MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE
National University "Zaporizhzhia Polytechnic"

GUIDELINES

for practical classes

in
"Engineering calculations and design in CAE systems"
for students of 13 "Mechanical engineering" course
specialty G9 "Applied mechanics"
educational program "Welding technologies and equipment",
"Restoration and improvement of parts and structures wear resistance"
of all modes of study

2024

Guidelines for practical classes in the discipline "Engineering calculations and design in CAE systems" for students for students of the field of knowledge 13 "Mechanical engineering" specialty G9 "Applied mechanics", educational program "Welding technologies and equipment", "Restoration and improvement of parts and structures wear resistance"of all forms of education / Incl.: H.M. Laptieva, I.M. Bilonyk - Zaporizhzhia: Zaporizhzhia Polytechnic National University, 2024. - 57 p.

Compiler: H.M. Lapteva, associate professor, candidate of technical sciences
I.M. Bilonyk, associate professor, candidate of technical sciences
Reviewer: M.Y.Osipov, associate professor, candidate of technical sciences

Responsible for the publication: H.M. Lapteva.

Approved
at the meeting of the WIP and SM subdepartment
Protocol No.5 d.d. 29.11, 2024
Recommended
for publication by the Science and Methodological Commission of Engineering and Physics Department
Protocol No.4 d.d 17.12, 2024

CONTENT

1 PRACTICAL CLASS NO. 1 ANALYSIS OF FEASIBILITY AND SCOPE OF USING OF THE FINITE-ELEMENT METHOD. STAGES OF FINITE ELEMENT ANALYSIS

1.1 Purpose of the study

Acquaintance with the feasibility and scope of using of the finite element analysis method. Mastering the sequence of stages of finite element analysis.

1.2 General information

The main task of an engineer is the design and calculation of various constructions. This requires knowledge of many subjects in different areas, diligence, attentiveness and a large amount of time. Modern programs make work much easier and allow you to make calculations much faster and better. They also visualize the process of force, torque, medium exertion and show what will happen to a construction or a component after a certain period of time.

Currently, there are many computer calculation systems, but they can be divided into two groups: CAD (Computer-Aided Design) and CAE (Computer-Aided Engineering). CAE systems are designed for modeling and analysis of various physical processes, such as beam loading, structural failure, air flow in ventilation shafts, etc. Most modern CAE systems are based on complete mathematical models, which are systems of partial differential equations that are solved using finite difference, finite volume, and finite element methods, in particular Ansys Mechanical APDL [1] (license version 4.0 is used during practical classes).

The main stages of finite element analysis:

1) **Preferences** – indicates the type of calculation (Fig. 1.1).

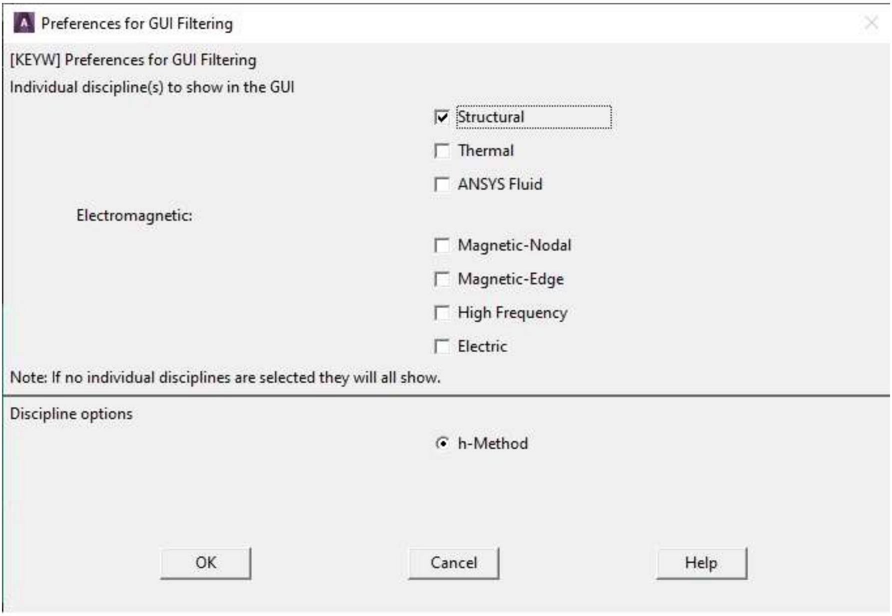


Figure 1.1 – Structure of the Preferences main menu section

2) **Preprocessor** – at the stage of preprocessor preparation, selection of the calculations type, modeling and application of loads (including boundary conditions) are performed. The output data required for the solution are specified here. The user selects coordinate systems and types of finite elements, specifies elastic constants and physico-mechanical properties of a material, builds a solid model and mesh of finite elements, performs the necessary actions with nodes and mesh elements, sets connection equations and constraints. It is also possible to use the statistical accounting module to estimate expected file sizes and memory resource consumption (Fig. 1.2).

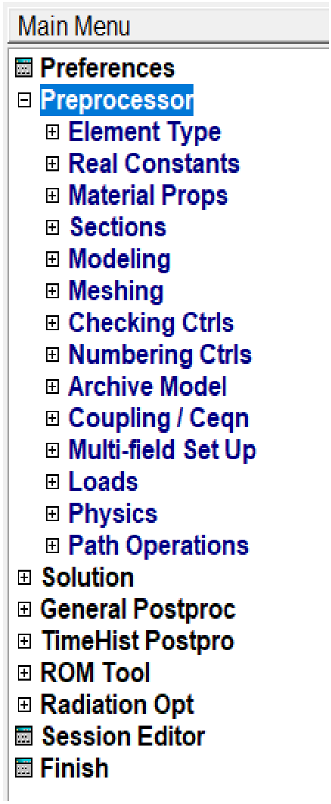


Figure 1.2 – Structure of the Preprocessor main menu section

3) **Solution** – After the calculation model has been built during preprocessing, you can proceed to the stage of solving the problem. This stage includes tasks of the type of analysis and its options, loads, solution step and ends with the launch of the finite-element task (Fig. 1.3).

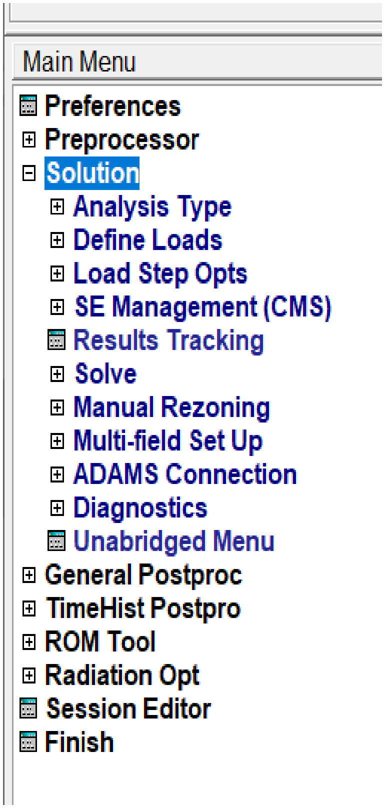


Figure 1.3 – Structure of the Solution main menu section

4) **General Postproc** – In the *ANSYS* program, the stage of post-processing follows the stages of pre-processor preparation and solution . With the help of the post-processing tools of the program, it is possible to refer to the results of the solution and interpret them in the desired way . The results of the solution include values of movements, temperatures, stresses, strains, velocities, and heat flows. The result of the program's work at the post-processing stage is a graphical and (or) tabular presentation of the results. The graphic image can be displayed on the monitor in interactive mode during post- processing or converted into a hard copy (Fig. 1.4) [3].

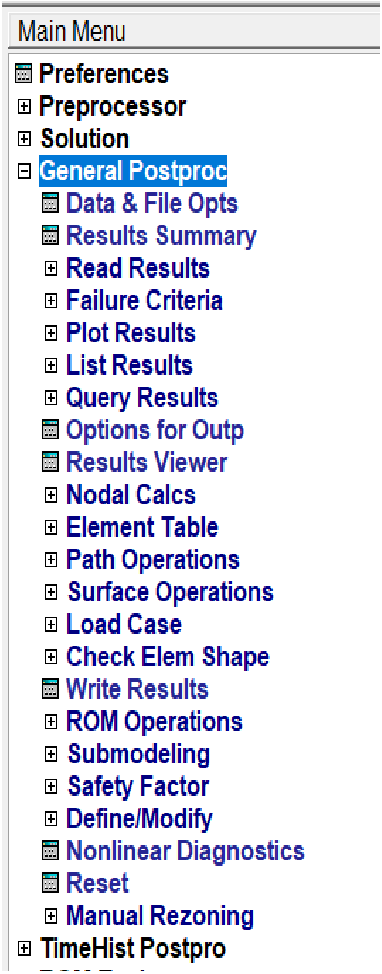


Figure 1.4 – The structure of the General Postproc main menu section

1.3 Control questions for self-checking and control of students' competence

1. List the main problems that are solved using ANSYS?
2. Lists the main ANSYS main menu sections.
3. List the stages of calculations in ANSYS.

4. Name the purpose of the main sections of the main menu.
5. List the most frequently used toolbar commands.
6. Name the purpose (feasibility) of the main toolbar commands.
7. Specify the main requirements for the location of ANSYS calculation files and their names.

1.4 Safety instructions

1. Students are admitted to practice after instruction on labor protection and fire safety.
2. It is forbidden to turn on electrical devices without the permission of the head of laboratory or the teacher.
3. In the event of a fire or electric shock, students must act in accordance with the approved instructions of labor protection and fire safety.

1.5 Working sequence

1. Open the file in the Mechanical APDL Product Launcher (the location and name of the file is specified by the teacher).
2. With the help of the teacher, familiarize yourself with the main sections of the main menu (Main Menu) (Fig. 1.5).

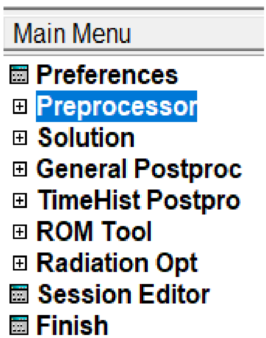


Figure 1.5 – The main sections of the main menu

3. Calculate the given model (Solution – Solve – Current LS).
4. Output results using General Postproc.
5. Prepare a file for the report according to the task and save the file with the calculation.

1.6 Content of the report

Purpose and method of practice. Brief theory. Procedure of finding calculation results. Printed file with the results (if available). Conclusions.

Recommended reading

1. A.Ya. Karvatskyi Finite element method in solid media mechanics problems. Program implementation and visualization of results [Electronic resource]: textbook - K.: NTUU "KPI", 2015. - 391 p.

2. I.V. Pavlenko Finite element method in problems of oscillations of mechanical systems: textbook – Sumy: Publishing House of Sumy State University, 2007. – 180 p.

3. ANSYS Mechanical APDL Structural AnalysisGuide.- ANSYS, Inc. Southpointe 275 Technology Drive, 2013. - 522 p.

2 PRACTICAL CLASS No. 2
DEVELOPMENT OF A GEOMETRIC MODEL OF THE PRODUCT. MODELING IN A PREPROCESSOR

2.1 Purpose of the study

Learning of developing a finite element model of a volumetric part or a welded construction.

2.2 General information

2.1.1 Basic tools of the Modeling main menu section.

Preprocessor – at the stage of preprocessor preparation, the selection of the type of calculations, **construction** is performed **models** and application of loads (including boundary conditions). The output data required for the solution are specified here. The user selects coordinate systems and types of finite elements, specifies elastic constants and physico-mechanical properties of a material, builds a solid model and mesh of finite elements, performs the necessary actions with nodes and mesh elements, sets connection equations and constraints. It is also possible to use the statistical accounting module to estimate expected file sizes and memory resource consumption.

Modeling main menu section is required for working with a geometric model: building, deleting, copying, moving.

Create – is a command for creating/building elements:

– Keypoints – allows you to create points, lines, nodes on the desktop.

– Lines – construction of lines, arcs, splines.

– Areas – creation of various geometric planes: circle, rectangle, polygon, arbitrary plane (Arbitrary).

– Volumes – construction of volumetric figures: block, cylinder, prism, sphere, cone, truncated pyramid.

Move – is a command created for movements of elements.
Copy – allows you to copy selected elements.
Delete – deletes selected elements.

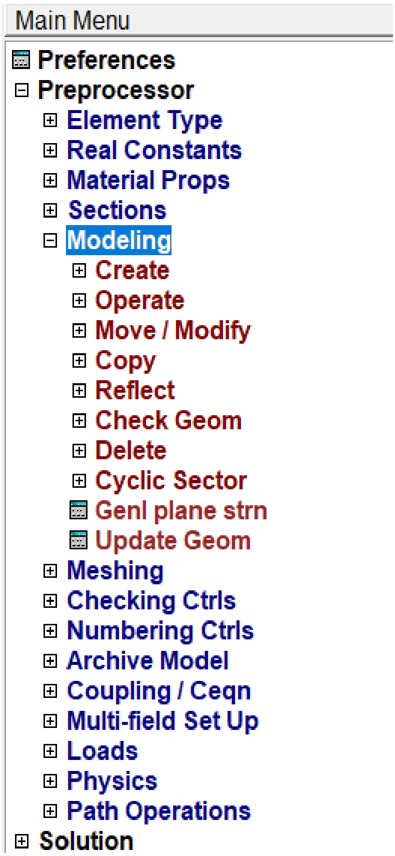


Figure 2.1 – Subsections of the Modeling main menu section

2.2.2 Plot element

When working with various projects, the task sometimes arises to view the construction in a different way. Thus, the Plot element (Fig. 2.2) allows you to show points, planes, volumes, application of forces and torques, etc.

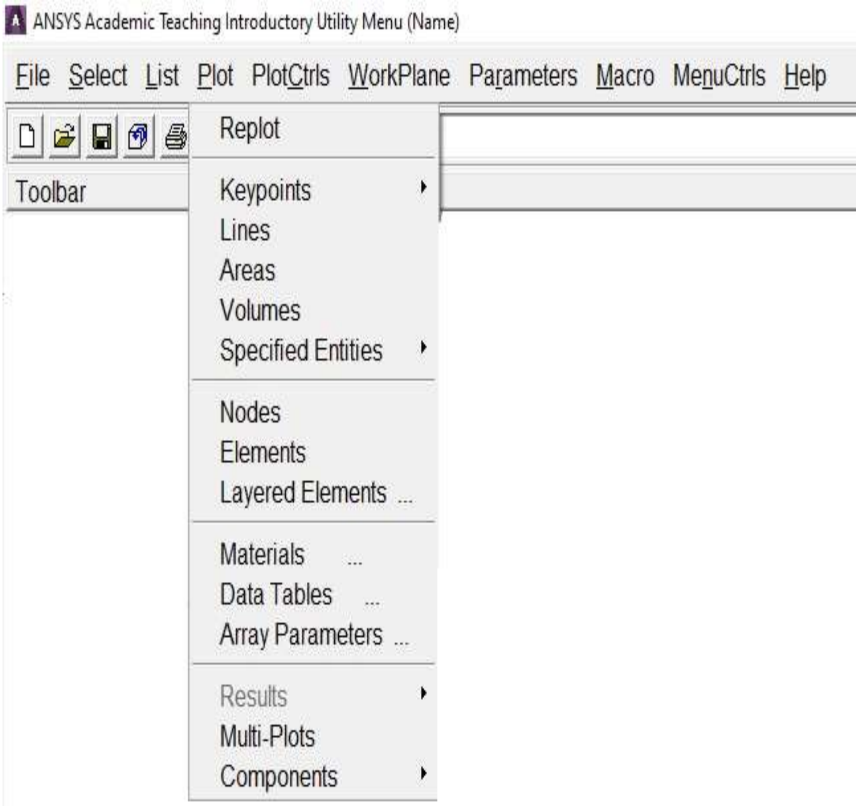


Figure 2.2 – Location of the Plot element

2.2.3 PlotCtrls element

During the building of a three-dimensional model, it often happens that the project is difficult to evaluate due to the accumulation of a large number of structural elements. Therefore, the PlotCtrls element is used to organize and adjust the drawing (Fig. 2.3).

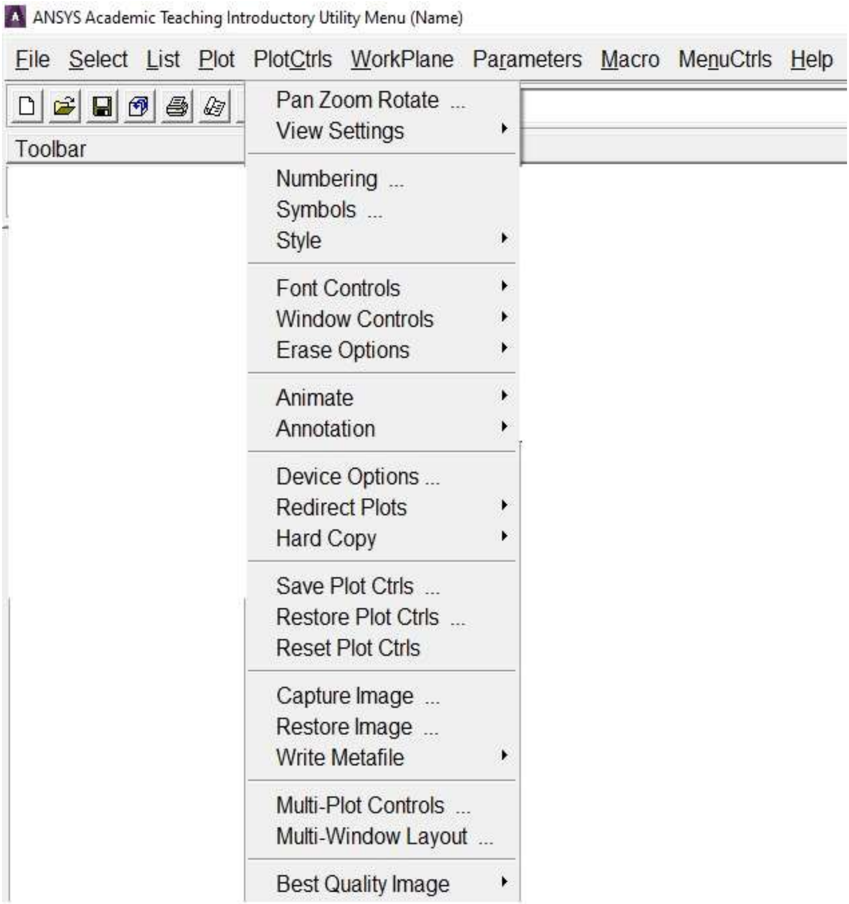


Figure 2.3 – Location of the PlotCtrls element

Using the Numbering item, the user can number the main elements: points, lines, planes, etc. (Fig. 2.4).

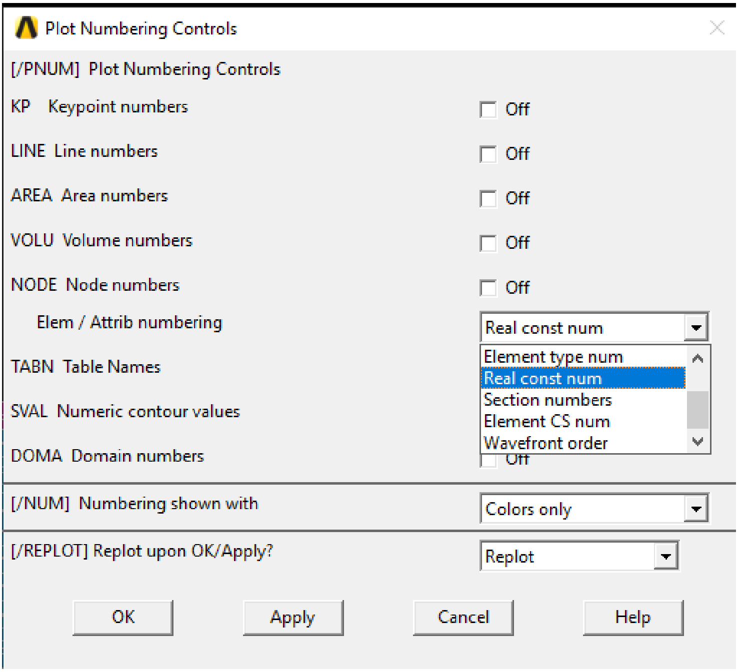


Figure 2.4 – Structure of the Numbering item

When opening Symbols, in a new window you can specify different symbols for displaying different loads in the program: application of force, pressure, moments, color filters, etc. (Fig. 2.5).

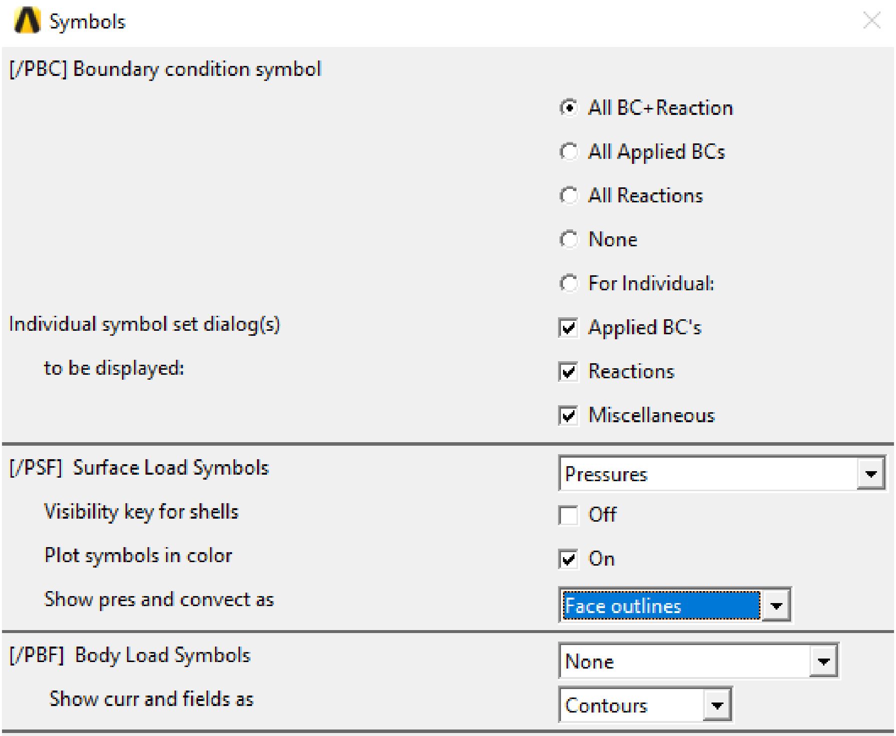


Figure 2.5 – Structure of the Symbols item

2.3 Control questions for self-checking and control of students' competence

1. List the hierarchy (block diagram) of the structural components involved in the creation of three-dimensional models (points, lines, surfaces, volumes).
2. Name the main advantages and disadvantages of displaying a geometric model using individual structural components.
3. Indicate in which section of the main menu are the tools for building a geometric model located?
4. List the operations that can be performed when creating a geometric three-dimensional model?

5. Indicate by which features it is possible to number and display individual structural components of a geometric model?

6. Describe when it is possible to build a geometric model using only lines?

1.4 Safety instructions

1. Students are admitted to practice after instruction on labor protection and fire safety.

2. It is forbidden to turn on electrical devices without the permission of the head of laboratory or the teacher.

3. In the event of a fire or electric shock, students must act in accordance with the approved instructions of labor protection and fire safety.

1.5 Working sequence

1. Open the Mechanical APDL Product Launcher "..." (all names should be indicated in latin).

2. With the help of the teacher, familiarize yourself with the main sections of the main menu (Main Menu).

3. Create a calculation model (Preprocessor – Modeling – Create) according to the task (the sketch number is the last digit of the grade book number (Appendix 1), the number of the line with dimensions is the penultimate digit of the grade book number (Appendix 2).

4. Show the finished model according to the task using lines (surfaces, points).

5. Choose specific surfaces, lines, surfaces, according to the task, in contact with the selected lines.

6. Prepare a file for the report according to the task and save the file with the built model.

2.6 Content of the report

Purpose and method of practice. Brief theory. The sequence of a geometric model building. Printout of the file with the image of the model (if possible). Conclusions.

Recommended reading

1. A.Ya. Karvatskyi Finite element method in solid media mechanics problems. Program implementation and visualization of results [Electronic resource]: textbook - K.: NTUU "KPI", 2015. - 391 p.

2. I.V. Pavlenko Finite element method in problems of oscillations of mechanical systems: textbook – Sumy: Publishing House of Sumy State University, 2007. – 180 p.

3. ANSYS Mechanical APDL Structural AnalysisGuide.- ANSYS, Inc. Southpointe 275 Technology Drive, 2013. - 522 p.

3 PRACTICAL CLASS No. 3 STAGES OF FORMATION OF THE PHYSICAL MODEL OF THE MATERIAL

3.1 Purpose of the study

Familiarize yourself with the main ways of taking into account the properties of materials in calculations and ways of their modeling.

3.2 General information

When calculating various projects in the CAE system, the problem formulation requires the specification of material properties. Depending on the type of problem, material properties can be: linear (stresses are proportional to deformations) or non-linear (stresses are not proportional to loads); isotropic (the material exhibits the same mechanical properties in all directions), anisotropic (the material exhibits different mechanical properties in two directions), orthotropic (symmetric about three orthogonal planes); temperature-dependent or independent.

The interface for setting the material model opens as follows: Main Menu – Preprocessor – Material Props – Material Models (Fig. 3.1).

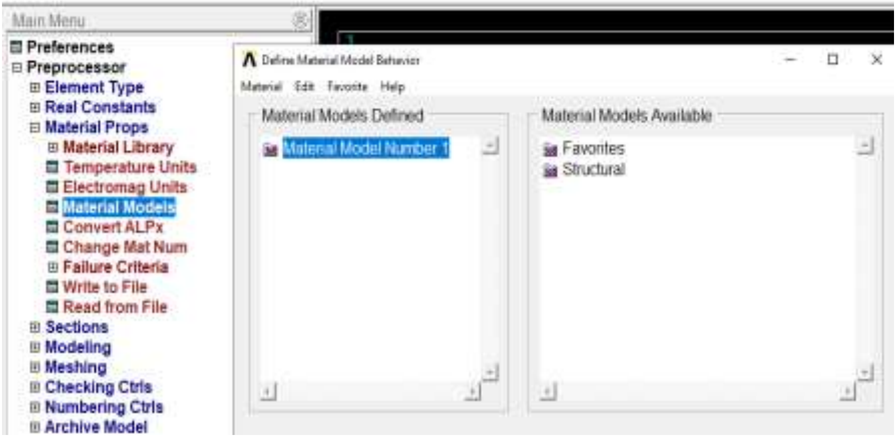


Figure 3.1 – Interface for specifying a material model

The main linear material properties for ANSYS MECHANICAL: modulus of elasticity (EX) and Poisson's ratio (PRXY) (Fig. 3.2). Values are entered in exponential form in MPa.

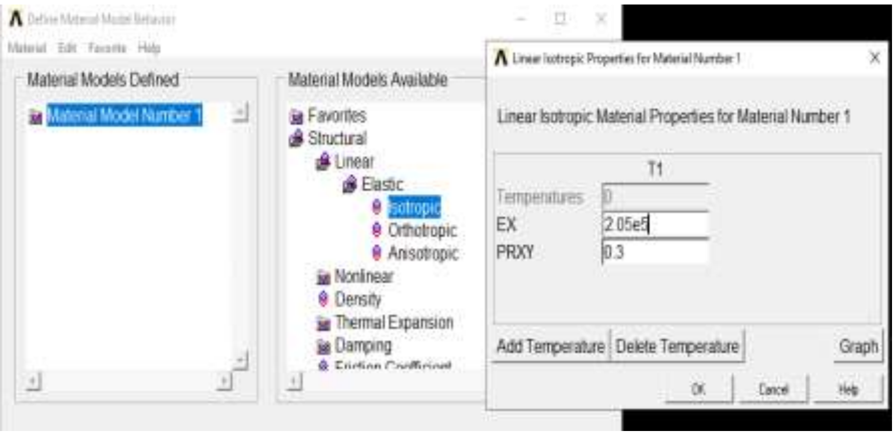


Figure 3.2 – Specifying the linear properties of the material

Basic nonlinear material properties for ANSYS MECHANICAL: yield strength (Fig. 3.3). Values are entered in MPa.

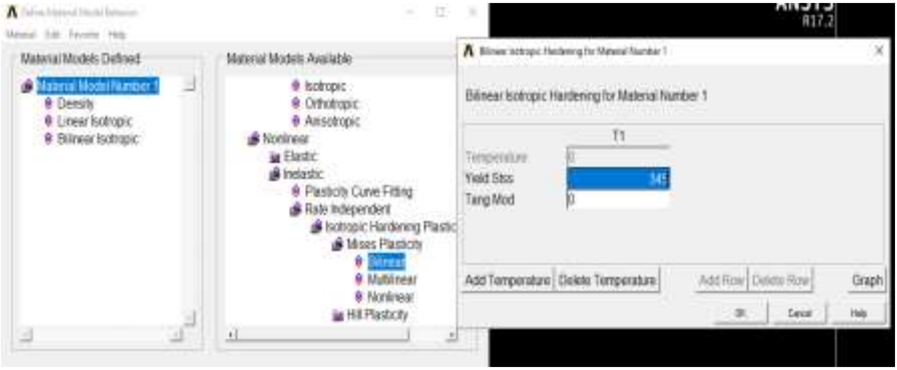


Figure 3.3 – Specifying nonlinear properties of the material

Also, when solving the problem, it is necessary to take into account the own weight of the construction, therefore, the density of the material must be specified (Fig. 3.4).

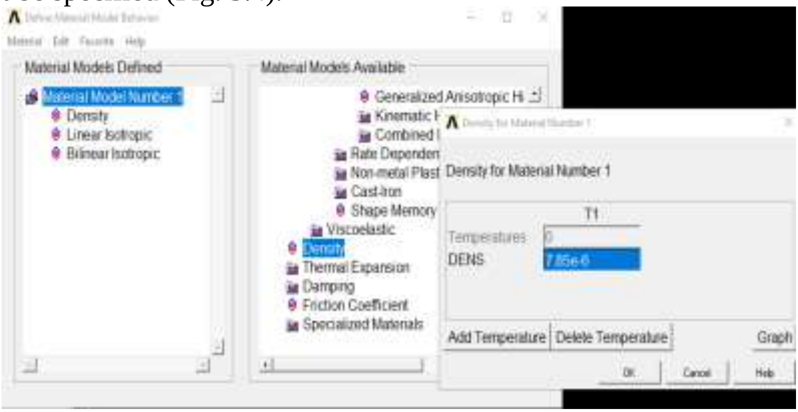


Figure 3.4 – Specifying the density of the material

If the problem statement requires the creation of several materials with different properties, then it is necessary to add a new model: Material – New Model (Fig. 3.5).

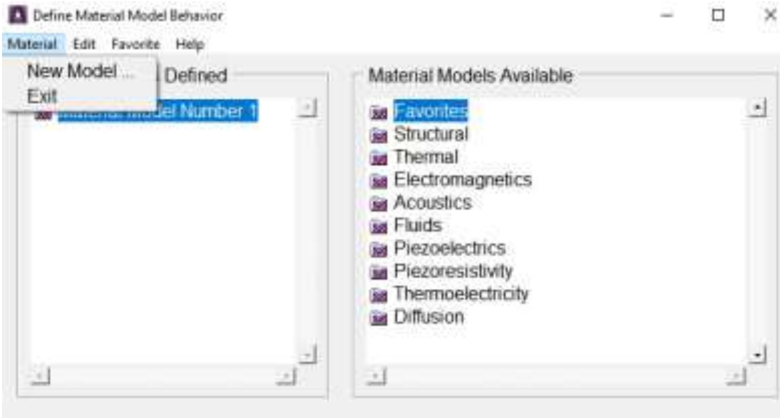


Figure 3.5 – Creation of a new material model

In order to assign the corresponding material model to each element of the construction, you need to go to the Main Menu - Preprocessor – Meshing – Mesh Attributes section of the main menu. Choose the appropriate surface (line, volume). Assign the number of the material type to the selected part (Fig. 3.6). Repeat for all components.

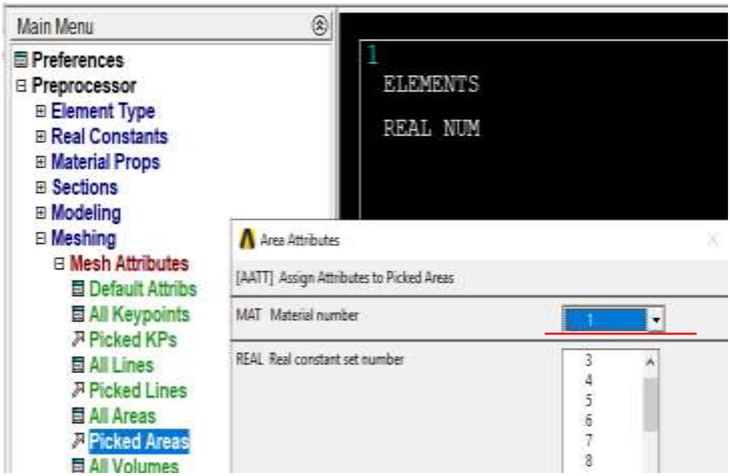


Figure 3.6 – Assigning a material model to a specific part of the construction

In order to make sure that the correct material model is assigned to each part of the structure, it is necessary to go to the PlotCtrls – Numbering section on the toolbar (Fig. 3.7).

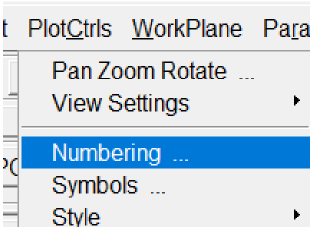


Figure 2.7 – Location of the Numbering section

Choose numbering by material type and set the numbering option by color and material model number (Fig. 3.8).

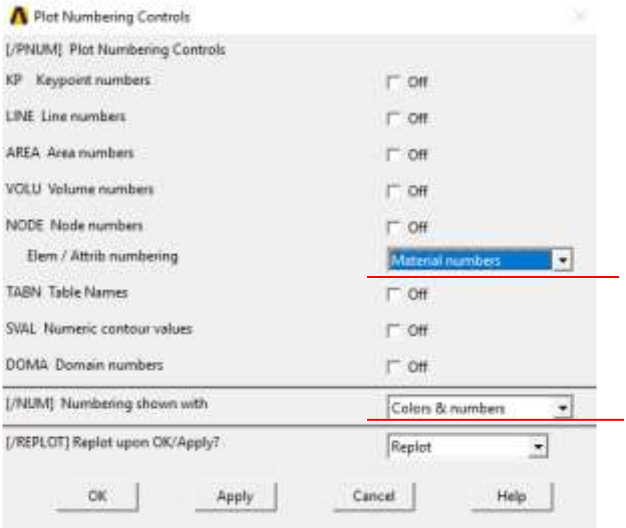


Figure 3.8 – Setting the numbering of structural elements by material type

3.3 Control questions for self-checking and control of students' competence

1. List the main physical and mechanical characteristics of the material which the construction is made of for ANSYS MECHANICAL.
2. Give an example in which form you can record large values of physical and mechanical characteristics of the material?
3. Specify the purpose for setting the density of the material.
4. Name the units of measurement which should be specified for material properties in ANSYS MECHANICAL?
5. Explain what physical and mechanical characteristic of the material is described by Poisson's ratio?
6. In what case is it possible to ignore the density of the material?

3.4 Safety instructions

1. Students are admitted to practice after instruction on labor protection and fire safety.
2. It is forbidden to turn on electrical devices without the permission of the head of laboratory or the teacher.
3. In the event of a fire or electric shock, students must act in accordance with the approved instructions of labor protection and fire safety.

3.5 Working sequence

1. Open the file in the Mechanical APDL Product Launcher, created in the previous practice (all names should be indicated in latin).
2. Display the geometric three-dimensional model built in the previous practical class.
3. Go to the main menu section Main Menu – Preprocessor – Material Properties – Material Models (Fig. 3.9).

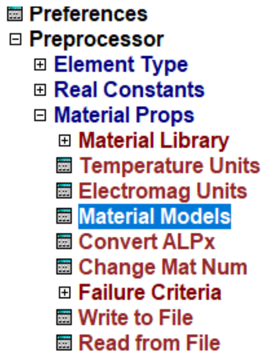


Figure 3.9 – Section of the main menu Material Properties

4. Enter the linear and non-linear properties of the material from which the construction will be made (version number – the last digit of the grade book, Appendix 3) and material density in the corresponding windows.

5. Save the changes to the file.

3.6 Content of the report

Purpose and method of practice. Brief theory. The sequence of material properties setting. Printout of the file section with material properties (if available). Conclusions.

Recommended reading

1. A.Ya. Karvatskyi Finite element method in solid media mechanics problems. Program implementation and visualization of results [Electronic resource]: textbook – K.: NTUU "KPI", 2015. – 391 p.

2. I.V. Pavlenko Finite element method in problems of oscillations of mechanical systems: textbook – Sumy: Publishing House of Sumy State University, 2007. – 180 p.

3. ANSYS Mechanical APDL Structural AnalysisGuide.– ANSYS, Inc. Southpointe 275 Technology Drive, 2013. – 522 p.

4 PRACTICAL CLASS No. 4 DEVELOPMENT OF A FINITE ELEMENT MODEL. CHARACTERISTICS OF FINITE ELEMENTS OF DIFFERENT TYPES. MESH QUALITY MANAGEMENT

4.1 Purpose of the study

Familiarize yourself with the main types of finite elements and their purpose, features and expediency of use.

4.2 General information

The built model is divided into finite elements of a fairly simple form. There are several typical forms (table 4.1) of finite elements, in which the field of displacements is determined by displacements of nodes using some interpolation functions. The stress and strain fields are determined by the displacements calculated in this way. The most time-consuming stage of solving tasks using MFE is the creation of a finite-element model at the stage of preprocessor preparation (preprocessor), since the automatic construction of the mesh of elements does not guarantee the appearance of errors.

The element type determines:

Finite element degrees of freedom (which in turn affect the type of mechanical, thermal, magnetic, electrical analysis).

Object model – one-dimensional, two-dimensional or three-dimensional.

The beam element BEAM4, for example, has 6 degrees of freedom (UX, UY, UZ, ROTX, ROTY, ROTZ) at the node and is used for modeling of frame constructions in 3-dimensional space. The plane element PLANE77 has nodal temperatures as degrees of freedom and can be used for modeling only flat objects [4].

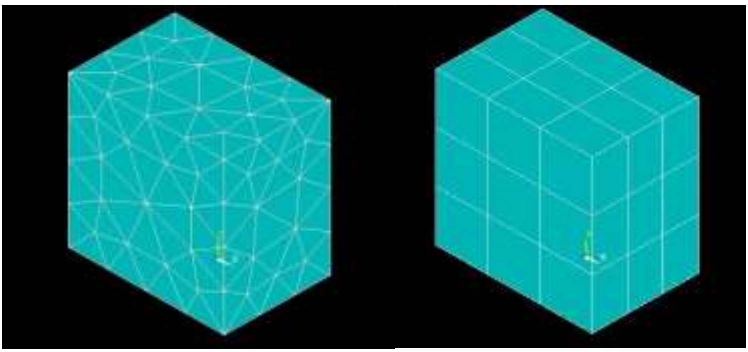
One of the most important stages in the finite element analysis is the construction of a finite elements mesh on the model, that is, the division of the entire model into small pieces (finite elements) connected to each other at nodes.

Table 4.1 – Types of finite elements

Element type	Modeling
LINK	Modeling of truss-constructions, cables, ropes, etc.
BEAM	Modeling of frame constructions
SHELL	Modeling of thin-wall constructions
SOLID	Modeling of three-dimensional objects
PIPE	Modeling of bar systems - pipe + liquid
MASS	Modeling of an absolutely solid body and a material point
CONTACT	Modeling of contact relations

In the ANSYS software complex, there are two main methods of constructing a mesh: constructing an arbitrary mesh (Fig. 4.1 a) and constructing an ordered mesh (Fig. 4.1 b).

An arbitrary mesh is built automatically, while neighboring elements can differ significantly in size. An ordered mesh is built by dividing the geometric elements of the model into a certain number of parts.



a b

Figure 4.1 – Types of finite elements mesh

In automatically constructed meshes with a large number of elements, the number of nodes prevails over the number of elements. The ratio of nodes to elements is approximately 2:1 for planar arbitrary meshes and 6:1 for arbitrary 3D meshes with tetrahedral elements.

The practice of calculations using FEM allows us to give the following recommendations (Fig. 4.2): 1) linear elements require a more frequent mesh than quadratic elements (with one intermediate node) or cubic elements (with two intermediate nodes); 2) an ordered mesh (b) is better than an arbitrary mesh (a); 3 rectangular mesh with 4 nodes (c) is better than mesh with triangular elements (b); 4) the mesh of triangular elements with intermediate nodes (d) has at least the same accuracy as the mesh of rectangular elements with 4 nodes (c); 5) a rectangular mesh with 8 nodes (e) is better than a mesh of triangular elements with intermediate nodes (d), despite the larger size of the rectangular elements; 6) approximation of displacements by a cubic polynomial (e) does not require a finer mesh [1].

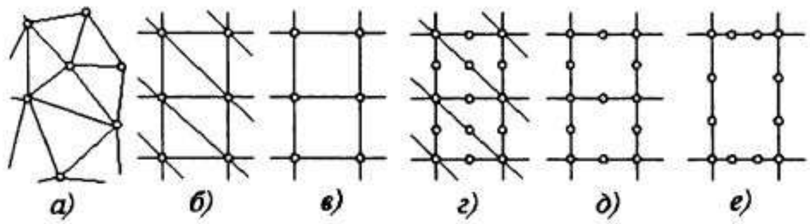


Figure 4.2 – Types of nodes with different types of finite elements mesh

4.3 Control questions for self-checking and control of students' competence

1. Name the types of finite elements available in ANSYS.
2. Describe the purpose of planar (two-dimensional) finite elements.
3. Describe the purpose of three-dimensional finite elements.
4. List the main ways of accumulation of errors in the results in computer modeling of welded constructions.

5. Specify the main criteria for choosing the appropriateness of using a mesh with a smaller size of finite elements when calculating a mathematical model.

7. Name the main advantages and disadvantages of a mesh with large elements for model calculation.

4.4 Safety instructions

1. Students are admitted to practice after instruction on labor protection and fire safety.

2. It is forbidden to turn on electrical devices without the permission of the head of laboratory or the teacher.

3. In the event of a fire or electric shock, students must act in accordance with the approved instructions of labor protection and fire safety.

4.5 Working sequence

1. Open the file in the Mechanical APDL Product Launcher, created in the previous practice (all names should be indicated in latin).

2. Display the geometric three-dimensional model built in the previous practical class.

3. Go to the section of the main menu Main Menu - Preprocessor - Element Type - Add/Edit/Delete - Add. Choose the type of end element in accordance with the type of construction (Fig. 4.3).

4. Set real constants to the corresponding structural elements to take into account the physical thickness in the calculation. To do this, enter the command r, 2, 8 in the command line, where r is the command for assigning a real constant, 2 is the serial number of the constant, and 8 is the physical thickness of the element in mm (Fig. 4.4).

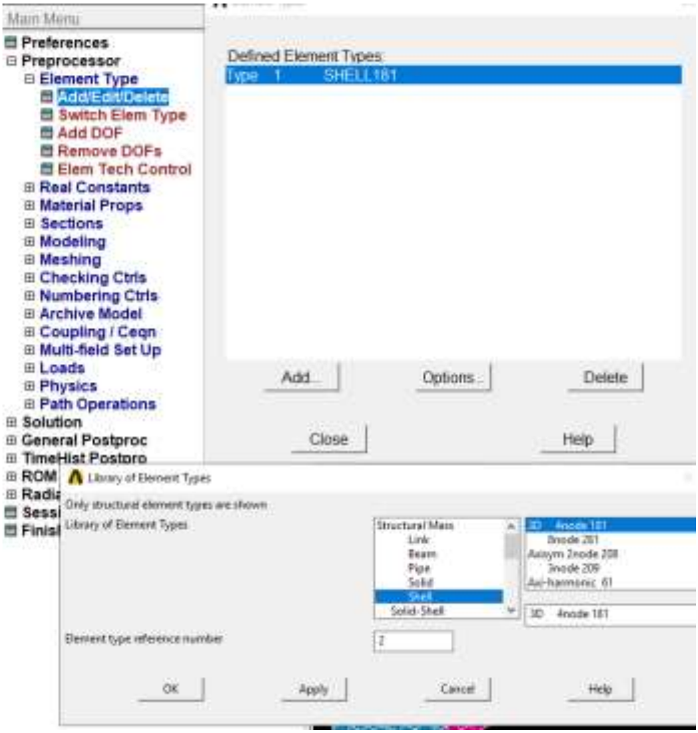


Figure 4.3 - Element Type main menu section interface

Figure 4.4 – Setting of real constants

5. Assign each model element the corresponding real constant, material model, and finite element type. To do this, go to the main menu section Main Menu – Preprocessor – Meshing – Mesh Attributes. Choose the appropriate surface (line, volume). Assign the material type number, the real constant number and the final element type number to the selected part (Fig. 4.5). Repeat for all components.

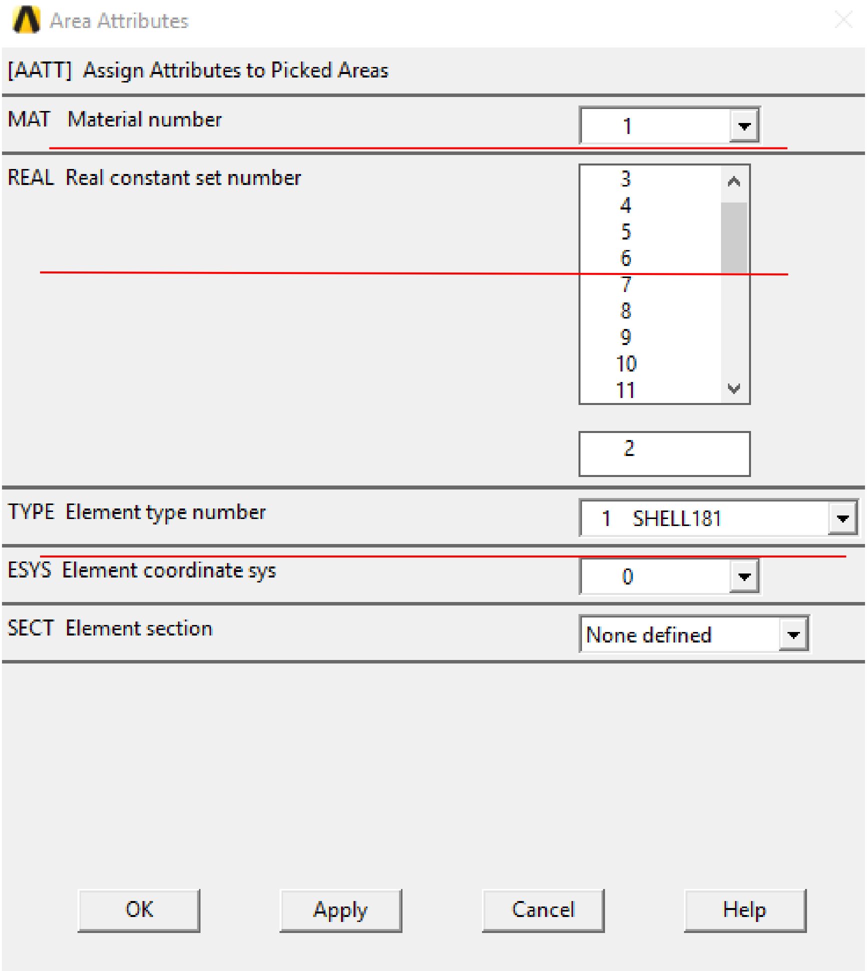


Figure 4.5 – Assignment of attributes

6. Break the three-dimensional model into elements. To do this, go to the main menu section Main Menu – Preprocessor – Meshing – MeshTool – Mesh. (To choose the size of the final elements, consult with the teacher). Change the shape and size of the final elements according to the task (Fig. 4.6). Draw conclusions about the quality of the mesh after the changes.

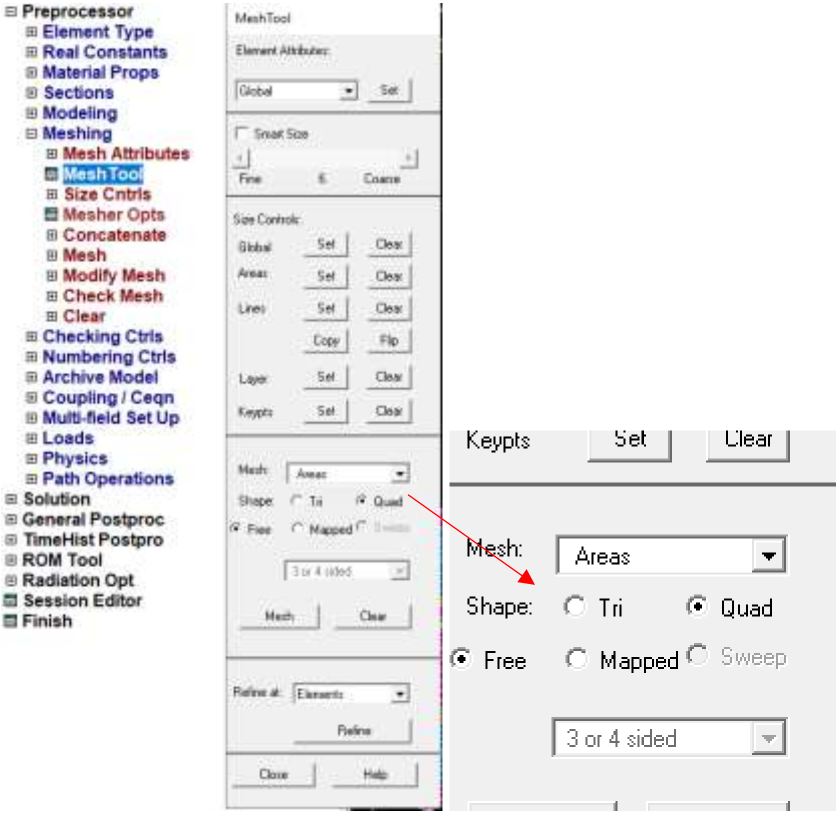


Figure 4.6 – Meshing main menu section interface

7. Save the changes to the file.

4.6 Content of the report

Purpose and method of practice. Brief theory. Procedure of setting the parameters of the finite elements mesh. Printout of the file with the obtained finite element model (if possible). Conclusions.

Recommended reading

1. A.Ya. Karvatskyi Finite element method in solid media mechanics problems. Program implementation and visualization of results [Electronic resource]: textbook - K.: NTUU "KPI", 2015. - 391 p.

2. I.V. Pavlenko Finite element method in problems of oscillations of mechanical systems: textbook – Sumy: Publishing House of Sumy State University, 2007. – 180 p.

3. ANSYS Mechanical APDL Structural AnalysisGuide.- ANSYS, Inc. Southpointe 275 Technology Drive, 2013. - 522 p.

5 PRACTICAL CLASS No. 5
TYPES OF BOUNDARY CONDITIONS. DEFINITION OF MOVEMENTS. SPECIFYING LOADS

5.1 Purpose of the study

Familiarize yourself with the sequence of application of boundary conditions and loads. Learn the basic principles of selection of boundary conditions and the most realistic reproduction of the action of loads.

5.2 General information

Loads mean both external and internal forces, boundary conditions in the form of restrictions on movement.

In the ANSYS MECHANICAL, the way to apply loads is in the main menu section (Main menu – Preprocessor – Loads – Define Loads – Apply (Fig. 5.1).

Figure 5.1 – Location of the Loads main menu section

Loads are divided into the following categories:

– constraints along all axes (x, y, z) – linear and in a specific plane – rotated relative to any of the axes (Fig. 5.2).

Figure 5.2 – Setting the degrees of freedom

– concentrated forces and moments of forces (Fig. 5.3);

Figure 5.3 – Application of forces and moments

– surface loads (pressure) (Fig. 5.4);

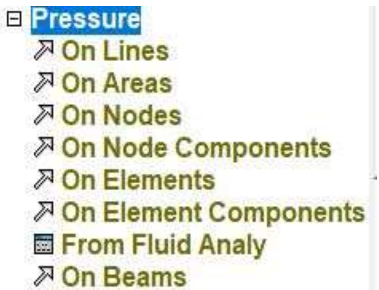


Figure 5.4 – Application of surface forces

– volumetric forces (Fig. 5.5);

Figure 5.5 – Application of volume forces

– inertial loads, in particular gravity (Fig. 5.6).

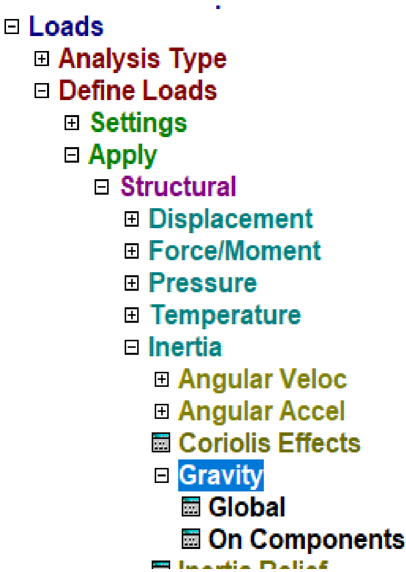


Figure 5.6 – Application of inertial forces

As can be seen from the above figures, the boundary conditions are set by lines, points, nodes, surfaces, volumes. All loads must be applied in units accepted in the international system of measurements.

5.3 Control questions for self-checking and control of students' competence

1. List the boundary conditions in movements relative to the coordinate axes.
2. Explain why it is necessary to take into account the force of gravity when calculating welded constructions?
3. Give an example when the effect of gravity can be ignored in calculations?
4. Explain how the correct determination of the degrees of freedom of the construction model affects the accuracy of the calculation.
5. Specify the case of the expediency of applying symmetric boundary conditions.
6. Explain the need to check the performance of the model before applying loads.

5.4 Safety instructions

1. Students are admitted to practice after instruction on labor protection and fire safety.
2. It is forbidden to turn on electrical devices without the permission of the head of laboratory or the teacher.
3. In the event of a fire or electric shock, students must act in accordance with the approved instructions of labor protection and fire safety.

5.5 Working sequence

1. Open the file in the Mechanical APDL Product Launcher, created in the previous practices (all names should be indicated in latin).
2. Display the geometric three-dimensional model built in the previous practical class.
3. Go to the main menu section Main Menu - Preprocessor – Loads –

Define Loads – Apply – Structural (Fig. 5.7).

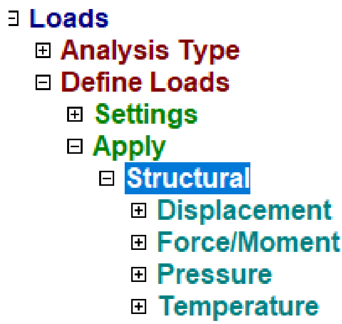


Figure 5.7 – Location of the Loads main menu section

4. According to the individual task, apply the load (force, pressure) (Fig. 5.8).

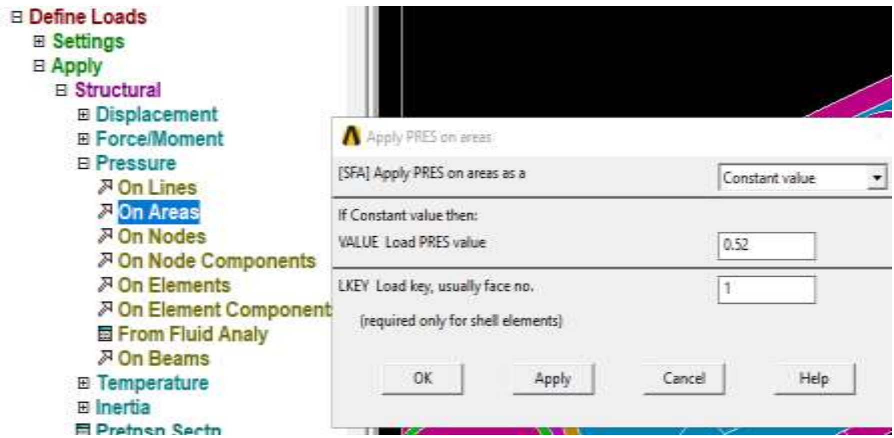


Figure 5.8 – Applying pressure to the surface

5. Apply boundary conditions (fix from movements) (Fig. 5.9).

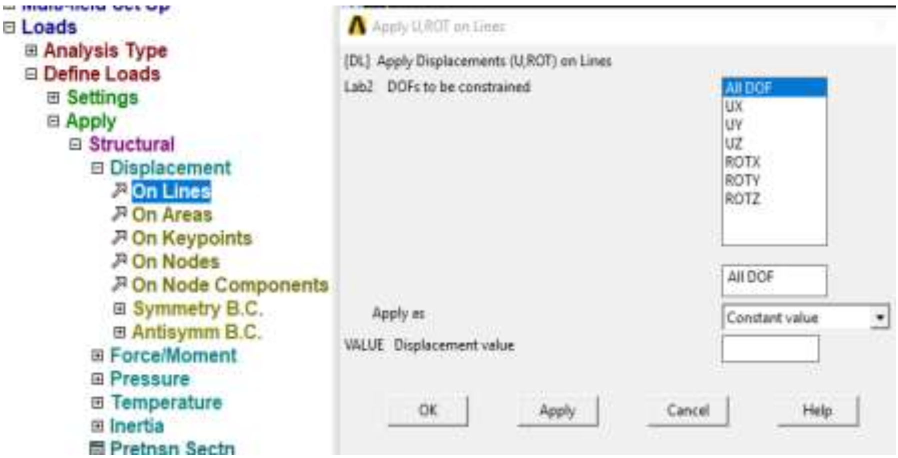


Figure 5.9 – Application of boundary conditions

6. Set the effect of gravity on the construction (Fig. 5.10).

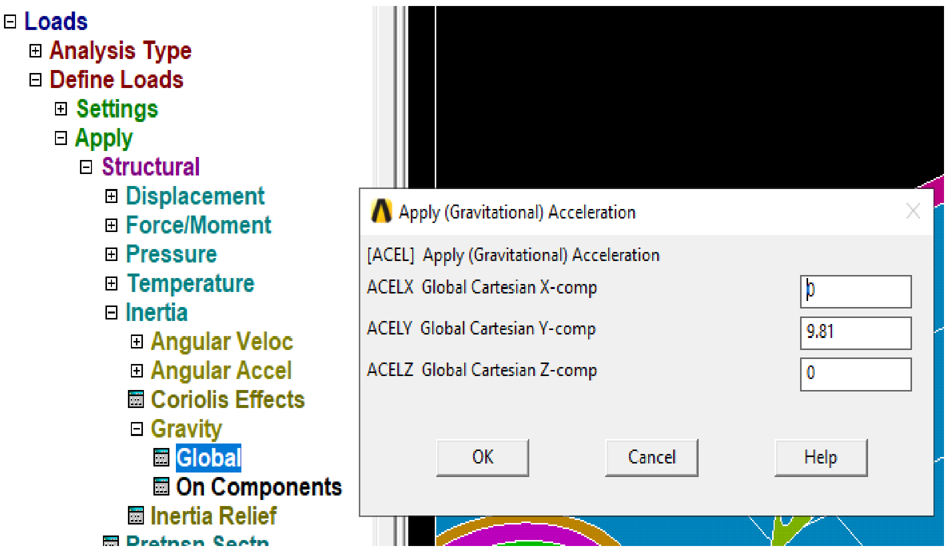


Figure 5.10 - Application of gravity

7. Save the changes to the file.

5.6 Content of the report

Purpose and method of practice. Brief theory. Steps for application of load and boundary conditions. Printout of the file showing applied loads and boundary conditions (if available). Conclusions.

Recommended reading

1. A.Ya. Karvatskyi Finite element method in solid media mechanics problems. Program implementation and visualization of results [Electronic resource]: textbook - K.: NTUU "KPI", 2015. - 391 p.

2. I.V. Pavlenko Finite element method in problems of oscillations of mechanical systems: textbook – Sumy: Publishing House of Sumy State University, 2007. – 180 p.

3. ANSYS Mechanical APDL Structural AnalysisGuide.- ANSYS, Inc. Southpointe 275 Technology Drive, 2013. - 522 p.

6 PRACTICAL CLASS NO. 6
SELECTION OF THE TYPE OF CALCULATION (SOLVER). ANALYSIS OF THE STRESS-STRAIN STATE

6.1 Purpose of the study

Acquaintance with the existing types of calculation (solver). Mastering the basic principles of choosing a solver type. Mastering the main steps of stress-strain analysis.

6.2 General information

The next step after the preprocessing operations is the solution operation (Solve). This stage includes the selection of the solver type, its options, the solution step and the problem launch.

The ANSYS program offers the following types of analysis (Fig. 6.1):

Static – static analysis. It is used to determine movements, stresses, deformations and forces that occur in the construction or its constituent parts under the action of loads that are not accompanied by energy dissipation processes or the appearance of significant inertial effects. Constancy of load and system response is assumed, that is, very slow changes of these parameters over time can be neglected.

Modal – modal analysis - analysis of the construction for its natural frequencies and forms, with its help, the natural frequencies and forms of oscillations are determined. It is used only for the problems of the mechanics of solid deformable body.

Harmoni c – Dynamic analysis. It is used for the problems of the mechanics of solid deformable body, the mechanics of liquids and gases, and electromagnetic analysis, to determine the response of a construction (in the form of movements, deformations, stresses, and forces) to the action of an arbitrary load that varies over time in such a way that inertial forces have to be taken into account effects and processes of energy dissipation.

Transient – non-stationary analysis. It is used for the analysis of transient processes.

Spectrum – spectral analysis. A preliminary modal analysis is necessary. It is used only for the problems of the mechanics of solid deformable body.

Eigen buckling – loss of stability. It is used to determine critical loads and forms of loss of stability in a linear installation. A steady-state analysis with calculation of the prestressed state should be performed beforehand. It is used only for the problems of the mechanics of solid deformable body.

Substructuring – analysis using the method of substructures. It is used to solve all types of tasks.

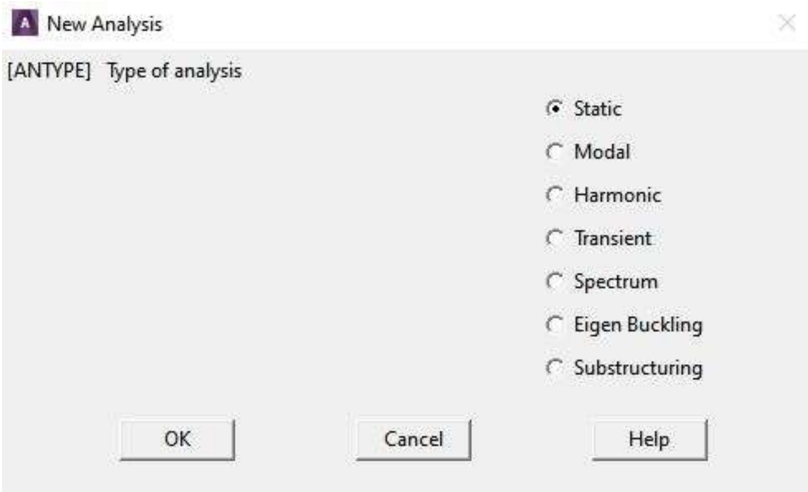


Figure 6.1 – Types of analysis in ANSYS MECHANICAL

The way to choose the type of analysis:

Main Menu → Preprocessor → Loads → Analysis Type - New Analysis or Main Menu → Solve → Analysis Type - New Analysis.

After choosing the type of solver, it is necessary to set the progress of the problem solution (Fig. 6.2), where the scale of movements, the time of completion of the problem solution, the number of steps and substeps of the program, as well as the frequency of recording the steps of the solution are selected.

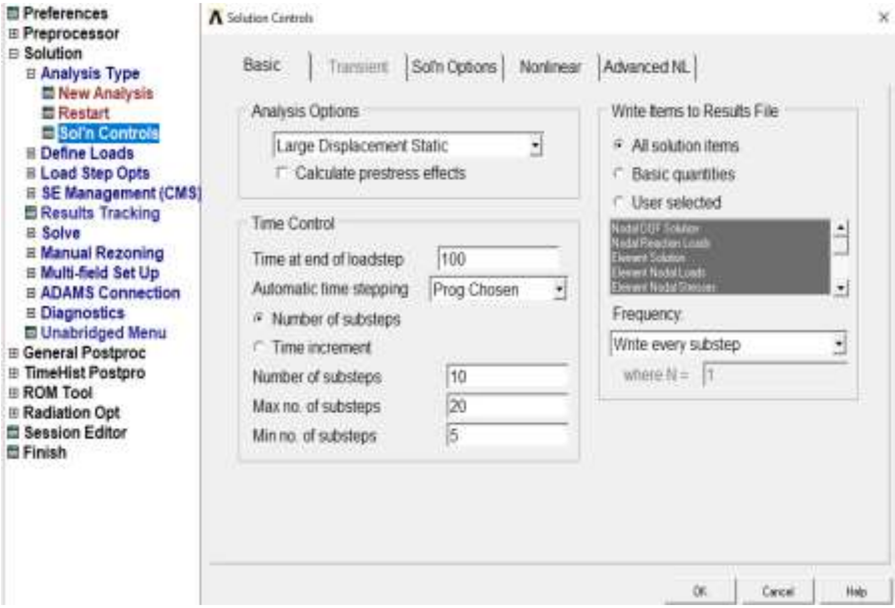


Figure 6.2 – Setting the progress of the problem solution

After all the operations described above (building a model, determining material properties, specifying loads, setting a problem), you need to calculate the project – Main Menu → Solution → Solve → Current LS (Fig. 6.3).

Figure 6.3 – Location of the Solve main menu section

In addition, the ANSYS program has a Load Step function. It allows you to perform the specified number of loads with each step. For example, in the first step, gravity acts on the bar, in the second step it is subjected to compression deformation, in the third to torsional deformation (Fig. 6.4).

Figure 6.4 – Location of the main menu section for Setting load steps

The final stage of the calculation is the analysis of the deformed body (General Postproc). To display the results on the screen, you need to open – General Postproc → Plot Results → Contour plot → Nodal Solution (Fig. 6.5).

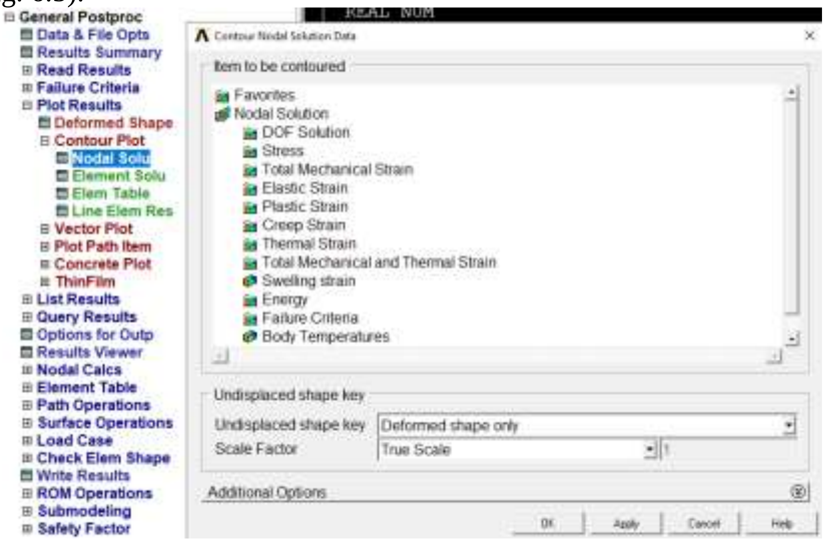


Figure 6.5 – Location of the main menu section for Analysis of calculation results

Depending on what needs to be evaluated and analyzed, select the corresponding sub-item of the menu (Fig. 6.6, 6.7).

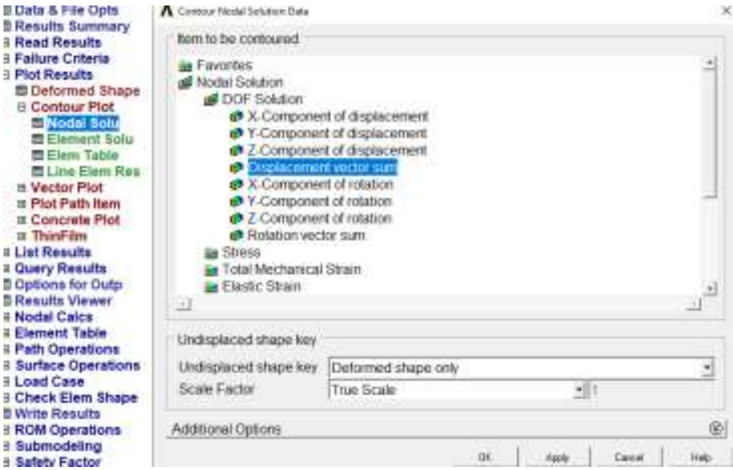


Figure 6.6 – Location of the main menu section for Analysis of movements

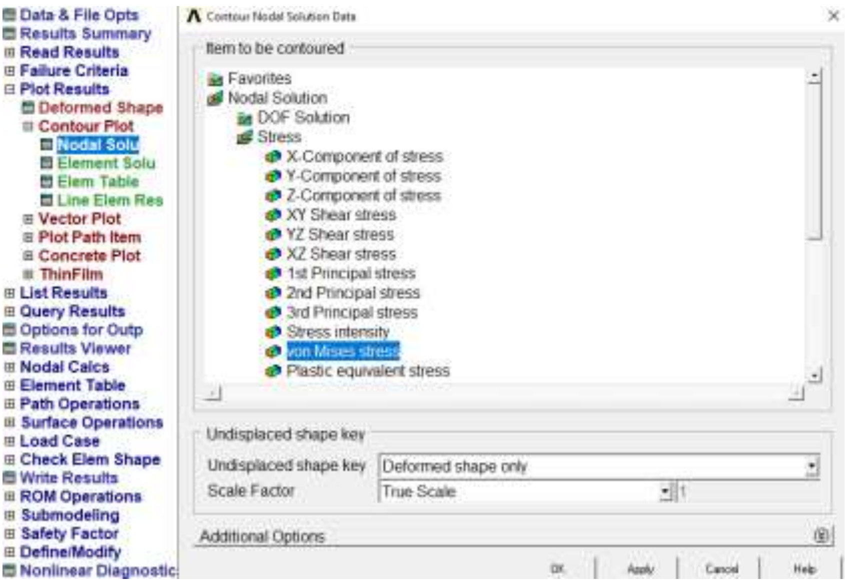


Figure 6.7 – Location of the Stress analysis main menu section

6.3 Control questions for self-checking and control of students' competence

1. List the main problems that are solved using ANSYS?
2. Explain what purpose contact problems can be used for?
3. Name the purpose of modal analysis in ANSYS.
4. State the general purpose of harmonic analysis in ANSYS.
5. Explain the feasibility of the program recording each step of the solution.
6. Give an example of using a static calculation.

6.4 Safety instructions

1. Students are admitted to practice after instruction on labor protection and fire safety.
2. It is forbidden to turn on electrical devices without the permission of the head of laboratory or the teacher.
3. In the event of a fire or electric shock, students must act in accordance with the approved instructions of labor protection and fire safety.

6.5 Working sequence

1. Open the file in the Mechanical APDL Product Launcher (the location and name of the file is specified by the teacher).
2. Select the type of solver according to the task.
3. Make settings before calculation (after consulting with the teacher).
4. Run the file for calculation.
5. Analyze the obtained results.
6. According to the teacher's task, prepare a file for the report and save the file with the calculation.

6.6 Content of the report

Purpose and method of practice. Brief theory. Steps for assignment of solver-type and calculation time settings. Conclusions.

Recommended reading

1. A.Ya. Karvatskyi Finite element method in solid media mechanics problems. Program implementation and visualization of results [Electronic resource]: textbook – K.: NTUU "KPI", 2015. – 391 p.

2. I.V. Pavlenko Finite element method in problems of oscillations of mechanical systems: textbook – Sumy: Publishing House of Sumy State University, 2007. – 180 p.

3. ANSYS Mechanical APDL Structural AnalysisGuide. – ANSYS, Inc. Southpointe 275 Technology Drive, 2013. – 522 p.

7 PRACTICAL CLASS NO. 7 ASSESSMENT OF THE ADEQUACY OF THE MATHEMATICAL MODEL OF THE WELDED STRUCTURE. MESH QUALITY ASSESSMENT

7.1 Purpose of the study

Learn the basic principles of assessing the adequacy of the obtained results. Learn to evaluate the influence of the quality of the finite elements mesh on the accuracy of the obtained results.

7.2 General information

After the calculation is completed, the result is displayed on the screen in the form of a colored isoscale, where the lines of the same color indicate the sections on which the same stresses act (Fig. 7.1) or the same movements are observed (Fig. 7.2).

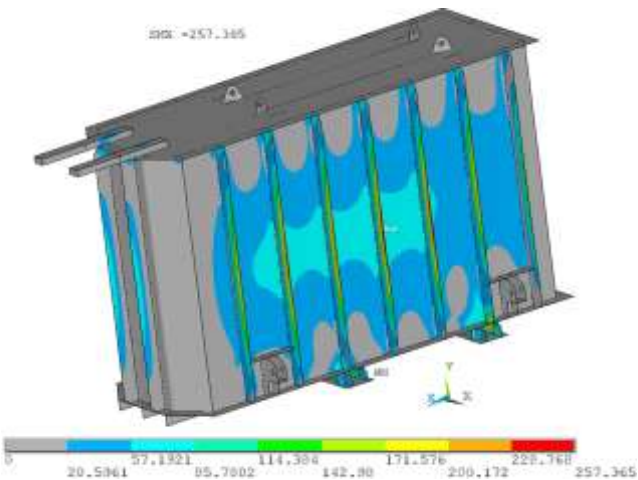


Figure 7.1 – Equivalent stresses when calculating strength, MPa

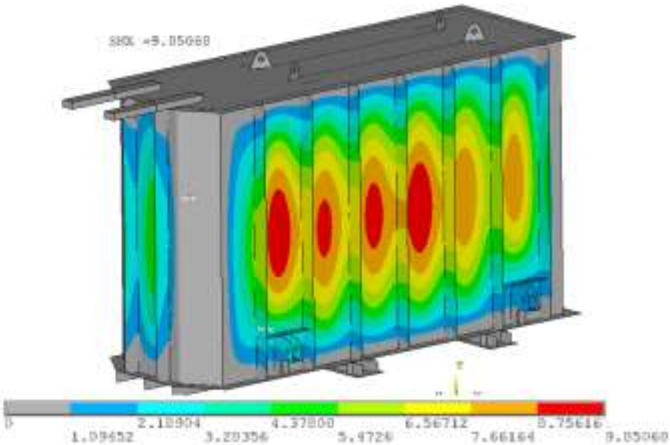


Figure 7.2 – Total movements when calculating the strength, Mpa

Sometimes, for a comprehensive assessment of the behavior of the construction and material, it is necessary to consider the results layer by layer, setting the appropriate options.

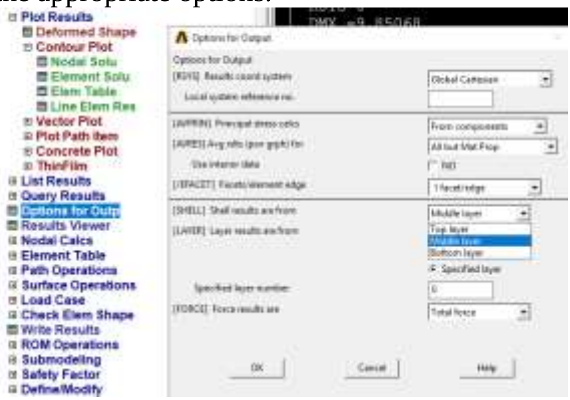


Figure 7.3 – Location of the section for setting options for displaying results on the screen

To assess the accuracy of the calculation results, it is necessary to take into account the quality of the mesh of finite elements. Elements of the wrong shape or size can accumulate errors that distort the picture of the results (Fig. 7.4).

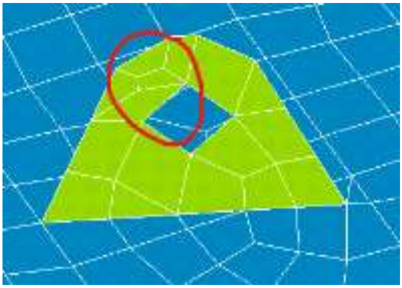


Figure 7.4 – An example of an irregularly shaped element that can lead to a calculation error

The ability to recognize errors caused by a poor mesh comes with experience. This allows you not to redo the model if there are doubts and, thus, to save calculation time.

7.3 Control questions for self-checking and control of students' competence

1. Specify the extension of the database file created by the ANSYS program.
2. Give an example of the content of a file with the extension .dbb?
3. How is a file with the extension .dbb created?
4. Explain in which of the metal layer sections the most reliable calculation results are obtained: upper, middle, lower?
5. Specify the steps for changes to the geometric model or physical and mechanical properties of the material.
6. Specify the sequence of the following actions to change the type of finite elements in order to improve the accuracy of further calculations after obtaining the results.

7.4 Safety instructions

1. Students are admitted to practice after instruction on labor protection and fire safety.
2. It is forbidden to turn on electrical devices without the permission

of the head of laboratory or the teacher.

3. In the event of a fire or electric shock, students must act in accordance with the approved instructions of labor protection and fire safety.

7.5 Working sequence

1. Open the file in the Mechanical APDL Product Launcher (with the task completed in the previous classes).

2. Present the results of the calculation in one form or another according to the instructions.

3. Indicate the places where low-quality elements could cause a calculation error. Suggest ways to eliminate this error.

4. Make the necessary changes to the model. Demonstrate a new result.

7.6 Content of the report

Purpose and method of practice. Brief theory. Steps for viewing the obtained results on a PC and displaying them in reports. Printout of the file with a graphical display of the results (if possible). Conclusions.

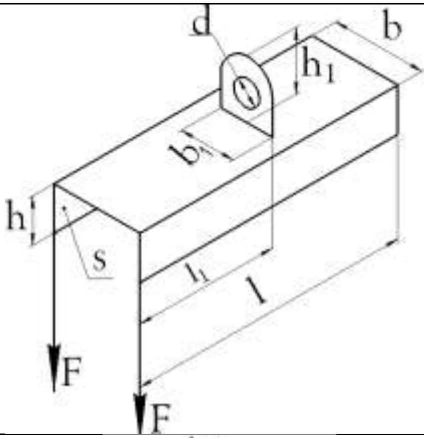
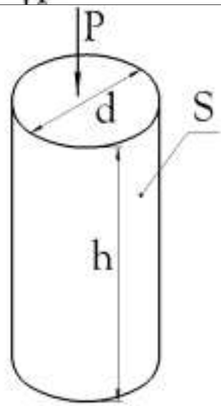
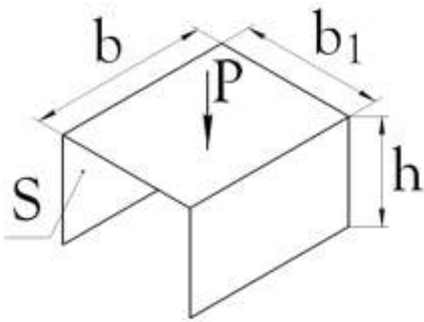
Recommended reading

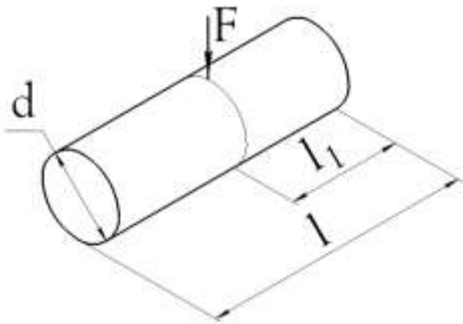
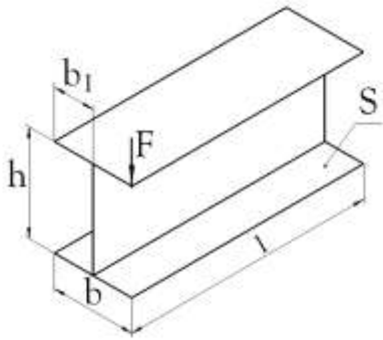
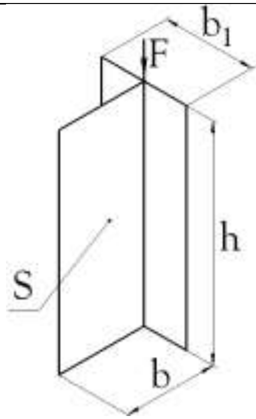
1. A.Ya. Karvatskyi Finite element method in solid media mechanics problems. Program implementation and visualization of results [Electronic resource]: textbook – K.: NTUU "KPI", 2015. – 391 p.

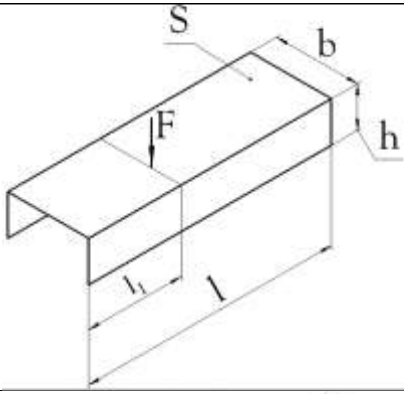
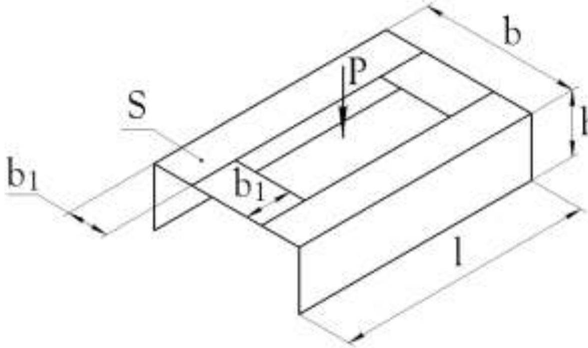
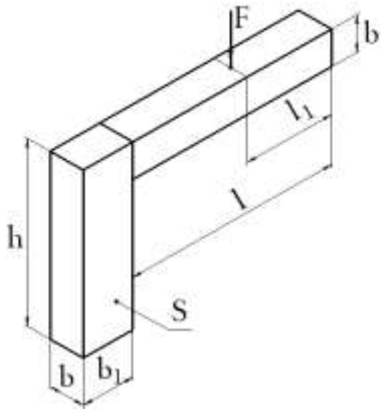
2. I.V. Pavlenko Finite element method in problems of oscillations of mechanical systems: textbook – Sumy: Publishing House of Sumy State University, 2007. – 180 p.

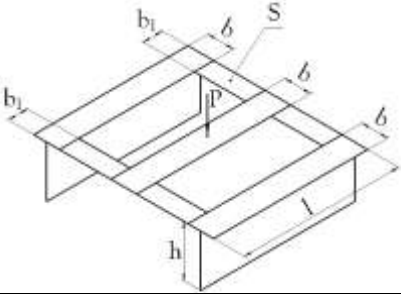
3. ANSYS Mechanical APDL Structural AnalysisGuide. – ANSYS, Inc. Southpointe 275 Technology Drive, 2013. – 522 p.

Appendix 1

<table>
<tr><th>No.</th><th>Construction sketch</th></tr>
<tr><td>1</td><td>
</td></tr>
<tr><td>2</td><td>
</td></tr>
<tr><td>3</td><td>
</td></tr>
</table>

No.	Construction sketch
4	
5	
6	

<table>
<tr><th>No.</th><th>Construction sketch</th></tr>
<tr><td>7</td><td>
</td></tr>
<tr><td>8</td><td>
</td></tr>
<tr><td>9</td><td>
</td></tr>
</table>

No.	Construction sketch
10	

Appendix 2

No.	Dimensions, mm									
	l	l1	b	b1	h	h1	d	S	F, N	P, MPa
1	1000	500	200	100	200	400	25	3	75	0,5
2	1200	600	100	50	200	400	50	2	50	0,25
3	1300	650	200	100	100	200	35	3	75	0,5
4	1400	700	300	150	150	300	50	5	100	1
5	1500	750	400	200	200	400	100	4	120	0,8
6	1600	800	500	250	250	500	150	5	130	1,1
7	1700	850	600	300	300	600	200	6	140	1,2
8	1800	900	700	350	350	700	250	4	150	0,9
9	1900	950	800	400	400	800	300	5	160	1,3
10	2000	1000	900	450	450	900	350	6	170	1,4

Appendix 3

No.	Material
1	09G2S Steel
2	St3 steel
3	12X18H10T Steel
4	Aluminum
5	65G Steel
6	45 Steel
7	13X Steel
8	45G17Ю3 Steel
9	12X18N9T Steel
10	09G2S Steel