modify table 1 cell 2 2 value -1000
modify table 1 cell 3 1 value 3
bcdep pressure 1 table 1
block 2 step 1 2 material 2
block 2 step 3 material 3
block 1 step all
output nodalforce off midresults on record3d on log on vtu on material on
analysis type static elasticity dim2 planestrain
static steps 3
calculation start path 'C:/fidesys01.pvd'
```

2.10. Test case №2.10

Problem Description

We consider the plane problem of the formation in a preloaded, infinitely extended body (the mechanical properties of the material of which are described by the Murnaghan potential) of a circular one at the moment of the onset of the inclusion. The mechanical properties of the inclusion material are described by the Murnaghan potential. A variant of the model of the formation of an elastic inclusion is considered, which (at the moment of formation) completely repeats the shape of the removed part of the body in the case when forces act on the surface of the inclusion opposite to the forces acting on the newly formed boundary of the body (through the replacement of the material in steps). The test case checks the correctness:

- physically nonlinear mathematical model of the material;
- change of material properties between loading steps.

Input values

Geometrical model:

There is an inclusion in the plate. During the calculation, the material properties of the inclusion change.

- Plate length 100 m;
- Plate width 100 m;
- The inclusion: circle with radius $R = 1$ m.

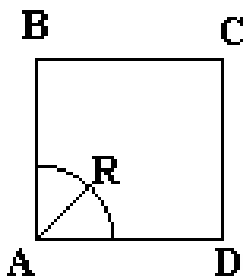

Fig 19 - Geometric model of the problem

Boundary conditions:

- In view of symmetry, ¼ part of the model is consider;
- The AB side is fixed along the X-axis;
- The AD side is fixed along the Y-axis;
- The pressure applied to the side CD with value 0.00315 Pa.

Material Properties:

- Matrix material:
  - λ_{matrix}=0.39;
  - G_{matrix} = 0.186;
  - C_{3mat} = -0.013;
  - C_{4mat} = -0.07;
  - C_{5mat} = 0.063.

- Material for the inclusion:

- $\lambda_{inclusion}$=1.07;
- $G_{inclusion}$ = 0.477;
- C_{3inc} = -0.093;
- C_{4inc} = 1.72;
- C_{5inc} = -5.31.

The material for the model is entered in tabular form:

- Step 1: Matrix material;
- Step 2: Material for the inclusion.

Mesh:

- Conformal mesh.
- Quadrangular finite elements second order.

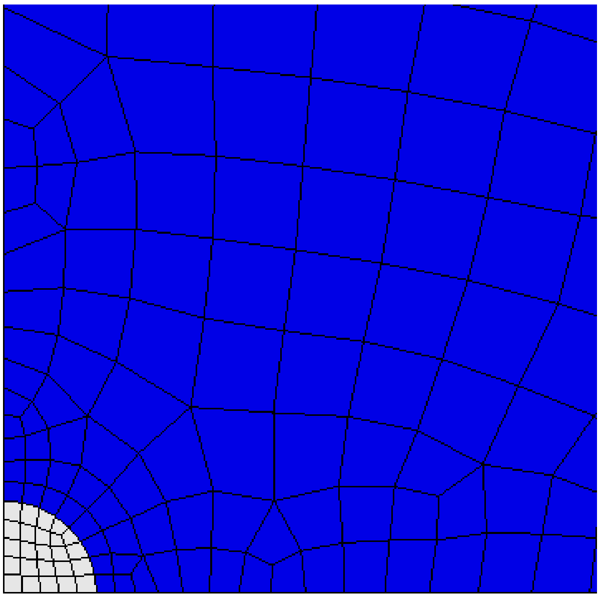

Fig 20 - Finite element mesh model

Calculation settings:

- static analisys;
- 2D plane strain state;
- elasticity;
- number of loading steps: 2.

Output Values

№	Value	Description	Unit	Target
1	Stress σ_{xx} at a point (0,0,0)	Stress XX	Pa	0.00275

Numerically approximate analytical solution

The solution algorithm is presented in [1]. Below is the result of the solution for the stresses for the inclusion and the matrix. For the criterion of this test case, the linear case is considered.

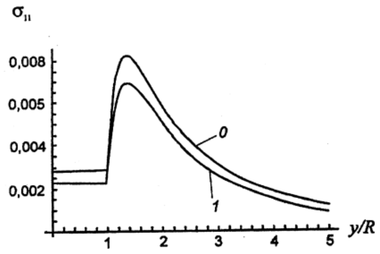

Fig 21 - Distribution for inclusion and matrix: 0 – linear solution, 1 – nonlinear solution

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Stress σ_{xx} at a point (0,0,0)	Stress XX	Pa	0.00275	2.674E-03	2.77

CAE Fidesys script:

```
reset
create surface rectangle width 100 zplane
#create surface circle radius 1 zplane
create surface ellipse major radius 0.5 minor radius {0.5-0.005601016} zplane
subtract surface 2 from surface 1 keep_tool
webcut body all with plane xplane offset 0
webcut body all with plane yplane offset 0
delete Body 4 3 7 8
delete Body 1
delete Body 2
merge all
curve 30 32 26 size 0.1
curve 30 32 26 scheme equal
curve 30 32 26 size 0.1
curve 30 32 26 scheme equal
mesh curve 30 32 26
surface 11 size auto factor 5
undo group begin
surface 11 size auto factor 5
mesh surface 11
undo group end
curve 24 7 scheme bias fine size 0.1 factor 1.09 start vertex 22 8
curve 24 7 scheme bias fine size 0.1 factor 1.09 start vertex 22 8
mesh curve 24 7
surface 9 size auto factor 5
undo group begin
```

```
surface 9 size auto factor 5
mesh surface 9
undo group end
create material 1
modify material 1 name 'Matrix'
create material 2
modify material 2 name 'Inclution'
modify material 1 set property 'MUR_SHEAR' value 0.186
modify material 1 set property 'MUR_LAME' value 0.39
modify material 1 set property 'MUR_C3' value -0.013
modify material 1 set property 'MUR_C4' value -0.07
modify material 1 set property 'MUR_C5' value 0.063
modify material 2 set property 'MUR_LAME' value 1.07
modify material 2 set property 'MUR_SHEAR' value 0.477
modify material 2 set property 'MUR_C3' value -0.93
modify material 2 set property 'MUR_C4' value 1.72
modify material 2 set property 'MUR_C5' value -5.31
modify material 2 set property 'INIT_STRESS_XZ' value 0
modify material 2 set property 'INIT_STRESS_YZ' value 0
modify material 2 set property 'INIT_STRESS_XY' value 0
modify material 2 set property 'INIT_STRESS_ZZ' value 0
modify material 2 set property 'INIT_STRESS_YY' value 0
modify material 2 set property 'INIT_STRESS_XX' value 0
set duplicate block elements off
block 1 add surface 9
block 1 name 'Matrix'
set duplicate block elements off
block 2 add surface 11
block 2 name 'Inclution'
block 1 material 1 cs 1 element plane order 2
block 2 material 1 cs 1 element plane order 2
create displacement on curve 11 dof 2 fix {0.05*0.063}
delete displacement 1
create displacement on curve 30 24 dof 2 fix
create displacement on curve 32 7 dof 1 fix
create pressure on curve 25 magnitude {-0.05*0.063}
static steps 2
block 2 step 2 material 2
analysis type static elasticity dim2 planestrain
static steps 2
calculation start path 'C:/fidesys01.pvd'
```

Reference

1. В. А. Левин, И. А. Мишин, А. В. Вершинин, Плоская задача об образовании включения в упругом нагруженном теле. Конечные деформации, Вестн. Моск. ун-та. Сер. 1. Матем., мех., 2006, номер 1, 56–59

2.11. Test case №2.11

Problem Description

We consider a tunnel heated from the inside (the temperature on the inner surface acts as a load).

Input values

Material Properties:

- Elastic modulus E = 18500 Pa;
- Poisson's ratio ν = 0.3333;
- Density ρ = 1e-8;
- Cohesion = 11;
- Internal friction angle = 35;
- Dilatancy angle = 35;
- Specific heat coefficient = 1.23;
- Conductivity = 1;
- Coefficient of thermal expansion = 1.72e-5.

Boundary conditions:

- The inner surface of the tunnel is affected by a temperature of 250°C, the temperature on the outer surface of the tunnel 0°C;
- Fixing from symmetry conditions.

Mesh:

- First order hexahedrons.

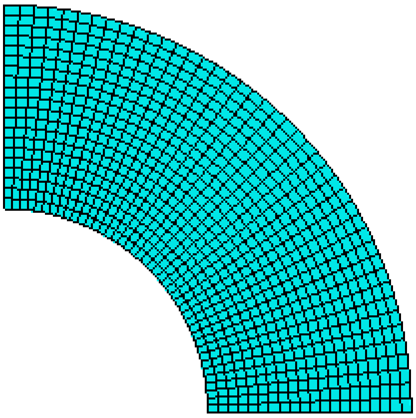

Fig 22 - Finite element mesh model

Output Values

Presented with the calculation results.

Numerically approximate analytical solution

For this problem, a numerical solution was considered obtained in the ANSYS.

Results

First order hexahedral mesh

№	Value	Point	Description	Unit	Target	CAE Fidesys result	Error, %
1	Displacement component u_x	(0.5, 0,0)	Displacement X	M	0.1558e-2	1.558E-03.	0.01
2	Displacement component u_x	(0.6, 0,0)	Displacement X	M	0.2119e-2	2.120E-03	0.03
3	Displacement component u_x	(0.7, 0,0)	Displacement X	M	0.2458e-2	2.459E-03	0.03
4	Displacement component u_x	(0.8, 0,0)	Displacement X	M	0.2668e-2	2.668E-03	0.02
5	Displacement component u_x	(0.94, 0,0)	Displacement X	M	0.278e-2	2.780E-03	0.01
6	Displacement component u_x	(0.1, 0,0)	Displacement X	M	0.2765e-2	2.765E-03	0.00
7	Plastic strain	(1, 0,0)	Plastic_Strain _XX	-	-0.225e-3	-2.315e-04	2.72
8	Plastic strain	(0.9, 0,0)	Plastic_Strain _XX	-	0.769e-4	7.381E-05	4.02
9	Plastic strain	(0.78, 0,0)	Plastic_Strain _XX	-	0.267e-3	2.636E-04	1.29
10	Plastic strain	(0.7, 0,0)	Plastic_Strain _XX	-	0.113e-3	1.120E-04	0.84
11	Plastic strain	(0.67, 0,0)	Plastic_Strain _XX	-	0	-2.879E-07	0.00
12	Plastic strain	(1, 0,0)	Plastic_Strain _YY	-	0.198e-2	1.978E-03	0.08
13	Plastic strain	(0.9, 0,0)	Plastic_Strain _YY	-	0.15e-2	1.495E-03	0.31
14	Plastic strain	(0.8, 0,0)	Plastic_Strain _YY	-	0.878e-3	8.789E-04	0.10

№	Value	Point	Description	Unit	Target	CAE Fidesys result	Error, %
15	Plastic strain	(0.7, 0,0)	Plastic_Strain_YY	-	0.175e-3	1.767E-04	0.97
16	Plastic strain	(0.67, 0,0)	Plastic_Strain_YY	-	0	-3.636E-07	0.00
17	Plastic strain	(1, 0,0)	Plastic_Strain_ZZ	-	0.1736e-3	1.811E-04	4.29
18	Plastic strain	(0.9, 0,0)	Plastic_Strain_ZZ	-	-0.1243e-3	-1.215E-04	2.23
19	Plastic strain	(0.8, 0,0)	Plastic_Strain_ZZ	-	-0.23e-3	-2.284E-04	0.71
20	Plastic strain	(0.7, 0,0)	Plastic_Strain_ZZ	-	-0.747e-4	-7.482E-05	0.16
21	Plastic strain	(0.67, 0,0)	Plastic_Strain_ZZ	-	0	1.796E-07	0.00
22	Elastic strain component ε_{xx}	(1, 0,0)	Elastic_Strain_X	-	-0.303e-3	-3.038E-04	0.27
23	Elastic strain component ε_{xx}	(0.9, 0,0)	Elastic_Strain_X	-	-0.26e-3	-2.588E-04	0.45
24	Elastic strain component ε_{xx}	(0.8, 0,0)	Elastic_Strain_X	-	-0.898e-4	8.893E-05	0.97
25	Elastic strain component ε_{xx}	(0.7, 0,0)	Elastic_Strain_X	-	0.308e-3	3.085E-04	0.17
26	Elastic strain component ε_{xx}	(0.67, 0,0)	Elastic_Strain_X	-	0.119e-2	1.190E-03	0.02
27	Elastic strain component ε_{xx}	(0.5, 0,0)	Elastic_Strain_X	-	0.274e-2	2.733E-03	0.25
28	Elastic strain component ε_{yy}	(1, 0,0)	Elastic_Strain_Y	-	0.787e-3	7.869E-04	0.01
29	Elastic strain component ε_{yy}	(0.9, 0,0)	Elastic_Strain_Y	-	0.928e-3	9.288E-0	0.08
30	Elastic strain component ε_{yy}	(0.8, 0,0)	Elastic_Strain_Y	-	0.107e-2	1.073E-03	0.28
31	Elastic strain component ε_{yy}	(0.7, 0,0)	Elastic_Strain_Y	-	0.112e-2	1.123E-03	0.29
32	Elastic strain component ε_{yy}	(0.67, 0,0)	Elastic_Strain_Y	-	0.363e-3	3.634E-04	0.10

№	Value	Point	Description	Unit	Target	CAE Fidesys result	Error, %
33	Elastic strain component ε_{yy}	(0.5, 0,0)	Elastic_Strain_Y	-	-0.1184e-2	-1.184E-03	0.01
34	Elastic strain component ε_{zz}	(1, 0,0)	Elastic_Strain_Z	-	-0.181e-3	-1.811E-04	0.03
35	Elastic strain component ε_{zz}	(0.9, 0,0)	Elastic_Strain_Z	-	-0.529e-3	-5.321E-04	0.58
36	Elastic strain component ε_{zz}	(0.8, 0,0)	Elastic_Strain_Z	-	-0.115e-2	-1.156E-03	0.52
37	Elastic strain component ε_{zz}	(0.7, 0,0)	Elastic_Strain_Z	-	-0.214e-2	-2.138E-03	0.10
38	Elastic strain component ε_{zz}	(0.67, 0,0)	Elastic_Strain_Z	-	-0.317e-2	-3.169E-03	0.03
39	Elastic strain component ε_{zz}	(0.5, 0,0)	Elastic_Strain_Z	-	-0.43e-2	-4.300E-03	0.00

CAE Fidesys script:

```
reset
set default element hex
create cylinder height 0.1 radius 0.5
create cylinder height 0.1 radius 1
subtract body 1 from body 2
webcut body 2 with plane xplane offset 0
webcut body 2 with plane yplane offset 0
delete body 2
delete body 3
create material 1
modify material 1 set property 'POISSON' value 0.3333
modify material 1 set property 'MODULUS' value 1.85e+04
modify material 1 set property 'DENSITY' value 1e-8
modify material 1 set property 'COHESION' value 11
modify material 1 set property 'DILATANCY_ANGLE' value 35
modify material 1 set property 'INT_FRICTION_ANGLE' value 35
modify material 1 set property 'SPECIFIC_HEAT' value 1.23
modify material 1 set property 'ISO_THERMAL_EXPANSION' value 1.72e-05
modify material 1 set property 'ISO_CONDUCTIVITY' value 1
set duplicate block elements off
block 1 volume 4
block 1 material 1
block 1 element solid order 2
surface 31 size 0.025
mesh surface 31
```

```
curve 11 13 40 42 interval 1
mesh curve 11 13 40 42
mesh volume 4
create temperature on surface 30 value 250
create temperature on surface 28 value 0
create displacement on surface 11 dof 1 fix 0
create displacement on surface 27 dof 2 fix 0
create displacement on surface 29 31 dof 3 fix 0
analysis type static elasticity plasticity heattrans dim3
```

2.12. Test case №2.12

Problem Description

We consider the problem of slope stability taking into account the formation of plastic zones according to the Drucker-Prager criterion. The test case checks the correctness:

- taking into account the plastic properties of the material when calculating the stress-strain state of the medium;
- Drucker-Prager plasticity criterion with symmetric hardening;
- nonlinear model for calculating mechanical strength.

Input values

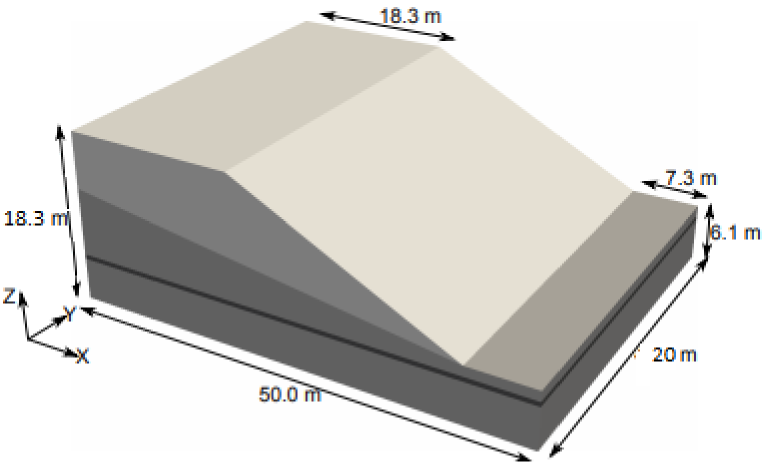

Fig. 23. Geometric model of the problem

Geometrical model:

- Typical dimensions are shown in Figure 23.

Material Properties:

- Elastic modulus E = 1e+8 Pa;
- Poisson's ratio ν = 0.3;
- Density ρ = 1918.37;
- Cohesion = 12889;
- Internal friction angle = 9.189;
- Dilatancy angle = 0.

Boundary conditions:

- The body is affected by gravity;
- Fixing from symmetry conditions.

Mesh:

- Second order hexahedra.

- Hexahedrons of the second order.

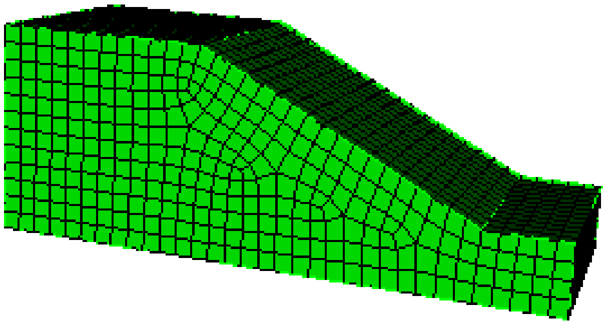

Fig. 24. Finite element mesh model

Output Values

№	Value	Description	Unit	Target
1	Displacement component u_z at a point (12.595, -20, 17.584)	Displacement Z	m	-0.0366
2	Displacement component u_x at a point(27.389, -20, 7.190)	Displacement X	m	0.01199
3	Plastic strain at a point (18.411, -20, 6.7264)	Plasticity_strain YY	-	0.59e-3
4	Plastic strain at a point (18.411, -20, 6.7264)	Plasticity_strain XX	-	0.000888

Numerically approximate analytical solution

A numerically approximate solution is presented in [1] (figures 4-5).

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Displacement component u_z at a point (12.595, -20, 17.584)	Displacement Z	m	-0.0366	-3.656E-02	0.11
2	Displacement component u_x at a point(27.389, -20, 7.190)	Displacement X	m	0.01199	1.194E-02	0.43
3	Plastic strain at a point (18.411, -20, 6.7264)	Plasticity_strain YY	-	0.59e-3	5.906E-04	0.10
4	Plastic strain at a point (18.411, -20, 6.7264)	Plasticity_strain XX	-	0.000888	8.871E-04	0.10

CAE Fidesys script:

```
reset
set node constraint on
set default element hex
create vertex 0 0 0
create vertex 50 0 0
create vertex 50 0 6.1
create vertex 42.7 0 6.1
create vertex 18.3 0 18.3
create vertex 0 0 18.3
create surface vertex 1 2 3 4 5 6
sweep surface 1 perpendicular distance 20
create material 1
modify material 1 name "dry"
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 1e+8
modify material 1 set property 'DENSITY' value 1918.367
modify material 1 set property 'DILATANCY_ANGLE' value 0
modify material 1 set property 'INT_FRICTION_ANGLE' value 9.189
modify material 1 set property 'COHESION' value 12889
set duplicate block elements off
block 1 volume 1
block 1 material "dry"
block 1 element solid order 2
curve all size 1.5
mesh curve all
mesh volume 1
create displacement on surface 7 8 dof 1 dof 2 dof 3 fix 0
create displacement on surface 1 dof 2 fix 0
create displacement on surface 2 6 dof 1 fix 0
create gravity on volume 1
modify gravity 1 dof 3 value -9.8
analysis type static elasticity plasticity dim3
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 30 tolerance 5e-2
calculation start path "kp.pvd"
```

Reference

1. Hom Nath Gharti1, Dimitri Komatitsch, Volker Oye1, Roland Martin and Jeroen Tromp Application of an elastoplastic spectral-element method to 3D slope stability analysis, INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN ENGINEERING. Int. J. Numer. Meth. Engng 2011.

2.13. Test case №2.13

Problem Description

We consider the Hertz problem for the two-dimensional case [1] for three different values of the applied force (25 N, 50 N, 100 N). Half of the cylinder is located with a convex part on a rigid base, a load is applied to the sheared part of the cylinder. The test case checks the correctness:

- setting parameters of general contact without friction in the interface;
- static solution with general contact without friction;
- the correctness of the output of the fields Contact status, Stress in contact.

Input values

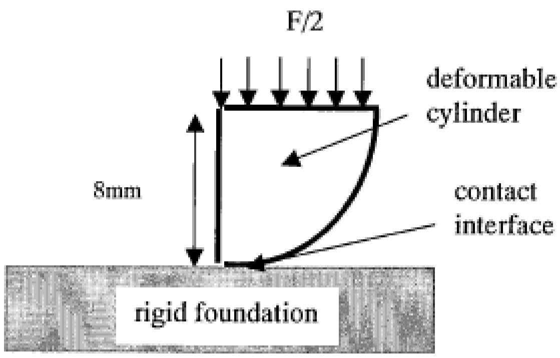

Fig. 25. Geometric model of the problem

Material Properties:

- $E_{cylinder}$=500 MPa, $\nu_{cylinder}$=0.3.

Boundary conditions:

- The base is fixed in all directions;
- The cylinder is fixed in the horizontal direction according to the symmetry condition;
- Three load cases: force F=25, 50, 100 N.

Contact:

- Nonconformal mesh;
- Friction μ=0;
- Type: general without friction.

Mesh:

- 8-node finite elements.

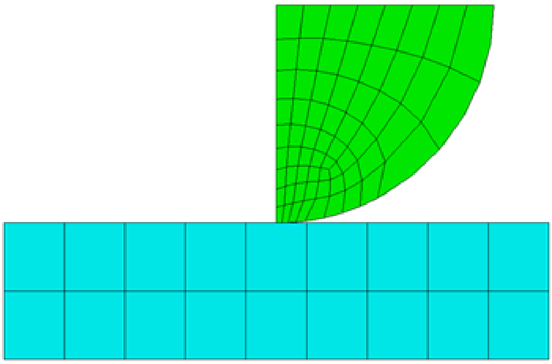

Fig. 26. Finite element mesh model

Calculation settings:

- Static analisys;
- 3D;
- Elasticity.

Output Values

№	Value	Description	Unit	Target
1	Contact status in the contact region at a point (0,0,0)	contact_status	-	2
2	Component of the stress tensor in the contact zone at a point (0,0,0) for F=25 N	contact_stress	MPa	24
3	Component of the stress tensor in the contact zone at a point (0,0,0) for F=50 N	contact_stress	MPa	35
4	Component of the stress tensor in the contact zone at a point (0,0,0) for F=100 N	contact_stress	MPa	47.5

Numerically approximate analytical solution

The problem has a numerical approximate solution published in [1] and shown in Figure 27.

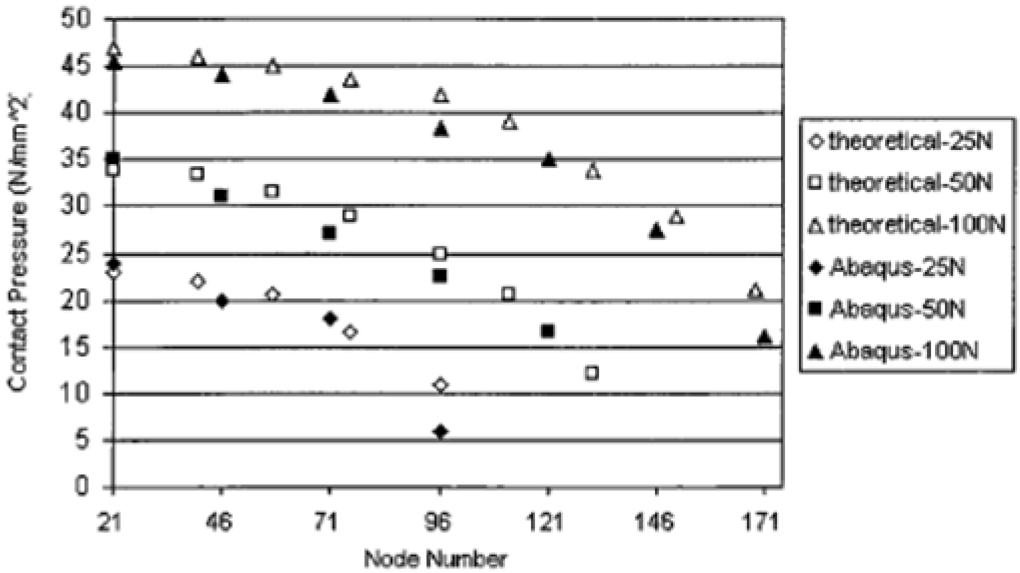

Fig.27. Results of the numerical solution of the problem

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Contact status in the contact region at a point (0,0,0)	contact_status	-	2	2	0.00
2	Components of the stress tensor in the contact zone at a point (0,0,0) for F=25 N	contact_stress	MPa	25	2.590E+01	3.60
3	Component of the stress tensor in the contact zone at a point (0,0,0) for F=50 N	contact_stress	MPa	35	3.644E+01	4.10
4	Component of the stress tensor in the contact zone at a point (0,0,0) for F=100 N	contact_stress	MPa	47	4.863E+01	3.47

CAE Fidesys script:

```
F=25
reset
set default element hex
create surface circle radius 8 zplane
```

```
webcut body 1 with plane xplane
webcut body 1 with plane yplane
delete Body 3
delete Body 2
move Surface 4 y 8 include_merged
create surface rectangle width 20 height 5 zplane
move Surface 6 y -2.499 include_merged
partition create curve 8 position 3.716651 0.915756 0
partition create curve 8 position 1.061858 0.070785 0
curve 8 scheme bias fine size 0.1 factor 1.1 start vertex 3
curve 17 interval 8
curve 17 scheme equal
curve 16 interval 9
curve 16 scheme equal
curve 6 interval 8
curve 6 scheme equal
curve 7 interval 9
curve 7 scheme bias factor 1.1 start vertex 3
surface 4 size auto factor 7
mesh surface 4
surface 6 size auto factor 7
mesh surface 6
create material 1
modify material 1 name 'mat_foun'
modify material 1 set property 'MODULUS' value 5e6
modify material 1 set property 'POISSON' value 0.3
create material 2
modify material 2 name 'mat_cyl'
modify material 2 set property 'MODULUS' value 500
modify material 2 set property 'POISSON' value 0.3
block 1 add surface 6
block 2 add surface 4
block all element plane order 2
block 1 material 'mat_foun'
block 2 material 'mat_cyl'
create displacement on surface 6 dof all fix
```

create displacement on curve 7 dof 1 fix

#25/2/17=0.705882353

create force on curve 6 force value 0.705882353 direction ny

create contact master curve 12 slave curve 8 tolerance 0.0005 type general friction 0.0 preload 0.0 offset 0.0 ignore_overlap off method mpc

analysis type static findefs elasticity dim2 planestrain

nonlinearopts maxiters 50 minloadsteps 1 maxloadsteps 100 tolerance 1e-3 targetiter 5
output nodalforce on energy off midresults on record3d on log on vtu on material off

F=50

reset

set default element hex

create surface circle radius 8 zplane

webcut body 1 with plane xplane

webcut body 1 with plane yplane

delete Body 3

delete Body 2

move Surface 4 y 8 include_merged

create surface rectangle width 20 height 5 zplane

move Surface 6 y -2.499 include_merged

partition create curve 8 position 3.716651 0.915756 0

partition create curve 8 position 1.061858 0.070785 0

curve 8 scheme bias fine size 0.1 factor 1.1 start vertex 3

curve 17 interval 8

curve 17 scheme equal

curve 16 interval 9

curve 16 scheme equal

curve 6 interval 8

curve 6 scheme equal

curve 7 interval 9

curve 7 scheme bias factor 1.1 start vertex 3

surface 4 size auto factor 7

mesh surface 4

surface 6 size auto factor 7

mesh surface 6

create material 1

modify material 1 name 'mat_foun'

modify material 1 set property 'MODULUS' value 5e6

modify material 1 set property 'POISSON' value 0.3

create material 2

modify material 2 name 'mat_cyl'

modify material 2 set property 'MODULUS' value 500

modify material 2 set property 'POISSON' value 0.3

block 1 add surface 6

block 2 add surface 4

block all element plane order 2

block 1 material 'mat_foun'

block 2 material 'mat_cyl'

create displacement on surface 6 dof all fix

create displacement on curve 7 dof 1 fix

#50/2/17=1.470589

create force on curve 6 force value 1.470589 direction ny

create contact master curve 12 slave curve 8 tolerance 0.0005 type general friction 0.0 preload 0.0 offset 0.0 ignore_overlap off method mpc

analysis type static findefs elasticity dim2 planestrain

nonlinearopts maxiters 50 minloadsteps 1 maxloadsteps 100 tolerance 1e-3 targetiter 5
output nodalforce on energy off midresults on record3d on log on vtu on material off

F=100

reset

set default element hex

create surface circle radius 8 zplane

webcut body 1 with plane xplane

webcut body 1 with plane yplane

delete Body 3

delete Body 2

move Surface 4 y 8 include_merged

create surface rectangle width 20 height 5 zplane

move Surface 6 y -2.499 include_merged

partition create curve 8 position 3.716651 0.915756 0

partition create curve 8 position 1.061858 0.070785 0

curve 8 scheme bias fine size 0.1 factor 1.1 start vertex 3

curve 17 interval 8

curve 17 scheme equal

```
curve 16 interval 9
curve 16 scheme equal
curve 6 interval 8
curve 6 scheme equal
curve 7 interval 9
curve 7 scheme bias factor 1.1 start vertex 3
surface 4 size auto factor 7
mesh surface 4
surface 6 size auto factor 7
mesh surface 6
create material 1
modify material 1 name 'mat_foun'
modify material 1 set property 'MODULUS' value 5e6
modify material 1 set property 'POISSON' value 0.3
create material 2
modify material 2 name 'mat_cyl'
modify material 2 set property 'MODULUS' value 500
modify material 2 set property 'POISSON' value 0.3
block 1 add surface 6
block 2 add surface 4
block all element plane order 2
block 1 material 'mat_foun'
block 2 material 'mat_cyl'
create displacement on surface 6 dof all fix
create displacement on curve 7 dof 1 fix
#100/2/17=2.9411765
create force on curve 6 force value 2.9411765 direction ny
create contact master curve 12 slave curve 8 tolerance 0.0005 type general friction 0.0 preload 0.0 offset 0.0 ignore_overlap off method mpc
analysis type static findefs elasticity dim2 planestrain
nonlinearopts maxiters 50 minloadsteps 1 maxloadsteps 100 tolerance 1e-3 targetiter 5
output nodalforce on energy off midresults on record3d on log on vtu on material off
```

Reference

1. NAFEMS R0081 - Benchmark Tests for Finite Element Modelling of Contact, Gapping and Sliding (задача CGS3).

2.14. Test case №2.14

Problem Description

We consider the problem of finding the eigenfrequencies of a beam, which is divided into three parts, between which the condition of general contact is valid. The test case is intended to check the correctness of the result of the calculation of the modal analysis, taking into account the general contact.

Input values

Geometrical model:

- Length $DD' = 10$ m;
- Width $AB = 2$ m;
- Height $AD = 2$ m.

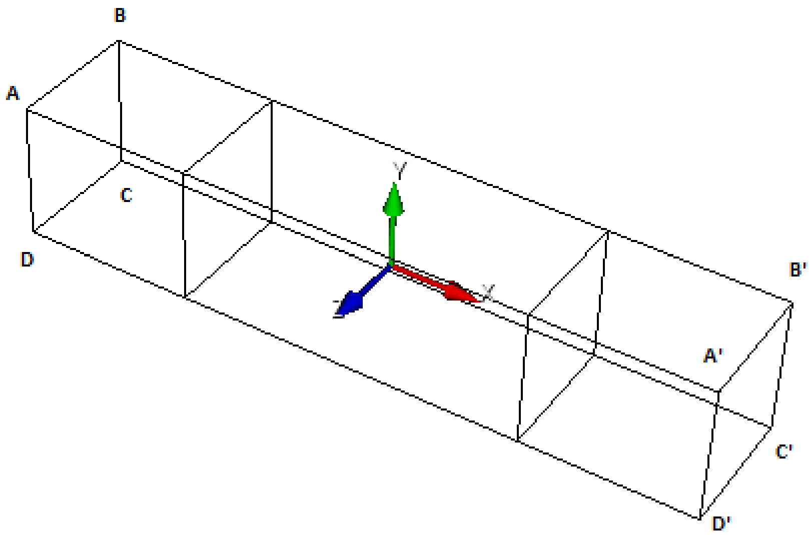

Fig. 28. Geometric model of a beam

Boundary conditions:

- Face BC is fixed to $u_x = u_z = 0$;
- Face B'C' is fixed to $u_z = 0$;
- Surface nodes DCD'C' are fixed to $u_y = 0$.

Material Properties:

- Elastic modulus $\mathrm{E} = 2\mathrm{e}11$ Pa;
- Poisson's ratio $\nu = 0.3$;
- Density $\rho = 8000$ kg/m³.

Mesh:

- Hexahedrons of the second order.

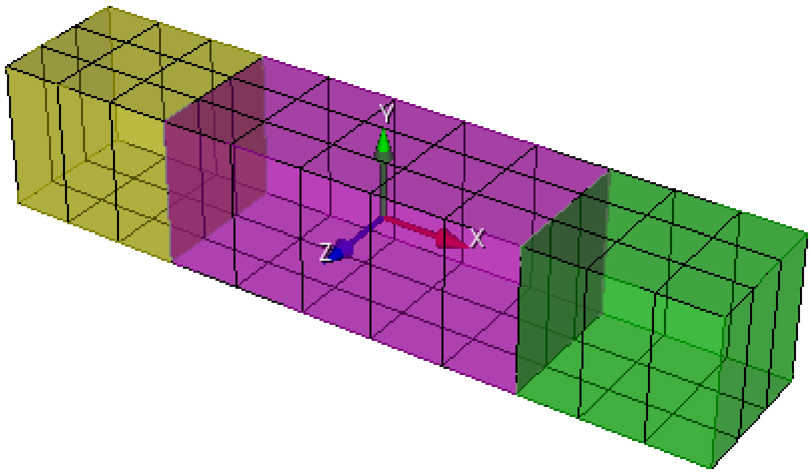

Fig. 29. Finite element mesh model

Contact:

- General;
- Method: mpc.

Calculation settings:

- Modal analysis;
- Search for the first lowest frequency.

Output Values

№	Value	Description	Target
1	Eigen Values	Eigen Values 1, Hz	38.254

Numerically approximate analytical solution

The solution from NAFEMS [1] acts as a reference.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Eigen Values	Eigen Values 1	Hz	38.254	3.677E+01	3.87

CAE Fidesys script:

```
reset
set default element hex
brick x 10 y 2 z 2
webcut volume 1 with plane xplane offset -2.5
webcut volume 1 with plane xplane offset 2.5
curve 28 41 36 26 43 35 25 44 33 28 27 42 34 size 1
curve 28 41 36 26 43 35 25 44 33 28 27 42 34 scheme equal
curve 3 15 37 7 13 39 1 5 23 21 29 31 size 2
curve 3 15 37 7 13 39 1 5 23 21 29 31 scheme equal
curve 11 16 40 12 9 14 38 10 22 24 32 30 size 0.67
curve 11 16 40 12 9 14 38 10 22 24 32 30 scheme equal
volume all scheme Auto
mesh volume all
create material 1
modify material 1 set property 'DENSITY' value 8000
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 2e11
set duplicate block elements off
block 1 volume all
block 1 material 1
create displacement on curve 7 dof 1 dof 3 fix 0
create displacement on curve 5 dof 3 fix 0
create displacement on node 56 59 60 53 55 63 64 57 58 62 61 54 33 80 79 38 74 92 91 84 83 89 90 75 76 88 87 82 81 85 86 77 2 7 8 6 14 30 29 25 26 31 32 13 12 28 27 24 dof 2 fix 0
block 1 element solid order 2
create contact master surface 17 slave surface 22 tolerance 0.0005 type general method auto
create contact master surface 7 slave surface 12 tolerance 0.0005 type general method auto
analysis type eigenfrequencies dim3
eigenvalue find 10 smallest
```

Reference

[1] NAFEMS Selected Benchmarks for Natural Frequency Analysis, Test 51.

2.15. Test case №2.15

Problem Description

We consider the problem of the stability of a compressed bar with the addition of a rigid contact condition. The test case checks the correctness of the calculation for the analysis of the buckling of the model, taking into account the contact interaction "general contact".

Input values

Geometrical model:

- Height h = 1 m;
- Radius R = 0.156 m;
- Thickness t = 0.006 m.

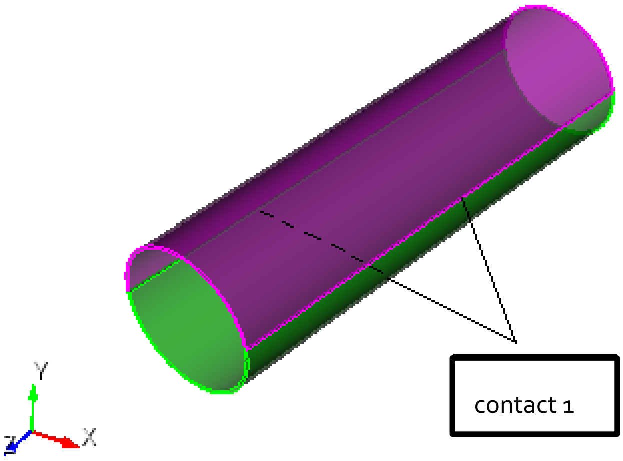

Fig 30 - Geometric model of the problem

Boundary conditions:

- Bottom circle is fixed in all directions;
- Pressure applied to the top circle p = 1 МПа;
- Contact pair - selection of main and secondary entity, Tied, Autoselect method.

Material Properties:

- Elastic modulus E = 200 GPa;
- Poisson's ratio $\nu = 0.3$.

Mesh:

- Hexahedral mesh.

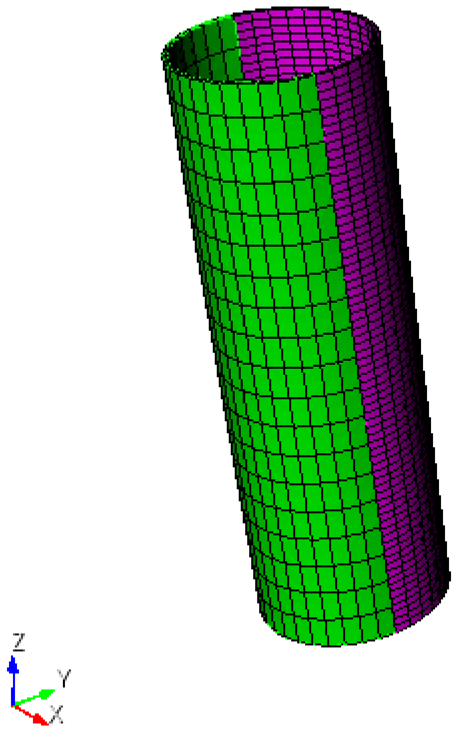

Fig 31 - Finite element mesh model

Calculation settings:

- Buckling analisys;
- 3D;
- Number of buckling forms: 1.

Output Values

№	Value	Description	Unit	Target
1	First coefficient of critical load	load multipliers(1)	-	44527

Numerically approximate analytical solution

The ANSYS solution acts as a reference.

Results

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	First coefficient of critical load	Critical Values 1	-	44527	4.458E+04	0.12

CAE Fidesys script:

```
reset
set default element hex
brick x 2.54 y 0.0508 z 0.0508
webcut volume 1 with plane yplane
webcut volume all with plane zplane
surface 19 26 33 31  scheme map
mesh surface 19 26 33 31
curve 2 4 6 8 interval 50
curve 2 4 6 8  scheme equal
mesh curve 2 4 6 8
volume all size auto factor 4
mesh volume all
create material 1
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 2.1e11
set duplicate block elements off
block 1 volume all
block 1 material 1
block 1 element solid order 2
create displacement on surface 23 35 29 21 dof all fix 0
create pressure on surface 19 26 33 31 magnitude 388
create contact autoselect tolerance 0.0005 type general method auto
analysis type stability elasticity dim3
eigenvalue find 1 smallest
```

3. Test casrs for cloud version

3.1. Test case №3.1

Problem Description

We consider the problem of static temperature loading of a hollow sphere. The model is in two parts, between the separator, there is a contact. The test case checks the correctness of the calculation under static temperature loading, taking into account the tied contact.

Input values

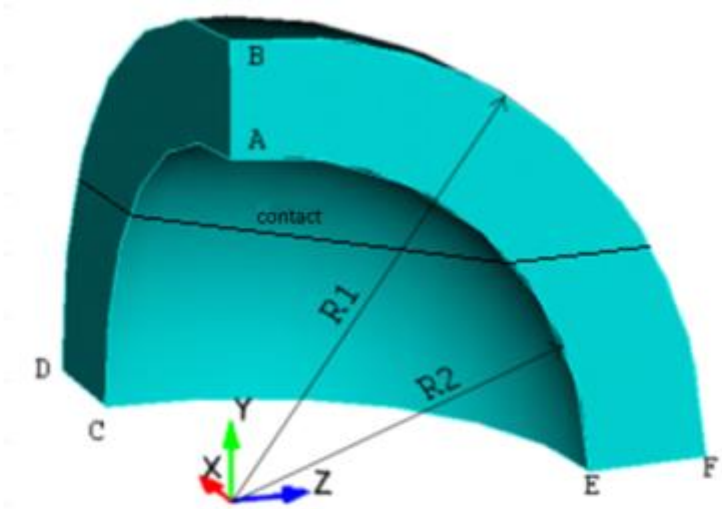

Fig.3.1. Geometric model for a hollow sphere

Geometrical model:

- Radius $R_1 = 4\ m$;
- Radius$R_2 = 3\ m$;
- In view of symmetry, we consider 1/8 of the sphere.

Boundary conditions:

- Zero displacements along the X-axis on the ABEF plane;
- Zero displacements along the Y-axis in the EFCD plane;
- Zero displacements along the Z axis in the ABCD plane;
- Solid temperature on the inner ACE surface of the sphere;
- Temperature $T = 30℃$.

Material Properties:

- Isotropic;
- Elastic modulus E = 200 GPa;
- Poisson's ratio ν = 0.3;
- Thermal expansion $\mu = 0.0001\ 1/℃$.

Mesh:

- Tetrahedrons of the second order.

Contact:

- Tied;
- Method: auto.

Calculation settings:

- Static calculation;
- Elasticity, thermal conductivity.

Output Values

№	Value	Description	Unit	Target
1	X-component of the displacement vector at the nodes of the grid at a point (4, 0, 0)	Displacement X	m	0.012

Analytical solution

The analytical solution is as follows [1]:

$$u_R = \mu T R_1.$$

Results in Prove.Design

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	X-component of the displacement vector at the nodes of the grid at a point (4, 0, 0)	Displacement X	m	0.012	0.012	0.00

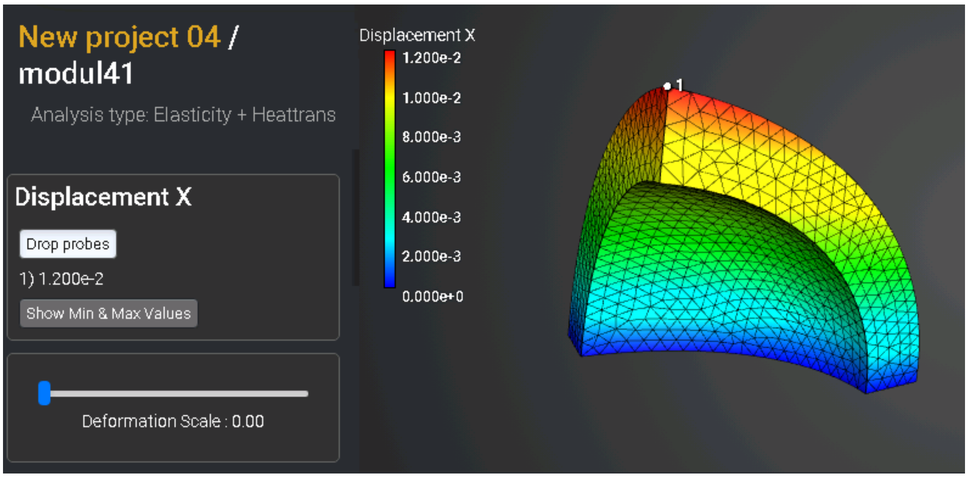

Fig. 3.2. Result of displacements X

CAE Fidesys script:

```
reset
create sphere radius 4
create sphere radius 3
subtract body 2 from body 1
webcut body 1 with plane yplane offset 0
webcut body 1 with plane zplane offset 0
webcut body 3 with plane zplane offset 0
webcut body 3 with plane xplane offset 0
delete Body 1
delete Body 6
delete Body 5
delete Body 4
webcut volume 4 with plane yplane offset 2
volume all scheme TetMesh
volume all size .3
mesh volume 4all
set duplicate block elements off
block 1 volume all
block 1 element solid order 2
create material 1
modify material 1 set property 'ISO_THERMAL_EXPANSION' value 1e-4
modify material 1 set property 'ISO_CONDUCTIVITY' value 1
modify material 1 set property 'POISSON' value .3
modify material 1 set property 'MODULUS' value 2e11
block 1 material 1
create displacement on surface 27 dof 2 dof 5 fix 0
create displacement on surface 37 44 dof 3 fix 0
create displacement on surface 39 43 dof 1 fix 0
create temperature on surface 40 42 value 30
create temperature on surface 38 45 value 30
create contact autoselect tolerance 0.0005 type tied method auto
analysis type static elasticity heattrans dim3
```

Reference

[1] Боли Б., Дж.Уэйнер. Теория температурных напряжений. М., Наука, 1974 г. –259 стр.

3.2. Test case №3.2

Problem Description

We consider the problem of static temperature loading of a solid sphere. The model is divided into two parts, between which the rigid contact condition acts. The test task is designed to check the correctness of the calculation under static temperature loading, taking into account the tied contact.

Input values

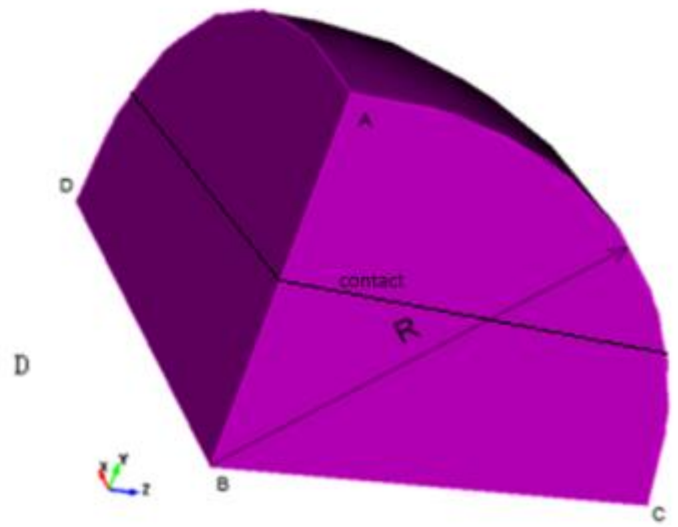

Fig.3.2.1. Geometric model for a hollow sphere

Geometrical model:

- Radius $R = 4\ m$;
- In view of symmetry, we consider 1/8 of the sphere.

Boundary conditions:

- Zero displacements along the X-axis on the ABC surface;
- Zero displacements along the Y-axis in the DBC surface;
- Zero displacements along the Z-axis in the ABD surface;
- Solid temperature on the inner ACD surface of the sphere;
- Temperature $T = 30$℃.

Material Properties:

- Isotropic;

- Elastic modulus E = 200 GPa;
- Poisson's ratio ν = 0.3;
- Thermal expansion $\mu = 0.0001\ 1/℃$.

Mesh:

- Tetrahedrons of the second order.

Contact:

- Tied;
- Method: auto.

Calculation settings:

- Static calculation;
- Elasticity, thermal conductivity.

Output Values

№	Value	Description	Unit	Target
1	X-component of the displacement vector at the nodes of the grid at a point (0, 4, 0)	Displacement X	m	0.012

Analytical solution

The analytical solution is as follows [1]:

$$u_R = \mu T R_1.$$

Results in Prove.Design

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	X-component of the displacement vector at the nodes of the grid at a point (0, 4, 0)	Displacement X	m	0.012	0.012	0.00

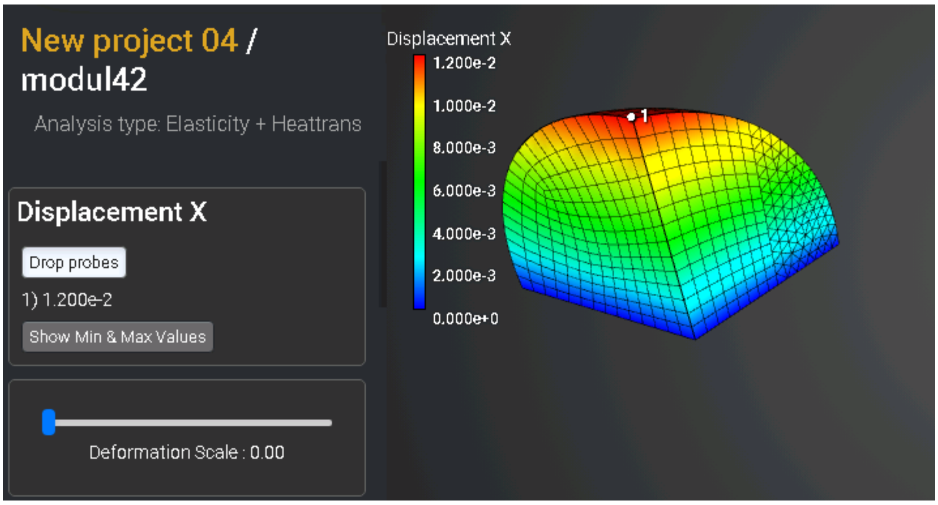

Fig. 3.2.2. Result of displacements X

CAE Fidesys script:

```
reset
create sphere radius 4
create sphere radius 3
subtract body 2 from body 1
webcut body 1 with plane yplane offset 0
webcut body 1 with plane zplane offset 0
webcut body 3 with plane zplane offset 0
webcut body 3 with plane xplane offset 0
delete Body 1
delete Body 6
delete Body 5
delete Body 4
webcut volume 4 with plane yplane offset 2
volume all scheme TetMesh
volume all size .3
mesh volume 4all
set duplicate block elements off
block 1 volume all
block 1 element solid order 2
create material 1
modify material 1 set property 'ISO_THERMAL_EXPANSION' value 1e-4
modify material 1 set property 'ISO_CONDUCTIVITY' value 1
modify material 1 set property 'POISSON' value .3
modify material 1 set property 'MODULUS' value 2e11
block 1 material 1
create displacement on surface 27 dof 2 dof 5 fix 0
create displacement on surface 37 44 dof 3 fix 0
create displacement on surface 39 43 dof 1 fix 0
create temperature on surface 40 42 value 30
create temperature on surface 38 45 value 30
create contact autoselect tolerance 0.0005 type tied method auto
analysis type static elasticity heattrans dim3
```

Reference

[1] Боли Б., Дж.Уэйнер. Теория температурных напряжений. М., Наука, 1974 г. –259 с.

3.3. Test case №3.3

Problem Description

We consider the problem a three-dimensional problem of a hollow cylinder under the influence of constant temperatures. The model is divided into two parts, between which the rigid contact condition acts. The test case is designed to check the correctness of the calculation under static temperature loading, taking into account the tied contact.

Input values

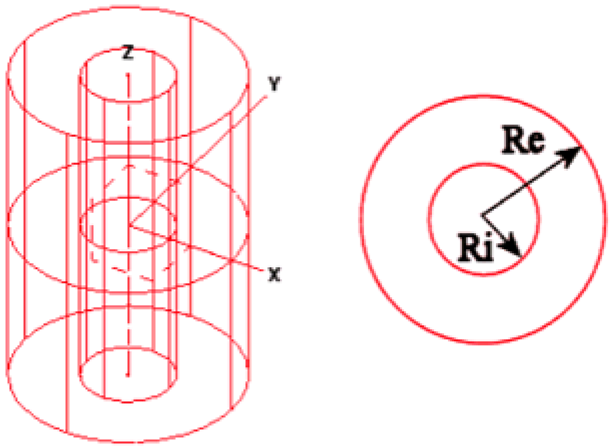

Fig. 3.3.1. Geometric model of a hollow cylinder

Geometrical model:

- Radius $R_i = 0.30\ m$;
- Radius $R_e = 0.35\ m$.

Boundary conditions:

- Internal temperature $T_i = 100\ °C$;
- External temperature $T_e = 20\ °C$;
- The ends of the cylinder are fixed along Z.

Material Properties:

- Isotropic;
- Thermal conductivity coefficient $V = 1\ W/(m \cdot °C)$.

Mesh:

- Tetrahedrons of the first order.

Contact:

- Tied;
- Method: auto.

Calculation settings:

- Static calculation;
- Thermal conductivity.

Output Values

№	Value	Description	Unit	Target
1	Temperature at a point (0.3,0,0)	Temperature	°C	100.0
2	Heat flux at a point (0.3,0,0)	Heat Flux	W/m2	1730
3	Temperature at a point (0.31,0,0)	Temperature	°C	82.98
4	Heat flux at a point (0.31,0,0)	Heat Flux	W/m2	1674
5	Temperature at a point (0.32,0,0)	Temperature	°C	66.51
6	Heat flux at a point (0.32,0,0)	Heat Flux	W/m2	1622
7	Temperature at a point (0.33,0,0)	Temperature	°C	50.54
8	Heat flux at a point (0.33,0,0)	Heat Flux	W/m2	1 573
9	Temperature at a point (0.34,0,0)	Temperature	°C	35.04
10	Heat flux at a point (0.34,0,0)	Heat Flux	W/m2	1 526
11	Temperature at a point (0.35,0,0)	Temperature	°C	20.00
12	Heat flux at a point (0.35,0,0)	Heat Flux	W/m2	1 483

Numerically approximate analytical solution

The solution from the Nastran Verification Manual [1] acts as a reference.

Results in Prove.Design

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Temperature at a point (0.3,0,0)	Temperature	°C	100.0	100	0.00
2	Heat flux at a point (0.3,0,0)	Heat Flux	W/m2	1730	1717.36	0.74
3	Temperature at a point (0.31,0,0)	Temperature	°C	82.98	82.92	0.07
4	Heat flux at a point (0.31,0,0)	Heat Flux	W/m2	1674	1676	-0.12
5	Temperature at a point (0.32,0,0)	Temperature	°C	66.51	66.51	0.00
6	Heat flux at a point (0.32,0,0)	Heat Flux	W/m2	1622	1622	0.00
7	Temperature at a point (0.33,0,0)	Temperature	°C	50.54	50.44	0.2
8	Heat flux at a point (0.33,0,0)	Heat Flux	W/m2	1 573	1574	-0.10
9	Temperature at a point (0.34,0,0)	Temperature	°C	35.04	35.06	-0.06
10	Heat flux at a point (0.34,0,0)	Heat Flux	W/m2	1 526	1523.94	0.13
11	Temperature at a point (0.35,0,0)	Temperature	°C	20.00	20	0.00
12	Heat flux at a point (0.35,0,0)	Heat Flux	W/m2	1 483	1492.7	-0.65

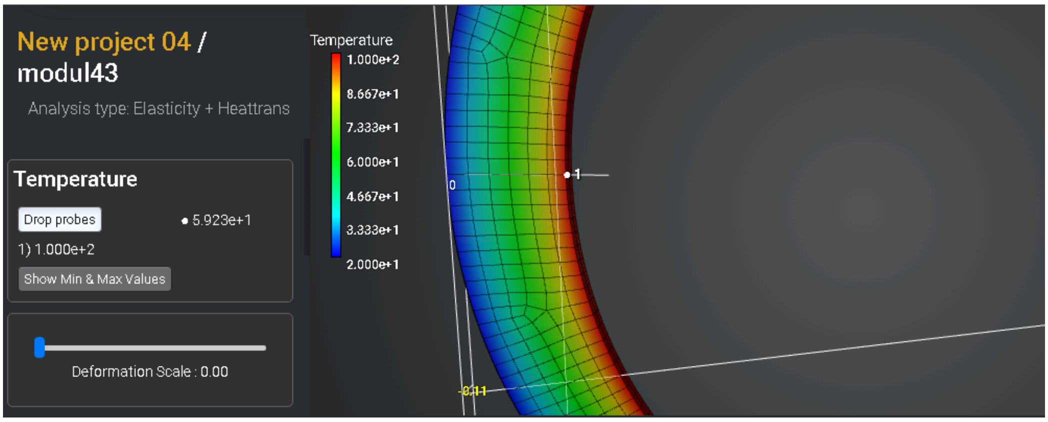

Fig. 3.3.2. Temperature at a point (0.3, 0, 0)

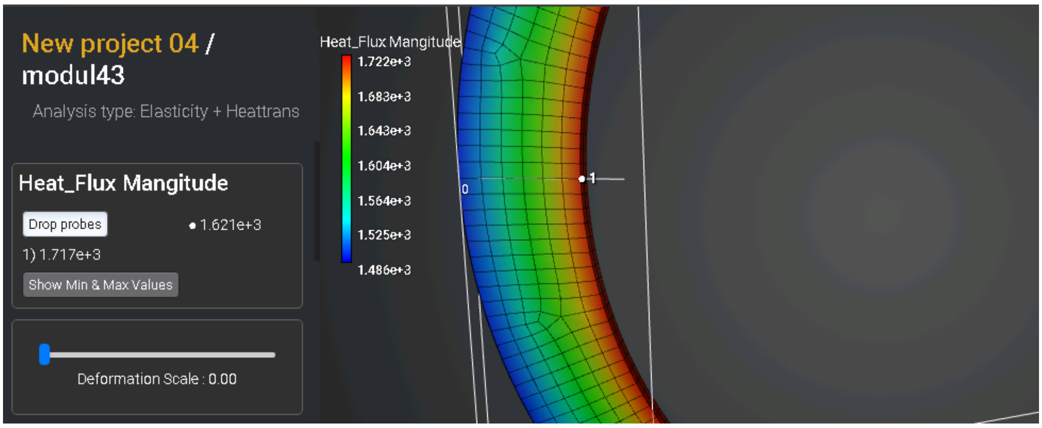

Fig. 3.3.3. Heat flux at a point (0.3, 0, 0)

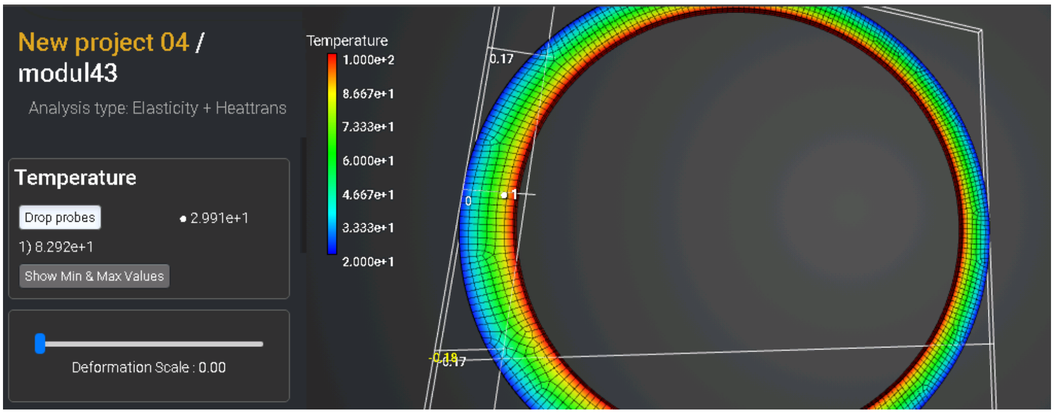

Fig. 3.3.4. Temperature at a point (0.31, 0, 0)

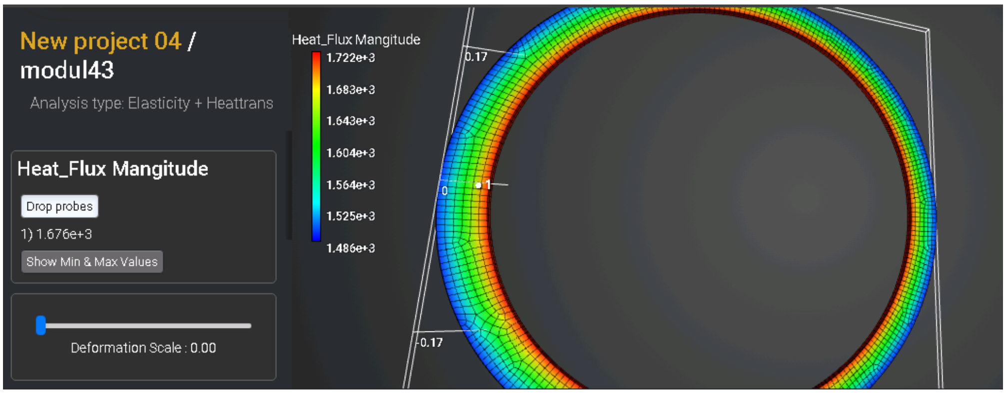

Fig. 3.3.5. Heat flux at a point (0.31, 0, 0)

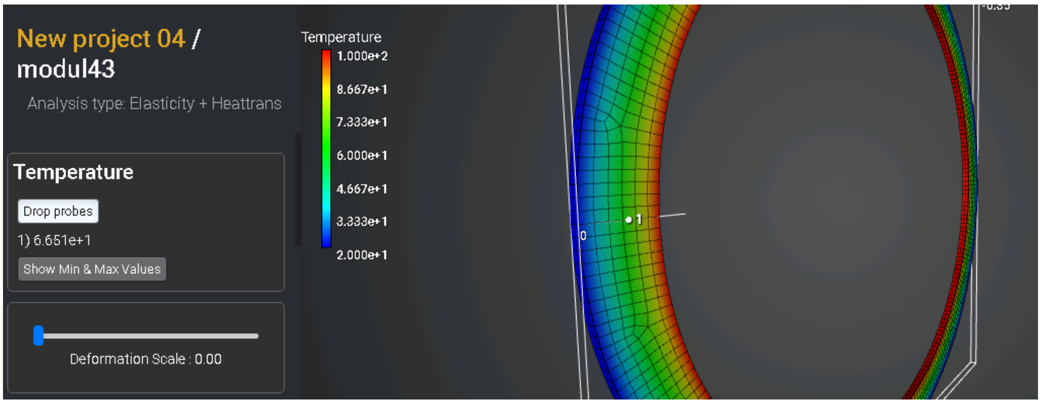

Fig. 3.3.6. Temperature at a point (0.32, 0, 0)

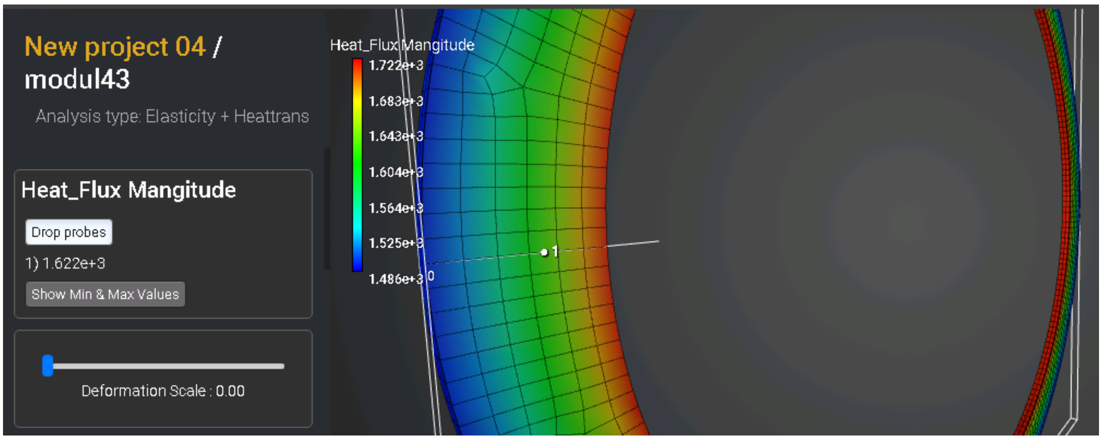

Fig. 3.3.7. Heat flux at a point (0.32, 0, 0)

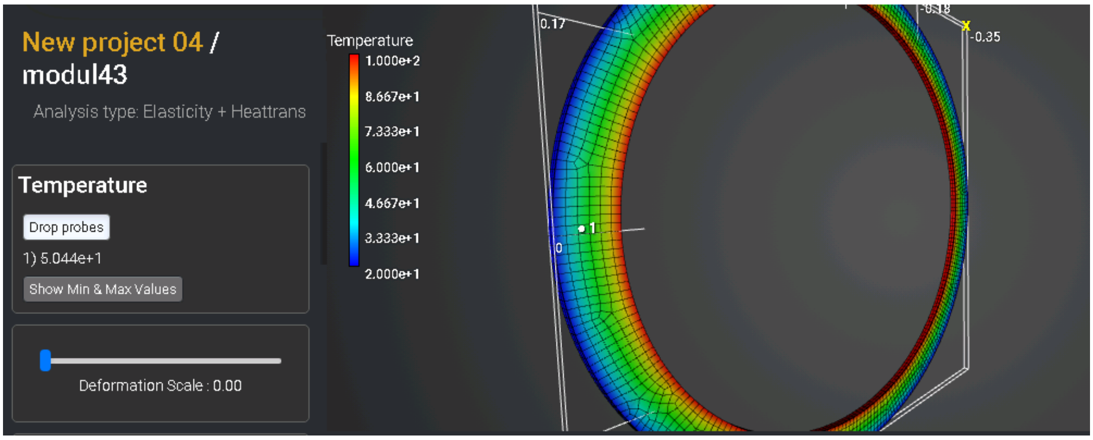

Fig. 3.3.8. Temperature at a point (0.33, 0, 0)

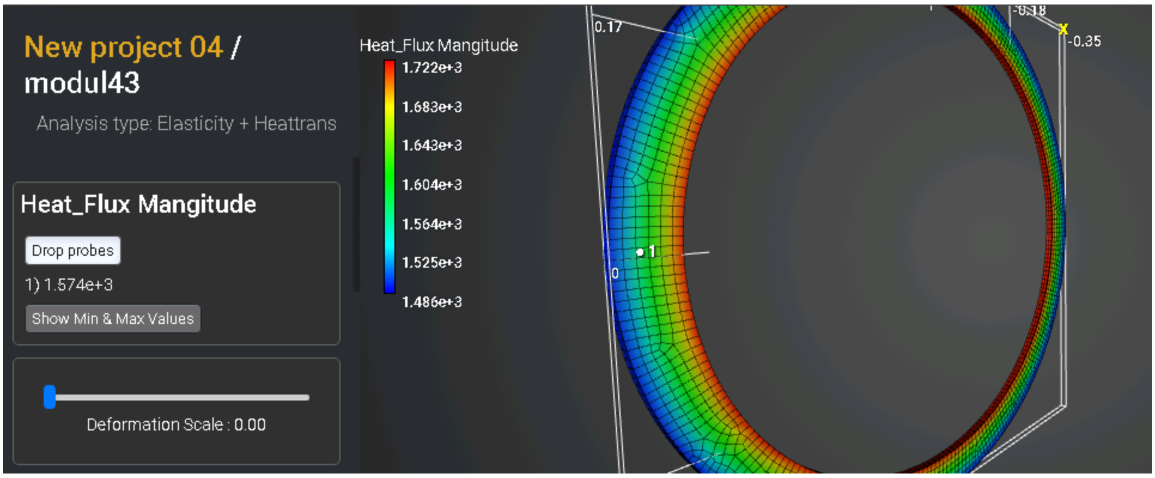

Fig. 3.3.9. Heat flux at a point (0.33, 0, 0)

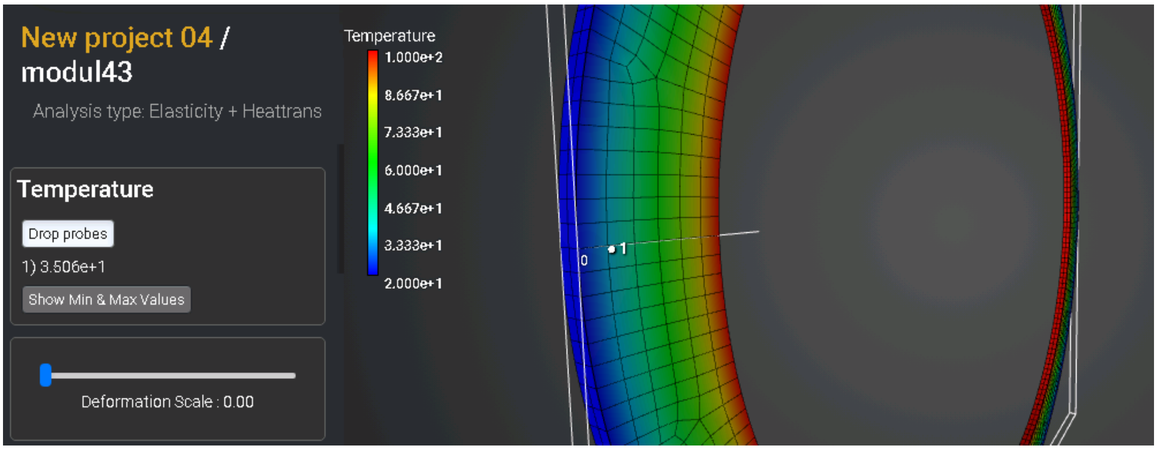

Fig. 3.3.9. Temperature at a point (0.34, 0, 0)

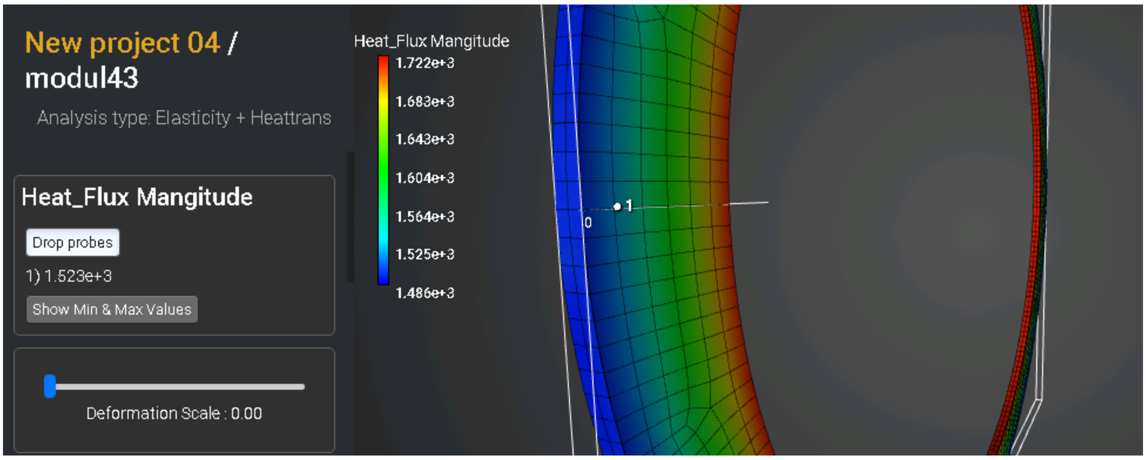

Fig. 3.3.10. Heat flux at a point (0.34, 0, 0)

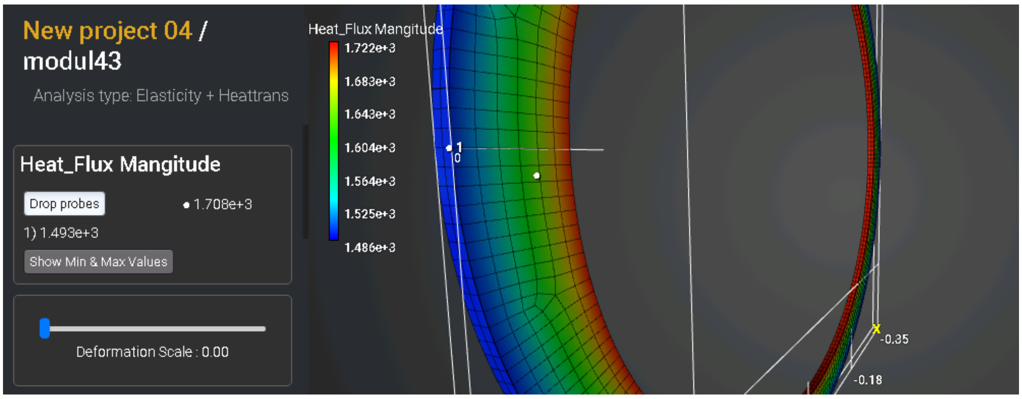

Fig. 3.3.11. Heat flux at a point (0.35, 0, 0)

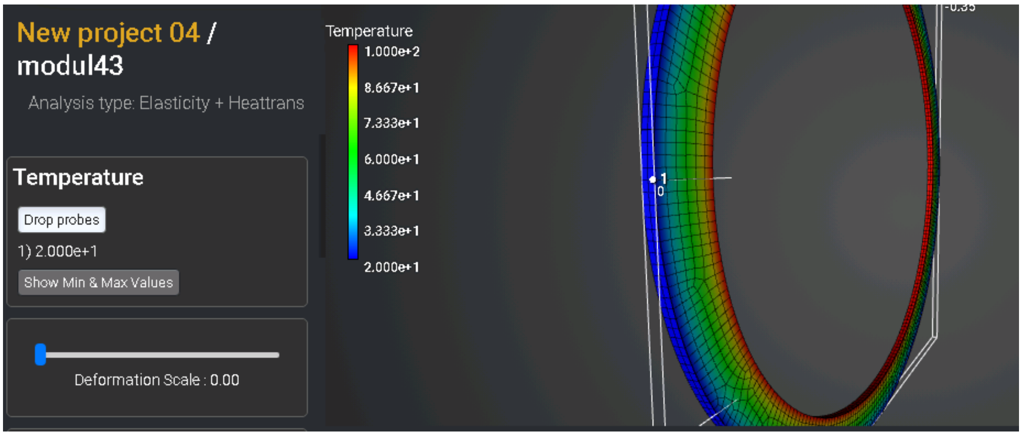

Fig. 3.3.12. Temperature at a point (0.35, 0, 0)

CAE Fidesys script:

```
reset
create Cylinder height 0.01 radius 0.3
create Cylinder height 0.01 radius 0.35
subtract body 1 from body 2  imprint
webcut volume 2  with plane zplane offset 0
curve 4 6 9 interval 400
curve 4 6 9 scheme equal
volume all scheme TetMesh
mesh volume all
create material 1
modify material 1 name 'Material 1'
modify material 1 set property 'ISO_CONDUCTIVITY' value 1
block 1 volume all
block 1  material 'Material 1'
block 1 element solid order 1
create temperature  on surface 14 11 value 100
create temperature  on surface 15 12 value 20
create displacement on surface 8 9 dof 3 fix 0
create contact autoselect tolerance 0.0005 type tied method auto
analysis type static heattrans dim3
```

Reference

[1] Societe Francaise des Mecaniciens. Guide de validation des progiciels de calcul de structures. Paris, Afnor Technique, 1990. Test No. TPLA01/89

3.4. Test case №3.4

Problem Description

We consider the problem a three-dimensional problem of a hollow cylinder under the influence of constant temperatures. The model is divided into two parts, between which the rigid contact condition acts. The test case is designed to check the correctness of the calculation under static temperature loading, taking into account the tied contact.

Input values

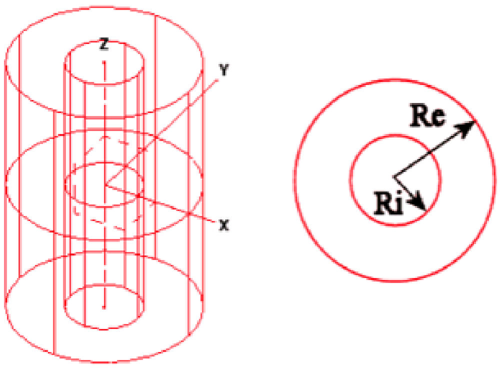

Fig. 3.4.1. Geometric model of a hollow cylinder

Geometrical model:

- Radius $R_i = 0.30\ m$;
- Radius $R_e = 0.391\ m$.

Boundary conditions:

- Convection on the internal surface $h_i = 150 \frac{Вт}{м^2 °C}$;
- Internal temperature $T_i = 500$ °C;
- Convection on the external surface $h_e = 142\ \frac{Вт}{м^2 °C}$;
- External temperature $T_e = 20$ °C;
- The ends of the cylinder are fixed along Z.

Material Properties:

- Isotropic;
- Thermal conductivity coefficient $V = 40\ W/(m \cdot °C)$.

Mesh:

- Tetrahedrons of the second order.

Contact:

- Tied;
- Method: auto.

Calculation settings:

- Static calculation;
- Thermal conductivity.

Output Values

№	Value	Description	Unit	Target
1	Temperature at a point (0.3,0,0)	Temperature	℃	272.3
2	Heat flux at a point (0.3,0,0)	Heat Flux	W/m2	3.416e4
3	Temperature at a point (0.391,0,0)	Temperature	℃	205.1
4	Heat flux at a point (0.391,0,0)	Heat Flux	W/m2	2.628e4

Results in Prove.Design

№	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Temperature at a point (0.3,0,0)	Temperature	℃	272.3	272.3	0.00
2	Heat flux at a point (0.3,0,0)	Heat Flux	W/m2	3.416e4	3.382 e4	0.1
3	Temperature at a point (0.391,0,0)	Temperature	℃	205.1	205.1	0.00
4	Heat flux at a point (0.391,0,0)	Heat Flux	W/m2	2.628e4	2.642e4	-0.53

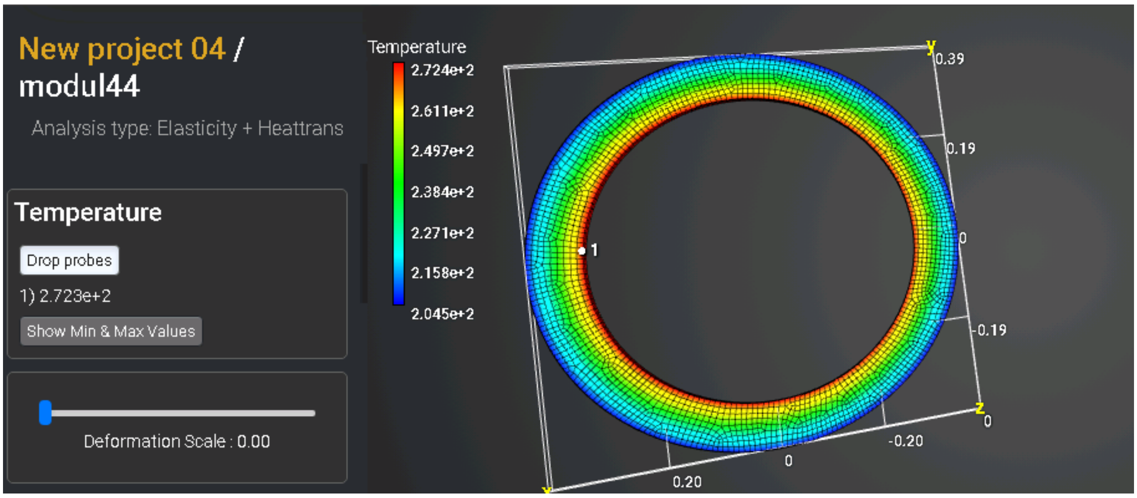

Fig. 3.4.2. Temperature at a point (0.3, 0, 0)

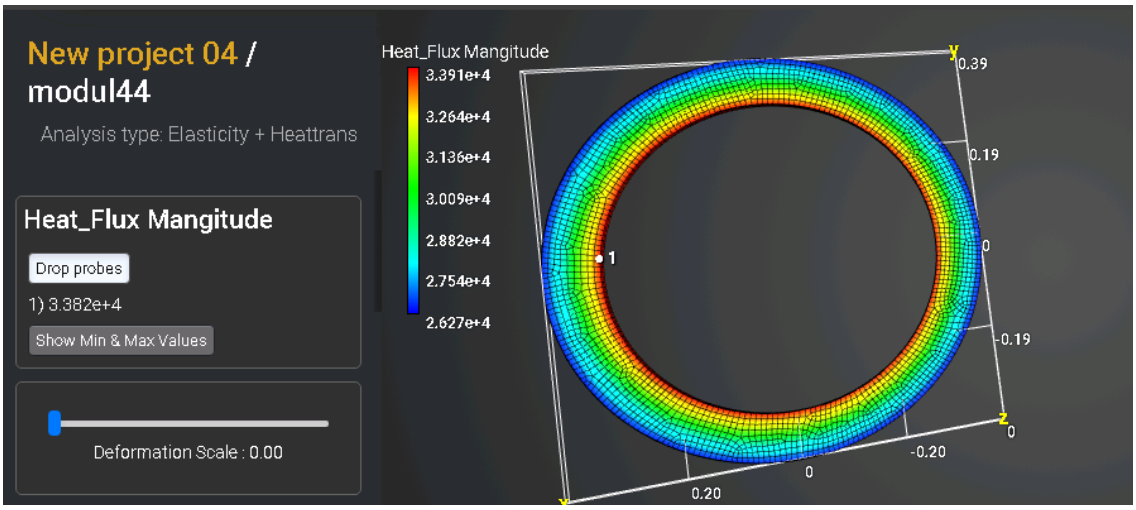

Fig. 3.4.3. Heat flux at a point (0.3, 0, 0)

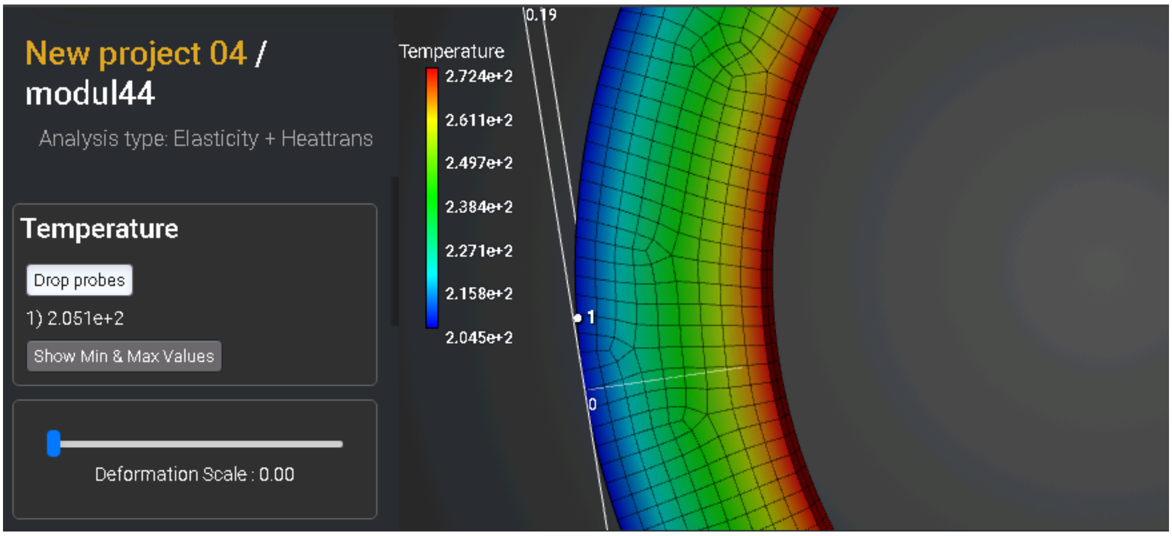

Fig. 3.4.4. Temperature at a point (0.391, 0, 0)

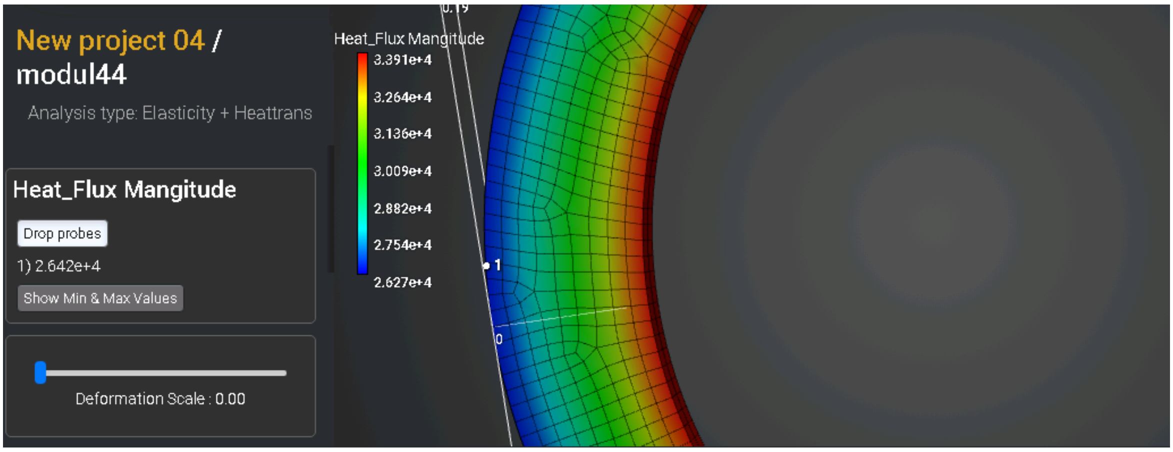

Fig. 3.4.5. Heat flux at a point (0.391, 0, 0)

CAE Fidesys script:

```
reset
create Cylinder height 0.01 radius 0.3
create Cylinder height 0.01 radius 0.391
subtract body 1 from body 2 imprint
webcut volume 2  with plane zplane offset 0
curve 4 6 9 interval 200
curve 4 6 9 scheme equal
volume all scheme TetMesh
mesh volume all
create material 1
modify material 1 set property 'ISO_CONDUCTIVITY' value 40
block 1 volume all
block 1 material 1
block 1 element solid order 2
create convection on surface 11 14 surrounding 500 coefficient 150
create convection on surface 12 15 surrounding 20 coefficient 142
create displacement on surface 8 9 dof 3 fix 0
create contact autoselect tolerance 0.0005 type tied method auto
analysis type static heattrans dim3
```

Reference

[1] Societe Francaise des Mecaniciens. Guide de validation des progiciels de calcul de structures. Paris, Afnor Technique, 1990. Test No. TPLA03/89

4. Contacts

http://www.cae-fidesys.com

support@cae-fidesys.com

+7 (495) 177-36-18