FIDESYS

strength analysis system

Version 5.1

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;
- Transient;
- Buckling;
- Mode Frequency;
- Spectrum;
- Effective Properties;
- Topological Optimization;
- External Integration MBD.

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 11	Ubuntu 16.04, Ubuntu 18.04, Ubuntu 20.04, Ubuntu 22.04
Windows Server 2022	Alt Linux 9.2
Windows 10	Debian 9, Debian 10, Debian 11
Windows Server 2019	RHEL 7, RHEL 8, RHEL 9
Windows Server 2016	Astra Linux Special Edition РУСБ.10015-01
Windows 8.1	Astra Linux 1.6, Astra Linux 1.7
Windows 8	RedOS
Windows Server 2012	Centos 7, Centos 8, Centos 9
Windows Server 2012 R2	Oracle Linux Server 9
Windows 7 SP1	OpenSUSE 15.3, OpenSUSE 15.4
Windows Server 2008 R2 SP1	Rocky Linux 8.5
	Scientific Linux 7
	Fedora 36

1. Test cases with analytical solutions

1.1. Test Case No.1.1

Problem Description

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

Input Values

Material Properties:

- Isotropic;
- Young's modulus = 1 Pa;
- Poisson ratio XY = 0.25;
- Density = 1 kg/m^3.

Geometric model:

- Solid cube with side 1m;
- Homogeneous material.

Boundary conditions:

- Non-periodic.

Mesh:

- Hexahedron (order 1, order 2), Tetrahedron (order 1, order 2);

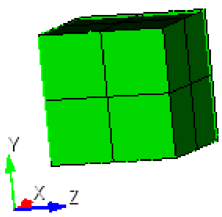


Fig 1.1 – Mesh 3D – Hexahedron

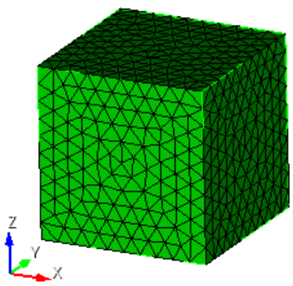


Fig 1.2 – Mesh 3D - Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	1.2
2	Effective elastic modulus	C_1122	Pa	0.4
3	Effective elastic modulus	C_1133	Pa	0.4
4	Effective elastic modulus	C_1212	Pa	0.4
5	Effective elastic modulus	C_1313	Pa	0.4
6	Effective elastic modulus	C_2222	Pa	1.2
7	Effective elastic modulus	C_2233	Pa	0.4
8	Effective elastic modulus	C_2323	Pa	0.4
9	Effective elastic modulus	C_3333	Pa	1.2

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e :

1. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

2. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

3. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

4. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

5. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

6. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$\alpha_{ij}^{e} = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^{e}}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11} q,\ \ \sigma_{ij} = \alpha_{ij}^{(1)} q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22} q,\ \ \sigma_{ij} = \alpha_{ij}^{(2)} q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q,\ \ \sigma_{ij} = \alpha_{ij}^{(3)} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q,\ \ \sigma_{ij} = \alpha_{ij}^{(4)} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q,\ \ \sigma_{ij} = \alpha_{ij}^{(5)} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32}) q,\ \ \sigma_{ij} = \alpha_{ij}^{(6)} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)};$

2) $C_{ij22} = \alpha_{ij}^{(2)};$

3) $C_{ij33} = \alpha_{ij}^{(3)};$

4) $C_{ij12} = C_{ij21} = \frac{1}{2} \alpha_{ij}^{(4)};$

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)}$;

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}$.

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

Hexahedron (order 1, order 2)

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	1.2	1.2	0
2	Effective elastic modulus	C_1122	Pa	0.4	0.4	0
3	Effective elastic modulus	C_1133	Pa	0.4	0.4	0
4	Effective elastic modulus	C_1212	Pa	0.4	0.4	0
5	Effective elastic modulus	C_1313	Pa	0.4	0.4	0
6	Effective elastic modulus	C_2222	Pa	1.2	1.2	0
7	Effective elastic modulus	C_2233	Pa	0.4	0.4	0
8	Effective elastic modulus	C_2323	Pa	0.4	0.4	0
9	Effective elastic modulus	C_3333	Pa	1.2	1.2	0

Script CAE Fidesys:

```
reset
brick x 1.0
volume 1 scheme Map
volume 1 size 0.5
mesh volume 1
```

```
create material 1
modify material 1 set property 'MODULUS' value 1.0
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'DENSITY' value 1.0
block 1 volume 1
block 1 material 1
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
periodicbc on
```

Tetrahedron mesh order 1, order 2

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	1.2	1.2	0
2	Effective elastic modulus	C_1122	Pa	0.4	0.4	0
3	Effective elastic modulus	C_1133	Pa	0.4	0.4	0
4	Effective elastic modulus	C_1212	Pa	0.4	0.4	0
5	Effective elastic modulus	C_1313	Pa	0.4	0.4	0
6	Effective elastic modulus	C_2222	Pa	1.2	1.2	0
7	Effective elastic modulus	C_2233	Pa	0.4	0.4	0
8	Effective elastic modulus	C_2323	Pa	0.4	0.4	0
9	Effective elastic modulus	C_3333	Pa	1.2	1.2	0

Script CAE Fidesys:

```
reset
brick x 1.0
volume 1 scheme Tetmesh
volume 1 size 0.1
mesh volume 1
create material 1
modify material 1 set property 'MODULUS' value 1.0
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'DENSITY' value 1.0
block 1 volume 1
block 1 material 1
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
periodicbc on
```

1.2. Test Case No.1.2

Problem Description

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

Input Values

Material Properties:

- Orthotropic
- Young’s modulus X = 12 Pa;
- Young’s modulus Y = 8 Pa;
- Young’s modulus Z = 4 Pa;
- Poisson ratio XY = 0.375;
- Poisson ratio XZ = 0.75;
- Poisson ratio YZ = 0.5;
- Density = 1 kg/m^3.
- Shear modulus XY =3 Pa;
- Shear modulus XZ = 2 Pa;
- Shear modulus YZ = 1 Pa;
- Thermal expansion coefficient X =1;
- Thermal expansion coefficient Y =1;
- Thermal expansion coefficient Z =1.

Geometric model:

- Solid cube with side 1m;
- Homogeneous material.

Boundary conditions:

- Non-periodic.

Mesh:

- Hexahedron (order 1, order 2), Tetrahedron mesh order 1, order 2;

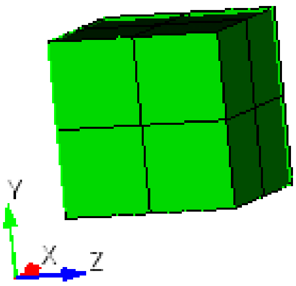


Fig 1.3 – Mesh 3D – Hexahedron

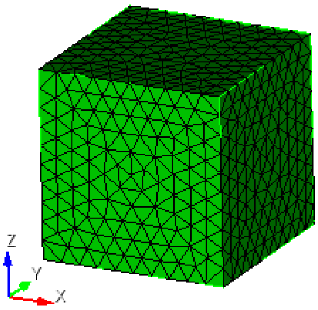


Fig 1.4 – Mesh 3D – Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	21
2	Effective elastic modulus	C_1122	Pa	9
3	Effective elastic modulus	C_1133	Pa	7.5
4	Effective elastic modulus	C_1212	Pa	3
5	Effective elastic modulus	C_1313	Pa	2
6	Effective elastic modulus	C_2222	Pa	13
7	Effective elastic modulus	C_2233	Pa	5.5
8	Effective elastic modulus	C_2323	Pa	1
9	Effective elastic modulus	C_3333	Pa	7.25
10	Effective density	Density	kg/m^3	1.0

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e :

7. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

8. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

9. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

10. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

11. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

12. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a_{ij}^e = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^e}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11} q, \ \sigma_{ij} = \alpha_{ij}^{(1)} q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22} q, \ \sigma_{ij} = \alpha_{ij}^{(2)} q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q, \ \sigma_{ij} = \alpha_{ij}^{(3)} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q, \ \sigma_{ij} = \alpha_{ij}^{(4)} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q, \ \sigma_{ij} = \alpha_{ij}^{(5)} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32}) q, \ \sigma_{ij} = \alpha_{ij}^{(6)} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)}$;

2) $C_{ij22} = \alpha_{ij}^{(2)}$;

3) $C_{ij33} = \alpha_{ij}^{(3)}$;

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)}$;

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)}$;

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}$.

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

Hexahedron (order 1, order 2)

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	21	21.0	0
2	Effective elastic modulus	C_1122	Pa	9	9.00	0
3	Effective elastic modulus	C_1133	Pa	7.5	7.5	0
4	Effective elastic modulus	C_1212	Pa	3	3.00	0
5	Effective elastic modulus	C_1313	Pa	2	2.0	0
6	Effective elastic modulus	C_2222	Pa	13	13.00	0
7	Effective elastic modulus	C_2233	Pa	5.5	5.50	0

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
8	Effective elastic modulus	C_2323	Pa	1	1	0
9	Effective elastic modulus	C_3333	Pa	7.25	7.250	0
10	Effective density	Density	kg/m^3	1.0	1.00	0

Script CAE Fidesys:

```
reset
brick x 1
volume 1 scheme Map
volume 1 size 0.5
mesh volume 1
create material 1
modify material 1 name 'Material1'
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
block 1 volume 1
block 1 material 'Material1'
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
periodicbc off
```

Tetrahedron (order 1, order 2)

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	21	21.0	0
2	Effective elastic modulus	C_1122	Pa	9	9.00	0
3	Effective elastic modulus	C_1133	Pa	7.5	7.5	0
4	Effective elastic modulus	C_1212	Pa	3	3.00	0
5	Effective elastic modulus	C_1313	Pa	2	2.0	0
6	Effective elastic modulus	C_2222	Pa	13	13.00	0
7	Effective elastic modulus	C_2233	Pa	5.5	5.50	0
8	Effective elastic modulus	C_2323	Pa	1	1	0

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
9	Effective elastic modulus	C_3333	Pa	7.25	7.250	0
10	Effective density	Density	kg/m^3	1.0	1.00	0

Script CAE Fidesys:

```
reset
brick x 1
volume 1 scheme Tetmesh
volume 1 size 0.1
mesh volume 1
create material 1
modify material 1 name 'Material1'
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
block 1 volume 1
block 1 material 'Material1'
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
periodicbc off
```

Reference:

Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 с.

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

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

Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

1.3. Test Case No.1.3

Problem Description

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

Input Values

Material Properties:

- Transversely-isotropic;
- Young's modulus T = 3 Pa;
- Young's modulus L = 4 Pa;
- Poisson ratio T = 0.25;
- Poisson ratio TL = 0.5;
- Density = 1 kg/m^3.
- Shear modulus TL = 1 Pa.

Geometric model:

- Solid cube with side 1m;
- Homogeneous material.

Boundary conditions:

- Non-periodic.

Mesh:

- Hexahedron (order 1, order 2), Tetrahedron (order 1, order 2);

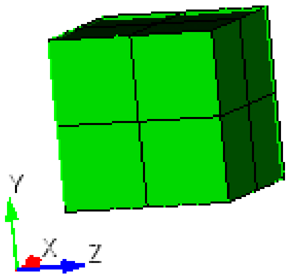


Fig 1.5 – Mesh 3D – Hexahedron

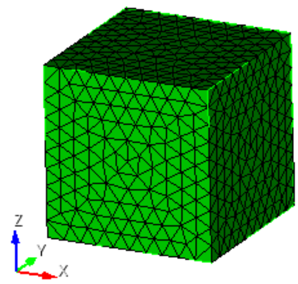


Fig 1.6 – Mesh 3D - Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	19.2
2	Effective elastic modulus	C_1122	Pa	16.8
3	Effective elastic modulus	C_1133	Pa	24
4	Effective elastic modulus	C_1212	Pa	1.2
5	Effective elastic modulus	C_1313	Pa	1
6	Effective elastic modulus	C_2222	Pa	19.2
7	Effective elastic modulus	C_2233	Pa	24
8	Effective elastic modulus	C_2323	Pa	1
9	Effective elastic modulus	C_3333	Pa	36
10	Effective Young's modulus	Ex	Pa	3.0
11	Effective Young's modulus	Ey	Pa	3.0
12	Effective Young's modulus	Ez	Pa	4.0
13	Effective Poisson ratio	ν_{yx}	-	0.25
14	Effective Poisson ratio	ν_{zx}	-	0.6667
15	Effective Poisson ratio	ν_{zy}	-	0.6667
16	Effective shear modulus	Gxy	Pa	1.2
17	Effective shear modulus	Gxz	Pa	1.0
18	Effective shear modulus	Gyz	Pa	1.0
19	Effective density	Density	kg/m^3	1.0

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e:

13. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

14. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

15. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

16. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

17. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

18. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a_{ij}^e = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^e}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11} q, \;\; \sigma_{ij} = \alpha_{ij}^{(1)} q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22} q, \;\; \sigma_{ij} = \alpha_{ij}^{(2)} q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q, \;\; \sigma_{ij} = \alpha_{ij}^{(3)} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q, \;\; \sigma_{ij} = \alpha_{ij}^{(4)} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q, \;\; \sigma_{ij} = \alpha_{ij}^{(5)} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32})q, \ \sigma_{ij} = \alpha_{ij}^{(6)} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)}$;

2) $C_{ij22} = \alpha_{ij}^{(2)}$;

3) $C_{ij33} = \alpha_{ij}^{(3)}$;

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)}$;

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)}$;

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}$.

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

Hexahedron (order 1, order 2)

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	19.2	19.20	0
2	Effective elastic modulus	C_1122	Pa	16.8	16.80	0
3	Effective elastic modulus	C_1133	Pa	24	24.00	0
4	Effective elastic modulus	C_1212	Pa	1.2	1.20	0

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
5	Effective elastic modulus	C_1313	Pa	1	1.00	0
6	Effective elastic modulus	C_2222	Pa	19.2	19.20	0
7	Effective elastic modulus	C_2233	Pa	24	24.00	0
8	Effective elastic modulus	C_2323	Pa	1	1.00	0
9	Effective elastic modulus	C_3333	Pa	36	36.00	0
10	Effective Young's modulus	Ex	Pa	3.0	3.00	0
11	Effective Young's modulus	Ey	Pa	3.0	3.00	0
12	Effective Young's modulus	Ez	Pa	4.0	4.00	0
13	Effective Poisson ratio	ν_{yx}	-	0.25	0.25	0
14	Effective Poisson ratio	ν_{zx}	-	0.6667	0.6666	<<0.01
15	Effective Poisson ratio	ν_{zy}	-	0.6667	0.66666	<<0.01
16	Effective shear modulus	Gxy	Pa	1.2	1.20	0
17	Effective shear modulus	Gxz	Pa	1.0	1.00	0
18	Effective shear modulus	Gyz	Pa	1.0	1.000	0
19	Effective density	Density	kg/m^3	1.0	1.00	0

Script CAE Fidesys:

```
reset
set default element hex
brick x 1
volume 1 size 0.5
mesh volume 1
block 1 volume 1
create material 1
modify material 1 set property 'TR_ISOT_E_T' value 3
modify material 1 set property 'TR_ISOT_E_L' value 4
modify material 1 set property 'TR_ISOT_G_TL' value 1
modify material 1 set property 'TR_ISOT_PR_T' value 0.25
modify material 1 set property 'TR_ISOT_PR_TL' value 0.5
modify material 1 set property 'DENSITY' value 1
block 1 material 1
block 1 element solid order 1  # update automatical from 1 to 2
```

analysis type effectiveprops elasticity dim3
periodicbc on

Tetrahedron (order 1, order 2)

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	19.2	19.20	0
2	Effective elastic modulus	C_1122	Pa	16.8	16.80	0
3	Effective elastic modulus	C_1133	Pa	24	24.00	0
4	Effective elastic modulus	C_1212	Pa	1.2	1.20	0
5	Effective elastic modulus	C_1313	Pa	1	1.00	0
6	Effective elastic modulus	C_2222	Pa	19.2	19.20	0
7	Effective elastic modulus	C_2233	Pa	24	24.00	0
8	Effective elastic modulus	C_2323	Pa	1	1.00	0
9	Effective elastic modulus	C_3333	Pa	36	36.00	0
110	Effective Young’s modulus	Ex	Pa	3.0	3.00	0
111	Effective Young’s modulus	Ey	Pa	3.0	3.00	0
112	Effective Young’s modulus	Ez	Pa	4.0	4.00	0
113	Effective Poisson ratio	ν_{yx}	-	0.25	0.25	0
114	Effective Poisson ratio	ν_{zx}	-	0.6667	0.6666	0.0049
115	Effective Poisson ratio	ν_{zy}	-	0.6667	0.66666	0.0049
116	Effective shear modulus	Gxy	Pa	1.2	1.20	0
117	Effective shear modulus	Gxz	Pa	1.0	1.00	0
118	Effective shear modulus	Gyz	Pa	1.0	1.000	0
119	Effective density	Density	kg/m^3	1.0	1.00	0

Script CAE Fidesys:

reset
brick x 1
volume 1 scheme Tetmesh
volume 1 size 0.1

```
mesh volume 1
block 1 volume 1
create material 1
modify material 1 set property 'TR_ISOT_E_T' value 3
modify material 1 set property 'TR_ISOT_E_L' value 4
modify material 1 set property 'TR_ISOT_G_TL' value 1
modify material 1 set property 'TR_ISOT_PR_T' value 0.25
modify material 1 set property 'TR_ISOT_PR_TL' value 0.5
modify material 1 set property 'DENSITY' value 1
block 1 material 1
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
```

1.4.Test Case No.1.4

Problem Description

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

Input Values

Material Properties:

- Murnaghan material;
- Lame modul = 2;
- Density = 1 kg/m^3.;
- Shear modulus = 1 Па;
- Coefficient C3 = -0.1;
- Coefficient C4 = -0.2;
- Coefficient C5 = -0.3.

Geometric model:

- Solid cube with side 1m;
- Homogeneous material.

Boundary conditions:

- Non-periodic.

Mesh:

Hexahedron mesh order 1, order 2;

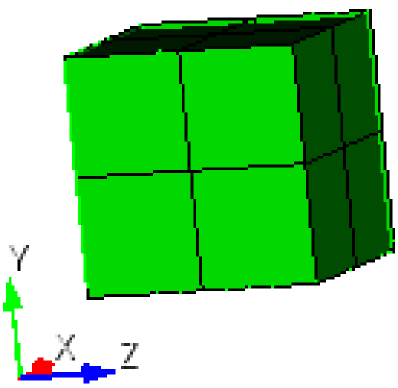


Fig 1.7 – Mesh 3D – Hexahedron

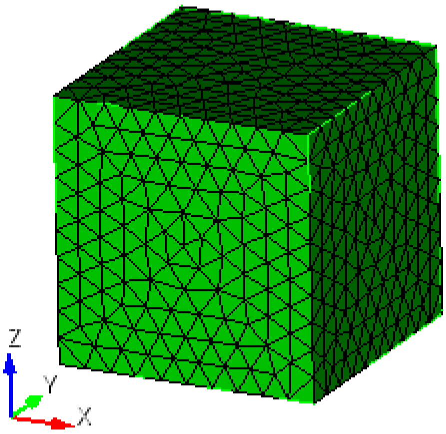


Fig 1.8 – Mesh 3D - Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	4
2	Effective elastic modulus	C_1122	Pa	2
3	Effective elastic modulus	C_1133	Pa	2
4	Effective elastic modulus	C_1212	Pa	1
5	Effective elastic modulus	C_1313	Pa	1
6	Effective elastic modulus	C_2222	Pa	4
7	Effective elastic modulus	C_2233	Pa	2
8	Effective elastic modulus	C_2323	Pa	1
9	Effective elastic modulus	C_3333	Pa	4
10	Young's modulus	E	Pa	2.6667
11	Poisson ratio	ν	-	0.3333
12	Density	Density	kg/m^3	1.0

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e :

19. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

20. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

21. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

22. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

23. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

24. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a^e_{ij} = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma^e_{ij}}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11} q, \ \sigma_{ij} = \alpha^{(1)}_{ij} q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22} q, \ \sigma_{ij} = \alpha^{(2)}_{ij} q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q, \ \sigma_{ij} = \alpha^{(3)}_{ij} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q, \ \sigma_{ij} = \alpha^{(4)}_{ij} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q, \ \sigma_{ij} = \alpha^{(5)}_{ij} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32}) q, \ \sigma_{ij} = \alpha^{(6)}_{ij} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha^{(1)}_{ij};$

2) $C_{ij22} = \alpha^{(2)}_{ij};$

3) $C_{ij33} = \alpha^{(3)}_{ij};$

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)};$

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)};$

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}.$

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

Hexahedron mesh order 1, order 2

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	4	4.00	0
2	Effective elastic modulus	C_1122	Pa	2	2.00	0
3	Effective elastic modulus	C_1133	Pa	2	2.00	0
4	Effective elastic modulus	C_1212	Pa	1	1.00	0
5	Effective elastic modulus	C_1313	Pa	1	1.00	0
6	Effective elastic modulus	C_2222	Pa	4	4.00	0
7	Effective elastic modulus	C_2233	Pa	2	2.00	0
8	Effective elastic modulus	C_2323	Pa	1	1.00	0
9	Effective elastic modulus	C_3333	Pa	4	4.00	0

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
10	Young’s modulus	E	Pa	2.6667	2.66666	<<0.01
11	Poisson ratio	ν	-	0.3333	0.33333	<<0.01
12	Density	Density	кг/м^3	1.0	1.0	0

Script CAE Fidesys:

```
reset
set default element hex
brick x 1
volume 1 size 0.5
mesh volume 1
block 1 volume 1
create material
modify material 1 set property 'MUR_LAME' value 2
modify material 1 set property 'MUR_SHEAR' value 1
modify material 1 set property 'MUR_C3' value -0.1
modify material 1 set property 'MUR_C4' value -0.2
modify material 1 set property 'MUR_C5' value -0.3
modify material 1 set property 'DENSITY' value 1
block 1 material 1
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
periodicbc on
```

Tetrahedron mesh order 1, order 2.

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	4	4.00	0
2	Effective elastic modulus	C_1122	Pa	2	2.00	0
3	Effective elastic modulus	C_1133	Pa	2	2.00	0
4	Effective elastic modulus	C_1212	Pa	1	1.00	0
5	Effective elastic modulus	C_1313	Pa	1	1.00	0
6	Effective elastic modulus	C_2222	Pa	4	4.00	0
7	Effective elastic modulus	C_2233	Pa	2	2.00	0
8	Effective elastic modulus	C_2323	Pa	1	1.00	0
9	Effective elastic modulus	C_3333	Pa	4	4.00	0
10	Young’s modulus	E	Pa	2.6667	2.66666	<<0.01
11	Poisson ratio	ν	-	0.3333	0.33333	<<0.01

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
12	Density	Density	kg/m^3	1.0	1.0	0

Script CAE Fidesys:

```
reset
brick x 1
volume 1 scheme Tetmesh
volume 1 size 0.1
mesh volume 1
block 1 volume 1
create material
modify material 1 set property 'MUR_LAME' value 2
modify material 1 set property 'MUR_SHEAR' value 1
modify material 1 set property 'MUR_C3' value -0.1
modify material 1 set property 'MUR_C4' value -0.2
modify material 1 set property 'MUR_C5' value -0.3
modify material 1 set property 'DENSITY' value 1
block 1 material 1
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
periodicbc on
```

1.5.Test Case No.1.5

Problem Description

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

Input Values

Material Properties:

Matrix

- Young's modulus E = 200 GPa;
- Poisson ratio ν = 0.3.
- Density = 1000 kg/m^3.

Thread:

- Young's modulus E = 2000 Pa;
- Poisson ratio =0.2;
- Density = 2000 kg/m^3

Geometric model:

- Rectangular parallelepiped 25 x 16 x 16;
- In the center along the X runs a thread with a length of 25 and a radius 2.85459861019.

Boundary conditions:

- Periodic.

Mesh:

- Hexahedron (order 1, order 2);

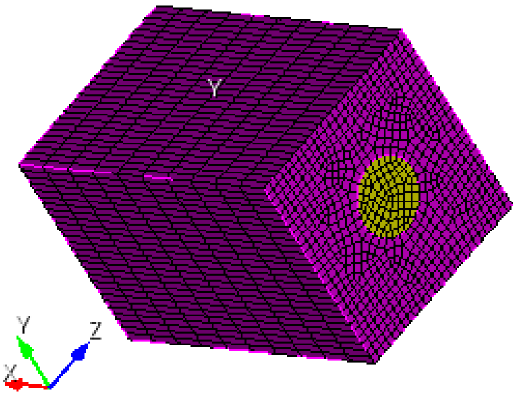


Fig 1.9 – Mesh 3D – Hexahedron

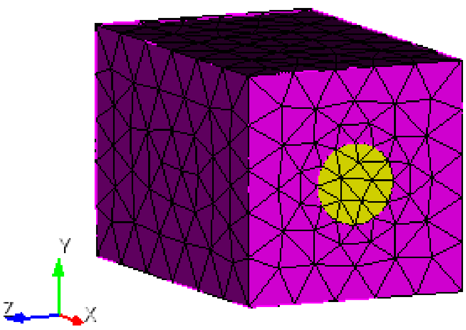


Fig 1.10 – Mesh 3D- Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	202.48
2	Effective elastic modulus	C_1122	Pa	1.22711
3	Effective elastic modulus	C_1133	Pa	1.22711
4	Effective elastic modulus	C_1212	Pa	0.938421
5	Effective elastic modulus	C_1313	Pa	0.938421
6	Effective elastic modulus	C_2222	Pa	3.11029
7	Effective elastic modulus	C_2233	Pa	1.33286
8	Effective elastic modulus	C_2323	Pa	0.888717
9	Effective elastic modulus	C_3333	Pa	3.11029
10	Effective Young's modulus	E1	Pa	201.803
11	Effective Young's modulus	E2	Pa	2.53669
12	Effective Young's modulus	E2	Pa	2.53669
13	Effective Poisson ratio	ν 12 =13	-	0.27618
14	Effective shear modulus	G12=G13	Pa	0.938421
15	Effective shear modulus	G23	Pa	0.888717
16	Density	Density	кг/м^3	1100

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e :

25. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

26. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

27. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

28. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

29. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

30. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a_{ij}^e = a_{ij}q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^e}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11}q, \ \sigma_{ij} = \alpha_{ij}^{(1)}q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22}q, \ \sigma_{ij} = \alpha_{ij}^{(2)}q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33}q, \ \sigma_{ij} = \alpha_{ij}^{(3)}q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21})q, \ \sigma_{ij} = \alpha_{ij}^{(4)}q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31})q, \ \sigma_{ij} = \alpha_{ij}^{(5)}q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32})q, \ \sigma_{ij} = \alpha_{ij}^{(6)}q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)}$;

2) $C_{ij22} = \alpha_{ij}^{(2)}$;

3) $C_{ij33} = \alpha_{ij}^{(3)}$;

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)}$;

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)}$;

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}$.

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

Hexahedron mesh order 1

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	202.48	200.91644	0.77
2	Effective elastic modulus	C_1122	Pa	1.22711	1.2725948	3.71
3	Effective elastic modulus	C_1133	Pa	1.22711	1.2725917	3.71
4	Effective elastic modulus	C_1212	Pa	0.938421	0.93935018	0.1

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
5	Effective elastic modulus	C_1313	Pa	0.938421	0.93929083	0.9
6	Effective elastic modulus	C_2222	Pa	3.11029	3.135307	0.8
7	Effective elastic modulus	C_2233	Pa	1.33286	1.3047198	2.11
8	Effective elastic modulus	C_2323	Pa	0.888717	0.89198678	0.37
9	Effective elastic modulus	C_3333	Pa	3.11029	3.1352915	0.8
10	Effective Young's modulus	E1	Pa	201.803	200.18694	0.8
11	Effective Young's modulus	E2	Pa	2.53669	2.5896062	2.09
12	Effective Young's modulus	E2	Pa	2.53669	2.5895934	2.09
13	Effective Poisson ratio	ν 12 =13	-	0.27618	0.28661851	3.78
14	Effective shear modulus	G12=G13	Pa	0.938421	0.93935018	0.10
15	Effective shear modulus	G23	Pa	0.888717	0.89198677	0.37
16	Density	Density	кг/м^3	1100	1099.2705	0.07

Hexahedron mesh order 2

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	202.48	202.37216	0.05
2	Effective elastic modulus	C_1122	Pa	1.22711	1.2731799	3.75
3	Effective elastic modulus	C_1133	Pa	1.22711	1.2731803	3.75
4	Effective elastic modulus	C_1212	Pa	0.938421	0.93983167	0.15
5	Effective elastic modulus	C_1313	Pa	0.938421	0.9398312	0.15
6	Effective elastic modulus	C_2222	Pa	3.11029	3.1359977	0.83
7	Effective elastic modulus	C_2233	Pa	1.33286	1.3069533	1.94

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
8	Effective elastic modulus	C_2323	Pa	0.888717	0.89142408	0.3
9	Effective elastic modulus	C_3333	Pa	3.11029	3.1359996	0.83
10	Effective Young's modulus	E1	Pa	201.803	201.64247	0.08
11	Effective Young's modulus	E2	Pa	2.53669	2.5885826	2.05
12	Effective Young's modulus	E2	Pa	2.53669	2.5885842	2.05
13	Effective Poisson ratio	ν 12 =13	-	0.27618	0.2865618	3.76
14	Effective shear modulus	G12=G13	Pa	0.938421	0.93983167	0.15
15	Effective shear modulus	G23	Pa	0.888717	0.89142408	0.3
16	Density	Density	кг/м3	1100	1099.9996	<<0.001

Script CAE Fidesys:

```
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 = 0.6}
# geometry
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
# meshing
volume all size {size}
curve 18 20 22 24 interval 10
mesh volume all
# materials
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 'DENSITY' value 2000
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 'DENSITY' value 1000
# blocks
block 1 volume 2
block 2 volume 3
block 1 material 'fiber'
```

```
block 2 material 'matrix'
block 1 element solid order 1  # update automatical from 1 to 2
analysis type effectiveprops elasticity dim3
periodicbc on
```

Tetrahedron mesh order 2.

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	202.48	202.35716	0.06
2	Effective elastic modulus	C_1122	Pa	1.22711	1.2758773	3.97
3	Effective elastic modulus	C_1133	Pa	1.22711	1.2727573	3.72
4	Effective elastic modulus	C_1212	Pa	0.938421	0.94320854	0.51
5	Effective elastic modulus	C_1313	Pa	0.938421	0.94085293	0.26
6	Effective elastic modulus	C_2222	Pa	3.11029	3.1516379	1.33
7	Effective elastic modulus	C_2233	Pa	1.33286	1.306944	1.94
8	Effective elastic modulus	C_2323	Pa	0.888717	0.90263071	1.57
9	Effective elastic modulus	C_3333	Pa	3.11029	3.1464708	1.16
10	Effective Young’s modulus	E1	Pa	201.803	201.6283	0.09
11	Effective Young’s modulus	E2	Pa	2.53669	2.6059074	2.73
12	Effective Young’s modulus	E2	Pa	2.53669	2.6016836	2.56
13	Effective Poisson ratio	ν 12 =13	-	0.27618	0.28642531	3.71
14	Effective shear modulus	G12=G13	Pa	0.938421	0.94317109	0.51
15	Effective shear modulus	G23	Pa	0.888717	0.90261837	1.56
16	Density	Density	кг/м^3	1100	1099.9915	<<0.001

Script CAE Fidesys:

```
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 = 4}
# geometry
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
# meshing
volume all scheme Tetmesh
volume all size {size}
mesh volume all
# materials
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 'DENSITY' value 2000
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 'DENSITY' value 1000
# blocks
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 elasticity dim3
periodicbc on
```

1.6. Test case No.1.6

Problem Description

The Lamb problem is considered, which is a dynamic action model of a concentrated load on the elastic half-plane boundary. Applied load depends on time according to Berlage's law.

Input Values

Geometric model:

- Length a=1000 m;
- Width b=500 m.

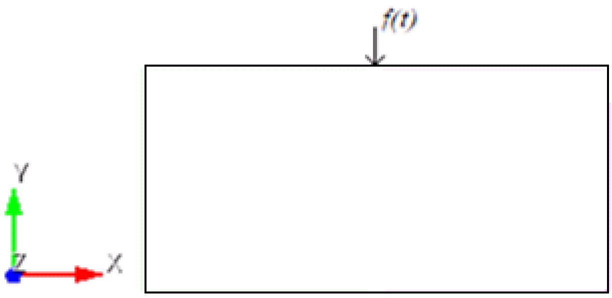


Fig. 1.11 - Geometric model of the Lamb problem

Border conditions:

- Point force is given using the Berlage formula:

$$f(t) = A\frac{\omega_1^2 e^{-\omega_1 t}}{4}\cdot\left(\sin(\omega_0 t)\left(-\frac{t^2}{\omega_1}+\frac{t}{\omega_1^2}+\frac{1}{\omega_1^3}\right)-\cos(\omega_0 t)\sqrt{3}\left(\frac{t^2}{\omega_1}+\frac{t}{\omega_1^2}\right)\right),$$

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

where A - amplitude, ω - frequency, t - time.

Non-reflective conditions applied to the bottom and side faces.

Material parameters:

- Young’s modulus $E = 2e+08$ Pa;
- Poisson ratio $\nu = 0.3$;
- Density $\rho = 1900\ kg/\mathrm{m}^3$;
- Cohesion $K = 29000$;
- Angle of internal friction $\alpha = 20$;

- Angle of dilatancy $\beta = 10$.

Mesh:

- Spectral elements of the 3rd order.

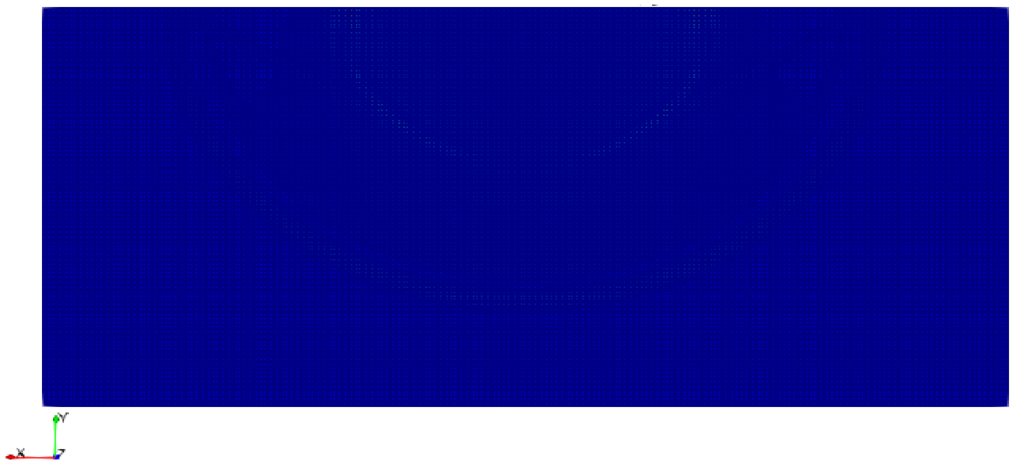


Fig. 1.12 - Spectral elements of the 3rd order for Lamb’s problem

The mesh should be of plane quadrangles, the height of the element is calculated according to the wavelength. The wave propagation speed is calculated by the formula [1]:

$$v = \sqrt{\frac{\lambda + 2G}{\rho}},$$

where h – geometry height, $t = \frac{h}{v}$ – time, $\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}$ – Lame module, $G = \frac{E}{2+2\nu}$ – shear modulus.

Calculation settings:

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

Calculation method used for the reference solution

Equations for the movement of the Rayleigh wave on the surface [1]

$$u_R \sim \frac{1}{2\pi}\int_{-\infty}^{\infty} -2iQe^{\frac{i\pi}{4}}\left[\frac{2}{c_R}\left(\frac{1}{c_R^2}-\frac{1}{\beta^2}\right)^{\frac{1}{2}}\right]\exp\left[i\omega\left(\frac{r}{c_R}-t\right)\right]\exp\left[-\omega\left(\frac{1}{c_R^2}-\frac{1}{\alpha^2}\right)^{\frac{1}{2}}h\right]\exp(i\omega t)d\omega,$$

$$w_R \sim \frac{1}{2\pi}\int_{-\infty}^{\infty} -2iQe^{\frac{i\pi}{4}}\left[\frac{2}{c_R^2}-\frac{1}{\beta^2}\right]\exp\left[i\omega\left(\frac{r}{c_R}-t\right)\right]\exp\left[-\omega\left(\frac{1}{c_R^2}-\frac{1}{\alpha^2}\right)^{\frac{1}{2}}h\right]\exp(i\omega t)d\omega.$$

where $$Q = A\left(\frac{2\pi\omega}{rc_R}\right)^{1/2} \frac{\omega}{\beta^2 R'(\frac{1}{c_R})}.$$ [1]

The physical parameters values of the propagation velocity of longitudinal and transverse waves, as well as the velocity of the Rayleigh wave, are found by the following formulas [1]

$$\alpha = \sqrt{\frac{K + \frac{4}{3}\mu}{\rho}},\ \beta = \sqrt{\frac{\mu}{\rho}},\ c = \frac{0.87 + 1.12\nu}{1+\nu}\beta,$$

where $\mu = \frac{E}{2(1+\nu)}$ - shear modulus, $K = \frac{E}{3(1-2\nu)}$ - compression module.

When changingν from 0 to 0.5, the phase velocity of the Rayleigh wave monotonically changes from 0.87 to 0.96 β.

Reference:

[1] Aki. K., Richards P. Quantitative seismology: Theory and methods. T. 1. Per. from English - M .: Mir, 1983 .- 520 p.

Result comparison

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

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Displacement vector components at mesh nodes at step 6	Displacement X	m	-0.00110025	-0.0011002537	<<0.01
2	Displacement vector components at mesh nodes at step 6	Displacement Y	m	0.000517095	0.00051707876	<<0.01
3	Displacement vector components at mesh nodes at step 8	Displacement X	m	-4.78016e-05	-4.7799808 e-05	<<0.01
4	Displacement vector components at mesh nodes at step 8	Displacement Y	m	0.000445372	0.00044537138	<<0.01

Script CAE Fidesys:

```
reset
set default element hex
create surface rectangle width 1000 zplane
webcut body 1 with plane xplane offset 0
webcut body 1 with plane yplane offset 0
delete Surface 3
rotate Surface 4 5 angle -90 about Z include_merged
```

```
webcut body 3 1 with plane yplane offset -250
surface all size 7
mesh surface all
imprint all
merge all
create material 1
modify material 1 name 'material'
modify material 1 set property 'MODULUS' value 2e+08
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 1900
modify material 1 set property 'COHESION' value 29000
modify material 1 set property 'INT_FRICTION_ANGLE' value 20
modify material 1 set property 'DILATANCY_ANGLE' value 10
set duplicate block elements off
block 1 add surface all
block 1 material 1
block 1 element plane order 3
create absorption on curve 28 24 13 15 19 21
create force on vertex 10 force value 1 direction 0 -1 0
bcdep force 1 value 'berlage(1e+8, 10, time)'
create receiver on curve 16 displacement 1 1 1
#create receiver on curve 16 velocity 1 1 1
#create receiver on curve 16 principalstress 1 1 1
#create receiver on curve 16 pressure
analysis type dynamic elasticity dim2 planestrain preload off
dynamic method full_solution scheme explicit maxtime 3 maxsteps 2025
output nodalforce off energy off record3d on log on vtu on material off results everystep 135
```

1.7. Test case No.1.7

Problem Description

In this problem, an infinite space is modeled, filled with a homogeneous isotropic elastic medium, in which a concentrated force acts, applied to a point and acting according to the Berlage law (Stokes problem [1]). It is considered that the source is point, that is, it is small in comparison with the distances to the receiver and just as small in comparison with the characteristic Units of space. The problem has an analytical solution.

Input Values

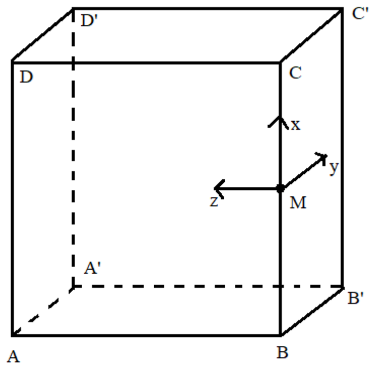


Fig. 1.13 - Geometric model Stokes problems

Geometric model:

- Cube 100×100×100 m;
- Geometry moved to coordinates (0, 50, 50), that M = (0, 0, 0).

Material parameters:

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

Border conditions:

- Symmetry condition: surface ABCD displacement $u_y = 0$;
- Symmetry condition: surface BB`C`C displacement $u_z = 0$;
- Symmetry Conditions: edge A`D` displacement $u_x = 0$;
- At the point M = (0, 0, 0), a force of 100 kN is applied, directed along the 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: since a quarter of the real model is considered in CAE Fidesys, the amplitude used to implement the analytical solution in the mathematical package should be divided by 4;

- Non-reflective HA 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).

Mesh:

- Element height h = 10 m;
- Spectral elements of the third order.

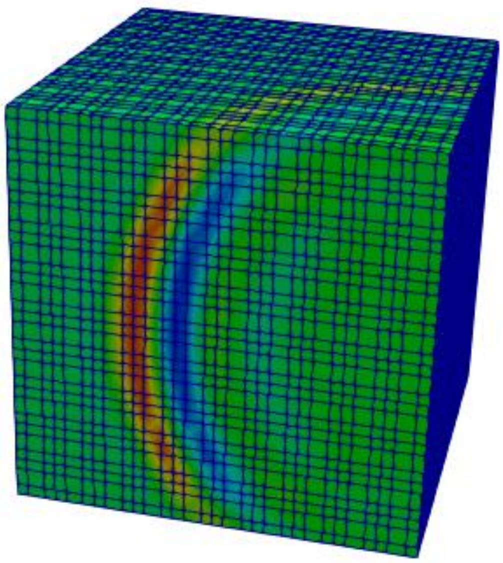


Fig. 1.14 - Spectral elements of the third order for Stokes problems

Element height was calculated using the formula:

$$h = \frac{L(n+1)}{10},$$

where $L = \frac{\upsilon}{\omega}$ - wavelength, υ - wave speed, ω - cyclic wave frequency, n - item order.

Calculation settings:

- Dynamic calculation;
- Maximum time – 0.4 s;
- Maximum number of steps 50 000;
- Output of every tenth step to .vtu file.

The maximum calculation time was chosen based on the analysis of the analytical solution in order to largely show the attenuation of the emerging waves.

Calculation method used for the reference solution

Let a concentrated force applied at a point $(x_0,\ y_0,\ z_0)$ and directed along a certain axis x_j act on an infinite space filled with a homogeneous isotropic elastic medium. 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, $\mu=\frac{E}{2(1+\nu)}$, $\lambda=\frac{\nu E}{(1+\nu)(1-2\nu)}$ – Lamé constants, ρ – Density of the environment in which waves propagate.

Kronecker symbol δ_{ij} is interpreted as follows:

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

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

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

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

here A – vibration amplitude, ω – cyclic oscillation frequency.

Having analyzed 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 Stokes problems in mathematical packages are $A,\ \omega,\ E,\ \nu,\ \rho$.

Reference:

[1] Aki K. Quantitative seismology / Richards P. - M .: Mir, t. 1, 1983. - 880 p.

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

Result comparison

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

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Component X of the displacement vector at the mesh nodes at timestep 0.136	Displacement X	m	5.384e-06	5.389e-06	1.13
2	Component Y of the displacement vector at the mesh nodes at timestep 0.144	Displacement Y	m	4.785e-06	4.799 e-06	0.19
3	Component Z of the displacement vector at the mesh nodes at timestep 0.144	Displacement Z	m	9.571e-06	9.424e-06	1.63
4	Component X of the displacement vector at the mesh nodes at timestep 0.2	Displacement X	m	1.842e-05	1.827e-05	0.79
5	Component Y of the displacement vector at the mesh nodes at timestep 0.2	Displacement Y	m	-7.33e-06	-7.339e-06	0.12
6	Component Z of the displacement vector at the mesh nodes at timestep 0.2	Displacement Z	m	-1.466e-05	-1.431e-05	2.37
7	Component X of the displacement vector at the mesh nodes at timestep 0.248	Displacement X	m	-1.024e-05	-1.006e-05	1/9

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
8	Component Y of the displacement vector at the mesh nodes at timestep 0.256	Displacement Y	m	3.258e-06	3.437e-06	2.09
9	Component Z of the displacement vector at the mesh nodes at timestep 0.256	Displacement Z	m	6.953e-06	6.899e-06	1.74

CAE Fidesys script:

```
reset
set default element hex
brick x 100 y 100 z 100
move Volume 1 location 0 50 50 include_merged
partition create curve 6  position 0 0 0
volume all size 10
mesh volume all
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 3
create displacement  on surface 3  dof 2 fix 0
create displacement  on surface 2  dof 3 fix 0
create displacement  on curve 2 dof 1 fix 0
create force  on vertex 9 force value 1 direction 1 0 0
bcdep force 1 value 'berlage(25e6, 10, time)'
create absorption on surface 1 5 6 4
create receiver on node 1566 137 1565 136 1564 135 1563 134 1562 123  displacement 1 1 1
create receiver on node 1566 137 1565 136 1564 135 1563 134 1562 123  velocity 1 1 1
create receiver on node 1566 137 1565 136 1564 135 1563 134 1562 123  principalstress 1 1 1
create receiver on node 1566 137 1565 136 1564 135 1563 134 1562 123  pressure
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
```

1.8. Test case No.1.8

Problem Description

Explosive pressure in a spherical cavity. The problem considers the behavior of an elastic infinite medium with a spherical cavity after applying pressure to the surface of the cavity. The solution was carried out for an explicit scheme.

Input Values

Geometric model:

- Presented at Fig 14;
- The considered area of the medium is limited by the volume of a sphere with a radius of 1.5 m;
- The cavity is located in the center of the sphere and has a radius of 0.5 m;
- Due to the symmetry of the problem, 1/8 of the original volume is considered.

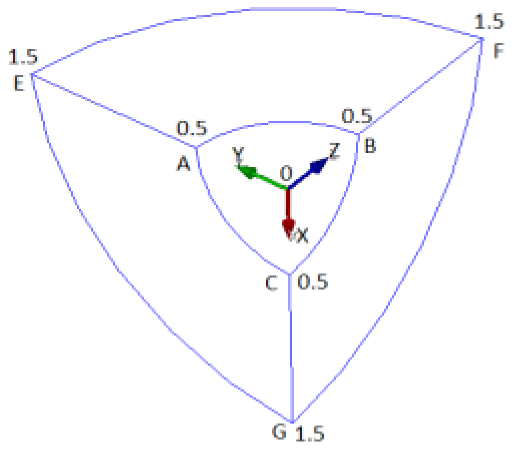


Fig. 1.15 - Spherical cavity

Border conditions:

- Symmetry condition: surface ABFE displacement $u_x = 0$;
- Symmetry condition: surface BCGF displacement $u_y = 0$;
- Symmetry Conditions: surface ACGE displacement $u_z = 0$;
- Pressure is applied to the surface of the spherical cavity ABC, which varies with time according to the formula

$$p(t) = 10^8 \sin(40000t).$$

Material parameters:

- Isotropic;
- Elastic modulusE = 200 GPa;
- Poisson ratio $\nu = 0.3$;

- Density ρ = 7900 kg/m^3.

Mesh:

- Spectral third-order hexahedra.

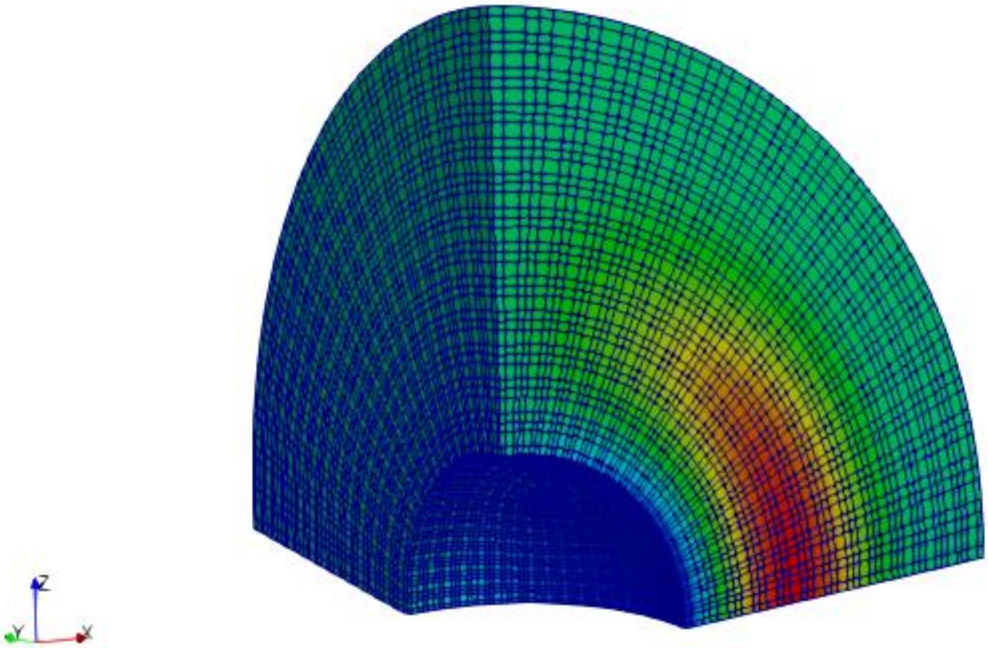


Fig. 1.16 - Spectral third order hexahedra for a spherical cavity

Calculation settings:

- Dynamic calculation;
- Maximum time – 1.35· 10−4 s;
- Maximum number of steps 50 000;
- Output of every tenth step to .vtu file.

Calculation method used for the reference solution

The displacement and stress values are calculated according to the following formulas [1]:

$$\tau = t - \frac{r-a}{c},$$

$$f(\tau) = \frac{a}{(\beta-\alpha)\rho}\int_0^\tau p(\xi)\left[e^{\alpha(\tau-\xi)} - e^{\beta(\tau-\xi)}\right]d\xi,$$

$$u_R = -\frac{-f\prime(\tau)}{c\cdot r} - \frac{f(\tau)}{r^2},$$

$$\sigma_R = \frac{\rho}{r}f''(\tau) + 2\frac{\rho c}{r^2}\frac{1-2\nu}{1-\nu}\left[f'(\tau) + \frac{c}{r}f(\tau)\right],$$

$$\sigma_\Theta = \frac{\rho}{r}\frac{\nu}{1-\nu}f''(\tau) - \frac{\rho c}{r^2}\frac{1-2\nu}{1-\nu}\left[f'(\tau) + \frac{c}{r}f(\tau)\right]$$

Reference:

[1] Timoshenko SP, Goodyer J. Theory of elasticity, transl. from English - M .: Nauka, 1975 - 576 p.

Result comparison

Below are the values for the components of the displacement vector and the stress tensor at the point (0.75, 0, 0) at the last moment in time.

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Displacement vector components at mesh points in spherical coordinates	Displacement X	m	4.08106e-05	4.089e-05	0.19
2	Stress tensor components at mesh nodes in spherical coordinates	Stress RR	MPa	48.75	48.04	1.45
3	Stress tensor components at mesh nodes in spherical coordinates	Stress TT	MPa	36.44	35.57	2.39

CAE Fidesys script:

```
reset
set default element hex
create sphere radius 1.5
webcut volume 1  with plane xplane offset 0
webcut volume 1 2 with plane yplane offset 0
webcut volume 1 2 3 4 with plane zplane offset 0
delete volume 1 2 4 5 6 7 8
create sphere radius 0.5
subtract volume 9  from volume 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
modify material 1 set property 'DENSITY' value 7900
volume all size auto factor 4
mesh volume all
set duplicate block elements off
block 1 add volume 3
block 1 material 1 cs 1 element solid order 3
create displacement  on surface 54  dof 1 fix 0
create displacement  on surface 52  dof 2 fix 0
create displacement  on surface 53  dof 3 fix 0
create pressure  on surface 51  magnitude 1
bcdep pressure 1 value "1e8*sin(40000*t)"
analysis type dynamic elasticity dim3 preload off
dynamic method full_solution scheme explicit maxtime 1.35e-04 maxsteps 50000
output nodalforce off energy off record3d on log on vtu on material off results everystep 10
```

1.9. Test case No.1.9

Problem Description

Checking the correctness of the solution based on the spectral element method for solving the Stokes problems, given in Section 1.6, on a non-conformal grid. The simulation results on a non-conformal mesh should match the results from Section 1.6.

Input Values

Geometric model:

- Cube 100×100×100 m;
- Geometry moved to coordinates(0, 50, 50), that M = (0, 0, 0).

Material parameters:

- Isotropic;
- Elastic modulus E = 2e8 Pa;
- Poisson ratio ν = 0,3;
- Density ρ = 1900 kg/m^3.

Border conditions:

- Symmetry condition: surface ABCD displacement $u_y = 0$;
- Symmetry condition: surface BB`C`C displacement $u_z = 0$;
- Symmetry Conditions: edge A`D` displacement $u_x = 0$;
- Dependence of force on time according to the Berlage formula with an amplitude of 25e6 m and a cyclic frequency of 10 Hz. Note: since a quarter of the real model is considered in CAE Fidesys, the amplitude used to implement the analytical solution in the mathematical package should be divided by 4 ;
- Dependence of force on time according to the Berlage formula with an amplitude of 25e6 m and a cyclic frequency of 10 Hz. Note: since a quarter of the real model is considered in CAE Fidesys, the amplitude used to implement the analytical solution in the mathematical package should be divided by 4;
- Non-reflective HA in planes AA`D`D, A`B`C`D`, DCC`D`, ABB`A`;
- Along the line of action of the force, receivers are assigned in all directions for each field from the drop-down list (displacement, speed, principal stresses, pressure).

Mesh:

- Element height first block h = 10 m;
- Spectral third order hexahedra for the first block;
- Element height second block h = 9 m;

- Spectral fourth order hexahedra for the first block.

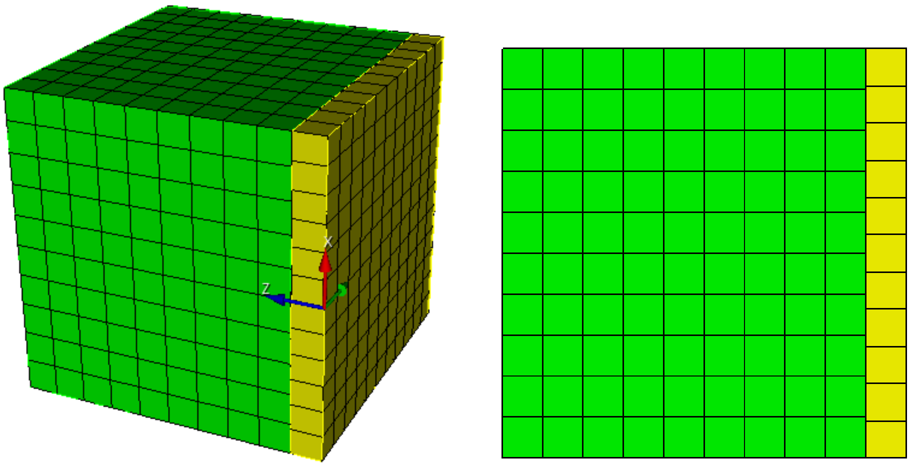


Fig. 1.17 - Non-conformal finite element mesh for Stokes problems

Element height was calculated using the formula

$$h = \frac{L(n+1)}{10},$$

where $L = \frac{\upsilon}{\omega}$ - wavelength, υ - wave speed, ω - cyclic wave frequency, n - item order.

Contact settings:

- Type: knitted;
- Accuracy: 0.11;
- Method: MPC.

Calculation settings:

- Dynamic calculation;
- Maximum time – 0.4 s;
- Maximum number of steps 50 000;
- Output of every tenth step to .vtu file.

Calculation method used for the reference solution

The analytical solution is presented in the section 1.7.

Result comparison

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

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Component X of the displacement vector at the mesh nodes at timestep 0.136	Displacement X	m	5.384e-06	5.333e-06	0.09
2	Component Y of the displacement vector at the mesh nodes at timestep 0.144	Displacement Y	m	4.785e-06	4.764 e-06	0.56
3	Component Z of the displacement vector at the mesh nodes at timestep 0.144	Displacement Z	m	9.571e-06	9.416e-06	1.72
4	Component X of the displacement vector at the mesh nodes at timestep 0.2	Displacement X	m	1.842e-05	1.845e-05	0.15
5	Component Y of the displacement vector at the mesh nodes at timestep 0.2	Displacement Y	m	-7.33e-06	-7.347e-06	0.24
6	Component Z of the displacement vector at the mesh nodes at timestep 0.2	Displacement Z	m	-1.466e-05	-1.4441e-05	1.7
7	Component X of the displacement vector at the mesh nodes at timestep 0.248	Displacement X	m	-1.024e-05	-1.001e-05	2.33
8	Component Y of the displacement vector at the mesh nodes at timestep 0.256	Displacement Y	m	3.258e-06	3.385e-06	3.59
9	Component Z of the displacement vector at the mesh nodes at timestep 0.256	Displacement Z	m	6.953e-06	6.887e-06	1.92

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 3 #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
create receiver on curve 6  displacement 1 1 1
create receiver on curve 6  velocity 1 1 1
create receiver on curve 6  principalstress 1 1 1
create receiver on curve 6  pressure
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
```

1.10. Test case No.1.10

Problem Description

Checking the correctness of the solution based on the spectral element method for solving the Lamb problem presented in Section 1.7 on a non-conformal mesh. The simulation results on a non-conformal mesh should match the results from section 1.7.

Input Values

Geometric model:

- Length $a = 1000$ m;
- Width $b = 500$ m;
- The model is divided into two layers of equal height.

Border conditions:

- The point force is specified using the Berlage formula:

$$f(t) = A\frac{\omega_1^2 e^{-\omega_1 t}}{4}\cdot\left(\sin(\omega_0 t)\left(-\frac{t^2}{\omega_1}+\frac{t}{\omega_1^2}+\frac{1}{\omega_1^3}\right)-\cos(\omega_0 t)\sqrt{3}\left(\frac{t^2}{\omega_1}+\frac{t}{\omega_1^2}\right)\right),$$

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

where A - amplitude (A=1e8), ω - frequency (ω=10), t - time.

- Non-reflective conditions applied to bottom and side faces.

Material parameters:

- Young's modulus $E = 2e+08$;
- Poisson ratio $\nu = 0.3$;
- Density $\rho = 1900$;
- Cohesion $K = 29000$;
- Angle of internal friction $\alpha = 20$;
- Angle of dilatancy $\beta = 10$.

Mesh:

- Spectral elements of the 3rd order;
- Mesh size for the top layer =7;

- Mesh size for the bottom layer=8.

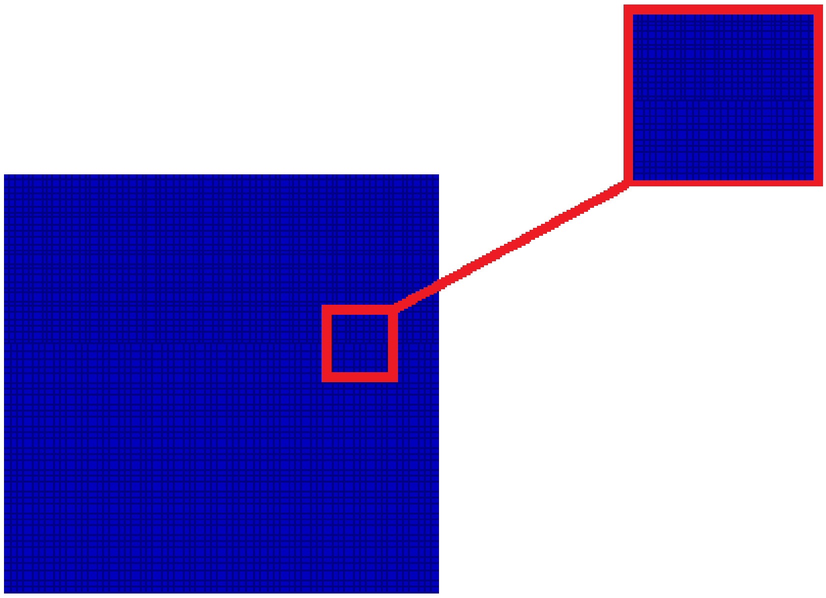


Fig. 1.18 - Non-conformal grid of spectral elements for the Lamb problem

The mesh should be of flat quadrangles, the height $h = 500$ of the element is calculated according to the wavelength (see paragraph 2.1).

Calculation settings:

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

Calculation method used for the reference solution

The analytical solution is given in the section 1.6.

Result comparison

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

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Displacement vector components at mesh nodes at timestep 0.416	Displacement X	m	-0.00110025	-0.001100	<<0.01
2	Displacement vector components at mesh nodes at timestep 0.416	Displacement Y	m	0.000517095	0.0005171	<<0.01

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
3	Displacement vector components at mesh nodes at timestep 0.555	Displacement X	m	-4.78016e-05	-4.780 e-05	0.01
4	Displacement vector components at mesh nodes at timestep 0.555	Displacement Y	m	-0.000445372	-0.000454	<<0.01

CAE Fidesys script:

```
reset
set default element hex
create surface rectangle width 1000 zplane
webcut body 1 with plane xplane offset 0
webcut body 1 with plane yplane offset 0
delete Surface 3
rotate Surface 4 5 angle -90 about Z include_merged
webcut body 3 1 with plane yplane offset -250
merge curve 18 25
merge curve 22 27
surface 9 7 size 7
mesh surface 9 7
surface 8 6 size 8
mesh surface 8 6
create material 1
modify material 1 name 'Material1'
modify material 1 set property 'MODULUS' value 2e+08
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 1900
modify material 1 set property 'COHESION' value 29000
modify material 1 set property 'INT_FRICTION_ANGLE' value 20
modify material 1 set property 'DILATANCY_ANGLE' value 10
create material 2
modify material 2 name 'Material2'
modify material 2 set property 'MODULUS' value 2e+08
modify material 2 set property 'POISSON' value 0.3
modify material 2 set property 'DENSITY' value 1900
modify material 2 set property 'COHESION' value 29000
modify material 2 set property 'INT_FRICTION_ANGLE' value 20
modify material 2 set property 'DILATANCY_ANGLE' value 10
set duplicate block elements off
block 1 add surface 9 7
set duplicate block elements off
block 2 add surface 8 6
block 1 material 1
block 2 material 2
block 1 2 element plane order 3
create absorption on curve 28 24 13 15 19 21
create force on vertex 10 force value 1 direction 0 -1 0
bcdep force 1 value 'berlage(1e+8, 10, time)'
create receiver on curve 16 displacement 1 1 1
create receiver on curve 16 velocity 1 1 1
create receiver on curve 16 principalstress 1 1 1
create receiver on curve 16 pressure
create contact master curve 17 23 slave curve 20 26 tolerance 0.0005 type tied method auto
analysis type dynamic elasticity dim2 planestrain preload off
dynamic method full_solution scheme explicit maxtime 5 maxsteps 2025
output nodalforce off energy off record3d on log on vtu on material off results everystep 135
```

1.11. Test case No.1.11

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 4 loading steps. In the problem, the correctness of the boundary pressure condition setting for stage-by-stage loading is checked.

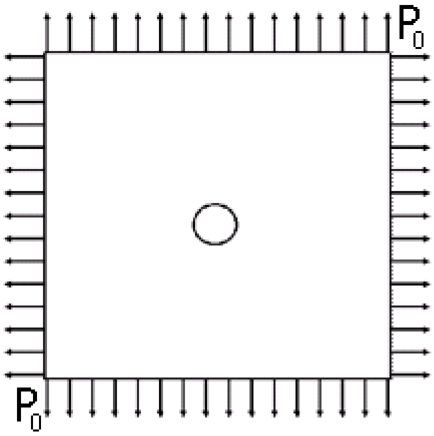


Fig. 1.19 - Geometric model for a plate with full tension

Input Values

Geometric model:

- Because of the problem symmetry, 1/4 of the plate is considered;
- BC = 5 m;
- Hole diameter 0.5 m;
- Polar coordinates are used.

Border conditions:

- Symmetry condition: curve AB displacement $u_x = 0$;
- Symmetry condition: curve ED displacement $u_y = 0$;
- P_0 = 0.25 MPa, 0.5 MPa, 0.75 MPa, 1 MPa.

Material parameters:

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

Mesh:

- 2D-quadrangular third-order spectral elements;

- 2D triangular spectral elements of the third order.

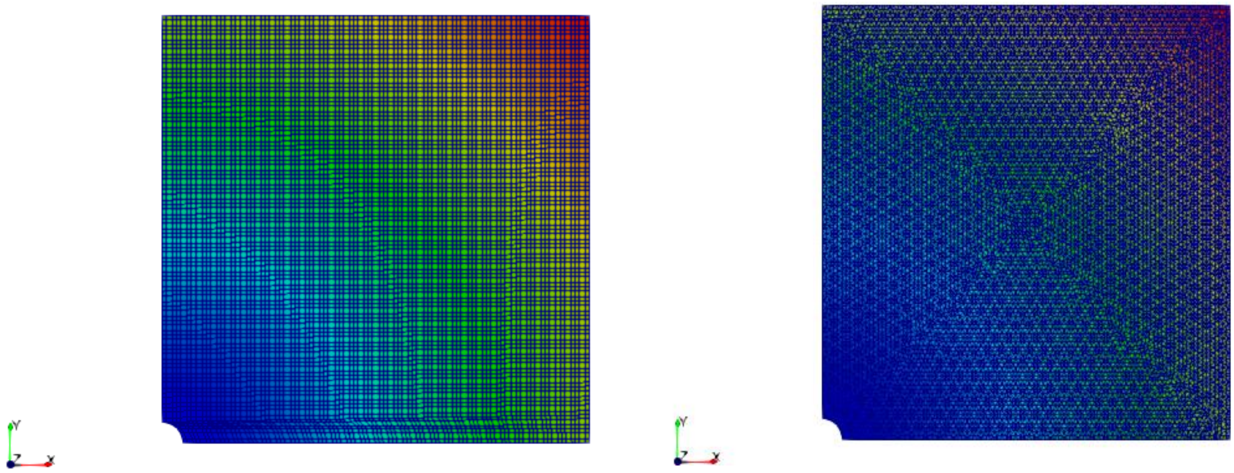


Fig. 1.20 - Spectral elements

Calculation method used for the reference solution

The values are calculated using the formula [1]:

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

Reference:

[1] Sedov L.I. “Continuum Mechanics, Volume 2”. M .: Science, 1970.

Result comparison

Below is the stress σ_θ at the boundary of the cut circle at the last loading step at the point (0.25,0,0).

Quadrangular third order spectral elements

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Stress tensor components at grid nodes in polar coordinates	Stress FF	MPa	2	2.0022149	0.11
2	Step number	step	-	4	4	-

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 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 1 2 value -250000
modify table 1 cell 2 2 value -500000
modify table 1 cell 3 2 value -750000
modify table 1 cell 4 2 value -1e+06
bcdep pressure 1 table 1
analysis type static elasticity dim2 planestrain
static steps 4
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5
static steps 4

Triangular third order spectral elements

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Stress tensor components at grid nodes in polar coordinates	Stress FF	MPa	2	1.9993749	0.03
2	Step number	step	-	4	4	-

CAE Fidesys script:
reset
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 trimesh geometry approximation angle 15
Trimesher surface gradation 1.3
Trimesher geometry sizing on
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 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 1 2 value -250000
modify table 1 cell 2 2 value -500000
modify table 1 cell 3 2 value -750000
modify table 1 cell 4 2 value -1e+06
bcdep pressure 1 table 1
analysis type static elasticity dim2 planestrain
static steps 4
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5
```

1.12. Test case No.1.12

Problem Description

The problem of an infinite cylindrical tube under the influence of internal and external pressures is considered. 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 2 loading steps. In the problem, the correctness of setting several boundary conditions for stage-by-stage loading is checked.

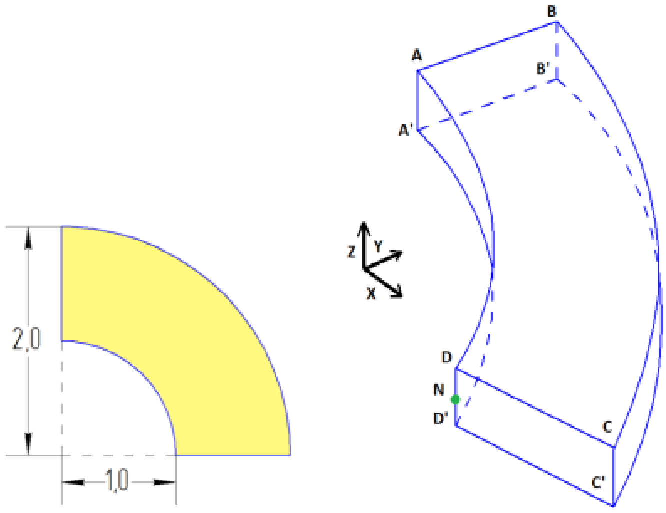


Fig. 1.21 - Geometric model of the problem of an infinite cylindrical pipe

Input Values

Geometric model:

- Due to the symmetry of the problem, a quarter of the wide section of the pipe is considered;
- Cut thickness 0.5 m;
- A cylindrical coordinate system is used.

Border conditions:

- Symmetry condition: surface ABB'A'displacement $u_x = 0$;
- Symmetry condition: surface CDD'C'displacement $u_y = 0$;
- Symmetry Conditions: surfaces ABCD and A’B’C’D’ displacement $u_z = 0$;
- A pressure p=0.5 MPa, 1 MPa is applied to the surface AA'D'D.
- A pressure p=0. 25 MPa, 0.5 MPa is applied to the surface B’B’C’C.

Material parameters:

- Isotropic;

- Elastic modulusE = 200 GPa;
- Poisson ratio ν = 0.3.

Mesh:

- Spectral third order hexahedra;
- Third-order spectral tetrahedra.

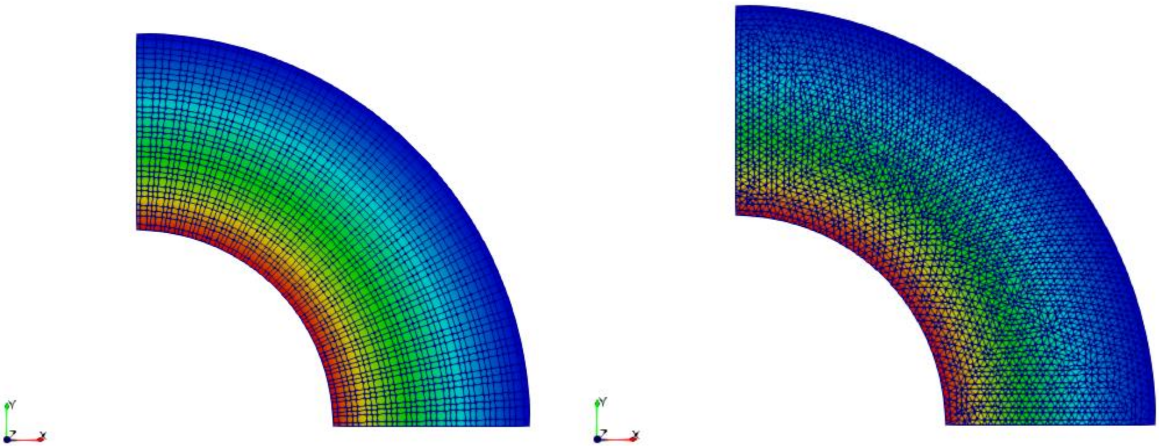


Fig. 1.22 - Spectral elements

Calculation method used for the reference solution

The values are calculated using the following formulas [1]:

$$\sigma_{rr} = \sigma_{11} = \frac{a^2 p_a}{b^2 - a^2}\left(1 - \frac{b^2}{r^2}\right) - \frac{b^2 p_b}{b^2 - a^2}\left(1 - \frac{a^2}{r^2}\right)$$

$$\sigma_{\theta\theta} = r^2 \sigma_{22} = \frac{a^2 p_a}{b^2 - a^2}\left(1 + \frac{b^2}{r^2}\right) - \frac{b^2 p_b}{b^2 - a^2}\left(1 + \frac{a^2}{r^2}\right) \qquad \sigma_{zz} = \sigma_{33} = \frac{\lambda}{\lambda + \mu}\frac{a^2 p_a - b^2 p_b}{b^2 - a^2}$$

Reference

[1] Sedov L.I. “Continuum Mechanics, Volume 2”. M .: Science, 1970., 568 стр.

Result comparison

Below are the stresses σrr, σθθ, σzz at the point N (1,0,0) at the last loading step.

Spectral third order hexahedra

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Stress tensor components at grid nodes in cylindrical coordinates	Stress RR	MPa	-1	-0.999807812	0.02

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
2	Stress tensor components at grid nodes in cylindrical coordinates	Stress FF	MPa	0.33	0.333412094	1.03
3	Stress tensor components at grid nodes in cylindrical coordinates	Stress ZZ	MPa	-0.2	-0.199918719	0.04
4	Step number	step	-	2	2	-

CAE Fidesys script:

```
reset
set default element hex
create Cylinder height 0.5 radius 2
create Cylinder height 0.5 radius 1
subtract volume 2 from volume 1
webcut volume 1 with plane xplane offset 0
webcut volume 1 3 with plane yplane offset 0
webcut volume 1 3 with plane yplane offset 0
delete volume 1 3 5
volume 4 size auto factor 5
volume 4 scheme auto
mesh volume 4
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 volume 4
block 1 material 1 cs 1 element solid order 3
create displacement on surface 11 dof 1 fix 0
create displacement on surface 27 dof 2 fix 0
create displacement on surface 31 29 dof 3 fix 0
create pressure on surface 30 magnitude 0
create pressure on surface 28 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 cell 1 1 value 1
modify table 1 cell 2 1 value 2
modify table 1 cell 1 2 value 500000
modify table 1 cell 2 2 value 1e+06
bcdep pressure 1 table 1
create table 2
modify table 2 dependency time
modify table 2 insert row 1
modify table 2 insert row 1
modify table 2 cell 1 1 value 1
modify table 2 cell 2 1 value 2
modify table 2 cell 1 2 value 250000
modify table 2 cell 2 2 value 500000
bcdep pressure 2 table 2
analysis type static elasticity dim3
```

static steps 2
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5

Third-order spectral tetrahedra

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Stress tensor components at grid nodes in cylindrical coordinates	Stress RR	MPa	-1	-0.99999975	<<0.01
2	Stress tensor components at grid nodes in cylindrical coordinates	Stress FF	MPa	0.33	0.3333	1.01
3	Stress tensor components at grid nodes in cylindrical coordinates	Stress ZZ	MPa	-0.2	-0.2	0.01
4	Step number	step	-	2	2	-

CAE Fidesys script:

```
reset
create Cylinder height 0.5 radius 2
create Cylinder height 0.5 radius 1
subtract volume 2 from volume 1
webcut volume 1 with plane xplane offset 0
webcut volume 1 3 with plane yplane offset 0
webcut volume 1 3 with plane yplane offset 0
delete volume 1 3 5
volume 4 size auto factor 5
volume 4  scheme tetmesh proximity layers off geometry approximation angle 15
volume 4  tetmesh growth_factor 1
Trimesher surface gradation 1.3
Trimesher volume gradation 1.3
Trimesher geometry sizing on
mesh volume 4
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 volume 4
block 1 material 1 cs 1 element solid order 3
create displacement on surface 11 dof 1 fix 0
create displacement on surface 27 dof 2 fix 0
create displacement on surface 31 29 dof 3 fix 0
create pressure on surface 30 magnitude 0
create pressure on surface 28 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 cell 1 1 value 1
modify table 1 cell 2 1 value 2
modify table 1 cell 1 2 value 500000
modify table 1 cell 2 2 value 1e+06
bcdep pressure 1 table 1
create table 2
modify table 2 dependency time
modify table 2 insert row 1
modify table 2 insert row 1
modify table 2 cell 1 1 value 1
modify table 2 cell 2 1 value 2
modify table 2 cell 1 2 value 250000
modify table 2 cell 2 2 value 500000
bcdep pressure 2 table 2
analysis type static elasticity dim3
static steps 2
```

nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5

1.13. Test case No.1.13

Problem Description

Uniform compression of a cube under the action of displacement is considered. 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 2 loading steps. In the task, the correctness of setting the displacement boundary condition for stage-by-stage loading is checked.

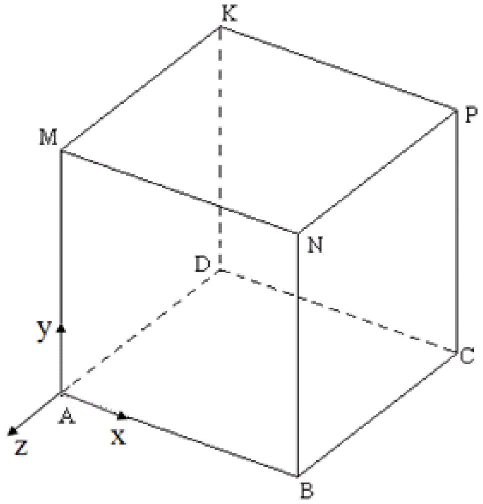


Fig. 1.23 - Geometric model of the problem of a cube uniform compression

Input Values

Geometric model:

- Cube with sides $0 \leq x \leq 10$ m, $0 \leq y \leq 10$ m, -10 m $\leq z \leq 0$.

Border conditions:

- Symmetry condition: surface AMKD displacement $u_x = 0$;
- Symmetry condition: surface ADCB displacement $u_y = 0$;
- Symmetry Conditions: surface DKPC displacement $u_z = 0$;
- Surface AMNB: $u_z = -0.5$ м, -1 м.

Material parameters:

- Isotropic;
- Elastic modulusE = 200 ГПа;
- Poisson ratio ν = 0.3.

Mesh:

- Spectral third order hexahedra;
- Third-order spectral tetrahedra.

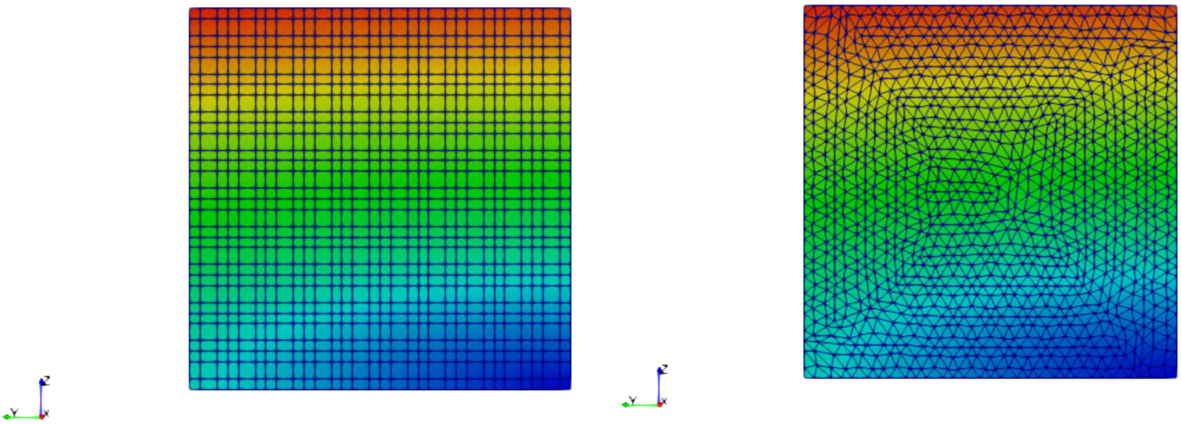


Fig. 1.24 - Spectral elements

Calculation settings:

- Statics;
- Elasticity;
- Number of loading steps: 2.

Calculation method used for the reference solution

The values are calculated using the following formulas [1]:

$$\varepsilon_{zz} = \frac{u_z}{L}; \varepsilon_{xx} = \varepsilon_{yy} = -\nu\frac{\sigma_{zz}}{E};$$

$$\sigma_{zz} = \varepsilon_{zz}E;\ \sigma_{xx} = \sigma_{yy} = 0;$$

$$u_z = -1\ \text{м};\ u_x = \frac{\varepsilon_{xx}}{L};\ u_y = \frac{\varepsilon_{yy}}{L}.$$

Where σ – stress tensor, ε – strain tensor, u – вектор перемещений, E – Young’s modulus, ν - Poisson ratio, L – side of the cube.

Reference

[1] Sedov L.I. “Continuum Mechanics, Volume 2”. M .: Science, 1970., 568 стр.

Result comparison

Below are the values for displacements, deformations and stresses at a point with coordinates (10,10,0) at the last loading step.

Spectral third order hexahedra

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Displacement vector components at grid nodes	Displacement Z	m	-1	-1	0
2	Displacement vector components at grid nodes	Displacement X	m	0.3	0.3	0
3	Displacement vector components at grid nodes	Displacement Y	m	0.3	0.3	0
4	Deformation tensor components at mesh nodes	Strain ZZ	-	-0.1	-0.1	0
5	Deformation tensor components at mesh nodes	Strain XX	-	0.03	0.03	0
6	Deformation tensor components at mesh nodes	Strain YY	-	0.03	0.03	0
7	Stress tensor components at mesh nodes	Stress ZZ	Pa	-2e10	-2e10	0
8	Step number	step	-	2	2	-

CAE Fidesys script:

```
reset
set default element hex
brick x 10 y 10 z 10
move Volume 1 location 5 5 5 include_merged
rotate Volume 1 angle 180 about Y include_merged
rotate Volume 1 angle -90 about Y include_merged
volume 1 size auto factor 5
volume 1 scheme auto
mesh volume 1
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 volume 1
block 1 material 1 cs 1 element solid order 3
create displacement on surface 2 dof 1 fix 0
create displacement on surface 3 dof 2 fix 0
create displacement on surface 6 dof 3 fix 0
create displacement on surface 4 dof 3 fix 1
create table 1
modify table 1 dependency time
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 1 2 value -0.5
```

modify table 1 cell 2 2 value -1
bcdep displacement 4 table 1
analysis type static elasticity dim3
static steps 2
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5

Third-order spectral tetrahedra

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Displacement vector components at grid nodes	Displacement Z	m	-1	-1	0
2	Displacement vector components at grid nodes	Displacement X	m	0.3	0.3	0
3	Displacement vector components at grid nodes	Displacement Y	m	0.3	0.3	0
4	Deformation tensor components at mesh nodes	Strain ZZ	-	-0.1	-0.1	0
5	Deformation tensor components at mesh nodes	Strain XX	-	0.03	0.03	0
6	Deformation tensor components at mesh nodes	Strain YY	-	0.03	0.03	0
7	Stress tensor components at mesh nodes	Stress ZZ	Па	-2e10	-2e10	0
8	Step number	step	-	2	2	-

CAE Fidesys script:

reset
brick x 10 y 10 z 10
move Volume 1 location 5 5 5 include_merged
rotate Volume 1 angle 180 about Y include_merged
rotate Volume 1 angle -90 about Y include_merged
volume 1 size auto factor 5
volume 1 scheme tetmesh proximity layers off geometry approximation angle 15
volume 1 tetmesh growth_factor 1
Trimesher surface gradation 1.3
Trimesher volume gradation 1.3
Trimesher geometry sizing on
mesh volume 1
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 volume 1
block 1 material 1 cs 1 element solid order 3
create displacement on surface 2 dof 1 fix 0

```
create displacement on surface 3 dof 2 fix 0
create displacement on surface 6 dof 3 fix 0
create displacement on surface 4 dof 3 fix 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 2 1 value 1
modify table 1 cell 3 1 value 2
modify table 1 cell 2 2 value -0.5
modify table 1 cell 3 2 value -1
bcdep displacement 4 table 1
analysis type static elasticity dim3
static steps 2
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5
```

1.14. Test Case No1.14

Problem Description

Uniform cube compression. The cube is divided into 13 parts, between which the conditions of tied contact with different values of the gaps are set. The test case checks the absence of point concentrators for the stress field value.

Input values

Geometric model:

- Cube with sides $0 \le x \le 10$ m, $0 \le y \le 10$ m, -10 m $\le z \le 0$.

Boundary conditions:

- Symmetry conditions;
- Pressure acts on the top of the cube 1e6 Pa;
- Contact surfaces: Autoselect;
- Tolerance: 0.0005;
- Type: Tied.

Material P

roperties:

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

Mesh:

- Spectral elements (order 3);
- Non-conformal meshes;

The geometry and finite element mesh for this test case is created using the CAE Fidesys script below.

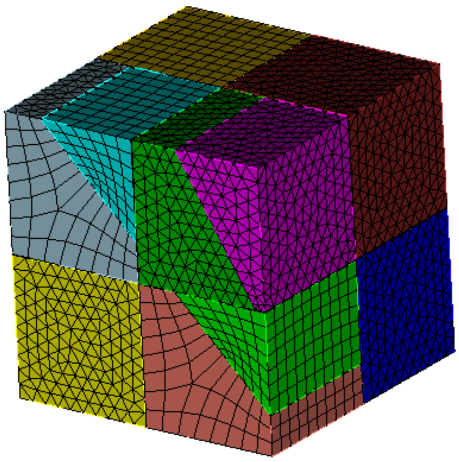


Fig. 1.25 - Finite-element mesh for a 13-part cube model

Calculation Settings:

- Static;
- Elasticity.

Output Values

No	Value	Description	Unit	Target
1	Component σ_{yy} of stress tensor	Stress YY	Pa	From -1000000.0 To -1000000.0

Calculation method used for the reference solution

Analytical solutions are calculated by the following formulas [1]:

$$\sigma_{yy} = P;\ \sigma_{xx} = \sigma_{zz} = \sigma_{xy} = 0.$$

Reference:

[1] Седов Л.И. “Механика сплошной среды, том 2”. М.: Наука, 1970г., 568 стр.

Results comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Component σ_{yy} of stress tensor	Stress YY	Pa	From -1000000.0 To -1000000.0	From -1000000 To 1000000	<<0.01

The distribution of the von Mises stress is shown in Figure 1.26.

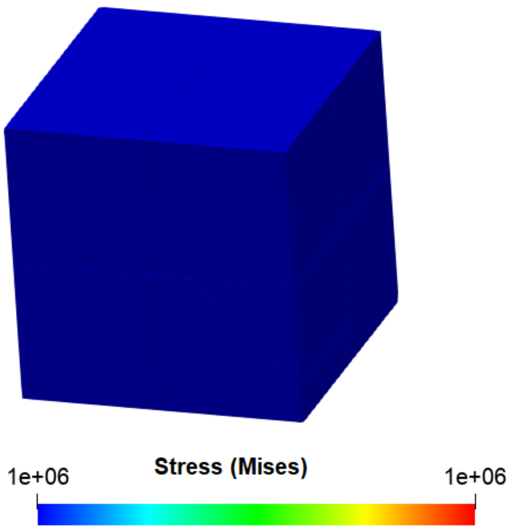


Fig. 1.26 - Von Mises Stress

CAE Fidesys Script:

```
reset
brick x 10
webcut body 1  with plane xplane offset 0
webcut body all with plane yplane offset 0
webcut volume all with plane zplane offset 0
webcut volume 4  with plane xplane offset 1 rotate 45 about z center 0 0 0
webcut volume 3  with plane xplane offset 4 rotate 30 about z center 0 0 0
webcut volume 1  with plane xplane offset 1 rotate 45 about z center 0 0 0
webcut volume 7  with plane xplane offset 2.5 rotate 45 about z center 0 0 0
webcut volume 6  with plane xplane offset -4 rotate 45 about z center 0 0 0
move Volume 3 10 1 11 7 5 12  x 0.01 include_merged
move Volume 7 5  z -0.01 include_merged
move Volume 10 3 1 11  z 0.01 include_merged
move Volume 8 6 13 4 9 2  x -0.01 include_merged
move Volume 8 6 13  z -0.01 include_merged
move Volume 4 9 2  z 0.01 include_merged
move Volume 4 3 10 9 7 8  y 0.01 include_merged
move Volume 2 11 1 13 6 5  y -0.01 include_merged
move Volume 4 3  y 0.01 include_merged
move Volume 11  y -0.01 include_merged
move Volume 13  y -0.01 include_merged
move Volume 7  y 0.02 include_merged
move Volume 3 4  y -0.01
move Volume 12 z -0.01
webcut volume all with plane from surface 48
webcut volume all with plane from surface 91
delete Volume 14 15 16
surface 95 89 42 21 84 85 75 scheme polyhedron
surface 95 89 42 21 84 85 75 size 0.95 #order,quality: 1,0.4 ; 2,0.95 ; 3,1.5
mesh surface 95 89 42 21 84 85 75
surface 76  scheme trimesh geometry approximation angle 15
surface 76 size 0.95 #order,quality: 1,0.6 ; 2,0.9; 3,1.35
Trimesher surface gradation 1.3
mesh surface 76
volume 4 6 12 1 11 scheme polyhedron
volume 1 12 size 0.9 #order,quality: 1,0.65 ; 2,0.9; 3,1.4
volume 6 11 size 0.95 #order,quality: 1,0.6 ; 2,0.9; 3,1.35
mesh volume 4 6 12 1 11
#cube2 tetra
volume 2 3 10 5 9 7 13 scheme tetmesh proximity layers off
volume 2 3 5 7 13 10 size 0.9 #order,quality: 1,0.6 ; 2,0.9; 3,1.3
volume 9 size 0.95 #order,quality: 1,0.65 ; 2,0.9; 3,1.45
Trimesher geometry sizing on
mesh volume 2 3 10 5 9 7 13
volume 8  redistribute nodes off
volume 8  scheme Sweep  source surface 76    target surface 73   sweep transform least squares
volume 8  autosmooth target on  fixed imprints off  smart smooth off
volume 8 size 0.9 #order,quality: 1,0.5 ; 2,0.9; 3,1.25
mesh volume 8
create material 1 name "LinearMat"
modify material 1 set property 'POISSON' value 0.3
```

```
modify material 1 set property 'MODULUS' value 2e+11
block 1 add volume all
block 1 material 1
block 1 element solid order 2
create displacement  on surface 76 69 51 138 119  dof 1 fix
create displacement  on surface 137 48 121 46 127  dof 2 fix 0
create displacement  on surface 150 115 35 21 118 141  dof 3 fix 0
create pressure  on surface 83 81 93 91 147 75  magnitude 1e6
create contact master surface 101 103 slave surface 95 89 tolerance 0.1 type tied method auto
create contact master surface 50 slave surface 70 tolerance 0.1 type tied method auto
create contact master surface 74 slave surface 54 tolerance 0.1 type tied method auto
create contact master surface 113 111 slave surface 44 tolerance 0.1 type tied method auto
create contact master surface 131 slave surface 49 tolerance 0.1 type tied method auto
create contact master surface 47 slave surface 139 120 tolerance 0.1 type tied method auto
create contact master surface 55 slave surface 43 tolerance 0.1 type tied method auto
create contact master surface 42 slave surface 100 130 tolerance 0.1 type tied method auto
create contact master surface 59 slave surface 79 86 tolerance 0.1 type tied method auto
create contact master surface 84 80 slave surface 72 tolerance 0.1 type tied method auto
create contact master surface 73 slave surface 116 149 tolerance 0.1 type tied method auto
create contact master surface 148 114 slave surface 90 94 tolerance 0.1 type tied method auto
create contact master surface 102 slave surface 97 tolerance 0.1 type tied method auto #vol 11 (hex) & vol 1 (hex)
create contact master surface 92 slave surface 87 tolerance 0.1 type tied method auto
create contact master surface 82 slave surface 77 tolerance 0.1 type tied method auto
create contact master surface 140 slave surface 117 tolerance 0.1 type tied method auto
create contact master surface 112 slave surface 107 tolerance 0.1 type tied method auto

analysis type static elasticity dim3
```

1.15. Test Case No1.15

Problem Description

The problem of integration of CAE Fidesys with the Euler software package on the stand model is considered.

Input Values

Geometric model is shown in Figure. 1.27:

- Volume 1 - a rectangular parallelepiped with dimensions of 0.01x0.01x0.005 m along the axes Ox, Oy, Oz respectively (surfaces bounding volume 1 are parallel to the coordinate planes);

- Surface 3 - simulates a plate 0.005 m thick, the median surface of which is a square 0.01x0.01 m. Surface 3 is parallel to the plane Oxy. The plate is similar to volume 1, but is modeled by shell elements;

- Lines 2 - beams element 0.0275 m long with a square section of 0.001x0.001 m, connect volume 1 and surface 3 and are orthogonal to them.

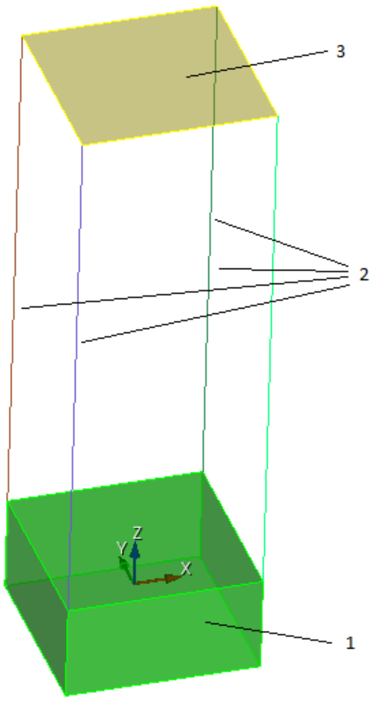


Figure. 1.27 - Geometrical model

Boundary conditions:

- since the problem of auto mechanics is considered, a series of calculations is performed. In this case, displacement restrictions are imposed on the vertices of the lower surface along the axis of volume 1.

Material Properties (Volume 1):

- Young's modulus $E_1 = 72000$ Pa;
- Poisson ratio $\nu_1 = 0.3$;
- Density ρ_1=2800 kg/m^3.

Material Properties (for Plate 3 and Beams):

- Young's modulus $E_2 = 200$ GPa;
- Poisson ratio $\nu_2 = 0.3$;
- Density ρ_2=8000 kg/m^3.

Mesh:

- Mesh including hexahedral, quadrangular shell and beam elements.

Calculation settings:

- Auto mechanic.

Output Values

As follows from the analytical representations [1], as a result of the calculation of the auto mechanics, matrices of stiffness, masses, as well as the eigenmodes of the model, which are the input parameters for the Euler software package, should be obtained. In this case, the correctness of these matrices follows from the fulfillment of the following conditions: - diagonal matrix, - unit matrix. Hence, the reference results for the problem are the parameters indicated in the table:

No	Value	Description	Unit	Target
1	Multiplication $E^T \cdot K \cdot E$	-	-	Diagonal matrix
2	Multiplication $E^T \cdot M \cdot E$	-	-	Unit matrix

Reference:

[1] В. Г. Бойков, И. В. Гаганов, Ф. Р. Файзуллин, А. А. Юдаков, «Моделирование движения механической системы, состоящей из деформируемых упругих тел, путём интеграции двух пакетов: EULER и Fidesys», Чебышевский сб., 18:3 (2017), 131–153.

Result comparison

No	Value	Description	Unit	Target	Error,%
1	Multiplication $E^T \cdot K \cdot E$	-	Diagonal matrix	diagonal	0
2	Multiplication $E^T \cdot M \cdot E$	-	Unit matrix	identity (E)	0

CAE Fidesys Script:

```
#{h=0.01}
reset
set default element hex
```

```
brick x {h} y {h} z {h/2}
#move volume 1 x {h/2} y {h/2} z {h/2}
create surface rectangle width {h} zplane
move body 2 z {3*h}
create curve vertex 1 12
create curve vertex 4 11
create curve vertex 3 10
create curve vertex 2 9
merge all
curve all size {h}
mesh curve all
mesh surface all
mesh volume 1
create material 1
modify material 1 name 'mat_bottom'
modify material 1 set property 'MODULUS' value 7.2e+4
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 2800
create material 2
modify material 2 name 'mat'
modify material 2 set property 'MODULUS' value 2e+11
modify material 2 set property 'POISSON' value 0.3
modify material 2 set property 'DENSITY' value 8000
block 1 add volume 1
block 1 material 1
block 1 element solid order 2
block 2 add surface 7
block 2 element shell order 2
block 2 material 2
create shell properties 2
modify shell properties 2 thickness {h/2}
modify shell properties 2 eccentricity 0.5
block 2 shell properties 2
block 3 add curve 17 to 20
block 3 material 2
block 3 element beam order 2
create beam properties 3
modify beam properties 3 type 'Rectangle'
modify beam properties 3 ey 0.0
modify beam properties 3 ez 0.0
modify beam properties 3 angle 0.0
modify beam properties 3 mesh_quality 6
modify beam properties 3 warping_dof off
modify beam properties 3 geom_H {h/10}
modify beam properties 3 geom_B {h/10}
block 3 beam properties 3
nodeset 1 add vertex 5 6 7 8
analysis type automechanics dim3 preload off
eigenvalue find 10 smallest
```

1.16. Test Case No1.16

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.

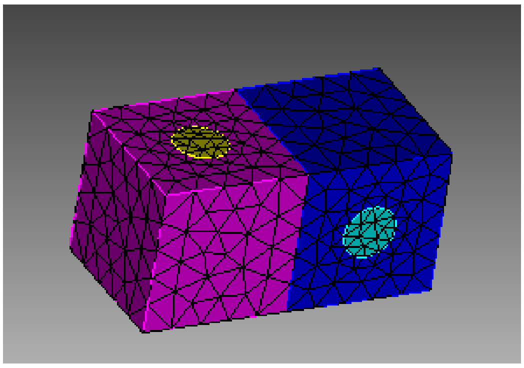


Fig 1.28 – Tetrahedral mesh

Target results

No	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

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficients	λ _11	$\frac{W}{m*K}$.	2.54285	2.534	-0.34%
2	Effective thermal conductivity coefficients	λ _22	$\frac{W}{m*K}$.	2.54285	2.531	-0.47%
3	Effective thermal conductivity coefficients	λ _33	$\frac{W}{m*K}$.	2.17647	2.292	0.26%

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.

1.17. Test Case No1.17

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: $\lambda = 10$;
- Matrix: $\lambda = 2$.

Boundary conditions:

- Periodic.

Mesh:

- First order tetrahedrons.

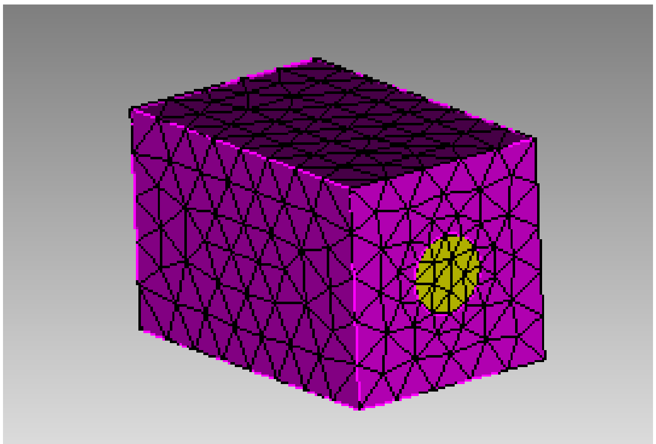


Fig 1.29 – Tetrahedral mesh

Target results

No	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:

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

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

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

First order tetrahedral mesh

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficients	λ _11	$\frac{W}{m*K}$.	2.8	2.773	0.95
2	Effective thermal conductivity coefficients	λ _22	$\frac{W}{m*K}$.	2.28571	2.2283	0.12
3	Effective thermal conductivity coefficients	λ _33	$\frac{W}{m*K}$.	2.28571	2.292	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.

1.18. Test Case No1.18

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.

Mesh:

- Second order hexahedrons.

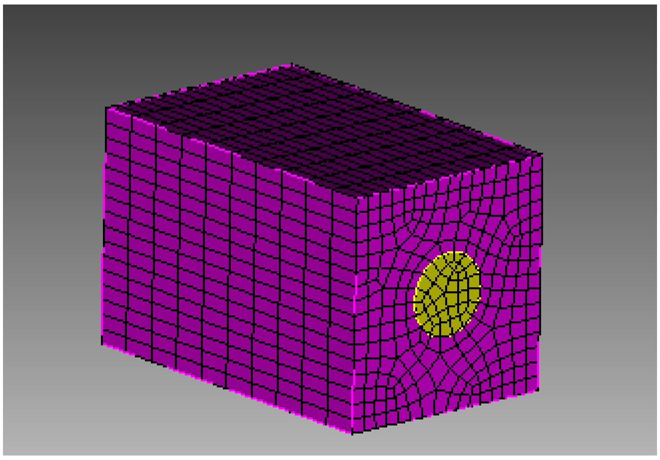


Fig 1.30 – Hexahedral mesh

Target results

No	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

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficients	λ _11	$\frac{W}{m*K}$.	$1.35709 * 10^{-5}$	$1.358 * 10^{-5}$	0.08%
2	Effective thermal conductivity coefficients	λ _22	$\frac{W}{m*K}$.	$8.58878 * 10^{-5}$	$8.484 * 10^{-5}$	1.22%
3	Effective thermal conductivity coefficients	λ _33	$\frac{W}{m*K}$.	$8.58878 * 10^{-5}$	$8.484 * 10^{-5}$	1.22%

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.

1.19. Test Case No1.19

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

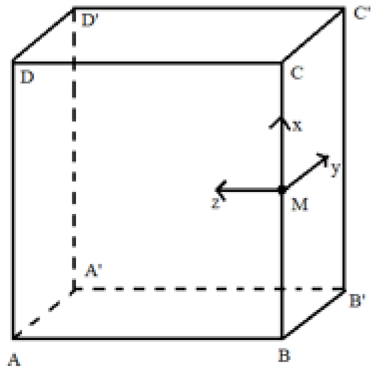


Fig 1.31 – Geometric model for Stokes problem

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).Mesh:

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.

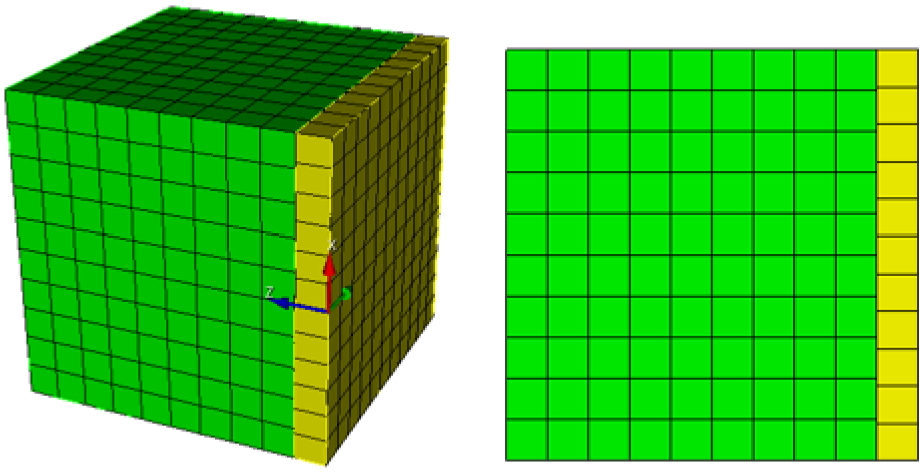


Fig 1.32 – Non-conformal finite element mesh for the Stokes problem

Target results

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

No	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
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

No	Value	Description	Unit	Target
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]:

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

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

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

No	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

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
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.

1.20. Test case No1.20

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

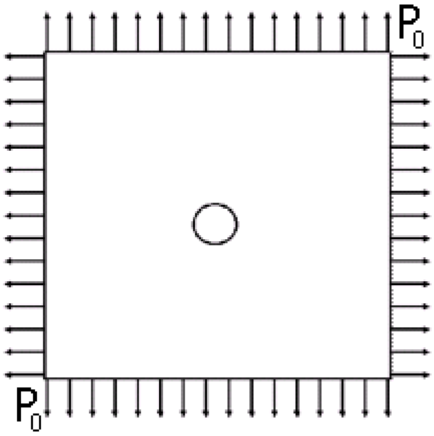


Figure 1.33 – Geometric model for a plate with ful all-round tension

Material prorerties:

- Young’s modulus E = 200 GPa;
- Poisson ratio v = 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

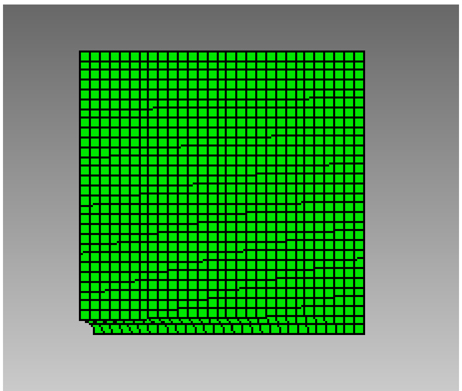


Fig 1.34 – 2D third order quadrangular spectral elements mesh

Target results

No	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

No	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.

1.21. Test case No1.21

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

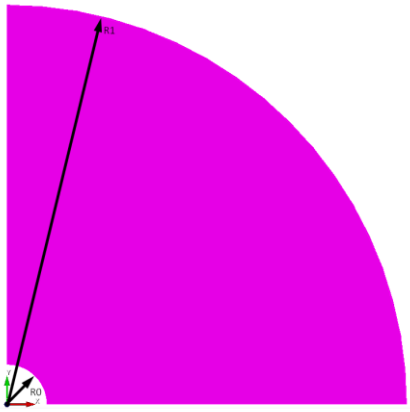


Fig 1.35 – Geometrical model

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.

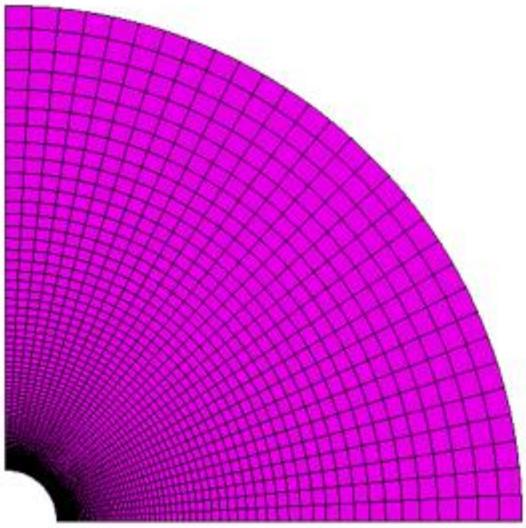


Fig 1.36 – 3rd order spectral elements for the Lamb problem

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).

No	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
5	Stress component σ_{yy}	(1.38922, 0,0)	Stress_YY	Pa	-9.68E+07	-9.757E+07	-0.78

No	Value	Point coordinates	Description	Unit	Target	CAE Fidesys result	Error, %
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

No	Value	Point coordinates	Description	Unit	Target	CAE Fidesys result	Error, %
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
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

No	Value	Point coordinates	Description	Unit	Target	CAE Fidesys result	Error, %
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
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

1.22. Test case No1.22

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

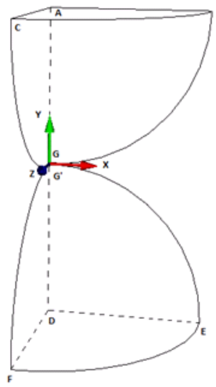


Fig 1.37 – Geometrical model

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

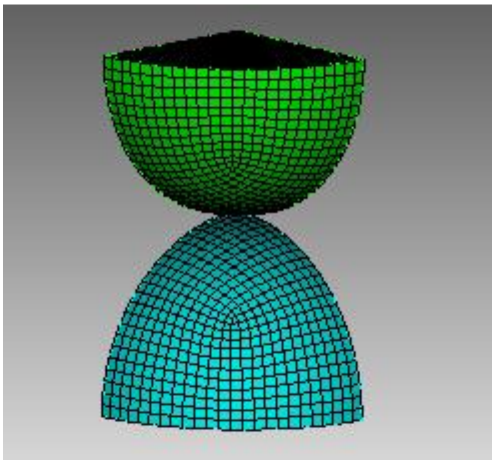


Fig 1.38 – Hexahedrons

Calculation settings:

- Static analysis;
- Elasticity;
- 3D.

Target results

No	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]:

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

Results

First order hexahedral mesh

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	σ_{yy} component of stress tensor	Stress YY	MPa	-2798.3	-2.744E+03	1.95

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.

1.23. Test case No1.23

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

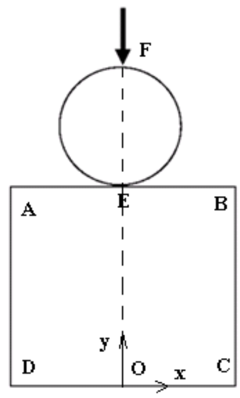


Fig 1.39 - Geometrical model

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

No	Value	Description	Point	Unit	Target
1	Stress tensor components in the contact zone	Contact Stress Node N	(0, -50, 0)	Pa	3600

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:

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

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:

$$R^* = \frac{R_1R_2}{R_1+R_2} .$$

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

No	Value	Description	Point	Unit	Target	CAE Fidesys result	Error, %
1	Stress tensor components in the contact zone	Contact Stress Node N	(0, -50, 0)	Pa	3600	3535	1.8

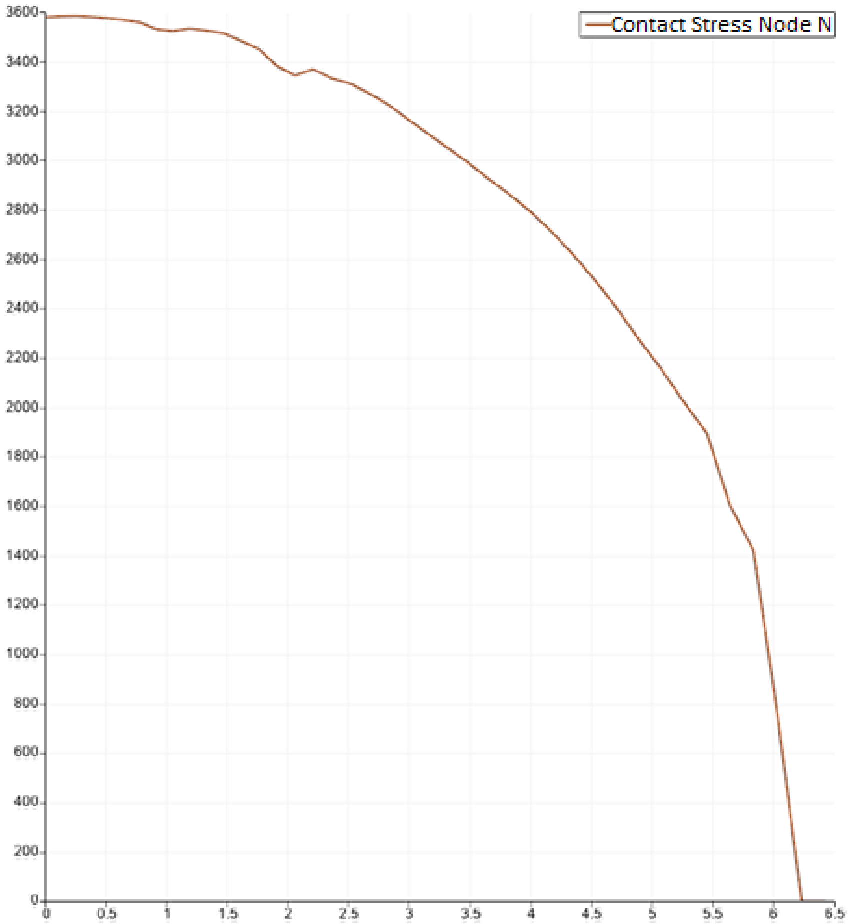


Fig. 1.40 - Graph of contact stress node stress N distribution with the contact zone 6.2 mm

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.

1.24. Test case No1.24

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.

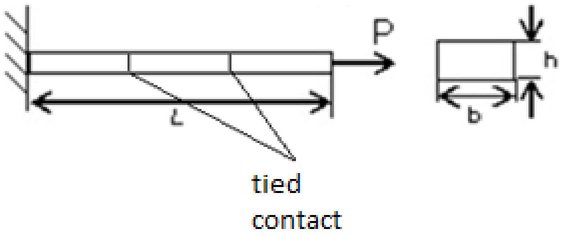


Fig 1.41 - Geometrical model

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/m^3.

Mesh:

- second order tetrahedral mesh.

Contact cettings:

- Rigid;
- Method: auto.

Analysis settings:

- Modal analysis;
- Preloaded model;

- Search for the first lowest frequency.

Target results

No	Value	Description	Value
1	Natural frequency	Eigen Values 1, Hz	86.16

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).

No	Value	Description	Unit	Value	CAE Fidesys result	Error, %
1	Natural frequency	Eigen Values 1	Hz	86.16	86.19	0.04

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)

1.25. Test case No1.25

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.

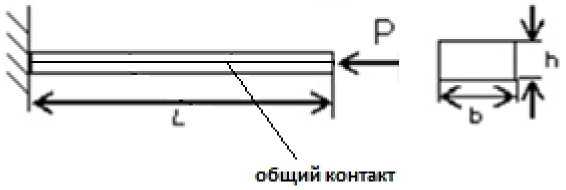


Fig 1.42 - Geometrical model

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

No	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).

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Critical force	Critical Values 1	-	44527	44590	0.15

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.

1.26. Test case No1.26

Problem description

Compression of an elastoplastic rectangular parallelepiped (multilinear isotropic hardening)

Input values

Geometrical model:

- Parallelepiped 5x1x1;

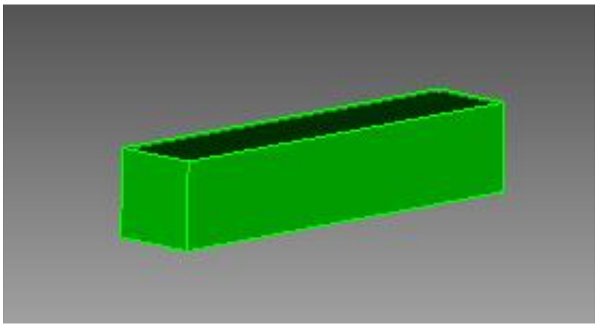


Fig 1.43 - Geometrical model

Boundary 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:

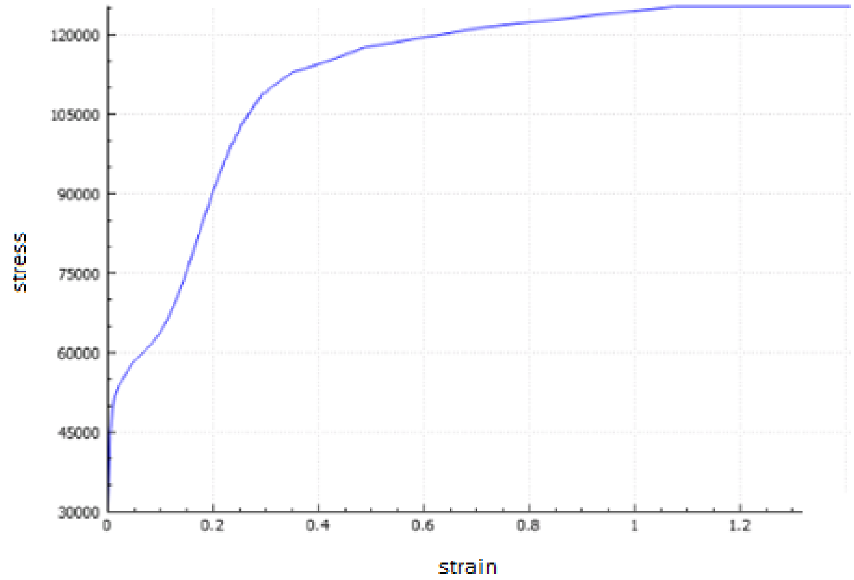


Fig 1.44 – Stress / plastic strain curve

Mesh:

- Second order hexahedrons.

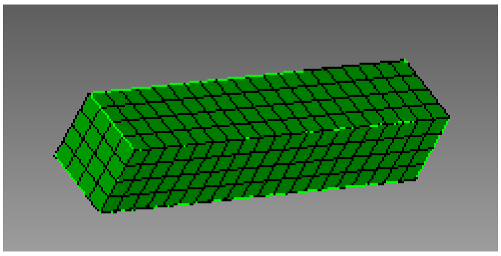


Fig 1.45 – Mesh

Target results

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

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

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

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

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

$$\varepsilon_{11}^p = -\varepsilon_{eq}^p$$

$$\varepsilon_{22}^p = \frac{1}{2}\varepsilon_{eq}^p = \varepsilon_{33}^p$$

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:

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

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

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

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

Results

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

1.27. Test case No1.27

Problem description

Tension of an elastoplastic rectangular parallelepiped (multilinear isotropic hardening)

Input values

Geometrical model:

- Parallelepiped 5x1x1;

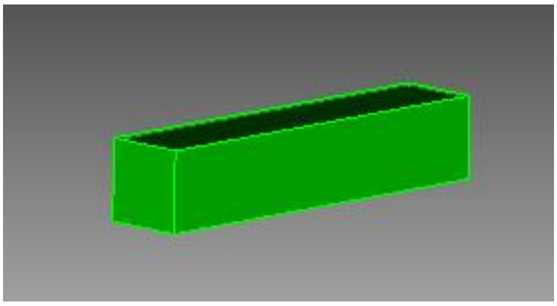


Fig 1.46 – Geometrical model

Boundary 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:

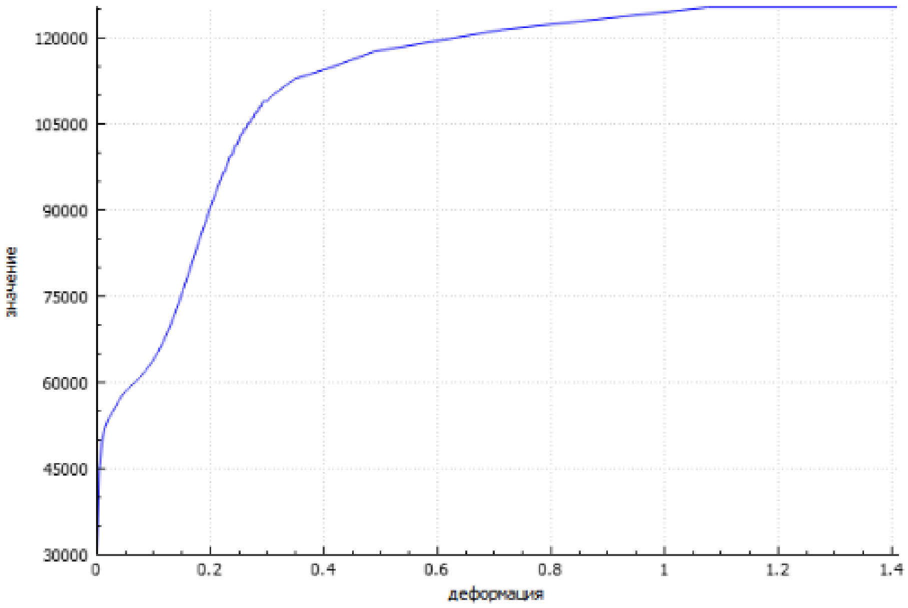


Fig 1.48 – Hardening curve

Mesh:

- Second order hexahedrons.

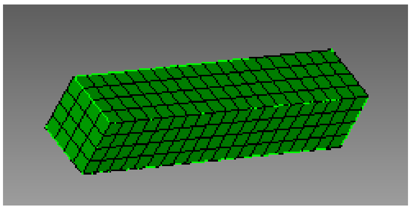


Fig 1.47 – Mesh

Target results

No	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

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

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:

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

$$\varepsilon_{11}^p = -\varepsilon_{eq}^p$$

$$\varepsilon_{22}^p = \frac{1}{2}\varepsilon_{eq}^p = \varepsilon_{33}^p$$

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

No	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
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)

1.28. Test case No 1.28

Problem description

Определение эффективных пороупругих механических для куба пористого материала (модель Био).

Input values

Material properties:

- Isotropic;
- Young's modulus E = 1e9 Pa;
- Poisson ratio ν = 0.25;
- Density ρ = 1800 кг/м^3;
- Cohesion 5.43712e6 Pa;
- Internal friction angle 21.43;
- Dilatancy angle 21.43;
- Porosity 0.25;
- Fluid's viscosity 0.005;
- Biot modulus 1;
- Fluid's bulk modulus 1e9 Pa;
- Fluid's density 1000 kg/m^3.

Geometrical model

- A solid cube with a side of 1m.

Boundary conditions:

- Nonperiodic.

Mesh:

- Hexahedron, first order.

Target results модуля

No	Value		Unit	Target
1	Effective Biot coefficients	BIOT_ALPHA	-	0

Analytical solution

Consider a representative volume V_0, chosen in the initial state, before deformation. On its boundary, we set boundary conditions in the form of zero displacements

$$u|_{\Gamma_0} = 0$$

We apply an internal pressure p to the inner surface of all pores and solve the boundary value problem of elasticity theory on a representative volume

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

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

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

As a result, we have that zero displacements of the boundary were set for the representative volume, providing zero effective deformations - and as a result of averaging, the effective deformation tensor σ^e was obtained. In general, this tensor will be non-zero due to the applied pore pressure. Effective poroelastic characteristics will be sought in the form

$$\sigma^e = -\alpha_{ij} p$$

For a homogeneous material, the numerically approximate analytical solution is trivial: due to the absence of pores, the effective Biot coefficients will be equal to zero. This works for isotropic, transversely isotropic and orthotropic materials.

Results

No	Value	Unit	Target	CAE Fidesys result	Error, %
1	Effective Biot coefficients	–	0	0	0.00

CAE Fidesys script:

```
reset
brick x 1
volume 1 size 1
mesh volume 1
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'MODULUS' value 1e+09
modify material 1 set property 'POISSON' value 0.25
modify material 1 set property 'DENSITY' value 1800
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 'SOLID_BULK_MODULUS' value 0
modify material 1 set property 'POROSITY' value 0.25
modify material 1 set property 'PERMEABILITY' value 0
modify material 1 set property 'FLUID_VISCOCITY' value 0.005
modify material 1 set property 'BIOT_ALPHA' value 1
modify material 1 set property 'BIOT_MODULUS' value 0
modify material 1 set property 'FLUID_BULK_MODULUS' value 1000
modify material 1 set property 'FLUID_DENSITY' value 1000
set duplicate block elements off
block 1 add volume 1
block 1 material 1 cs 1 element solid order 1
analysis type effectiveprops elasticity poroelast dim3 preload off
periodicbc off
```

Reference:

[1] P.C. Carman. Fluid flow through granular beds // Transactions, Institution of Chemical Engineers, London, Vol. 15, 1937. – P. 150–166.
[2] P.C. Carman. Flow of gases through porous media (Butterworths, London, 1956).

1.29. Test case No 1.29

Problem description

The problem of the movement of a load with an initial velocity along an inclined plane (taking into account the stiffness of the spring) is considered. The control task checks:

- the correctness of the calculation of the dynamic calculation of the model, taking into account the contact interaction "sliding contact with friction";
- solution for mismatched grids for spectral elements.

Input values модуля

Geometrical model:

- See figure 1.49;
- Cargo - cube 1x1 m;
- Base - Spring K = 30 kN/m.

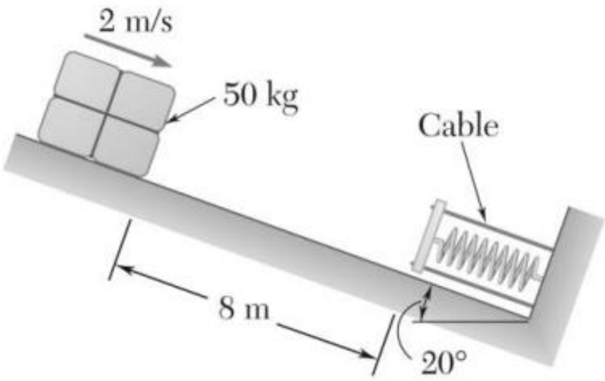


Fig. 1.49 - Geometrical model

Boundary conditions:

- Initial stress for spring 50mm;
- Initial velocity 2м/с;
- The base is rigidly fixed;
- Contact pare: general contact with frictioan, method Auto, Friction: 0.2.

Material properties:

- Young’s modulus Егруза = 2e11 Па;
- Poisson ratio ν=0.3;
- Base is rigid.

Mesh:

- Hexahedron;
- Order 3 and more.

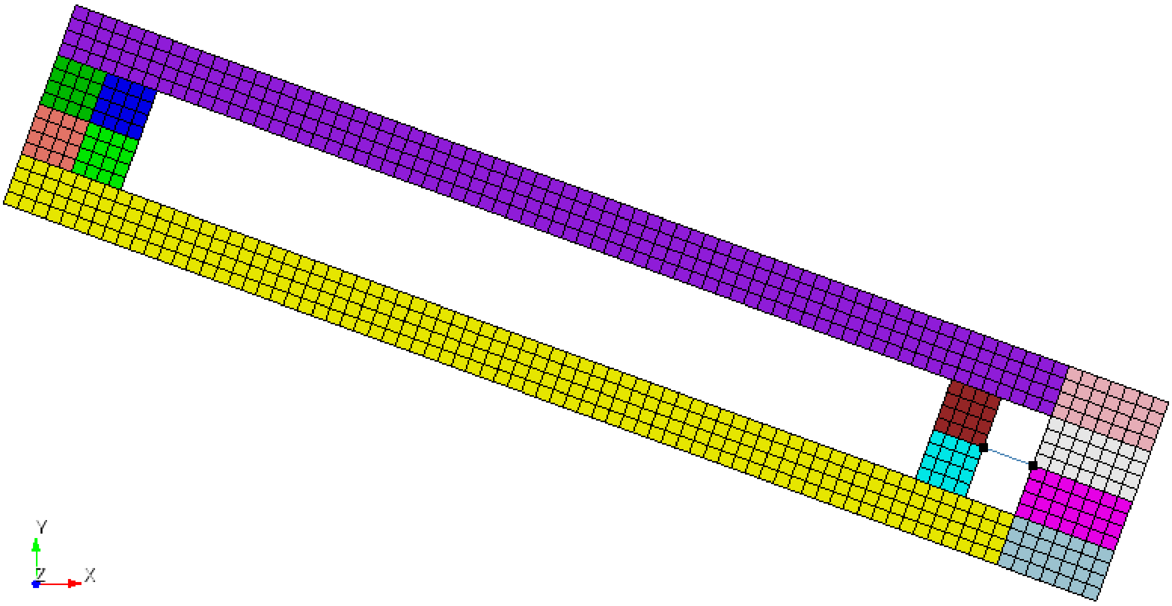


Fig. 1.50 – Finite elements mesh

Настройки расчета:

- Transient;
- 3D.

Target results модуля

No	Value	Description	Unit	Target
1	Displacement of the spring at the moment of complete stop of the load	Displacement_SUM	м	0.1744
2	The X component of the displacement vector at a point (8, -3, 0)	Displacement_X	м	0.164
3	The Y component of the displacement vector at a point (8, -3, 0)	Displacement_Y	м	0.06

Analytical solution

Analytical solution lies in solving the laws of conservation of work and energy[1].

$$T_1 + V_1 + U_{1\to 2} = T_2 + V_2$$

Position 1:

$$T_1 = \frac{1}{2} m v_1^2 = \frac{1}{2}(50)(2)^2 = 100\text{ J}$$

$$V_{1g} = mgh_1 = (50)(9.81)(8\sin 20°) = 1342.09\text{ J}$$

$$V_{1e} = \frac{1}{2} k e_1^2 = \frac{1}{2}(30\times 10^3)(0.05)^2 = 37.5\text{ J}$$

Position 2:

$$T_2 = \frac{1}{2} m v_2^2 = 0 \quad \text{since} \quad v_2 = 0.$$

$$V_{2g} = mgh_2 = (50)(9.81)(-x \sin 20°) = -167.76x$$

$$V_{2e} = \frac{1}{2} k e_2^2 = \frac{1}{2}(30 \times 10^3)(0.05 + x)^2 = 37.5 + 1500x + 15{,}000x^2$$

The work of friction forces:

$$+\nearrow \ \Sigma F_n = 0$$

$$N - mg \cos 20° = 0$$

$$\begin{aligned} N &= mg \cos 20° \\ &= (50)(9.81)\cos 20° \\ &= 460.92 \text{ N} \\ F_f &= \mu_k N \\ &= (0.2)(460.92) \\ &= 92.184 \\ U_{1\to 2} &= -F_f d \\ &= -92.184(8 + x) \\ &= -737.47 - 92.184x \end{aligned}$$

Reference:

[1] Vector mechanics for engineers (dynamics), 13.68(13.69)

Results

Spectral Hexahedrons (3 order) with friction

NoNo	Value	Description	Unit	Target	CAE Fidesys result	Error, %
11	Displacement of the spring at the moment of complete stop of the load (2.26 sec)	Displacement_SUM	м	0.1744	0.17741	1.7
22	The X component of the displacement vector at a point (8, -3, 0)	Displacement_X	м	00.164	0.166714	1.7
33	The Y component of the displacement vector at a point (8, -3, 0)	Displacement_Y	м	00.06	0.06067	1.1

CAE Fidesys script:

```
reset
create surface rectangle width 1 height 1 zplane
create surface rectangle width 10 height 0.5 zplane
Surface 1 copy move x 10
move Surface 2 x 4.5 y -0.75 include_merged
```

```
sweep curve 4 vector 1 0 0 distance 0.5 keep
webcut body all with plane yplane
Surface 7 copy move y -0.5
move Surface 10 9 x 8 include_merged
create curve vertex 26 24
webcut Surface 6 5 with plane xplane
Surface 11 2 copy move y 1.5
move Surface 12 to 15 x 6 include_merged
merge all
move Surface 12 to 15 x -6 include_merged
rotate Surface all angle -20 about Z include_merged
create cs type cartesian origin vertex 26 dir1 vertex 22 dir2 vertex 13
surface all size auto factor 4
mesh surface all
curve 39 interval 1
curve 39 scheme equal
mesh curve 39
create material 1
modify material 1 name 'mat 1'
modify material 1 set property 'MODULUS' value 2e+12
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 50
create material 2
modify material 2 name 'mat 2'
modify material 2 set property 'MODULUS' value 2e+11
modify material 2 set property 'POISSON' value 0.3
modify material 2 set property 'DENSITY' value 5e-3
set duplicate block elements off
block 1 add surface 2 17 11 7 8 16 9 10
block 1 material 2 cs 1 element plane order 3
block 2 add surface 13 12 14 15
block 3 add curve 39
block 2 material 1 cs 1 element plane order 3
create spring properties 1
modify spring properties 1 type 'linear_spring'
modify spring properties 1 stiffness 30000
modify spring properties 1 spring_constant_damping 0
modify spring properties 1 spring_linear_damping 0
modify spring properties 1 spring_mass 0
block 3 element spring
```

block 3 spring properties 1

move Surface 12 to 15 y -6 include_merged

create displacement on surface 17 2 8 16 7 11 dof all fix

create contact master curve 44 42 slave curve 58 type general friction 0 ignore_overlap off offset 0.0 tolerance 0.005 method penalty normal_stiffness 0.000001 tangent_stiffness 0.001

create contact master curve 51 47 slave curve 5 type general friction 0.2 ignore_overlap off offset 0.0 tolerance 0.005 method penalty normal_stiffness 1 tangent_stiffness 0.5

create contact master curve 19 21 slave curve 31 33 type general friction 0 ignore_overlap off offset 0.0 tolerance 0.005 method penalty normal_stiffness 0.01 tangent_stiffness 0.5

create directionrestraint on curve 14 16 displacement value 0 normal

move Surface 12 to 15 y 6 include_merged

create initial velocity on surface 14 15 13 12

modify initial velocity 1 dof 1 value 1.87938524

modify initial velocity 1 dof 2 value -0.684040291

modify initial velocity 1 cs 1

create gravity on surface all

modify gravity 1 dof 2 value -9.8

analysis type dynamic elasticity dim2 planestrain preload off

dynamic method full_solution scheme implicit steps 400 newmark_gamma 0.01 maxtime 2.4

output nodalforce on energy off record3d on log on vtu on material off results everystep 1

nonlinearopts maxiters 100 minloadsteps 1 maxloadsteps 1000 tolerance 0.000001 targetiter 5

create force on vertex 25 force value 1500 direction x

bcdep force 1 value 'if(t<2.21,0,-1500)'

bcdep force 1 cs 2

Hexahedron (order 2) with friction

NoNo	Value	Description	Unit	Target	CAE Fidesys result	Error, %
11	Displacement of the spring at the moment of complete stop of the load (2.26 sec)	Displacement_SUM	м	0.1744	0.17741	1.7
22	The X component of the displacement vector at a point (8, -3, 0)	Displacement_X	м	00.164	0.166714	1.7
33	The Y component of the displacement vector at a point (8, -3, 0)	Displacement_Y	м	00.06	0.06067	1.1

CAE Fidesys script:

reset
create surface rectangle width 1 height 1 zplane
create surface rectangle width 10 height 0.5 zplane
Surface 1 copy move x 10
move Surface 2 x 4.5 y -0.75 include_merged
sweep curve 4 vector 1 0 0 distance 0.5 keep
webcut body all with plane yplane
Surface 7 copy move y -0.5
move Surface 10 9 x 8 include_merged

```
create curve vertex 26 24
webcut Surface 6 5 with plane xplane
Surface 11 2 copy move y 1.5
move Surface 12 to 15 x 6 include_merged
merge all
move Surface 12 to 15 x -6 include_merged
rotate Surface all angle -20 about Z include_merged
create cs type cartesian origin vertex 26 dir1 vertex 22 dir2 vertex 13
surface all size auto factor 4
mesh surface all
curve 39 interval 1
curve 39 scheme equal
mesh curve 39
create material 1
modify material 1 name 'mat 1'
modify material 1 set property 'MODULUS' value 2e+12
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 50
create material 2
modify material 2 name 'mat 2'
modify material 2 set property 'MODULUS' value 2e+11
modify material 2 set property 'POISSON' value 0.3
modify material 2 set property 'DENSITY' value 5e-3
set duplicate block elements off
block 1 add surface 2 17 11 7 8 16 9 10
block 1 material 2 cs 1 element plane order 3
block 2 add surface 13 12 14 15
block 3 add curve 39
block 2 material 1 cs 1 element plane order 3
create spring properties 1
modify spring properties 1 type 'linear_spring'
modify spring properties 1 stiffness 30000
modify spring properties 1 spring_constant_damping 0
modify spring properties 1 spring_linear_damping 0
modify spring properties 1 spring_mass 0
block 3 element spring
block 3 spring properties 1
move Surface 12 to 15 y -6 include_merged
create displacement on surface 17 2 8 16 7 11 dof all fix
create contact master curve 44 42 slave curve 58 type general friction 0 ignore_overlap off offset 0.0 tolerance 0.005 method penalty normal_stiffness 0.000001 tangent_stiffness 0.001
create contact master curve 51 47 slave curve 5 type general friction 0.2 ignore_overlap off offset 0.0 tolerance 0.005 method penalty normal_stiffness 1 tangent_stiffness 0.5
create contact master curve 19 21 slave curve 31 33 type general friction 0 ignore_overlap off offset 0.0 tolerance 0.005 method penalty normal_stiffness 0.01 tangent_stiffness 0.5
create directionrestraint on curve 14 16  displacement value 0 normal
move Surface 12 to 15 y 6 include_merged
create initial velocity on surface 14 15 13 12
modify initial velocity 1 dof 1 value 1.87938524
modify initial velocity 1 dof 2 value -0.684040291
modify initial velocity 1 cs 1
create gravity on surface all
modify gravity 1 dof 2 value -9.8
analysis type dynamic elasticity dim2 planestrain preload off
dynamic method full_solution scheme implicit steps 400 newmark_gamma 0.01 maxtime 2.4
output nodalforce on energy off record3d on log on vtu on material off results everystep 1
nonlinearopts maxiters 100 minloadsteps 1 maxloadsteps 1000 tolerance 0.000001 targetiter 5
create force  on vertex 25 force value 1500 direction x
bcdep force 1 value 'if(t<2.21,0,-1500)'
bcdep force 1 cs 2
```

1.30. Test case No 1.30

Problem description

The problem of beam bending under the action of axial and small transverse loads is considered. The beam is subjected to pure compression until a critical bending load is reached, after which the beam deflects with large transverse displacements. The control task checks:
- calculation taking into account finite deformations;
- stepped load (change of boundary conditions between steps).

Input values

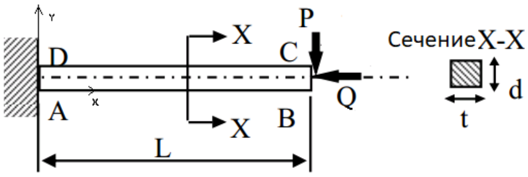


Fig. 1.51 - Geometrical model

Geometrical model:

- See figure 1.51;
- Beam 3.2×0.1×0.1 м;
- A = (0, 0, 0).

Boundary conditions:

- Left end fixed in all directions;
- The entire volume is fixed along the axis Oz: uz = 0;
- P = 3.844e2 N, 3.844e4 N, Q=3.844e3 N, 3.844e6 N;
- Forces Q and P are applied to the vertices of the right end along the axes Ox and Oy, respectively.

Material properties:

- Isotropic;
- Young's modulus E = 200 ГПа;
- Poisson ratio ν = 0.0.

Mesh:

- Hexahedron.

Target results модуля

No	Value	Description	Unit	Target
1	The Ux component of the displacement vector at the point (3.2,0.05, -0.05) at step 2	Displacement_X	m	-5.0464
2	The Uy component of the displacement vector at the point (3.2,0.05, -0.05) at step 2	Displacement_Y	m	-1.3472

No	Value	Description	Unit	Target
3	The Uz component of the displacement vector at the point (3.2,0.05, -0.05) at step 2	Displacement_Z	m	0
4	Step number	step	-	2

Analytical solution

An analytical solution to this problem can be obtained on the basis of the fundamental theory of Bernoulli-Euler, taking into account large displacements of the beam. The solution is a non-linear second-order differential equation from which displacements and curvature can be derived using elliptic integrals. The reference solution is presented in [1] based on [2].

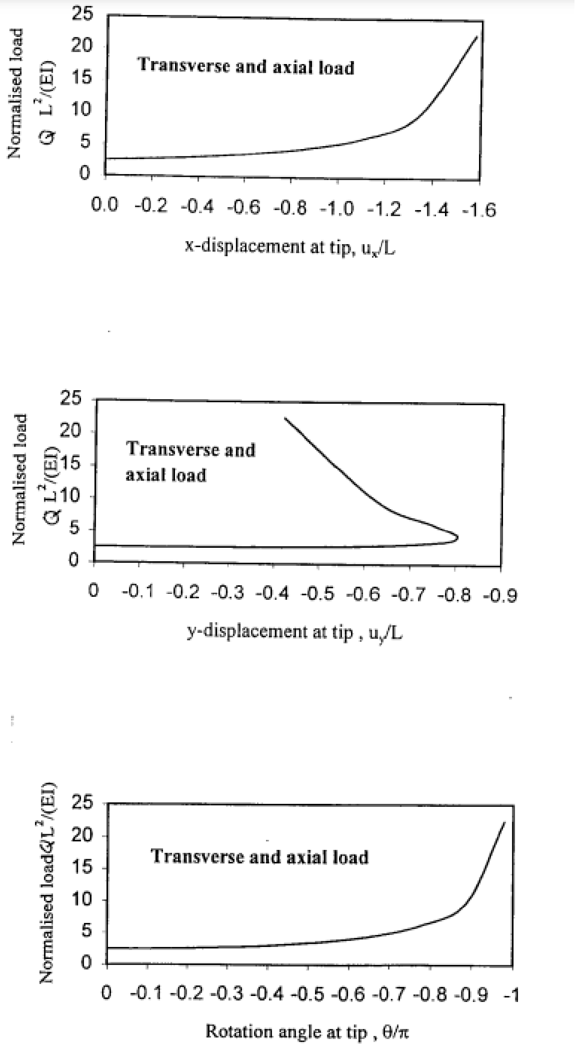


Fig. 1.52 - Referense solution presented in[1,2]

Reference

[1] NAFEMS R0072 Introduction to Non-Linear Finite Element Analysis (FE Example 5: Cantilever Problem) (пример 5.b, страница 196)
[2] Lyons P. and Holsgrove, S. [1989]

Finite elements benchmarks for 2D beams and axisymmetric shells involving geometry non-linearity, NAFEMS Report, P10

Results

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	The Ux component of the displacement vector at the point (3.2,0.05, -0.05) at step 2	Displacement_X	m	-5.0464	-5.06008	0.27
2	The Uy component of the displacement vector at the point (3.2,0.05, -0.05) at step 2	Displacement_Y	m	-1.3472	-1.36135	1.05
3	The Uz component of the displacement vector at the point (3.2,0.05, -0.05) at step 2	Displacement_Z	m	0	0	<0.01
4	Step number	step	-	2	2	-

CAE Fidesys script:

```
#{P=3.844e4}
#{Q=3.844e6}
reset
set default element hex
brick x 3.2 y 0.1 z 0.1
move Volume 1 x 1.6 y 0.05 z -0.05 include_merged
volume 1 size 0.05 #order,quality: 1,0.01
mesh volume 1
create material 1
modify material 1 name 'material'
modify material 1 set property 'POISSON' value 0
modify material 1 set property 'MODULUS' value 210e9
set duplicate block elements off
block 1 add volume 1
block 1 material 1 cs 1 element solid order 2
create displacement on surface 4 dof all fix 0
create displacement on volume 1 dof 3 fix 0
create force on vertex 1 2 5 6 force value 1 direction nx
create force on vertex 1 2 5 6 force value 1 direction ny
create table 1
modify table 1 dependency time
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 1 2 value 96.1
modify table 1 cell 2 2 value 9610
bcdep force 2 table 1
create table 2
modify table 2 dependency time
modify table 2 insert row 1
modify table 2 insert row 1
modify table 2 cell 1 1 value 1
modify table 2 cell 2 1 value 2
modify table 2 cell 1 2 value 961
modify table 2 cell 2 2 value 961000
bcdep force 1 table 2
static steps 2
analysis type static elasticity findefs dim3
```

1.31. Test case No 1.31

The problem of compression of an elastic-plastic sample with asymmetric hardening is solved.

Input values

- Young's modulus E = 5.1e+6 Pa
- Poisson ratio ν = 0.25
- Yield strength σt = 1.5541e+4 Pa
- Yield compressive strength σc = 4.3414e+4 Pa
- Stress(Strain) curve for compression (table):

ε_p^{eq}	σ_{eq}, Па
0	43414
1	1043414

Geometrical model:

- Rectangular brick with sides 0≤ x ≤ 5, 0 ≤ y ≤ 1, 0 ≤ z ≤ 1

Boundary conditions:

- For y = 0 uy = 0
- For z = 0 uz = 0
- For volume ux = -0.12*x/5

Mesh:

- Hexahedron (first order).

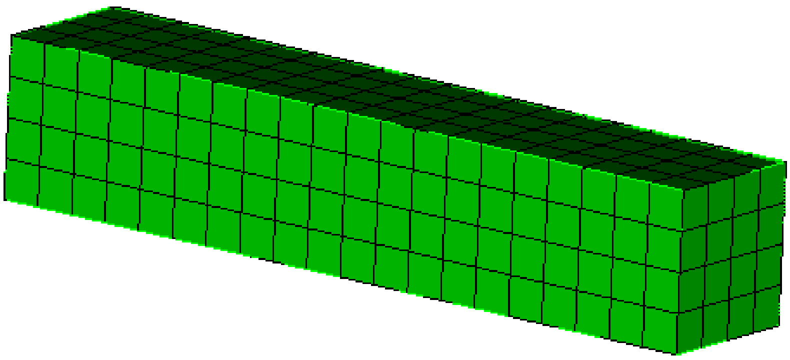


Fig. 1.53 – Finite Elements Mesh

Target results модуля

No.	Coordinates	Description	Unit	Target
1	(0; 0; 0)	Stress σ_{xx}	Pa	-43414.2
2	(0; 0; 0)	Stress σ_{xx}	Pa	-64826.3838

Analytical solution

Analytical solution based on the method proposed in [1]:
Expression for σ11:

$$\sigma_{11}(t) = \frac{\sigma_c(\beta - 1) + E_c \varepsilon_{11}(t)}{1 - \beta + \frac{E_t}{E}}$$

where E_c is the compressive strength modulus calculated for linear hardening as follows:

$$E_c = \frac{\sigma_c^u - \sigma_c}{\varepsilon_u^p}$$

where σ_c^u - ultimate stress, ε_c^u - ultimate plastic strain.

Reference:

[1] Code_Aster Integration of the elastoplastic mechanical behaviors of Drucker-Prager, associated (DRUCK_PRAGER)and non-aligned (DRUCK_PRAG_N_A) and postprocessings https://www.code-aster.org/V2/doc/v12/en/man_r/r7/r7.01.16.pdf

Results

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Stress σ_{xx} at the point (5, 0, 0) in 0.36 sec	Stress XX	Pa	-43414.2	-4.359E+04	0.41
2	Stress σ_{xx} at the point (5, 0, 0) in 1 sec	Stress XX	Pa	-64826.38	-6.483E+04	0.0

CAE Fidesys script:

```
reset
set default element hex
brick x 5 y 1 z 1
move volume 1 x 2.5 y 0.5 z 0.5
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 'DP_YIELD_STRENGTH_COMPR' value 4.3414e4
modify material 1 set property 'DP_YIELD_STRENGTH' value 1.5541e4
```

```
create table 1
modify table 1 dependency strain
modify table 1 insert row
modify table 1 cell 1 1 value 0
modify table 1 cell 1 2 value 4.3414e4
modify table 1 insert row
modify table 1 cell 2 1 value 1
modify table 1 cell 2 2 value 104.3414e4
modify material 1 set property 'SIGMA_CURVE_COMPR' 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 0.25
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
bcdep displacement 3 value '-0.12*x/5'
analysis type static elasticity plasticity dim3
nonlinearopts maxiters 1000 minloadsteps 100 maxloadsteps 1000000 tolerance 1e-3 targetiter 5
```

1.32. Test case No 1.32

Problem description

The problem of tension of an elastic-plastic sample with asymmetric strengthening is considered.

Input values

Material properties:

- Young's modulus E = 5.1e+6;
- Poisson ratio $\nu = 0.25$;
- Yield strength $\sigma_t = 1.5541e+4$;
- Yield compressive strength $\sigma_c = 4.3414e+4$;
- Stress(Strain) curve for compression (table):

ε_p^{eq}	σ_{eq}, Па
0	43414
1	1043414

Геометрия:

- Rectangular brick with sides $0 \le x \le 5$, $0 \le y \le 1$, $0 \le z \le 1$

Boundary conditions:

- For y = 0 $u_y = 0$
- For z = 0 $u_z = 0$
- For volume $u_x = 0.12*x/5$

Mesh:

- Hexahedron (first order).

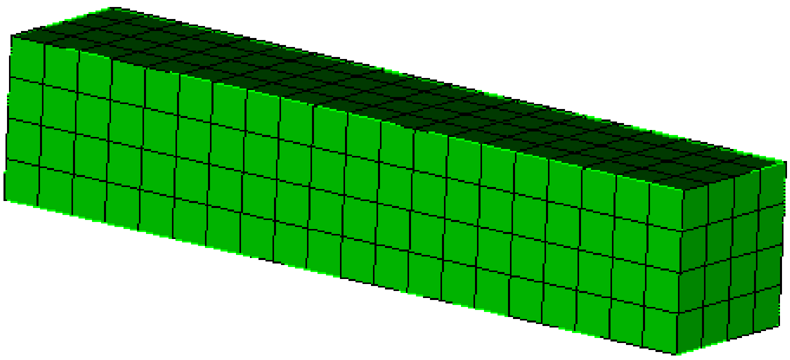


Fig. 1.54 – Finite Elements Mesh

Target results

No.	Coordinates	Value	Unit	Target
1	(0; 0; 0)	Stress σ_{xx} t=0.13 c	Pa	15541
2	(0; 0; 0)	Stress σ_{xx} t=1 c	Pa	20402.0642

Analytical solution

Analytical solution based on the method proposed in [1]:
Expression for σ11:

$$\sigma_{11}(t) = \frac{\sigma_t (1+\beta)^2 + E_c (1-\beta)\varepsilon_{11}(t)}{(1+\beta)^2 + \frac{E_c}{E}(1-\beta)}$$

where E_c is the compressive strength modulus calculated for linear hardening as follows:

$$E_c = \frac{\sigma_c^u - \sigma_c}{\varepsilon_u^p}$$

where σ_c^u - ultimate stress, ε_p^u - ultimate plastic strain.

Reference:

[1] Code_Aster Integration of the elastoplastic mechanical behaviors of Drucker-Prager, associated (DRUCK_PRAGER)and non-aligned (DRUCK_PRAG_N_A) and postprocessings https://www.code-aster.org/V2/doc/v12/en/man_r/r7/r7.01.16.pdf

Results

No.	Coordinates	Value	Unit	Target	CAE Fidesys result	Error, %
1	(0; 0; 0)	Stress σ_{xx}	Па	15541	1.556E+04	0.11
2	(0; 0; 0)	Stress σ_{xx}	Па	20402.0642	2.040E+04	<0.01

CAE Fidesys script:

```
reset
set default element hex
brick x 5 y 1 z 1
move volume 1 x 2.5 y 0.5 z 0.5
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 'DP_YIELD_STRENGTH_COMPR' value 4.3414e4
```

```
modify material 1 set property 'DP_YIELD_STRENGTH' value 1.5541e4
create table 1
modify table 1 dependency strain
modify table 1 insert row
modify table 1 cell 1 1 value 0
modify table 1 cell 1 2 value 4.3414e4
modify table 1 insert row
modify table 1 cell 2 1 value 1
modify table 1 cell 2 2 value 104.3414e4
modify material 1 set property 'SIGMA_CURVE_COMPR' table 1 #a
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 0.25
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
bcdep displacement 3 value '0.12*x/5'
analysis type static elasticity plasticity dim3
nonlinearopts maxiters 1000 minloadsteps 10 maxloadsteps 1000000 tolerance 1e-3 targetiter 5
```

1.33. Test case No 1.33

Problem description

The problem of plate stability is considered with the addition of a contact condition. In the control task, the correctness of the calculation of the analysis of the buckling of the model is checked, taking into account the contact interaction "sliding contact with friction".

Input values

Geometrical model:

- See figure 1.55;
- Width b = 0,1 м;
- Thickness h = 0,002 м;
- Length a = 0,1 м.

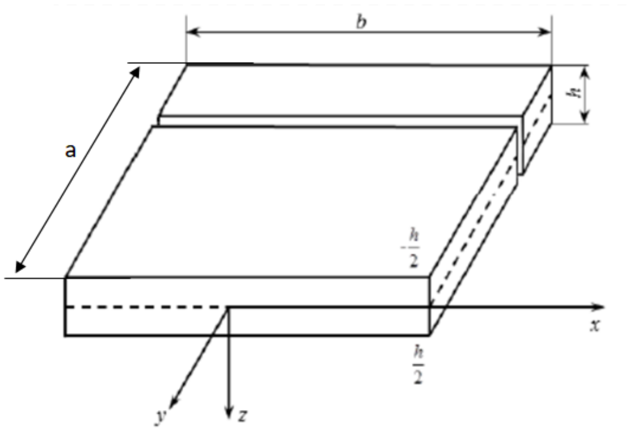


Fig. 1.55 - Geometrical model

Boundary conditions:

- See figure 1.56;
- Both ends of the rod rest on hinges;
- Contact pair - selection of main and secondary entities, General contact with friction, Autoselect method, Friction: 0, 0.2, 1;
- There is a compressive force.

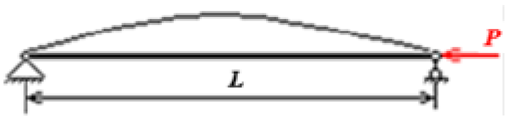


Fig. 1.56 – Boundary conditions

Material properties:

- Young's modulus E = 2e11 Па;
- Poisson ratio ν=0.3;
- Density ρ=1000 кг/м3.

Mesh:

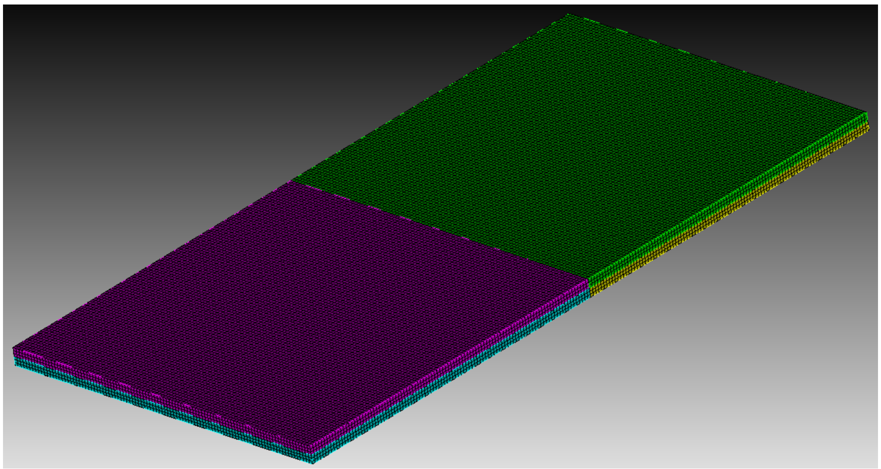
- Hexahedron (2 order).
- 

Fig. 1.57 – Finite Elements Mesh

Calculation settings:

- Buckling;
- 3D;
- Number of buckling modes: 1.

Target results модуля

No	Value	Description	Unit	Target
1	First critical load factor	load multipliers(1)	-	56220.0

Analytical solution

The problem has an approximate analytical solution [1] given below.

The approximate formula for the critical stress of the plate becomes:

$$\sigma_{crit} = E\left(\frac{\delta}{b}\right)^2$$

The critical buckling force of the plate is determined by the formulas:

$$P_{crit} = E\frac{\delta^3}{b}$$

Reference:

[1] Е.И. Орешко, В.С. Ерасов, А.Н Луценко Особенности расчетов устойчивости стержней и пластин // Авиационные материалы и технологии. – 2016 - No4(45)
УДК 517.25, DOI 10.18577/207-9140-2016-0-4-74-79.

Results

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	First critical load factor	load multipliers(1)	-	56220.0	5.475E+03	2.78

CAE Fidesys script:

```
reset
brick x 0.1 y 0.1 z 0.002
webcut volume 1 with plane zplane offset 0
webcut volume all with plane yplane offset 0
curve 18 26 20 25 interval 2
curve 18 26 20 25 scheme equal
move Volume 3 4 y .02 include_merged
merge all
move Volume 3 4 y -.02 include_merged
volume all size auto factor 7
mesh volume all
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 7.9e+10
modify material 1 set property 'DENSITY' value 100
block 1 add volume all
block 1 material 1 cs 1 element solid order 2
create pressure on surface 31 19 33 25 magnitude 500 # p = 0.1 H
create displacement on curve 35 43 dof 2 dof 3 fix 0
create displacement on curve 41 36 dof 1 dof 2 dof 3 fix 0
create contact master surface 17 27 slave surface 32 22 type general friction 0.1 ignore_overlap off offset 0.0 tolerance 0.0005 method auto
analysis type buckling elasticity dim3
eigenvalue find 1 smallest
```

2. Test cases with numerically approximate analytical solutions

2.1. Test Case No2.1

Problem Description

Determination of effective mechanical characteristics for a two-layer layer-fiber composite. A numerical approximate solution is used.

Input Values

Material Properties:

Steel:

- Young's modulus E = 200 kPa;
- Poisson ratio ν = 0.25.

Rubber:

- Young's modulus E = 2 Pa;
- Poisson ratio ν = 0.49.

Geometric model:

Generate automatically using interface with parameters:

- Laminated fiber composite;
- Thread diameter – 6.0;
- Filament angle – 30 degrees;
- Thread pitch– 8.0;
- Layer thickness– 16.0.

Boundary conditions:

- Periodic.

Mesh

- Tetrahedron mesh order 2.

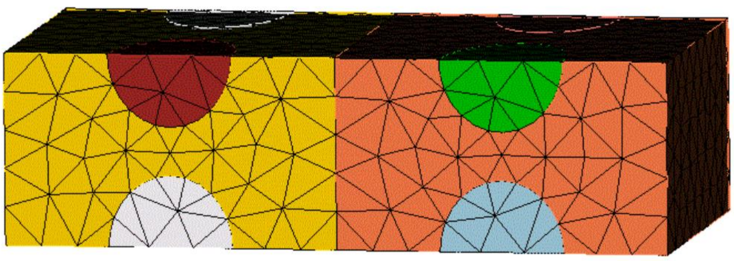


Fig 2.1 – Mesh 3D - Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	24852.4
2	Effective elastic modulus	C_1122	Pa	8281.54
3	Effective elastic modulus	C_2222	Pa	2763.12
4	Effective elastic modulus	C_1212	Pa	8283.5
5	Density	Density	$кг / м^3$	0

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e:

31. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

32. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

33. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

34. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

35. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

36. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a_{ij}^e = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^e}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11}q,\ \ \sigma_{ij} = \alpha_{ij}^{(1)}q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22}q,\ \ \sigma_{ij} = \alpha_{ij}^{(2)}q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33}q,\ \ \sigma_{ij} = \alpha_{ij}^{(3)}q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21})q,\ \ \sigma_{ij} = \alpha_{ij}^{(4)}q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31})q,\ \ \sigma_{ij} = \alpha_{ij}^{(5)}q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32})q,\ \ \sigma_{ij} = \alpha_{ij}^{(6)}q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)};$

2) $C_{ij22} = \alpha_{ij}^{(2)};$

3) $C_{ij33} = \alpha_{ij}^{(3)};$

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)};$

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)};$

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}.$

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	24852.4	24854.847	0.01
2	Effective elastic modulus	C_1122	Pa	8281.54	8308.11363	0.32
3	Effective elastic modulus	C_2222	Pa	2763.12	2799.135749	1.3
4	Effective elastic modulus	C_1212	Pa	8283.5	8283.85704	0.004
5	Density	Density	$кг / м^3$	0	0	0

Script CAE Fidesys:

```
reset
create brick width 16 depth 9.2376 height 16
create cylinder height 50.4752 radius 3
volume 2 rotate 90.0 about y
volume 2 rotate 30 about z
volume 2 move y -9.2376
create cylinder height 50.4752 radius 3
volume 3 rotate 90.0 about y
volume 3 rotate 30 about z
volume 3 move y 0
create cylinder height 50.4752 radius 3
volume 4 rotate 90.0 about y
volume 4 rotate 30 about z
volume 4 move y 9.2376
intersect volume 1 2 keep
intersect volume 1 3 keep
intersect volume 1 4 keep
delete volume 2
delete volume 3
delete volume 4
```

```
subtract volume 5 6 7 from volume 1 keep
delete volume 1
volume all move z 8
volume all move z 16 copy
volume 9 10 11 12 reflect 1.0 0.0 0.0
imprint volume all
merge volume all
block 1 volume 5 6 7 9 10 11
block 2 volume 8 12
#{steel_E = 2.0e5}
#{steel_nu = 0.25}
#{rub_E = 2.0}
#{rub_nu = 0.49}
#{mesh_size = 2.0}
create material 1
modify material 1 name 'steel'
modify material 1 set property 'POISSON' value {steel_nu}
modify material 1 set property 'MODULUS' value {steel_E}
create material 2
modify material 2 name 'rubber'
modify material 2 set property 'POISSON' value {rub_nu}
modify material 2 set property 'MODULUS' value {rub_E}
block 1 material 1
block 2 material 2
block 1 2 element solid order 2
volume all scheme Tetmesh
set tetmesher interior points on
set tetmesher optimize level 3 overconstrained off sliver off
set tetmesher boundary recovery off
volume all tetmesh growth_factor 1.0
volume all size {mesh_size}
volume all size {mesh_size}
mesh volume all
analysis type effectiveprops elasticity dim3
periodicbc on
```

2.2.Test Case No2.2

Problem Description

Determination of effective mechanical characteristics for a dispersed composite of periodic structure, reinforced with spherical inclusions.

Input Values

Material Properties:

Matrix:

- Young's modulus = 1 Pa;
- Poisson ratio ν = 0.4;
- Density ρ = 1000 kg/m^3.

Inclusion:

- Young's modulus E = 10 Pa;
- Poisson ratio ν = 0.25;
- Density ρ = 10000 kg/m^3.

Geometric model:

- Solid cube with side 1 m;
- In the center, an inclusion in the form of a ball with radius 0.228542449538.

Boundary conditions:

- Periodic.

Mesh

- Tetrahedron mesh order 2.

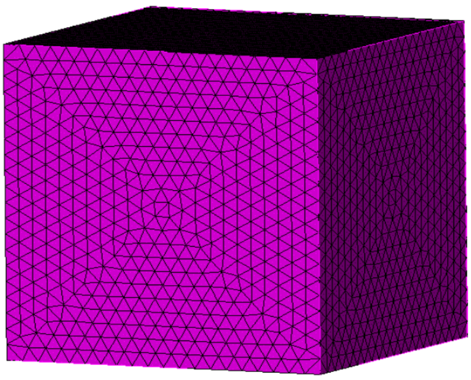


Fig 2.2 – Mesh 3D – Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	2.26175
2	Effective elastic modulus	C_1122	Pa	1.48163
3	Effective elastic modulus	C_1133	Pa	1.48163
4	Effective elastic modulus	C_1212	Pa	0.39006
5	Effective elastic modulus	C_1313	Pa	0.39006
6	Effective elastic modulus	C_2222	Pa	2.26175
7	Effective elastic modulus	C_2233	Pa	1.48163
8	Effective elastic modulus	C_2323	Pa	0.39006
9	Effective elastic modulus	C_3333	Pa	2.26175
10	Density	Density	$кг / м^3$	1450
11	Young's modulus	E	Pa	1.08889201676
12	Poisson ratio	ν	-	0.395799805264

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e :

37. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

38. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

39. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

40. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

41. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

42. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a_{ij}^e = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^e}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11} q, \ \sigma_{ij} = \alpha_{ij}^{(1)} q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22} q, \ \sigma_{ij} = \alpha_{ij}^{(2)} q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q, \ \sigma_{ij} = \alpha_{ij}^{(3)} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q, \ \sigma_{ij} = \alpha_{ij}^{(4)} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q, \ \sigma_{ij} = \alpha_{ij}^{(5)} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32}) q, \ \sigma_{ij} = \alpha_{ij}^{(6)} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)};$

2) $C_{ij22} = \alpha_{ij}^{(2)};$

3) $C_{ij33} = \alpha_{ij}^{(3)};$

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)}$;

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)}$;

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}$.

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	2.26175	2.27029	0.38
12	Effective elastic modulus	C_1122	Pa	1.48163	1.481390	0.02
13	Effective elastic modulus	C_1133	Pa	1.48163	1.481391	0.02
14	Effective elastic modulus	C_1212	Pa	0.39006	0.390033	0.01
15	Effective elastic modulus	C_1313	Pa	0.39006	0.390014	0.01
16	Effective elastic modulus	C_2222	Pa	2.26175	2.270277	0.38
17	Effective elastic modulus	C_2233	Pa	1.48163	1.48141	0.01
18	Effective elastic modulus	C_2323	Pa	0.39006	0.390035	0.01
19	Effective elastic modulus	C_3333	Pa	2.26175	2.270278	0.38
20	Density	Density	kg/m^3	1450	1449.7445	0.02
21	Young’s modulus	E	Pa	1.08889201676	1.1003858	1.06
22	Poisson ratio	ν	-	0.395799805264	0.39486241	0.24

Script CAE Fidesys:

```
reset
#{Pi = 3.1415926}
#{cube_size = 1.0}
#{ratio = 0.05}
#{E_m = 1.0}
#{nu_m = 0.4}
#{ro_m = 1000}
#{E_i = 10.0}
#{nu_i = 0.25}
#{ro_i = 10000}
#{sphere_rad = ( 0.75 * ratio * cube_size^3 / Pi )^0.33333}
create brick width {cube_size}
create sphere radius {sphere_rad}
subtract volume 2 from volume 1 keep
delete volume 1
imprint volume all
merge volume all
create material 1 name 'matr'
modify material 1 set property 'MODULUS' value {E_m}
modify material 1 set property 'POISSON' value {nu_m}
modify material 1 set property 'DENSITY' value {ro_m}
create material 2 name 'incl'
modify material 2 set property 'MODULUS' value {E_i}
modify material 2 set property 'POISSON' value {nu_i}
modify material 2 set property 'DENSITY' value {ro_i}
volume all size 0.1 #order,quality: 3,1
```

```
volume all scheme Tetmesh
mesh volume all
block 1 volume 2
block 2 volume 3
block 1 material 'incl'
block 2 material 'matr'
set node constraint on
block 1 element solid order 2
block 2 element solid order 2
#{G_m = E_m / (2.0 + 2.0*(nu_m))} # shear modules from Young's modulus and Poisson's ratio
#{G_i = E_i / (2.0 + 2.0*(nu_i))}
#{K_m = E_m / (3.0 - 6.0*(nu_m))} # bulk modules from Young's modulus and Poisson's ratio
#{K_i = E_i / (3.0 - 6.0*(nu_i))}
#{G_eff = G_m * ( 1.0 - 15.0*(1 - (nu_m))*(1 - G_i/G_m)*ratio / (7.0 - 5.0*(nu_m) + 2.0*(4.0 - 5.0*(nu_m))*G_i/G_m) )}
#{K_eff = K_m + (K_i - K_m)*ratio / ( 1.0 + (K_i - K_m)/(K_m + 1.33333*G_m) )}
#{E_eff = 9.0*K_eff*G_eff / (3.0*K_eff + G_eff)} # Young's modulus from shear modulus and bulk modulus
#{nu_eff = (3.0*K_eff - 2.0*G_eff) / (6.0*K_eff + 2.0*G_eff)} # Poisson's ratio from shear modulus and bulk modulus
analysis type effectiveprops elasticity dim3
periodicbc on
```

2.3. Test Case No.2.3

Problem Description

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

Input Values

Material Properties:

Steel:

- Young's modulus E = 200 кПа;
- Poisson ratio ν = 0.25;
- Density ρ = 7800 $кг/м^3$.

Rubber:

- Young's modulus E = 2 Па;
- Poisson ratio ν = 0.49;
- Density ρ = 1300 $кг/м^3$.

Geometric model:

- Solid cube with side 1.3m;
- In the middle (perpendicular to the Z-axis) there is a layer of steel with thickness 0.3.

Boundary conditions:

- Periodic.

Mesh

- Tetrahedron mesh order 2.

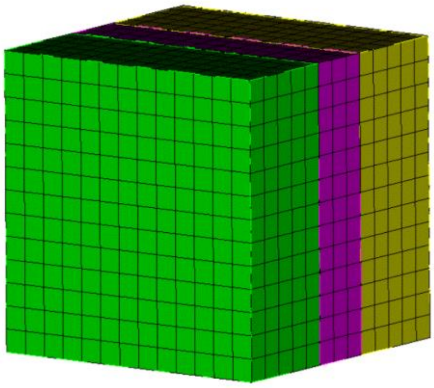


Fig 2.3 – Mesh 3D - Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Pa	49262.4200024
2	Effective elastic modulus	C_1122	Pa	12338.3105548
3	Effective elastic modulus	C_1133	Pa	36.3071714214
4	Effective elastic modulus	C_1212	Pa	18462.0547238
5	Effective elastic modulus	C_1313	Pa	0.872481025635
6	Effective elastic modulus	C_2222	Pa	49262.4200024
7	Effective elastic modulus	C_2233	Pa	36.3071714214
8	Effective elastic modulus	C_2323	Pa	0.872481025635
9	Effective elastic modulus	C_3333	Pa	44.4947405774
10	Density	Density	$кг / м^3$	2800
11	Young's modulus	E1=E2	Pa	46155.5
12	Young's modulus	E3	Pa	44.4519
13	Poisson ratio	ν12=ν21	-	0.25001
14	Poisson ratio	ν13=ν31	-	0.611983

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e:

1. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

2. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

3. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

4. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

5. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

6. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a^e_{ij} = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma^e_{ij}}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11} q, \ \sigma_{ij} = \alpha^{(1)}_{ij} q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22} q, \ \sigma_{ij} = \alpha^{(2)}_{ij} q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q, \ \sigma_{ij} = \alpha^{(3)}_{ij} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q, \ \sigma_{ij} = \alpha^{(4)}_{ij} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q, \ \sigma_{ij} = \alpha^{(5)}_{ij} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32}) q, \ \sigma_{ij} = \alpha^{(6)}_{ij} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha^{(1)}_{ij};$

2) $C_{ij22} = \alpha_{ij}^{(2)}$;

3) $C_{ij33} = \alpha_{ij}^{(3)}$;

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)}$;

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)}$;

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}$.

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	49262.4200024	49229.61829	0.07
2	Effective elastic modulus	C_1122	Pa	12338.3105548	12330.11285	0.07
3	Effective elastic modulus	C_1133	Pa	36.3071714214	36.3136	0.02
4	Effective elastic modulus	C_1212	Pa	18462.0547238	18462.05472	<<0.01
5	Effective elastic modulus	C_1313	Pa	0.872481025635	0.872481026	<<0.01
6	Effective elastic modulus	C_2222	Pa	49262.4200024	49229.61829	0.07
7	Effective elastic modulus	C_2233	Pa	36.3071714214	36.3136	0.02
8	Effective elastic modulus	C_2323	Pa	0.872481025635	0.872481	<<0.01
9	Effective elastic modulus	C_3333	Pa	44.4947405774	44.494741	<<0.01
10	Density	Density	$кг / м^3$	2800	2800	0
11	Young’s modulus	E1=E2	Pa	46155.5	46124.7424	0.07
12	Young’s modulus	E3	Pa	44.4519	44.451898	<<0.01
13	Poisson ratio	ν12=ν21	-	0.25001	0.250	<<0.01
14	Poisson ratio	ν13=ν23	-	0.611983	0.6121	0.02
15	Poisson ratio	ν31=ν32	-	0.000589395	0.0005899	0.08
16	Shear modulus	G12	Pa	18462.1	18462.055	<<0.01
17	Shear modulus	G13=G23	Pa	0.872481	0.8725	<<0.01

Script CAE Fidesys:

```
cubit.cmd("reset")
fidesys.cmd("set default element hex")
rub_thick = 1.0
steel_thick = 0.3
rub_number = 1
length = 1.3
width = 1.3
height = rub_number*(rub_thick + steel_thick)
```

```
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_E = 2.0e5
steel_nu = 0.25
steel_lambda = lambda_Calc_E_nu(steel_E, steel_nu)
steel_G = G_Calc_E_nu(steel_E, steel_nu)
steel_rho = 7800.0
 rub_E = 2.0
rub_nu = 0.49
rub_lambda =  lambda_Calc_E_nu(rub_E,rub_nu)
rub_G = G_Calc_E_nu(rub_E, rub_nu)
rub_rho = 1300.0
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" )
command1 = "block 2 volume"
for i in range(1, rub_number+2): command1 = command1 + " " + str(i)
cubit.cmd(command1)
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")
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 'DENSITY' value " + str(steel_rho))
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 'DENSITY' value " + str(rub_rho))
cubit.cmd("block 1 material 'steel'")
cubit.cmd("block 2 material 'rubber'")
cubit.cmd("block 1 2 element solid order 2")
cubit.cmd("volume all scheme Sweep")
cubit.cmd("volume all size " + str(mesh_size) )
cubit.cmd("mesh volume all")
cubit.cmd("analysis type effectiveprops elasticity dim3")
cubit.cmd("periodicbc on")
cubit.cmd("analysis type effectiveprops elasticity dim3")
cubit.cmd("periodicbc on")
```

2.4.Test Case No.2.4

Problem Description

Determination of effective mechanical characteristics for a layered composite containing layers of two materials, one of which is modeled by a shell.

Input Values

Material Properties:

Steel:

- Young's modulus E = 2.0e5 Pa;
- Poisson ratio ν = 0.25.
- Density ρ = 7800 $кг/м^3$.

Rubber:

- Young's modulus E = 2 Па;
- Poisson ratio ν = 0.49.
- Density ρ = 1300 kg/m^3.

Geometric model:

- Solid cube with side 1.3m;
- In the middle (perpendicular to the Z axis) there is a layer of steel 0.05 thick, modeled by shell elements

Boundary conditions:

- Periodic.

Mesh

- Tetrahedron mesh order 2.

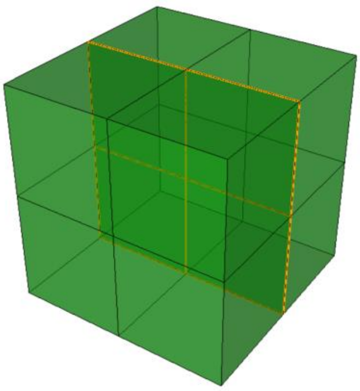


Fig 2.4 – Mesh 3D - shell (volumetric view)

Output Values

No	Value	Description	Unit	Target
1	Effective elastic modulus	C_1111	Па	8238.889
2	Effective elastic modulus	C_1122	Pa	2083.752
3	Effective elastic modulus	C_1133	Pa	33.342
4	Effective elastic modulus	C_1212	Pa	3077.568
5	Effective elastic modulus	C_1313	Pa	0.698
6	Effective elastic modulus	C_2222	Pa	8238.889
7	Effective elastic modulus	C_2233	Pa	33.342
8	Effective elastic modulus	C_2323	Pa	0.698
9	Effective elastic modulus	C_3333	Pa	35.597
10	Density	Density	kg/m^3	1600
11	Young’s modulus	E1=E2	Pa	7694.38
12	Young’s modulus	E3	Pa	35.3816
13	Poisson ratio	ν12=ν21	-	0.250074
14	Poisson ratio	ν13=ν23	-	0.70242
15	Poisson ratio	ν31=ν32	-	0.00322999
16	Shear modulus	G12	Pa	3077.57
17	Shear modulus	G13=G23	Pa	0.698

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e :

7. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

8. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

9. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

10. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

11. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

12. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a_{ij}^e = a_{ij} q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^e}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11} q, \ \sigma_{ij} = \alpha_{ij}^{(1)} q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22} q, \ \sigma_{ij} = \alpha_{ij}^{(2)} q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q, \ \sigma_{ij} = \alpha_{ij}^{(3)} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q, \ \sigma_{ij} = \alpha_{ij}^{(4)} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q, \ \sigma_{ij} = \alpha_{ij}^{(5)} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32})q, \ \sigma_{ij} = \alpha_{ij}^{(6)} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)}$;

2) $C_{ij22} = \alpha_{ij}^{(2)}$;

3) $C_{ij33} = \alpha_{ij}^{(3)}$;

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)}$;

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)}$;

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}$.

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Effective elastic modulus	C_1111	Pa	8238.889	8239.3564	3.0775942
22	Effective elastic modulus	C_1122	Pa	2083.752	2084.168	0.02
23	Effective elastic modulus	C_1133	Pa	33.342	32.885906	1.37
34	Effective elastic modulus	C_1212	Pa	3077.568	3077.5942	<<0.001

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
55	Effective elastic modulus	C_1313	Pa	0.698	0.67114094	3.85
66	Effective elastic modulus	C_2222	Pa	8238.889	8239.3564	0.01
77	Effective elastic modulus	C_2233	Pa	33.342	32.885906	1.37
88	Effective elastic modulus	C_2323	Pa	0.698	0.67114094	3.85
99	Effective elastic modulus	C_3333	Pa	35.597	34.228188	3.85
110	Density	Density	$кг/м^3$	1600	1600	0
111	Young's modulus	E1=E2	Pa	7694.38	7694.4592	<<0.001
112	Young's modulus	E3	Pa	35.3816	34.018670	3.85
113	Poisson ratio	$\nu 12 = \nu 21$	-	0.250074	0.25007696	<<0.001
114	Poisson ratio	$\nu 13 = \nu 23$	-	0.70242	0.72051429	2.58
115	Poisson ratio	$\nu 31 = \nu 32$	-	0.00322999	0.0031855309	1.38
116	Shear modulus	G12	Pa	3077.57	3077.5942	<<0.001
117	Shear modulus	G13=G23	Pa	0.698	0.67114094	3.85

Script CAE Fidesys:

```
fidesys.cmd("reset")
fidesys.cmd("set default element hex")
rub_thick = 1.25
steel_thick = 0.05
steel_E = 2.0e5
steel_nu = 0.25
steel_alpha = 1.3e-5
steel_lambda = 40.0
steel_rho = 7800.0
 rub_E = 2.0
rub_nu = 0.49
rub_alpha = 7.7e-5
rub_lambda = 1.0
rub_rho = 1300.0
mesh_size = 0.65
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)
# averaging over the volume
def aver(st, rub): return (steel_thick * st + rub_thick * rub) / (rub_thick + steel_thick)
fidesys.cmd("brick x " + str(rub_thick + steel_thick) )
fidesys.cmd("webcut volume all with plane zplane offset 0 merge ")
fidesys.cmd("create material 1")
fidesys.cmd("modify material 1 name 'steel'")
```

```
fidesys.cmd("modify material 1 set property 'MODULUS' value " + str(steel_E) )
fidesys.cmd("modify material 1 set property 'POISSON' value " + str(steel_nu) )
fidesys.cmd("modify material 1 set property 'ISO_THERMAL_EXPANSION' value " + str(steel_alpha) )
fidesys.cmd("modify material 1 set property 'ISO_CONDUCTIVITY' value " + str(steel_lambda) )
fidesys.cmd("modify material 1 set property 'DENSITY' value " + str(steel_rho) )
fidesys.cmd("create material 2 ")
fidesys.cmd("modify material 2 name 'rubber'")
fidesys.cmd("modify material 2 set property 'MODULUS' value " + str(rub_E) )
fidesys.cmd("modify material 2 set property 'POISSON' value " + str(rub_nu) )
fidesys.cmd("modify material 2 set property 'ISO_THERMAL_EXPANSION' value " + str(rub_alpha) )
fidesys.cmd("modify material 2 set property 'ISO_CONDUCTIVITY' value " + str(rub_lambda) )
fidesys.cmd("modify material 2 set property 'DENSITY' value " + str(rub_rho) )
fidesys.cmd("set duplicate block elements off")
fidesys.cmd("block 1 add surface 7")
fidesys.cmd("set duplicate block elements off")
fidesys.cmd("block 2 add volume 1 2")
fidesys.cmd("block 1 material 1")
fidesys.cmd("block 2 material 2")
fidesys.cmd("block 1 element shell order 1")
fidesys.cmd("block 2 element solid order 1")
fidesys.cmd("create shell properties 1")
fidesys.cmd("modify shell properties 1 thickness " + str(steel_thick))
fidesys.cmd("modify shell properties 1 eccentricity 0.5")
fidesys.cmd("block 1  shell properties 1")
fidesys.cmd("volume all scheme Sweep")
fidesys.cmd("volume all size " + str(mesh_size) )
fidesys.cmd("mesh volume all")
fidesys.cmd("analysis type effectiveprops heattrans elasticity dim3 preload off")
fidesys.cmd("periodicbc on")
fidesys.cmd("solver method direct use_uzawa no try_other off")
fidesys.cmd("output nodalforce off energy off record3d on log on vtu on material off")
```

2.5.Test Case No.2.5

Problem Description.

Determination of effective mechanical characteristics for a porous material of periodic structure, with spherical pores.

Input Values

Material:

- Young's modulus E = 1 Pa;
- Poisson ratio ν = 0.4;
- Density ρ = 1 $кг/м^3$.

Geometric model:

- Solid cube with side 1m;
- In the center there is a hole in the form of a ball with a radius 0.228542449528 .

Boundary conditions:

- Periodic.

Mesh

- Tetrahedron mesh order 2.

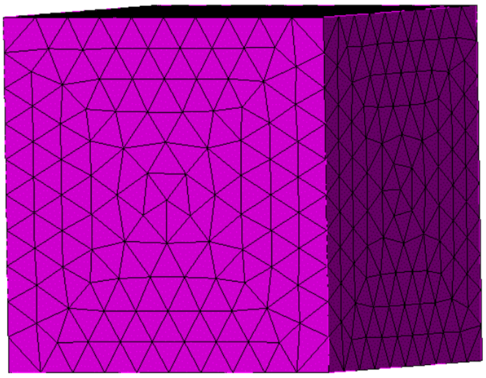


Fig 2.5 – Mesh 3D - Tetrahedron

Output Values

No	Value	Description	Unit	Target
1	Poisson ratio	ν	-	0.383928510975
2	Young's modulus	E	Pa	0.899553532133
3	Density	Density	$кг/м^3$	0.95

Calculation method used for the reference solution

Six deformation types are applied to a representative volume, each of which has its own type of effective strain tensor E^e :

1. $E_e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis X;

2. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – stretching/compression along the axis Y;

3. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix}$ - stretching/compression along the axis Z;

4. $E_e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ – in plane shear XY;

5. $E_e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix}$ – in plane shear XZ;

6. $E_e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix}$ – in plane shear YZ.

So, for each of the six boundary value problems, an effective strain tensor E^e was given and the effective stress tensor σ^e is obtained.

The linear dependence of σ^e on q is presented by the formula:

$$a_{ij}^e = a_{ij}q$$

Since the magnitude of deformations q and the corresponding tensor σ^e are known, the tensor coefficient of the dependence a_{ij} can be calculated simply:

$$a_{ij} = \frac{\sigma_{ij}^e}{q}$$

Since the effective properties are estimated in the form of the generalized Hooke's law

$$\sigma_{ij} = Cijkle_{kl},$$

considering the form E^e in each problem, the formulas for $Cijkl$ will look like this:

1) $E^e = \begin{pmatrix} q & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij11}q, \ \ \sigma_{ij} = \alpha_{ij}^{(1)}q;$

2) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & q & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij22}q, \ \ \sigma_{ij} = \alpha_{ij}^{(2)}q;$

3) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & q \end{pmatrix} \Rightarrow \sigma_{ij} = \sigma_{ij33} q,\ \sigma_{ij} = \alpha_{ij}^{(3)} q;$

4) $E^e = \begin{pmatrix} 0 & q & 0 \\ q & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij12} + \sigma_{ij21}) q,\ \sigma_{ij} = \alpha_{ij}^{(4)} q;$

5) $E^e = \begin{pmatrix} 0 & 0 & q \\ 0 & 0 & 0 \\ q & 0 & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij13} + \sigma_{ij31}) q,\ \sigma_{ij} = \alpha_{ij}^{(5)} q;$

6) $E^e = \begin{pmatrix} 0 & 0 & 0 \\ 0 & 0 & q \\ 0 & q & 0 \end{pmatrix} \Rightarrow \sigma_{ij} = (\sigma_{ij23} + \sigma_{ij32}) q,\ \sigma_{ij} = \alpha_{ij}^{(6)} q.$

Hence, the coefficients $Cijkl$ are calculated from the corresponding α_{ij}:

1) $C_{ij11} = \alpha_{ij}^{(1)};$

2) $C_{ij22} = \alpha_{ij}^{(2)};$

3) $C_{ij33} = \alpha_{ij}^{(3)};$

4) $C_{ij12} = C_{ij21} = \frac{1}{2}\alpha_{ij}^{(4)};$

5) $C_{ij13} = C_{ij31} = \frac{1}{2}\alpha_{ij}^{(5)};$

6) $C_{ij23} = C_{ij32} = \frac{1}{2}\alpha_{ij}^{(6)}.$

Reference:

[1] Бидерман В.Л., Гуслицер Р.Л., Захаров С.П., Ненахов Б.В., Селезнев И.И., Цукерберг С.М. Автомобильные шины (конструкция, расчет, испытание, эксплуатация). – Под общей редакцией Бидермана В.Л. – М.: Государственное научно-техническое изд-во химической литературы, 1963. – 384 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.

[4] Mohammad Ali Bagheri, Antonin Settari Effects of fractures on reservoir deformation and flow modeling // Canadian Geotechnical Journal, Vol 43, 2006. - P. 574-586.

[5] Hashin, Z, and Shtrikman, S, 1963, A variational approach to the elastic behavior of multiphase minerals. Journal of the Mechanics and Physics of Solids, 11 (2), 127-140.

[6] Kachanov M., Tsukrov I., Shafiro B. Effective modulus of solids with cavities of various shapes // Applied Mechanics Reviews, Vol. 47, No. 1, Part 2, 1994 - P. 151-174.

Result comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Poisson ratio	ν	-	0.383928510975	0.388215587	1.12
2	Young’s modulus	E	Pa	0.899553532133	0.916450489	1.88
3	Density	Density	$кг / м^3$	0.95	0.95342265	0.36

Script CAE Fidesys:

```
reset
set node constraint off
#{Pi = 3.1415926}
 #{cube_size = 1.0}
#{ratio = 0.05} # porosity
#{E_m = 1.0}
#{nu_m = 0.4}
##{E_i = 100.0}
##{nu_i = 0.25}
#{sphere_rad = ( 0.75 * ratio * cube_size^3 / Pi)^0.33333}
create brick width {cube_size}
create sphere radius {sphere_rad}
subtract volume 2 from volume 1
 create material 1 name 'matr'
modify material 1 set property 'MODULUS' value {E_m}
modify material 1 set property 'POISSON' value {nu_m}
modify material 1 set property 'DENSITY' value 1.0
volume all size 0.1 #order,quality: 3,1
volume all scheme Tetmesh
mesh volume all
block 1 volume 3
block 1 material 'matr'
block 1 element solid order 2
#{G_m = E_m / (2.0 + 2.0*nu_m)} # shear modulus from Young's modulus and Poisson's ratio
#{K_m = E_m / (3.0 - 6.0*nu_m)} # bulk modulus from Young's modulus and Poisson's ratio
#{G_eff = G_m * ( 1.0 - 15.0*(1 - nu_m)*ratio / (7.0 - 5.0*nu_m) )}
#{K_eff = K_m - K_m*ratio / ( 1.0 -  K_m/(K_m + 1.33333*G_m) )}
#{E_eff = 9.0*K_eff*G_eff / (3.0*K_eff + G_eff)} # Young's modulus from shear modulus and bulk modulus
#{nu_eff = (3.0*K_eff - 2.0*G_eff) / (6.0*K_eff + 2.0*G_eff)} # Poisson's ratio from shear modulus and bulk modulus
analysis type effectiveprops elasticity dim3
periodicbc on
```

2.6. Test case No.2.6

Problem Description

The dynamic problem of a square is considered, the sides of which move according to the given laws from time to time. In this setting, the square is divided into 4 parts along the diagonals - three contact pairs.

Input Values

Geometric model:

- Side of a square a=10 m.

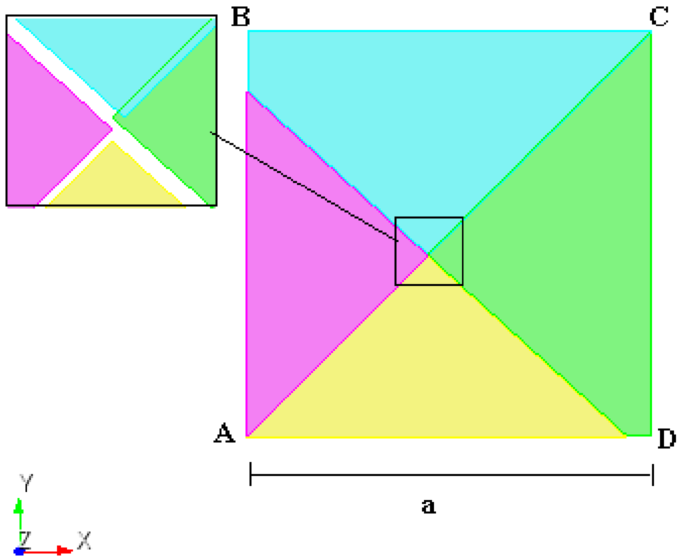


Fig 2.6 - Geometric model of test case

Border conditions:

- Symmetry condition: curve AB displacement $u_x = 0$;
- Symmetry condition: curve AD displacement $u_y = 0$;
- The displacement of the CD side along the X axis is $-4 \cdot e^{-70 \cdot t} \cdot sin(314 \cdot t + 3.14)$;
- The displacement of the BC side along the Y axis is $-8 \cdot e^{-70 \cdot t} \cdot sin(314 \cdot t + 3.14)$;
- Three contact pairs - (automatic selection of the main and secondary entity);
- Contact accuracy - 0.03 for all contact pairs.

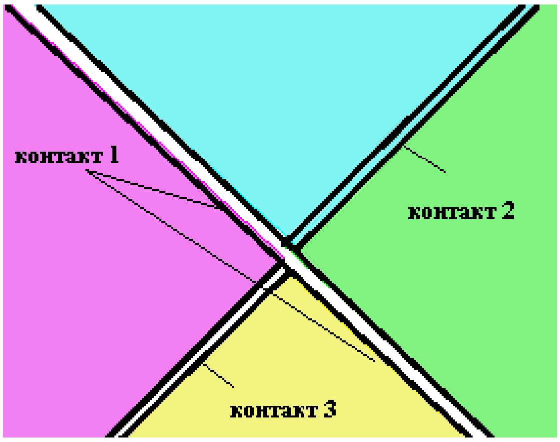


Fig 2.7 - Contact pairs

Material parameters:

- Elastic modulus $E = 2e+11$ Pa;
- Poisson's ratio $\nu = 0.3$;
- Density $\rho = 7900$ kg/m^3.

Mesh options:

- Mixed non-conformal mesh of the 2cd order;
- Finite elements: squares and triangles.

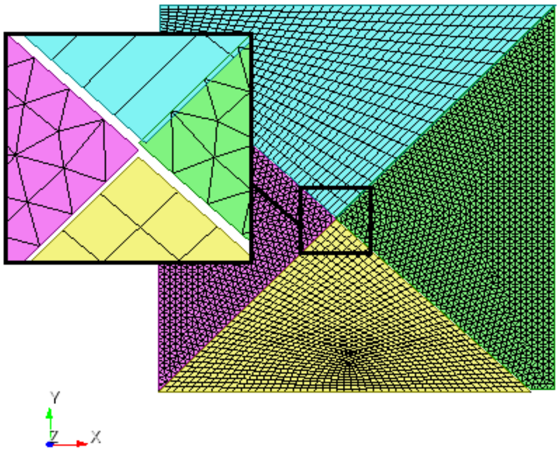


Fig 2.8 - Finite element mesh model

Calculation settings:

- Time analysis;
- Scheme: Implicit;
- Maximum time: 2e-2;

- Time step: 6e-5.

Calculation method used for the reference solution

The data obtained in the ANSYS package are used as a reference solution.

Result comparison

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Component X of the displacement vector at the point (0, 0, 0)	Displacement X	m	0.423	0.40984529	3.11
2	Component Yof the displacement vector at the point (0, 0, 0)	Displacement Y	m	0.845	0.81715182	3.3

Script Fidesys:

```
reset
create surface rectangle width 10.1 zplane
webcut body 1 with general plane location 0 -1 0 direction 1 1.1 0
webcut body 2 1  with plane xplane rotate -45 about z center 0 0 0
move Surface 5  y -0.025 include_merged
move Surface 4  y -0.025 include_merged
move Surface 4  y -0.025 include_merged
move Surface 7  x 0.025 include_merged
webcut body 1 2 3  with general plane location 0 -4.95 0 direction 0 1 0
delete surface 9 11
webcut body 3 4 with general plane location -4.98 0 0 direction 1 0 0
webcut body 1 4 with general plane location 5.02 0 0 direction 1 0 0
delete surface 13 15 17 18 20
surface 21 19 size auto factor 1
surface 14 10 size auto factor 1
surface 14 10 scheme trimesh
surface 21 19 scheme auto
mesh surface all
create material 1
modify material 1 name 'Material 1'
modify material 1 set property 'DENSITY' value 7900
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 2e11
block 1 surface all
block 1 material 'Material 1'
block 1 element plane order 2
create displacement on curve 43 37 dof 1 fix 0
create displacement on curve 25 38 54 dof 2 fix 0
create displacement on curve 52 58 dof 1 fix 1
create displacement on curve 59 dof 2 fix 1
set node constraint on
```

bcdep displacement 3 value '-4*exp(-70*t)*sin(314*t+3.14)'
bcdep displacement 4 value '-8*exp(-70*t)*sin(314*t+3.14)'
create contact master curve 27 slave curve 32 tolerance 0.08 type tied method auto
create contact master curve 39 slave curve 44 tolerance 0.08 type tied method auto
create contact master curve 60 slave curve 53 tolerance 0.08 type tied method auto
create contact master curve 21 slave curve 26 tolerance 0.08 type tied method auto
analysis type dynamic elasticity dim2
dynamic method full_solution scheme implicit maxtime 2e-2 timestep 6e-5 newmark_gamma 0.005
nonlinearopts maxiters 50 minloadsteps 1 maxloadsteps 10 tolerance 1e-3

output nodalforce off results everystep 1000

2.7. Test case No.2.7

Problem Description

Calculation of the stress-strain and thermal state of solid tires at axial loads from 0 to 100 tons. Consider a typical model of a solid tire (see Fig.2.9). We will assume that the mechanical properties of rubber and steel tires are described by Hooke's law.

Input Values

Geometric model:

- R1 = 0.1 m;
- R2 = 0.2 m;
- R3 = 0.21 m;
- R4 = 0.26 m;
- H1 = 0.07 m;
- H2 = 0.01 m.

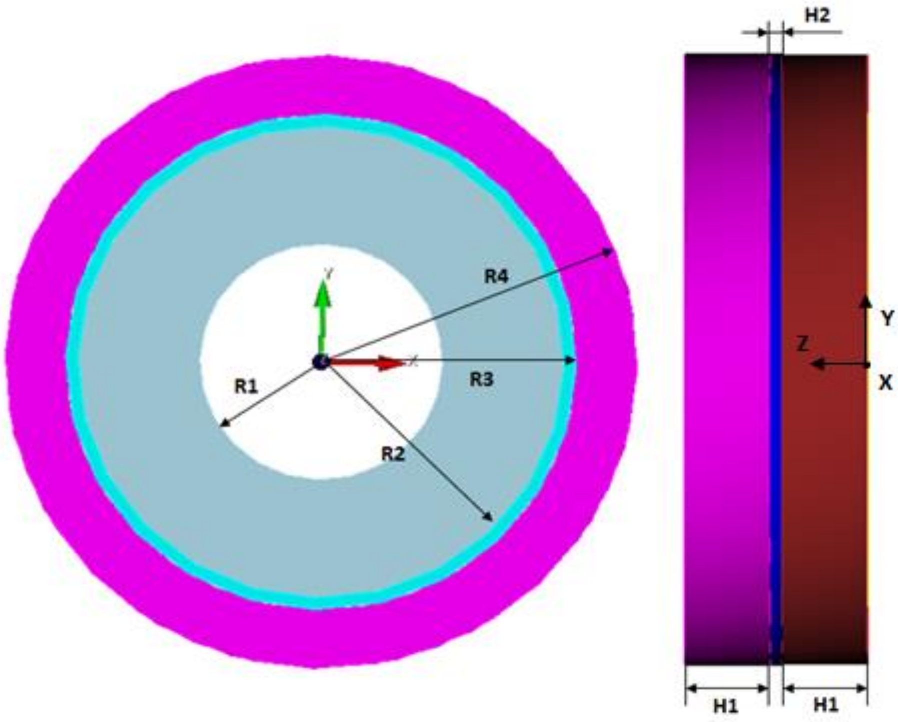


Fig 2.9 - Solid tire projections

Border conditions:

- Pinch condition: $\vec{u}\big|_{z=0} = \vec{0}$;
- Axial load, simulated by pressure at the left end of the tire (z = 2 H1 + H2): $p = 1000 \cdot t$;

- Axial acceleration on the inner surface $r = R1$ (cylindrical SC) a_z = 0.01 m/s^2;
- Temperature 25 °C throughout the solid tire.

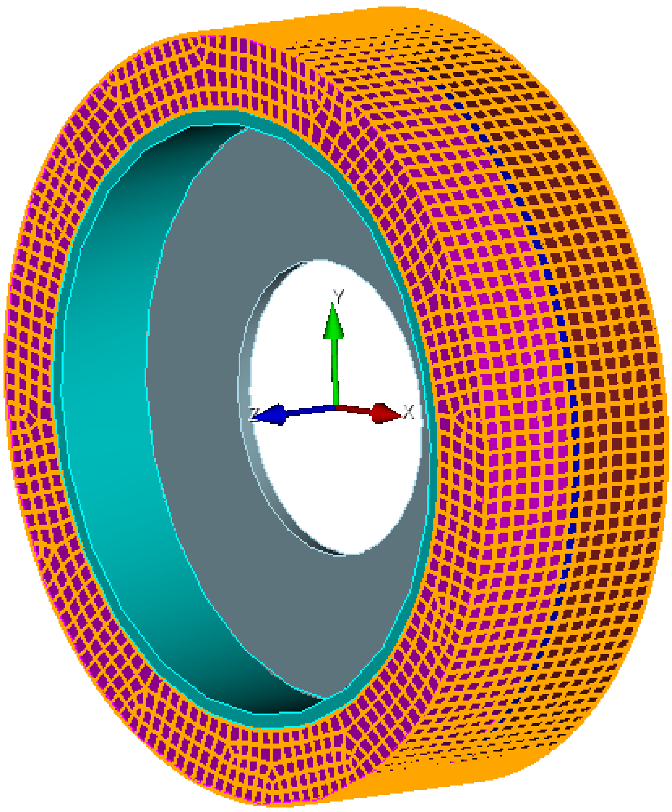


Fig 2.10 - 3D view of a massive tire

Material parameters:

- Steel:
  - Elastic modulus E = 210 GPa;
  - Poisson ratio $\nu = 0.3$;
  - Density ρ = 7800 kg/m^3;
  - Coefficient of thermal expansion α = 1.2e-5 °C^{-1};
  - Coefficient of thermal conductivity λ = 58 Vt/(m·K) ;
  - Coefficient of specific heat c = 462 Dg/(kg·K).
- Rubber**:**
  - Elastic modulus E = 5 MPa;
  - Poisson ratio ν = 0.45;
  - Density ρ = 1200 kg/m^3;
  - Coefficient of thermal expansion α = 7.7e-5 °C^{-1};
  - Coefficient of thermal conductivity λ = 0.1 Vt/(m·K);
  - Coefficient of specific heat c = 1420 Dg/(kg·K).

Mesh:

- Hexahedral mesh.

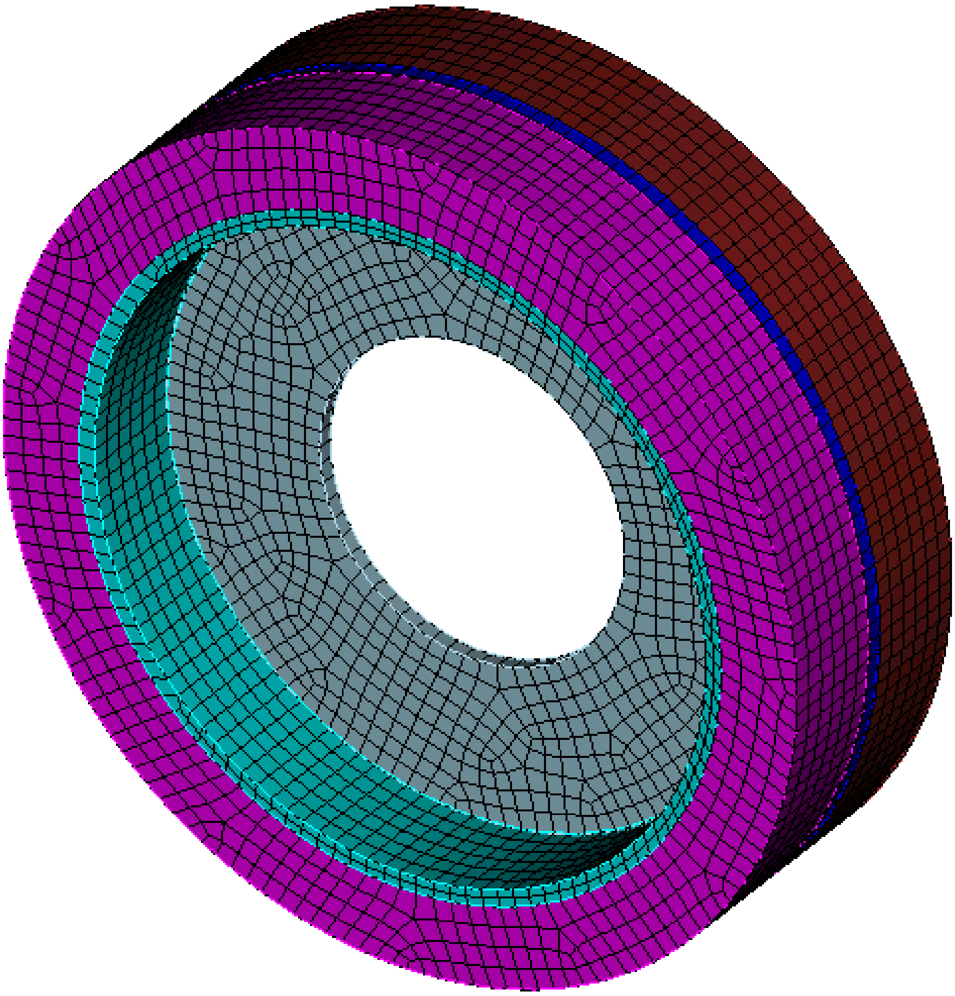


Fig 2.11 - Finite element mesh model

Calculation settings:

- Time analysis;
- Scheme: Implicit;
- Elasticity;
- Thermal conductivity;
- Maximum time: 1000;
- Number of steps: 10.

Calculation method used for the reference solution

The ANSYS solution acts as a reference.

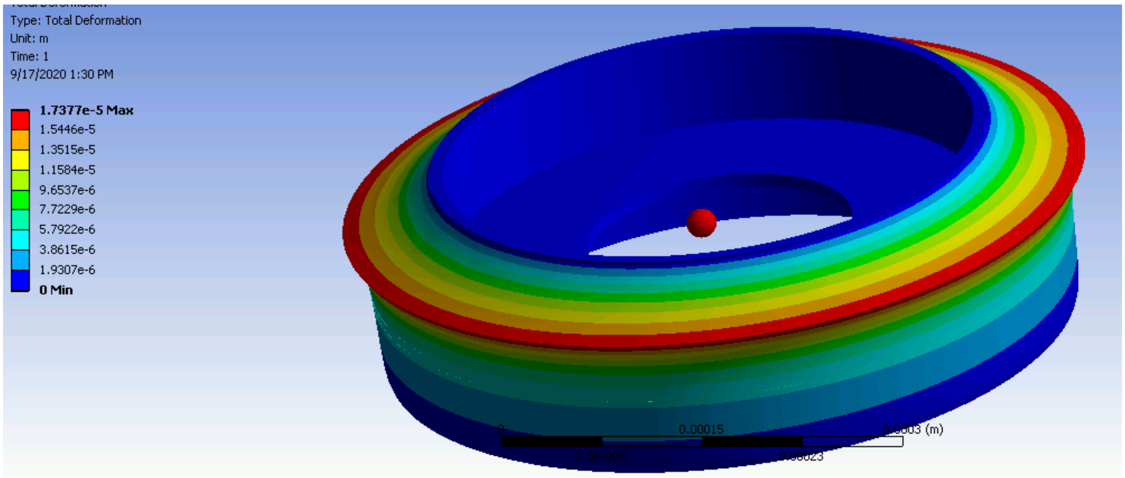


Fig 2.12 - Solving a problem in ANSYS

Result comparison

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Displacement vector components u_z at the point(0.26; 0; 0.15)	Displacement Z	m	0.0171	0.016878524	1.3

Script Fidesys:

```
reset
#{tireOuterHeight = 0.15}
#{tireThinHeight = 0.01}
#{wideSteelInnerR = 0.2}
#{thinSteelInnerR = 0.1}
#{wideSteelH = 0.01}
#{wideRubberH = 0.05}
set node constraint off
create Cylinder height {tireOuterHeight} radius {wideSteelInnerR}
create Cylinder height {tireOuterHeight} radius {wideSteelInnerR + wideSteelH}
create Cylinder height {tireOuterHeight} radius {wideSteelInnerR + wideSteelH + wideRubberH}
subtract body 1 from body 2 keep
subtract body 2 from body 3
delete volume 1
webcut body all with plane zplane offset {tireThinHeight/2}
webcut body all with plane zplane offset {-tireThinHeight/2}
create Cylinder height {tireThinHeight} radius {wideSteelInnerR}
create Cylinder height {tireThinHeight} radius {thinSteelInnerR}
subtract body 10 from body 9
merge all
```

```
create material 1
modify material 1 name 'steel'
modify material 1 set property 'MODULUS' value 2.1e+11
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 7800
modify material 1 set property 'SPECIFIC_HEAT' value 462
modify material 1 set property 'ISO_CONDUCTIVITY' value 58
modify material 1 set property 'ISO_THERMAL_EXPANSION' value 1.2e-05
create material 2
modify material 2 name 'rubber'
modify material 2 set property 'MODULUS' value 5e6
modify material 2 set property 'POISSON' value 0.45
modify material 2 set property 'DENSITY' value 1200
modify material 2 set property 'SPECIFIC_HEAT' value 1420
modify material 2 set property 'ISO_CONDUCTIVITY' value 0.1
modify material 2 set property 'ISO_THERMAL_EXPANSION' value 7.7e-05
block 1 volume 3 5 7
block 2 volume 4 6 8 9
block 1 material 2
block 2 material 1
block 1 element solid order 2
block 2 element solid order 2
volume all size auto factor 4
mesh volume all
create temperature on volume all value 25
create displacement on surface 12 15 dof all fix 0
create pressure on surface 13 16 magnitude 0
bcdep pressure 1 value '(1e3)*t'
#create acceleration on surface 47 dof 3 fix 0.01
analysis type dynamic heattrans elasticity dim3
dynamic method full_solution scheme implicit maxtime 1000 steps 10
nonlinearopts maxiters 50 minloadsteps 1 maxloadsteps 10 tolerance 1e-3 targetiter 5
```

2.8.Test case No.2.8

Problem Description

The transient process of loading a bar structure with a concentrated force is considered. The task checks the maximum total displacements at the moment of 1 sec, as well as the equality of the maximum total displacements to zero at the moment of 5 sec.

Input Values

Geometric model:

- Truss geometry is built in a third-party CAD package and imported as a file with the stp extension (Truss.stp).

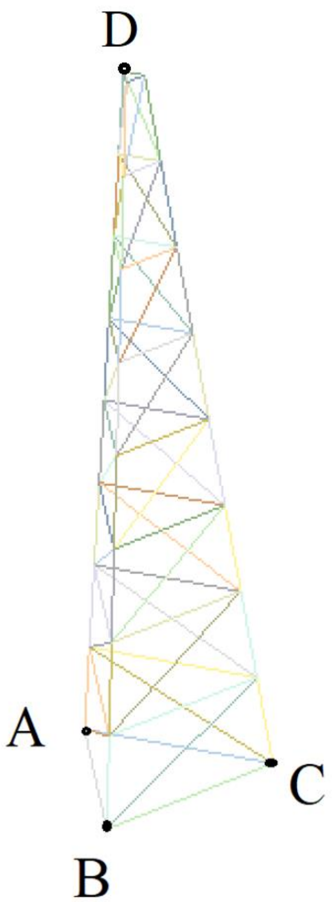


Fig 2.13 - Geometric model

Border conditions

- Fastening on all movements at points A, B, C.
- At point D, a point force is applied, depending on time according to the law in a tabular form (table 1)

Table 1 Setting the dependence on time for force

Time	Target force, N
0	0
1	10^5
2	0
5	0

Material parameters:

- Isotropic
- Elastic modulus E = 200 GPa;
- Density ρ = 7800 kg/m^3;
- Poisson ratio ν = 0.3.

Mesh:

- Beam finite elements of the first order;
- Section of beam elements: hollow tube, outer radius 100mm, inner radius 90mm.

Calculation settings:

- Dynamic calculation;
- Implicit scheme;
- Newton-Raphson method;
- At the first stage, from 0 sec to 2 sec, a step of 0.01 sec was used;
- At the second stage. From 2 sec to 5 sec, a step of 0.1 sec was used;
- Maximum time - 5 s;
- Maximum number of steps 230;
- Output of every 10th step to .vtu file.

Calculation method used for the reference solution

The problem has a numerical solution obtained in the ANSYS package.

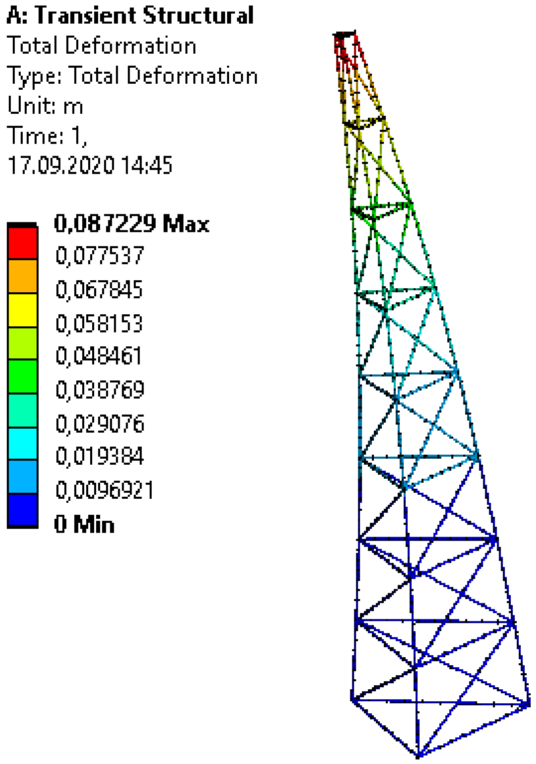


Fig 2.14 - Total displacements at time t = 1 s, m

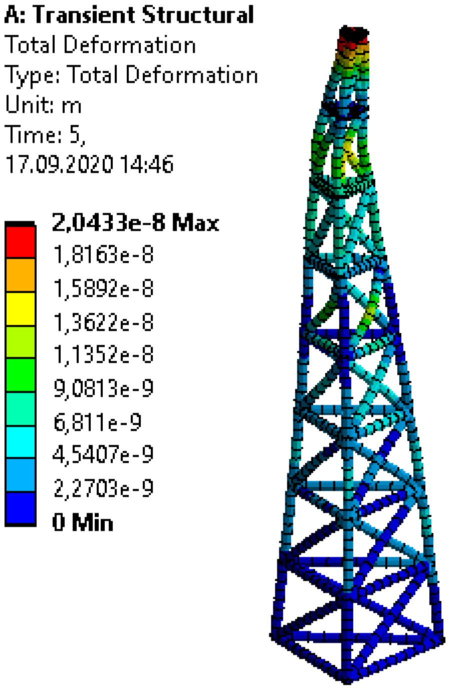


Fig 2.15 - Total displacements at time t = 5 s, m

Result comparison

Below are the values for the displacements at the point (6.06032, 4.81675, 49.3827) at the time t = 1 s.

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Component X of the displacement vector at the mesh nodes	Displacement X	m	0	5.4466067e-05	0.01
2	Component Y of the displacement vector at the mesh nodes	Displacement Y	m	8.72e-2	8.7121458e-02	0.09
3	Component Z of the displacement vector at the mesh nodes	Displacement Z	m	-3.48e-3	-3.4934805e-03	0.39

Below are the values for the displacements at the point (6.06032, 4.81675, 49.3827) at the last moment of time t = 5 s.

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Component X of the displacement vector at the mesh nodes	Displacement X	м	0	0	0
2	Component Y of the displacement vector at the mesh nodes	Displacement Y	м	0	0	0
3	Component Z of the displacement vector at the mesh nodes	Displacement Z	м	0	0	0

CAE Fidesys script:

```
reset
import step "C:\Users\ Truss.stp" heal
merge vertex all
curve all interval 5
curve all scheme equal
mesh curve all
create material 1 from 'Steel'
modify material 1 set property 'DENSITY' value 7800
set duplicate block elements off
block 1 add curve all
block 1 material 1
create beam properties 1
modify beam properties 1 type 'Circle With Offset Hole'
modify beam properties 1 angle 0.0
modify beam properties 1 ey 0.0
modify beam properties 1 ez 0.0
modify beam properties 1 geom_D1 200e-3
```

```
modify beam properties 1 geom_D2 180e-3
modify beam properties 1 geom_e 0
modify beam properties 1 mesh_quality 5
modify beam properties 1 warping_dof off
block 1 element beam order 1
block 1 beam properties 1
create displacement  on vertex 4 2 1  dof all fix
create force  on vertex 54  force value {1e5} direction 0 1 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 cell 2 1 value 1
modify table 1 cell 3 1 value 2
modify table 1 cell 4 1 value 5
modify table 1 cell 2 2 value 1e5
bcdep force 1 table 1
analysis type dynamic elasticity dim3 preload on
dynamic method full_solution scheme implicit maxtime 5 steps 500 newmark_gamma 0.005
damping mass_matrix 0 stiffness_matrix 0.05
solver method direct use_uzawa auto try_other off
output nodalforce off energy off record3d off log on vtu on material off results everystep 10
```

2.9.Test case No.2.9

Problem Description

The Lamb problem is considered, which is a dynamic action model of a concentrated load on the elastic half-plane boundary. Applied load depends on time according to Berlage's law. The model consists of two layers with different materials.

Input Values

Geometric model:

- See Fig 2.16.

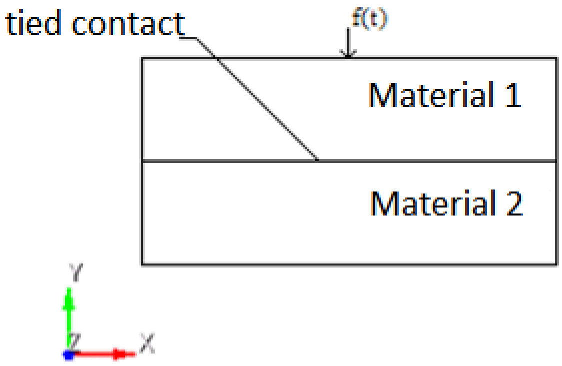


Fig 2.16 - Geometric model

Border conditions:

- Point force is given using the Berlage formula: $f(t) = A\cdot\frac{\omega_1^2 e^{-\omega_1 t}}{4}(\sin(\omega_0 t)(-\frac{t^2}{\omega_1}+\frac{t}{\omega_1^2}+\frac{1}{\omega_1^3})-\cos(\omega_0 t)\sqrt{3}(\frac{t^2}{\omega_1}+\frac{t}{\omega_1^2})), \omega_1 = \frac{\omega_0}{\sqrt{3}}, \omega_0 = 2\pi\omega$

where A - amplitude, ω - frequency, t - time;

Non-reflective conditions applied to bottom and side faces.

Material parameters of the top layer:

- Young's modulus $E = 2e+08$;
- Poisson ratio $\nu = 0.3$;
- Density $\rho = 1900$;
- Cohesion $K = 29000$;
- Angle of internal friction $\alpha = 20$;

- Angle of dilatancy $\beta = 10$.

Material parameters of the bottom layer:

- Young's modulus $E = 3e + 08$;
- Poisson ratio $\nu = 0.3$;
- Density $\rho = 1900$;
- Cohesion $K = 29000$;
- Angle of internal friction $\alpha = 20$;
- Angle of dilatancy $\beta = 10$.

Finite element mesh generation:

- Spectral elements of the 3rd order.

The mesh should be of flat quadrangles, the height of the element is calculated in accordance with the wavelength (see subclause 1.6).

Contact settings:

- Type: Tied;
- Tolerance 0.0005;
- Method: MPC.

Calculation settings:

- Dynamic calculation;
- Maximum time – 5 s;
- Maximum number of steps 2025;
- Output of every 135 steps to a .vtu file.

Calcuation method used for the reference solution

The values are compared to the full model, without using the associated contact (1.16)

Result comparison

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

No.	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Displacement vector components at mesh nodes at step 6	Displacement X	m	-0.00110025	-0.0011002537	<<0.01
2	Displacement vector components at mesh nodes at step 6	Displacement Y	m	0.000517095	0.00051707876	<<0.01
3	Displacement vector components at mesh nodes at step 8	Displacement X	m	-4.78016e-05	-4.7799808e-05	<<0.01
4	Displacement vector components at mesh nodes at step 8	Displacement Y	m	-0.000445372	-0.00044537138	<<0.01

Script CAE Fidesys:

```
reset
set default element hexzplane
webcut body 1 with plane xplane offset 0
webcut body 1 with plane yplane offset 0
delete Surface 3
rotate Surface 4 5 angle -90 about Z include_merged
webcut body 3 1 with plane yplane offset -250
merge curve 18 25
merge curve 22 27
surface all size 7
mesh surface all
create material 1
modify material 1 name 'Material1'
modify material 1 set property 'MODULUS' value 2e+08
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 1900
modify material 1 set property 'COHESION' value 29000
modify material 1 set property 'INT_FRICTION_ANGLE' value 20
modify material 1 set property 'DILATANCY_ANGLE' value 10
create material 2
modify material 2 name 'Material2'
modify material 2 set property 'MODULUS' value 2e+08
modify material 2 set property 'POISSON' value 0.3
modify material 2 set property 'DENSITY' value 1900
modify material 2 set property 'COHESION' value 29000
modify material 2 set property 'INT_FRICTION_ANGLE' value 20
modify material 2 set property 'DILATANCY_ANGLE' value 10
set duplicate block elements off
block 1 add surface 9 7
set duplicate block elements off
block 2 add surface 8 6
block 1 material 1
block 2 material 2
block 1 2 element plane order 3
create absorption on curve 28 24 13 15 19 21
create force on vertex 10 force value 1 direction 0 -1 0
bcdep force 1 value 'berlage(1e+8, 10, time)'
create receiver on curve 16 displacement 1 1 1
#create receiver on curve 16 velocity 1 1 1
#create receiver on curve 16 principalstress 1 1 1
```

```
#create receiver on curve 16 pressure
create contact master curve 17 23 slave curve 20 26 tolerance 0.0005 type tied method auto
analysis type dynamic elasticity dim2 planestrain preload off
dynamic method full_solution scheme explicit maxtime 3 maxsteps 2025
output nodalforce off energy off record3d on log on vtu on material off results everystep 135
```

2.10. Test case No.2.10

Problem Description

In the problem, a suspended beam with a square section is considered, fixed in the upper section. An axial tensile force is applied to the free end of the beam.

Input values

Geometric model:

- Beam height $L = 10$ in;
- Beam width $d = 2$ in;
- Geometry is imported from 01_model.stp file.

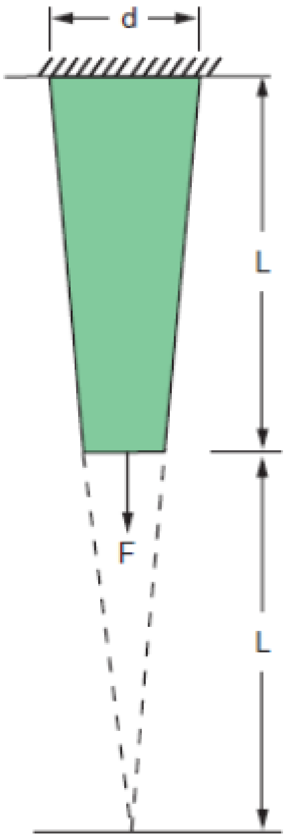


Fig 2.17 - Geometric model of the problem

Border conditions:

- Zero displacements along all axes on the Y = 0 plane;
- Axial force F = 5000, 7500, 10,000 lb, applied to all nodes of the Y = L plane.
- Number of loading steps: 3

Material parameters:

- Elastic modulus $E = 10.4e + 6$ psi;
- Poisson ratio $\nu = 0.3$.

Mesh:

- Second-order conformal mesh;
- Finite elements: hexahedrons.

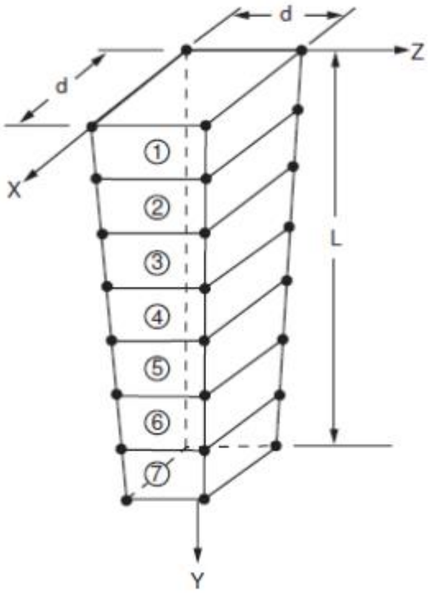


Fig 2.18 - Finite element mesh model

Calcuation method used for the reference solution

The ANSYS solution VM37 problem [1] acts as a reference.

Reference:

[1] Verification Manual for the Mechanical APDL Application, SAS IP, Inc 2009

Result comparison

No.	Loading steps	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Step 3	Stress tensor component σ_{yy} at point (1, L/2, 1)	Stress YY	psi	4444	4443.109	0.02
2	Step 3	Step number	dimensionless	-	3	3	-

CAE Fidesys script:

```
reset
import step "01_model.stp”
```

```
#import step "D:/Комплект численных решений/CAD/01_model.stp" heal
move Volume 1 x 1 y 0 z 1 include_merged
volume 1 size auto factor 10
mesh volume 1
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'MODULUS' value 1.04e+07
modify material 1 set property 'POISSON' value 0.3
set duplicate block elements off
block 1 add volume 1
block 1 material 1 cs 1 element solid order 2
create displacement on surface 4 dof all fix
create pressure  on surface 2  magnitude -10000
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 2 1 value 2
modify table 1 cell 3 1 value 3
modify table 1 cell 1 2 value -5000
modify table 1 cell 2 2 value -7500
modify table 1 cell 3 2 value -10000
bcdep pressure 1 table 1
analysis type static elasticity dim3
static steps 3
```

2.11. Test Case No2.11

Problem Description

The problem of testing the ability of contact algorithms to transmit uniform pressure using a non-conformal irregular mesh is considered.

Input Values

Geometric model:

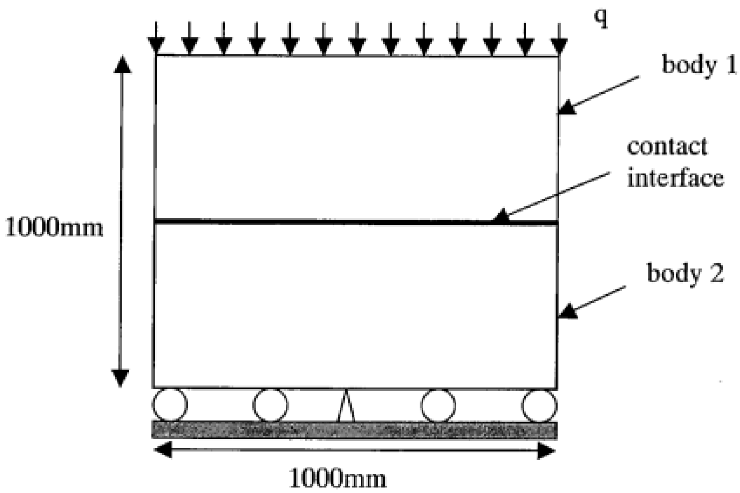


Fig 2.19 - Geometric model

Boundary conditions:

- The bottom side of the foundation is fixed in vertical movement, the center of this side is fixed in all directions;
- To improve the convergence of the problem, added pinning of vertex 2 along the x axis;
- Pressure q=40000 H/m^2.

Material Properties:

- Isotropic;
- Young's modulus $E = 100$ MPa;
- Poisson ratio $\nu = 0.3$.

Mesh:

- Non-conformal irregular mesh, first-order elements.

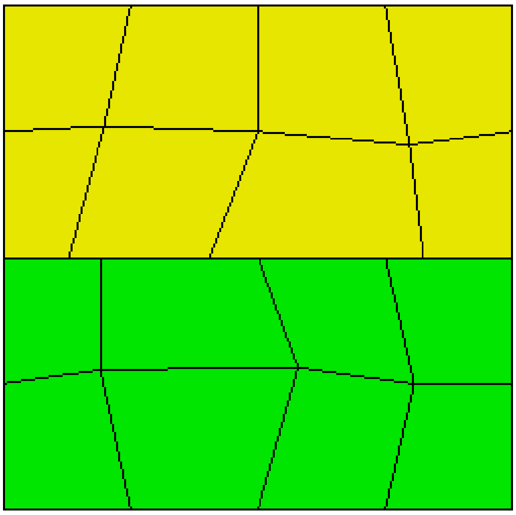


Fig 2.20 - Non-conformal irregular mesh

Contact Settings:

- Type: general;
- Tolerance: 0.0005;
- Method: auto.

Calculation Settings:

- Static;
- Elasticity.

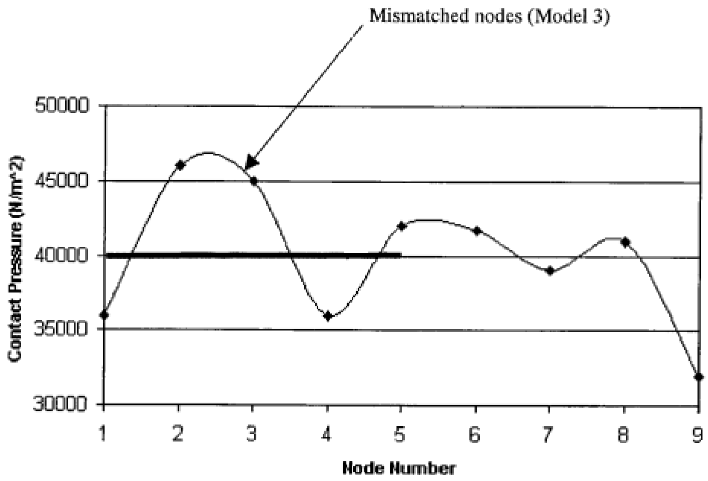
Output Values

The values for the voltage in contact at points with coordinates (-0.5,0.001,0), (0,0.001,0) and (0.5,0.001,0) are presented below.

No	Value	Description	Unit	Target
1	Stress tensor components in the contact zone at a point (-0.5,0.001,0)	Contact Stress N	Pa	40000
2	Stress tensor components in the contact zone at a point (0,0.001,0)	Contact Stress N	Pa	40000
3	Stress tensor components in the contact zone at a point (0.5,0.001,0)	Contact Stress N	Pa	40000

Calculation method used for the reference solution

The problem has a numerical solution [1]. Expected results:



Reference:

[1] NAFEMS R0081 - Benchmark Tests for Finite Element Modelling of Contact, Gapping and Sliding

Result comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error, %
1	Stress tensor components in the contact zone at a point (-0.5,0.001,0)	Contact Stress N	Pa	40000	39839.9999	0.4
2	Stress tensor components in the contact zone at a point (0,0.001,0)	Contact Stress N	Pa	40000	39836.085	0.41
3	Stress tensor components in the contact zone at a point (0.5,0.001,0)	Contact Stress N	Pa	40000	39840	0.4

CAE Fidesys Script:

```
reset
set default element hex
create surface rectangle width 1 height 1 zplane
webcut body 1 with plane yplane offset 0
partition create curve 3 position 0 -50 0
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'MODULUS' value 1e+8
modify material 1 set property 'POISSON' value 0.3
curve 1 interval 4
curve 1 scheme equal
mesh curve 1
```

```
curve 9 10  interval 2
curve 9 10  scheme equal
mesh curve 9 10
mesh surface 3
node 12  move X -0.120828 Y 0.000000 Z 0.000000
node 11  move X -0.094434 Y 0.000000 Z 0.000000
node 10  move X 0.074762 Y 0.000000 Z 0.000000
node 13  move X -0.058639 Y 0.049426 Z -0.000000
node 15  move X 0.047123 Y -0.026582 Z -0.000000
node 13  move X 0.006909 Y -0.039155 Z -0.000000
curve 7 6  interval 2
curve 7 6  scheme equal
mesh curve 7 6
curve 11 3  interval 2
curve 11 3  scheme equal
mesh curve 11 3
surface 2  size auto factor 10
mesh surface 2
node 27  move X -0.058639 Y 0.000000 Z 0.000000
node 30  move X -0.058639 Y 0.026582 Z -0.000000
node 29  move X 0.078311 Y 0.031188 Z -0.000000
node 28  move X 0.056336 Y -0.001057 Z -0.000000
block 1 add surface 2
block 2 add surface 3
block all material 'mat1'
block 1 element plane order 1
block 2 element plane order 1
create displacement on vertex 9 dof all fix
create displacement on curve 3 11 dof 2 fix
create displacement  on vertex 2  dof 1 fix
create pressure  on curve 1  magnitude 40000
create contact master curve 5 slave curve 8 type general friction 0.0 ignore_overlap off offset 0.0 tolerance 0.0005 method auto
analysis type static elasticity dim2 planestrain
```

2.12. Test Case No2.12

Problem Description

The problem of testing the ability of contact algorithms to transfer total displacements using a non-conformal irregular mesh with a rigid contact is considered.

Input Values

Geometric model:

- 02_model.stp.

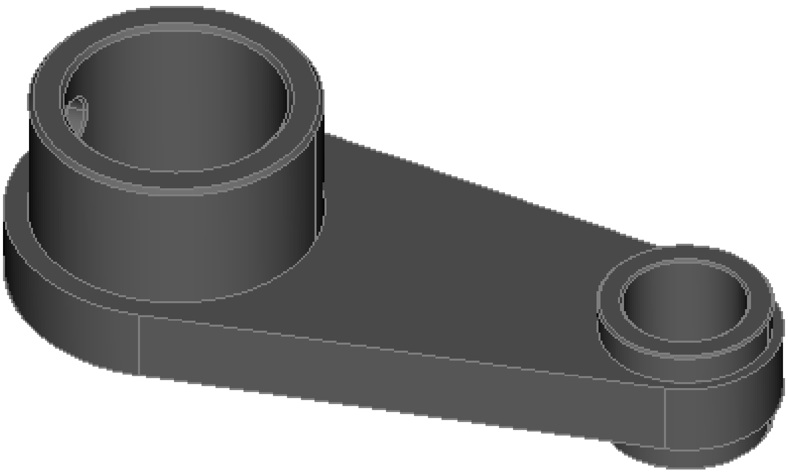


Fig 2.21 - Geometric model

Material Properties:

- Isotropic;
- Young's modulus E = 2e11 Пa;
- Poisson ratio ν = 0.3;
- Density ρ = 7850 kg/m^3.

Mesh:

- Finite-elements mesh (order 2).

Boundary conditions:

- The inner surface of the larger cylinder is rigidly fixed;
- Pressure p=1e5 МПa acts on the upper surface of the small cylinder;

- Tolerance for tied contact settings is 0.25.

Before starting the calculation, the model should be scaled by 0.001 for correct results.

Calculation Settings:

- Static;
- Elasticity.

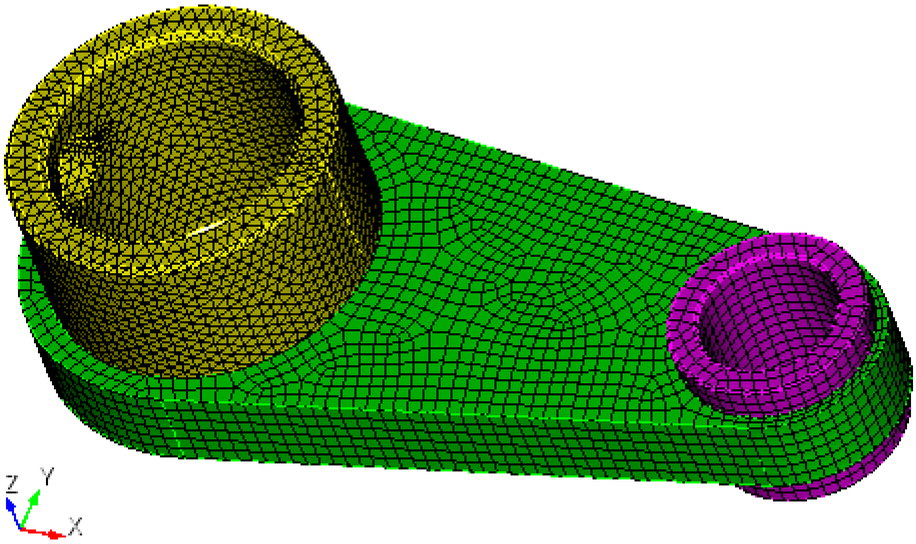


Fig 2.22 - Finite-element mesh

Output Values

No	Value	Description	Unit	Target
1	Maximum value of total displacements on a mesh with element size 0.0025	Displacement sum	m	3.2011e-6

Calculation method used for the reference solution

The ANSYS solution acts as a reference. For the correctness of the comparison, a study was carried out for mesh convergence (Figures 2.24-2.26).

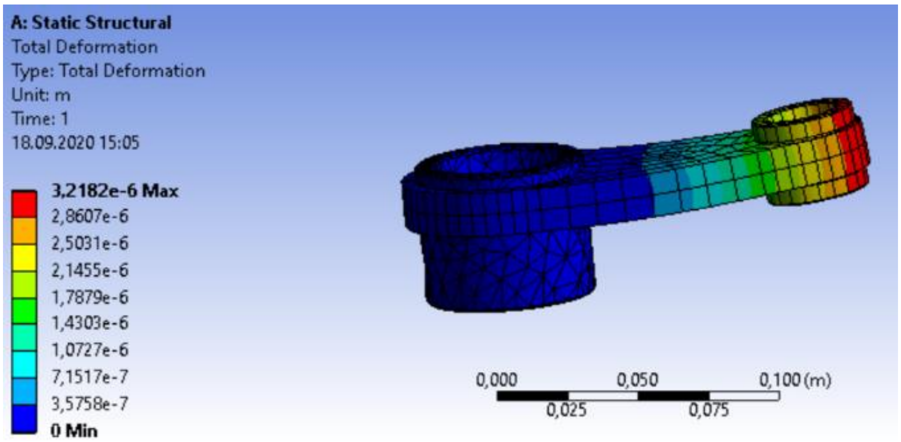


Fig 2.24 - Values of total displacements with an element size = 0.01

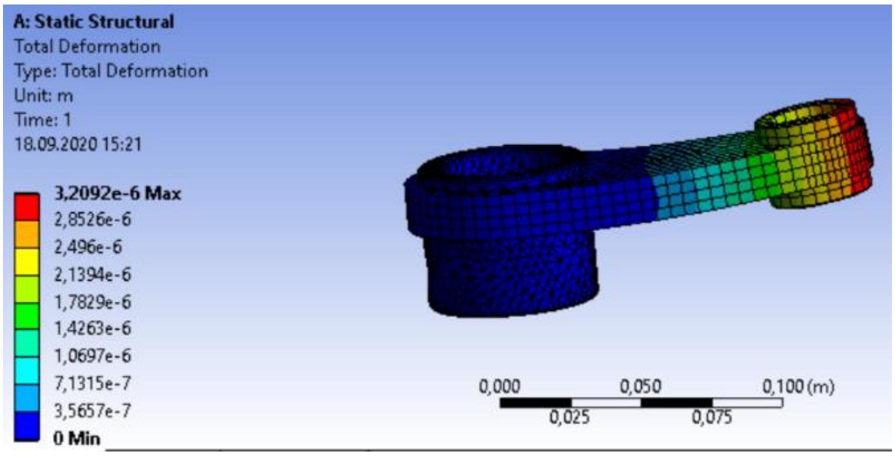


Fig 2.25 - Values of total displacements with an element size = 0.005

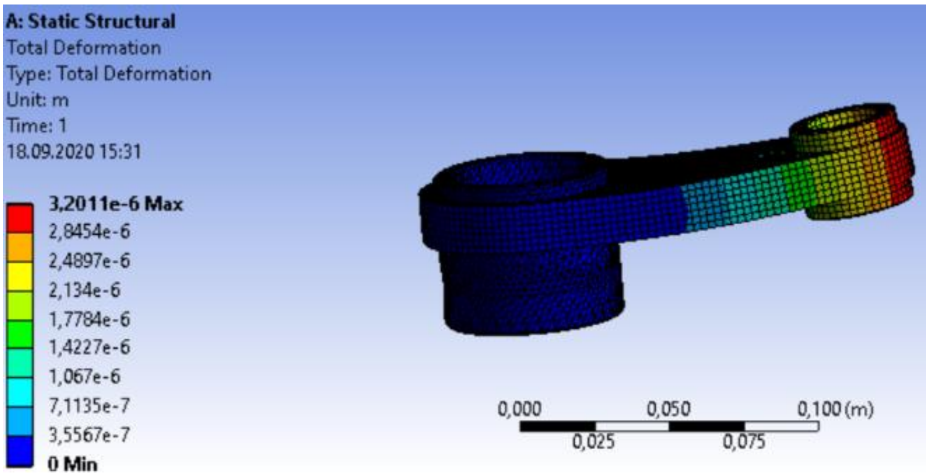


Fig 2.26 - Values of total displacements with an element size = 0.0025

Result comparison

No	Value	Description	Unit	Target	CAE Fidesys	Error,%
1	Maximum value of total displacements on a mesh with element size = 0.0025	Displacement sum	m	3.2011e-6	3. 1847985e-6	0.51

CAE Fidesys Script:

```
reset
import step "C: /02_model.step" heal
Volume all scale 0.001
volume all size 0.0025
mesh volume 1
volume 3 redistribute nodes off
volume 3 scheme Sweep source surface 24 target surface 23 sweep transform least squares
volume 3 autosmooth target on fixed imprints off smart smooth off
volume 3 redistribute nodes off
volume 3 scheme Sweep source surface 24 target surface 23 sweep transform least squares
volume 3 autosmooth target on fixed imprints off smart smooth off
mesh volume 3
volume 2 scheme tetmesh proximity layers off geometry approximation angle 15
volume 2 tetmesh growth_factor 1
Trimesher surface gradation 1.3
Trimesher volume gradation 1.3
Trimesher geometry sizing on
mesh volume 2
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
modify material 1 set property 'DENSITY' value 7850
set duplicate block elements off
block 1 add volume all
block 1 material 1 cs 1 element solid order 2
create displacement on surface 19 dof all fix 0
create pressure on surface 24 magnitude 1e5
create contact autoselect tolerance 0.00025 type tied method auto
analysis type static elasticity dim3
```

2.13. Test case No2.13

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

No	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:

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

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

No	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.

2.14. Test case No2.14

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 α_x = 1 K^{-1};
- Thermal expansion coefficient α_y = 2 K^{-1};
- Thermal expansion coefficient α_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

No	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

No	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.

2.15. Test case No2.15

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

No	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

No	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.

2.16. Test case No2.16

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.

<u>Mesh:</u>

- First order tetrahedrons.

Target results

No	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

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Effective thermal conductivity coefficient	λ_11	$\frac{W}{m*K}$	2.8	2.773E+00	0.95
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m*K}$	2.28571	2.283E+00	0.12
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m*K}$	2.28571	2.292E+00	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 2
analysis type effectiveprops heattrans dim3
periodicbc on
```

Reference:

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

2.17. Test case No2.17

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

No	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

No	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	0.08
2	Effective thermal conductivity coefficient	λ_22	$\frac{W}{m * K}$	$8.58878 * 10^{-5}$	8.484E-05	1.22
3	Effective thermal conductivity coefficient	λ_33	$\frac{W}{m * K}$	$8.58878 * 10^{-5}$	8.484E-05	1.22

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.

2.18. Test case No2.18

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

No	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

No	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.

2.19. Test case No2.19

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

No	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

No	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.

2.20. Test caseNo2.20

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

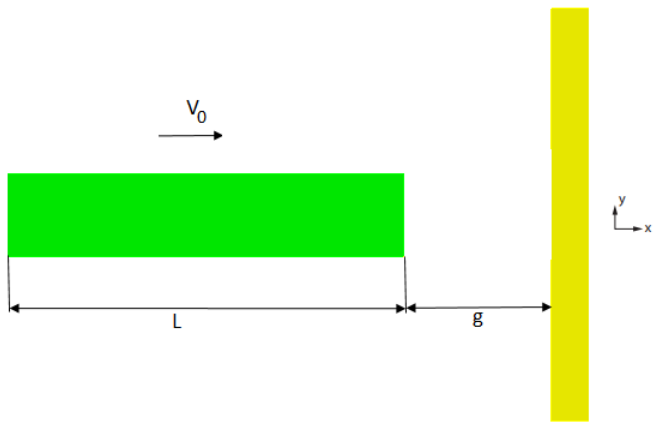


Fig 2.34 - Geometrical model

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.

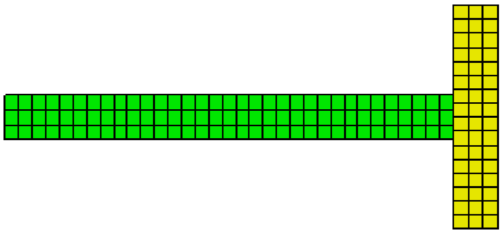


Fig 2.35 - Mesh

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

No	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
	Contact status in contact region at point (5,0,0) at t=0.00015 sec.	contact_status	-	0

No	Value	Description	Unit	Target
	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

No	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
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.

2.21. Test case No2.23

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.

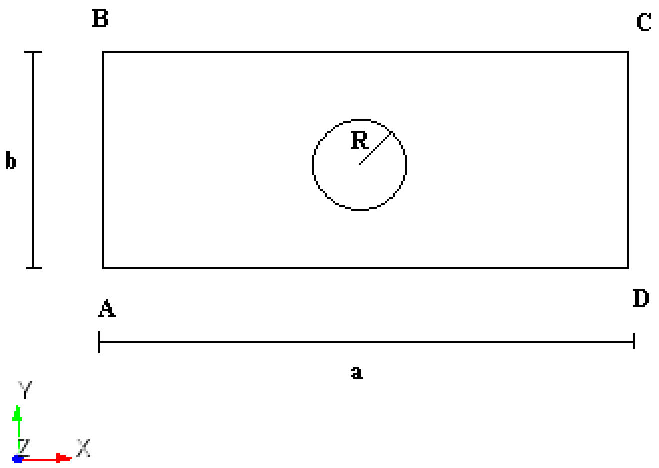


Fig 2.36 - Geometric model of the problem

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.

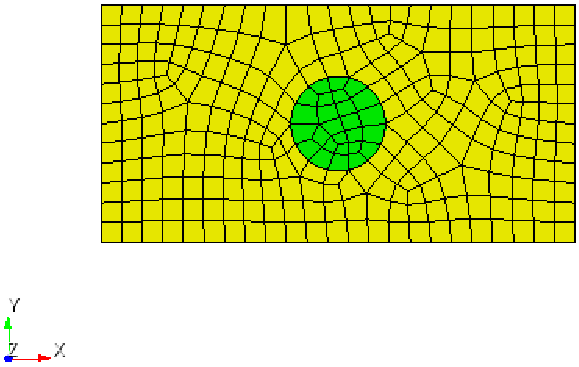


Fig 2.37 - Finite element mesh model

Calculation settings:

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

Output Values

No	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

No	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
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
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5
```

2.22. Test case No2.24

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.

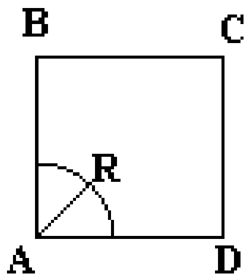


Fig 2.38 - Geometric model of the problem

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.

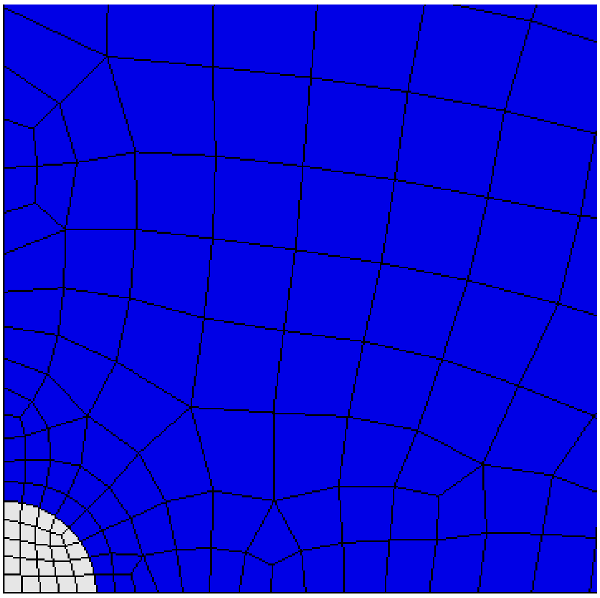


Fig 2.39 - Finite element mesh model

Calculation settings:

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

Output Values

No	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.

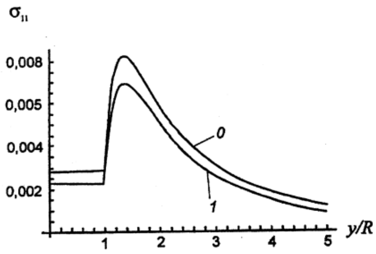


Fig 2.40 - Distribution for inclusion and matrix: 0 – linear solution, 1 – nonlinear solution

Results

No	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 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
mesh surface 11
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
mesh surface 9
create material 1
modify material 1 name 'Матрица'
create material 2
modify material 2 name 'Включение'
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 'Матрица'
set duplicate block elements off
block 2 add surface 11
block 2 name 'Включение'
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
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5
```

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

2.23. Test case No2.23

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.

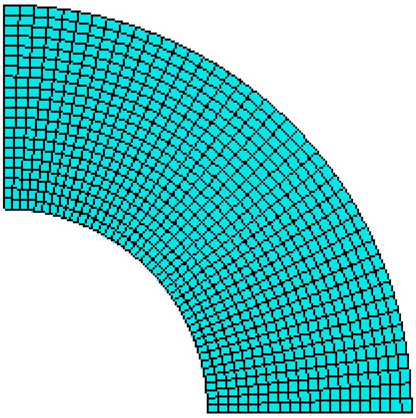


Fig 2.39 - Finite element mesh model

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

No	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	2.119E-03	0.01
3	Displacement component u_x	(0.7, 0,0)	Displacement X	M	0.2458e-2	2.458E-03	0.01
4	Displacement component u_x	(0.8, 0,0)	Displacement X	M	0.2668e-2	2.668E-03	0.01
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	0.01
7	Plastic strain	(1, 0,0)	Plastic_Strain_XX	-	-0.225e-3	-2.295e-04	1.84
8	Plastic strain	(0.9, 0,0)	Plastic_Strain_XX	-	0.769e-4	7.678E-05	0.15
9	Plastic strain	(0.78, 0,0)	Plastic_Strain_XX	-	0.267e-3	2.668E-04	0.09
10	Plastic strain	(0.7, 0,0)	Plastic_Strain_XX	-	0.113e-3	1.133E-04	0.27
11	Plastic strain	(0.67, 0,0)	Plastic_Strain_XX	-	0	-3.630E-07	0.00
12	Plastic strain	(1, 0,0)	Plastic_Strain_YY	-	0.198e-2	1.980E-03	0.02
13	Plastic strain	(0.9, 0,0)	Plastic_Strain_YY	-	0.15e-2	1.496E-03	0.25
14	Plastic strain	(0.8, 0,0)	Plastic_Strain_YY	-	0.878e-3	8.786E-04	0.07

No	Value	Point	Description	Unit	Target	CAE Fidesys result	Error, %
15	Plastic strain	(0.7, 0,0)	Plastic_Strain_YY	-	0.175e-3	1.761E-04	0.60
16	Plastic strain	(0.67, 0,0)	Plastic_Strain_YY	-	0	-5.081E-07	0.00
17	Plastic strain	(1, 0,0)	Plastic_Strain_ZZ	-	0.1736e-3	1.784E-04	2.78
18	Plastic strain	(0.9, 0,0)	Plastic_Strain_ZZ	-	-0.1243e-3	-1.242E-04	0.05
19	Plastic strain	(0.8, 0,0)	Plastic_Strain_ZZ	-	-0.23e-3	-2.300E-04	0.01
20	Plastic strain	(0.7, 0,0)	Plastic_Strain_ZZ	-	-0.747e-4	-7.519E-05	0.66
21	Plastic strain	(0.67, 0,0)	Plastic_Strain_ZZ	-	0	2.339E-07	0.00
22	Elastic strain component ε_{xx}	(1, 0,0)	Elastic_Strain_X	-	-0.303e-3	-3.047E-04	0.54
23	Elastic strain component ε_{xx}	(0.9, 0,0)	Elastic_Strain_X	-	-0.26e-3	-2.599E-04	0.05
24	Elastic strain component ε_{xx}	(0.8, 0,0)	Elastic_Strain_X	-	-0.898e-4	-8.983E-05	0.04
25	Elastic strain component ε_{xx}	(0.7, 0,0)	Elastic_Strain_X	-	0.308e-3	3.081E-04	0.02
26	Elastic strain component ε_{xx}	(0.67, 0,0)	Elastic_Strain_X	-	0.119e-2	1.190E-03	0.03
27	Elastic strain component ε_{xx}	(0.5, 0,0)	Elastic_Strain_X	-	0.274e-2	2.734E-03	0.24
28	Elastic strain component ε_{yy}	(1, 0,0)	Elastic_Strain_Y	-	0.787e-3	7.859E-04	0.01
29	Elastic strain component ε_{yy}	(0.9, 0,0)	Elastic_Strain_Y	-	0.928e-3	9.282E-04	0.02
30	Elastic strain component ε_{yy}	(0.8, 0,0)	Elastic_Strain_Y	-	0.107e-2	1.073E-03	0.28

No	Value	Point	Description	Unit	Target	CAE Fidesys result	Error, %
31	Elastic strain component ε_{yy}	(0.7, 0,0)	Elastic_Strain_Y	-	0.112e-2	1.123E-03	0.31
32	Elastic strain component ε_{yy}	(0.67, 0,0)	Elastic_Strain_Y	-	0.363e-3	3.629E-04	0.03
33	Elastic strain component ε_{yy}	(0.5, 0,0)	Elastic_Strain_Y	-	-0.1184e-2	-1.185E-03	0.05
34	Elastic strain component ε_{zz}	(1, 0,0)	Elastic_Strain_Z	-	-0.181e-3	-1.784E-04	1.42
35	Elastic strain component ε_{zz}	(0.9, 0,0)	Elastic_Strain_Z	-	-0.529e-3	-5.294E-04	0.07
36	Elastic strain component ε_{zz}	(0.8, 0,0)	Elastic_Strain_Z	-	-0.115e-2	-1.154E-03	0.38
37	Elastic strain component ε_{zz}	(0.7, 0,0)	Elastic_Strain_Z	-	-0.214e-2	-2.137E-03	0.12
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
nonlinearopts maxiters 100 minloadsteps 10 maxloadsteps 1000 tolerance 0.01 targetiter 5
```

2.24. Test case No2.24

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

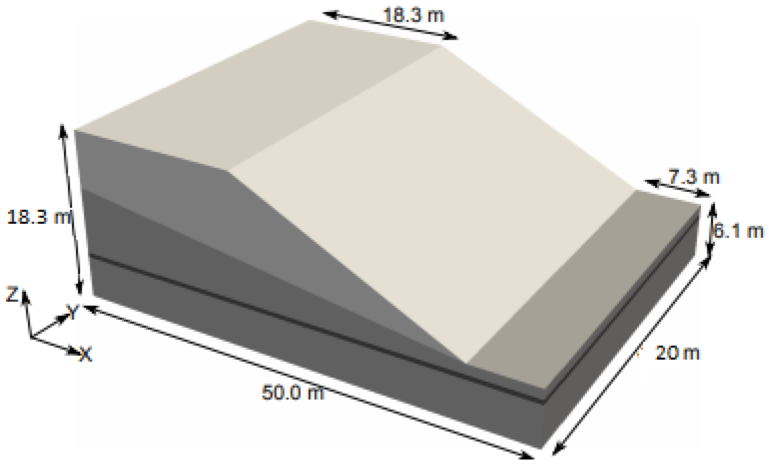


Fig 2.41 - Geometric model of the problem

Geometrical model:

- Typical dimensions are shown in Figure 2.41.

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.

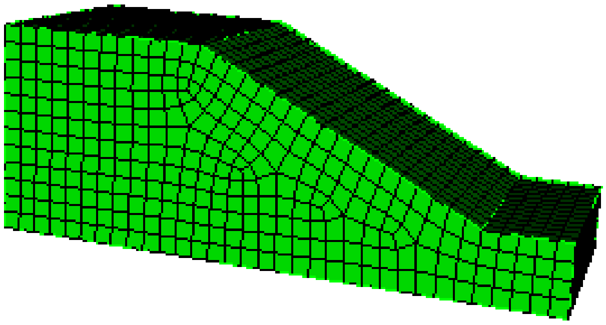


Fig 2.42 - Finite element mesh model

Output Values

No	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 2.41-2.42).

Results

No	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
```

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.

2.25. Test case No2.27

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

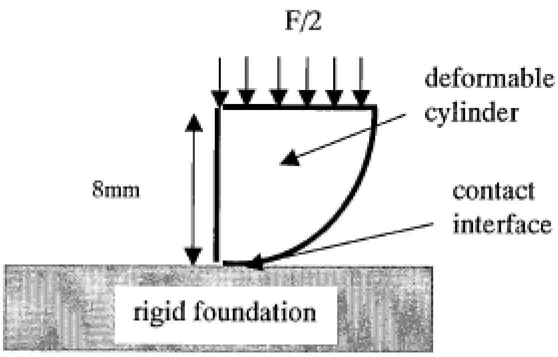


Fig 2.43 - Geometric model of the problem

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.

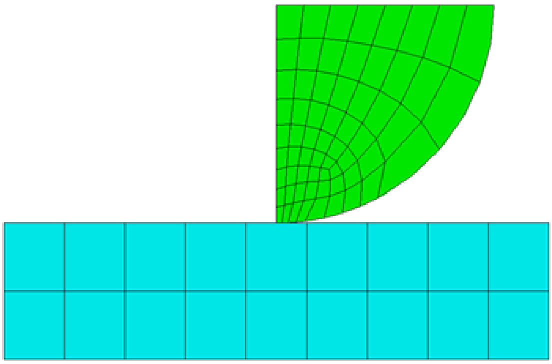


Fig 2.44 - Finite element mesh model

Calculation settings:

- Static analisys;
- 3D;
- Elasticity.

Output Values

No	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.

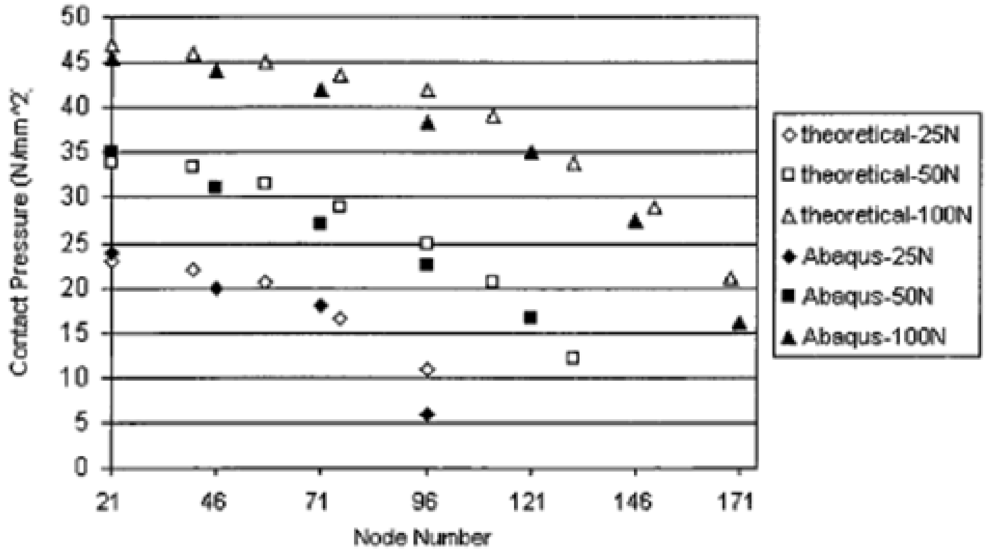


Fig 2.45 - Results of the numerical solution of the problem

Results

No	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).

2.26. Test case No2.26

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.

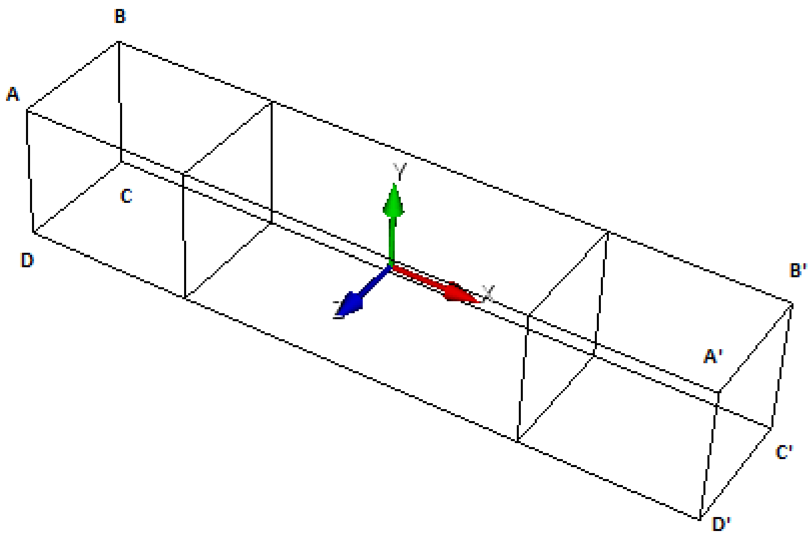


Fig 2.45 - Geometric model of a beam

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/m^3.

Mesh:

- Hexahedrons of the second order.

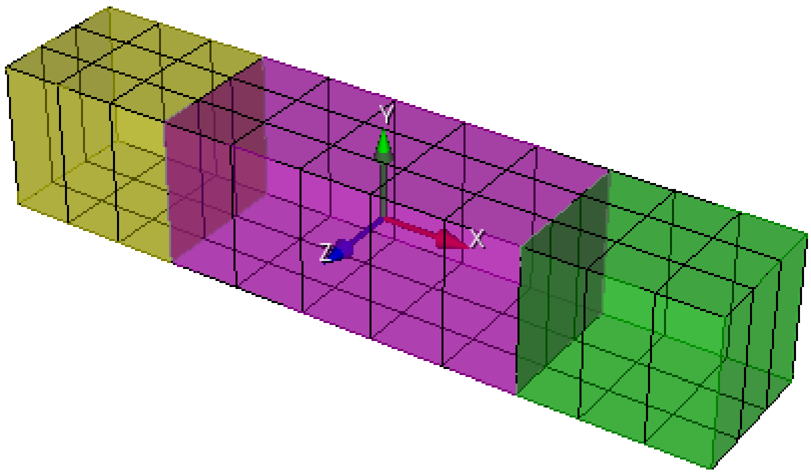


Fig 2.47 - Finite element mesh model

Contact:

- General;
- Method: mpc.

Calculation settings:

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

Output Values

No	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

No	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.

2.27. Test case No2.29

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.

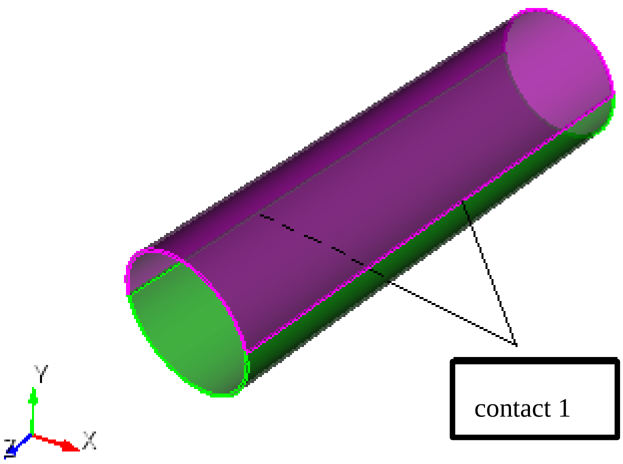


Fig 2.48 - Geometric model of the problem

Boundary conditions:

- Bottom circle is fixed in all directions;
- Pressure applied to the top circle p = 1 MПa;
- 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.

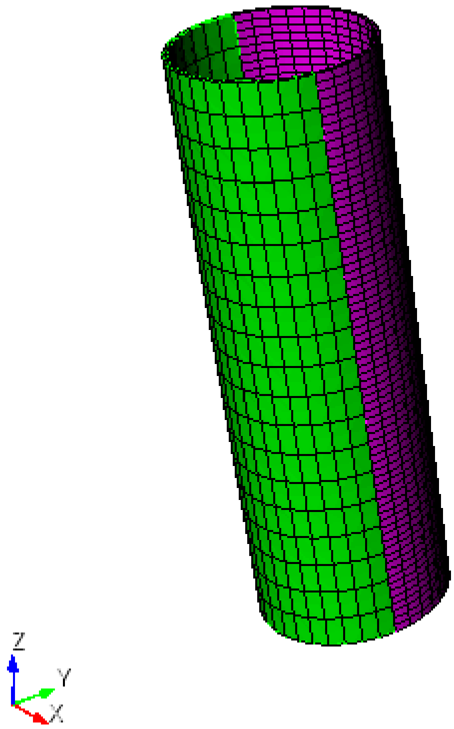


Fig 2.49 - Finite element mesh model

Calculation settings:

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

Output Values

No	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

No	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
```

2.28. Test case No 2.28

Problem Description

The problem of moving a load over a surface is considered, taking into account friction.

When interacting, the load contacts the surface (sliding contact with friction). The control task checks the correctness of the calculation:

- contact interaction "sliding with friction";

- interactions of nonconformally connected grids of spectral elements.

Input Value

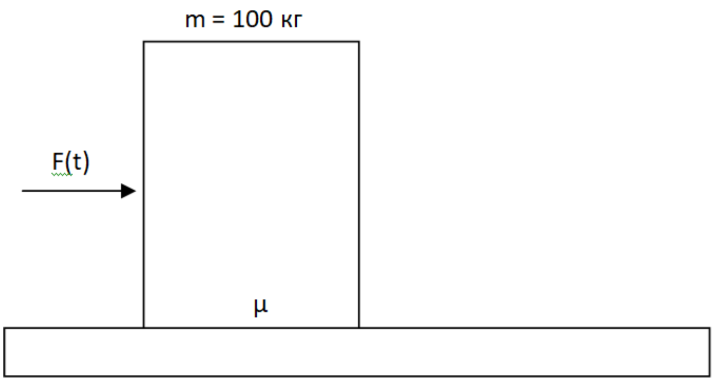


Figure 2.49 - Geometrical model

Geometrical model

- Base: rectangle (L=10 м, h=1 1);
- Cargo: rectangle (L=1 м, h=1 м);
- Coefficients of friction between base and load μ = 0,4; 0,6.

Material:

- E=2e11 Па, ν=0.3.

Boundary conditions:

- The base is fixed in all directions;
- The force acts on the load in the horizontal direction F(x) = 100t
- The force of gravity acts on the load

Mesh:

- Plane finite elements;
- Order 3.

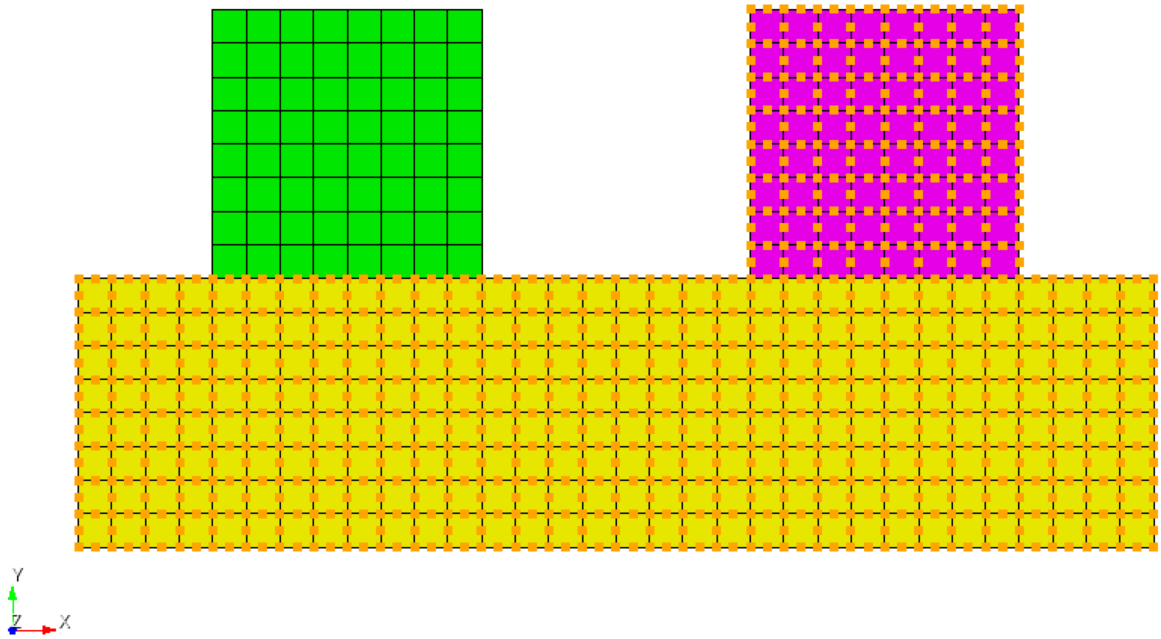


Fig 2.50 – Finite elements mesh

Contact settings:

- General (master – curve 6, slave - curve 4);
- Friction 0.4/0.6;
- Tolerance 0.0005;
- Method Penalty.

Calculation settings:

- Transient;
- 2D;
- Planestrain;
- Full solution;
- Implicit;
- Max time 8 c;
- Steps 1000.

Output Value

No	Value	Description	Unit	Target
1	Contact Status	Contact_Status	-	2
2	Displacement vector component ux for cargo in t = 1	Displacement_XX	м	0
3	Displacement vector component ux for cargo in t = 6	Displacement_XX	м	0
4	Displacement vector component ux for cargo in t = 8	Displacement_XX	м	> 0

Numerically approximate analytical solution

A numerical approximate solution is given in [1].

As a solution, the position of the cube at the final and intermediate moments of time is taken to check the correctness of the work of the static friction force (Fthrust = Ftr. rest) and sliding.

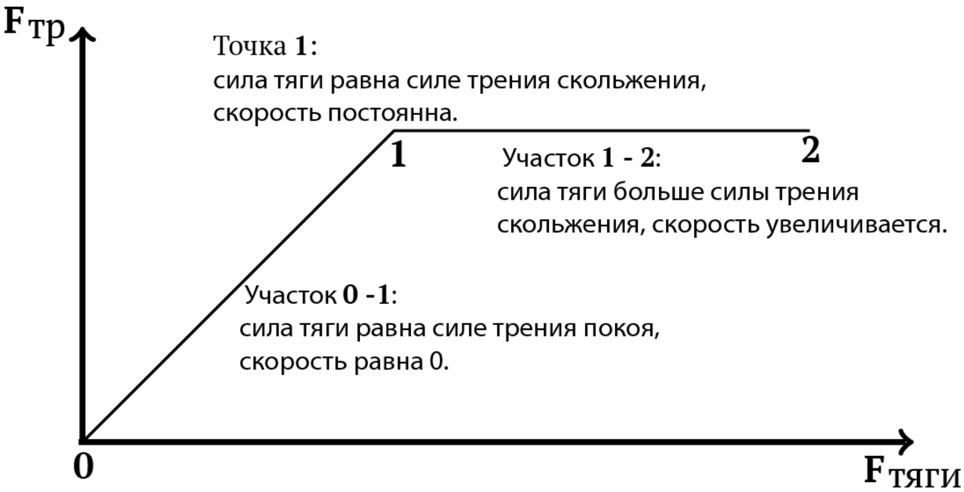


Fig 2.51 - Numerical solution for checking the correctness of the work of the force of static and sliding friction

Reference

[1] Полюшкин, Н.Г. Основы теории трения, износа и смазки: учеб. пособие / Н.Г. Полюшкин; Краснояр. гос. аграр. ун-т. – Красноярск, 2013 – 192 с.

Results

Spectral Elements

No	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Contact Status	Contact_Status	-	2	2	Критерий выполнен
2	Displacement vector component ux for cargo in t = 1	Displacement_XX	м	0	0	0
3	Displacement vector component ux for cargo in t = 6	Displacement_XX	м	0	0	0
4	Displacement vector component ux for cargo in t = 8	Displacement_XX	м	> 0	>0	Критерий выполнен

CAE Fidesys script:

```
reset
```

```
create surface rectangle width 1 height 1 zplane
create surface rectangle width 4 height 1 zplane
Surface 1 copy move x 2 y 0.5
move Surface 2 x 1 y -1 include_merged
surface all size auto factor 4
mesh surface all
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'MODULUS' value 2e+11
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 100
block 1 add surface all
block 1 material 1 cs 1 element plane order 3
create displacement on surface 2 3 dof all fix
create contact autoselect type general friction 0.4 ignore_overlap off offset 0.0 tolerance 0.0005 method penalty normal_stiffness 1.0 tangent_stiffness 0.5
create contact master curve 10 slave curve 4 type general friction 0 ignore_overlap off offset 0.0 tolerance 0.0005 method penalty normal_stiffness 1.0 tangent_stiffness 0.5
move Surface 3 y -0.5 include_merged
create pressure on curve 1 magnitude 1000
create pressure on curve 2 magnitude 1000
bcdep pressure 2 value '1000 * t'
analysis type dynamic elasticity dim2 planestrain preload off
dynamic method full_solution scheme implicit steps 200 newmark_gamma 0.005 maxtime 1
```

2.29. Test case No 2.29

Problem Description

Checking the correctness of the calculation of the interaction of a large elliptical hole with a small one, when the major axes of the ellipses are parallel. The center of the large ellipse is at (0,0). Holes are formed sequentially in an endless plate, previously uniaxially stretched. The solution results compare linear and non-linear formulations. The calculations were carried out for the Mooney material in a plane stress state.

Input Value

Geometrical model:

The plate has an inclusion. During the calculation, the properties of the inclusion material change.

- Length 100 м;
- Width 100 м;
- First ellipse: a_1=1, b_1=0.25, center's coordinate (0,0,0)
- Second ellipse: a_2=0.4, b_2=0.1, x_2=2, y_2=0.625, centers's coordinate (2, 0.625,0).

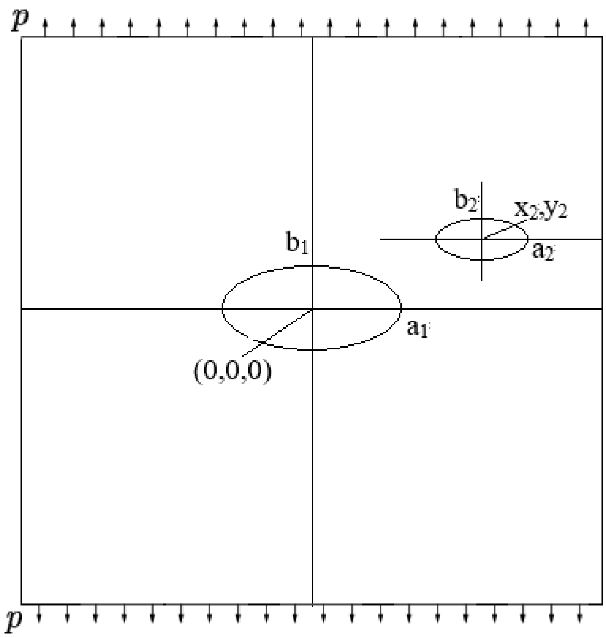


Fig 2.52 - Geometrical model

Boundary conditions:

- One node is fixed on the sides with the coordinate y=0 along the displacement u_y;
- On the upper and lower sides, one node is fixed with the coordinate x=0 along the displacement u_x;
- The plate is subjected to uniaxial incremental tensile pressure in increments:
  - Step 1: - 0.15 Па;
  - Step 2: - 0.15 Па;
  - Step 3: - 0.15 Па.

Materials:

- Mooney-Rivlin Material:

- C10=0.5;
- C01=0;
- D=1.04;
- β=1, G=1 Па, ν=0.48.

Mesh:

- Conformal;
- 2D.

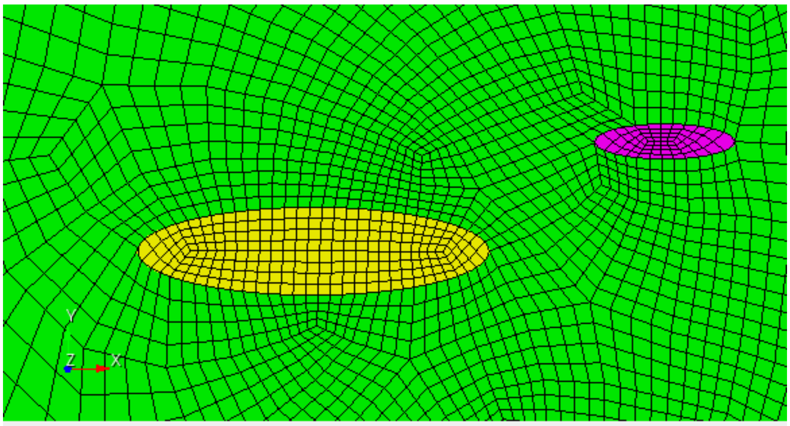


Fig 2.53 – Finite elements mesh near ellipses

Calculation settings:

- Static;
- 2D;
- Planestrain;
- Elasticity;
- Finite deformation (for case 2);
- Load step: 3.

Output Value

Case 1 (linear)

No	Шаги нагружения	Наименование переменной	Обозначение переменной	Размерность	Значение
1	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (1, 0, 0)	Stress_cylindrical_FF	Pa	1.15
2	Step 3	Stress σφφ at the point (-1, 0, 0)	Stress_cylindrical _FF	Pa	1
3	Step 3	Stress σφφ at the point (2.4, 0.625, 0)	Stress_cylindrical _FF	Pa	1.55
4	Step 3	Stress σφφ at the point (1.6, 0.625, 0)	Stress_cylindrical _FF	Pa	1.8

Вариант 2 (нелинейный случай)

No	Шаги нагружения	Наименование переменной	Обозначение переменной	Размерность	Значение
1	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (1, 0, 0)	Stress_cylindrical _FF	Pa	1.35
2	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (-1, 0, 0)	Stress_cylindrical _FF	Pa	1.25
3	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (2.4, 0.625, 0)	Stress_cylindrical _FF	Pa	1.75
4	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (1.6, 0.625, 0)	Stress_cylindrical _FF	Pa	2

Numerically approximate analytical solution

A numerical approximate solution is given in the source [1] (p. 183, Fig. 5.36) and is shown in Figure 2.54.

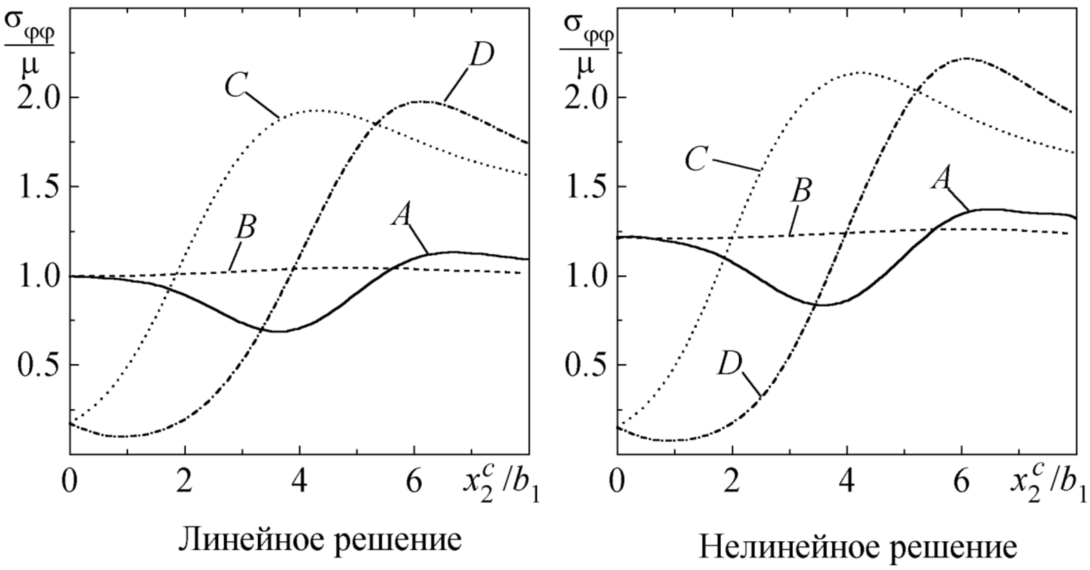


Fig 2.54 – The value of stresses at the tops of the holes according to [1]

Reference

[1] Левин В.А., Зингерман К.М. Плоские задачи теории многократного наложения больших деформаций. Методы решения. - М.: ФИЗМАТЛИТ, 2002. - 272 с. - ISBN 5-9221-0282-6

Results

Вариант 1 (линейный случай)

No	Load step	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (1.01, 0, 0)	Stress_cylindrical _FF	Pa	1.15	1.171	1.83
2	Step 3	Stress σφφ at the point (-1.0125, 0, 0)	Stress_cylindrical _FF	Pa	1	1.033	3.35
3	Step 3	Stress σφφ at the point (2.4015, 0.625, 0)	Stress_cylindrical _FF	Pa	1.55	1.556	0.37
4	Step 3	Stress σφφ at the point (1.599, 0.622934, 0)	Stress_cylindrical _FF	Pa	1.8	1.858	3.23

Case 2 (nonlinear geometry)

No	Load step	Value	Description	Unit	Target	CAE Fidesys result	Error, %
1	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (1.00278, 0.00559868, 0)	Stress_cylindrical _FF	Pa	1.35	1.407	4.2
2	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (-1.006, 0, 0)	Stress_cylindrical _FF	Pa	1.25	1.311	4.9
3	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (2.4, 0.625, 0)	Stress_cylindrical _FF	Pa	1.75	1.753	0.2
4	Step 3	Stress $\sigma_{\varphi\varphi}$ at the point (1.60009, 0.623, 0)	Stress_cylindrical _FF	Pa	2	2.080	3.98

CAE Fidesys script:

linear:

```
reset
set node constrain on
create surface rectangle width 50 zplane
create surface ellipse major radius 1 minor radius 0.25 zplane
subtract surface 2 from surface 1 keep_tool
create surface ellipse major radius 0.4 minor radius 0.1 zplane
move Surface 4 x 2 y 0.625 include_merged preview
move Surface 4 x 2 y 0.625 include_merged
subtract surface 4 from surface 3 keep_tool
webcut body all with plane xplane offset 0
webcut body all with plane yplane offset 0
merge all
```

compress all

#curve 12 9 scheme bias fine size 0.01 factor 1.05 start vertex 11

#curve 12 9 scheme bias fine size 0.01 factor 1.05 start vertex 11

#mesh curve 12 9

#curve 19 18 scheme bias fine size 0.01 factor 1.05 start vertex 13

#curve 19 18 scheme bias fine size 0.01 factor 1.05 start vertex 13

#mesh curve 19 18

surface all sizing function type skeleton min_size auto max_size auto max_gradient 1.5 min_num_layers_2d 1 min_num_layers_1d 1

mesh surface all

refine surface all numsplit 1 bias 1.0 depth 1

#refine surface all numsplit 1 bias 1.0 depth 1

#refine surface all numsplit 1 bias 1.0 depth 1

#refine surface all numsplit 1 bias 1.0 depth 1

create material 1

modify material 1 name 'mat1'

modify material 1 set property 'MOONEY_C01' value 0

modify material 1 set property 'MOONEY_C10' value 0.5

modify material 1 set property 'MOONEY_D' value 4e-06

block 1 add surface 3 2 6 7

set duplicate block elements off

block 2 add surface 1

set duplicate block elements off

block 3 add surface 8 4 5 9

block all material 1

block all element plane order 9

create displacement on vertex 6 9 dof 1 fix

create displacement on vertex 14 10 dof 2 fix

create pressure on curve 6 5 magnitude 1

create pressure on curve 7 4 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 2 1 value 2

modify table 1 cell 3 1 value 3

modify table 1 cell 1 2 value -0.15

modify table 1 cell 2 2 value -0.15

```
modify table 1 cell 3 2 value -0.15
bcdep pressure 1 table 1
create table 2
modify table 2 dependency time
modify table 2 insert row 1
modify table 2 insert row 1
modify table 2 insert row 1
modify table 2 cell 1 1 value 1
modify table 2 cell 2 1 value 2
modify table 2 cell 3 1 value 3
modify table 2 cell 1 2 value -0.15
modify table 2 cell 2 2 value -0.15
modify table 2 cell 3 2 value -0.15
bcdep pressure 2 table 2
static steps 3
block 3 step 1
block 2 step 1 2
analysis type static elasticity dim2 planestress
```

Nonlinear geometry:

```
reset
set node constrain on
create surface rectangle width 50 zplane
create surface ellipse major radius 1 minor radius 0.25 zplane
subtract surface 2 from surface 1 keep_tool
create surface ellipse major radius 0.4 minor radius 0.1 zplane
move Surface 4 x 2 y 0.625 include_merged preview
move Surface 4 x 2 y 0.625 include_merged
subtract surface 4 from surface 3 keep_tool
webcut body all with plane xplane offset 0
webcut body all with plane yplane offset 0
merge all
compress all
#curve 12 9 scheme bias fine size 0.01 factor 1.05 start vertex 11
#curve 12 9 scheme bias fine size 0.01 factor 1.05 start vertex 11
#mesh curve 12 9
#curve 19 18 scheme bias fine size 0.01 factor 1.05 start vertex 13
#curve 19 18 scheme bias fine size 0.01 factor 1.05 start vertex 13
#mesh curve 19 18
```

surface all sizing function type skeleton min_size auto max_size auto max_gradient 1.5 min_num_layers_2d 1 min_num_layers_1d 1

mesh surface all

refine surface all numsplit 1 bias 1.0 depth 1

#refine surface all numsplit 1 bias 1.0 depth 1

#refine surface all numsplit 1 bias 1.0 depth 1

#refine surface all numsplit 1 bias 1.0 depth 1

create material 1

modify material 1 name 'mat1'

modify material 1 set property 'MOONEY_C01' value 0

modify material 1 set property 'MOONEY_C10' value 0.5

modify material 1 set property 'MOONEY_D' value 4e-06

block 1 add surface 3 2 6 7

set duplicate block elements off

block 2 add surface 1

set duplicate block elements off

block 3 add surface 8 4 5 9

block all material 1

block all element plane order 9

create displacement on vertex 6 9 dof 1 fix

create displacement on vertex 14 10 dof 2 fix

create pressure on curve 6 5 magnitude 1

create pressure on curve 7 4 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 2 1 value 2

modify table 1 cell 3 1 value 3

modify table 1 cell 1 2 value -0.15

modify table 1 cell 2 2 value -0.15

modify table 1 cell 3 2 value -0.15

bcdep pressure 1 table 1

create table 2

modify table 2 dependency time

modify table 2 insert row 1

modify table 2 insert row 1

modify table 2 insert row 1

modify table 2 cell 1 1 value 1

modify table 2 cell 2 1 value 2

modify table 2 cell 3 1 value 3

modify table 2 cell 1 2 value -0.15

modify table 2 cell 2 2 value -0.15

modify table 2 cell 3 2 value -0.15

bcdep pressure 2 table 2

static steps 3

block 3 step 1

block 2 step 1 2
analysis type static findefs elasticity dim2 planestress

2.30. Test case No 2.30

Problem Description

Verification of the correctness of the solution of the problem of equilibrium of a load on an inclined plane, taking into account friction in the contact and taking into account the stiffness of the springs. This setup checks for:

- setting the parameters of a sliding contact with friction in the interface;

- static solution taking into account sliding contact with friction.

Input Value

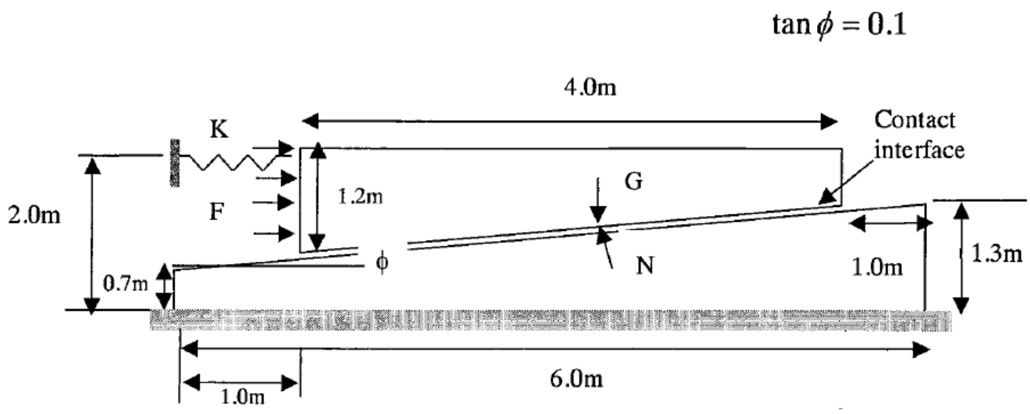


Fig 2.55 - Geometrical model

Material:

- E=206 ГПа, ν=0.3.

Boundary conditions:

- The bottom face is fixed in all directions;
- The left side of the upper surface is fixed along the X axis using the spring element [K(μ)];
- The force of gravity G = 3058 N;
- Force F_x=1500 N acts on the left side.

Contact settings:

- Nonconformal mesh;
- Friction μ=0;
- Type: Sliding with friction:
- Method: Penalty;
- Dependence of the spring stiffness coefficient on the coefficient of friction:
    - μ = 0.0; K = 132,6 N/m;
    - μ = 0.1; K = 98,0 N/m;
    - μ = 0.2; K = 62,6 N/m;
    - μ = 0.3; K = 26,5 N/m.

Mesh:

- Plane finite elements.

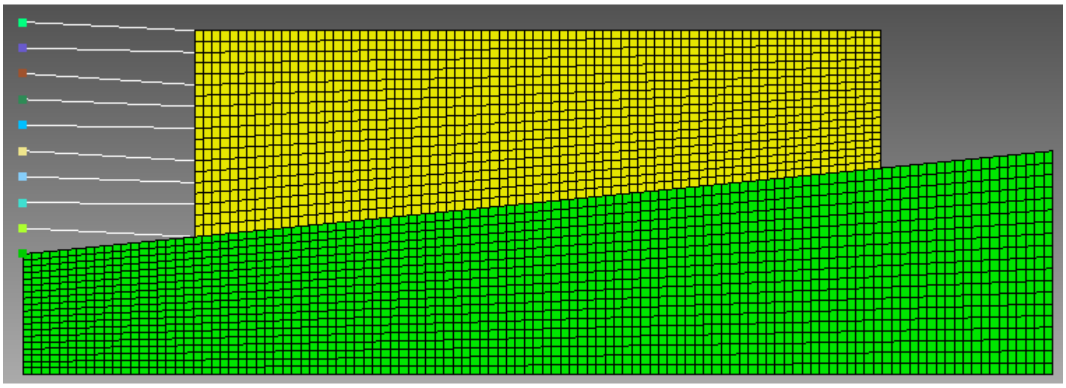


Fig 2.56 – Finite elements mesh

Calculation settings:

- Static;
- 2D;
- Elasticity.

Output Value

No	Friction	Spring stiffness	Value	Unit	Target
1	0	132.6	Horizontal offset U_x	m	1.0
2	0.1	98.0	Horizontal offset U_x	m	1.0
3	0.2	62.6	Horizontal offset U_x	m	1.67
4	0.3	26.5	Horizontal offset U_x	m	1.

Numerically approximate analytical solution

The problem has a numerical approximate solution published in NAFEMS [1]

Reference

[1] NAFEMS R0081 - Benchmark Tests for Finite Element Modelling of Contact, Gapping and Sliding (задача CGS4).

Results

First order elements

No	Friction	Spring stiffness	Value	Unit	Target	CAE Fidesys Results	Error, %
1	0	132.6	U_x	м	1.0	0.9981	0.19
2	0.1	98.0	U_x	м	1.0	0.9995	0.05
3	0.2	62.6	U_x	м	1.0	0.908	0.92
4	0.3	26.5	U_x	м	1.0	0.9866	1.34

Second order elements

No	Friction	Spring stiffness	Value	Unit	Target	CAE Fidesys Results	Error, %
1	0	132.6	U_x	м	1.0	0.97	0.3
2	0.1	98.0	U_x	м	1.0	1.006	0.61
3	0.2	62.6	U_x	м	1.0	0.9901	0.99
4	0.3	26.5	U_x	м	1.0	1.003	0.32

CAE Fidesys script:

First order mesh

```
reset
set default element hex
create surface rectangle width 6 height 1.3 zplane
create surface rectangle width 4 height 1.2 zplane
move Surface 2 y 0.8 include_merged
create vertex on curve 2 distance 0.7 from vertex 3
split surface 1 through vertex 1 9
delete Surface 3
create curve vertex 1 10
move Vertex 12 location vertex 7 include_merged
imprint surface 2 with curve 12
delete surface 6
delete curve 12
Vertex 6 copy move x -1 repeat 1 nomesh
Vertex 14 copy move y -0.15 repeat 8 nomesh
move Surface 5  y -0.05 include_merged
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'MODULUS' value 2.06e11
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 1
set duplicate block elements off
block 1 add surface 4
block 1 material 1 cs 1 element plane order 1
set duplicate block elements off
block 2 add surface 5
block 2 material 1 cs 1 element plane order 1
set node constraint on
curve 4 6 11 14  interval 19
curve 4 6 11 14  scheme equal
mesh curve 4 6 11 14
curve 3 9  interval 115
```

```
curve 3 9  scheme equal
mesh curve 3 9
curve 5 13  interval 79
curve 5 13  scheme equal
mesh curve 5 13
surface 4 5  size auto factor 5
mesh surface 4 5
create displacement on curve 3 dof all fix 0
#1500/20=75
create force  on curve 6  force value 75 direction x
create gravity on surface 5
modify gravity 1 dof 2 value -764.5#==3058N/4=764.5 where 4 is area of wedge
mesh vertex 14 to 22
create edge node 3921 21
create edge node 3922 24
create edge node 3923 27
create edge node 3924 29
create edge node 3925 31
create edge node 3926 34
create edge node 3927 36
create edge node 3928 38
create edge node 3929 22
create displacement on node 3921 to 3929 dof all fix 0
block 3 add edge 7605 to 7613
block 3 element type spring
create spring properties 3
modify spring properties 3 type 'linear_spring'
modify spring properties 3 spring_constant_damping 0
modify spring properties 3 spring_linear_damping 0
modify spring properties 3 spring_mass 0.00001
modify spring properties 3 stiffness 132.6    #friction 0
modify spring properties 3 stiffness_torsional 0
block 3 spring properties 3
block 4 add vertex 9 14 to 22
block 4 element lumpmass
create lumpmass properties 4
modify lumpmass properties 4 mass 1e-16
modify lumpmass properties 4 mass_inertia 0
block 4 lumpmass properties 4
create contact master curve 9 slave curve 13 tolerance 0.0005 type general friction 0.0 ignore_overlap off offset 0.0  method auto
```

output nodalforce on midresults on record3d on log on vtu on

analysis type static elasticity findefs dim2 planestrain

Second order mesh

reset

set default element hex

create surface rectangle width 6 height 1.3 zplane

create surface rectangle width 4 height 1.2 zplane

move Surface 2 y 0.8 include_merged

create vertex on curve 2 distance 0.7 from vertex 3

split surface 1 through vertex 1 9

delete Surface 3

create curve vertex 1 10

move Vertex 12 location vertex 7 include_merged

imprint surface 2 with curve 12

delete surface 6

delete curve 12

Vertex 6 copy move x -1 repeat 1 nomesh

Vertex 14 copy move y -0.15 repeat 8 nomesh

move Surface 5 y -0.05 include_merged

create material 1

modify material 1 name 'mat1'

modify material 1 set property 'MODULUS' value 2.06e11

modify material 1 set property 'POISSON' value 0.3

modify material 1 set property 'DENSITY' value 1

set duplicate block elements off

block 1 add surface 4

block 1 material 1 cs 1 element plane order 2

set duplicate block elements off

block 2 add surface 5

block 2 material 1 cs 1 element plane order 2

set node constraint on

curve 4 6 11 14 interval 19

curve 4 6 11 14 scheme equal

mesh curve 4 6 11 14

curve 3 9 interval 115

curve 3 9 scheme equal

mesh curve 3 9

curve 5 13 interval 79

curve 5 13 scheme equal

mesh curve 5 13

```
surface 4 5 size auto factor 5
mesh surface 4 5
create displacement on curve 3 dof all fix 0
#1500/20=75
create force on curve 6 force value 38.46154 direction x
#create force on node 41 40 42 43 44 45 46 47 48 49 50 51 52 53 54 55 56 57 58 59 force value 75 direction x
create gravity on surface 5
modify gravity 1 dof 2 value -764.5#==3058N/4=764.5 where 4 is area of wedge
mesh vertex 9
mesh vertex 22
mesh vertex 21
mesh vertex 20
mesh vertex 19
mesh vertex 18
mesh vertex 17
mesh vertex 16
mesh vertex 15
mesh vertex 14
create edge node 11526 41
create edge node 11527 76
create edge node 11528 55
create edge node 11529 71
create edge node 11530 50
create edge node 11531 66
create edge node 11532 45
create edge node 11533 43
create edge node 11534 40
create displacement on node 11525 to 11534 dof all fix 0
block 3 add edge 7605 to 7614
block 3 element type spring
create spring properties 3
modify spring properties 3 type 'linear_spring'
modify spring properties 3 spring_constant_damping 0
modify spring properties 3 spring_linear_damping 0
modify spring properties 3 spring_mass 0.00001
modify spring properties 3 stiffness 132.6 #friction 0
modify spring properties 3 stiffness_torsional 0
block 3 spring properties 3
block 4 add vertex 22 21 20 15 18 19 17 9 14 16
block 4 element lumpmass
```

create lumpmass properties 4

modify lumpmass properties 4 mass 1e-16

modify lumpmass properties 4 mass_inertia 0

block 4 lumpmass properties 4

create contact master curve 9 slave curve 13 tolerance 0.0005 type general friction 0.0 ignore_overlap off offset 0.0 method penalty normal_stiffness 0.001 tangent_stiffness 0.05

output nodalforce on midresults on record3d on log on vtu on

analysis type static elasticity findefs dim2 planestrain

2.31. Test Case No 2.31

Problem Description

Verification of the correctness of the solution of the problem of calculating the plate for natural frequencies with the addition of the condition of sliding contact with friction. The control task checks the correctness of the calculation of the modal analysis, taking into account the contact interaction "sliding contact with friction".

Input Value

Geometrical model:

- See figure 2.57;
- Width b = 0,1 м;
- Thickness h = 0,002 м;
- Length a = 0,1 м.

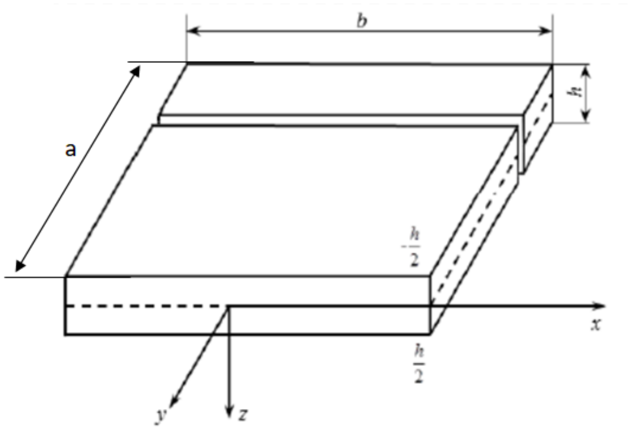


Рисунок 2.57 - Geometrical model

Boundary conditions:

- Case 1: None;
- Case 2: One side is fixed;
- Contact - General with friction, method Auto, Friction: 0, 0.2, 1.

Material:

- Elastic modulus E = 7e10 Па;
- Poisson's ratio ν=0.3;
- Density ρ=7850 кг/м3

Mesh:

- Hexhahedron.

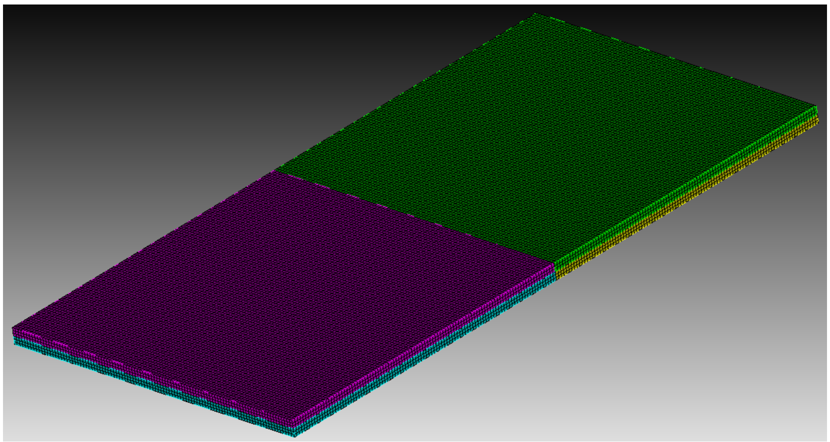


Fig 2.58 – Finite elements mesh

Calculation settings:

- Modal;
- 3D;
- Number of modes: 7.

Output Value

Case 1: No boundary conditions

No	Value	Description	Unit	Target
1	Mode 1	Eigen Values	Hz	0
2	Mode 2	Eigen Values	Hz	0
3	Mode 3	Eigen Values	Hz	0
4	Mode 4	Eigen Values	Hz	0
5	Mode 5	Eigen Values	Hz	0
6	Mode 6	Eigen Values	Hz	0
7	Mode 7	Eigen Values	Hz	650

Case 2: one of the side faces is fixed

No	Value	Description	Unit	Target
1	Mode 1	Eigen Values	Hz	170
2	Mode 2	Eigen Values	Hz	412
3	Mode 3	Eigen Values	Hz	1038
4	Mode 4	Eigen Values	Hz	1318
5	Mode 5	Eigen Values	Hz	1501
6	Mode 6	Eigen Values	Hz	2616
7	Mode 7	Eigen Values	Hz	2983

CAE Fidesys script:

```
reset
brick x 0.1 y 0.1 z 0.002
webcut volume 1 with plane zplane offset 0
webcut volume all with plane yplane offset 0
curve 18 26 20 25 interval 2
curve 18 26 20 25 scheme equal
move Volume 3 4 y .02 include_merged
merge all
move Volume 3 4 y -.02 include_merged
volume all  size auto factor 7
mesh volume all
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 7.9e+10
block 1 add volume all
block 1 material 1 cs 1 element solid order 2
create pressure  on surface 8 16  magnitude 5000
create displacement on curve 13 dof 1 dof 2 dof 3 fix 0
create displacement on curve 15 dof 1 dof 3 fix 0
create contact master surface 17 27  slave surface 32 22  type general friction 0.1 ignore_overlap off offset 0.0 tolerance 0.0005 method auto
analysis type buckling elasticity dim3
eigenvalue find 1 smallest
```

Numerically approximate analytical solution

The reference is the solution obtained in the ANSYS package.

Results

Case 1: No boundary conditions

No	Value	Description	Unit	Target	CAE Fidesys Result	Error, %
1	Mode 1	Eigen Values	Hz	0	0	<<0.01
2	Mode 2	Eigen Values	Hz	0	0	<<0.01
3	Mode 3	Eigen Values	Hz	0	0	<<0.01
4	Mode 4	Eigen Values	Hz	0	0	<<0.01
5	Mode 5	Eigen Values	Hz	0	0	<<0.01
6	Mode 6	Eigen Values	Hz	0	0	<<0.01
7	Mode 7	Eigen Values	Hz	650	649.4	0.01

Case 2: one of the side faces is fixed

No	Value	Description	Unit	Target	CAE Fidesys Result	Error, %
1	Mode 1	Eigen Values	Hz	170	169	0.62
2	Mode 2	Eigen Values	Hz	412	411.5	0.12
3	Mode 3	Eigen Values	Hz	1038	1034	0.4
4	Mode 4	Eigen Values	Hz	1318	1305	0.96
5	Mode 5	Eigen Values	Hz	1501	1496	0.32
6	Mode 6	Eigen Values	Hz	2616	2604	0.48
7	Mode 7	Eigen Values	Hz	2983	2968	0.49

CAE Fidesys script:

```
reset
brick x 0.1 y 0.1 z 0.002
webcut volume 1 with plane zplane offset 0
webcut volume all with plane yplane offset 0
curve 18 26 20 25 interval 2
curve 18 26 20 25 scheme equal
move Volume 3 4 y .02 include_merged
merge all
move Volume 3 4 y -.02 include_merged
volume all  size auto factor 7
mesh volume all
create material 1
modify material 1 name 'mat1'
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'MODULUS' value 7.9e+10
block 1 add volume all
block 1 material 1 cs 1 element solid order 2
create pressure  on surface 8 16  magnitude 5000
create displacement on curve 13 dof 1 dof 2 dof 3 fix 0
create displacement on curve 15 dof 1 dof 3 fix 0
create contact master surface 17 27  slave surface 32 22  type general friction 0.1 ignore_overlap off offset 0.0 tolerance 0.0005 method auto
analysis type buckling elasticity dim3
eigenvalue find 1 smallest
```

3. Test Cases for cloud version

3.1.Test Case No.3.1

Problem Description

The problem of an infinite cylindrical pipe under the influence of internal pressure is considered.

Input Values

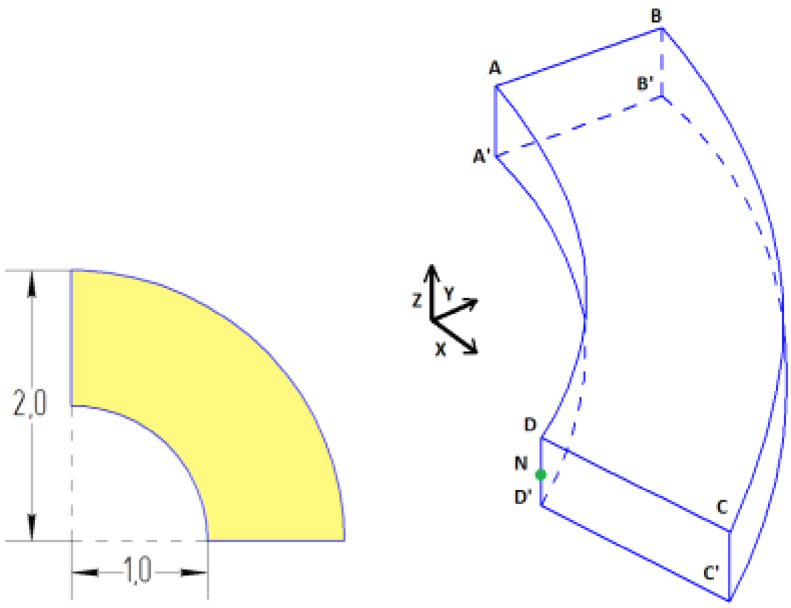


Fig 3.1 - Geometrical CAD-model

Geometric model:

- Due to the symmetry of the problem, a quarter of the wide cut of the pipe is considered;
- CAD-model 01.stp.

Boundary conditions:

- Symmetry condition: surface ABB'A' displacement $u_x = 0$;
- Symmetry condition: surface CDD'C' displacement $u_y = 0$;
- Symmetry condition: surfaces ABCD and A’B’C’D’ displacement $u_z = 0$;
- A pressure p = 1 MPa is applied to the surface AA’D’D;
- A pressure p = 0.5 MPa is applied to the surface B’B’C’C.

Material Properties:

- Isotropic;

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

Meshes:

Finite element mesh is shown in the figure 3.2

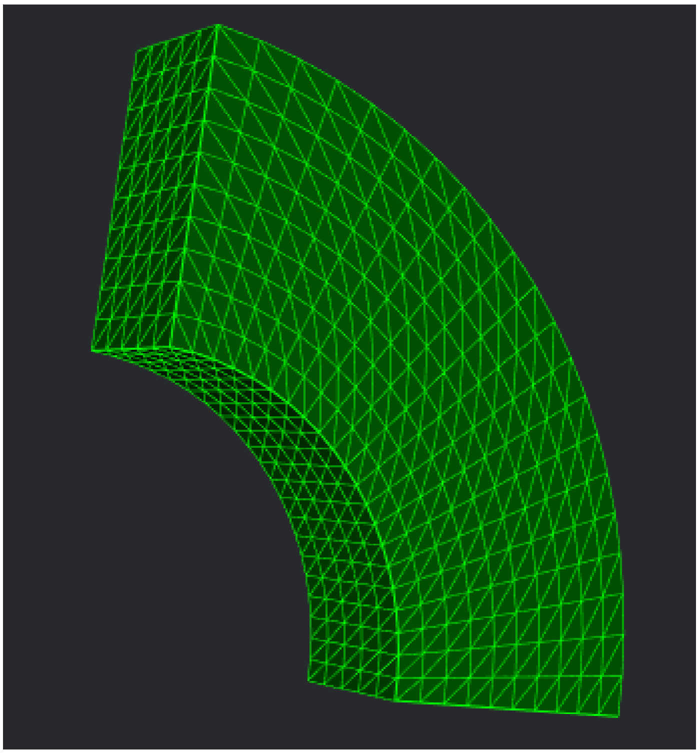


Fig 3.2 - Finite element mesh

Calculation Settings:

- Static;
- Elasticity.

Output Values

The stress values at point N (1,0,0) are given below.

No	Value	Description	Unit	Target
1	Component RR of the stress vector at the nodes	Stress RR	MPa	-1.00
2	Component TT of the stress vector at the nodes	Stress TT	MPa	0.33

No	Value	Description	Unit	Target
3	Component ZZ of the stress vector at the nodes	Stress ZZ	MPa	-0.2

Calculation method used for the reference solution

Analytical solutions are calculated by the following formulas:

$$\sigma_{rr} = \sigma_{11} = \frac{a^2 p_a}{b^2 - a^2}\left(1 - \frac{b^2}{r^2}\right) - \frac{b^2 p_b}{b^2 - a^2}\left(1 - \frac{a^2}{r^2}\right)$$

$$\sigma_{\theta\theta} = r^2 \ \sigma_{22} = \frac{a^2 p_a}{b^2 - a^2}\left(1 + \frac{b^2}{r^2}\right) - \frac{b^2 p_b}{b^2 - a^2}\left(1 + \frac{a^2}{r^2}\right)$$

$$\sigma_{zz} = \sigma_{33} = \frac{\lambda}{\lambda + \mu}\frac{a^2 p_a - b^2 p_b}{b^2 - a^2}$$

The values are taken at the point where the Cartesian coordinate system coincides with the cylindrical coordinate system.

Reference:

[1] Седов Л.И. “Механика сплошной среды, том 2”. М.: Наука, 1970г., 568 стр.

Result comparison

No	Value	Description	Unit	Target	ProveDesign Results	Error,%
1	Component XX of the stress vector at the nodes	Stress XX	MPa	-1.00	-0.95	5
2	Component YY of the stress vector at the nodes	Stress TT	MPa	0.33	0.34	3
3	Component ZZ of the stress vector at the nodes	Stress ZZ	MPa	-0.2	-0.195	5.0

The distribution of the stress field σ_{xx} is shown in Figure 3.3.

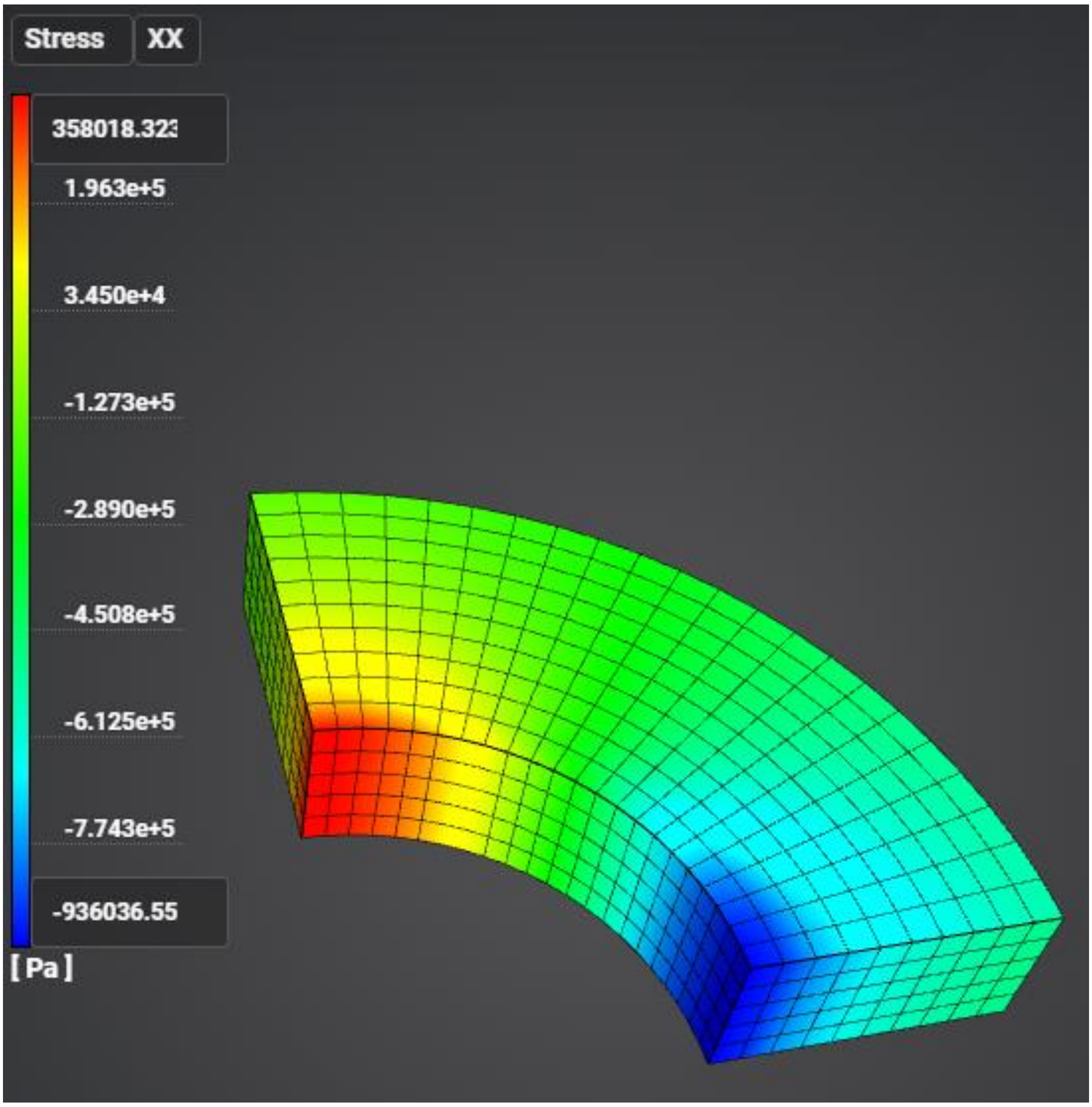


Fig 3.3 - Distribution of stress XX

3.2.Test Case No3.2

Problem Description

The cube is divided into 4 parts, each of which has its own element type. In view of symmetry, the 1/8 part of the cube is considered. A pressure of uniform compression was applied to the top face. The test case verifies tied contact condition.

Input Values

Geometric model:

- CAD-model 02.stp

Boundary conditions:

- Symmetry conditions;
- A pressure of 1e6 Pa is applied to the upper face.

Material Properties:

- Poisson ratio ν = 0.3;
- Young's modulus E = 2e11 Pa.

The model is divided into 4 blocks:

- Finite element mesh is shown in the figure 3.4.

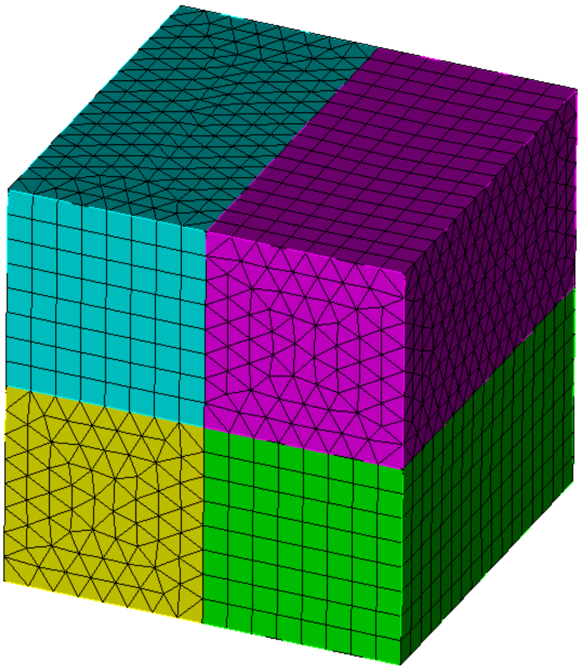


Fig 3.4 - Finite element mesh

Calculation Settings:

- Static;
- Elasticity.

Output Values

The values at point (5,5,-5) are given below.

No	Value	Description	Unit	Target
1	Component Y of the displacement vector at the nodes	Displacement Y	m	$-5{\cdot}10^{-5}$
2	Component YY of the stress tensor at the nodes	Stress YY	MPa	-1
3	Component Mises YY of the stress tensor at the nodes	Stress Mises	MPa	1

Calculation method used for the reference solution

Analytical solutions are calculated by the following formulas [1]:

$$\sigma_{yy} = P, \sigma_{xx} = \sigma_{zz} = \sigma_{xy} = 0;$$

$$\varepsilon_{yy} = \sigma_{yy} E$$

$$u_y = \varepsilon_{yy} L$$

Reference:

[1] Седов Л.И. “Механика сплошной среды, том 2”. М.: Наука, 1970г., 568 стр.

Result comparison

No	Value	Description	Unit	Target	ProveDesign Results	Error,%
1	Component Y of the displacement vector at the nodes	Displacement Y	m	$-5{\cdot}10^{-5}$	$-4.997{\cdot}10^{-5}$	0.06
2	Component YY of the stress tensor at the nodes	Stress YY	MPa	-1	-1	0
3	Component Mises YY of the stress tensor at the nodes	Stress Mises	MPa	1	1	0

3.3.Test Case No.3.3

Problem Description

The problem of uniaxial stretching of the cube is considered. In view of symmetry, the 1/8 part of the original model is viewed. A boundary movement condition is applied to the top face. This test case verifys the correct work of the model when the boundary condition Displacement is in effect.

Input Values

Geometric model:

- CAD-model 04.stp

Boundary conditions:

- Symmetry conditions;
- Surface for z = 5: u_z = -1 m.

Material Properties:

- Isotropic;
- Young's modulus E = 200 Гпа;
- Poisson ratio ν = 0.3.

Meshes:

Finite element mesh is shown in the figure 3.5.

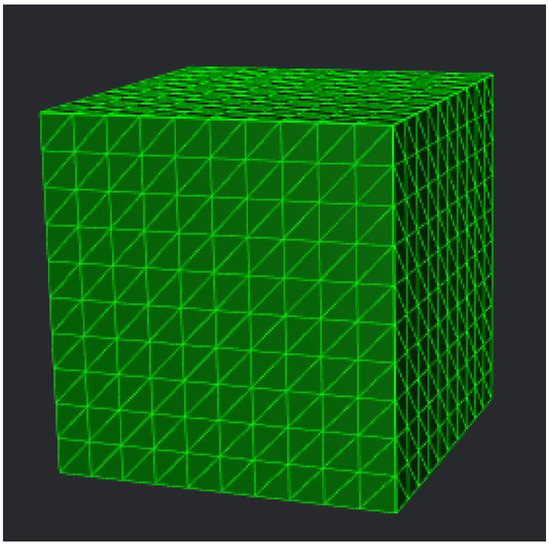


Fig 3.5 - Finite element mesh

Calculation Settings:

- Static;
- Elasticity.

Output Values

The values for displacements, strain and stresses at point (10,10,0) are given below.

No	Value	Description	Unit	Target
1	Components of the displacement vector at mesh nodes	Displacement Z	м	-1
2	Components of the displacement vector at mesh nodes	Displacement X	м	0.3
3	Components of the displacement vector at mesh nodes	Displacement Y	м	0.3
4	Components of the strain tensor at mesh nodes	Strain ZZ	-	-0.1
5	Components of the strain tensor at mesh nodes	Strain XX	-	0.03
6	Components of the strain tensor at mesh nodes	Strain YY	-	0.03
7	Components of the strain tensor at mesh nodes	Strain XY, Strain XZ, Strain YZ	-	0
8	Components of the stress tensor at mesh nodes	Stress ZZ	Па	-2e10
9	Components of the stress tensor at mesh nodes	Stress XX, Stress YY	Па	0

Calculation method used for the reference solution

Analytical solutions are calculated by the following formulas [1]:

$$\varepsilon_{zz} = \frac{u_z}{L}; \varepsilon_{xx} = \varepsilon_{yy} = -\nu\frac{\sigma_{zz}}{E};$$

$$\sigma_{zz} = \varepsilon_{zz}E;\ \sigma_{xx} = \sigma_{yy} = 0;$$

$$u_z = -1 \text{ м};\ u_x = \frac{\varepsilon_{xx}}{L};\ u_y = \frac{\varepsilon_{yy}}{L}.$$

Where σ – the stress tensor, ε – the strain tensor, u – the displacement vector, E – Young's modulus, ν - Poisson ratio, L – side of the cube.

Reference:

[1] Седов Л.И. "Механика сплошной среды, том 2". М.: Наука, 1970г., 568 стр.

Result comparison

No	Value	Description	Unit	Target	ProveDesign Results	Error,%
1	Components of the displacement vector at mesh nodes	Displacement Z	m	-1	-1	0
2	Components of the displacement vector at mesh nodes	Displacement X	m	0.3	0.3	0
3	Components of the displacement vector at mesh nodes	Displacement Y	m	0.3	0.3	0
4	Components of the strain tensor at mesh nodes	Strain ZZ	-	-0.1	-0.1	0
5	Components of the strain tensor at mesh nodes	Strain XX	-	0.03	0.03	0
6	Components of the strain tensor at mesh nodes	Strain YY	-	0.03	0.03	0
7	Components of the strain tensor at mesh nodes	Strain XY, Strain XZ, Strain YZ	-	0	0	0
8	Components of the stress tensor at mesh nodes	Stress ZZ	Pa	-2e10	-2e10	0
9	Components of the stress tensor at mesh nodes	Stress XX, Stress YY	Pa	0	0	0

The distribution of displacement u_z is shown in Figure 3.6.

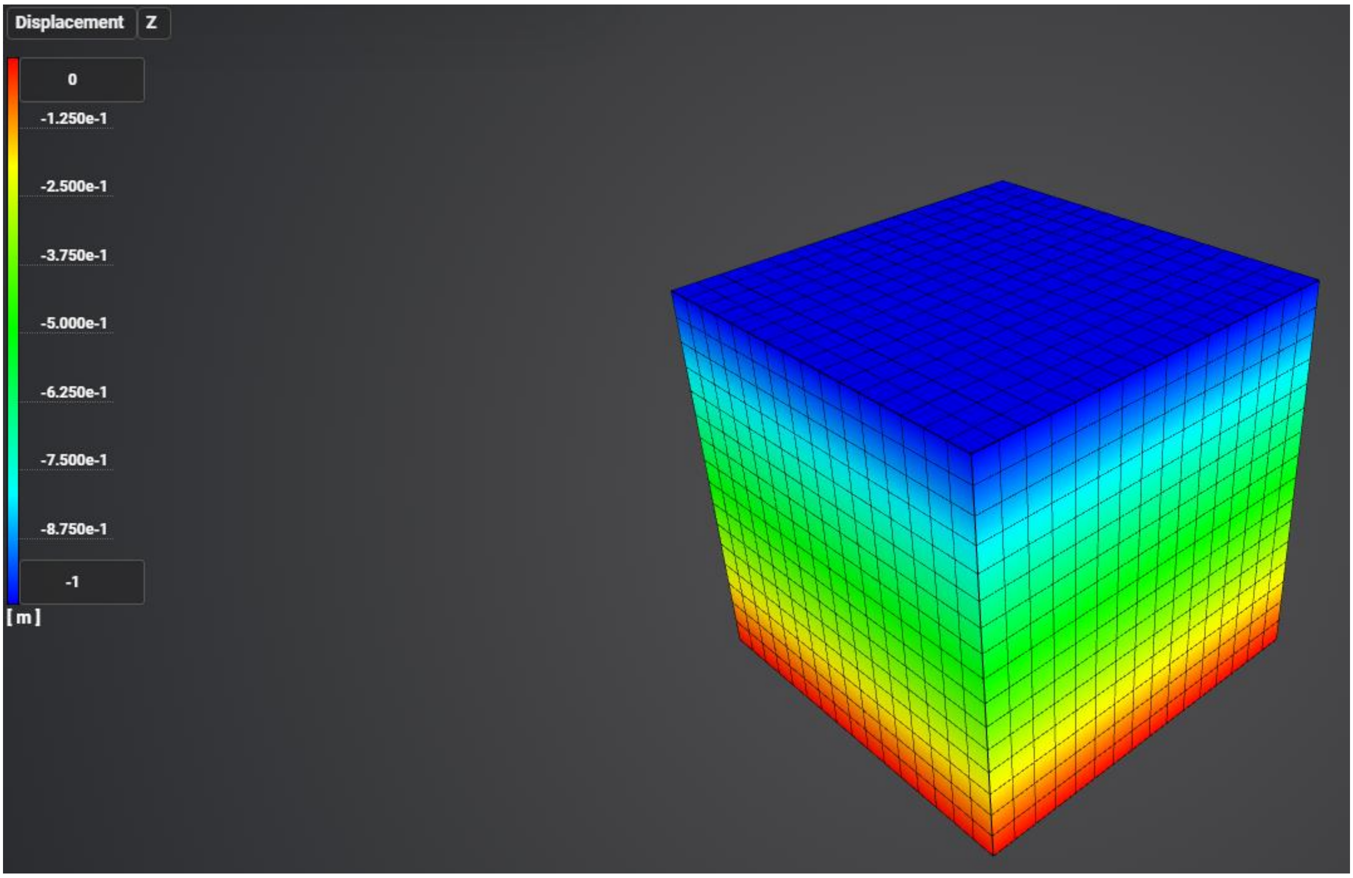


Fig 3.6 - Distribution of displacement u_z

3.4. Test Case No.3.4

Problem Description

In the problem, a suspended beam with a square section, fixed in the upper sections, is applicable. An axial tensile force is applied to the free end of the beam.

Input Values

Geometrical model:

- 01_model.stp.

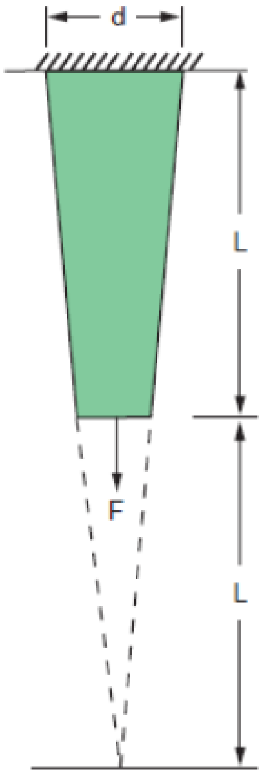


Fig 3.7 - Geometrical model

Boundary Conditions:

- Zero displacement along all axises in Y = 0 plane;
- Force F = 10 000 lb, applied to all nodes in Y = L plane.

Material properties:

- Young's modulus $E = 10.4e + 6$ psi;
- Poisson ratio $\nu = 0.3$.

Mesh:

- See figure 3.8.

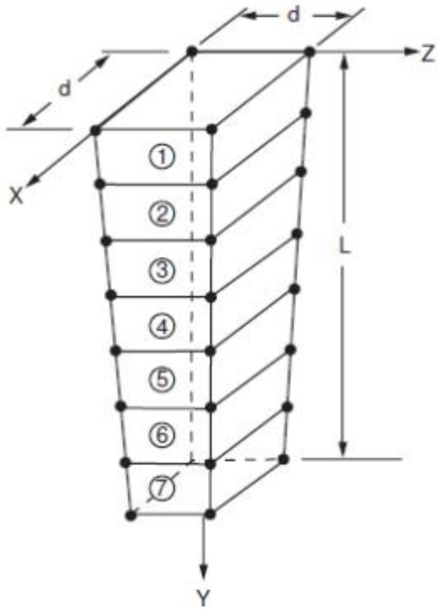


Fig 3.8 - Finite-element mesh

Output Values

No	Value	Discription	Unit	Target
1	Component σ_{yy} at point (1, L/2, 1)	Stress YY	psi	4444

Calculation method used for the reference solution

The ANSYS solution VM37 problem acts as a reference [1].

Reference

[1] Verification Manual for the Mechanical APDL Application, SAS IP, Inc 2009

Result comparison

No	Value	Discription	Unit	Target	ProveDesign Results	Error, %
1	Component σ_{yy} at point (1, L/2, 1)	Stress YY	psi	4444	4466	0.5

The distribution of stress σ_{yy} is shown in Figure 3.9.

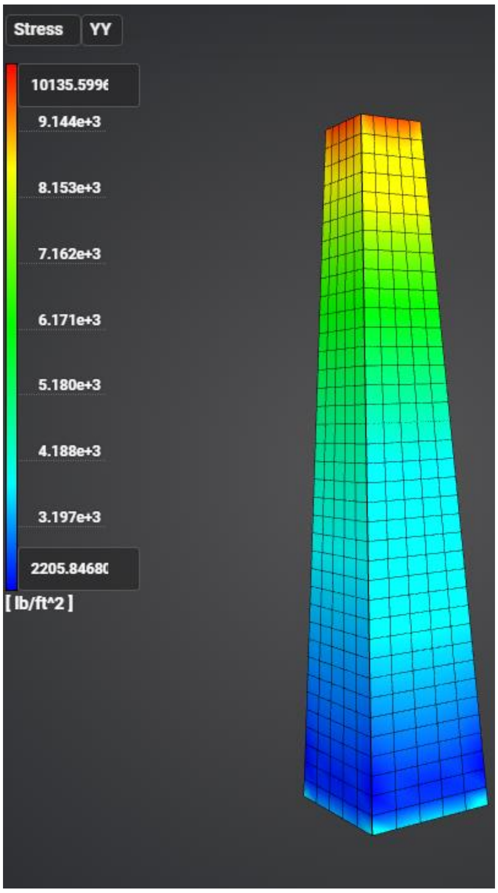


Fig 3.9 – The distribution of stress σ_{yy}

3.5. Test Case No.3.5

Problem Description

The problem of testing the ability of contact algorithms to transfer total displacements using a non-conformal irregular mesh with a rigid contact is considered.

Input Values

Geometrical model:

- 02_model.stp.

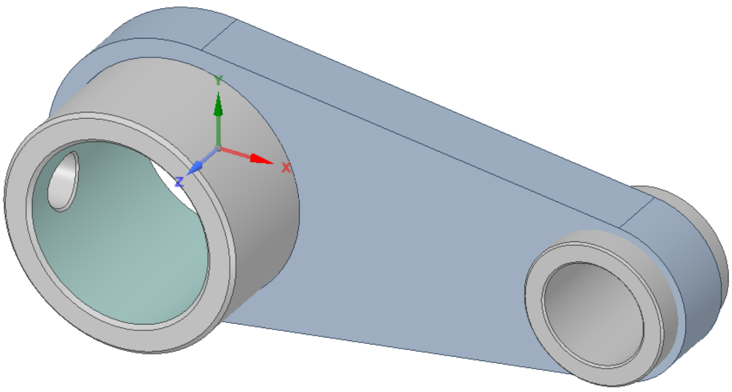


Fig 3.10 - Geometrical model

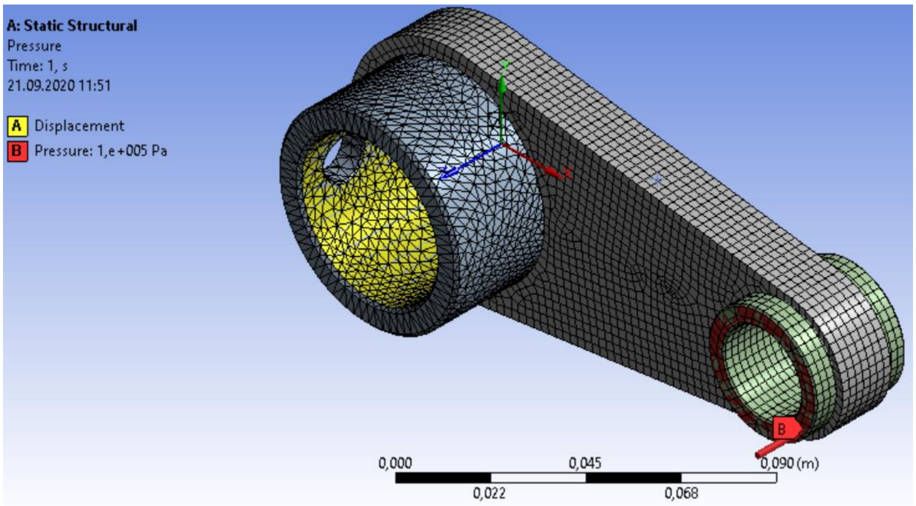


Fig 3.11 - Meshes and boundary conditions

Material:

- Isotropic;
- Young's modulus E = 2e11 Па;
- Poisson ratio ν = 0.3;
- Density ρ = 7850 kg/m^3.

Mesh:

- See figure 70.

Настройки расчета:

- Static, elasticity.

Output Values

Below are the numerical values of the displacements.

No	Value	Description	Unit	Target
1	Maximum Displacement Sum	Displacement sum	м	3.2092e-6

Calculation method used for the reference solution

See Test Case 2.12.

Result comparison

No	Value	Description	Unit	Target	ProveDesign Results	Error, %
1	Maximum Displacement Sum	Displacement sum	м	3.2092e-6	3.095e-6	3.69

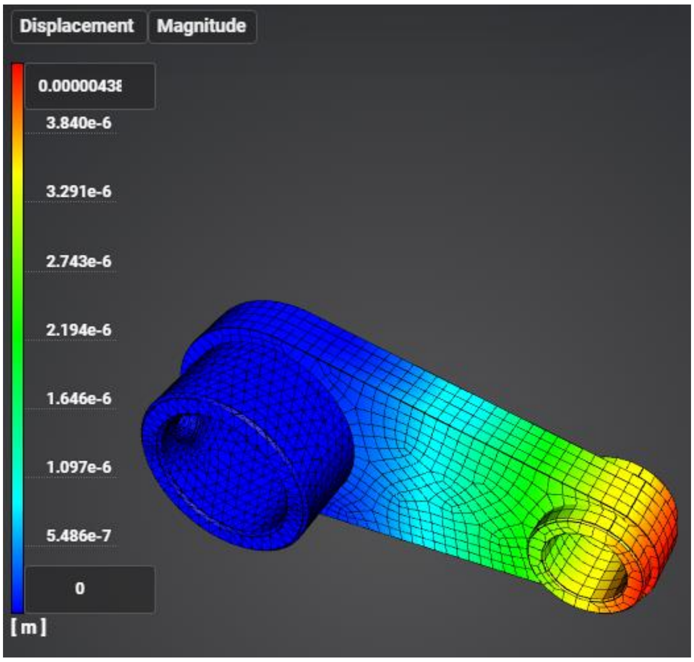


Fig 3.13 - The displacement Sum

3.6.Test case No3.6

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

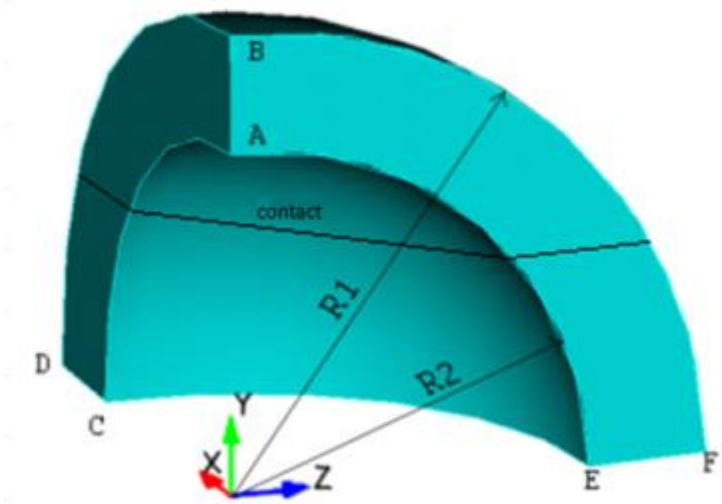


Fig 3.14 - Geometric model for a hollow sphere

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 $T = 30$°C.

Material Properties:

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

Mesh:

- Tetrahedrons of the second order.

Contact:

- Tied;
- Method: auto.

Calculation settings:

- Static calculation;
- Elasticity, thermal conductivity.

Output Values

No	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

No	Value	Description	Unit	Target	ProveDesign Results	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

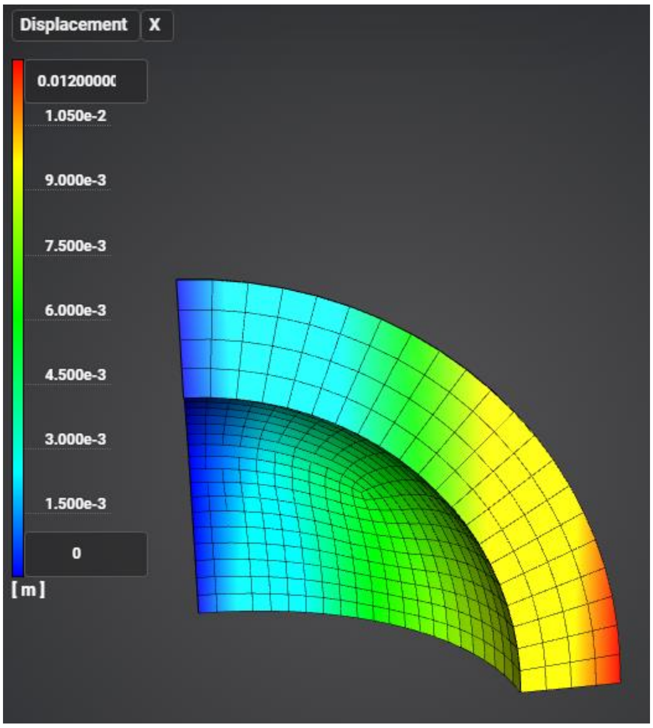


Fig 3.15 - Result of displacements X

Reference:

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

3.7.Test case No3.7

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

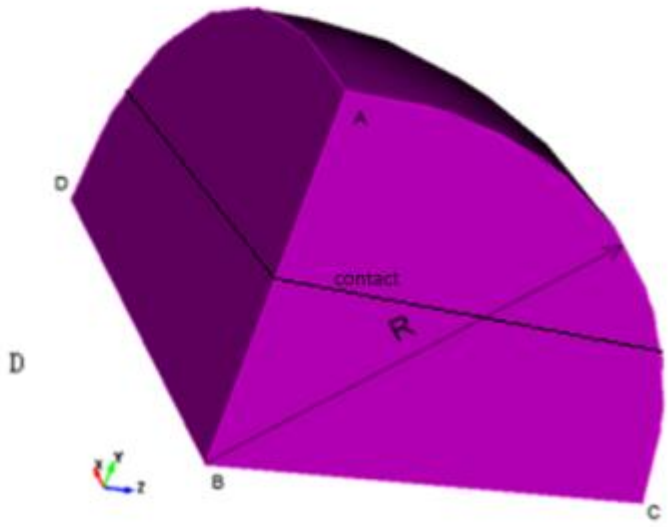


Fig 3.16 - Geometric model for a hollow sphere

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

No	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

No	Value	Description	Unit	Target	ProveDesign Results	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

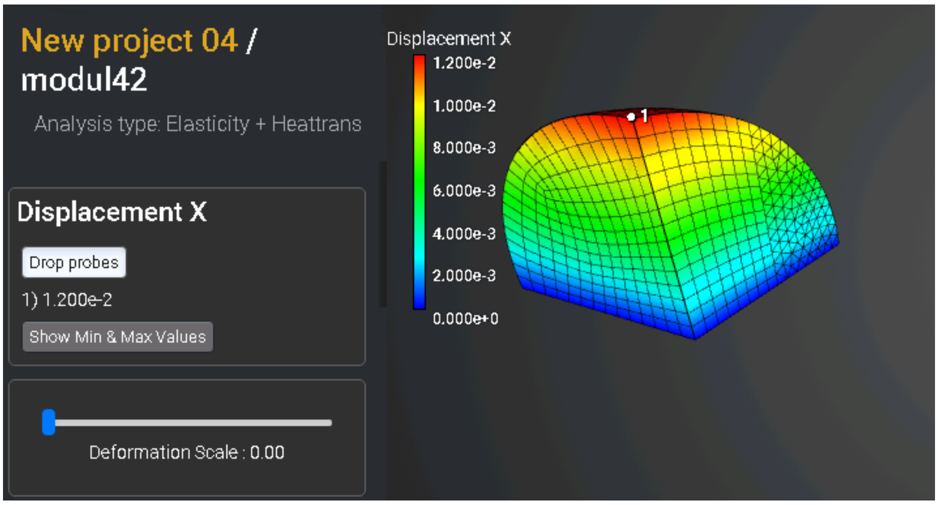


Fig 3.17 - Result of displacements X

Reference

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

3.8.Test case No3.3

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

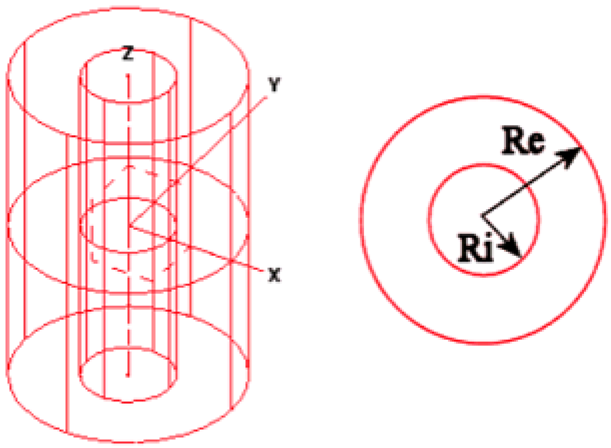


Fig 3.18 - Geometric model of a hollow cylinder

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

No	Value	Description	Unit	Target
1	Temperature at a point (0.3,0,0)	Temperature	℃	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	℃	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	℃	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	℃	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	℃	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	℃	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

No	Value	Description	Unit	Target	ProveDesign Results	Error, %
1	Temperature at a point (0.3,0,0)	Temperature	℃	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	℃	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	℃	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	℃	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	℃	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	℃	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

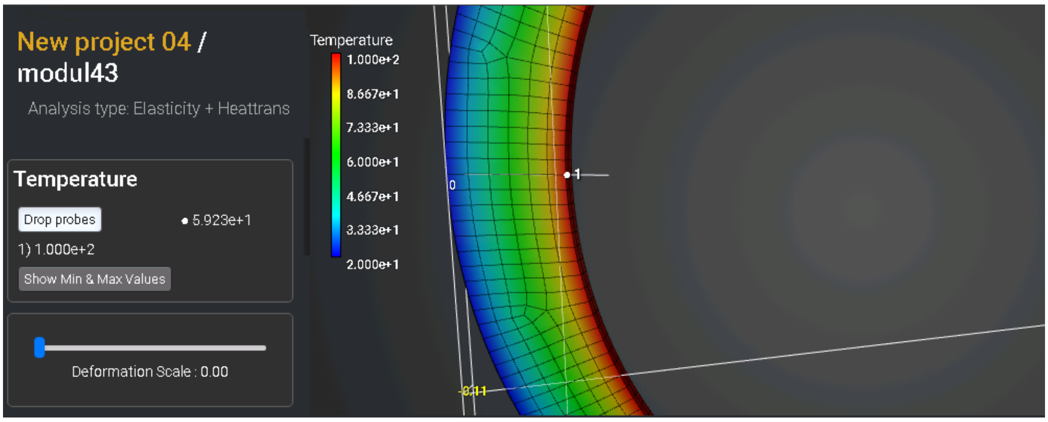


Fig 3.19 - Temperature at a point (0.3, 0, 0)

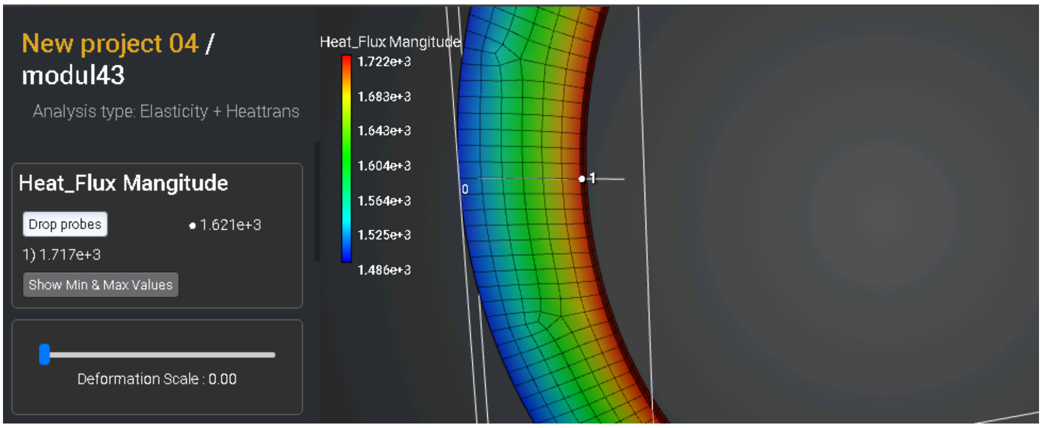


Fig 3.20 - Heat flux at a point (0.3, 0, 0)

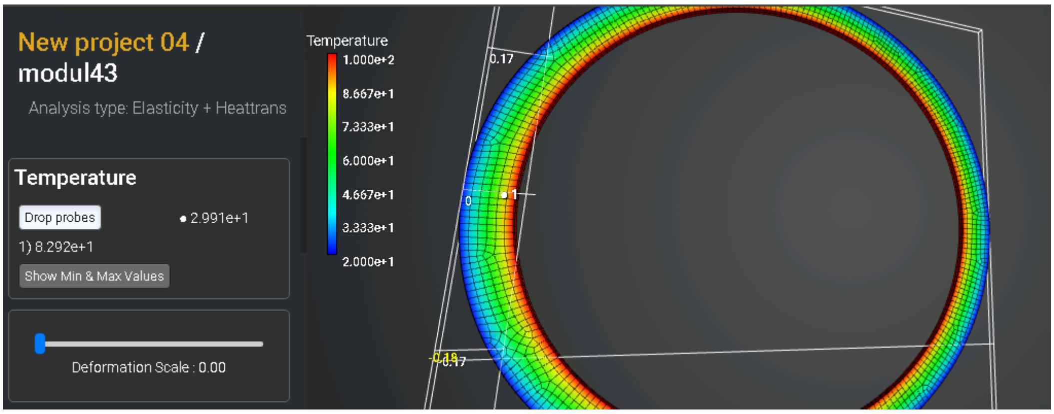


Fig 3.21 - Temperature at a point (0.31, 0, 0)

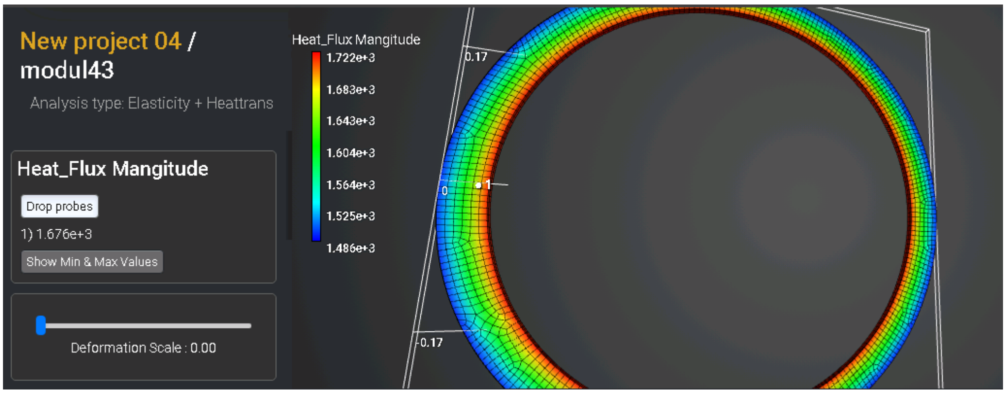


Fig 3.22 - Heat flux at a point (0.31, 0, 0)

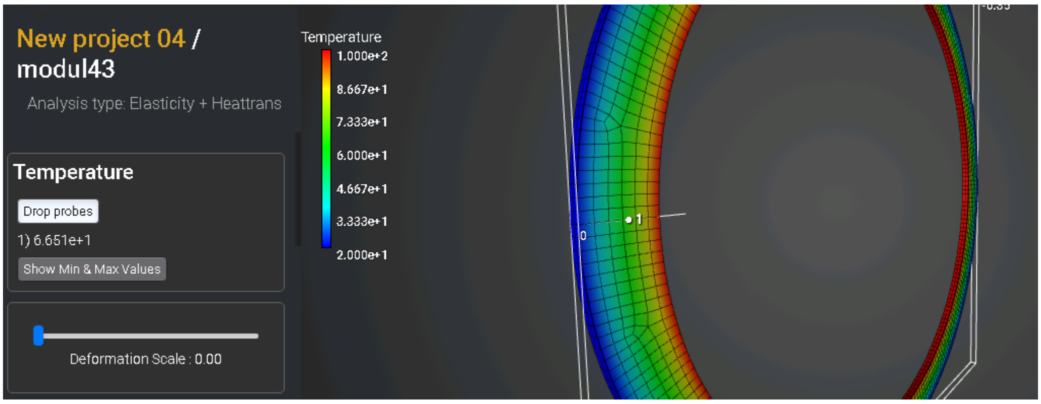


Fig 3.23 - Temperature at a point (0.32, 0, 0)

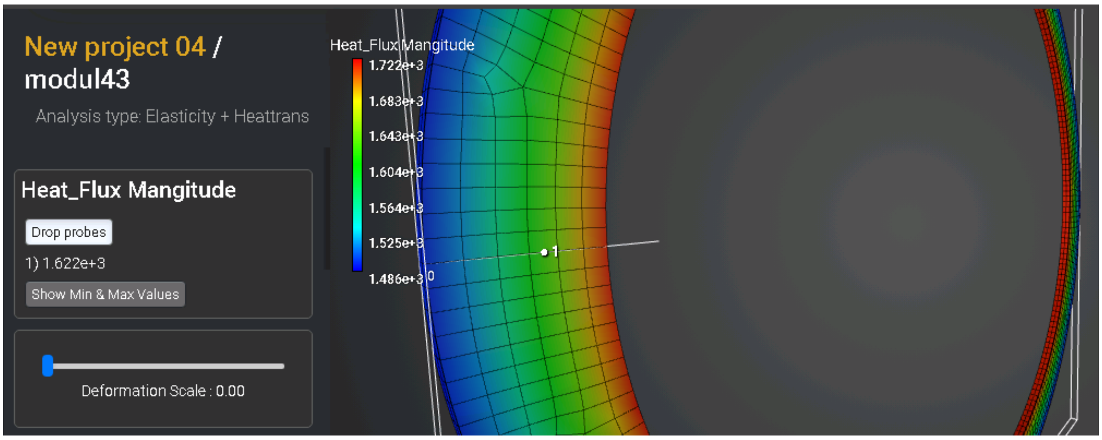


Fig 3.24 - Heat flux at a point (0.32, 0, 0)

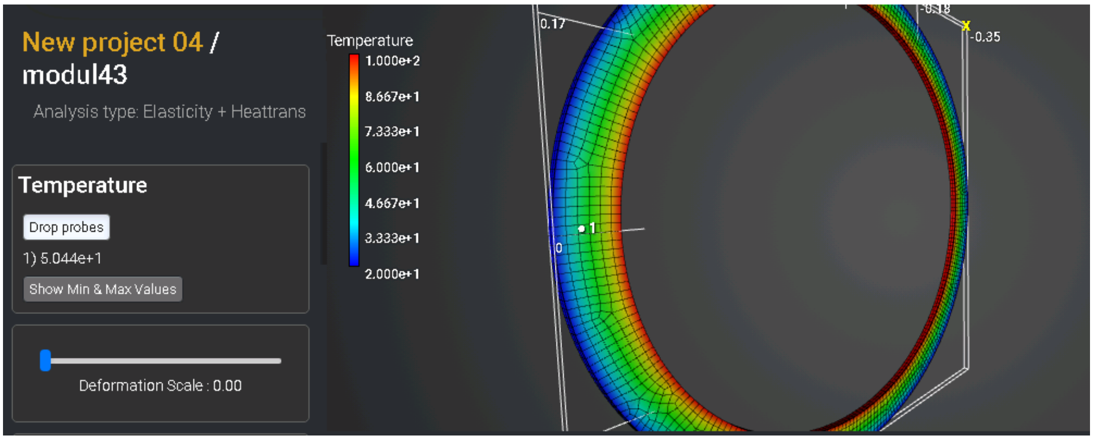


Fig 3.25 - Temperature at a point (0.33, 0, 0)

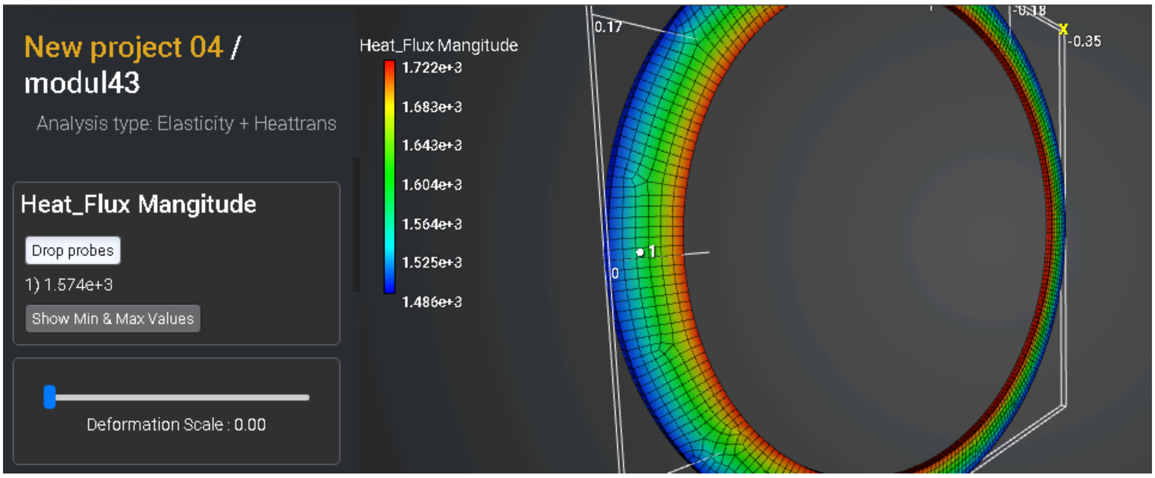


Fig 3.26 - Heat flux at a point (0.33, 0, 0)

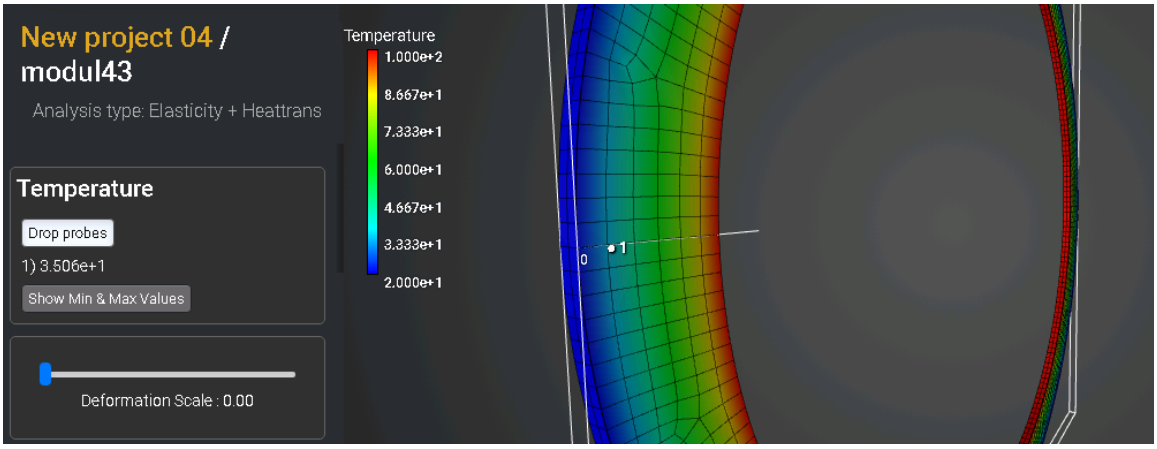


Fig 3.27 - Temperature at a point (0.34, 0, 0)

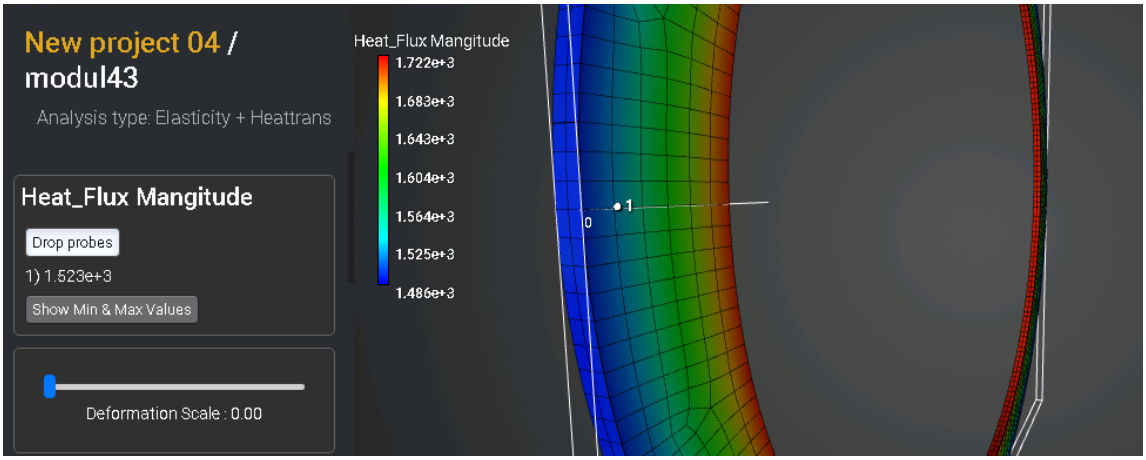


Fig 3.28 - Heat flux at a point (0.34, 0, 0)

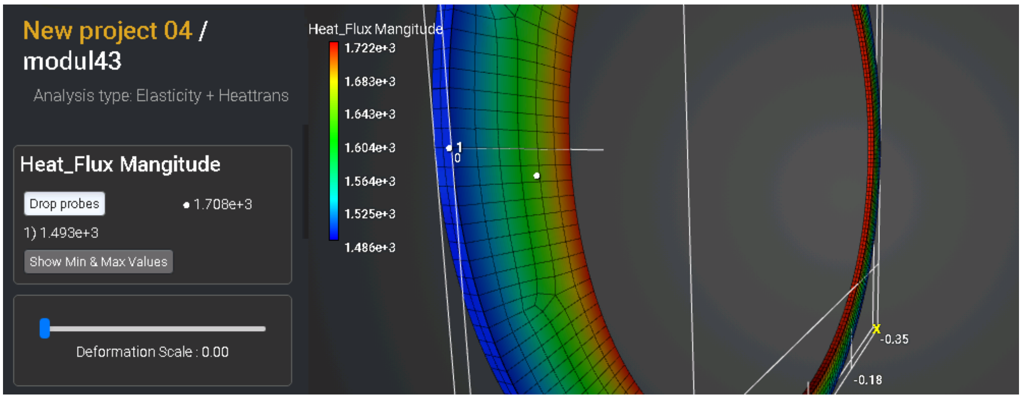


Fig 3.29 - Heat flux at a point (0.35, 0, 0)

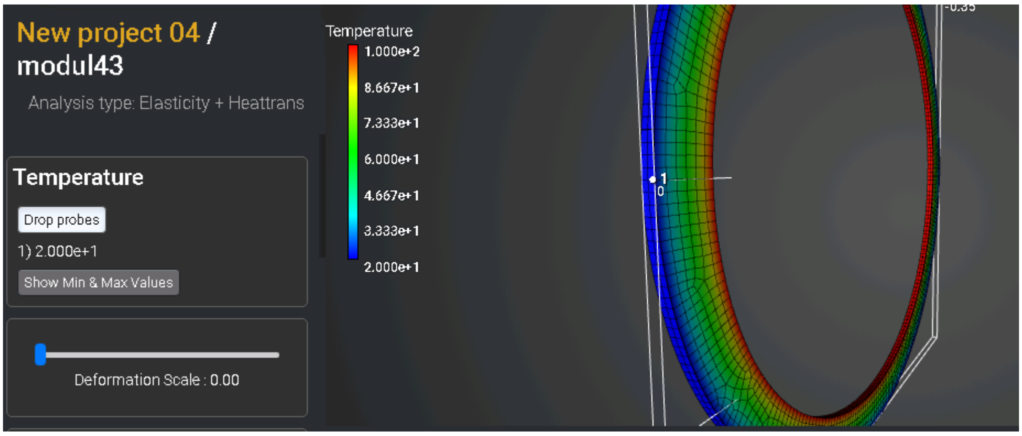


Fig 3.30 - Temperature at a point (0.35, 0, 0)

Reference

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

3.9.Test case No3.9

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

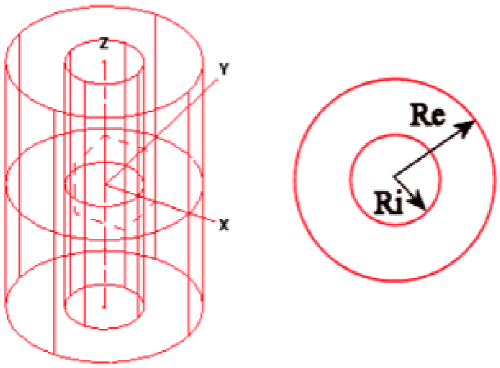


Fig 3.31 - Geometric model of a hollow cylinder

Geometrical model:

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

Boundary conditions:

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

Material Properties:

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

Mesh:

- Tetrahedrons of the second order.

Contact:

- Tied;
- Method: auto.

Calculation settings:

- Static calculation;
- Thermal conductivity.

Output Values

No	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

No	Value	Description	Unit	Target	ProveDesign Results	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

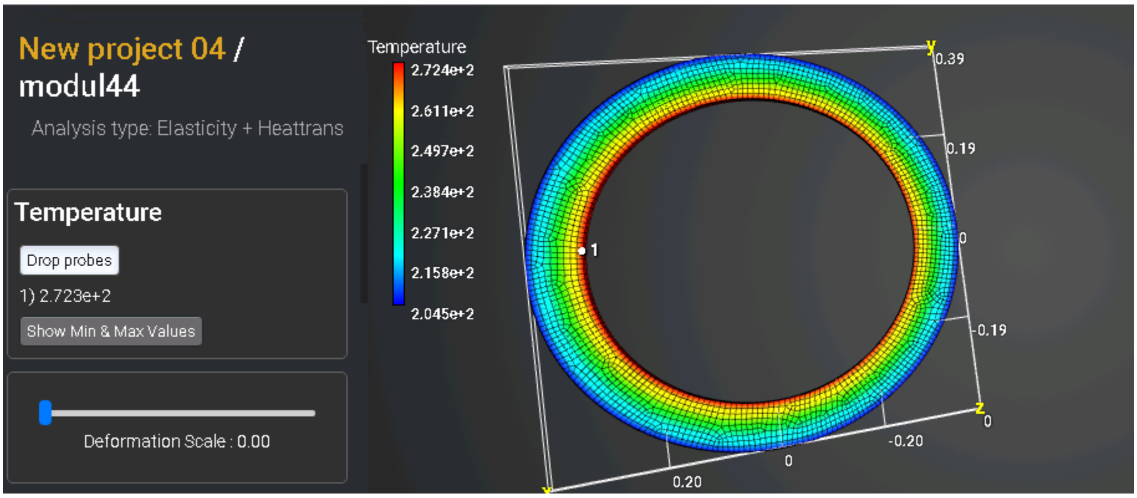


Fig 3.32 - Temperature at a point (0.3, 0, 0)

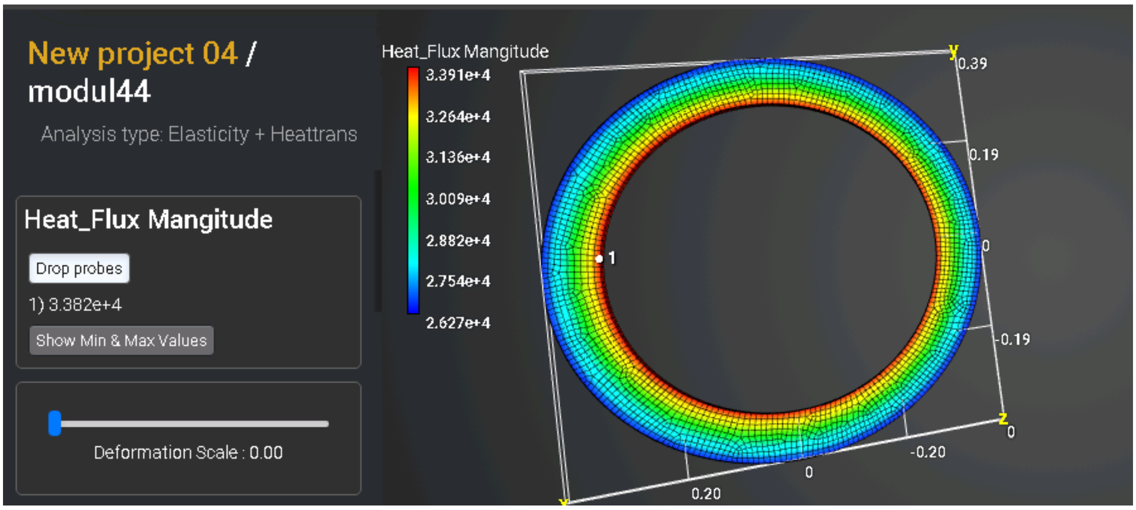


Fig 3.33 - Heat flux at a point (0.3, 0, 0)

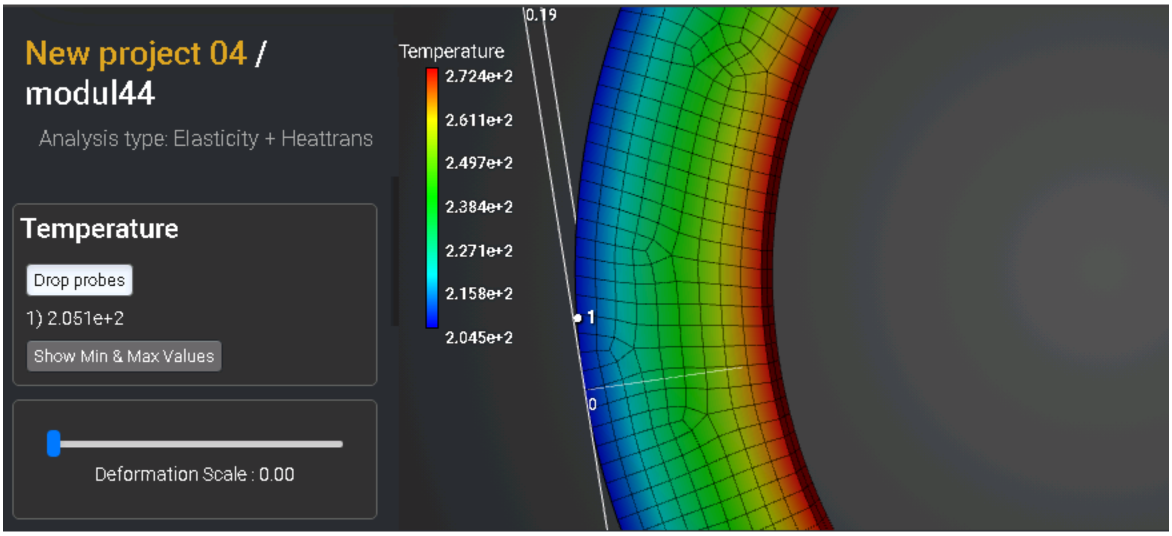


Fig 3.34- Temperature at a point (0.391, 0, 0)

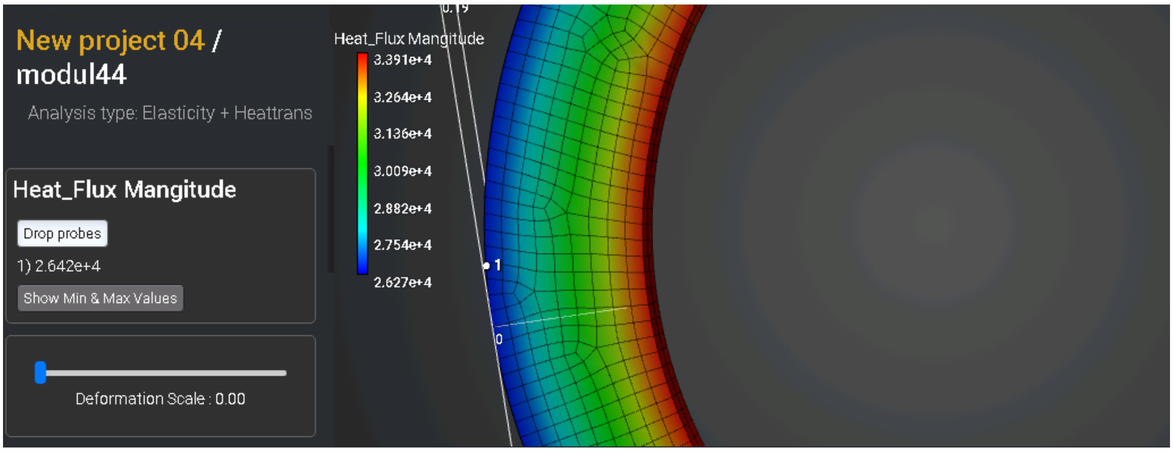


Fig 3.35 - Heat flux at a point (0.391, 0, 0)

Reference

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

3.10. Test Case No 3.10

Problem Description

Проверка правильности расчета нагружения полого цилиндра внутренним давлением 24 Н/мм^2. Для задания закона пластичности используется упрочнение. \

Input values

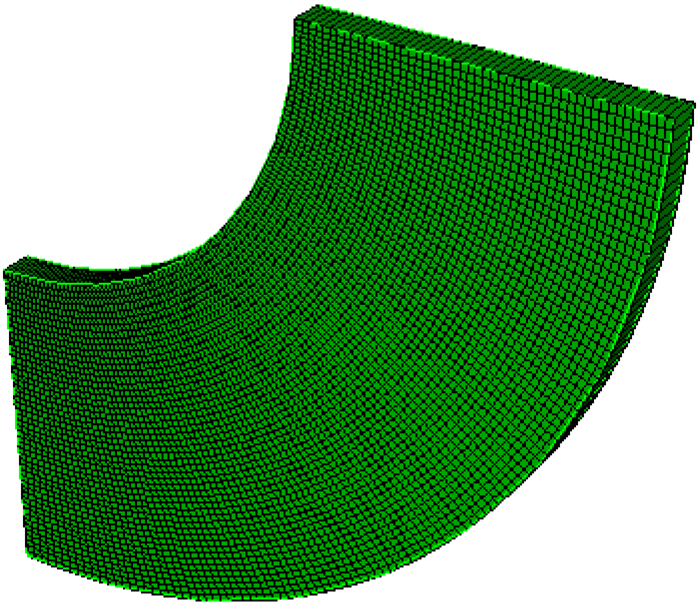


Fig 2.55 – Geometry model

Geometry model:

- Considered ¼ of the model;
- Height H=100 mm;
- Radius $R_i = 100$ mm;
- Radius $R_e = 200$ mm.

Boundary condition:

- Pressure 24 MPa applied in the inner surface;
- Symmetry.

Material:

- Young’s modulus E = 21000 Pa;
- Poisson ratio ν = 0.3;
- Ultimate strangth = 4219.2;
- Yield strength = 24 Pa;
- Ultimate strain = 1.

Calculation settings:

- Static;
- Plasticity.

Output Values

№	Value	Description	Unit	Target
1	Stress σ_{Mises} at the point (100, 0, 0)	σ_{Mises}	MPa	39.564
2	Displacement u_x at the point (100, 0, 0)	u_x	mm	0.4044
3	Stress σ_{Mises} at the point (100, 0, 0)	σ_{Mises}	MPa	24.027
4	Displacement u_x at the point (100, 0, 0)	u_x	mm	0.233

Numerically approximate analytical solution

The reference solution is from Nafems [1].

In addition, a comparison was made with the numerical solution in the Ansys package.

Ansys Script:

```
FINISH
/CLEAR
/PREP7

MPTEMP,1,0
et,1,plane183
KEYOPT,1,3,2
MPDATA,EX,1,,2.1e+4
MPDATA,PRXY,1,,0.3
TB,BISO,1,,,
TBMODIF,1,1,0
TBMODIF,2,1,24
TBMODIF,3,1,4200

PCIRC,200,100,0,90

AESIZE,1,5
!MSHAPE,1
AMESH,1

DL,2,1,UX,0
DL,4,1,UY,0
SFL,3,PRES,24   !500 for findefs

/SOL
ANTYPE,0
NLGEOM,0
NSUBST,10,30,10
!OUTRES,ERASE
!OUTRES,ALL,ALL
```

```
!RESCONTRL,DEFINE,ALL,ALL,1
TIME,1
SOLVE
```

Reference:

[1] NAFEMS R0072 Introduction to Non-Linear Finite Element Analysis (Plasticity example 2: 2D Plane stress, случай изотропного упрочнения)

Results in Prove.Design

№	Value	Description	Unit	Target	ProveDesign Results	Error, %
1	Stress σ_{Mises} at the point (100, 0, 0)	σ_{Mises}	MPa	39.564	39.8472	0.72
2	Displacement u_x at the point (100, 0, 0)	u_x	mm	0.4044	0.403928	0.12
3	Stress σ_{Mises} at the point (100, 0, 0)	σ_{Mises}	MPa	24.027	23.9906	0.15
4	Displacement u_x at the point (100, 0, 0)	u_x	mm	0.233	0.232638	0.16

3.11. Test Case No3.11.

Problem Description

The elastic-plastic equilibrium of a hollow ball under internal pressure is considered. By virtue of symmetry, the segment of the ball located in the first octant stands out.

Input values

Geometry model:

- Inner radius of the ball: a = 2.5 м
- Outer radius of the ball: b = 5 м;
- Due to the symmetry of the problem, 1/8 of the sphere is considered.

Boundary condition:

- On the coordinate planes, displacements along perpendiculars are equal to zero
- Pressure p = 30 Pa is applied on the inner surface

Material:

- Isotropic;
- E = 21e3 Н/м²;
- ν = 0.3;
- σy = 24 Н/м².

Calculation Settings:

- Static;
- Elasticity, Plasticity.

Output Values

№	Value	Description	Unit	Target
1	The X component of the displacement vector in the mesh nodes at a point (3,0,0)	Displacement X	m	4.219·10-3
2	The XX component of the stress tensor in the mesh nodes at a point (3,0,0)	Stress XX	Pa	-21.249

№	Value	Description	Unit	Target
3	The X component of the displacement vector in the mesh nodes at a point (4.5,0,0)	Displacement X	m	$2.165 \cdot 10^{-3}$
4	The XX component of the stress tensor in the mesh nodes at a point (3,0,0)	Stress XX	Pa	-2.908

Numerically approximate analytical solution

The stress-strain state is determined by the formulas [1]:

- In the plastic zone ($a \le r \le c$)

$$\sigma_{rr}(r) = 2\sigma_y \ln(r/a) - p, \qquad \sigma_{\phi\phi} = \sigma_{rr}(r) + \sigma_y,$$
$$\varepsilon_{rr} = \psi(r) \cdot \big(\sigma_{rr}(r) - \sigma(r)\big) + \mathrm{k} \cdot \sigma(r), \quad \varepsilon_{\phi\phi} = \psi(r) \cdot \big(\sigma_{\phi\phi}(r) - \sigma(r)\big) + \mathrm{k} \cdot \sigma(r),$$
$$u_{plast} = \varepsilon_{\phi\phi} \cdot r$$

where

$$\psi(r) = -2k + \left(\frac{1}{2G} + 2k\right) \cdot \left(\frac{c}{r}\right)^3, \qquad k = \frac{1-2\nu}{E}, \qquad \sigma(r) = \frac{1}{3}\big(\sigma_{rr}(r) + 2\sigma_{\phi\phi}(r)\big),$$

c – boundary of the plastic zone founded from the equation

$$\ln\left(\frac{c}{a}\right) - \frac{1}{3}\left(\frac{c}{b}\right)^3 = \frac{p}{2\,\sigma_y} - \frac{1}{3}$$

- In the elastic zone ($c \le r \le b$)

$$\sigma_{rr}(r) = p^* \cdot (1 - (b/r)^3)\,, \quad \sigma_{\phi\phi}(r) = p^* \cdot (1 + b^3/(2r^3))$$

1.1. $\varepsilon_{rr} = du_{elastic}/dr, \qquad \varepsilon_{\phi\phi} = u_{elastic}/r\,,$

where $p^* = \big(p - 2\sigma_y \ln(c/a)\big) \cdot \left(\frac{c^3}{b^3 - c^3}\right)$, $u_{elastic} = p^*.\left(k + \frac{b^3}{4Gr^3}\right)$

Reference:

[1] Л.М. Качанов. Основы теории пластичности. М., 1969г., 420 стр

Results in Prove.Design

№	Value	Description	Unit	Target	ProveDesign Results	Error, %
1	The X component of the displacement vector in the mesh nodes at a point (3,0,0)	Displacement X	m	$4.219 \cdot 10^{-3}$	$4.1953 \cdot 10^{-3}$	0.56
2	The XX component of the stress tensor in the mesh nodes at a point (3,0,0)	Stress XX	Pa	-21.249	-21.252	0.01
3	The X component of the displacement vector in the mesh nodes at a point (4.5,0,0)	Displacement X	m	$2.165 \cdot 10^{-3}$	$2.150 \cdot 10^{-3}$	0.71
4	The XX component of the stress tensor in the mesh nodes at a point (3,0,0)	Stress XX	Pa	-2.908	-2.90437	0.12

4. Contacts

http://www.cae-fidesys.com

support@cae-fidesys.com

+7 (495) 177-36-18