MINISTRY OF EDUCATION AND SCIENCE OF THE RUSSIAN FEDERATION

FEDERAL STATE AUTONOMOUS EDUCATIONAL
INSTITUTION OF HIGHER EDUCATION
«SAMARA NATIONAL RESEARCH
UNIVERSITY named after academician S.P. KOROLEV»
(SAMARA UNIVERSITY)

A.O. SHKLOVETS

WORK IN CAE-PACKAGE «ANSYS MECHANICAL»: STRUCTURAL ANALYSIS BY THE FINITE ELEMENT METHOD

Recommended the editorial-and-publishing committee of the Institute of engine and power plant engineering as textbook of speciality 24.04.05, 24.03.05 Aircraft Engines

SAMARA
Publisher of the Samara University
2017

UDC 621.787
ББК 39.55
S

Reader: professor V. B. Baljakin doctor of engineering science

Shklovets, Aleksandr Olegovich
S **Work in CAE-package «Ansys Mechanical»: structural analysis by the finite element method: textbook** / *A. O. Shklovets.* – Samara: Publisher of the Samara University, 2017. – 44 p.

ISBN 978-5-7883-1363-4

The tutorial describes the work in the software package for the finite element analysis Ansys. Examples of structural analysis for various structures are given. Calculations using beam, shell and volume finite elements are considered. The basic methods for creating the geometry of the model, generating a finite element mesh are described. Various types of loads applied in Ansys are described.

This textbook is proposed for master students and bachelors of speciality 24.04.05, 24.03.05 Aircraft Engines.

Get ready at the Department of Construction and Design of Aircraft Engines.

UDC 621.787
ББК 39.55

ISBN 978-5-7883-1363-4

Contents

INTRODUCTION

Many of the problems that researchers and engineers currently have to deal with do not lend themselves to an analytical solution or require huge problems for experimental implementation. Often the only possibility of express analysis of an engineering problem is computer mathematical modeling. Progress in the development of numerical methods has made it possible to significantly expand the range of problems available to analysis. In practice, the results obtained on the basis of these methods are used in virtually all fields of science and technology.

In the analysis of constructions, the finite element method finds its most important application. In structural analysis, constructions are bridges, buildings, ship hulls, aircraft components, machine parts, pistons, tools, in other words, any engineering structures.

The primary variables that are calculated during the construction analysis are the displacements. Further, based on the calculated displacements at the mesh nodes, other important parameters are determined, such as stresses, elastic or plastic deformation, etc.

In this tutorial basic position of computer engineering analysis discussed in the examples solved using Ansys Mechanical that belongs to the class «heavy» systems and has advanced computing capabilities.

1 PREPARING FOR WORK AND LAUNCHING ANSYS

1.1 Create new folder on the disk

1) **Start Menu – My Computer – Teacher specified disk – Right mouse button – Create – Folder**
2) **Enter the name of the folder (name of the student).**
3) **Close Menu.**

1.2 Launching Ansys

1) **Start Menu → Programs → Ansys 13.0 → Mechanical APDL Product Launcher.**

The menu is shown in Fig. 1.1. In the **Working Directory** column, specify the path to the previously created working folder. In the **Job Name** column, enter the name of the task (you can leave it by default). Click **Run**.

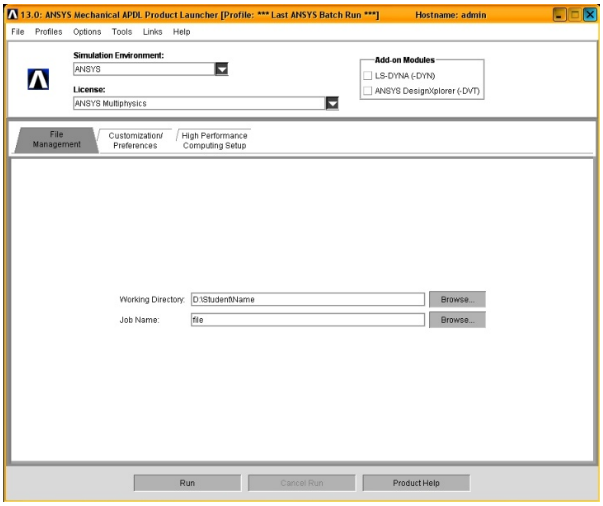

Fig. 1.1. Ansys Mechanical Start menu

2) After entering the interactive mode, the user's dialogue with the program is carried out through a multi-window "**Graphical User Interface (GUI)**" (Fig. 1.2). Briefly describe some of them.

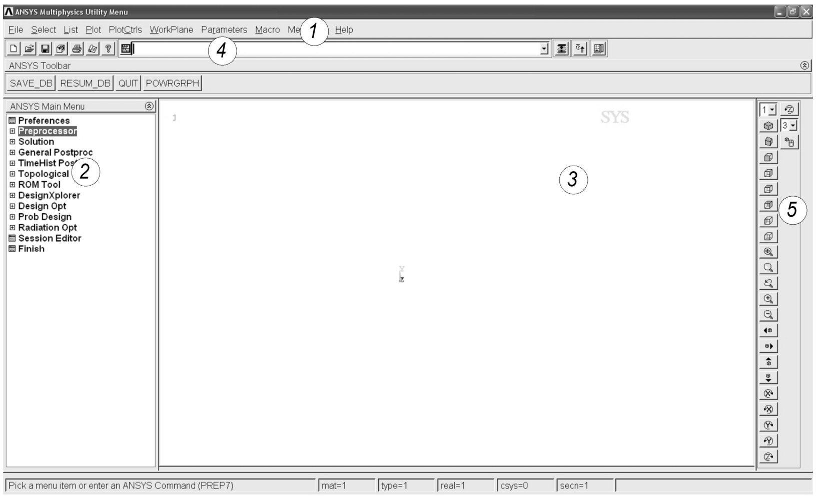

Fig. 1.2. Graphical User Interface

The top horizontal window 1 is the **Utility Menu (UM)**, contains a set of often used commands for working with the model, such as saving and running models, displaying various model elements, working with coordinate planes, etc.

Window 2 is the **Main Menu (MM)**. It contains the main functions and stages of program execution, which are grouped into pop-up (dynamic) menus, which are dependent on the progress of the program.

Window 3 is the **Graphic window**. It is an area for displaying such graphical information as a finite element model or graphs of analysis results.

Window 4 is an **Input window (Ansys input)**. The area for the command set and the output of messages in the **Output Window**. It is possible to access the list of previously entered commands. Commands duplicate actions that are conducted interactively.

Window 5 allows you to control the position of geometry in space, zoom in and out of the model, rotate, etc.

2 STRUCTURAL ANALYSIS OF A BEAM WITH I-SECTION AND DISTRIBUTED LOADS

We consider the classical problem of strength of materials about direct bending in one plane of an elastic multispan beam with piecewise constant bending stiffnesses. External force factors causing such a bend may be concentrated forces, moments and distributed loads with a piecewise linear change in intensity.

2.1 Modeling of points and lines that define beam

1) As an example, take beam with I-section (Fig. 2.1).

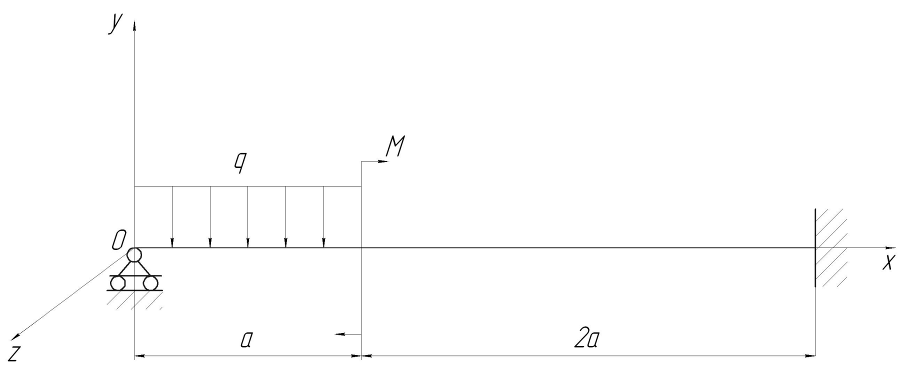

Fig. 2.1. Beam geometry

This beam is located relative to the Cartesian coordinate system OXYZ so that the x-axis is directed along the axis of the beam (Fig. 2.1). The y- and z-axes are directed along the main axes of inertia of the section, and the zero coordinate is located at the left end of the beam. Direct transverse bending is carried out in the OXY plane. The characteristics of the cross-section (Fig. 2.2) are taken from GOST 19425-74, we select a beam with the profile number 20C:

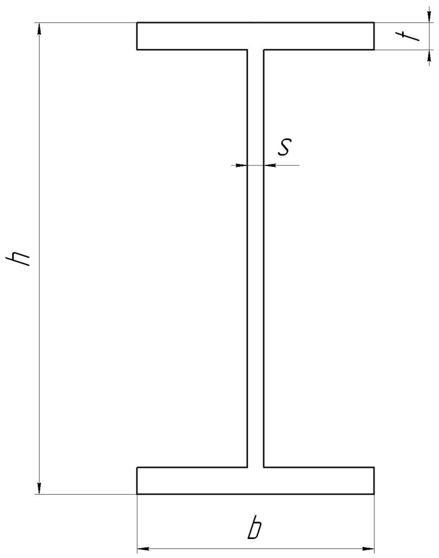

Fig. 2.2. I-section of beam

Tab. 2.1 – Initial data

Beam parameters	a, м	q, kN/м	M, kN·м	
	2	40	20	
Section parameters	h, мм	b, мм	S, мм	t
	200	100	7	11,4

2) Create the start and end points defining the beam, as well as the point to which the load is applied: **PrPr → Modeling → Create → Keypoints in Active CS**. In the window that appears, enter the coordinates of the points:

	x	y	z
1	0	0	0
2	2	0	0
3	6	0	0

3) Connect the resulting points in two segments: **PrPr → Modeling → Create → Lines → Lines → Straight Line.** Select points 1 and 2 → **Apply**, then 2 and 3 → **OK** .

2.2 Creating a finite element model

1) Element type: **PrPr → Element Type → Add/Edit/Delete**. Select the element type **Beam 188 → OK**. You must specify the cross-section type: **PrPr → Sections → Beam → Common Sections**, select the I-section on the **Sub-Type** (Fig. 2.3)

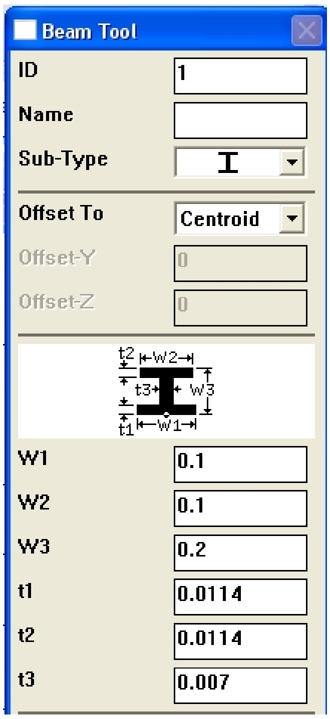

Fig. 2.3. Beam Tool

2) Material properties. **PrPr → Material Props → Material Models**. Set: modulus of elasticity EX=$2 \cdot 10^{11}$ (Па); Density DENS = 7800 (kg/$м^3$); Poisson's ratio PRXY=0,3;

3) To orient the section of the beam in space, an orientation point is given. To orient the I-section along the Y-axis (Fig. 2.4), create a point 4 (0,1,0).

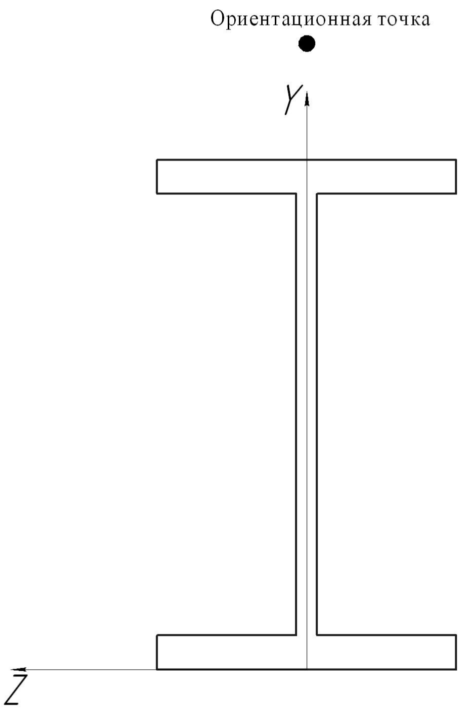

Fig. 2.4. Orientation point location

4) Specify the attributes of the created finite elements: **PrPr → Meshing → Mesh Attributes → All Lines**. In the column **Pick Orientation Keypoint** click **Yes**. Select point 4 as the orientation;

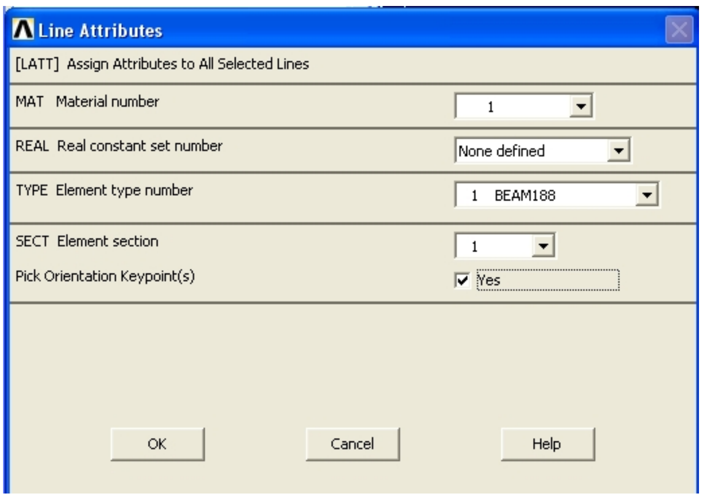

Fig. 2.5. Finite Element Attributes

5) Specify the number of elements to be created on the lines. **PrPr → Meshing → Size Cnrtls → ManualSize → Lines**. Select **L1, NDIV=10 → Apply**. For **L2 NDIV=20**

6) Generate the finite element mesh: **PrPr → Meshing → Mesh–Lines → Pick All**. To display the finite elements of a beam type, you need to do the following: **UM → PlotCtrls → Style → Size and Shape → Display of Element → On**

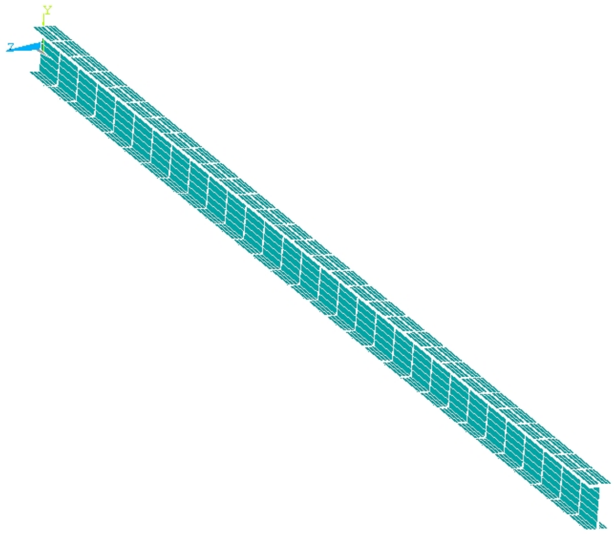

Fig. 2.6. Finite Element Beam Model

2.3 Application of boundary conditions and loads

1) Simulate the hinged movable support (UY=UZ=0): **PrPr → Loads → Define Loads → Apply → Structural → Displacement → On Keypoints**. **Select point 1 – Apply**. In the drop-down menu, select UY, then select UZ. Simulate a fixed-end: **PrPr → Loads → Define Loads → Apply → Structural → Displacement → On Keypoints**. **Select point 3 → All DOF → OK**;

2) Simulate a linear load. A positive direction of linear load is the direction opposite the OY axis. Select the elements belonging to line 1: **UM → Select → Entities → Lines By Num/Pick → Line 1**. Then, instead of **Lines**, **select Elements, Attached to Lines, Reselect** (only the selected line is used) **→ OK** (Fig. 2.7). After clicking **Plot Elements**, you will see only the selected items; To apply the load: **PrPr → Loads → Define Loads → Apply → Structural → Pressure → On Beam → Pick All**, **VaII=40000 →** ***OK***. Select now the hidden lines and elements: **UM → Select → Everything;**

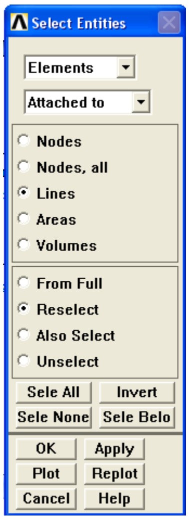

Fig. 2.7 – Select elements

3) Apply the torque to the point. Positive is the moment directed counterclockwise: **PrPr → Loads → Define Loads → Apply → Structural → Force/Moment →On Keypoints** , select point 2, set **MZ=-20000**

2.4 Running on the calculation and analysis of results

1) Run on calculation. Set the analysis type: **MM → Solution → Analysis Type → New Analysis – Static**. Run the solution: **Solution → Solve → Current LS → OK**.

2) Display the deformation of the model relative to the OY axis (Figure 2.7), combined with an undeformed model: **MM → General Postproc → Plot Results → Contour Plot → Nodal Solu → DOF Solution → Y-Component of displacement**, in the column **Undisplaced shape key Deformed shape with undeformed model**

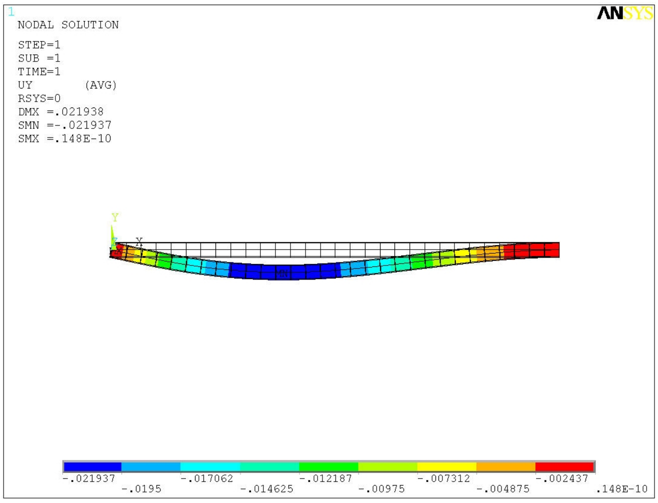

Fig. 2.8. Деформация модели относительно оси OY

3) Construct the curve of shearing forces Qy. The shearing force in Ansys has the opposite sign in comparison with the curve of shearing forces adopted. Using **Help** for the **Beam188**, we find the necessary commands for setting curves. To construct curve of shearing forces, enter on the command line as follows:

ETABLE,QYI,SMISC,5

ETABLE,QYJ,SMISC,18

PLLS,QYI,QYJ

To construct bending-moment curve, enter on the command line as follows:

ETABLE,MZI,SMISC,2

```
ETABLE,MZJ,SMISC,15
PLLS,MZI,MZJ
```

3 CALCULATION OF CONNECTED BARS LOADED WITH TEMPERATURE LOAD AND TENSILE FORCE.

Consider a system of bars connected with each other by a beam:

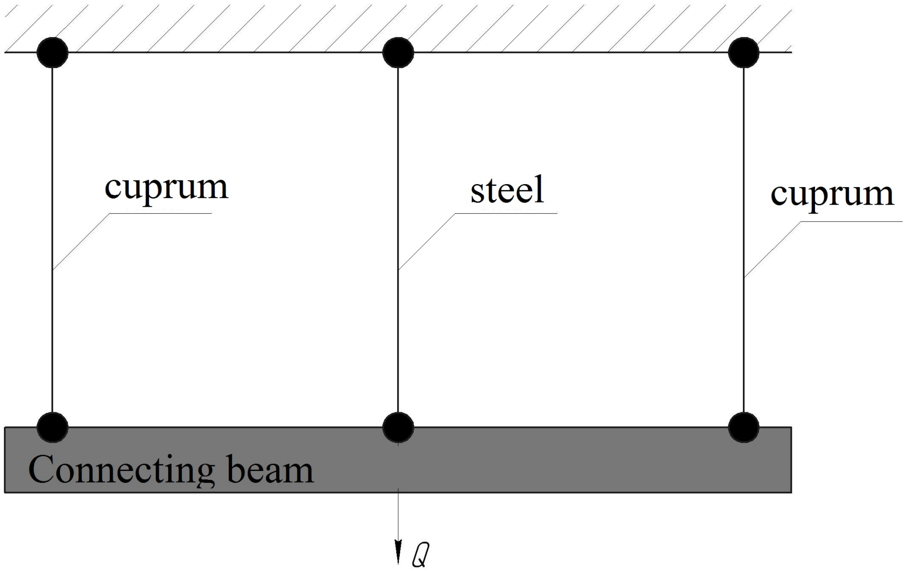

Fig. 3.1. System of bars

Three bars of the same length are joined together by a beam, the central bar is loaded with a tensile force $Q=10$ kN. In addition, the bars are loaded with a temperature gradient. The two bars are made of copper, the central bar of steel.

3.1 Geometry modeling

1) Create start and end points defining the bars: **MM → PreProcessor → Modeling → Create → Keypoints in Active CS**. In the window that appears, enter the coordinates of the points:

	x	y	z
1	0	0	0
2	0.04	0	0
3	0.08	0	0
4	0	0.08	0
5	0.04	0.08	0
6	0.08	0.08	0

2) Create line between points 1 and 4, 2 and 5, 3 and 6 (Fig. 3.2)

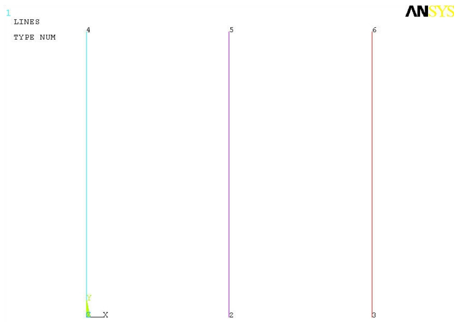

Fig. 3.2. Construction of lines

MM → PreProcessor → Modeling → Create → Lines → Lines → Straight Line. Select points 1 and 4, → **Apply**, then 2 and 5 → **Apply**, then 3 and 6 → **OK**.

3.2 Finite element type and material properties

1) Element type: **MM → Preprocessor → Element Type → Add/Edit/Delete**. Select the element type **Beam 188 → OK**. Specify the type of cross section: **MM → Preprocessor → Sections → Beam → Common Sections**, select rectangular cross-section, setting B=H= 0,001 м;

2) Set the material properties of copper and steel. **MM → Preprocessor → Material Props → Material Models**. For steel: $E=2\cdot10^{11}$ (Pa); density 7800 (kg/m^3); Poisson's ratio 0,3; coefficient of thermal expansion (**Structural – Thermal Expansion – Secant Coefficient**) $\alpha=11{,}85\cdot10^{-6}$ 1/^{0}C. For copper: $E=1{,}2\cdot10^{11}$ (Pa); density 8950 (kg/m^3); Poisson's ratio 0,3; coefficient of thermal expansion $\alpha=16{,}8\cdot10^{-6}$ 1/^{0}C;

3) Specify the number of elements to be created on the lines. **MM → PrPr → Meshing → Size Cnrtls → ManualSize → Lines**. For all lines, specify 10 elements;

4) Generate a finite element mesh on copper bars: **PrPr → Meshing → Mesh Attribute → Picked Lines**. Select line 1 and 3, in the appeared window in the **Material number** column, set 2 (copper material number), then select line 2 and set material number 1. Create the finite elements along the lines. To display the finite elements of a beam type, you need to do the following: **UM → PlotCtrls → Style → Size and Shape → Display of Element → On**;

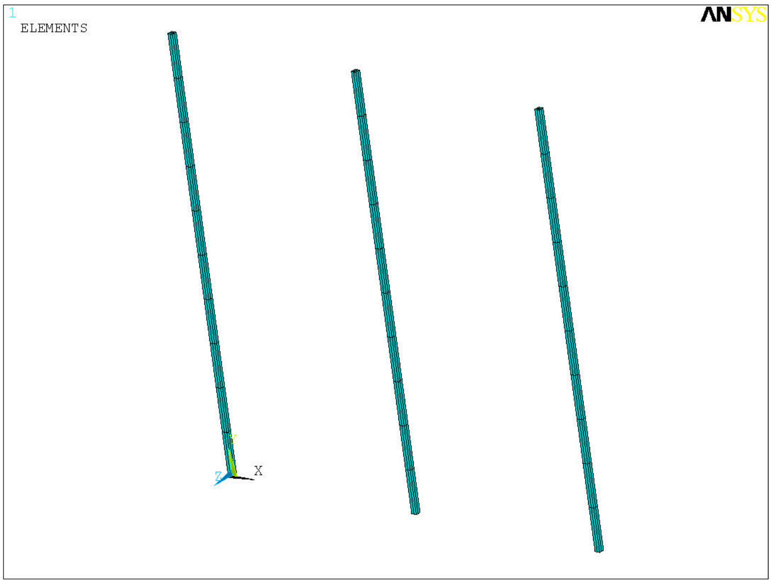

Fig. 3.3. Finite Element Model

3.3 Boundary conditions and load

1) Connect the displacement of ends of the bars in the direction of the axis Y. **MM → Prpr → Coupling/Ceqn → Couple DOFs**. Select nodes with numbers 2,13, 24, NSET set 1, Lab set UY;

2) Fix the bars in the base to all degrees of freedom: **PrPr → Loads → Define Loads → Apply → Structural → Displacement → On Nodes**. Select nodes with numbers 1, 12, 23 → All DOF → OK;

3) Apply a temperature load **PrPr → Loads → Define Loads → Apply → Structural → Temperature → On Lines**, set the temperature to 300 ° C for all lines;

4) Apply a concentrated force in the direction of Y-axis to the end of steel bar: **PrPr → Loads → Define Loads → Apply → Structural → Force/Moment**, select nodes 24, set FY=1000.

3.4 Running on the calculation and analysis of results

1) Run on calculation. Set the analysis type: **MM → Solution → Analysis Type → New Analysis → Static.** Run the solution **Solution → Solve → Current LS → OK**.

2) Compare the stresses arising in the steel bar with stresses arising in the copper bar. Run on the command line the following macro:

```
FINISH
/POST1
STEEL_N  = NODE (0.04,,,)
COPPER_N = NODE (0.08,0,0)
```

```
STEEL_E = ENEARN (STEEL_N)
COPPER_E = ENEARN (COPPER_N)
ETABLE,STRS_ST,LS,1
ETABLE,STRS_CO,LS,1
*GET,STRSS_ST,ELEM,STEEL_E,ETAB,STRS_ST
*GET,STRSS_CO,ELEM,COPPER_E,ETAB,STRS_CO
```

Do the following calculation on your own:

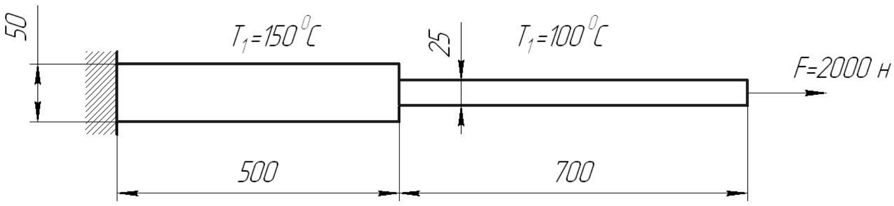

Fig. 3.4. Cantilevered beam

The cantilevered, unevenly heated beam with square section is loaded with axial force *F*. Determine the maximum tensile stresses in the fixed-end, as well as the axial displacement of the bar.

4 CREATING AREAS IN ANSYS

4.1 Creating area by points

1) Create points with coordinates:

	x	y	z
1	0	0	0
2	1	0	0.1
3	1	1	-0.1
4	0	2	0.3

2) Create area from the constructed points **MM → PrPr → Create → Areas → Arbitrary → through KPs;**
3) Consistently in order of increasing numbers, select created points;
4) Click **OK**.

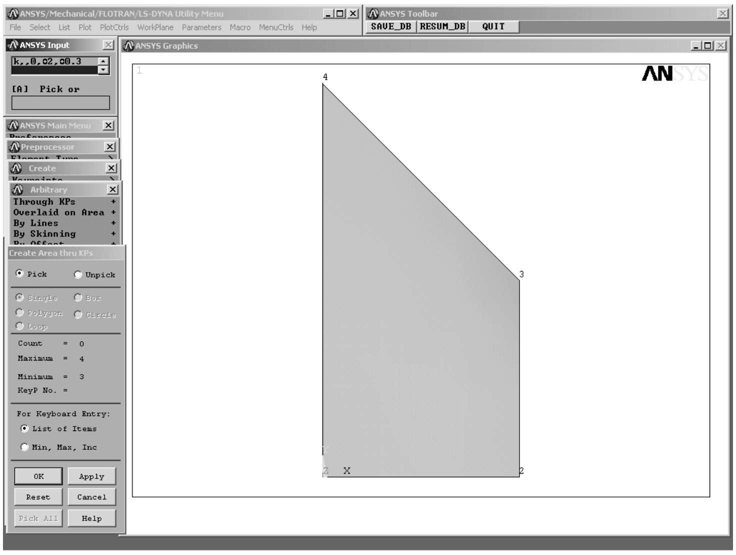

Fig. 4.1. Creating area by points

4.2 Delete area

1) **PrPr → Delete → Areas only**;
2) Select area to be deleted;
3) Click **OK**.

4.3 Creating area by lines

1) **UM → Plot → Lines**;
2) **MM → PrPr → Create → Areas → Arbitrarys By lines**;
3) Select lines for the area to be created;
4) Click **OK**;

5) Delete all object data: **UM → Files → Clear & Start New → OK – yes**.

4.4 Creating area by skinning

1) Create points with coordinates:

	x	y	z
1	-2.0	0.0	0.0
2	-1.0	0.5	0.0
3	0.0	0.8	0.0
4	1.0	0.4	0.0
5	2.0	0.0	0.0

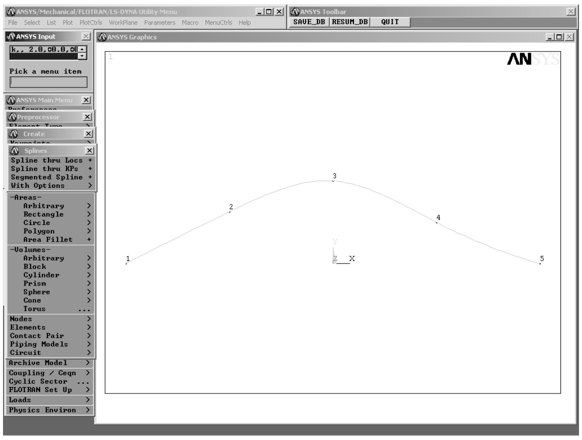

Fig 4.2. First section

2) Create a spline by points: **MM → PrPr → Create → Splines → Spline through KPs**.

3) Create points with coordinates:

	x	y	z
1	-2.0	-0.2	2.0
2	-1.0	0.6	2.0
3	0.0	0.8	2.0
4	1.0	0.5	2.0
5	2.0	-0.1	2.0

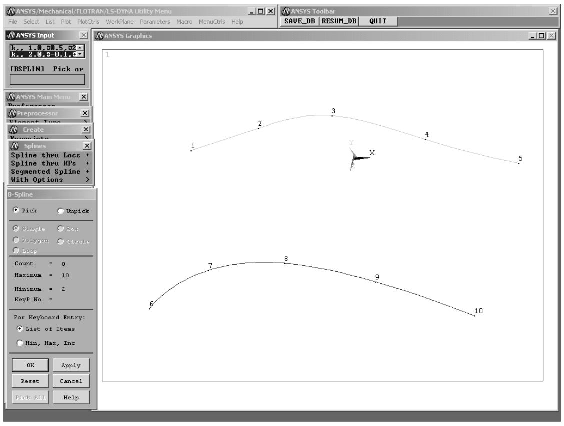

Fig 4.3. Second section

4) Create a spline by points.

5) Create points with coordinates:

	x	y	z
1	-2.0	0.1	3.0
2	-1.0	0.5	3.0
3	0.0	0.7	3.0
4	1.0	0.3	3.0
5	2.0	0.1	3.0

6) Create a spline by points.

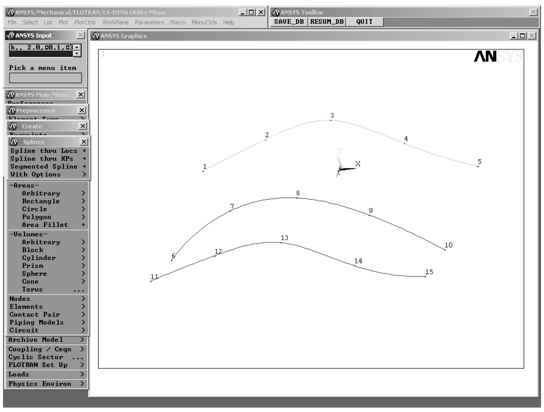

Fig 4.4 Third section

7) Create area by splines **MM → PrPr → Create → Areas → Arbitrary → By Skinning**.

8) Consistently select three created splines.

9) Click **OK**.

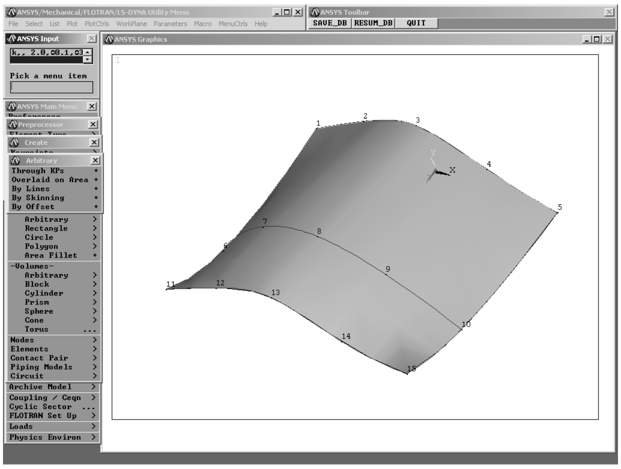

Fig 4.5. Area by section

10) Delete all object data.

4.5 Creating area by rotating the line around an axis

1) Create points with coordinates:

	x	y	z
1	-5	0	0
2	10	0	0
3	-5	2	0
4	-2	2	0
5	2	1	0
6	6	3	0
7	8	4	0
8	10	4.5	0

2) Create line between points KP3…KP4 и KP4…KP5.

3) Create spline by points KP5…KP8.

4) Display line numbers:

UM → PlotCtrls → Numbering → LINE → ON → OK;

UM → Plot → LINE .

5) Perform pairing of lines:

1. **PrPr → Create → Line Fillet**;
2. Select L2 и L3;
3. Click OK.
4. Set the radius of the pairing RAD = 5.
5. Click OK.

6. **UM → Plot → Replot**.

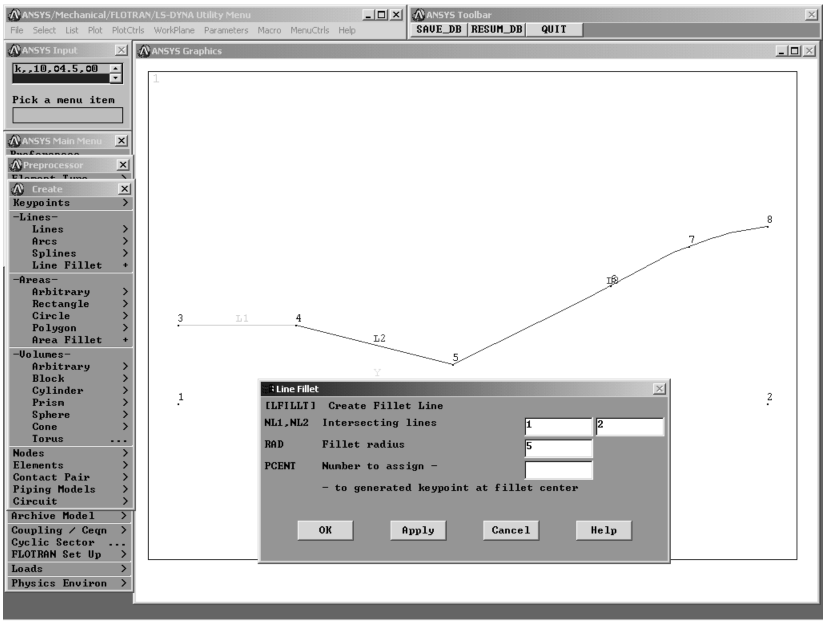

Fig. 4.6. Construction of fillets

6) Create area by rotating lines:

1. **PrPr → Modeling → Operate → Extrude → Lines → About Axis**.
2. Click **Pick All**.
3. Select KP1 and KP2, which determine the axis of rotation.
4. Click OK.
5. Set ARC=360.
6. Click OK.

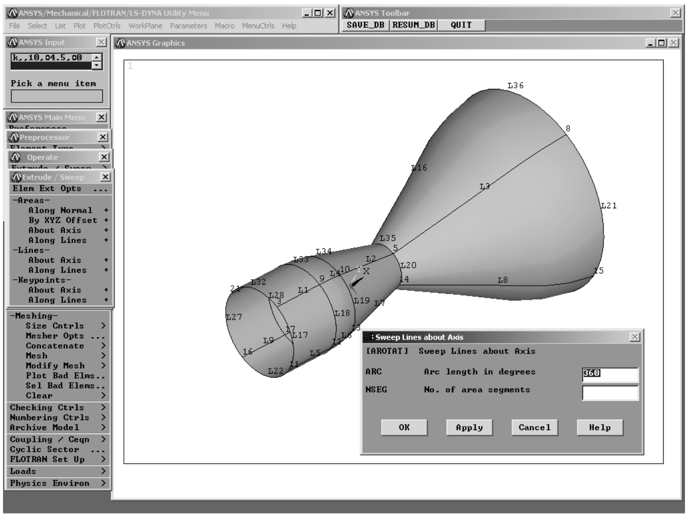

Fig 4.7. Area of rotation

7) Delete all object data.

4.6 Creating area as a circle segment or ring

1) **PrPr → Modeling → Create → Areas → Circle → Partial Annulus**.

2) Enter the coordinates of the circle center: wpx = 1; wpy = 1.

3) Enter the inner radius of the segment **Rad – 1 = 1**(To create segment of circle must be specified **Rad – 1 = 0**).

4) Enter the angle at which the radius is located relative to the x-axis, which defines the right boundary of the segment: **Theta –1 = 45**.

5) Enter the outer radius of the segment: **Rad – 2 = 2**.

6) Enter the angle at which the radius is located relative to the x-axis, which defines the left boundary of the segment: **Theta –2 = 180**.

7) Click OK.

8) Delete all object data.

4.7 Creating area in the form of circle or ring

1) **PrPr → Modeling → Create → Areas → Circle → Partial Annulus**.

2) Enter the coordinates of the circle center: wpx = 1; wpy =1.

3) Enter the inner radius of the circle **Rad – 1 = 0**.

4) Enter the outer radius of the circle **Rad – 2 = 2**.

5) Click OK.

6) Delete all object data.

5 MODELING OF SHELL FINITE ELEMENTS MESH

5.1 Select finite element type:

1) **PrPr → Element type → Add/Edit/Delete → Add**;

2) Select in the left window **Shell**;

3) Select in the right window **Elastic 4 node 181**;

4) Click OK;

5) Click Close.

5.2 Set material properties and thickness of element:

1) **PrPr → Material Props → Material Models**. For steel: $E=2\cdot10^{11}$ (Pa); density 7800 (kg/m^3); Poisson's ratio 0,3

2) **PrPr → Sections → Shell → Lay-up → Add/Edit**. Set: Thickness = 0,002 (m)

3) Click OK;

5.3 Creating area in the form of rectangle:

1) **PrPr → Modeling → Create → Areas → Rectangle → By Dimensions**;

2) Set: x1=-2(м); x2=2(м); y1=-1(м); y2=1(м);

3) Click OK.

5.4 Creating area in the form of circle with radius R = 0.5m and center located at the point with coordinates: x=0; y=0:

1) **PrPr → Modeling → Create → Areas → Circle → Solid Circle**;

2) Subtract the area A2 from the area A1 by the operation **Subtract**. **PrPr → Modeling → Operate → Booleans → Subtract → Areas.**

5.5 Set the created area attributes:

1) **PrPr → Meshing → MeshAttributes → Picked Areas**. Select created area;

2) Click OK;

3) Set: material number **MAT=1**; element type **TYPE = 1**; coordinate system type **ESYS=0**;

4) Click OK.

5.6 Set the edge size of the generated finite elements

1) **PrPr → Meshing → Size Cntrls → ManualSize → Lines → Picked Lines**. Select all lines, set **Size 0,001 м.**

5.7 Generate arbitrary finite element mesh:

1) **PrPr → Meshing → Mesh → Areas → Free → Pick All**.

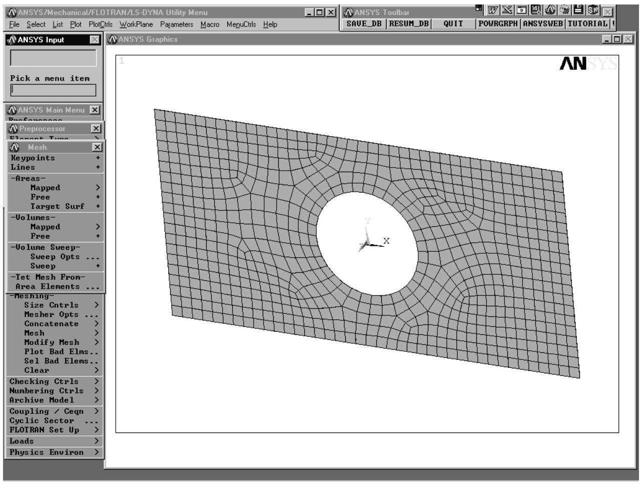

Fig. 5.1. Free mesh of finite elements

5.8 Grinding of mesh

PrPr → Modify Mesh → Elements.

1) Mark in the **Circle** menu **Refine mesh at elements**;

2) Move cursor to the center of hole, press left mouse button and, without releasing it, cover the circle with two layers of elements;

3) Put labels in the **Unpick** and **Single** menus **Refine mesh at elements**;

4) Cancel the cursor selection of randomly covered elements (leave only two layers of elements);

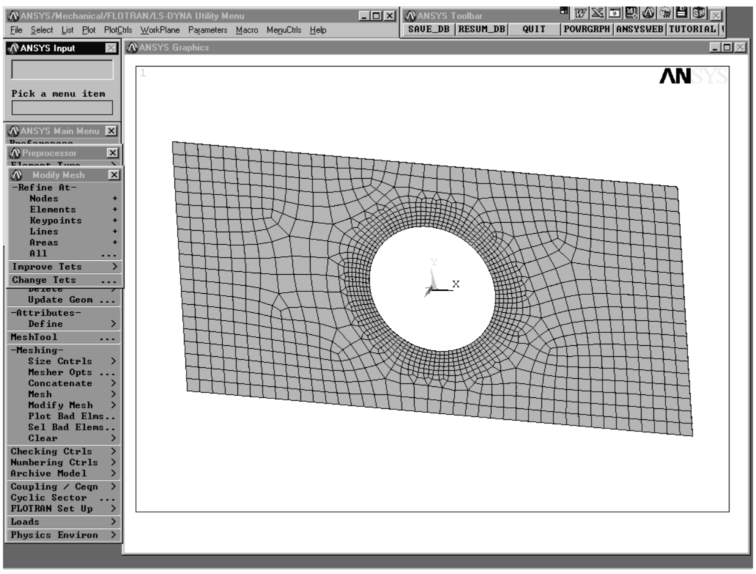

Fig 5.2. Grinding mesh

5) Click OK.

6) Set the grinding level LEVEL=2.

7) Click OK.

5.9 Cleaning area from mesh

1) **PrPr →Meshing → Clear → Areas**.

2) Select area on which the mesh of elements is cleaned.

3) Click OK.

4) **UM → Plot → Areas**.

5.10 Generating an ordered mesh of shell finite elements

1) Create point KP9 with coordinates: x=0; y=0; z=0.

2) Create lines between points KP9 and KP4, KP9 and KP3, KP9 and KP2, KP9 and KP1.

3) Subtract by operation **Divide** from area A3 of lines L11, L10, L9 and L12:

4) **PrPr → Operate → Divide → Areas by line**:

1. Select area A3;
2. Click OK;
3. Select area L9, L10, L11 and L12;
4. Click OK.

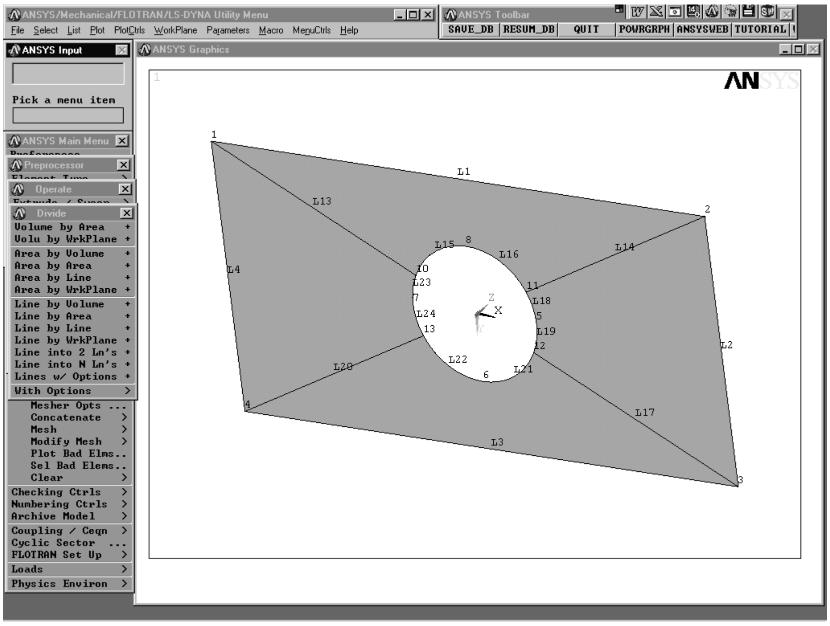

Fig. 5.3. Create intersecting lines

5) Perform a pairwise summation of lines forming the circumference of the hole to reduce their number:

1. **PrPr → Operate → Add → Lines**;

2. Select lines L15 and L16;

3. Click OK;

4. Click Apply;

5. Select lines L18 and L19;

6. Click OK;

7. Click Apply;

8. Select lines L21 and L22;

9. Click OK;

10. Select lines L24 and L23;

11. Click OK.

6) Specify the number of elements along lines **PrPr → Size Cntrls → All Lines**.

1.Set: NDIV=20.

2.Click OK.

7) Generate an ordered finite element mesh **PrPr → Mesh → Areas Mapped → 3 or 4 sided → Pick All**.

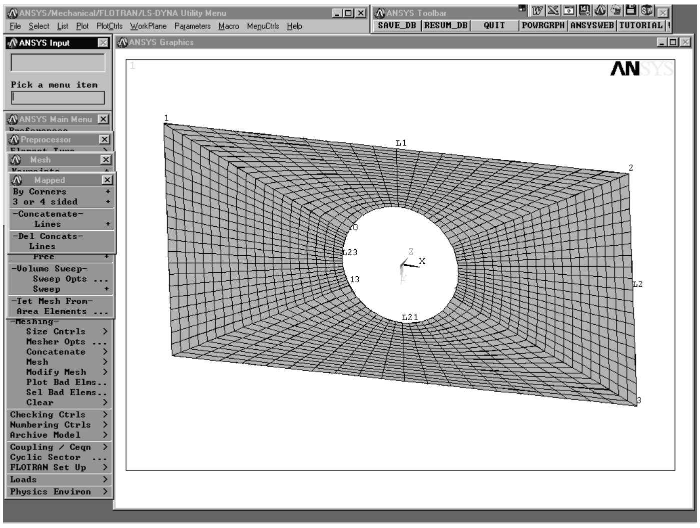

Fig.5.4. Ordered finite element mesh

6 VOLUME MESH OF FINITE ELEMENTS

Consider the calculation of the bladed disk of the compressor

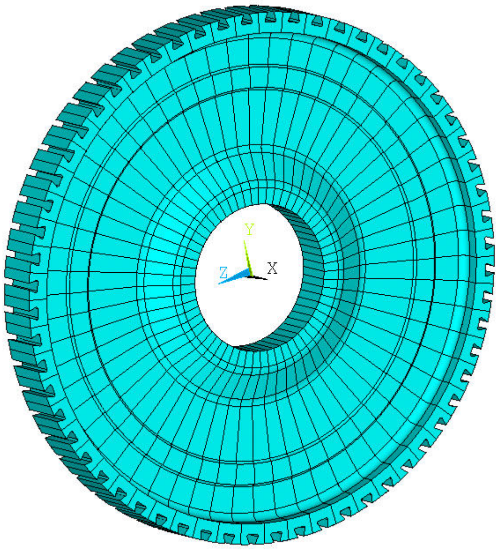

Fig. 6.1. Volume model of bladed disk

6.1 Creating a sectorial volume model of bladed disk

1) Create points that define the shape of the disk: **PrPr → Create → KeyPoints → in Active CS**

Номера точек	X	Y	Z
1	-0.025	0.09	0
2	-0.025	0.115	0
3	-0.004	0.155	0
4	-0.004	0.23	0
5	-0.007	0.230	0
6	-0.007	0.255	0
7	-0.004	0.255	0
8	-0.004	0.29	0
9	-0.014	0.29	0
10	-0.014	0.3	0
11	-0.03	0.3	0
12	-0.03	0.325	0

2) Since the bladed disk is symmetrical about the x-axis, reflect the second half of the points by the symmetry operation: **PrPr → Modeling → Reflect → all → Y-Z plane**;

3) Create two points with numbers 500 and 501 defining the axis of rotation of the disk with coordinates 500 (-0.01,0,0) , 501 (0.01,0,0);

4) Create lines between points KP10 and KP11, KP11 and KP12, KP12 and KP24, KP24 and KP23, KP23 and KP22, KP20 and KP19, KP19 and KP18, KP18 and KP17, KP17 and KP16, KP16 and KP15, KP15 and KP14, KP14 and KP13, KP13 and KP1;

5) Create arcs between points KP10 and KP8 with center at point KP9 and radius 0,01, and between points KP20 and KP22 with center at point KP21 and radius 0,01;

6) Create fillet radius of 0.02 between the following lines L1 and L2, L18 and L19, L2 L3, L18, L17. **PrPr → Modeling → Create → Lines → Line Fillet**;

7) Create fillet radius of 0,003 between the following lines L6 and L4, L13 and L17. In Fig. 6.2. shows the result of all the operations performed;

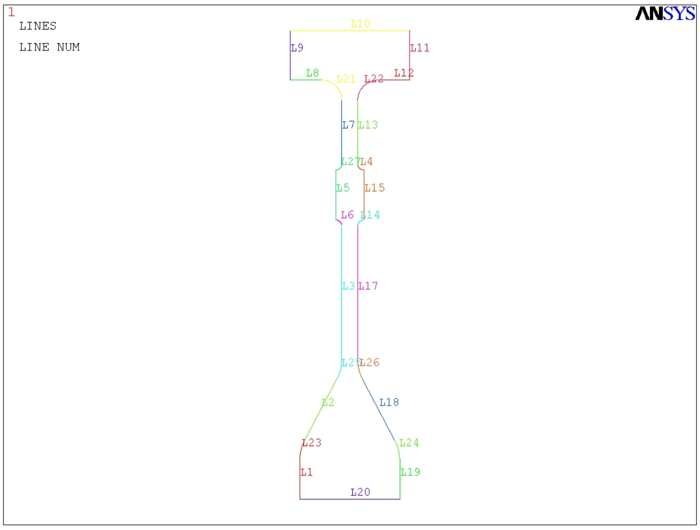

Fig. 6.2. Disk contour

8) To create a finite element mesh, you must create additional lines. Create line between points K25 and K28, K26 and K27, K29 and K31, K30 and K32, K34 and K19, K33 and K35, K8 and K20;

9) Create an additional point K16 with coordinates (0, 0.306, 0);

10) Create arc by three points KP10, KP22, KP 16: **PrPr → Modeling → Create → Lines → Arcs → Thtough 3 KPs**

11) Create areas along the lines as shown in Fig. 6.3;

12) Create a cylindrical coordinate system:

1. Align the working coordinate system with the global: **MM → Workplane → Align WP with Global Cartesian**
2. Rotate the working coordinate system by 90^0 relative to the y-axis: **MM → Workplane → Offset WP by Increments**

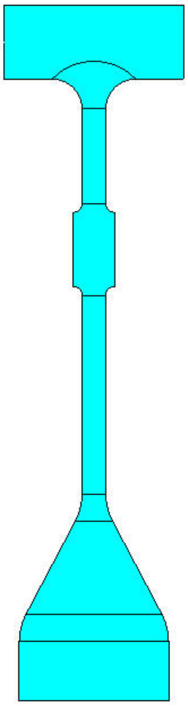

Fig. 6.3. Creating areas

3. Create a cylindrical coordinate system 11 that coincides with the working plane: **MM → Workplane → Local Coordinate Systems → Create Local CS → At WP origins**

13) Rotate areas by an angle of 3^0 relative to the axis of rotation disk:

1. Go to the created cylindrical coordinate system: **MM → Workplane → Change Active CS to → Specified Coord Sys → 11**
2. Set the rotation of the areas by 3^0 relative to the z-axis: **PrPr → Modeling → Move/Modify → Areas → Areas**

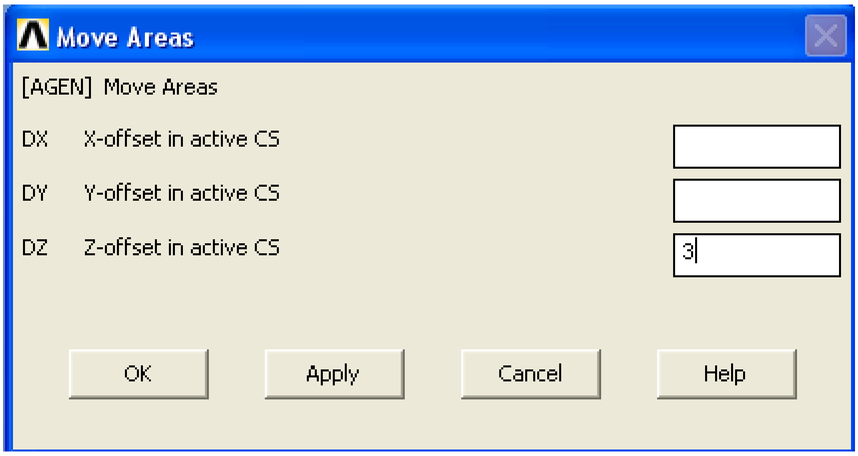

Fig. 6.4. Area rotation menu

14) Create a sector of disk by operation of extrusion of areas: **PrPr → Modeling → Operate → Extrude → About Axis**. Select as the two points defining an axis of rotation points 500 and 501.

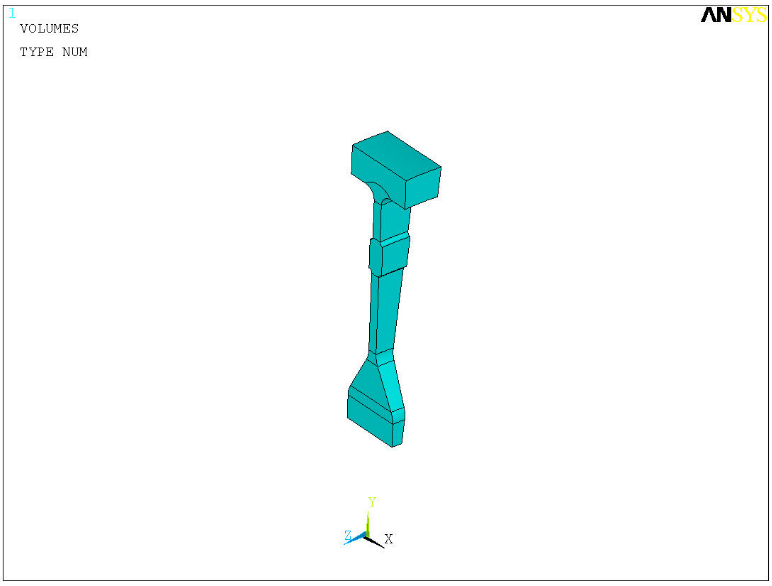

Fig. 6.5. Bladed disk sector

6.2 Cutting the groove in the rim of disc

1) Set the working coordinate system on rim of disk as shown in Fig. 6.6. **MM → WorkPlane → AlignWP with → Plane Normal to Line → L93 → Ratio Along Line=0,5**. The coordinate system must be installed in the middle of the line 93, while the line will be normal to the XY plane;

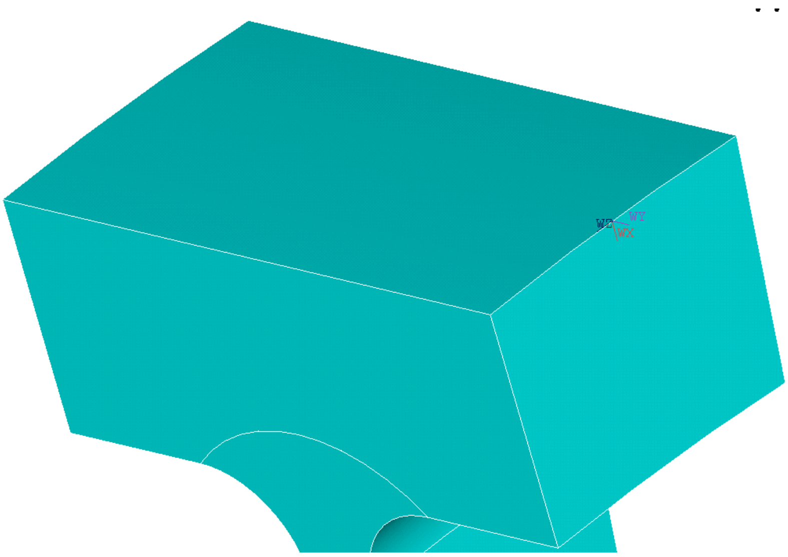

Fig. 6.6. Setting the working coordinate system

2) Make working coordinate system of active: **MM → WorkPlane → Change Active CS to → Working Plane;**

3) Create 4 points that define the groove of disk:

1	0.012	0	-0.0085
2	0.012	0	0.0085
3	0	0	-0.004132
4	0	0	0.004132

4) Create lines by points KP64 and KP65, KP65 and KP63, KP63 and KP62, KP62 and KP64;

5) Add a fillet radius 0,002 *м* between lines L96 and L97, and between lines L98 and L97 (Fig. 6.7);

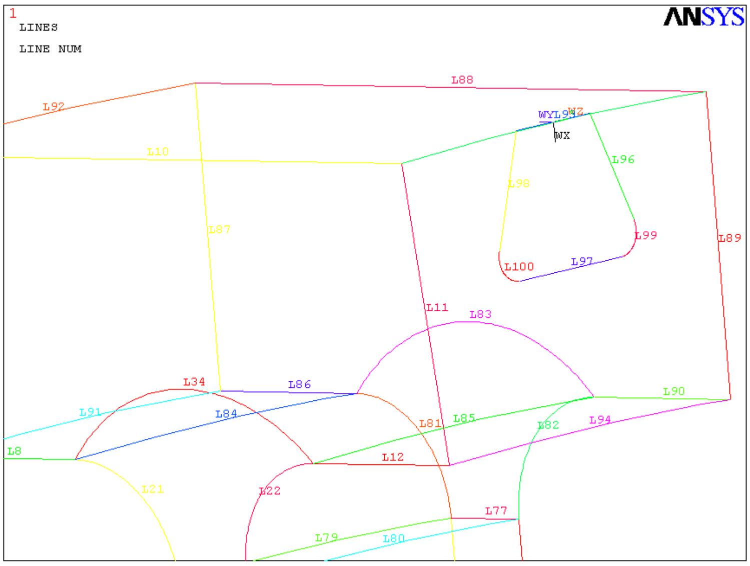

Fig. 6.7. Contour of groove of disc

6) Create area along the contour lines of groove: **PrPr → Modeling → Create → Areas → Arbitrary → By Lines**, select lines with numbers in order L95, L96, L99, L97, L100, L98;

7) Make the global coordinate system of active: **MM → WorkPlane → Change Active CS to → Global Cartesian**;

8) Extrude in direction opposite to x-axis created area by 0.1 m: **PrPr → Modeling → Operate → Extrude → Areas → By XYZ Offset**;

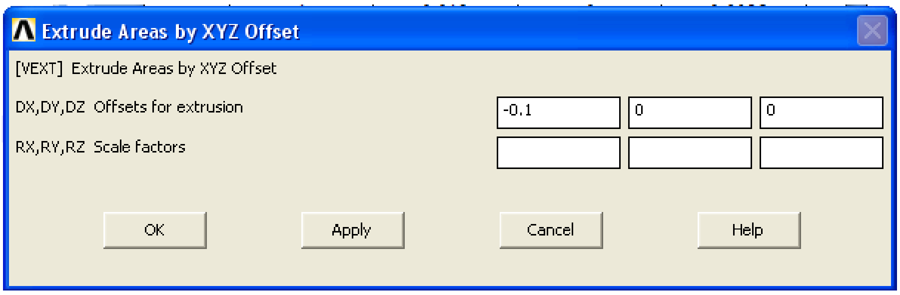

Fig. 6.8. Extrude areas menu

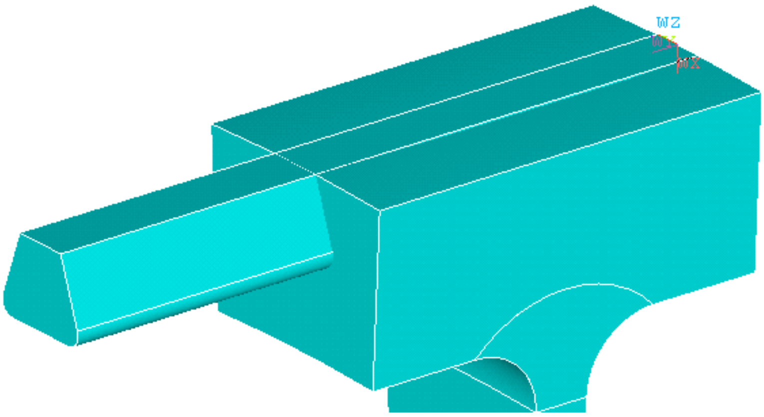

Fig. 6.9. Auxiliary volume for the groove cut-out

9) By subtracting one volume from another, create a groove in the disk sector: PrPr → **Modeling** → **Operate** → **Booleans** → **Subtract** → **Volumes**. Select the disk volume first, then the secondary volume.

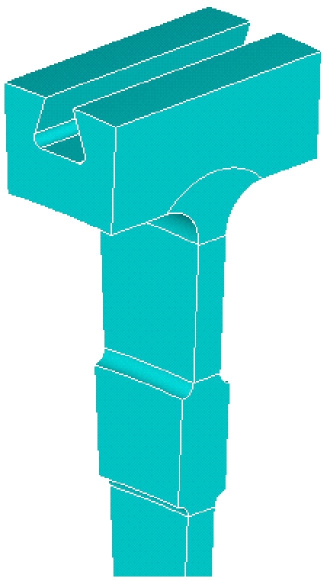

Fig. 6.10. Groove in disc

6.3 Preparing the model for mesh generation

To create an ordered finite element mesh, split the disk rim into several volumes, each of which easily creates an ordered mesh.

1) Create line from point 22 perpendicular to the line 10: **PrPr** → **Modeling** → **Create** → **Line** → **Lines** → **Normal to Line**. In this case, the line 10 is divided into two lines, with a separating point KP64 (Fig. 6.11);

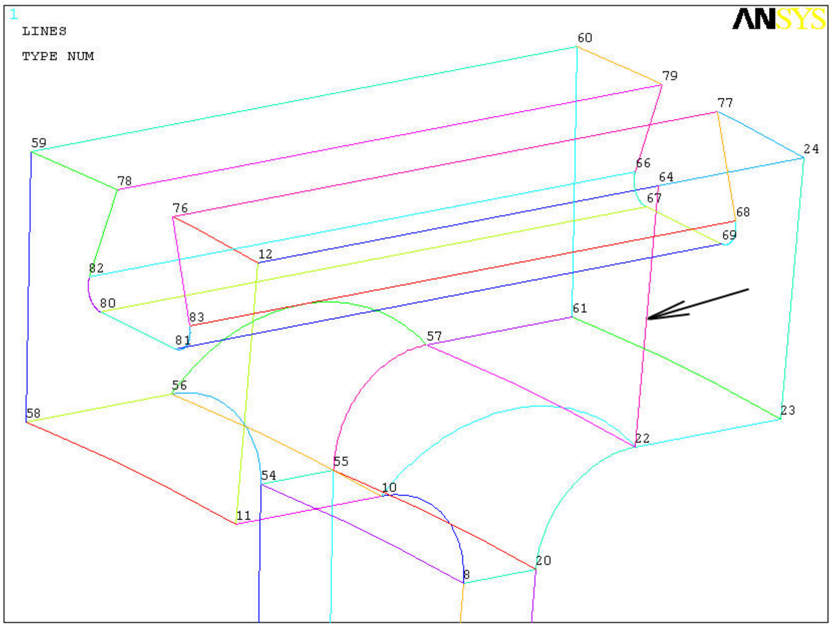

Fig.6.11. Auxiliary line

2) Cut the rim volume with the working plane (WorkPlane), for this you need to link it to the points KP64, KP22, KP57: **MM → WorkPlane → Align WP with Keypoints**;

3) Delete the auxiliary line: **PrPr → Delete → Line and Below → L93**;

4) Cut the rim volume with the working plane: **PrPr → Operate → Booleans → Divide → Volu by WrkPlane**;

5) Proceed in the same action on the other part of disk rim, an auxiliary line is created from point KP10;

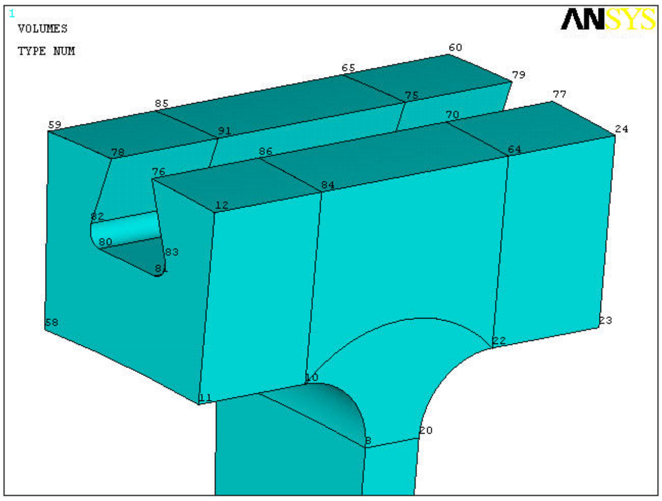

Fig. 6.12. Splitting rim of disc

6) Cut rim with the horizontal working plane. Combine the WorkPlane with the points KP82, KP83, KP68, then similarly to par. 4 cut three rim volumes by specifying them in the Boolean operation menu. Результат на Fig. 6.13;

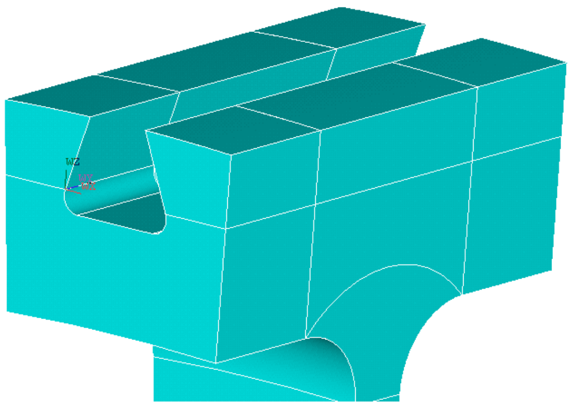

Fig. 6.13. Final splitting of disk rim into separate volumes

6.4 Generating finite element mesh

1) Set material properties of disk **PrPr → Material Props → Material Models**. For steel: $E=2\cdot10^{11}$ (Pa); density 7800 (kg/m^3); Poisson's ratio 0,3.

2) Set element type

1. **PrPr → Element type → Add/Edit/Delete → Add → Shell → 3D 4node 181**
2. **PrPr → Element type → Add/Edit/Delete → Add → Solid → Brick 8 Node 185**

3) Set the number of elements along the lines as shown in Fig. 6.14 and 6.15: **PrPr → Meshing → Size Cntrls → Lines → Picked Lines**.

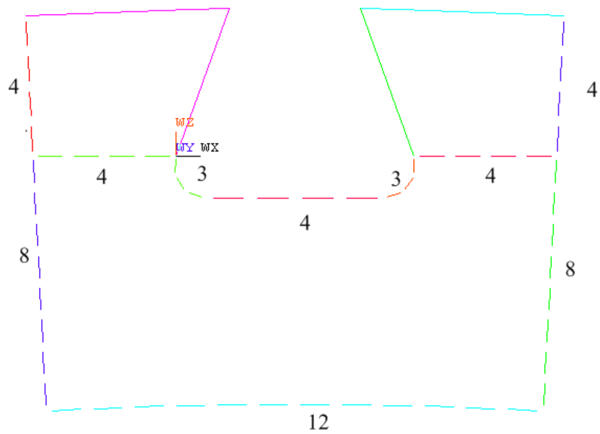

Fig. 6.14. Number of elements along the end face

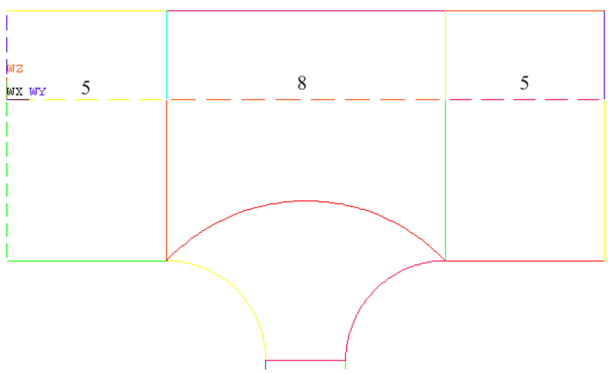

Fig. 6.15. Number of elements along the side surface

4) Set element type **Shell 181: PrPr → Meshing → Mesh Attributes**

5) Create unordered mesh of finite elements on the lower end surface of A105 (Fig. 6.14): **PrPr → Meshing → Mesh → Areas → Free**. Based on this mesh will be created volume mesh.

6) Set element type **Solid 185**, create the bottom half of the rim of the disk by extrusion (sweep): **PrPr → Meshing → Mesh → Volume Sweep → Sweep**.

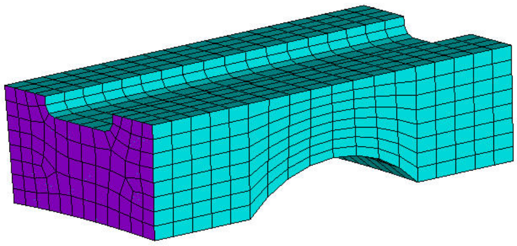

Fig. 6.16. Volume mesh on the bottom of disk rim

7) Create ordered mesh on the remaining volumes of disk rim: **PrPr → Meshing → Mesh → Volumes → Mapped 4 to 6 sided**

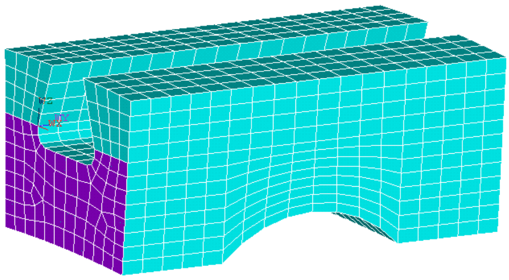

Fig. 6.17. Volume mesh on rim of disk

8) Specify the number of elements along edges of bladed disk as shown in Fig. 6.18.

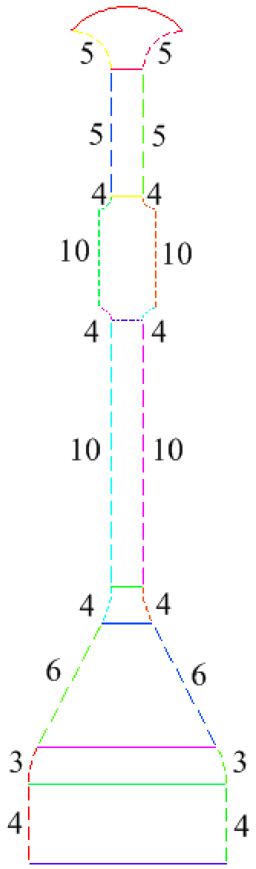

Fig. 6.18. Bladed disk

9) Starting from the top surface (A8), create a surface finite element mesh. For A6, specify type of mesh **Free**, for the rest type **Mapped**. The result in Fig. 6.19.

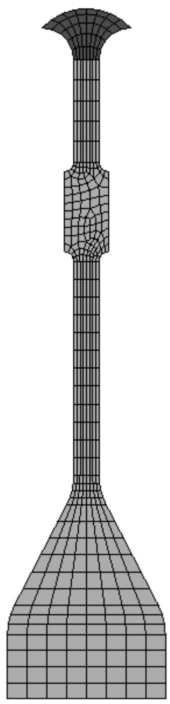

Fig. 6.19. 2D mesh on the bladed disk

10) Create volume finite element mesh by extrusion: **PrPr** → **Meshing** → **Mesh** → **Volume Sweep** → **Sweep**. Specify volumes with numbers from 8 to 1;

11) Remove the mesh from the areas: **PrPr** → **Meshing** → **Clear** → **Areas** → **Pick all**.

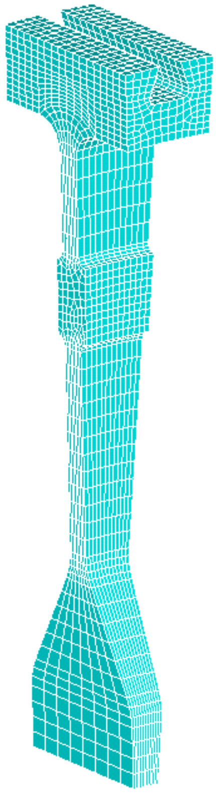

Fig. 6.20. Finite-element disk sector model

6.5 Boundary conditions and loads

Since the for calculation is used sector model, it is necessary to impose cyclic symmetry conditions on opposite faces of the disk sector, connecting the displacements of the nodes of these faces in a cylindrical coordinate system in the direction of rotation.

1) Select the areas that belong to the opposite faces of the sector: **UM** → **Select** → **Entities** → **Areas** → **By NumPick**;

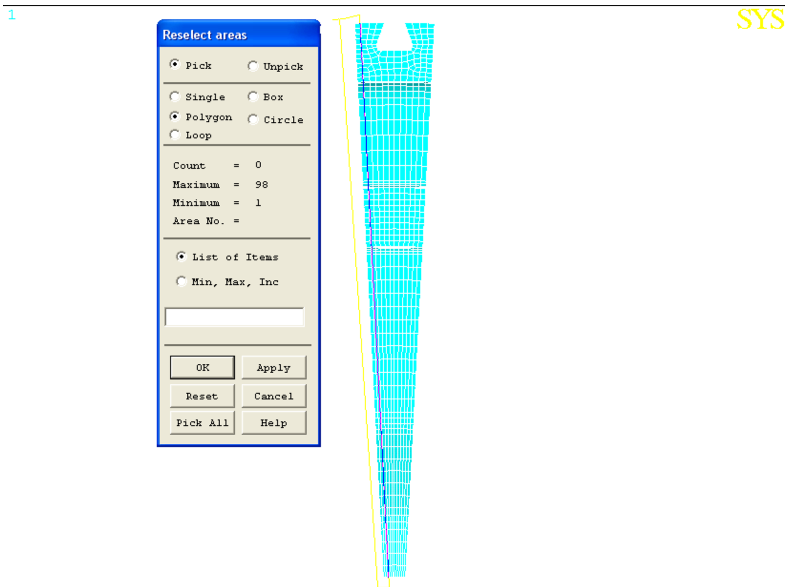

Fig. 6.21. Selecting areas using the **Polygon** operation

2) Select nodes that belong to these areas: **UM** → **Select** → **Entities** → **Nodes** → **Attached to** → **Areas all** → **Reselect**;

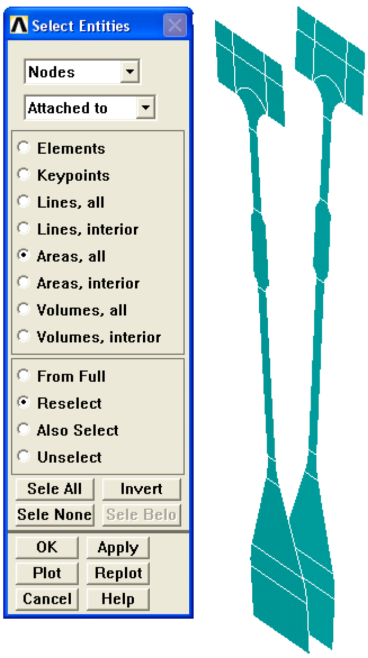

Fig. 6.22. Select of nodes belonging to the opposite faces of the sector

3) Connect the opposite nodes by the condition of cyclic symmetry: **PrPr** → **Coupling/ Ceqn** → **Offset Nodes**. Specify the coordinate system 11 (KCN), DY=6^0. Select all elements of model: UM → Select → Everything;

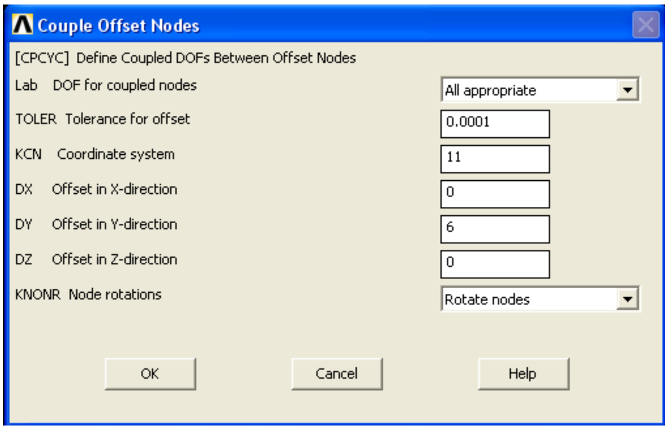

Fig. 6.23. The cyclic symmetry condition menu

4) Secure the disk sector. Select the areas A33 and A36, then the nodes belonging to these areas;

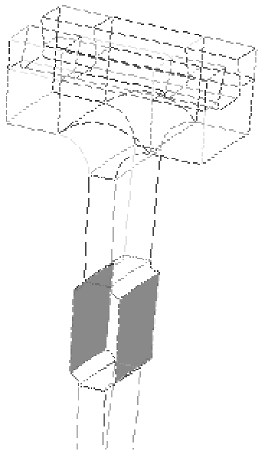

Fig. 6.24. Surfaces for fixing the model

5) Fixing model conveniently carried out in a cylindrical coordinate system. Go to the local cylindrical coordinate system 11: **UM → WorkPlane → Change Active CS to → Specified Coord Sys → 11**;

6) Move the selected nodes to the local coordinate system: **PrPr → Modeling → Move/Modify → Rotate Node CS → to Active CS**;

7) Secure the model in the axial (UZ) and radial (UY) directions: **PrPr → Loads → Define Loads → Apply → Structural → Displacement → On Nodes**;

8) Select all model: **MM → Select → Everything**;

9) Apply a centrifugal load 600 *рад*/с: **PrPr → Loads → Define Loads → Apply → Structural → Inertia → Angular Velocity → Global → OmegX=600**;

10) Run on calculation: **Solution → Analysis Type → New Analysis → Static. Solution → Solve → Curent LS**;

11) Display the circumferential and radial stresses in the disk:

1. Display the results in a cylindrical coordinate system: **General Postproc → Options for Outp** - In the **RSYS** column set the **Local system**

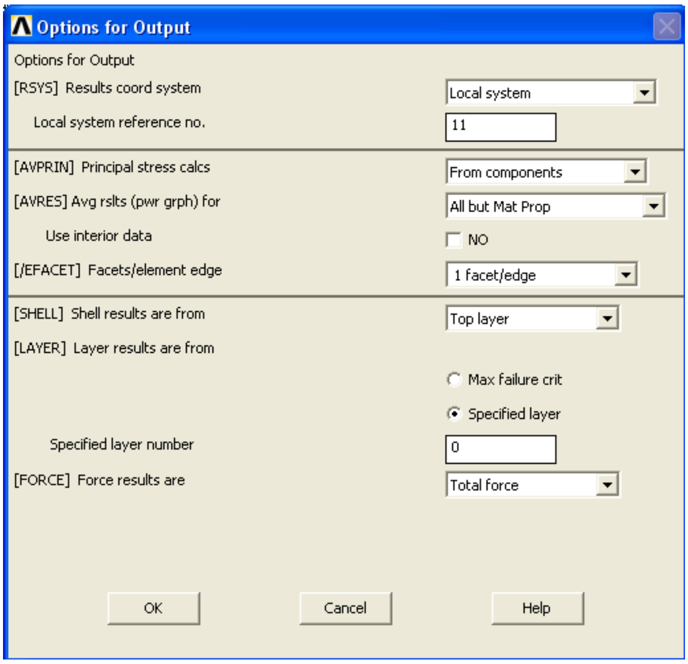

Fig. 6.25 Output of calculation results in a cylindrical coordinate system

2. Output the circumferential stresses: **General Postproc → Plot Results → Contour Plot → Nodal Solu → Stress – X-Component of stress**

3. Output the radial stresses: **General Postproc → Plot Results → Contour Plot → Nodal Solu → Stress – Y-Component of stress**

CONCLUSION

The examples in this tutorial are the base level for students using Ansys Mechanical for structural analysis. The example of analysis simple designs shows the capabilities of the software package for creating finite element models, applying loads and calculating. The examples are created in such a way that allow us to study a large number of functions Ansys Mechanical, which will be necessary in the performance of course and project work in the senior years.

REFERENCES

1 Ogorodnikova, O.M. Introduction to computer structural analysis (Введение в компьютерный конструкционный анализ) [Text]/ O.M. Ogorodnikova – Ekaterinburg: Ural State Technical University, 2001.-48 с.

2 ANSYS. Commands Reference. Rel. 11. / ANSYS Inc. Houston, 2006.

Учебное издание

Шкловец Александр Олегович

**WORK IN CAE-PACKAGE «ANSYS MECHANICAL»:
STRUCTURAL ANALYSIS BY THE FINITE ELEMENT METHOD**

Учебное пособие

В авторской редакции

Подписано в печать 20.12.2017. Формат 60х84 1/16.
Бумага офсетная. Печ. л. 2,75.
Тираж 25 экз. Заказ .

ФЕДЕРАЛЬНОЕ ГОСУДАРСТВЕННОЕ АВТОНОМНОЕ
ОБРАЗОВАТЕЛЬНОЕ УЧРЕЖДЕНИЕ ВЫСШЕГО ОБРАЗОВАНИЯ
«САМАРСКИЙ НАЦИОНАЛЬНЫЙ ИССЛЕДОВАТЕЛЬСКИЙ
УНИВЕРСИТЕТ имени академика С.П. КОРОЛЕВА»
(Самарский университет)
443086, Самара, Московское шоссе, 34.

Изд-во Самарского университета.
443086, Самара, Московское шоссе, 34.