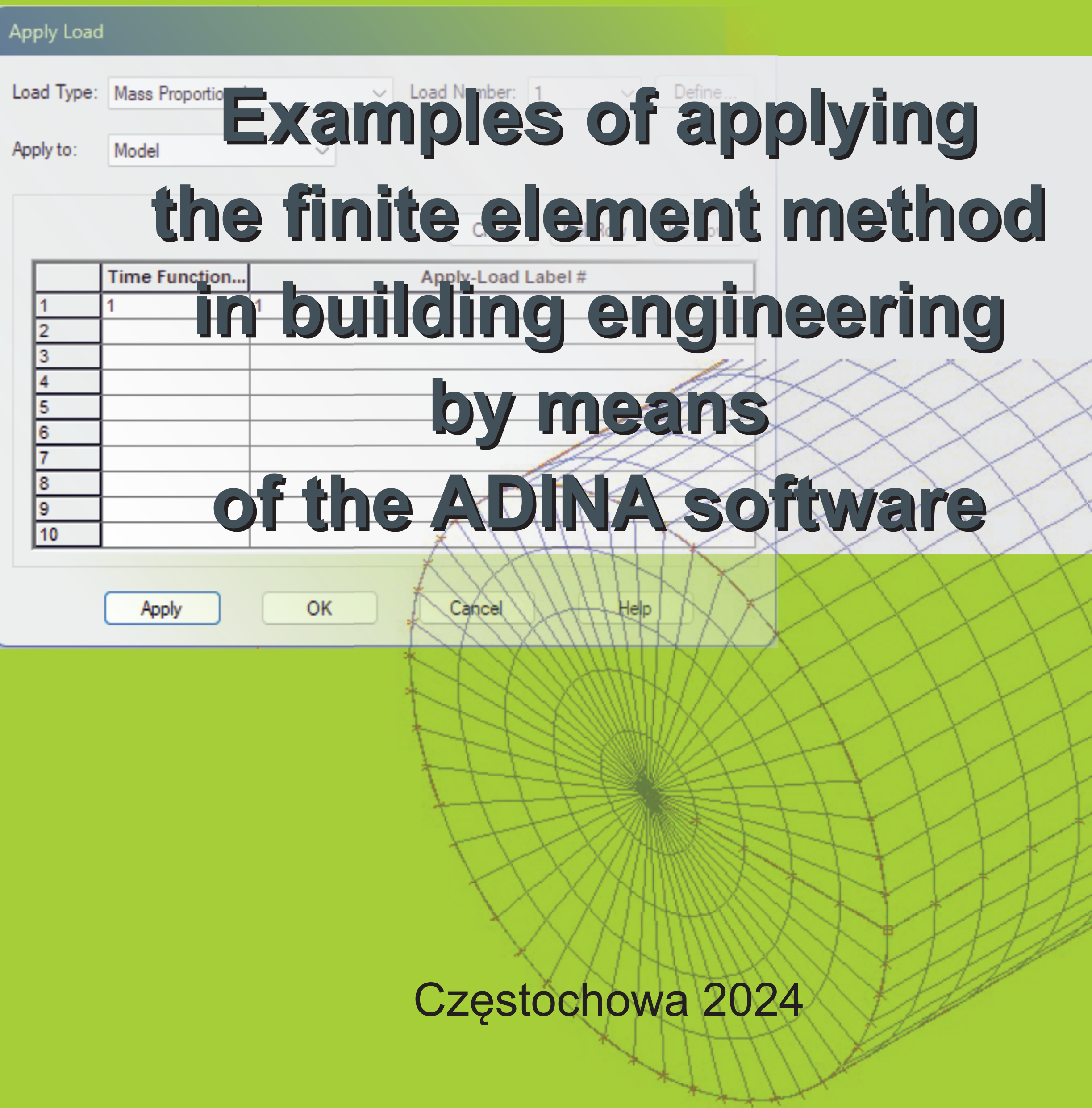
CZESTOCHOWA UNIVERSITY OF TECHNOLOGY

Izabela Major

Maciej Major

Krzysztof Kuliński



Examples of applying the finite element method in building engineering by means of the ADINA software

Częstochowa 2024

Czestochowa University of Technology

Izabela Major
Maciej Major
Krzysztof Kuliński

Examples of applying the finite element method in building engineering by means of the ADINA software

Part one

The Publishing Office of Czestochowa University of Technology

Częstochowa 2024

Reviewers
Krzysztof Cichocki, PhD. Eng. Prof. ANS
Jacek Selejdak, PhD. Eng. Assoc. Prof. PCz

Proofreading
Zdzisława Tasarz

Technical editor
Robert Świerczewski

Cover design
Dorota Boratyńska

e-ISBN 978-83-7193-997-6

DOI: 10.17512/CUT/9788371939976

The Publishing Office of Czestochowa University of Technology
Poland, 42-200 Częstochowa, al. Armii Krajowej 36 B
Editorial Team – wydawnictwo.pcz.pl, phone 34 325 04 80, e-mail: wydawnictwo@pcz.pl
Distribution – sklep.pcz.pl, phone 34 325 03 93, e-mail: sklep@pcz.pl

TABLE OF CONTENTS

FOREWORD

The present book has been written for students of building engineering and mechanics, who during their studies have contact with computer simulations which for their calculations use the finite element method in the ADINA software. The included examples provide simultaneous step-by-step presentation of both the operation of the software and the options related to the modeling of a given physical phenomenon. The examples address the application of both static and dynamic analysis for the calculation of flat rods and surface elements, as well as the calculation of spatial volumetric elements. The book intends to present selected techniques for modeling physical phenomena, and to facilitate work in the environment of the ADINA software.

The finite element method, abbreviated as FEM, is a basic element of computer-supported design. This method is eagerly used by engineers from all over the world who handle computer simulations, and the popularity of the method results from the fact that calculations may involve any model shape for a given type of analysis. We tend to be unaware that some items which are in everyday use have been produced on the basis of earlier analyses using FEM-type software. Such items include, e.g., shoe insoles, hard helmets protecting the head against impact, motors, etc. Of course, besides computer simulations, before a product reaches its receiver, experimental tests are performed on a prototype of a given element, in order to check and confirm the correctness of the analysis which uses the simulations. Moreover, the projects may be way larger, in example: testing the stability of a suspended rope bridge during blowing wind. The design team decides how complex and detailed the prepared model should be, with models which are more representative of the actual object meaning higher quality of the performed calculations. It should be kept in mind that larger models which are more representative of the actual object result in a directly proportional increase in the amount of time necessary to perform the calculations. In a way, this is a drawback; however, in such a moment it should be contemplated how many simplifications should have been made, and how performed calculations should be time costly in reference to the standard method (a piece of paper and a pen). One should also realize how much the results are different from reality when simplifications are used. When performing advanced simulations, it is required that the machine on which the calculations are to be performed should have proper technical parameters; otherwise, the calculations will most likely end with an error. The use of computers for FEM analyses requires certain financial expenses; however, for design offices, the return is relatively fast, since both the pace and the scope of possibilities in the performed simulations increase rapidly.

There are numerous commercial computer programs which use the finite element method for calculations. The most popular programs used all over the world include: ANSYS, ABAQUS, and SOLIDWORKS. These are advanced and complex programs which allow calculations for almost any structures. ADINA is also a commercial program with a similar level of advancement; however, it is mostly popular in the United States, albeit a certain part of European universities and industrial structures eagerly use it to perform their own simulations.

THE BASICS OF MODELING USING THE FINITE ELEMENT METHOD

The finite element method is currently the basic tool of the computer-supported design process. It is a method which involves determining the approximated solutions of partial differential equations. Each time, the fundamental element of a performed numerical analysis is usually some kind of a physical phenomenon or process, since why mathematical formulation of the problem should be developed in the very beginning. Subsequently, the choice of a proper mathematical model for producing a specific solution decides about the usefulness of the yielded results and their match with the actual model. It should be pointed out from the get-go that results do not match the reality perfectly, as models with a high degree of complexity are affected by the adopted simplifications, the size and number of finite elements, the type of the selected elements, etc.

In the ADINA software, before the design of a given object/phenomenon is initiated, it is necessary to choose a preprocessing module in which the alleged model will be created. The software is divided into 5 basic modules, i.e.:

- ADINA Structures
- ADINA Thermal
- ADINA CFD
- ADINA EM
- Post-Processing

The first four modules, i.e.: "ADINA Structures", "ADINA Thermal", "ADINA CFD" and "ADINA EM", are the so-called preprocessing modules, in which construction of the adopted model/phenomenon takes place. The individual modules correspond to the following types of the examined physical phenomena:

"ADINA Structures" – enables a linear/nonlinear analysis of stresses/deformations in rod elements, surface elements (2D and 3D) and volumetric elements (3D), both in statics and dynamics. Moreover, it is possible to declare the nonlinearity of a material model, assume heavy strains, perform a contact analysis, etc.

"ADINA Thermal" – enables an analysis of convection, heat flow, and thermal radiation in surface (2D and 3D) and volumetric (3D) elements. Moreover, it is possible to study the phenomena of freezing and thawing of materials.

"ADINA CFD" – a module related to an analysis of sound wave propagation phenomena as well as the flow of compressible and incompressible liquids. The calculations can be performed for any shape of the surface restraining – the flow of liquids/the propagation of waves. When studying the flow of liquids, both laminar and turbulent flow can be examined.

“ADINA EM” – a module related to the analysis of electromechanical phenomena. In this module, it is possible to define the phenomena of electrical conductivity, examine the propagation of an electromagnetic field, etc.

Although these modules are accessible separately in the software, the internal options enable the combination of various types of analyses. It is possible to simultaneously analyze two modules, e.g., analyze stresses affected by the heat supplied to the model – a combination of the “ADINA Structures” and “ADINA Thermal” modules.

The last module is “Post-Processor” – this is the module in which the results are analyzed in the form of maps, graphs, and tables. This module also enables the combination of various types of analyses from various pre-processing modules when preparing an identical model in terms of geometric and material characteristics.

Once a proper computational model of the preprocessor has been selected, declaration of geometric parameters is initiated. Geometry of the model may be completed directly in the ADINA software, or by means of any CAD-type software, keeping in mind that the file extension of the model exported from the CAD software must correspond to the importing software – ADINA. The supported file extensions include: *.brep, *.dat, *igs, *.nas, *.stl and *.stp. When declaring a geometric model, keep in mind that it should provide the best possible representation of the actual object; however, it should be kept in mind that the representation should not be overdone either, i.e., in a 10-meter concrete slab, there is no sense in declaration of an opening with a diameter of several millimeters. The same applies to roughness, which is not taken into account either. One should also consider dividing the model into parts, if it has axes of symmetry, and perform an analysis of only one part of this symmetric object. Sample divisions of models are presented in Figures 1 for a planar plate and in Figure 2 for a volumetric element.

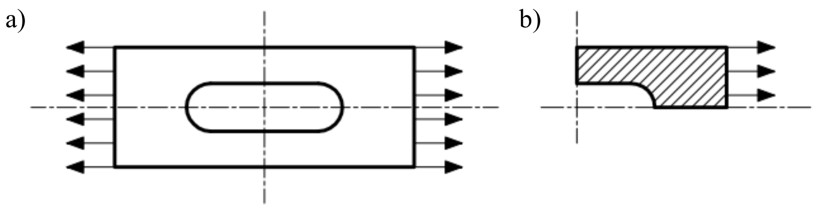


Fig. 1. Delimiting an area to be analyzed for a stretched flat plate. Dash-dotted lines define the symmetry of a given object; a) a model prior to division, b) a separated area which should have been taken to the analysis

Models usually use axial symmetry for planar models, plane symmetry for spatial models, and cyclic symmetry mainly for rotor elements. The division into subareas by means of symmetry or other delimiting criteria enables a more precise analysis of a model, and enables reducing the requirements in the form of: free disk space, the amount of main computer memory necessary to perform the calculations, and the use of the processor.

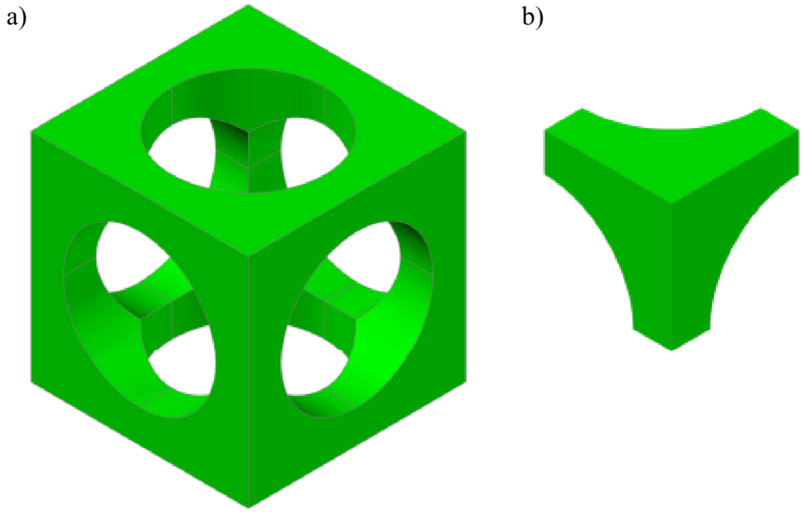
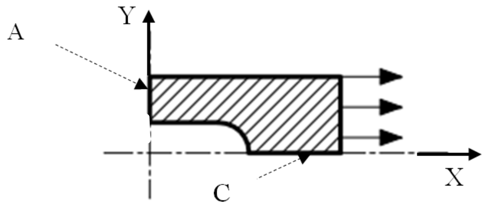


Fig. 2. a) a model of a cube with cut out openings subjected to triaxial stretching.
b) a separated fragment to be analyzed upon applying the abovementioned division of the model into symmetric elements

Declaration of the model or models of materials creating the examined object is commenced after selection of a proper model analysis module and declaration of the geometric model. To this end, the ADINA software offers over 30 various types of material models. The software includes the following material models: elastic, plastic, thermal, with variable and constant creep characteristics, rubber and foam, geotechnical, and others. Each of the material models also has its subcategories, and when the user is not satisfied with any of the definitions prepared in advance, they can access a definition tool, in which they can define their own type of material on the basis of strain and stress curves.

The next step is to declare boundary conditions. Boundary conditions can be assigned to a point, a line, a surface, a 3D figure, as well as a single node or finite element. It should be noted that boundary conditions are indispensable for the numerical analysis, otherwise the model becomes unstable or improperly defined and the calculations are terminated. This is especially significant for objects in which the applied external forces balance themselves out. In spite of the equilibrium of forces (e.g., Fig. 1), an object should be modeled so as to take away the necessary number of degrees of freedom from the body, to ensure blocking the motion of this body on all the considered axes of the coordinate system, but also retaining the freedom of body deformations due to the action of forces. In accordance with the above, for flat objects, it is necessary to take away 3 degrees of freedom, while for spatial objects this number increases to 6.

During one's work, it is also possible to encounter symmetric and antisymmetric boundary conditions. For elements such as those in Figure 1, it is necessary to assume symmetric boundary conditions in accordance with Figure 3.



A – linear fixity
$U_x = 0$

C – linear fixity
$U_y = 0$

Fig. 3. Symmetric boundary conditions

In accordance with the contents of Figure 3 (compared to Figure 1), it is clear that the X direction should be blocked for the line marked with the letter "A", while the Y direction should remain unblocked, since deformations occur in this place due to the applied load. A similar case applies to the line marked with the letter "C"; however, the blocked directions is Y, and the unblocked direction is X, since on the line of the axis of symmetry overlapping the X axis of the global coordinate system some deformations occur due to the load. Despite it results in taking away 2 degrees of freedom the analyzed object is unable to move. Discussed approach is the simplest for defining symmetric boundary conditions, which leaves the possibility of free rotations for finite elements along lines marked with "A" and "C". Despite that, in other different cases it may have been required to block the rotation in regard to the Z axis pointing upwards/downwards at the crossing of X and Y axes.

After the declaration of boundary conditions, the next step is usually to define and declare loads. Almost all types of loads have been implemented in the ADINA software, including the possibility to create their combinations. Since all the definitions in the software are without physical units, one should take extra care to ensure unit compliance. The easiest way is to operate on simple metric unit system (SI) or on imperial system, respectively. In practice, the SI system is the one used the most frequently.

Having a computational model created this way (virtual bar/shell/body defined, all boundary conditions, all loads, the number of time steps, the functions of the load factor over time, possible load combinations etc.) one progresses onto 3 final steps before the performance of calculations. The first among these steps is to define groups of elements – in the next step, this allows for assigning to a given geometric object a material model, a cross-section, the type of calculations (small/large displacements), the moment of bringing an element to "life" or excluding it from the analysis in a given time unit, etc.

The penultimate step is to divide geometry for the finite elements. The compliance of numerical calculations with the actual values of the examined characteristics/properties of a phenomenon/process depends mainly on discretization of the area (mesh density). It should be kept in mind that adopting a mesh with the same dimensions for the whole model may lead to errors in calculations. The mesh should have increased density in locations such as the corners, the connections of two different elements (made of two various material models), contact joints, places in direct vicinity of a concentrated force, etc. According to that it is clear that in all places where a concentration of stresses occurs, the number of mesh elements should have been increased.

As already previously mentioned, the finite element method involves the determination of approximated solutions of partial differential equations, therefore, using a coarse mesh (relatively large mesh size) for the whole model, it may end in underestimation of stresses in the place of their concentration. Figure 4 presents the essence of the problem.

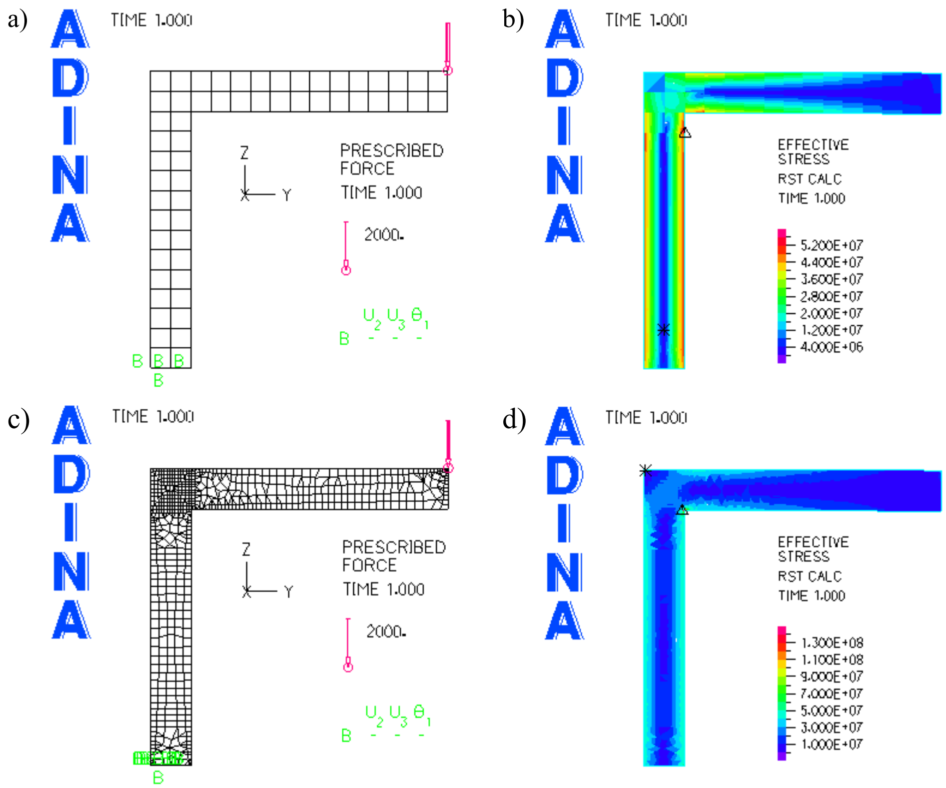


Fig. 4. a) an example of the application of a coarse mesh, b) results for a coarse mesh, c) an example of the application of a densified mesh. d) results for a densified mesh. The "jagging" of the map of stresses results from the fact of not using the smoothing function (an intentional action)

In accordance with what can be seen in the figures, the numerical values of stresses for a coarse mesh are underestimated, in contrast to an increased mesh density. It should be noted that the absence of map smoothing function use was intentional, in order to show how much the final results, depend on the regularity of the shape of finite elements, as well as on their size. In the case of Figure 4b, it is apparent that the colors of the presented resultant map do not change smoothly, passing from one finite element onto the other in a visible steps. These visible steps are connected with the adopted too large mesh size. For the mesh in Figure 4d, the change in map colors between one finite element and the other is rather smooth, while the effects of "jagging" are associated with the irregularity of the mesh elements.

When the smoothing technique function is applied, the resultant map of stresses ultimately looks like in Figure 5. It should be kept in mind that the division into finite elements in the case of planar (2D) and spatial (3D) models proceeds in FEM software semiautomatically – i.e., the user inputs the final parameters (the density of mesh division for finite elements) for the edges defining a given surface or 3D figure, then the software performs discretization of the inner area of the input body by means of complex algorithms.

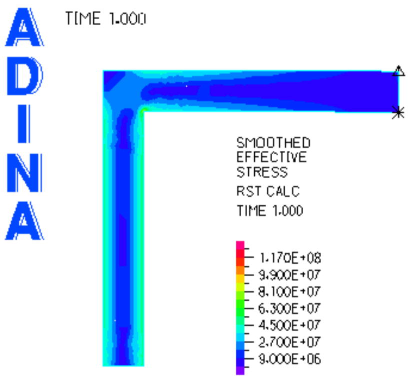


Fig. 5. A smoothened map of reduced stresses for a densified mesh of finite elements

The dimensions of finite elements are closely related to the division for elements declared by the user, and to the automatic internal discretization adopted by the software. The density of dividing the edges of the body adopted by the user usually causes the mesh created inside a given object to have a mesh size no larger than the largest assumed segment of discretization.

Finite elements are implemented when proper discretization of the area has been adopted-depending on type of analysis and material, finite elements should have assigned required properties. In example in static analysis there should have been adopted at least: Young's modulus and Poisson's ratio. The precision of calculations depends to a certain extent on the adopted finite element type (Fig. 6). For example, when designing a displacements of a beam made of a circular tube, results may highly deviate from reality pyramidal (4-node) spatial finite elements (3D) are used. When the same mesh is used, but with the implementation of prismatic (8-node) objects, the results will correspond to the actual displacement values (errors will only be related to the approximation of the solution). Generally, the higher the number of nodes in a simple finite element the higher the precision of reproducing the actual behavior of the model resulting in a greater precision of calculations. On the other hand, simultaneously the time needed to complete the calculations increases significantly as well as the demand for computer RAM size and processor unit speed. Moreover, some of the elements better describes one material, whereas the other ones describe better different material. Before choosing a given finite element type, it is necessary to get acquainted with guidelines regarding its usage. These guidelines can be found in technical documentation attached to the software.

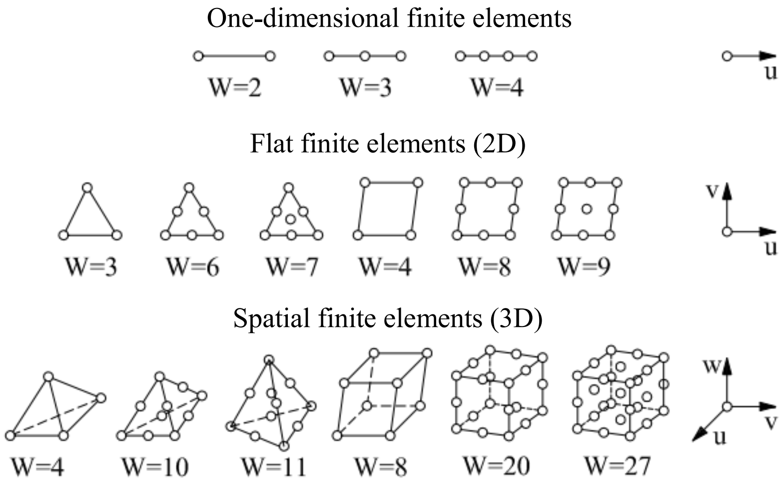


Fig. 6. Types of finite elements available in the ADINA software

When implementing the finite elements, the user can choose among three types of division of the internal structure of an object:

– "Rule-Based" – causes the elements to be evenly distributed. This option is usually applicable when the designed element has a simple shape.

As can be seen in the Figure 7, the applied mesh is regular and symmetric in the case of quarter circles, while the sizes and shapes of some finite elements are incorrect. Calculation of the abovementioned model would cause erroneous results in certain points of the model.

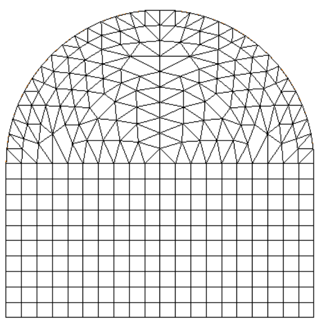


Fig. 7. Using the "Rule-Based" function with 4-node elements and forced triangular shapes of finite elements: "Preferred Cell Shape: Triangular"

"Free-Form" – any distribution of the elements (adopted automatically by the software on the basis of the input density of the division of edges describing an object).

That option works for larger projects with highly irregular shapes. Using this option in the model prepared in Figure 7 results in a division like in Figure 8.

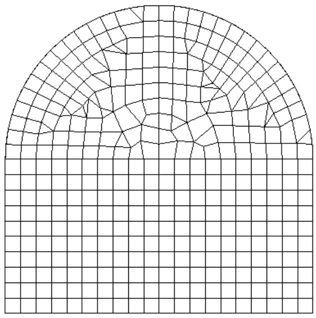


Fig. 8. Using the "Free-Form" function with 4-node elements

As seen in Figure 8, the applied division is irregular in the case of quarter circles. Moreover, similarly to the model of Figure 7, some of the finite elements have different shapes, which further down the line can lead to erroneous calculations in certain areas of the model.

– "Degenerated" – the last option which enables considering a surface with 3 vertices (triangular surfaces, quarter circles, etc.) as a surface in which one of the vertices is a main node, from which a mesh extends towards the remaining nodes specified by the user's density of division. The use of this function is presented in Figure 9.

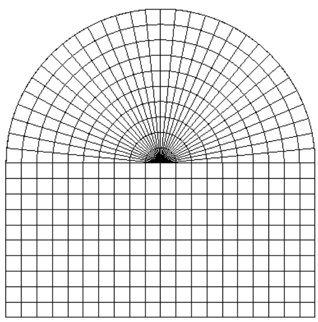


Fig. 9. Using the "Triangular Surfaces Treated as Degenerate" function adopting 4-node finite elements

It should be kept in mind that during the implementation of finite elements, a very important factor contributing to the correctness of future calculations is to fine-tune the discretization of the area, so that the transition between the consecutive elements would be smooth, and moreover, the discretized area would not contain elements with strongly deformed shapes.

Finally, it should be added that semiautomatic generation of a mesh by the software entails certain problems. One of them is the fact that the user inputs the density of division only for the envelope of a given body, having no impact on the distribution of finite elements within certain areas of this body. With the geometric model presented in Figure 10, the area should be densified near the application of the concentrated force.

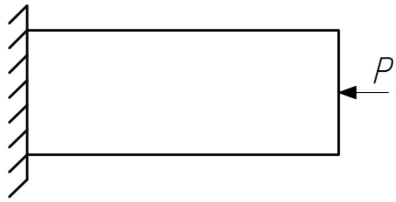


Fig. 10. A flat geometric model with an adopted concentrated force

The preparation of a regular mesh with elements in the shape of rectangles for the input area, and the division of the entire envelope by rectangular finite elements with identical dimensions and a regular mesh, cause the resulting mesh to look like in Figure 11.

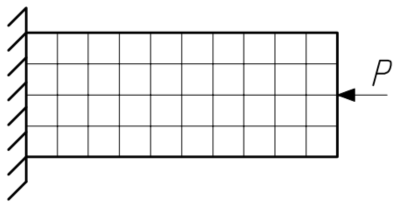


Fig. 11. A regular mesh of rectangular elements

For a very large object, the above schematic drawing causes the necessity to use a finer mesh. It is possible to increase the mesh density for the entire model, which automatically results in greater demand for the processing power of the computer, and in increasing the demand of the main memory. It is also possible to increase density on a single edge or several edges, allowing for resolution of the problem. A sample view of a finite element mesh with applied increased density of a single edge is presented in Figure 12.

In accordance with the figure above, when the software is ordered to use rectangular elements, it becomes impossible to discretize an area by means of the selected elements, which results in the appearance of triangular elements. Figure 13 presents the density increase of a mesh when several edges have been selected, so that the mesh would only contain rectangular elements.

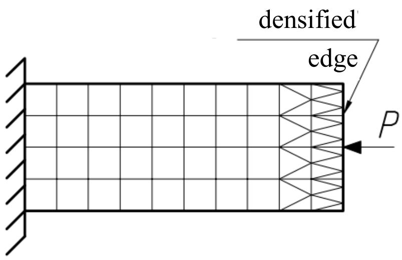


Fig. 12. Applying the densification of a single edge

An analysis of the Figures 12 and 13a, 13b suggest that Figure 13b presents the most preferable form of meshing. However, in such a case it is necessary to consider whether the elements located in close vicinity to the central point of the object are not too large.

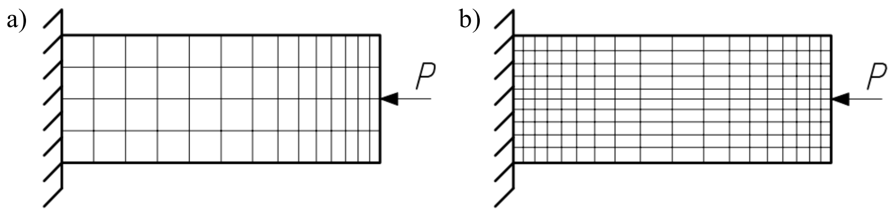


Fig. 13. a) densification of the upper and lower edge of an object near a concentrated force, b) densification of 4 edges near the places of stress concentration

When it is required for the zone near the concentrated force to be densified in a circular/elliptical manner, once again we obtain the presence of triangular elements. Moreover, in order to enable the preparation of such a model, it is necessary to create two separate geometric objects, i.e., a rectangle with a cut out circle/ellipse, and a part of an ellipse. The essence of the problem is presented in Figure 14.

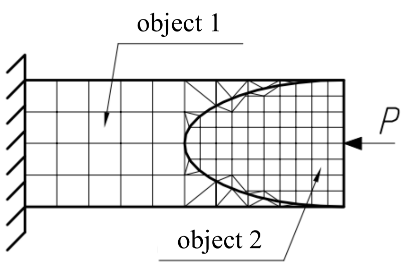


Fig. 14. Elliptical densification of a mesh

In Figure 14, object 1 and object 2 are two subareas of the main object, which is a bar in the form of a rectangle. The preparation of the abovementioned model is quite complicated, especially when the object is drawn in the ADINA software.

In such a case, the fastest method is to use CAD-type software, and import the prepared model into the ADINA software.

It is less problematic to prepare a model consisting of two rectangles in the ADINA software (Fig. 15); however, this also entails the problem of the generation of triangular elements along the contact of two areas – an area with an increased mesh density, and an area with standard mesh.

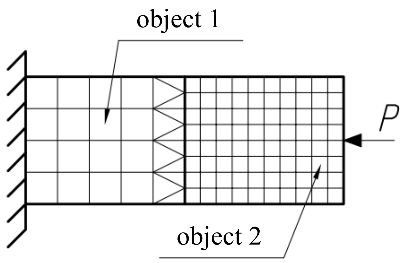


Fig. 15. Increase of mesh density in one of the rectangles of an existing model

In accordance with the abovementioned considerations, it should be assumed that it is not always possible to prepare a model from an identical type of elements. Moreover, already at the stage of inputting the geometric model, the user must be able to predict the places of the model in which concentration of stresses may occur, and create the geometric model in a way which enables mesh density increase for smooth transition of the dimensions of finite elements as well as prevent creation of these elements with irregular shape.

In addition, when analyzing the computational model, if only possible, separate partial calculations should have been performed which would enable checking the correctness of the adopted assumptions. It should be kept in mind that a machine is capable of calculating virtually anything, if all the criteria concerning declaration have been fulfilled; however, the final assessment of the usefulness of the results and the correctness of the performed calculations always constitutes the responsibility of designers modeling a given subject. Therefore, it is required for the design team to have knowledge on the studied subject, and to have knowledge regarding the operation of a given computational tool.

AN INTRODUCTION TO USING THE SOFTWARE

First work on the ADINA software started in 1974. The pioneer working on the software was Klaus-Jürgen Bathe, who in 1975 was given a position in the Department of Mechanical Engineering at the Massachusetts Institute of Technology. In 1986, the ADINA R&D, Inc. company was founded in order to support the development of the ADINA system. Almost 30 years have passed since the founding of the company and the commercial release of the first software. During this period, the system of the software was in constant development, undergoing multiple modifications with respect to the input of the geometric model as well as the optimization of the solver and the addition of new functional modules. The entirety of the present publication is based onto versions up to ADINA AUI 23.0.0. It is recommended for the users to have at least the same or newer version of the software, since older versions of the software may not include certain functions used in the examples, or the location of certain functions may be different from what is presented.

In order to run the software for the first time (having installed the ADINA suite), on a computer with the Windows system go to the "Start menu/All Programs/Bentley Engineering/ADINA CONNECT Edition/ADINA 23.00 User Interface (AUI)" or use the "AUI 23.00.exe" desktop shortcut. Alternatively, go to the location where the software is installed, e.g., "C:\ProgramFiles\Bentley\ADINA\23.00\bin\AUI.exe".

After running the software, a welcome window will appear, following which the software is ready to use. A view of the main window of the program is presented in Figure 16.

In order to increase the clarity of further presentation of the program's functionality, the background display parameters have been changed to white. The way in which this is accomplished will be described in a moment.

The user's communication with the software is quite simple, i.e., similarly to other programs in the Windows system, settings/selections are accomplished in menu tabs, by means of buttons, command lines, etc.

Areas of communication with the program are presented in Figure 17.

The areas of communication listed in Figure 17 stand for, respectively:
1. upper menu tabs
2. button and list box toolbars
3. computational model tree
4. main modeling window
5. program command line window
6. list of program messages
7. list of current session messages

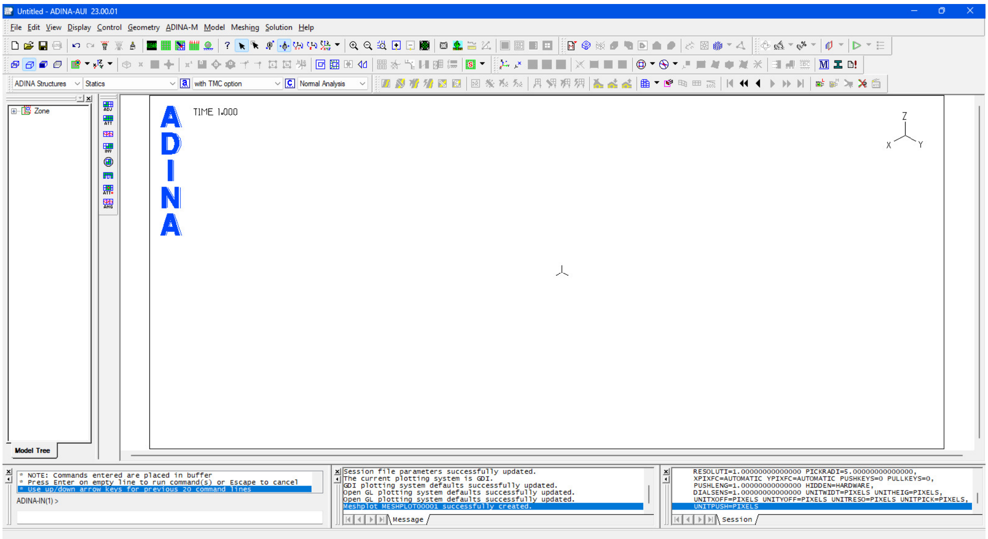


Fig. 16. Main program window

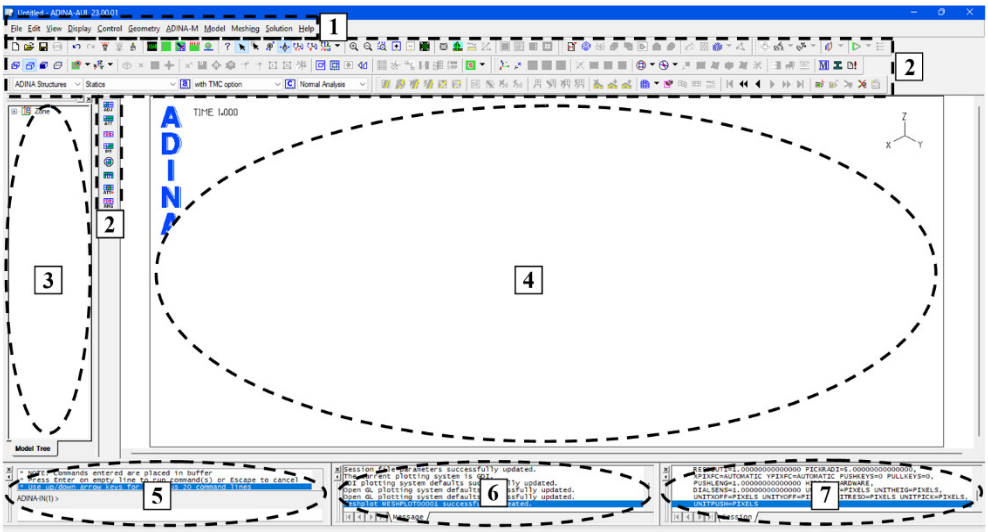


Fig. 17. Areas of the user's communication with the software. Description in the text

The upper menu tabs enable basic operations, such as saving/loading/closing, etc. of the program. The tabs also include all options possible to be input, related to object modeling – modeling both the shape geometry as well as computational parameters. All the available tabs for "ADINA Structures" along with the functions will be discussed in a separate paragraph.

The button and list box toolbars include the most frequently used options involving the modeling of shape geometry and computational parameters.

All the buttons for “ADINA Structures” and their functions are described in the ADINA primer, which can be accessed from “Help/Primer (pdf)…”. Buttons frequently used in the problem topics of this book will be described in the further separate chapter.

Note: Not every function of the program has its equivalent in the form of a button/ combo box.

The computational model tree comprises a list of currently used definitions and declarations, i.e., declarations of external forces, boundary conditions, material properties etc. and definitions of parameters related to the shape of the model. The functions in the tree can be used to modify the existing assumptions without the necessity to search for functions among buttons or upper menu tabs.

The main modeling window – this place shows in graphical form the model geometry along with other parameters, like, e.g., the applied external forces, the adopted boundary conditions, etc.

Note: Not all parameters may be displayed. For example, it is not possible to display the parameters of the initial state. In example it is not possible to display initial stresses graphically in the preprocessing module.

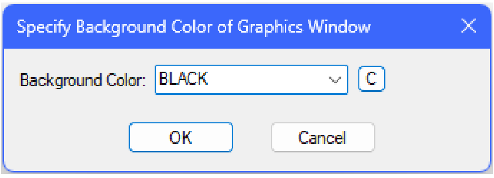
The program command line enables inputting computational and geometric parameters, material properties and others by means of specialized commands. A full list of commands can be found in the help files of the ADINA software.

List of program messages is a window in which the program uses text messages to present the function type which has currently been input, whether the opening/saving of a file was successful, etc.

The list of current session messages gives a possibility to read, what information are being saved in a temporary “xxx.ses” file. Each operation done in the ADINA environment i.e. change of the title, assigning material properties, input of geometry points, lines etc. are being reflected as a list of commands appended in the “xxx.ses” file. The “xxx.ses” file is a temporary file stored in the “C:\Users\Current User\AppData\Local\Temp\” folder. As soon as user saves a project file through the “File/Save as…” also the “xxx.ses” file is saved in the same location where the project save has been done. That action prevents the “xxx.ses” file to be deleted after the computer reboot.

As mentioned before, the background color of the main modeling window can be changed. In order to do this, choose “Edit/Background Color…” from the upper menu tabs. When a new window opens, choose one of the predefined colors from the selectable list, or click the “C” button and choose a color from the color palette of the Windows system. These windows are presented in Figure 18.

Note: In the drop-down list of the "Specify Background Color of Graphic Window" window, it is possible to enter colors of the RGB system using keyboard, e.g., #FFFFFF.

a) 

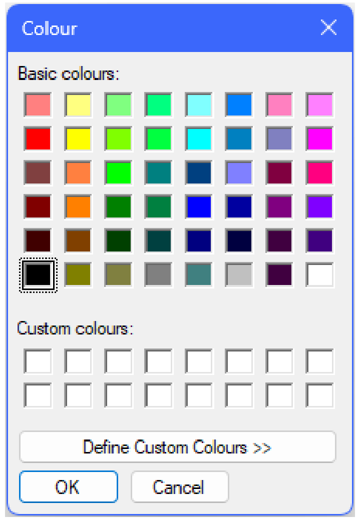
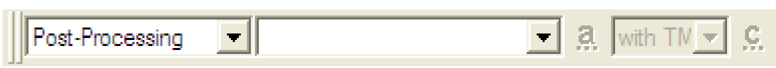
b) 

Fig. 18. a) A selectable list window; b) Choosing a color from the color palette of the Windows system after clicking the "C" button in the "Specify Background Color of Graphic Window" window

Regarding the examples provided in the present book, each time after running the program make sure that in the area of buttons and list boxes toolbar, the following drop-down list

has the "ADINA Structures" function active.

In order to go to the module responsible for displaying the results, choose the "Post--Processing" option from the same drop-down list.



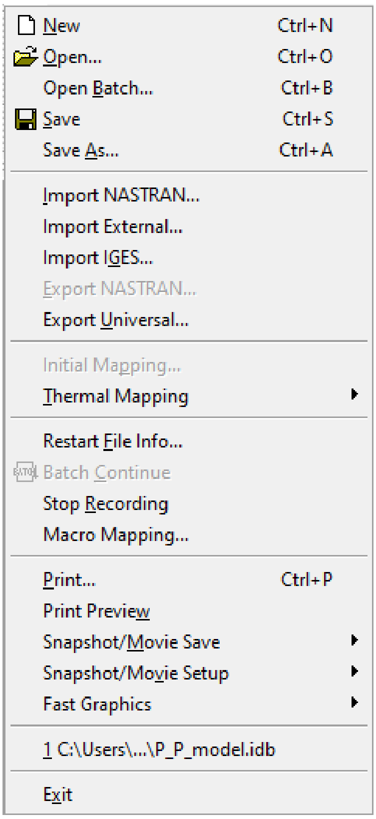
Note: Before commencing execution of the examples, it is recommended for the reader to once again get acquainted with the descriptions of the upper menu tabs and their functions, as well as the descriptions of individual buttons.

We wish you fruitful work, dear reader!

CHARACTERISTICS OF MAIN MENU TABS IN THE ADINA-STRUCTURES SOFTWARE

Starting from the top left side, all functions of the software will be presented along with short descriptions.

The “File” window



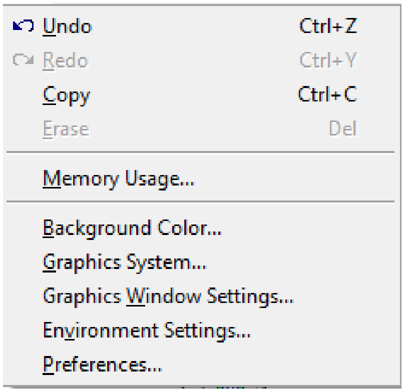
1. “New” → open a new model window
2. “Open” → open a file: a model (database) “*.idb”, a model (database) with a command line “*.in”, session file “*.ses”, plot with a command line “*.plo”, a porthole results file “*.por”, plot file (database) “*.pdb”,
3. “Open Batch” → open and process an input file written as commands
4. “Save” → save the current file

5. “Save As…” → save the current file as…
6. “Import NASTRAN…” → import a data file from the NASTRAN software
7. “Import IGES…” → import a data file from a CAD-type environment
8. “Export NASTRAN…” → export model file to a format supported by the NASTRAN software
9. “Export Universal…” → export file to the *.unv format
10. “Initial Mapping…” → load a file containing initial conditions
11. “Thermal Mapping”
    – “Define…”→ define a new thermal mapping file containing temperature and temperature gradient loads
    – “Delete…” → removes a thermal loads and gradients specified in a file from current model
12. “Restart File Info…” → list of information about existing restart file
13. “Batch Continue” → continue reading and processing a batch-type input file
14. “Stop Recording” → stops recording steps taken by the user in the session “*.ses” file

    “Macro Mapping…” → assign a text file as a macro mapping file
15. “Print…” → opens a window with print properties
16. “Print Preview” → opens a print preview window for the current model
17. “Snapshot/Movie Save”
    – “Snapshot (vector)…” → saves the current image as a vector graphic type
    – “Snapshot (bitmap)…” → saves the current image as a bitmap form
    – “Movie (bitmap)…” → saves the frames from a current movie created in the model as a movie picking bitmap graphics in the .avi file
    – “Movie (vector)…” → saves the frames from a current movie created in the model as a movie picking vector graphics in the .avi file
18. “Snapshot/Movie Setup”
    – “PostScript…” → gives the possibility of specifying parameters associated with the PostScript plotting system
    – “Adobe Illustrator…” → gives the possibility of specifying parameters associated with the Adobe Illustrator plotting system
19. “Fast Graphics”
    – “Save Animation (ani)…” → saves current model animation in the .ani format
    – “Load Animation (ani)…” → loads animation from an .ani file into the current model window
20. “C:\Users\ …” → a list of few previously used files

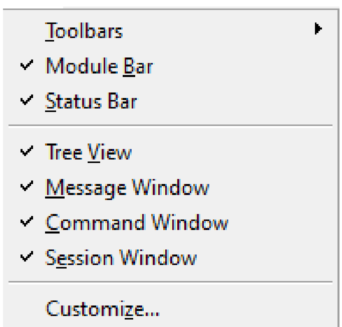
    “Exit” → exit the program

The “Edit” window



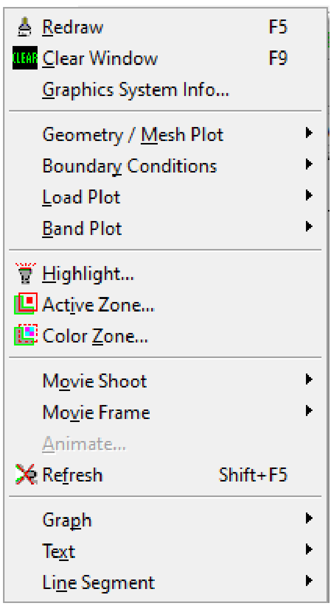
1. “Undo” → undo the last performed operation
2. “Redo” → redo a previously undone operation
3. “Copy” → gives the possibility of copying model/plot fragment
4. “Erase” → selected item is erased from the graphic display (The item is not deleted from the model!)
5. “Memory Usage…” → specify the amount of memory used by the program
6. “Background Color…” → the background color of the main window
7. “Graphics System…” → display mode options
8. “Graphics Window Settings…” → display options for the main window of the model
9. “Environment Settings…” → options allowing adjusting program interface settings
10. “Preferences” → interface preferences like prompting for labels, confirmation via space-bar for picking objects etc.

The “View” window



1. Toolbars
   - “General” → a bar with general buttons
   - “Modeling” → a bar with buttons responsible for modeling
   - “Adina-M” → a bar with buttons related to the Adina-M module
   - “Display” → a bar with display buttons
   - “Results” → a bar with buttons related to the results
   - “Macro” → a bar with buttons related to recorded macros
   - “Pick” → a bar with buttons related to pick/select items in the project
   - “Fast Graphics” → a bar with buttons related to fast graphics
   - “Reset” → resets the visibility of toolbars to the defaults settings
2. “Module Bar” → a bar with analysis selection buttons
3. “Status Bar” → toggles on/off status bar in the main window
4. “Tree View” → a side bar specifying the characteristics of the model
5. “Message Window” → toggles on/off the message window
6. “Command Window” → toggles on/off the command line window
7. “Session Window” → toggles on/off the session commands window
8. “Customize…” → enables customizing the buttons of the existing toolbars

<u>The “Display” window</u>

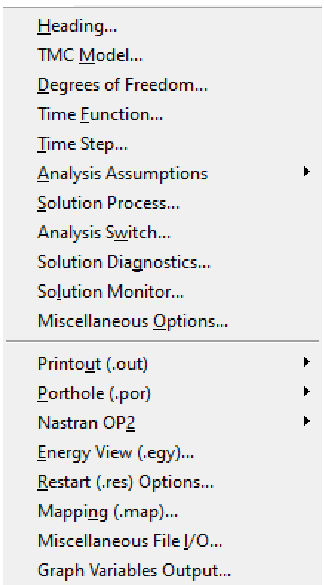


1. “Redraw” → refresh the model drawing
2. “Clear Window” → clear the main window of the program (the model is not deleted, becomes invisible)

3. “Graphics System Info” → information about project heading, current window width/height and plotting system
4. “Geometry / Mesh Plot”
   - “Use Default” → show the model with default settings
   - “Use Zone…” → allow creating a mesh plot of a zone using default settings
   - “Use Style” → allow to create/specifies the style used for mesh plots
   - “Modify…” → modify parameters responsible of displaying parts of/whole model
   - “Change Zone” → change the displayed model area
   - “Define Style” → define the display style
   - “Select Groups” → select contact groups/elements to display
   - “General Element Connectivity…” → indicate connections between nodes for general elements
5. “Boundary Conditions”
   - “Default” → show boundary conditions in the default style
   - “Custom…” → display settings for the boundary conditions
6. “Load Plot”
   - “Use Default” → show model loads with default settings
   - “Use Style…” → show model loads using a style
   - “Modify…” → modify the style of the model loads
   - “Define Style…” → define the display style for the model loads
7. “Band Plot”
   - “Create…” → create a plot with chosen properties for current model using user-defined or default style settings
   - “Modify…” → change of properties in an existing plot
   - “Define Style…” → various style options related to plot visualization
8. “Highlight…” → highlight the selected fragment/part/whole model
9. “Active Zone…” → specify the list of selected active areas
10. “Color Zone…” → assign color to the area
11. “Movie Shoot…”
    - “Rotation…” → create a movie through a rotation around the model
    - “Load Step” → record a movie in a specified time range
    - “Cut Plane…” → record the movement of a cut plane through the model
    - “Iso-Surface…” → record the movement of an iso-surface through the model
12. “Movie Frame…”
    - “Define…” → define a single movie frame
    - “Show…” → show the movie in the specified frames range
    - “List…” → a list of created and stored movie frames
    - “Delete…” → delete a specified range of frames in a movie
    - “Remove All” → delete all movies in the current project file

13. "Animate…" → create an animation by specifying time intervals between the consecutive movie frames and movie cycles
14. "Refresh" → refresh the model with the movie
15. "Graph"
    - "Material Stress-Strain Curve…" → display a stress-strain graph for a selected material
    - "Time-Dependent Material Curve…" → display a graph for a time-dependent material
    - "Define User Data…" → define data for a graph with X and Y axes
    - "Plot User Data…" → display a XY graph from the defined data
    - "Modify…" → modify elements related to graph visualization
    - "List…" → a list of the user's graphs along with defiend data
    - "Define Style…" → define the display style of the user's graphs
16. "Text"
    - "Define…" → displays a window for defining the user's texts
    - "Draw…" → displays the selected text defined by the user on the screen with a possibility to change text attributes
17. "Line segment"
    - "Define…" → displays a window for defining a line with the X- and Y-axis coordinate
    - "Draw…" → displays the selected line defined by the user on the screen with a possibility to change its attributes

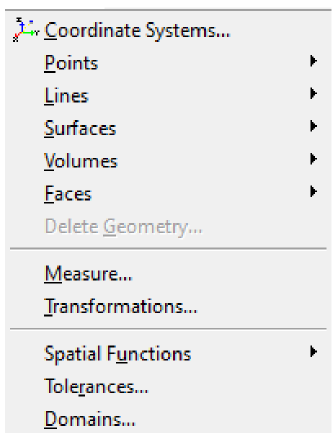
<u>The "Control" window</u>



1. “Heading…” → a window for entering the model heading
2. “TMC Model…” → settings of a thermomechanical coupling analysis
3. “Degrees of Freedom…” → specification of master boundary conditions used in the model
4. “Time Function…” → set the options of load value factors in specified time intervals
5. “Time Step…” → definition of total time steps and their intervals in performed analysis
6. “Analysis Assumptions…”
   – “Fluid Structure Interaction…” → options related to interaction of ADINA structures with the ADINA CFD module
   – “Kinematics…” → specify the default formulation regarding calculations of displacements/rotations, strains etc.
   – “Mass Matrix…” → specify the type of mass matrix used for a dynamic analysis
   – “Rayleigh Damping…” → specify Rayleigh damping coefficients
   – “Modal Damping…” → specify modal damping factors for a mode superposition analysis
   – “Default Temperature Settings…” → settings related to initial model temperature, prescribed temperature in time function, temperature gradient in regard to time function etc.
7. “Solution Process…” → specify various settings related to the calculations/ solution of the model
8. “Analysis Switch…” → allow to specify time, at which the current analysis will be switched into another one (i.e. from static to dynamic)
9. “Solution Diagnostics…” → specify the amount of information related to solution and contact between elements stored in the output file
10. “Solution Monitor…” → allows to define a variable/s being tracked during the solution
11. “Miscellaneous Options…” → other options not listed above
12. “Printout (.out)…”
    – “Volume…” → control of parameters stored in the output file, while working on a solution
    – “Result at Nodes…” → nodes selection, for which the information defined in “Volume…” will be stored in the output file
    – “Time Steps…” → the settings of time steps for which results are to be saved in the output file

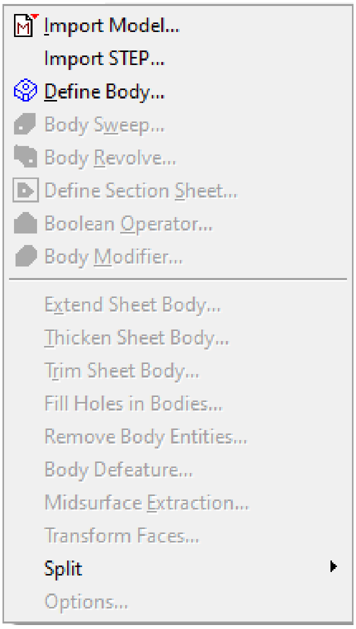
13. "Porthole (.por)…"
    - "Volume…" → control the options and data to be saved in results file
    - "Equilibrium Iteration Results…" → specifies options/parameters to be saved at equilibrium iterations
    - "Select Element Results…" → specifies options/parameters to be saved for 3D solid elements in the results file
    - "Result at Nodes…" → specifies options/parameters to be saved for selected nodes in the results file
    - "Time Steps (Nodal Results)…" → an option for selecting time steps for which nodal results will be stored in the resultant file
    - "Time Steps (Element Results)…" → an option for selecting time steps for which element results will be stored in the resultant file
14. "Nastran OP2"
    - "Output Options…" → options regarding data stored in the OP2 file
    - "Select Result Output…" → specifies parameters to be saved in the OP2 file
    - "Time Steps (Nodal Results)…" → an option for selecting time steps for which nodal results will be stored in the OP2 file
    - "Time Steps (Element Results)…" → an option for selecting time steps for which element results will be stored in the OP2 file
15. "Energy View (.egy)…" → specifies displaying the output energy for whole model/whole model and all element groups/whole model and selected groups
16. "Restart (.res) Options…" → options regarding the file for restarting calculations
17. "Mapping (.map)…" → options for creating and formatting a mapping file
18. "Miscellaneous File I/O" → specify an auxiliary file storage as well as input and output control
19. Graph Variables Output… → options which allow to specify which variables will be saved during solution for plotting with the graph viewer

The "Geometry" window



1. "Coordinate Systems…" → definition of a local coordinate systems
2. "Points…" → specify points to create a model
3. "Lines"
   - "Define…" → define a new model line
   - "Split…" → split the line into two separate lines with lengths specified by a non-dimensional length factor
   - "Extend…" → extends the existing line with specified options
   - "Thickness…" → line thickness
4. "Surfaces"
   - "Define…" → define a new model surface
   - "Thickness…" → define the thickness of the input surface
   - "Check Orientation" → checks the inconsistency in the surfaces connection
   - "Delete All…" → deletes all unused surfaces from the model
5. "Volumes"
   - "Define…" → define a new 3D figure in the model
   - "Delete All…" → delete all unused 3D figures from the model
6. "Faces"
   - "Thickness…" → define surface thickness
   - "Face Link…" → create face links in order to generate consistent meshing of finite elements
7. "Delete Geometry…" → window specifying delete of points, lines, surfaces, volumes etc.
8. "Measure…" → an option for measuring the distance, angles, etc.
9. "Transformations…" → an option for geometric transformations of the model (translation, rotation, scale etc.)
10. "Spatial Functions"
    - "Line…" → define a linear variable function in space for a spline
    - "Surface…" → define a variable function describing the shape of a surface in space
    - "Volume…" → define a variable function describing the shape of a 3D figure in space
11. "Tolerances…" → options regarding numerical tolerances
12. "Domains…" → create domains for controlling coincidence checking

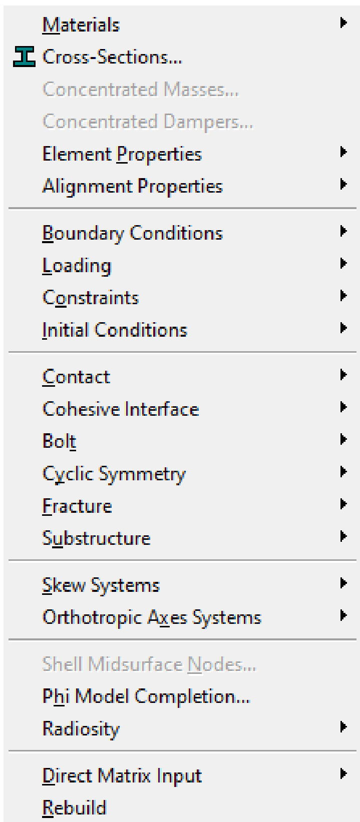
The “ADINA-M” window



1. “Import Model…” → imports CAD file with appropriate .x_t or .x_b extension containing a 1/2/3D model
2. “Import Step…” → imports CAD file with appropriate .stp extension containing surfaces/bodies
3. “Define body…” → define a parameterized 3D figure, the so-called ‘body’
4. “Body Sweep…” → create a 3D figure by sweeping the surface
5. “Body Revolve…” → create a 3D figure by revolving the surface around specified vector in space
6. “Define Section Sheet…” → create a segment of a plate/layer
7. “Boolean Operator…” → logical operations of a sum, difference, product, etc. for 3D figures
8. “Body Modifier…” → modify the features of a 3D figure
9. “Extend Sheet Body…” → extends sheet of a body
10. “Thicken Sheet Body…” → makes sheet of a body thicker of slimmer
11. “Trim Sheet Body…” → trims sheet of a body
12. “Fill Holes in Bodies…” → Allow to create holes in sheet bodies
13. “Remove Body Entities…” → removes points/edges from a body
14. “Body Defeature…” → delete small features from a 3D body on the basis of the adopted criteria

15. “Midsurface Extraction…” → options for extracting a layer/plate from a thin-walled 3D figure
16. “Transform Faces…” → allow to use predefined by user transformations on body faces (move face through translation, rotate face through rotation etc.)
17. “Split”
    – “Edge…” → split the edge of a 3D body into segments
    – “Face…” → split the surface of a 3D body into two separate surfaces on the basis of two points included in this surface
18. “Options…” → options for checking the errors of 3D figures

The “Model” window



1. “Materials”
   – “Manage Materials…” → manage all the available materials
   – “List Material Info…” → a list of defined materials in the model
   – “Elastic” → (elastic materials)
     a) “Isotropic…” → define an isotropic material

b) "Orthotropic…" → define an orthotropic material
c) "Nonlinear…" → define a material with nonlinear characteristics

– "Plastic" → (plastic materials)
  a) "Bilinear…" → define a bilinear material
  b) "Multilinear…" → define a multilinear material
  c) "Mroz Bilinear…" → define a bilinear Mroz material
  d) "Orthotropic…" → define an orthotropic material
  e) "Gurson…" → define a Gurson material
  f) "Cyclic…" → define a cyclic material
  g) "Cast Iron…" → define a cast iron material
– "Thermo" → (materials with thermal characteristics)
  a) "Isotropic…" → define an isotropic material
  b) "Orthotropic…" → define an orthotropic material
  c) "Bilinear Plastic…" → define a bilinear plastic material
  d) "Plastic Cyclic…" → define a plastic-cyclic material
  e) "Cast Iron…" → define a bilinear plastic material
– "Creep" → (materials with creep characteristics)
  a) "Elastic…" → define an elastic material
  b) "Irradiation…" → define an irradiated material
  c) "Thermo-Bilinear-Plastic…" → define a bilinear plastic material with thermal characteristics
  d) "Thermo-Multilinear-Plastic…" → define a multilinear plastic material with thermal characteristics
– "Creep Variable" → (materials with variable creep characteristics)
  a) "Elastic…" → define an elastic material
  b) "Thermo-Bilinear-Plastic…" → define a bilinear plastic material with thermal characteristics
  c) "Thermo-Multilinear-Plastic…" → define a multilinear plastic material with thermal characteristics
– "Rubber/Foam" → (materials made of rubber/foam)
  a) "Ogden…" → define an Ogden material
  b) "Mooney-Rivlin…" → define a Mooney-Rivlin material
  c) "Sussman-Bathe…" → define a Sussman-Bathe material
  d) "Arruda-Boyce…" → define an Arruda-Boyce material
  e) "Eight Chain…" → define a eight chain rubber material
  f) "Hyperfoam…" → define a hyperfoam material
– "Geotechnical" → (geotechnical materials)
  a) "Cam-Clay…" → define a Cam-Clay-type material model
  b) "Drucker-Prager…" → define a Drucker-Prager material
  c) "Curve Description…" → define a material by means of a described curve
  d) "Mohr-Coulomb…" → define a Mohr-Coulomb material

- "Others" → (the remaining materials)
  a) "Potential-based Fluid…" → define a potential-based fluid
  b) "Concrete…" → define a concrete-type material
  c) "Gasket…" → define a sealing material
  d) "Viscoelastic…" → define a viscoelastic material
  e) "Anand…" → define an Anand material
  f) "Shape Memory Alloy…" → define a shape memory alloy
  g) "Piezoelectric…" → define a piezoelectric material
  h) "Three-Network…" → define a three-network alloy
  i) "User Coded…" → define a material created by the user
- "Stress-Strain Curve…" → a material defined by the stress-strain curve
- "Stress-Strain Input Options…" → options regarding stress-strain material interpretation etc.
- "Porous Media Property…" → define the property of a porous material
- "Spring Property" → define a spring with specific properties
- "6-DOF Spring Property" → define a spring with specific properties, with 6 degrees of freedom
- "Beam Rigidity" → define beam rigidity

2. "Cross-Sections…" → define cross-sections properties
3. "Concentrated Masses…" → allow to assign concentrated masses on model entities or nodes
4. "Concentrated Dampers…" → allow to assign concentrated dampers on model entities or nodes
5. "Element properties" → (the characteristics of elements)
   - "Truss…" → define elements with a truss structure for lines or edges
   - "Beam…" → define a beam for lines or edges
   - "Isobeam…" → define an isobeam for lines or edges
   - "Pipe…" → define a pipe for lines or edges
   - "Plate…" → define a plate/layer for a surface
   - "Shell…" → define a shell for a surface
   - "2D Solid…" → define a layer element (constituting a cross-section of a 3D element) for a surface
   - "3D Solid…" → define a solid figure element for a 3D figure or 'body'
   - "2D Potential-based Fluid…" → define a potential-based fluid for a surface
   - "3D Potential-based Fluid…" → define a potential-based fluid for a 3D figure or 'body'
   - "1D General Element" → define characteristics for line- or edge-type elements
   - "2D General Element" → define characteristics for surface-type elements
   - "3D General Element" → define characteristics for 3D figure- or 'body'-type elements

6. “Alignment Properties” → (the characteristics of elements)
   - “Translation Alignment…” → defines stiffness for translational elements alignment and prescribed translational alignment as a function of solution time
   - “Distance Alignment…” → defines stiffness for distance elements alignment and prescribed distance alignment as a function of solution time
   - “Rotation Alignment…” → defines stiffness for elements rotation alignment and prescribed rotation alignment as a function of solution time
   - “Triad Set…” → defines the initial orientation of triadsets, which are used by elements alignment
   - “Assign Triad Set…” → Assigns triad set to nodes of alignment elements
7. “Boundary Conditions” → (boundary conditions)
   - “Define Fixity…” → creates a new boundary condition in the model
   - “Apply Fixity…” → assigns a boundary condition to an existing model entity
   - “Apply Fixity On Nodes…” → assigns a boundary condition to finite elements nodes
   - “Define End Release…” → define a release condition for the beam end
   - “FSI Boundary…” → defines a boundary conditions for fluid – structure interactions
   - “Potential Interface…” → define an interface between potential-based fluid elements and structural elements
   - “Surface Tension…” → define boundary conditions for surface tensions
8. “Loading” → (loads)
   - “Apply…” → allow to add load to model entities
   - “Apply On Nodes/Elements…” → add a load to existing nodes or elements
   - “Load Options…” → specify the treatment of multiple loads added to nodes
   - “Load Case…” → define a load case for a linear static analysis
   - “Load Combination…” → define a load combination
9. “Constraints” → (constraints/limitations)
   - “Constraint Equations…” → define constraints for geometry entities
   - “Constraint Equations (Multiple Slaves)…” → define constraints for multiple geometric entities, where one entity is a master entity and multiple entities are treated as slaves
   - “Constraint Equations (Nodes)…” → define constraints between nodes
   - “General Constraints (Nodes)…” → define generalized constraints for nodes
   - “Glue Mesh…” → allow to glue two dissimilar meshes together
   - “Glue Mesh Control…” → various options for controlling the glue mesh
   - “RBE3 Constraint…” → define Nastran RBE3 contraint
   - “Rigid Link Default Settings…” → settings regarding rigid links
   - “Rigid Links…” → define rigid links in the model
   - “Rigid Links (Nodes)…” → define rigid links between nodes

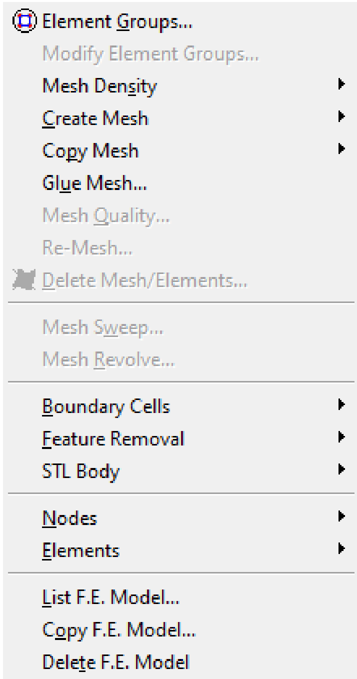
- "Ovalization Constraints…" → define ovalization constraints for pipe elements (at points)
- "Ovalization Constraints (Nodes)…" → define ovalization constraints for pipe elements (at nodes)

10. "Initial Conditions" → (initial conditions)
    - "Define…" → create a new initial condition
    - "Apply…" → apply the initial condition to the model entity
    - "Apply On Nodes…" → apply the initial condition on specified nodes of the model
    - "Rotational Velocity…" → creates and assigns the rotational velocity in the model
    - "Imperfection…" → define geometrical imperfections
    - "Geological Strain Field…" → define a ground strain field for 2D and 3D solid elements
11. "Contact" → (contacts)
    - "Contact Version…" → specifies contact version used in contact modelling
    - "Contact Control…" → specifies options regarding behavior of the algorithm used in contact modelling
    - "Contact Group…" → create groups elements in contact
    - "Contact Surface…" → define a contact surface
    - "Contact Pair…" → define pairs of contacting elements
    - "Contact Search…" → automatically create a contact surface (3D) within a specified search distance
    - "Mesh Rigid Contact Surface…" → mesh surfaces including rigid contact between them
    - "Delete Mesh…" → deletes the rigid contact surface mesh
    - "Contact Surface Offset…" → define the offset for individual contact surfaces
    - "Variable Coulomb Friction…" → define friction between contact parts using Coulomb friction coefficient
    - "Contact Body…" → define the contact between 'bodies' 3D figures
    - "Contact Point…" → define a contact between points
    - "Drawbead…" → define drawbeads for a metal forming analysis
    - "Analytical Rigid Target…" → define parameters for an analytical rigid target contact analysis
    - "Contact Surface (Element Set)…" → create a contact surface from a set of elements
    - "Contact Surface (Node)…" → define a contact surface using nodes
    - "Contact Surface (Face Node)…" → define a contact surface by means of nodes related to a Face-type surface
    - "Result At Nodes…" → define nodes for which results are saved in the output file

12. “Cohesive Interface” → (cohesive interface)
    - “Define…” → define cohesive interfaces, splits the mesh during datafile generation or when rebuilding the model
    - “Properties…” → specifies various properties for cohesive interface
    - “Split Interface…” → define interfaces where mesh will be split
13. “Bolt” → (bolt)
    - “Bolt Options…” → various options for bolt elements
    - “Bolt Sequence…” → define loads sequence applied on a bolt
    - “3D Bolt Plane…” → define a 3D plane for the direction of a bolt and its splitting
14. “Cyclic Symmetry” → (cyclic symmetry)
    - “Control…” → options related to cyclic symmetry
    - “Boundaries…” → define cyclic symmetry boundary conditions
    - “Loads…” → specifies on which cyclic part subsequent loads act on
15. “Fracture” → (a crack/fracture)
    - “Fracture Control…” → specifies various options for fracture analysis
    - “3D SVS Crack…” → define 3-D crack with SVS method used for calculation of stress intensity factors in 3-D analysis
    - “3D Crack Front (ADINA-M)” → (options related to 3D cracks)
      a) “Define…” → define a crack in the model
      b) “Subdivide…” → specify the finite element mesh density for a crack
      c) “Mesh Crack…” → create a mapping mesh for a crack
      d) “Mesh Body…” → create a free mesh for a crack ‘body’
      e) “Quarter Point…” → controls the placement of mid-side nodes in elements adjacent to the crack-tip
      f) “Virtual Shift…” → define a virtual shift for a crack front
    - “Virtual Crack Extension…” → options related to virtual crack extension in fracture analysis
    - “Line Contours…” → define J-integral line contours
    - “2D Crack Propagation…” → crack propagation control data used in 2D analysis
    - “Crack Generator…” → a tool enabling the generation of an initial crack point and/or a crack pathway
    - “Singular Elements…” → define singularities for fracture analysis
16. “Substructure” → (substructure)
    - “Define/Set…” → define a new substructure in current model
    - “Reuse…” → reuse a previously defined substructure
    - “From Element Group…” → create a new substructure from an existing element group

17. “Skew Systems” → (skew coordinate systems)
    – “Define…” → define a skew coordinate system
    – “Apply…” → apply a skew coordinate system for selected geometric elements
18. “Orthotropic Axes System” → (a local coordinate system of the elements of orthotropic materials)
    – “Define…” → define axes of a local coordinate system for an orthotropic material
    – “Assign (Material)…” → assign axes of a local coordinate system to an orthotropic material
    – “Assign (Initial Strain)…” → assign initial strain orthotropic axes system to geometry or elements
    – “Specify To Elements (Material)…” → assign local orthotropic material coordinate system to specified finite elements
    – “Specify To Elements (Initial Strain)…” → assign local initial strain coordinate system to specified finite elements
19. “Shell Midsurface Nodes…” → specify 5 or 6 degrees of freedom for shell midsurface nodes
20. “Phi Model Completion…” → a set of options completing the definition of characteristics for a potential-based fluid
21. “Radiosity” → (direct matrix input)
    – “Radiosity Group…” → define radiosity groups
    – “Radiosity Surface…” → define radiosity surfaces on model geometry
    – “Meshing…” → creates mesh of radiosity surface in model geometry with selected properties
    – “Delete Mesh…” → deletes mesh on a radiosity surface
    – “Radiosity Surface (Element Set)…” → define radiosity surface using element edge or face sets
    – “Radiosity Surface (Node)…” → define radiosity surface using nodes
22. “Direct Matrix Input” → (direct matrix input)
    – “Define…” → define direct input data included in matrices based on nodal degrees of freedom
    – “Apply…” → apply definitions of direct input data included in matrices
23. “Rebuild” → rebuilds an existing model

The "Meshing" window



1. "Element Groups…" → define a specific type of finite elements used with various options
2. "Modify Element Groups…" → modify properties of existing element groups in the model
3. "Mesh Density" → (mesh density)
   - "Point…" → specify the size of division under finite elements at geometry points
   - "Line…" → specify the size of division under finite elements for geometry lines
   - "Surface…" → specify the size of division under finite elements for surface objects
   - "Volume…" → specify the size of division under finite elements for 3D geometry objects
   - "Edge…" → specify the size of division under finite elements for edges of a 'body/ies'
   - "Face…" → specify the size of division under finite elements for faces (surfaces) of 'body/ies' objects
   - "Body…" → specify the size of division under finite elements for 3D 'bodies'

- "Complete model…" → specify the size of elements into which a complete model will be divided
- "Default Mesh Size…" → specify the default size of elements created in a model for new geometry
- "Point Size…" → specify the size of finite elements at geometry points providing various options
- "Mesh Size Function…" → specify a function for an element size located at X/Y/Z position
- "At Locations…" → define the size of a mesh elements for specific places in the coordinate system for association with surfaces and 'body' 3D figures

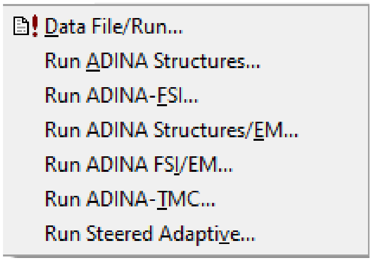
4. "Create Mesh" → (create a mesh)
   - "Point…" → generate nodes at geometry points
   - "Line…" → generate 1D elements on a line
   - "Surface…" → generate 2D finite elements on a surface
   - "Volume…" → generate 3D finite elements on a 3D figure
   - "Edge…" → generate 1D finite elements on edges
   - "Face…" → generate 2D finite elements on a 'body'-type surface
   - "Body…" → generate 3D finite elements on a 'body'-type 3D
   - "Body (Simulate Only)…" → adjusts division of the edges of a 'body' 3D figure in preparation for meshing
   - "Brick Dominant…" → generate a mesh of brick elements as dominant in a geometric 3D figure
   - "Lofted Body…" → generate a mapped mesh on lofted type of 'body'
   - "Brick Dominant…" → generate a brick-dominant mesh in a solid geometry 'body'
   - "Element Spider…" → generate beam/truss elements between entities located on two different sides
   - "Shell Transition…" → generate shell transition elements between a shell and a 3D figure
   - "From Element-Face Set…" → generate mesh from element-face sets
   - "Options…" → assignment of meshing algorithm regarding ADINA version

5. "Copy Mesh" → (copy a mesh)
   - "Body…" → copy a mesh from one 'body' 3D figure to another with a similar shape
   - "Triangulation…" → copy a mesh from the surface ('face') of a 'body' to another surface ('face') by means of triangulation technique
   - "List Triangulation…" → display a list of surfaces ('faces') which have a mesh copied by means of the triangulation technique
   - "Delete Triangulation…" → delete a copied mesh created with the use of the triangulation method

6. “Glue Mesh…” → glue dissimilar meshes between surfaces/’faces’
7. “Mesh Quality…” → checks the quality of created finite elements
8. “Re-Mesh…” → once again create mesh on a selected region
9. “Delete Mesh/Elements…” → delete mesh/finite elements from the selected model entities
10. “Mesh Sweep…” → sweep 2D mesh elements to create 3D elements
11. “Mesh Revolve…” → revolve 2D mesh elements to create 3D elements by means of revolving 2D elements around specified vector
12. “Boundary Cells” → (boundary cells – connected with ADINA fluids analyses)
    - “Define…” → define boundary cells
    - “Create 3-D Mesh…” → create a 3D mesh from boundary cells
13. “Feature Removal” → (the removal function)
    - “Body Cleanup…” → delete small features of a ‘body’ 3D figure before meshing
    - “Remove Edge…” → delete selected edges in the model
    - “Remove Face…” → delete selected ‘faces’ in the model
    - “Body Restore…” → restore all deleted edges and ‘faces’ removed with above presented options
    - “Discrete BREP…” → Creates a discrete boundary representation for a given ‘body’, which provides simply a triangular surface mesh (of the body) that has the advantage of being modified with feature removal options
    - “Body Defeature…” → deletes the features on a ‘body’ based on its discrete boundary representation
14. “STL Body” → (stereographic 3D figures)
    - “Import STL…” → import a 3D model file containing a stereographic part prepared in CAD software
    - “Convert Parasolid Body…” → converts Parasolid ‘body’ into the STL body type
    - “All-Brick Meshing…” → creates brick elements on an STL ‘body’
    - “Discrete BREP…” → creates a discrete boundary representation for a given STL ‘body’
    - “Adapt Discrete BREP…” → update mesh density on the basis of discrete representation of boundary elements of a ‘body’ in a stereographic part
    - “Export STL…” → export triangulation of 2D, 3D surfaces or shell elements to a file with the .stl extension
    - “Import Cloud Data…” → import points from the point-cloud data file

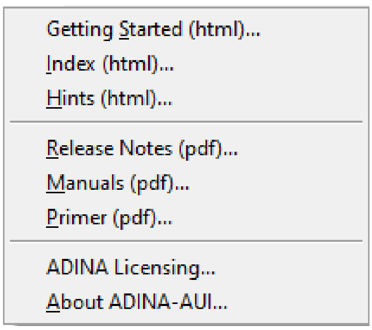
15. "Nodes" → (nodes)
    - "Define…" → define points to be meshed as nodes
    - "Node Set…" → define a set of points to be meshed as nodes
    - "Delete…" → deletes unused nodes in the model
    - "Renumber…" → renumber the nodes not associated with the geometry
    - "Snap Nodes…" → moves nodes to the most closed geometry
    - "Split mesh…" → split mesh at an interface
    - "Detach Mesh…" → detaches mesh from one part to another
    - "Join Mesh…" → joins two meshes by merging coincident nodes
16. "Elements" → (elements)
    - "Element Set…" → define a set of finite elements
    - "Element-Edge Set…" → define a set of finite elements consisting of element edges
    - "Element-Face Set…" → define a set of finite elements consisting of 'body'-type 'faces' (surfaces)
    - "Connector…" → define connector elements between model entities
    - "Truss…" → define truss-type elements across two geometry lines
    - "Axisymmetric Truss…" → define axisymmetric truss elements (ring) on specified geometry points
    - "Spring…" → define a spring on the basis of geometric points or lines
    - "6-DOF Spring…" → define a spring with six degrees of freedom on the basis of geometric points or lines
    - "Shell Thickness…" → specify shell elements thickness
    - "Shell Layer…" → define data for a complex shell layer
    - "Element Nodes…" → define connections between finite elements nodes
    - "Element Data…" → define data for individual selected element/s
    - "Renumber…" → renumber finite elements
    - "Change Group…" → allow to select elements which will be considered in different element group than preliminarily specified
    - "Transform…" → transform finite elements (copy / move elements)
    - "Convert…" → convert elements with 8 nodes (2D elements and shells) or elements with 20 nodes (3D elements) into elements with 9 or 27 nodes
    - "Settings…" → compatibility checking of CFD mesh options, checking for duplicate elements, possibility of creating unique element labels
17. "List F.E. Model…" → information in the form of a list about the current finite elements in the model
18. "Copy F.E. Model…" → copy elements/nodes from one model to another
19. "Delete F.E. Model…" → delete an entire finite elements in the model

The “Solution” window



1. “Data File/Run…” → creates an output data file, and optionally runs a solving module
2. “Run ADINA Structures…” → runs a solving module for solid models
3. “Run ADINA-FSI…” → runs a solving module for fluid based structures
4. “Run ADINA Structures/EM…” → runs a solving module for structures including electromechanical coupling
5. “Run ADINA FSI/EM…” → runs a solving module for fluid based structures including electromagnetic coupling
6. “Run ADINA TMC…” → runs a solving module for thermos-mechanical coupling problems
7. “Run Steered Adaptive…” → run a fluid structure interaction FSI/ CFD module analysis with adaptive meshing including various options concerning solution process

The “Help” window



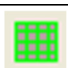
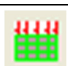
1. “Getting Started (html)…” → a “getting started with the program” help file in the HTML format.
2. “Index (html)…” → an index of typical keywords located in the help file
3. “Hints (html)…” → hints/troubleshooting list

4. “Release Notes (pdf)…” → information/news related to the currently used version of the ADINA software
5. “Manuals (pdf)…” → a set of .pdf documents containing materials for modeling/commands/verification etc. in the ADINA program
6. “ADINA Primer (pdf)…” → a vast set of examples presenting the program abilities
7. “About ADINA-AUI…” → information about the current version

CHARACTERISTICS OF MAIN MENU TOOLBAR BUTTONS USED ACROSS EXAMPLES

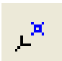
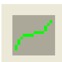
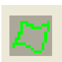
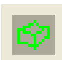
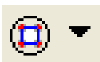
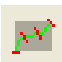
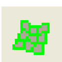
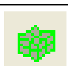
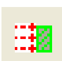
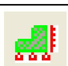
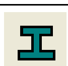
As previously mentioned all the buttons for “ADINA Structures” and their functions are described in the ADINA primer, which can be accessed from “Help/Primer (pdf)…”. In this chapter only the most frequently used buttons across presented in this book problems are shown and explained.

The “General” bar

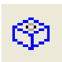
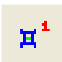
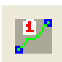
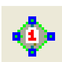
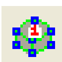
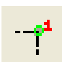
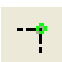
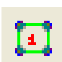
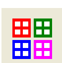
	“Open” → open a saved model file
	“Save” → save the current model to a file/overwrite a file
	“Redraw” → redraw/update the model
	“Clear” → clear the current model window (does not delete the model permanently)
	“Mesh Plot” → show the model with default settings
	“Modify Mesh Plot” → modify model display parameters
	“Load Plot” → display loads defined in the model
	“Boundary Plot” → display boundary conditions assigned in the model
	“Pick” → an option which enables picking elements of the model
	“Dynamic Resize” → dynamic resize of the model and the elements
	“Dynamic Pan” → dynamic pan of the model and the elements
	“Dynamic Rotate (XY)” → dynamic rotation of the XY plane
	“Zoom” → zoom to an indicated part of the screen
	“Unzoom All” → unzoom the whole screen until the default view

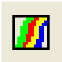
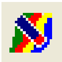
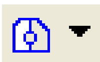
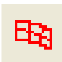
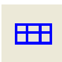
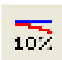
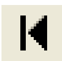
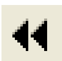
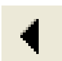
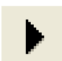
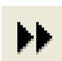
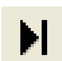
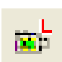
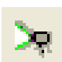
	"Unzoom Partially" → gradually unzoom the screen content until the default value
	"Refit" → exit the magnified selected mesh to a view of the entire model of finite elements
	"Measure" → an option enabling measurement of the distance between two points

The "Modeling" bar

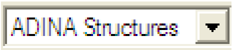
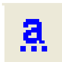
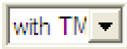
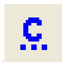
	"Define Points" → displays a window for defining points in space
	"Define Lines" → allows for defining lines by means of previously created points
	"Define Surfaces" → define a surface
	"Define Volumes" → define volumetric elements
	"Define Element Groups" → define a group with specified type of finite elements and assigns material to them as well as gives the possibility to modify additional options regarding finite elements
	"Mesh Lines" → generate finite elements on a line
	"Mesh Surface" → generate finite elements on a surface
	"Mesh Volume" → generate finite elements on a 3D volumetric figure
	"Apply Load" → define/assign loads
	"Apply Fixity" → define/assign boundary conditions
	"Manage Materials" → manage material models and its properties
	"Cross-Section" → define a cross-section
	"Data File/Solution" → create an output file/start calculations

The “Display” bar

	“Wire Frame”→ show the model along with hidden lines
	“Hidden Surfaces Removed”→ show the model with hidden surfaces removed
	“Shading”→ shading of the model
	“Cull Front Faces”→ do not show front faces in the model
	“ISO View” → view the model in accordance with the selected axes
	“Show Geometry” → activate/deactivate displaying the geometry of the model
	“Point Labels” → activate/deactivate displaying the numbers of points
	“Line/Edge Labels” → activate/deactivate displaying the numbers of lines and edges of the model
	“Surface/Face Labels” → activate/deactivate displaying surface numbers of the model
	“Volume Body Labels” → activate/deactivate displaying the numbers of 3D figures and ‘body’-type volumetric elements
	“Node Labels” → activate/deactivate displaying the numbers of nodes of a finite element mesh
	“Node Symbols” → activate/deactivate displaying the nodes of a finite element mesh
	“Element Labels” → activate/deactivate displaying the numbers of elements of a finite element mesh
	“Color Element Groups” → activate/deactivate displaying differentiation of groups of elements used in the model by means of colors

	“Create Band Plot” → create a plot of model characteristics
	“Modify Band Plot” → modify the created plot of characteristics
	“Clear Band Plot” → delete the plot of model characteristics
	“Smooth Plots” → smooth the plot of model characteristics
	“Group outline” → options for displaying finite elements and the model itself
	“Show Deformed Mesh” → show a deformed finite element mesh depending on the time step
	“Original Mesh” → show the original state of the finite element mesh
	“Scale Displacement” → show a rescaled (for better visibility of the strains) deformed finite element mesh of the model
	“First Solution” → go to the solution from the first time step
	“Fast Rewind” → fast rewind the time steps of the solution
	“Previous Solution” → rewind the solution by 1 time step
	“Next Solution” → fast forward the solution by 1 time step
	“Fast Forward” → fast forward the time steps of the solution
	“Last Solution” → go to the solution from the last time step
	“Movie Load Step” → create a stop motion movie of the model’s response to the input loads
	“Animate” → display the declared movie in the form of an animation
	“Refresh” → delete the stop motion movie from the cache memory
	“Save Avi” → save the stop motion movie from the cache memory on a drive in the form of an *.avi file

The “Module Bar”

	“Program Module” → choose an active program module
	“Analysis Type” → choose the type of the performed analysis of the model
	“Analysis Options” → options related to the performed analysis
	“Multiphysics Coupling” → define an additional performed analysis (models consisting of two analyses)
	“Coupling Options” → options of a coupling analysis

Examples regarding building statics and dynamics

• Statics

○ Two-dimensional problems

EXAMPLE 1
A SIMPLY SUPPORTED BEAM. DETERMINING THE MAXIMUM BENDING MOMENT AND DISPLACEMENT

The example presents the modeling of a beam subjected to a linear constant load. The consecutive steps present the manner of specifying the basic functions necessary to model a system and to perform an analysis. A static scheme of the analyzed beam is presented in Figure 19.

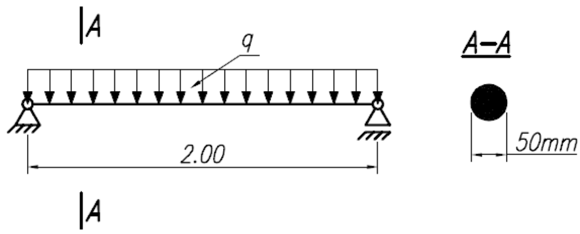


Fig. 19. Scheme of a simply supported beam

The span length of the beam between supports, diameter and value of constant linear load are:

$L = 2.00$ m
$D = 0.05$ m
$q = 5.00$ kN/m

A beam with a circular cross-section has been modeled from steel, assuming the following material constants:

$E = 210$ GPa $= 2.1\mathrm{e}^{11}$ Pa
$\upsilon = 0.30$
$\rho = 7859$ kg/m^3

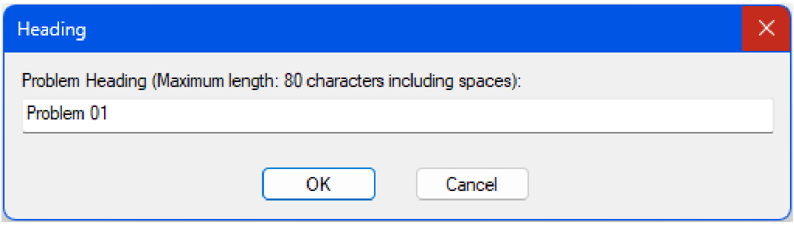
<u>Note</u>: It is recommended for models prepared in two-dimensional (2D) environment to be created in the first quarter of the coordinate system, preferably in the YZ plane. This is due to the fact that the previous versions of the ADINA software experienced problems with meshing when 2D elements were located in a different quarter and in a different plane than YZ. Currently it is possible to set the plane for the analysis of 2D solid elements in “Control→Miscellaneous Options…”. It is also recommended for all the values in the program to be entered in accordance with the basic units of the SI system – this will make the interpretation of results much easier.

STEP 1. Definition of the type of analysis

Upon opening the ADINA software, choose “ADINA Structures” from the “Module Bar” in the “Program Module” section, and choose “Statics” from the drop-down list next to “Analysis Type”.

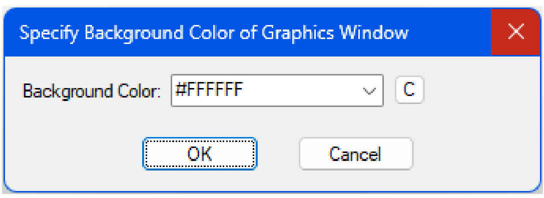
STEP 2. Entering the heading of the model

Choose “Control→Heading” from the upper menu tabs, and then in the newly displayed window enter the heading and confirm it with the “OK” button.



STEP 3. Changing the background color of the model window

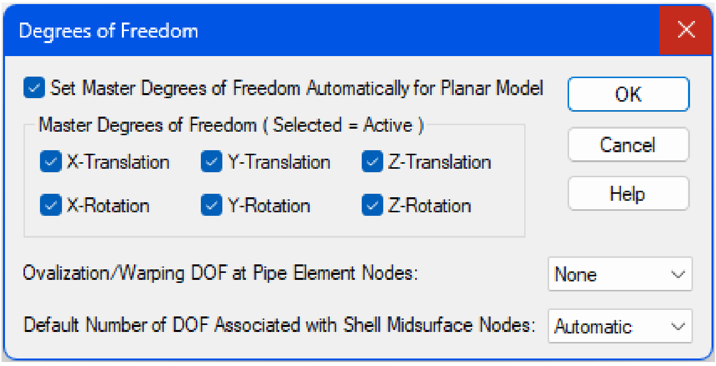
In order to change the displayed background color, change the color value in “Edit → Background Color” in the main window, choose the color desired by the user, and then confirm it with the “OK” button. As an example white color will be used.



Note: If the background color does not change, it may be connected with wrong graphics system chosen in the ADINA program. In order to change it, go to “Edit → Graphics System…” and change the “Graphics System:” option to “Windows GDI”.

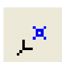
STEP 4. Definition of global boundary conditions

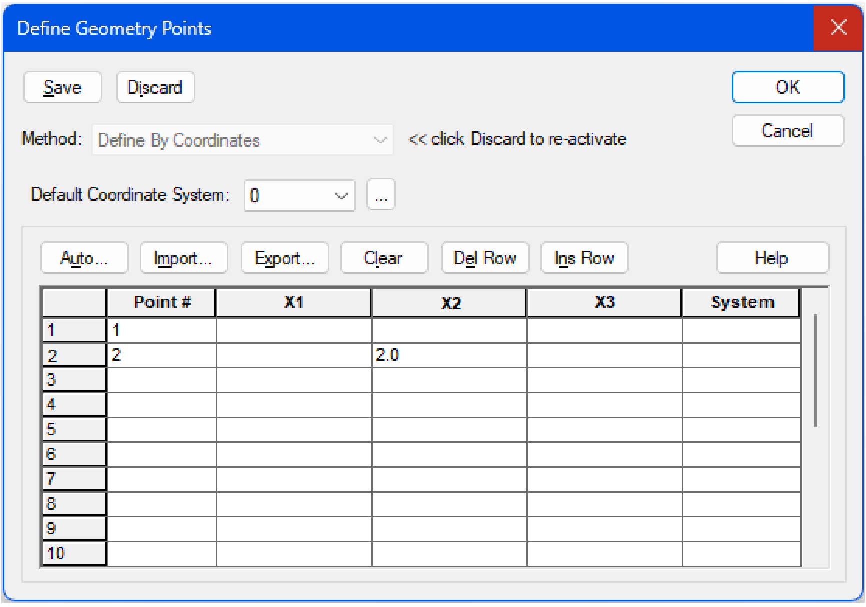
Global boundary conditions simply mean exclusion or inclusion of displacements and rotations related to the entire model. The current example deals only with the ability to move along the Y and Z axes and rotate with respect to the X axis. The remaining conditions do not occur in the present example, so they should be excluded. In order to open the window of global boundary conditions, choose "Control → Degrees of Freedom" from the upper menu tabs.



In the new versions of ADINA, one can leave the check mark enabled for the option "Set Master Degrees of Freedom Automatically for Planar Model". According to that ADINA will automatically adjust master boundary conditions in regard to the specified calculated problem. In older ADINA versions it was rather necessary to include/exclude master boundary conditions. For the older ADINA software taking into account studied problem, following boundary conditions should have check mark enabled: "Y-Translation", "Z-Translation" and "X-Rotation". Remaining boundary conditions should have empty check boxes. The "Ovalization/Warping DOF at Pipe Element Nodes:" as well as "Deafult Number of DOF Associated with Shell Midsurface Nodes:" should have been left as it is automatically adopted.

STEP 5. Definition of points

In order to define the points needed to create a model, go to the "Geometry → Points → Define…" tab, or click  in the toolbar available in the main menu of the program. To ensure backwards compatibility the model is created in the first quarter – the YZ plane. Therefore, the table opened in the new window should be filled out in accordance with the figure, in which X1 means values on the X axis, X2 – values on the Y axis, and X3 – values on the Z axis.



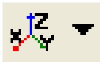
It is possible to choose the coordinate system specified by user changing the '0' value in the "Default Coordinate System", however the local coordinate system must have been introduced by the user. Moreover, the local coordinate system may be created through the "…" button in the line of "Default Coordinate System" option.

'System' in the table means the coordinate system currently used to specify the X1, X2, and X3 coordinates. Different points may be referred to different coordinate systems.

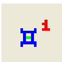
It should be noted that the numeration of points is not created automatically, and thus each new point must have a number assigned by the user in the "Point #" column. Upon clicking the "Save" button, the remaining values of the table will be automatically filled out as zeros. So far, no local coordinate system has been defined, and the global coordinate system always has a value of 0, which is why it was not required to enter any numbers in the "System" column. Currently, the window can be closed with the "OK" button. Upon clicking the "OK" button, the point definition window will disappear, and two orange points will appear in the main model window.

Although the program fills out the column with zeros automatically, for a larger number of points it is recommended for the user (especially a beginner) to enter the given zeros manually. This makes it easier to identify whether a given point with a value of 0 on one of the axes is actually supposed to have such a value, or whether it can have a different one.

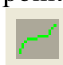
STEP 6. Model view

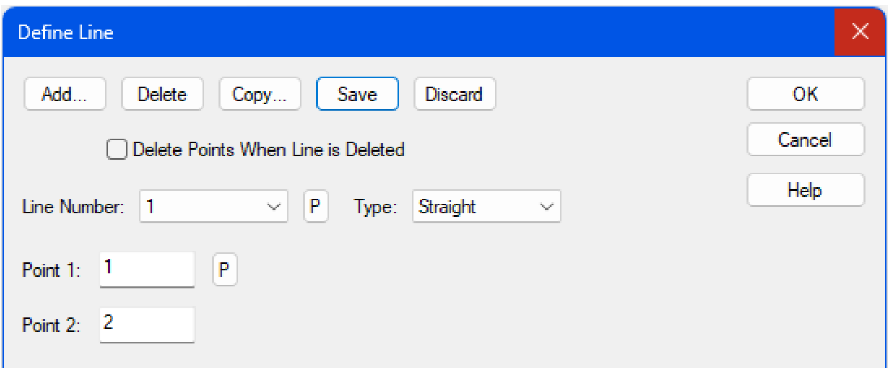
In older ADINA software program assumes certain display values automatically, which is not always satisfactory to the user, e.g., in this case, after defining two points, the program switched to the main model but with a perspective view, which makes it much harder to work with a two-dimensional model 2D; therefore, the button  should be used to choose "YZ View" in order to change the view.

STEP 7. Displaying the numbers of points

In order to display the points assigned numbers for their easier identification, click .

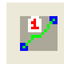
STEP 8. Definition of lines

In order to define a line between two visible points, choose "Geometry → Lines → Define…" from the upper menu tabs, or click  in the main menu of the program. In the newly opened window, click the "Add…" button for adding a new line, make sure that the line type is a straight line, meaning "Straight", and then in Point 1 and Point 2 enter numbers like in the figure below.



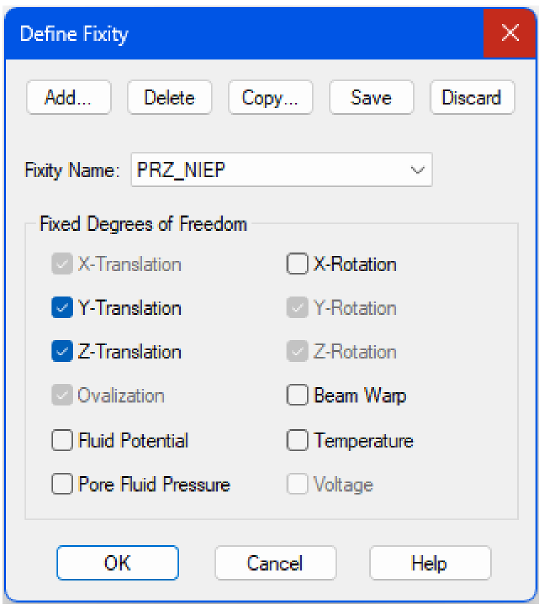
There is no relevance whether a line is made from left to right by entering 1 in Point 1: and 2 in Point 2:, or it is made from right to left in this problem. Upon entering the identification numbers of points, it is always advisable to first click the "Save" button, and then "OK" button. As it will turn out soon, such a procedure allows for adding further lines with no need to reopen the same window. An orange line connecting the two previously defined points should appear in the main model window.

STEP 9. Displaying the numbers of lines

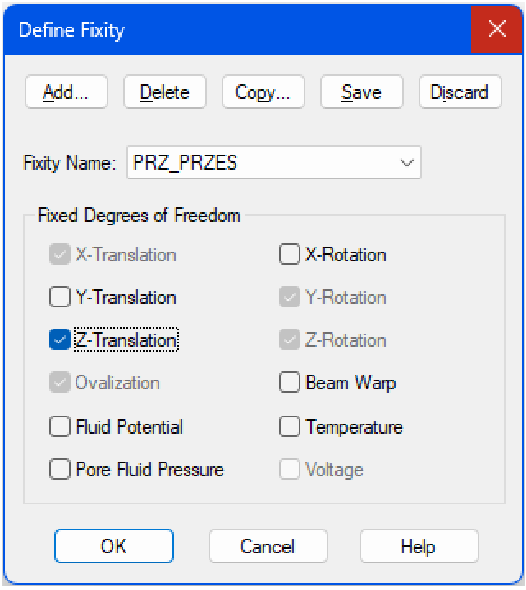
In order to display the numbers of lines for their easier identification, click .

STEP 10. Definition of boundary conditions (fixity characteristics)

In order to declare fixities in the model, choose "Model → Boundary Conditions → Define Fixity…" from the upper menu tabs, which will cause the user to go to a new window for defining boundary conditions. Two fixities are needed, i.e., immovable hinged support and the hinged support with the ability to move in the Y axis direction. In order to create an immovable support in the newly opened window, press the "Add…" button, input the name of the new fixity, and then the requested displacements and rotations will be blocked for all options with check mark enabled. The definition of a immovable hinged support fixity is presented in the figure below.



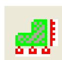
In this case, displacements along the Y and Z axes have been blocked, while rotation with respect to the X axis remains unrestricted. The remaining values are not taken into consideration in the abovementioned example. Some of the boundary conditions are gray – inactive, which was caused by the earlier setting of global boundary conditions in the model.

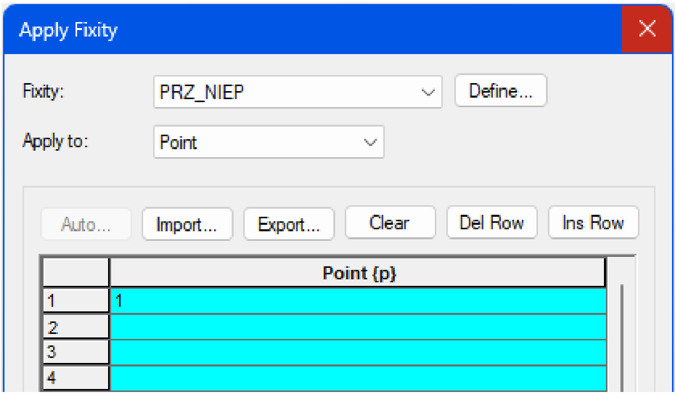


In order to define a movable hinged support, click the “Save” button, and then the “Add…” button in the current window, once again enter the name of the fixity, and then select proper translations to be blocked.

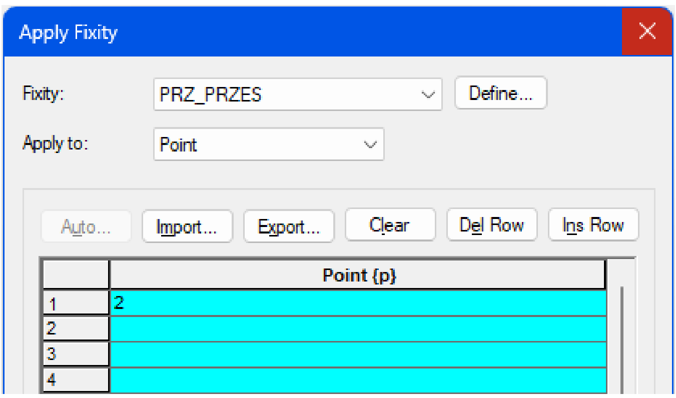
Once the abovementioned operations are completed, click the “Save” button, and then “OK”.

STEP 11. Definition of boundary conditions (applying fixities in nodes)

In order to apply created fixity to a node, choose “Model → Boundary Conditions → Apply Fixity…” from the upper menu tabs, or click  in the main program menu. In the new window, make sure that the “Apply to:” option shows “Points”, since the boundary conditions are to be applied to points. At first the immovable hinged support will be added to the one of the model points. In order to do that, in the “Fixity:” drop-down list select the name of immovable hinged support (in this case “PRZ_NIEP”) and check if in the “Apply to:” drop-down list “Point” is selected (if not, please select “Point”). Then enter the number of point manually in the “Points” column, or double-click the first row of the “Point {p}” column and choose the nodes manually. When the choice has been made, press the ESC key on the keyboard to return to the definition window. Then save the status of the boundary condition by means of the “Apply” button. Properly defined boundary condition of immovable hinged support is presented in the window below:



As a consecutive step, in the "Fixity:" drop-down list, one must choose the hinged support with the ability to move on the X axis (in this case one should chose "PRZ_PRZES"), leave the "Apply to:" option unchanged i.e. "Point" and in the first row of the table in column "Point {p}" introduce in the similar way as before the second point "2". Properly defined boundary condition of hinged support with the ability to move is presented in the window below:



Going back to the remaining options of the window for applying boundary conditions:

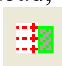
– "Define…" is used to create additional boundary conditions in the same way as by going to "Model → Boundary Conditions → Define Fixity…",
– "Apply to:" gives the possibility to choose to which model entity the boundary condition should have been applied. Once different entity is selected, the title of table reflects the chosen element, i.e. if in the "Apply to:" drop-down list one choose "Line" the title of table column will be "Line {p}". It should be noted that in order to have possibility of choosing different entities they must be created in the main model window beforehand.

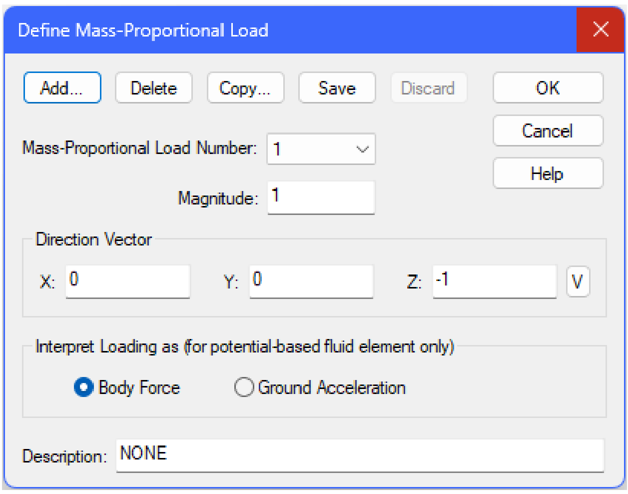
The remaining buttons in the window are:

– "Auto…" is used to input points, lines, etc. from a number to a number, with an increment number,

– “Import” and “Export” enable importing and exporting the numbers of points, lines, etc. from/to a text file *.txt.
– “Clear” enables complete clearing of the currently selected list of boundary conditions (e.g., having selected “Apply to: Points”, clicking the “Clear” button will allow for clearing a full list of boundary conditions related to points. Others such as lines etc. will remain unchanged, until the user’s intervention.).
– “Del Row” and “Ins Row” enable deleting a single row or several rows, or adding a new row in the column.

STEP 12. Definition of loads

In order to define a load, choose “Model → Loading → Apply…” from the upper menu tabs, or click  in the main menu. Then in the newly opened window define both the mass-proportional load and external load in the form of a linear continuous load. The mass-proportional load is the first one to be defined; therefore, choose “Mass Proportional” from the drop-down list in “Load Type:”, and then click the “Define…” button next to the “Load Number:” drop-down list. In order to define a new load, click the “Add…” button in the next newly opened window. The values should look like in the figure below.



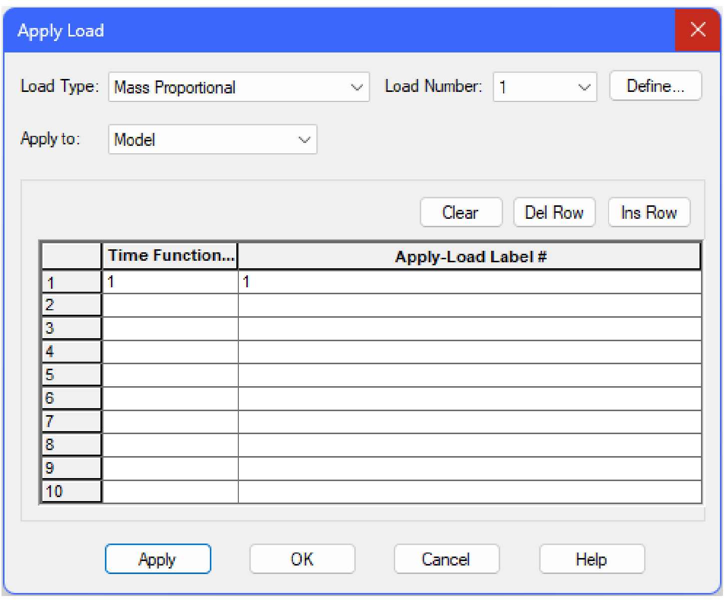
Since the options presented in the new window are satisfactory, click the “Save” button, and then “OK”. The description label may be omitted.

Going back to the mass-proportional load definition window, the following options are included:
– The “Mass-Proportional Load Number” option in the form of a drop-down list, informing us about the number of mass-proportional loads which have been defined for the entire model. Of course, there is usually one mass-proportional load, since the software calculates the bulk density of an element by itself; when

this type of load is applied, there is no need to differentiate that one mass-proportional load belongs to element A, and mass-proportional load no. 2 belongs to element B. A larger number of mass-proportional loads is usually used mainly for model tests.

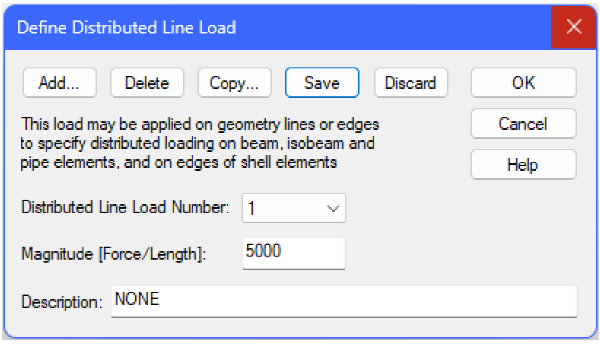
– The "Magnitude" option enables increasing or decreasing the distribution of the mass-proportional load of a model by means of values. For a value of one, the mass-proportional load is calculated as 100%, while for other values it is respectively higher or lower.
– The "Direction Vector" options indicate the direction of the applied load with respect to the global coordinate system. –1 next to "Z" indicates that the weight will act opposite to the Z axis. It is important to note that any number can be entered in this box (not only 1), but it will not affect the value of load. Only the +/– sign informs the software whether the load is acting in accordance with the direction of the axis, or against it. The load can also act obliquely to the global coordinate system with no problems; therefore, in each box it is possible to enter values of, e.g., X: 2, Y: 2, Z: 2, and the load will act in accordance with the directions of the axes, at an angle of 45° to each one of them.
– The "Interpret Loading as" options (for potential-based fluid element only) only apply to a potential-based fluid model. In any other module, they do not take part in the analysis.
– The "Description" field correspond to the 'naming' of load. One can write a name for a load, which will be furtherly visible in the model tree.

When the definition of mass-proportional load in its own window has been completed, the program returns to the window in which the load data are assigned to the model; for instance, in the case of mass-proportional load, the load definition window for the model should look like this:

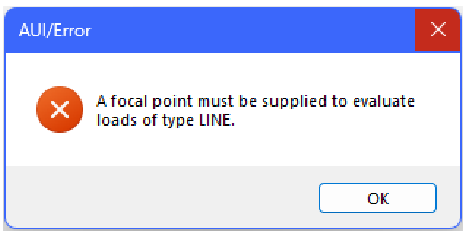


Note: Especially in older versions of ADINA it is important that each time the load is introduced, the user should enter the subsequent number in the "Label #" column, since when doing this automatically (with no input from the user), the program sometimes 'skips' the loads. When the values have been entered into the table, click the "Apply" button in order to apply the entered values, and do not leave the load definition window yet.

The next step corresponds of adding a continuous load. In order to do this, choose "Distributed Line Load" from the drop-down list next to "Load Type:", then click the "Define…" button near the drop-down list next to "Load Number:", and enter values like in the figure below.



Then click "Save" and "OK" in order to return to the main load definition window. When the digit 1 has been entered in the first row and the "Line {p}" column, and then after clicking the "Apply" button, the resulting information will be like in the figure below:



This means that the model needs an additional point, the so-called "Aux. Point", with respect to which the load will be assumed. However, we cannot use any of the currently provided points in the model.

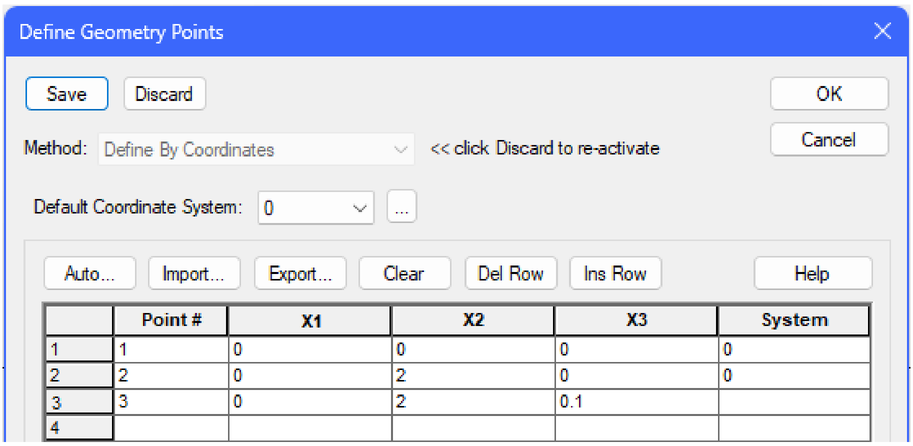
STEP 13. Definition of an additional point

In order to add another point to the model, use the same operation as in step 5 called "Definition of Points". Therefore, open the window by means of the upper

"Geometry → Points → Define…" tab, or by means of . Upon opening the new window, input point 3 in any place on the Y axis, and with any coordinate on the Z axis (however, one should think whether the load must have a direction same with the Z axis, or opposite). Since in the current example the load is opposite to the Z axis, the point should be set in the positive half of the Z axis. Then click the "Apply" button and "OK". If the data have been entered like in the figure below, point "P3" should appear in the model above the "P2" button.

Pay attention to the fact that the definition of a point specifying the direction of load may take place already in the beginning of the definition of the model. One or more of such points are defined in the model, later playing an important role for options related to the direction of load, the cross-section, etc.

<u>Note</u>: In the case of loads (if a load has been declared as positive), the assumed direction of a load extends from the auxiliary point towards the defined part of the model.



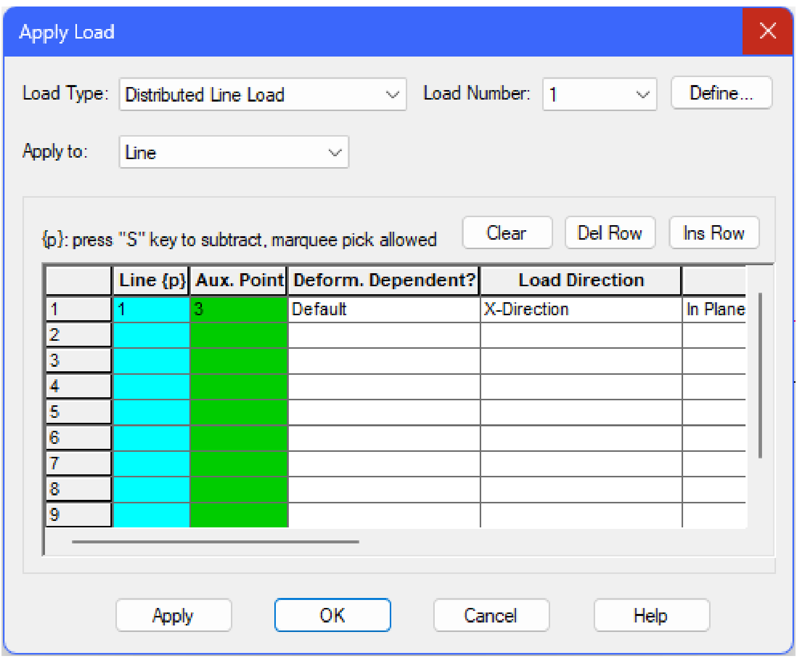
STEP 14. Definition of loads part 2

Let's go back to the "Apply Load" window once again. Options of the table for declaring a linear load are:

– "Line {p}" specifies the line to which the load is applied;
– "Aux. Point" is an auxiliary point identifying the direction of the load from the specified point towards the defined line;
– "Deform Dependent?", with "Default/Yes/No" options to choose from, specifies whether the load is calculated following deforming geometry under applied load "Yes" or the load does not change its line of action "No". The usual option used in here is "No".

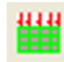
– “Load Direction” specifies the load direction. The choices are: “X-Comp. only/ Y-Comp. Only/Z-Comp. only”, meaning load acting opposite the X/Y/Z axes, respectively, “Tangential Traction” meaning a force acting tangentially to the axis of a given bar, and “Global X-Dir./Global Y-Dir./Global Z-Dir.”, where the action of a force declared as positive causes its action in accordance with the direction of the X/Y/Z axis, respectively. In most cases, it is best to leave the value as default – “Total (Normal)”.
– “Load Plane” specifies the plane in which the load works. It is best to leave the value as default – “In Plane” (when the auxiliary point has already been specified), which causes the load to be adopted in a plane between the defined point and the line of the model to which the load refers. The second one of the possible options is “Perpendicular to Plane”.
– “Time Function” refers to a time function specified by the user. In the current example, the value should be left as default.
– “Arrival Time” specifies the return of load over a time of x seconds. For a static analysis without declared time steps, like in the present example, the value should be left as default.
– “Spatial Function” – the spatial function describing the variation of a quantity along a line. Leave the value as default.
– “Label #” – a number specifying the next defined load. As described earlier, it should be controlled by the user, so that the program would not ‘skip’ the loads.

Since the model has an auxiliary point for load, all that remains is to add a linear load. Declaration of load is shown in the figure below:



As soon as all the values are introduced in the window, click the “Apply” button and then “OK”.

STEP 15. Displaying load values and boundary conditions

In order to display the defined loads in the main model window, click . In order to show boundary conditions, click .

The alternative method is to activate the view by the upper menu tabs of the program. In order to show the loads, choose “Display → Load Plot → Use Default”, while showing the fixities is done by “Display → Boundary Conditions → Default”. The user’s good eye might have noticed that the “Custom…” option for boundary conditions in the “Display” tab enables the introduction of changes in the view, while for loads such options are “Use Style…” and “Define Style…”. Since in most cases it is sufficient for the program to display the standard styles, modifications or creating new styles will not be discussed in the present example.

It often happens that the legend of the displayed parameters overlaps the model, in which case two buttons should be clicked in the main program menu:

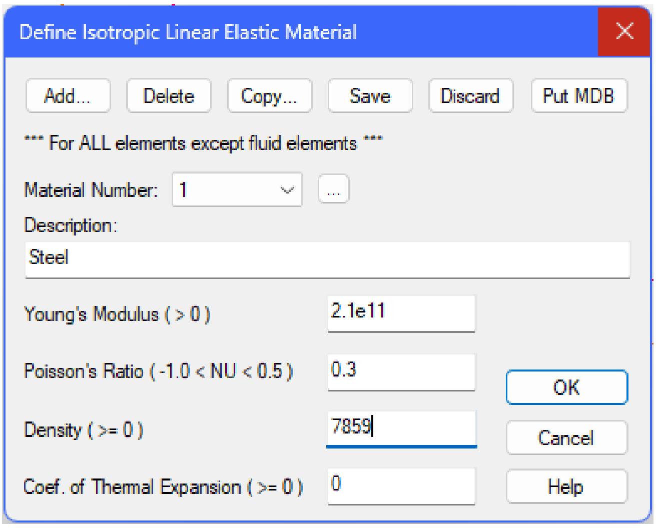
 – the “Pick” button enables the selection of elements

 – the “Dynamic Pan” button enables the panning of elements

For monitors with high screen resolutions, the descriptions of legends may be very small, however they can be enlarged by means of zooming in/out, which is done by means of the “Dynamic Resize” button: .

STEP 16. Definition of material constants

In order to define material constants, choose “Model → Materials → Manage Materials” from the upper menu tabs, or click . In the newly opened window, choose and click the “Isotropic” button in the “Elastic” group of materials. An alternative method of reaching the window with the characteristics of an isotropic material is to select “Model → Materials → Elastic → Isotropic…” in the upper tabs of the program. In the newly opened window, first click “Add…” in order to add a new material, and then enter the values of material constants like in the figure below:



The "Description" option means 'name' of a given material. The box does not need to be filled out; the program identifies the material anyway by its number presented next to "Material Number:". Unfortunately, the user must remember/write down the material number which they have defined, since later assignment of a given material to the model takes place via its number.

The "Young Modulus (> 0)" option specifies Young's modulus. It is allowed to use the format 210 000 000 000, or an engineering format as shown above. The box is obligatory.

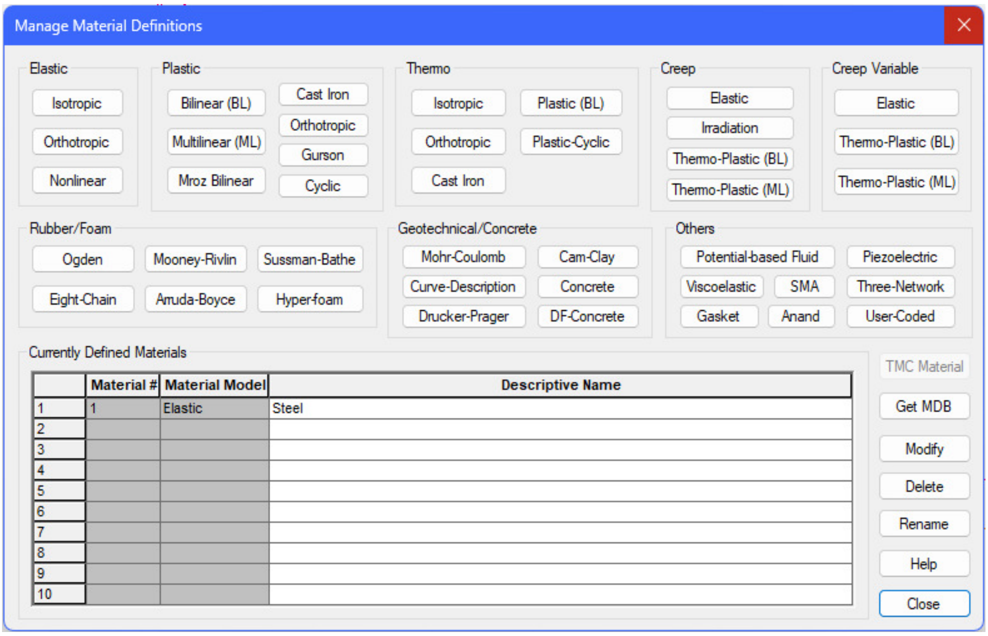
The "Poisson's Ratio (–1.0 < NU < 0.5)" option enables inputting the Poisson's ratio. The box is obligatory, although the default value is 0.0.

The "Density (>= 0)" option specifies the density of the material. The box is required especially in the case of an analysis with a mass-proportional load.

The "Coef. of Thermal Expansion (>= 0)" option is a box not required if the performed analysis is not a thermal analysis of a given material.

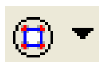
The "Put MDB" button – when the button is clicked, the declared material is added to the program database. This database is shared between all the opened ADINA AUI windows, however after closing down the program it is not stored, thus it cannot be imported in new problems.

Upon inputting all the data, click the "Save" button and "OK". When the program is in the "Manage Materials" window, this window can also be closed with the "Close" button. This window is presented in the figure below.

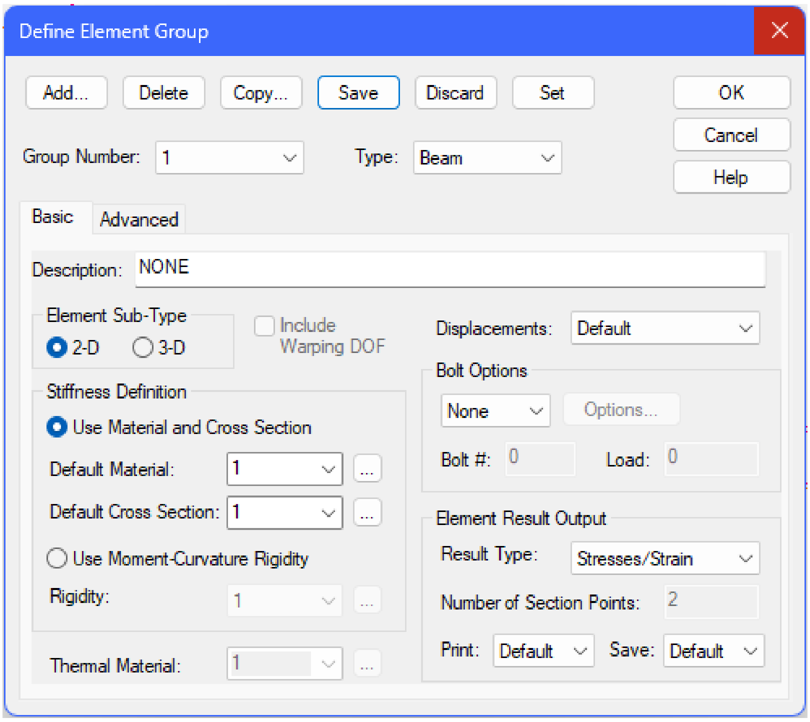


The “Manage Materials” window is more useful than the alternative method of adding materials through upper menu bars, as the user has an access to any material possible to declare.

STEP 17. Specifying the type of the analyzed construction

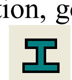
The next step is to define the element group/groups of the model. This means that certain constants, such as the type of material, the size of displacements, the group type, etc., are specified in a given group. In order to define a group, choose “Meshing → Element Groups…” from the upper tabs of the program, or choose  from the toolbars. Since a beam has been modeled in the present example, in the newly opened window click the “Add…” button in order to add one group, then choose the “Beam” option from the drop-down list next to the “Type:” description. Then in the “Description” bar in the same window it is possible to enter a title characterizing the group (however, like when defining a material, in the remaining declarations the program uses the number of the group rather than its description). The next step is to change the “Element Sub-Type” to “2-D”. In the “Stiffness Definition” group of options, make sure that the use of a defined cross-section and material “Use Material and Cross Section” is selected. Set the “Default Material” value as “1” (it is a previously defined steel material with the number 1), and set the value of the cross-section

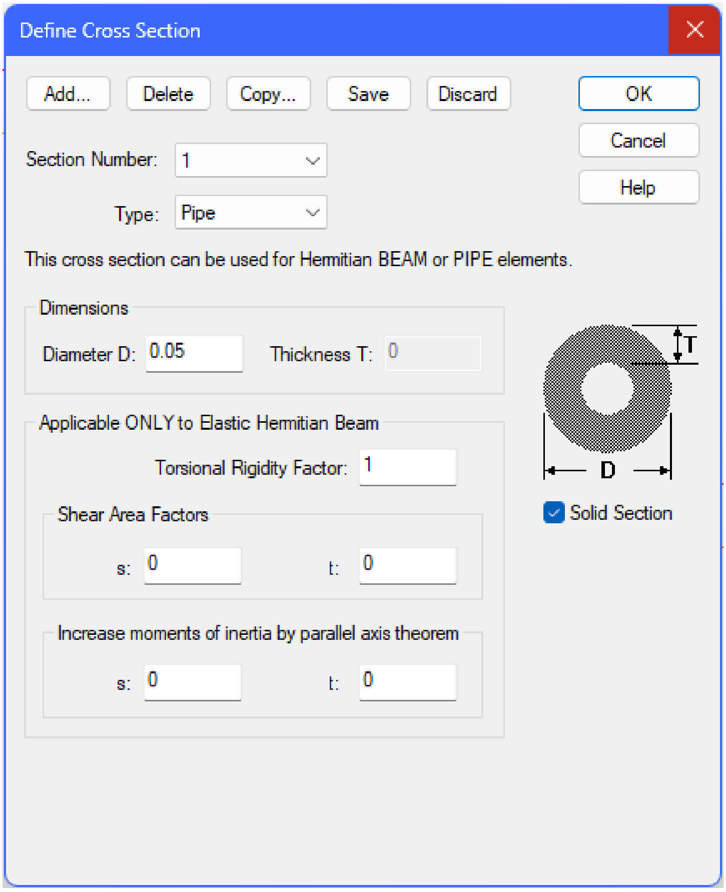
"Default Cross Section" as "1" as well. At this moment, no cross-section of the beam has been declared yet, but as soon as it is declared, it will have the number "1". The remaining values in the tabs remain unchanged. Due to the plurality of options in the window, not all functions have been described – they will be described in an ongoing manner when used in other examples. A properly filled out window is presented in the figure below:



Upon inputting all the data, click the "Save" button and "OK" in order to save the input data and leave the window.

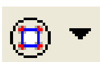
STEP 18. Definition of the cross-section of a beam

In order to define a cross-section, go to "Model → Cross-Sections…" in the upper tabs of the program, or click  in the program toolbars. In the newly opened window, click the "Add…" button. In the place where the bar type "Type:" is to be specified, choose the "Pipe" option from the drop-down list, and then input the diameter in "Diameter D:". In order to make the bar solid, select the "Solid Section" option. A properly filled out window should look as follows:



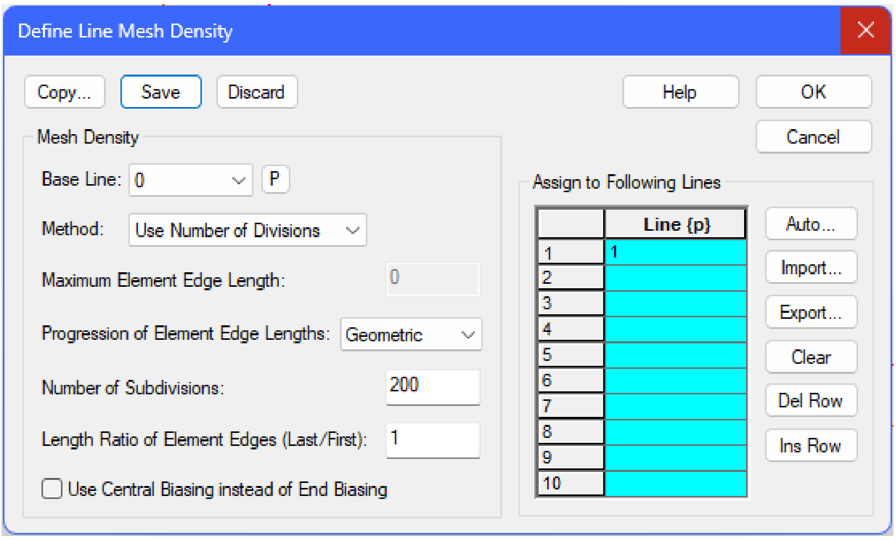
Click “Save” and leave the window through “OK” button.

STEP 19. Mesh subdivision

In order to divide an element into proper segments, choose “Meshing → Mesh Density → Line…” from the upper tabs of the program, or click the arrow next to the button  located in the toolbar, and then choose “Subdivide Lines” from the drop-down menu. The ‘number of divisions’ method will be used in the newly opened window. Therefore, in the “Mesh Density” group of options, choose the “Use Number of Divisions” option from the drop-down list in the “Method” box, then enter 200 in the “Number of Subdivisions” box (this means that each segment of the mesh has a length of 1.00 cm). Leave the remaining options unchanged.

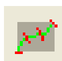
An alternative method of achieving the desired effect is to choose the “Use Length” option in the “Method:” drop-down list. Then enter a value of 0.01 in “Element Edge

Length”. Leave the remaining options unchanged. In both cases, the desired result produces segments with a length of 0.01 cm. In the table in “Line {p}” column introduce “1” in the first row, what means that only the line no. 1 is going to be divided. It is also possible to divide line, if from the “Base Line” drop-down list, the line no. 1 will be checked.



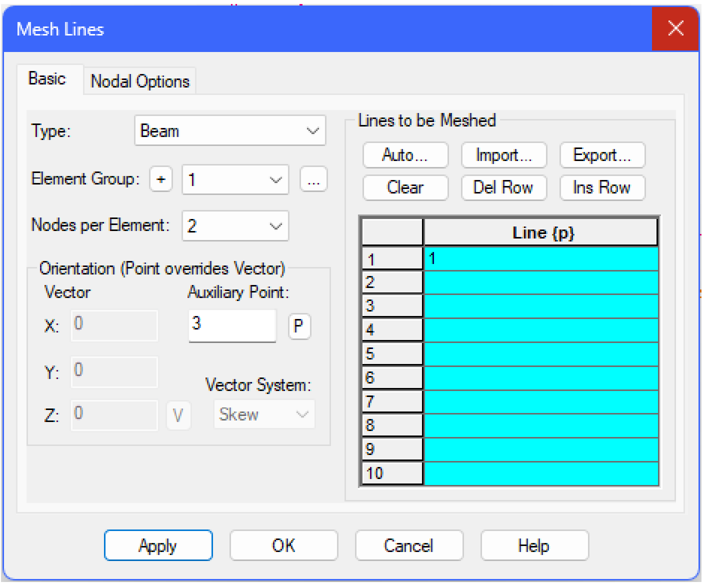
When the option has been selected, click the “Save” button and “OK”.

STEP 20. Definition of finite elements

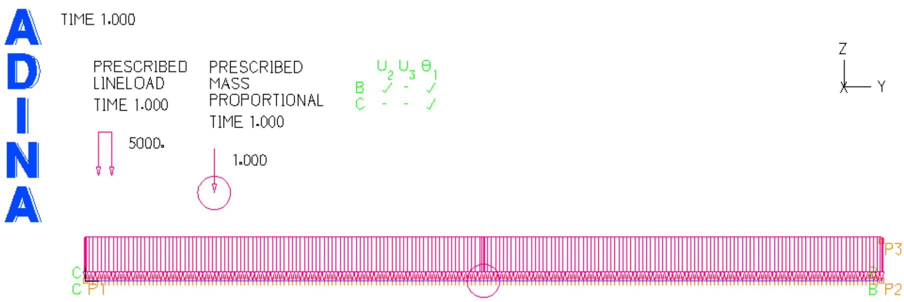
The next step is to prepare a finite elements on prepared division. In order to accomplish this, choose “Meshing → Create Mesh → Line…” from the upper tabs of the program, or achieve the same by clicking  from the toolbar.

In “Type:”, make sure that the type of the finite element mesh being defined is a beam – “Beam”. The “Element Group” option should have a value of “1”, meaning the assignment of a previously defined group of elements.

The previously modeled auxiliary point “P3” will be used in the “Orientation (Point Overrides Vector)” group of options. Therefore, in “Auxiliary Point” enter a number corresponding to auxiliary point “3”, or by clicking the “P” button choose the desired point from the model. Enter the number of the desired line “1” in first row of the table on the right side of the window, or choose a line from the model by double-clicking the table row, and after selection press ESC on the keyboard. The second tab called “Nodal Options” is left unchanged. A view of the window along with the input data is presented in the figure below:



Then click the “Apply” and “OK” buttons. The model should look as follows:



STEP 21. Starting calculations

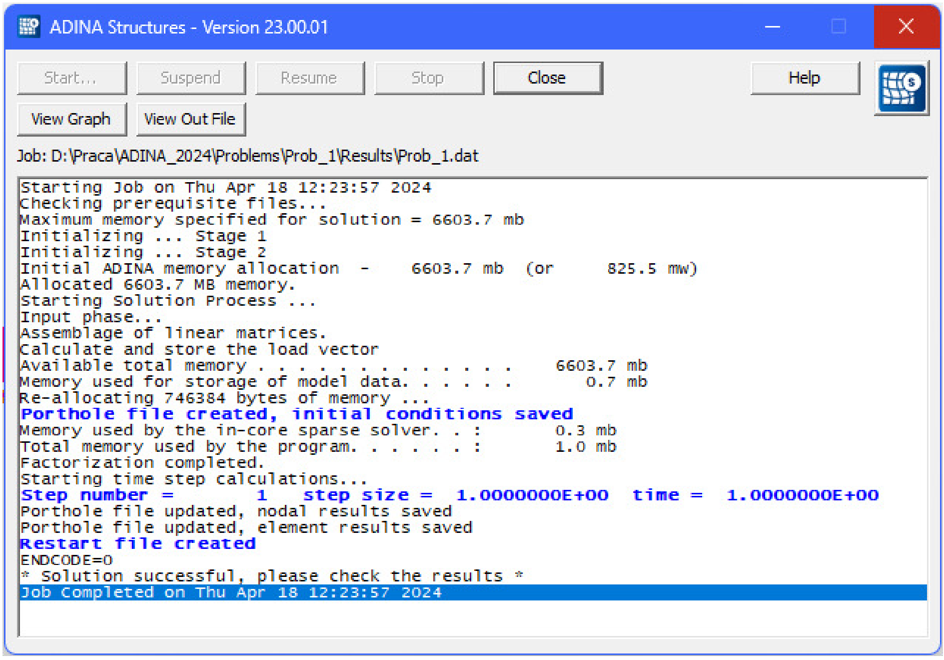
In order to start calculations, choose “Solution → Data File/Run” from the upper menu tabs, or choose  from the toolbars. Subsequently, input the path for saving the resultant file. However, before the user clicks the “Save” button, they should make sure that the “Run Solution” and “Automatic” function near the “Maximum Memory for Solution” options are selected in the window in the “Adina Structures Solution” group of options.

The “Run Solution” option enables solving a task; when unselected, it will result in only indicating a location for saving the resulting model file.

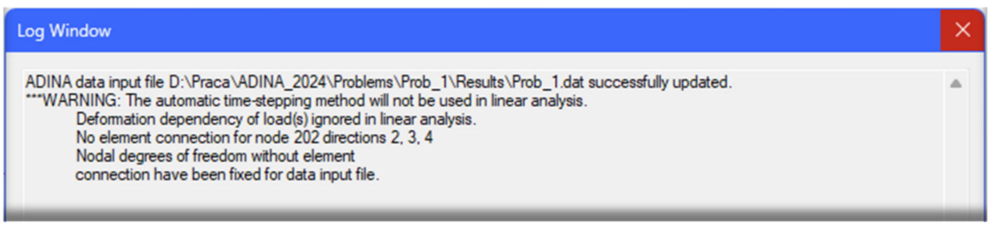
The “Automatic” option allows the program to automatically choose the necessary internal memory of the computer to solve a given problem.

The accompanying windows are presented below:

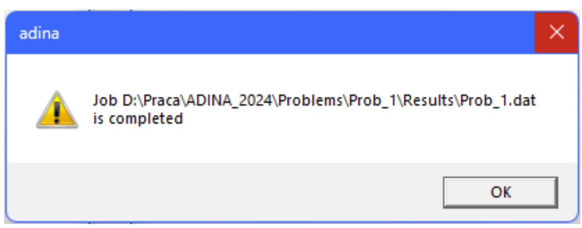
– a window showing the status of calculations, along with the used memory, etc.



– a window displaying comments and errors present in the model. When an error occurs, the program will not perform further calculations.



– a window informing whether the calculations were successful, or whether they were interrupted.



STEP 22. Transition to the post-processing module (results)

In order to go to the results module, change “ADINA Structures” to “Post-Processing” in the drop-down list of the “Module” toolbar.

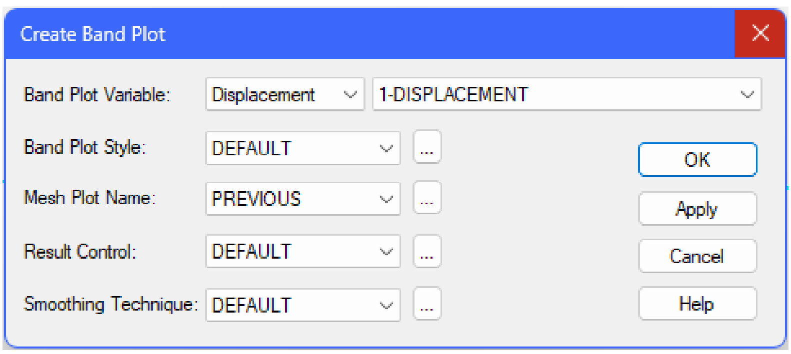
When switching between modules (as soon as the program recognizes the introduction of changes such as, e.g., recalculations of the construction, a window will be displayed, asking whether to reject the changes or save the file. The decision about saving is up to the user, although usually it is best to save the existing model, and do not go to the post-processing module before doing so).

STEP 23. Opening the resultant file

In order to open the resultant file, choose  from the toolbar, or choose “File → Open” or “File → Open porthole…” from the upper menu tabs. The “Open porthole…” option comprises several options, but they will not be discussed in this example. When the resultant file has been selected and confirmed, the model should be displayed on the main screen of the program.

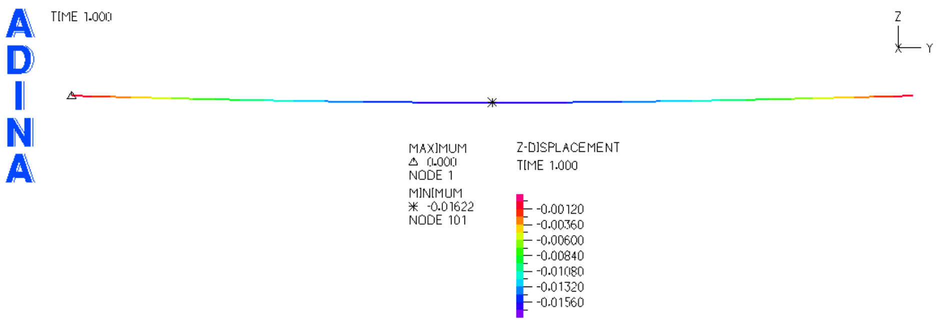
STEP 24. Displaying the results in the form of a map

In order to display the results in the form of maps, choose “Display → Band Plot → Create…” from the upper menu tabs, or choose  from the toolbars. The following window will open after this operation:



Displaying results related to displacements/flexures

In order to display results related to the beam displacements in the Z-axis, choose the “Displacement” option from the drop-down list in “Band Plot Variable:”, and “Z-Displacement” in the second drop-down list on the right. Then click the “OK” button (clicking “Apply” and then “OK” will result in the appearance of one model and two identical legends in the main model window. Of course, the legends can be deleted by clicking a given legend and pressing the DELETE key on the keyboard. It is recommended to only click the “OK” button). A window along with the generated vertical displacement is presented below:



Interpretation of the results

The maximum Z-axis displacement of the beam is in its center, and it is 0.01622 m.

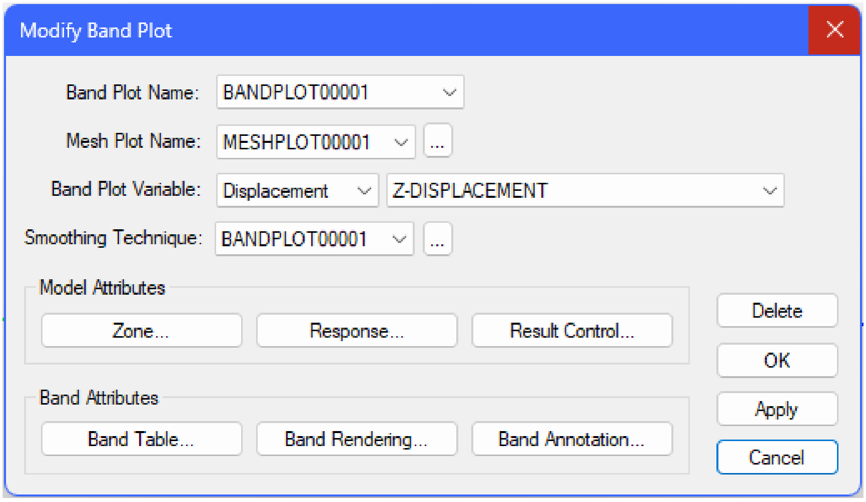
On the basis of an analytical calculations, the displacement of a simply supported beam with a constant load amounts to:

$$f = \frac{5 \cdot q \cdot L^4}{384 \cdot E \cdot J} = 0.01618 \text{ m}$$

According to the above, the result is almost perfect. The difference is due to the fact that the finite element method is an approximate method which may be ‘less’ precise than typical engineering calculations. Moreover, the approximation strongly depends on nodes of finite elements, number of divisions for a mesh, the shape of finite elements etc. In this case the small error is caused by the division into finite elements. The higher the mesh density, the better the results in this case, however, it is always important if the user needs such precision, which is costly in computing time and virtual memory necessity.

STEP 25. Changing the displayed plot/map

In order to display other resultant values, use “Display → Band Plot → Modify”, or click . In such a case, a new window will open, in which it is possible to introduce a larger number of modifications than just the type of the displayed results. The window in question is presented below:



The “Band Plot Name:” option specifies the style to which the plot is related. There can be different band plots in a single result file.

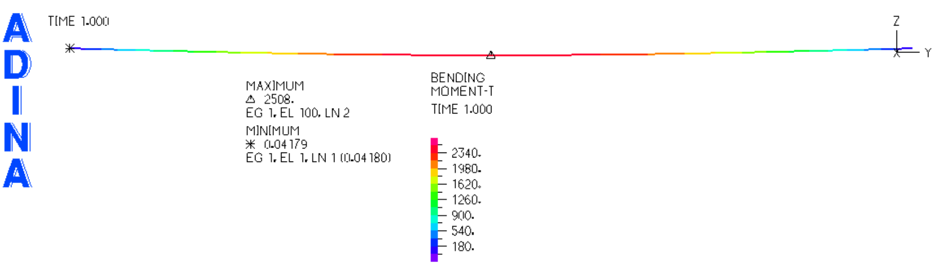
The “Mesh Plot Name:” option specifies the displayed plots to which the changes should apply.

The “Band Plot Variable:” options mean the type of the displayed results.

The “Smoothing Technique” option specifies the type of smoothing functions used in analyses of two- and three-dimensional objects with a mesh of finite elements consisting of geometry with 3 or more terminal nodes.

The remaining options will be described when they are used in the following examples.

Using the modification window for a plot/map, let us also try to display bending moments in the beam. Therefore, when using the “Modify” window, choose “Force” from the first drop-down list in “Band Plot Variable”, and “BENDING_MOMENT_T” in the second one, and then close the window with the “OK” button.



The maximum moment achieved in the software is 2508 N·m.

The value of the moment in the center of the beam determined using the analytical method amounts to:

$$M_{max} = \frac{q \cdot L^2}{8} = 2500 \text{ N·m}$$

As can be seen, the solutions produced in the ADINA software are similar to analytical solutions.

STEP 26. Displaying reaction values in the form of list of values

In order to check the values of support reactions, use the upper menu tabs: “List → Extreme Values → Zone…”. In the top right corner of the newly opened window, in the “Variables to List” group of options set the display type as “Reaction” in the first row/first column, and the type of displayed reactions as vertical reactions “Z-REACTION” in the second column, then click the “Apply” button. The remaining options in the window remain unchanged. If everything was done properly, the results should be displayed in the same window in the following form:

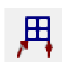
```
ADINA: AUI version 23.00.01.016, 18 April 2024: Problem 01
Licensed from Bentley Systems, Inc.
Finite element program ADINA, response range type load-step:
Listing for zone WHOLE_MODEL:
     POINT             Z-REACTION

Time 1.00000E+00

Node 1                5.01543E+03
Node 201              5.01543E+03



*** End of list.
```

In older ADINA versions the program is unable to display reactions on the model screen; the only possibility is to obtain them via a list.

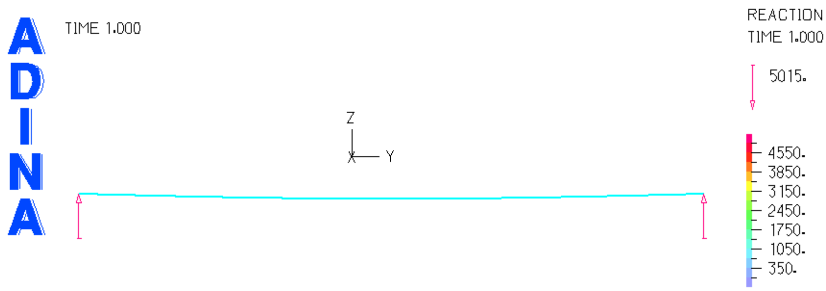
STEP 27. Deleting the displayed map

When wishing to delete the map (but not the model), use the button  from the toolbars.

STEP 28. Displaying reactions in the form of plot

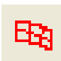
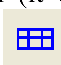
In the new ADINA 23.00.01 one can press the following button  corresponding to the option “Display → Reaction Plot → Create…”, choose the “Reaction Quantity:”

as “Reaction”, remaining fields leave as its defaults and click “OK”. The main window should look alike:



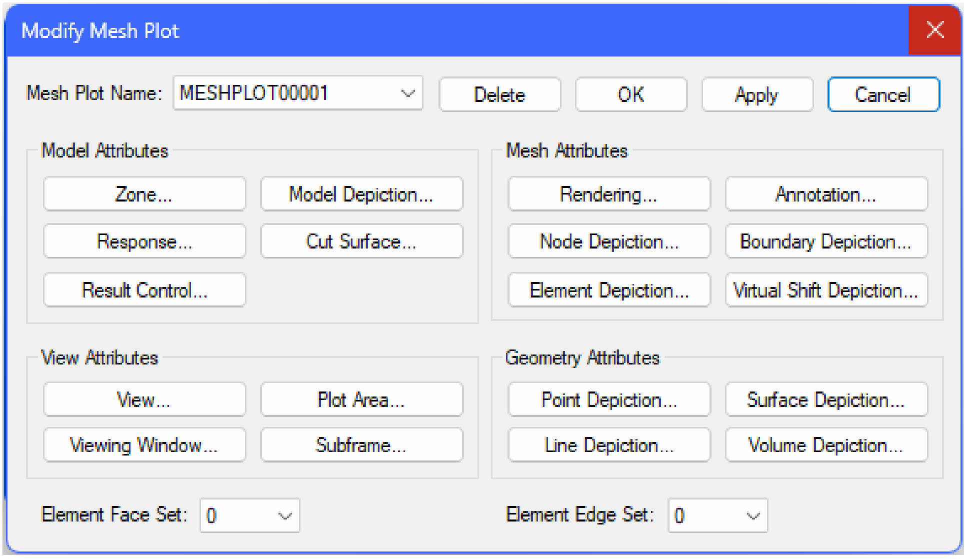
Once the reactions are shown, one can delete the plot (not the model), through the following button  located on the toolbars.

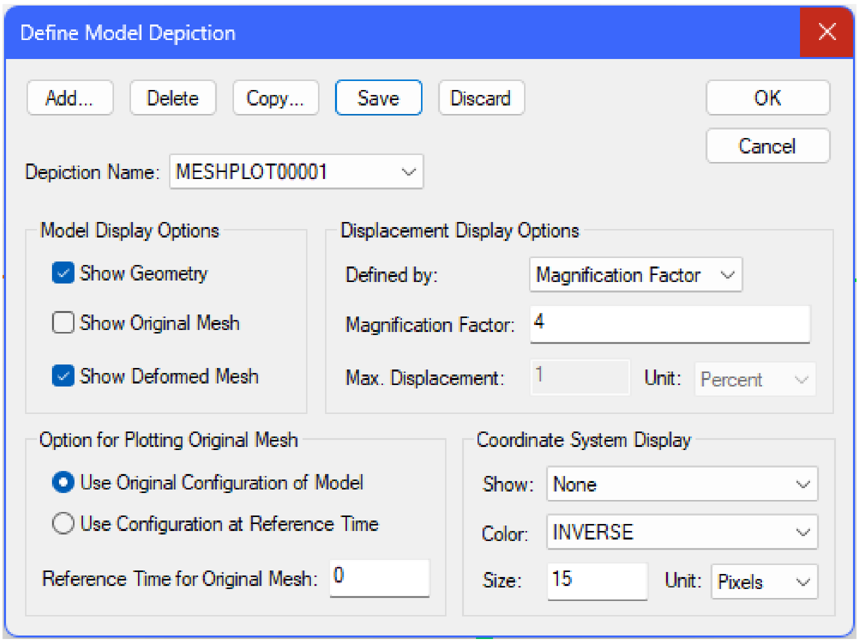
STEP 29. Options for displaying deformations of a finite element mesh

In order to display a deformed finite element mesh with respect to the construction, just use . When the displacements are quite large, it may be necessary to display the original finite element mesh (it can be activated along with displaying the deformed mesh). This is done by .

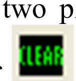
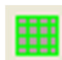
It is not uncommon for the displacements/flexures of the model to be very small; they may be invisible on the screen. In order to improve legibility, or check whether an anticipated flexure took place, just use .

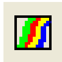
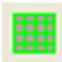
Of course, if the scale of flexure is still unsatisfactory, it can be set manually by means of options included in “Display → Geometry / Mesh Plot → Modify…”, or by means of . When choosing the “Model Depiction” button in the “Displacement Display Options” group of options, change the value in the “Magnification Factor” box to a higher number. Then click the “Save” button and “OK” in this window; the program will return to the main “Modify…” window; while there, click the “Apply” button, and then “OK”. Changes related to the view should take place in the model.



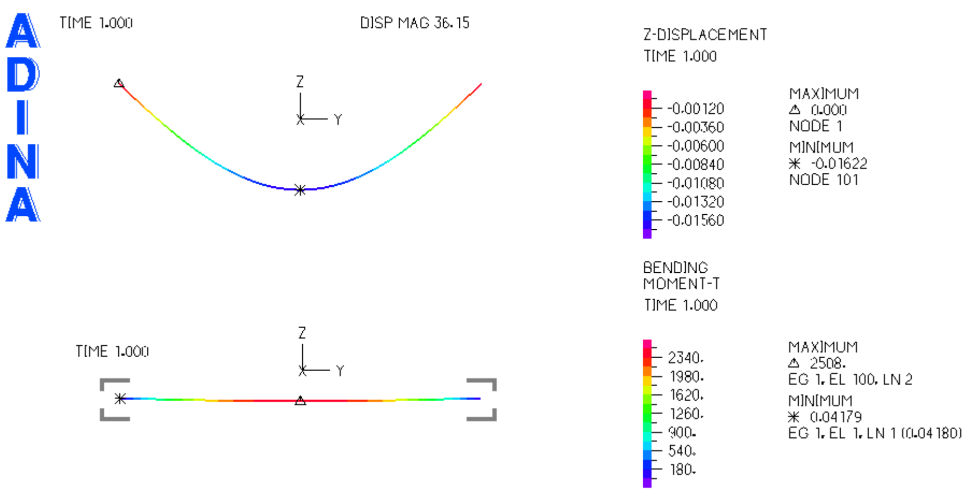


STEP 30. Simultaneous view of two or more different plots/maps

In order to display two plots simultaneously, clear the window of the previous models by means of . Subsequently, in order to add one plot/map, use 

and add a plot/map with . The next step is to shrink the model, so that it would occupy no more than ½ of the window along with descriptions. This is done by means of scaling  and moving the components of the plot/map . In order to add another plot, repeat the action of adding another model to the window by means of . The use of the remaining functions remains unchanged.

A user may want to introduce changes to one of the existing models (if there are more than one). In order to do this, just use the left mouse button to select the proper plot (the current choice will be indicated by a frame with wider gray lines next to the vertices of the frame), and then use the function which interests the user. The figure below presents two plots in one window:



EXAMPLE 2
A TRUSS SYSTEM. DISPLACEMENT OF A NODE DUE TO THE ACTION OF A CONCENTRATED FORCE. DETERMINING ZERO FORCE MEMBERS

The present example shows the consecutive steps related to the modeling of a truss system. Some steps will be analogical to the previous example, thus no commentary will be added to the applied functions which were discussed previously. Scheme of the analyzed truss is presented in Figure 20.

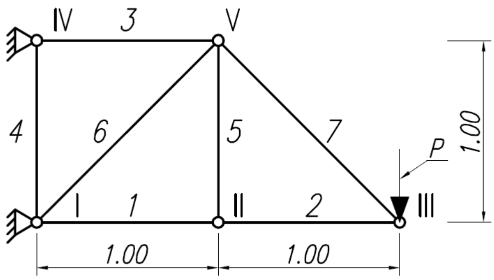


Fig. 20. Truss scheme

Bar lengths L and H as well as the load value are equal:
$L = 1.00$ m
$H = 1.00$ m
$P = 5.00$ kN

Material constants for steel material:
$E = 210$ GPa $= 2.1\mathrm{e}^{11}$ Pa
$\upsilon = 0.30$

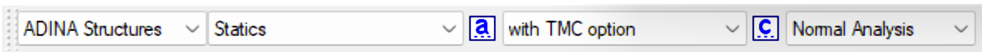
The diameter of each truss bar is assumed as:
$d = 0.10$ m
therefore, the cross-section area of a single bar needed in further computations is equal to:

$$A = \pi \cdot \frac{d^2}{4} = 7.853981 \cdot 10^{-3}\ \mathrm{m}^2$$

<u>Note</u>: In this example mass-proportional load is not included in the calculations. The truss system is a planar system, thus the definition should take place in the positive quarter of the coordinate system (the YZ axes).

STEP 1. Definition of the type of analysis

Upon opening the ADINA software, choose “ADINA Structures” from the “Module Bar” toolbar in the window comprising a drop-down list, and choose “Statics” in the second drop-down list.

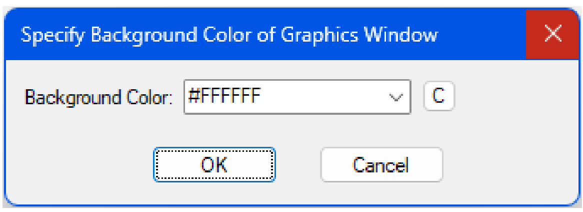


STEP 2. Entering the heading of the model

In order to specify a heading, choose “Control → Heading” from the upper menu tabs, and then enter a heading, e.g., “Planar truss system”. Upon entering a heading, click the “OK” button.

STEP 3. Definition of the background color of the main model window

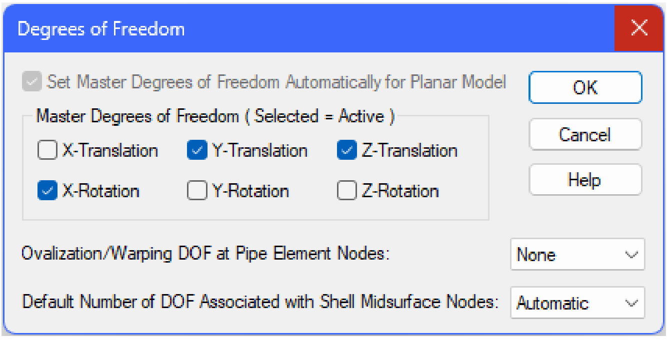
In order to define the background color, go to “Edit → Background Color…”. Then, in the newly opened window choose the color white from the drop-down list. Choose the item called “WHITE” from the drop-down list, and confirm the choice with the “OK” button.



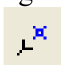
<u>Note</u>: The background color is not saved along with the model. This means that after each opening of the file, the background color will return to default – “black”.

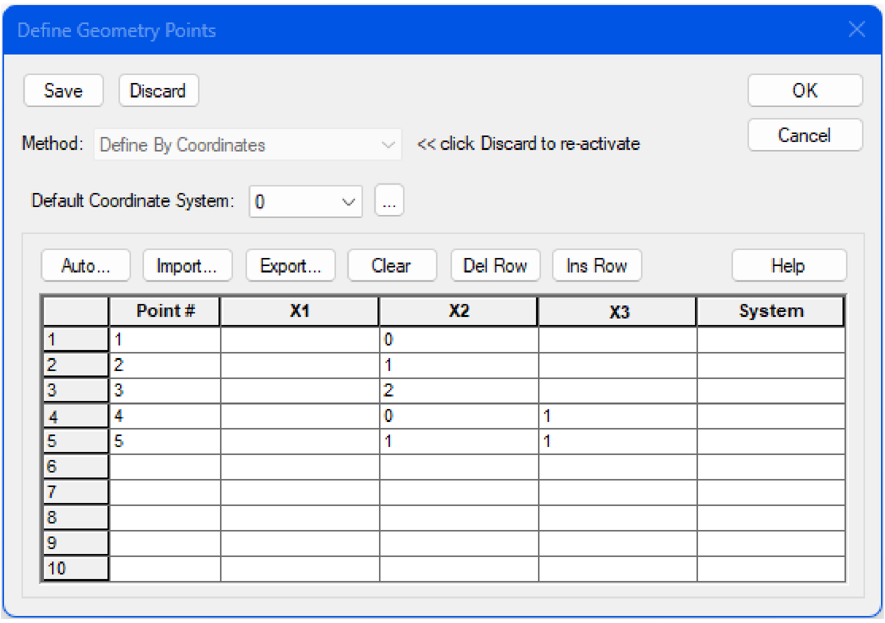
STEP 4. Definition of global boundary conditions

Because the analysis is performed in two dimensions (2D), one should select the master boundary conditions to give the ability to move along the Y and Z axes, as well rotate with respect to the X axis.



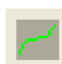
STEP 5. Definition of points

The definition of points takes place by selecting "Geometry → Points → Define" from the upper menu tabs, or by means of . Input the data in accordance with the figure below:



Then confirm the data by clicking "Save" and "OK".

STEP 6. Definition of lines

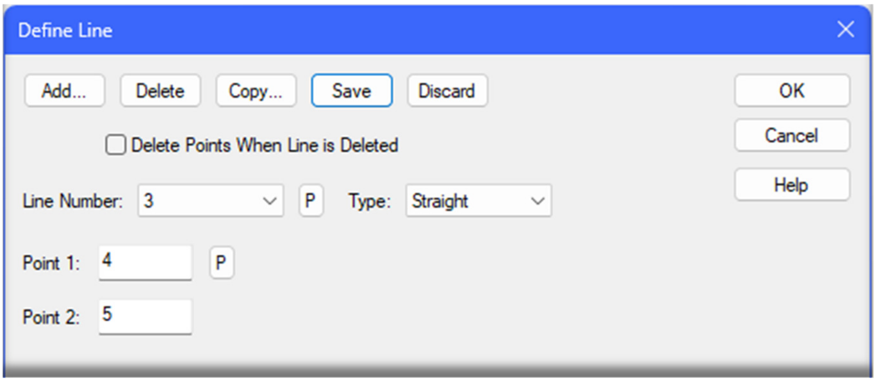
In order to define a line, go to "Geometry → Lines → Define…", or use . In order to add a larger number of lines, the pattern is as follows: "Add… → Type:

Straight → Point 1: … → Point 2: … → Save”. This action should be performed as many times as the number of lines needed in the figure.

The definition of lines may take place in any sequence. In the example the following sequence is used:

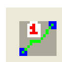
Line Number:	Point 1:	Point 2:
1	1	2
2	2	3
3	4	5
4	1	4
5	2	5
6	1	5
7	5	3

A view of the window with data input for line no. “3” is presented in the figure below:

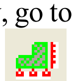


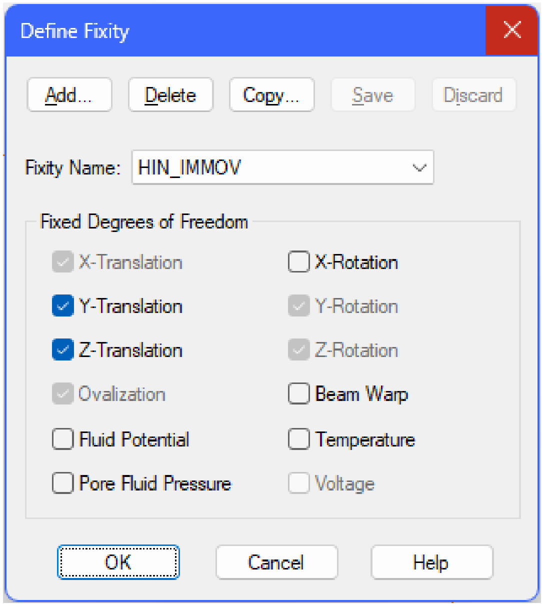
As soon as all the lines are introduced, confirm changes with the “OK” button.

STEP 7. Displaying the lines numbers

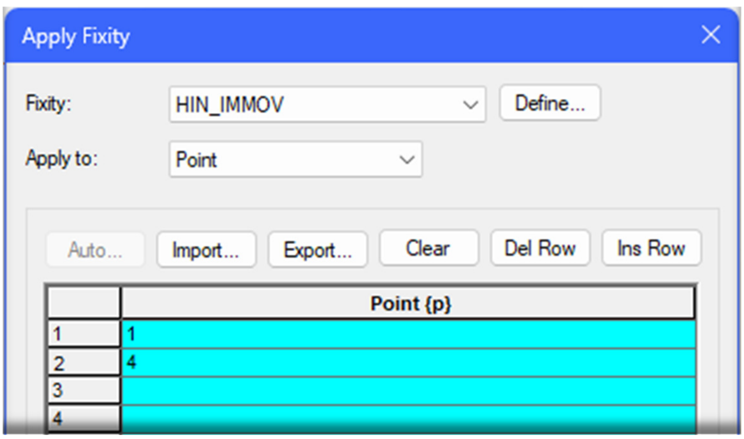
In order to display the lines numbers for their easier identification, click  .

STEP 8. Definition of boundary conditions (fixity characteristics)

In order to define and apply a fixity, go to “Model → Boundary Conditions → Apply Fixity…”, or do it by means of . In the newly opened window, click the “Define…” button, and then “Add…” in order to add a new fixity (immovable hinged support). Enter “Hin_immov” as the title, and then select Y and Z translations. Leave rotation around the X axis unrestrained. The remaining entities are unrelated to the current model, so it may remain unchanged. Confirm the introduced changes with the “Save” button.

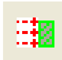


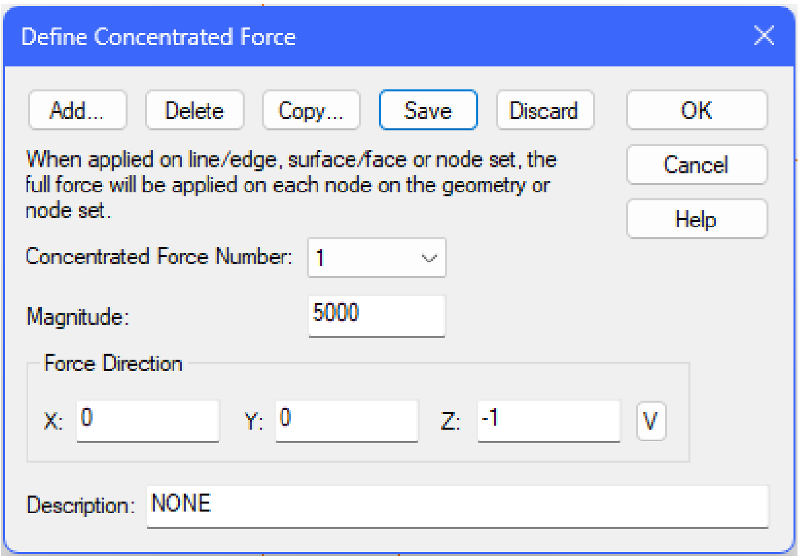
Following this, confirm changes in the fixity definition window by clicking “OK” and move on to the applying of boundary conditions window. Make sure that the “Apply to:” option is set to “Point” and the “Fixity” option is set to “HIN_IMMOV”. Then in the table rows insert points 1 and 4 (nodes I and IV shown in Figure 20).



Then confirm the changes by clicking the “Apply” and “OK” buttons.

STEP 9. Definition of loads

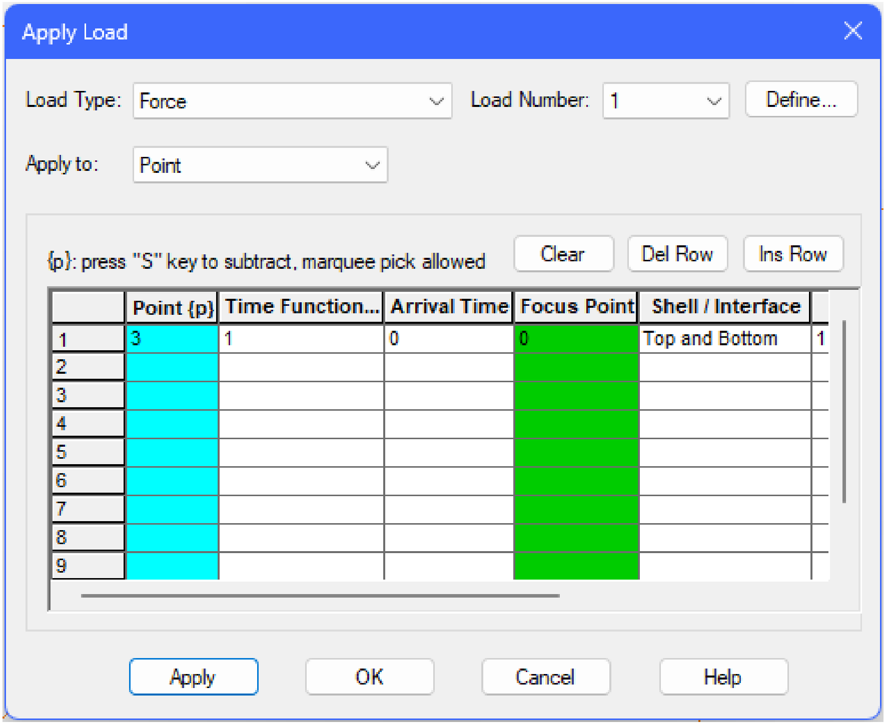
In order to add a load to the model, go to "Model → Loading → Apply…", or choose this option by means of . In the newly opened window make sure that "Load Type:" shows a concentrated force – "Force". Then click the "Define…" button, "Add…" button and enter 5000 in the "Magnitude:" box, and a value of –1 in the "Force Direction → Z:" box, because the force acts opposite to the Z axis. Description may be left unchanged. When the values have been specified, click the "Save" button and leave the window with the "OK" button.



Upon exiting, make sure that the "Apply to:" option in the main load definition window in the model displays "Point".

Subsequently, enter the node number "3" manually in the "Point {p}" column of the table, or choose the construction node by double-clicking the row of the column. After this operation, press the ESC key on the keyboard. Leave other options unchanged.

Then close the window by clicking the "Apply" and "OK" buttons.



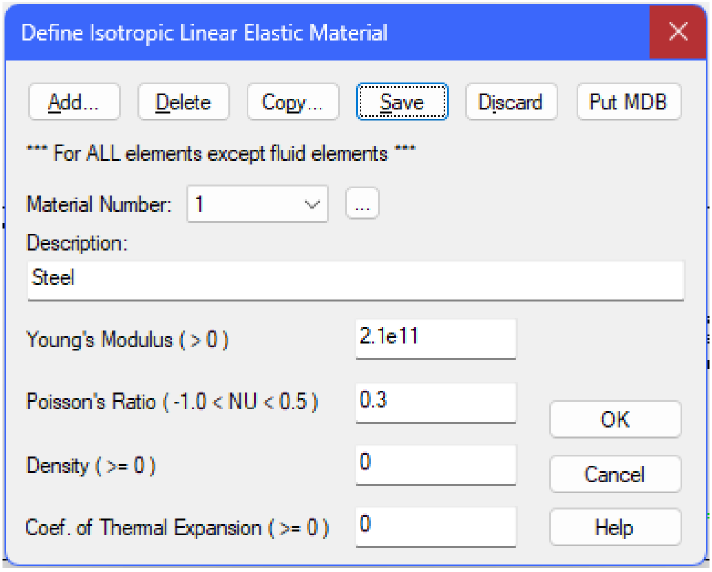
STEP 10. Displaying boundary conditions and loads

In order to display the defined loads in the main model window, click . In order to show boundary conditions, click .

STEP 11. Definition of material constants

In order to go to the declaration of material constants, choose the "Model → Materials → Manage Materials" tab, or click . In the newly opened window, find and click the "Isotropic" button in the "Elastic" group of materials. An alternative method of reaching the window with the characteristics of an isotropic material is to find "Model → Materials → Elastic → Isotropic..." in the upper tabs of the program.

In this window, first click "Add..." in order to add a new material, and then enter the values of material constants like in the figure below:

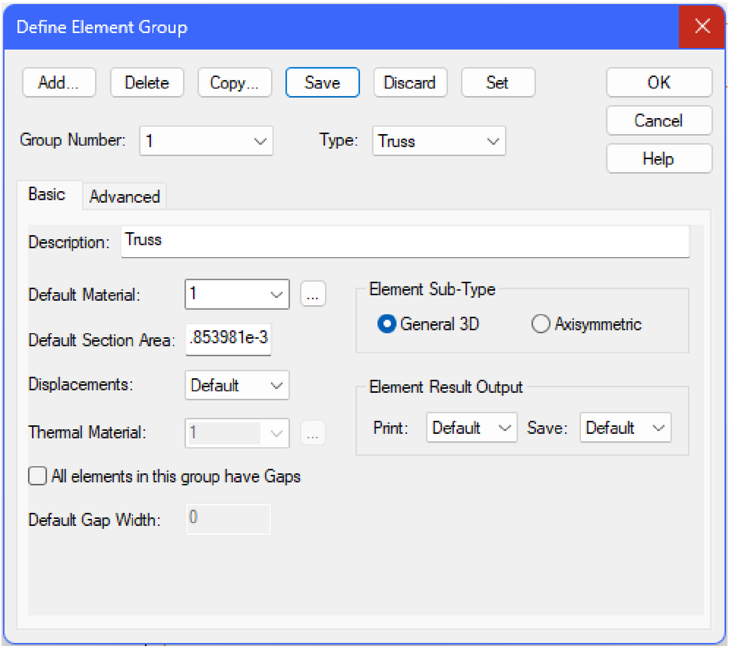


The density of the material may be omitted, since the mass-proportional load of the truss is not taken into consideration in the calculations. Upon declaring the values for a material, changes are confirmed by clicking “Save” and “OK”, respectively. In the material manager window, press the “Close” button to leave the window.

STEP 12. Specifying the type of the analyzed construction

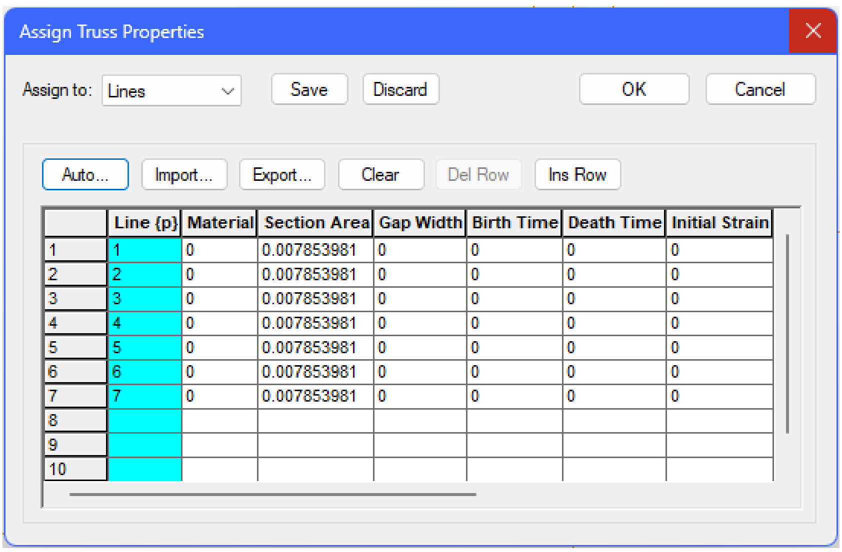
In order to furtherly create finite elements treated as truss elements, one need to add the proper group by means of “Meshing → Element Groups…”, or by means of . Then in the newly opened window click “Add…”, and from the drop-down list choose “Truss” for the “Type:” option. Make sure that the “General 3D” option is selected in “Element Sub-Type”. The “Axisymmetric” option is used only for trusses that has one degree of freedom per node along the Y axis direction. According to that the Z axis is the axis of rotational symmetry and the Y axis is the radial direction. Such elements may be subjected only to circumferential force. It should be noted that most of truss systems will be created with “General 3D” option.

Apart from the presented options, enter a value of “7.853981e-3” in the “Default Section Area” box in order to define a cross-sectional area for each of the truss bar element in this element group.



It should be noted that some trusses have bars with various cross-sectional areas; therefore, for this purpose, one enters "1" in the "Default Section Area:" box, while "Model → Element Properties → Truss…" provides the ability to define both the material and the cross-section areas as well as a vast of other options for each bar separately.

For the truss from this example, the figure below demonstrates a general idea how to fill out the table of characteristics for each bar, assuming that a value of "1" would be entered in the "Define Element Group" window next to the "Default Section Area" option.

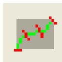


	Line {p}	Material	Section Area	Gap Width	Birth Time	Death Time	Initial Strain
1	1	0	0.007853981	0	0	0	0
2	2	0	0.007853981	0	0	0	0
3	3	0	0.007853981	0	0	0	0
4	4	0	0.007853981	0	0	0	0
5	5	0	0.007853981	0	0	0	0
6	6	0	0.007853981	0	0	0	0
7	7	0	0.007853981	0	0	0	0
8							
9							
10							

Both methods of introducing cross-section areas in the presented problem would lead to obtaining the same numerical results.

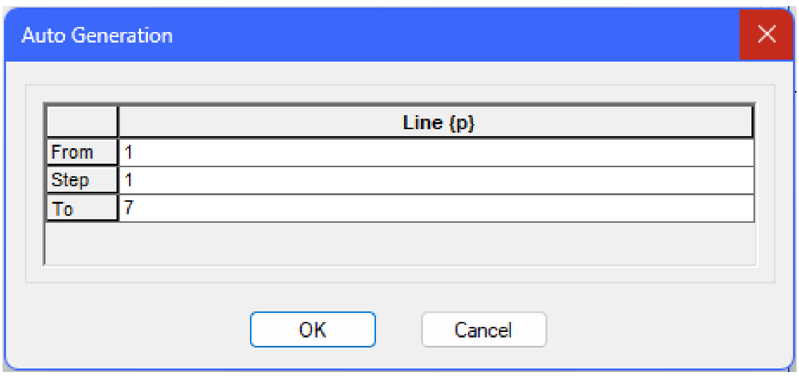
STEP 13. Definition of finite elements

A truss is a structural arrangement consisting of a large number of bar elements which can be considered as segments. An attempt of subdividing them via mesh density will produce error in the window during the model calculations, and the structure will not be calculated.

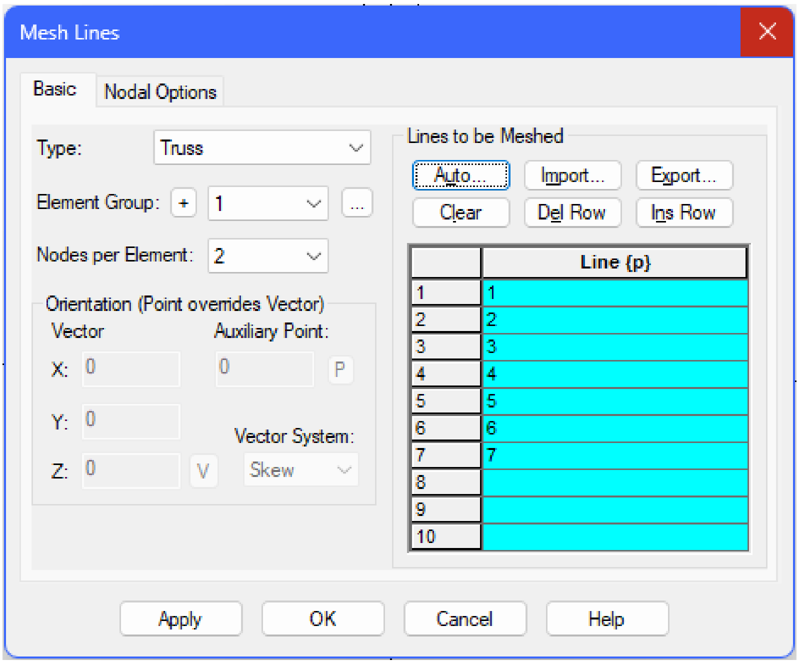
In order to generate finite elements on the bars of a truss, go to "Meshing → Create Mesh → Line", or choose  from the toolbars.

Then make sure that the "Type:" option shows "Truss". According to that only stretching/compressing forces are considered in the truss, the "Nodes per Element:" option should be left as "2". Then, fill out the table manually with values (bar numbers) from 1 to 7. One alternative way is to click any row of the table and choose lines from the model, and then confirm selection by pressing the ESC key on the keyboard.

Another way to fill out the table is to use the "Auto…" button located over the table. Enter a value of "1" in "From" – this option specifies the line number from which the program start; subsequently, in the "Step" box also enter a value of "1", since this is the value of the increment of line numbers. In this case, enter the number of the last bar in the "To" option, i.e., "7". The figure below presents a properly filled out window:



Confirm the introduced changes by clicking "OK"; this will also cause the table to be filled out with numbers from 1 to 7. Further, in the "Mesh Lines" window, click "Apply" and "OK" to return to the main model window.



STEP 14. Starting calculations

In order to start calculations, choose "Solution → Data File/Run" from the upper menu tabs, or choose  from the toolbars. Subsequently, input the path for saving the resultant file. However, before the user clicks the "Save" button, they should make sure that the "Run Solution" and "Automatic" function near the "Maximum Memory for Solution" options are selected in the window in the "Adina Structures Solution" group of options. After starting the calculations and recalculating the model, close all three dialogue windows which appeared during the calculations.

STEP 15. Post-processing module (results)

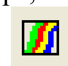
In order to go to the results module, change "ADINA Structures" to "Post-Processing" in the drop-down list of the "Module" toolbar.

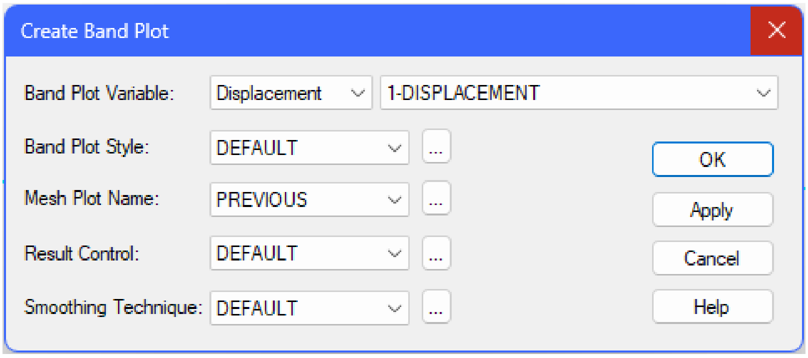
When a message appears stating that changes in the drawing have not been saved, it is recommended to save the model.

STEP 16. Opening the resultant file

In order to open the resultant file, choose  from the toolbar, or choose "File → Open" from the upper menu tabs.

STEP 17. Displaying the results in the form of a map

In order to display the results in the form of maps, choose "Display → Band Plot → Create…" from the upper menu tabs, or choose  from the toolbars. The following window will open after this operation:



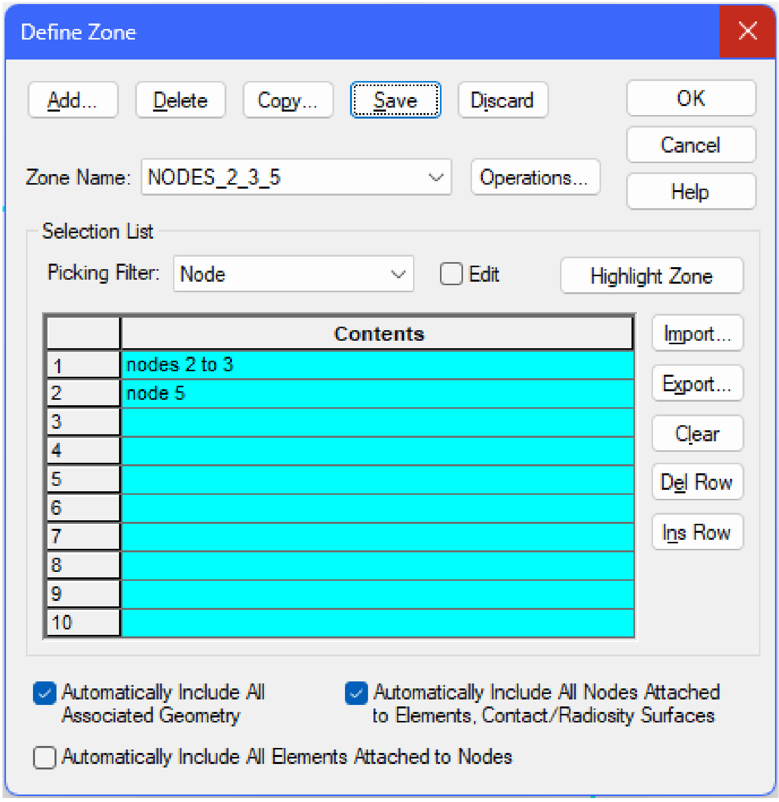
In the window, choose the map of resultant values which interests the user.

STEP 18. Determining the displacements of nodes

Before defining the group of nodes which are the points of interest one should click following buttons on the toolbars:  – this button shows the numbers assigned to finite elements nodes and  – this button shows the finite elements nodes, which can be furtherly chosen by another windows from the ADINA program.

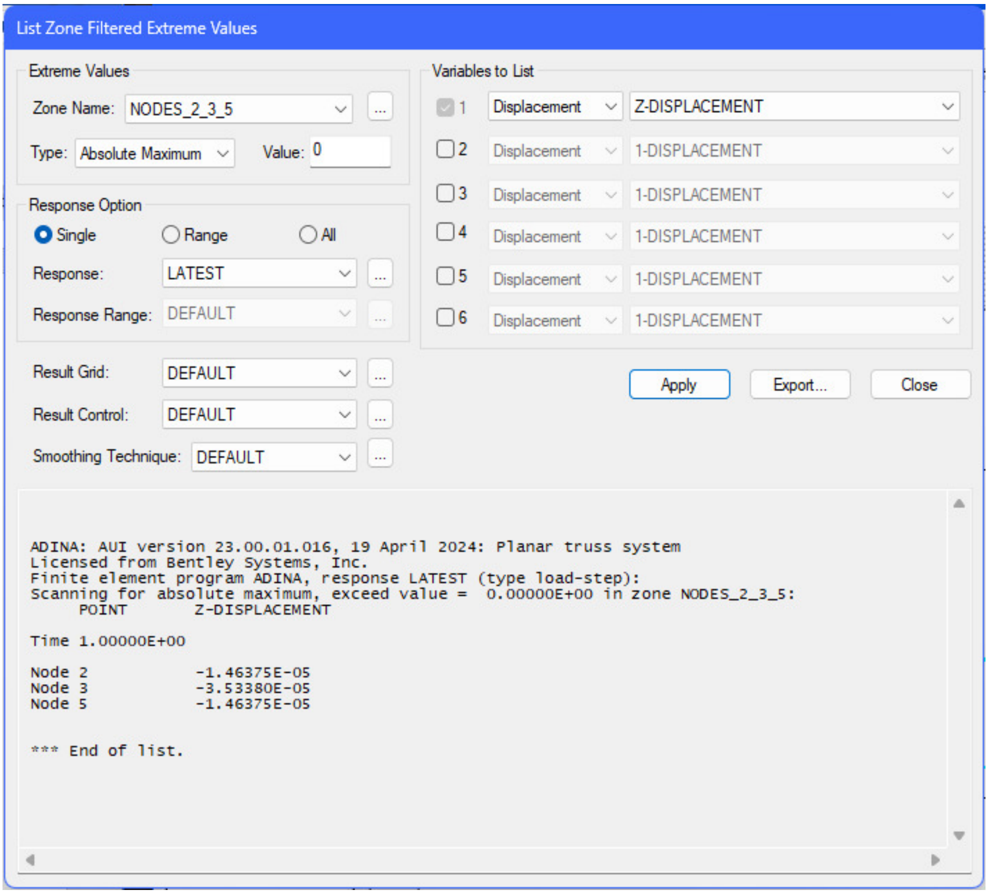
Now let's go back to determine the displacement values of nodes II, III and V. In order to do this, go to "Definitions → Zone…", and then in the newly opened window click the "Add…" button and enter the name of a new set, e.g., "Nodes_2_3_5". For the "Picking Filter:" option, change the value from "ALL" to the value which refers to the nodes – "Node". Once the picking option has been selected, double-click the first row of the table and choose the nodes which interest the user, then use the ESC key on the keyboard for confirmation. The numbers of nodes in the table can also be input manually (however, the name "node" should also be specified before the number).

In this case, the remaining options remain unchanged. When the nodes have been selected, confirm changes by means of the "Save" button, and then "OK". A properly defined window of nodes is shown in the "Define Zone" figure below:



In order to display values of displacement in selected nodes, go to "List → Filtered Values → Zone…" (the window shown below). In order to show only final values for the selected nodes, for the "Zone Name:" option choose the name of the nodes defined by the user from the drop-down list, in this case "Nodes_2_3_5". Then make sure that the "Type:" option displays "Absolute Maximum". In the "Response Option" group of options, select the "Single" option, and in "Response" choose "LATEST" from the drop-down list. In this case, all that remains is to specify the type of the displayed final results for the selected nodes; therefore, choose the result which interests the user in the right part of the "Variables to List" window.

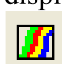
In this example, displacements along the Z axis have been selected, so this type of results should be "Displacement", and more precisely "Z-DISPLACEMENT". It is also possible to display a larger number of results for the input nodes, just by selecting the boxes next to a value of 2 or 3, etc., in the "Variables To List" group of options, and then choose the requested type of the displayed results.



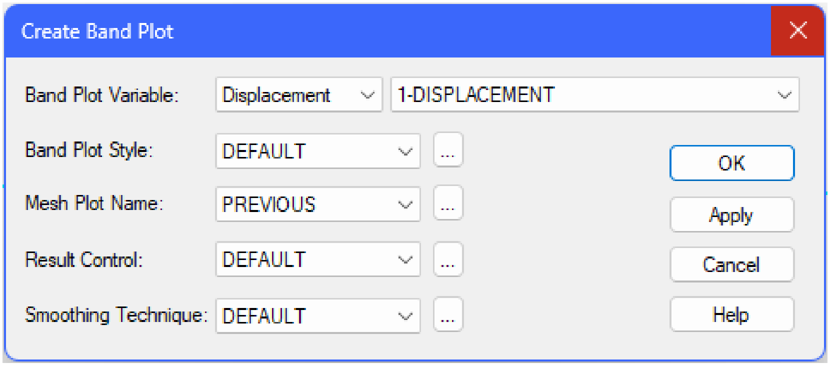
Each time, all changes in the above window should be approved by the "Apply" button, while the "Close" button should be clicked to leave the window.

It should be noted that the 'presented' at the moment list of results can be exported to a *.txt file by means of the "Export…" button.

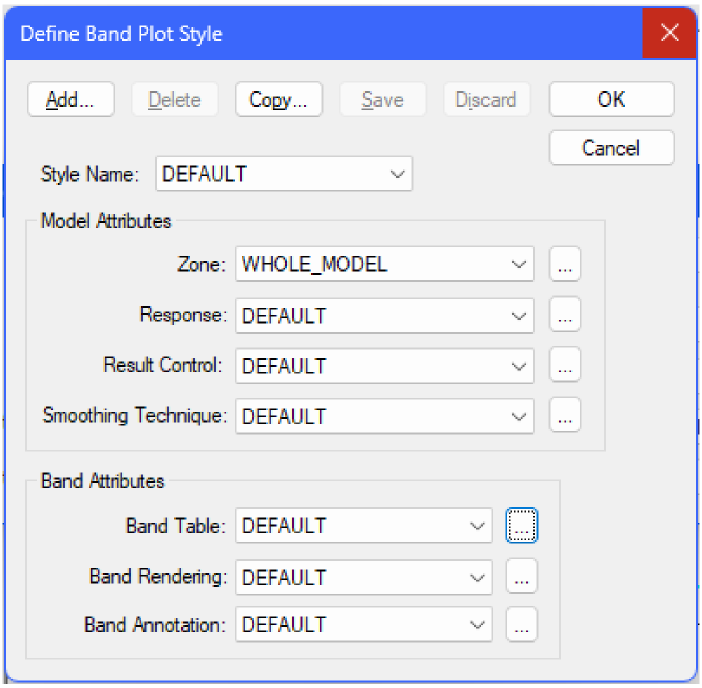
STEP 19. Determining the "zero force members" of a truss by means of maps

This operation can be performed by displaying a force value map. In order to display the zero force members, go to "Display → Band Plot → Create…", or click .

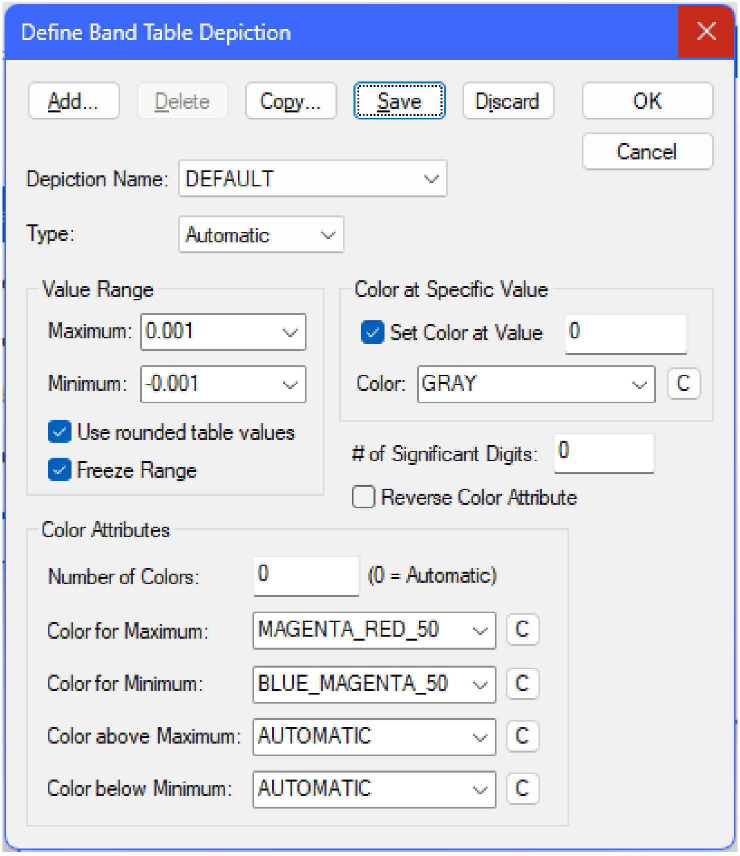
In the newly opened window, click the button with three dots next to the "Band Plot Style" option.



Subsequently, in the “Define Band Plot Style” window, click the button with three dots next to the “Band Table” option. The next window called “Define Band Table Depiction” will open, in which it is possible to define the values of the sought results.



In order to define the sought values, change the values corresponding to “Minimum” and “Maximum” in the “DEFAULT” style in the “Value Range” group of options to the value very close to “0”. This is caused by a fact, that the numerical values are not equal zero, they are very close to, but not equal. It should be taken into account that the “Minimum” value obligatorily have a negative sign. Subsequently, in the “Color at Specific Value” group of options (indicating a specific value with an individual color), select the box belonging to the “Set Color at Value” option.



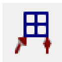
Enter a value of "0" in the value box, and specify the color as desired. Then click the "Save" button and "OK".

Note: Definition of one's own style in the window (along with its name) will cause its absence from the "Band Table" drop-down list of the "Define Band Plot Style" window; therefore, it is recommended to change the "DEFAULT" style.

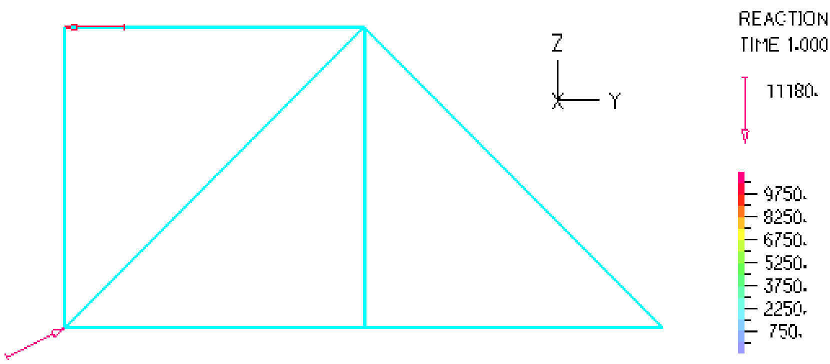
Upon returning to the "Define Band Plot Style" window, click the "Save" and "OK" buttons, until the "Create Band Plot" window is displayed. In this window, from "Band Plot Variable" choose "Force" as the type, and "AXIAL_FORCE" as the sub-type. Subsequently, the "OK" button will display the map in the window.

Note: An important problem of this method is that on the next plot/map after changing the requested parameters, e.g., from forces to displacements etc., the program will still pick previously set values. In such a case, delete the plot completely by means of , and then repeat this entire step, until reaching the "Maximum" and "Minimum" options. In order to let the program automatically adopt a legend of results, enter the word "automatic" in both "Maximum" and "Minimum" option. Moreover, unselect highlighting the value "0" with a specified color.

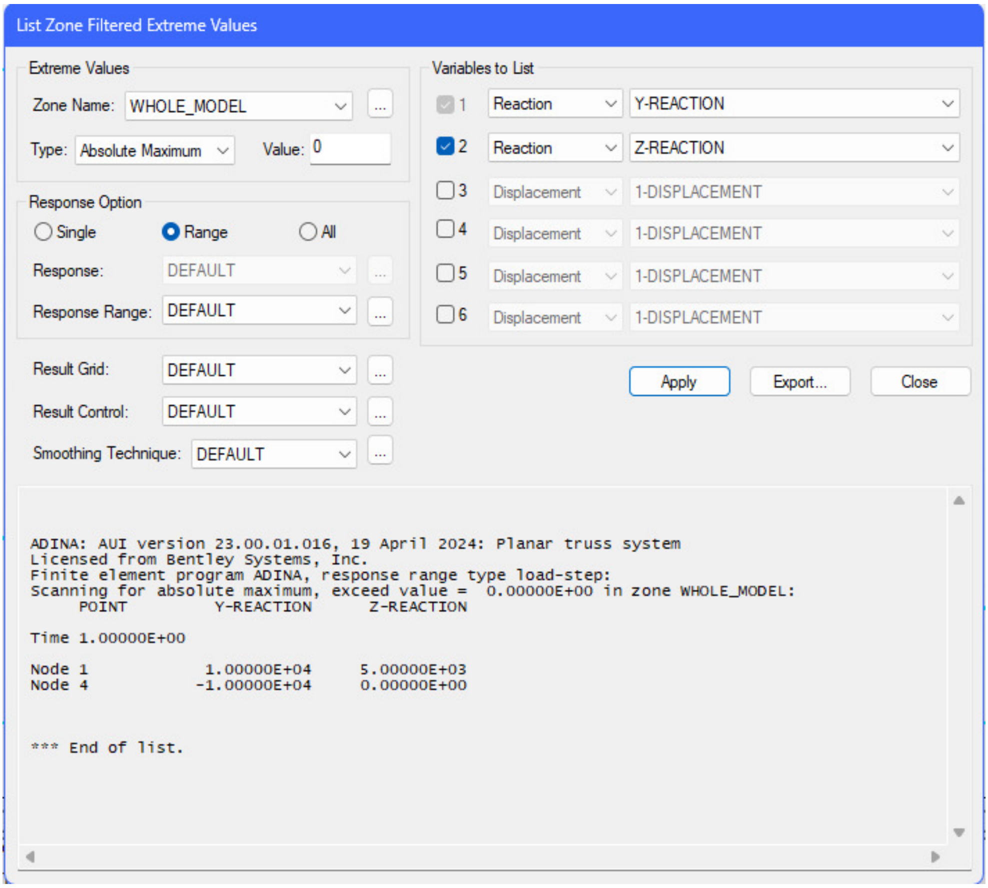
STEP 20. Determining a reaction in the support nodes of a truss

In order to display support reactions (the direction of their impact), go to "Display/ Reaction Plot/Create…" or click , and then in the newly opened window choose "REACTION" from the "Reaction Quantity" menu. Then click the "OK" button.

The window should look like this:

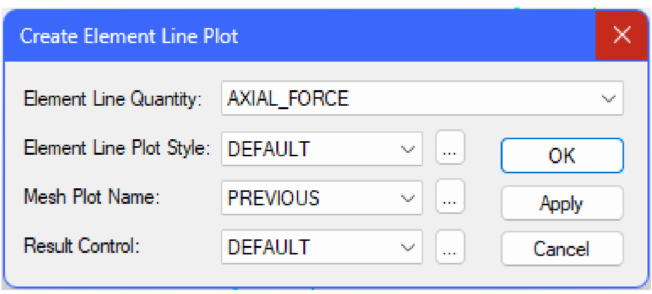


As can be seen, the displayed reactions have two directions of impact. In order to learn the reaction values, go to "List/Filtered Values/Zone…", then in the "Variables to List" group of options in the newly opened window choose "Reaction" in the first box, and choose "Y-REACTION" from the neighboring drop-down list. Select the second row so that it would also be included when displaying the results, and change the right drop-down lists to "REACTION" and "Z-REACTION". When these actions are completed, click the "Apply" button. The program window should look like in the figure below:

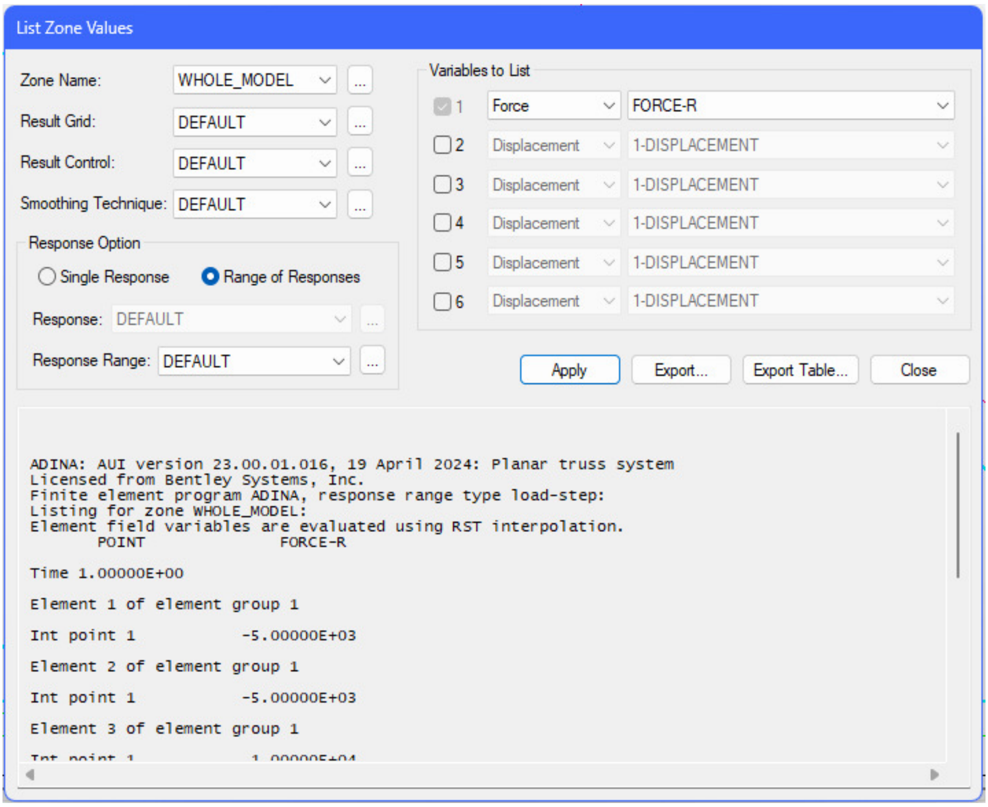


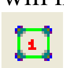
STEP 21. Determining normal forces in the bars of a truss

Similarly to specifying the reactions, it is possible to start with displaying normal forces occurring in the bars of a truss in the form of a plot. To this end, go to "Display → Element Line Plot → Create…", and then in the "Element Line Quantity" box choose "AXIAL_FORCE", leave the remaining options unchanged, and then click "OK" in order to confirm the changes and leave the window.



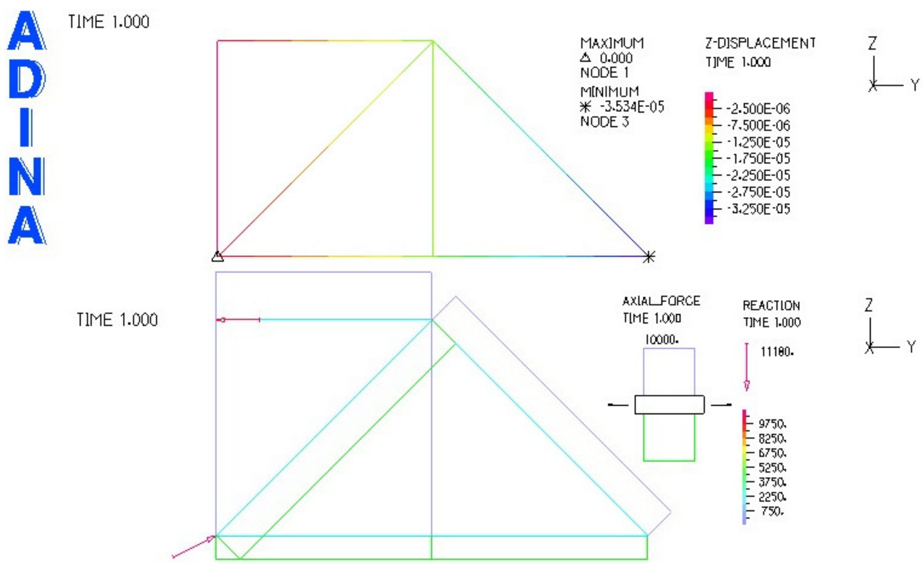
The plot only indicates which bars are stretched and which ones are compressed. It is impossible to read the precise values of axial forces in the bars from that plot. In order to read the real values of forces in bars, go to “List → Value List → Zone…”, and in the “Variables to List” group of options in the newly opened window choose “Force” in the first row, and the “FORCE-R” subtype from the neighboring drop--down list. Then click the “Apply” button.



Note: Since the results values are presented for the finite elements “Element” itself, and not directly are referred to the lines, there may be a situation in which a given line will have a different number of finite element. This should be checked by click-ing  in the model window. Upon clicking this button next to each line (in the truss), a proper element number will appear for a given line.

STEP 22. Creating a raster object of the current model

If there is an interesting graph, or simply a need to save the current model in the form of a snapshot, click  or go to “File → Snapshot/Movie Save → Snapshot (bitmap)…", then choose a file save location and a format (*.jpg or *.bmp). A sample snapshot looks like this:



<u>Note</u>: A snapshot results in displaying only the main resultant model window. Therefore, all the buttons, toolbars, etc. become hidden.

EXAMPLE 3
DEFORMATION OF A PLANAR FRAME WITH RIGID CONNECTIONS AT NODES SUBJECTED TO THE VARIABLE EXTERNAL FORCE

In this example the activities related to modeling of a frame with stiffeners at the connection nodes using a perfectly stiff material are presented. Each stiffener is assumed to be 0.25 m long, measuring from the center of the node along a given bar. The reader is advised to refer to the previous examples as some of the features have been discussed earlier. The diagram of the analyzed model is shown in Figure 21.

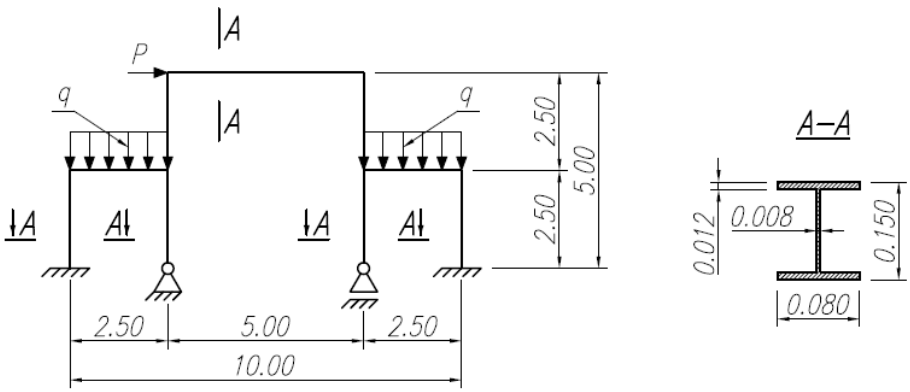


Fig. 21. Scheme of analyzed frame

The following data have been assumed in the calculations:
linear load $q = 10\ \mathrm{kN/m}$ (static and sinusoidal)
concentrated force $P_{max} = 20$ kN

Material: steel
$E = 210$ GPa $= 2.1\mathrm{e}^{11}$ Pa
$\upsilon = 0.30$
$\rho = 7860\ \mathrm{kg/m^3}$

Boundary conditions: Clamped supports on the external parts of the aisle (removal of 3 degrees of freedom for planar calculations), pinned support (release of rotation around the X axis in the left part of the inner aisle at the connection with the ground) and pinned support with the ability to move in the horizontal direction (release of rotation around the X axis and release of displacement in the Y direction in the right part of the internal aisle at the connection with the ground)

STEP 1. Specifying the type of analysis

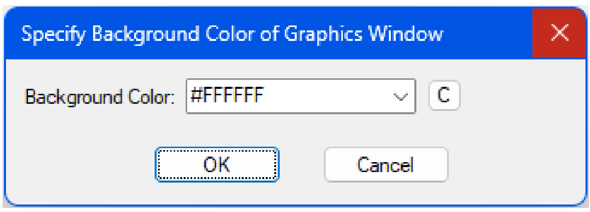
Choose “Adina Structures” from the drop-down list in the “Module Bar” toolbar, and “Static” as the analysis type from the second drop-down list on the right.

STEP 2. Entering the heading of the model

In order to specify a heading, choose “Control → Heading” from the upper menu tabs, and then enter a heading, e.g., “Rigid frame”. Upon entering a heading, click the “OK” button.

STEP 3. Definition of the background color of the main model window

In order to define the background color, go to “Edit → Background Color…”. Then, in the newly opened window choose the color white from the drop-down list. Choose the item called “WHITE” from the drop-down list, and confirm the choice with the “OK” button.



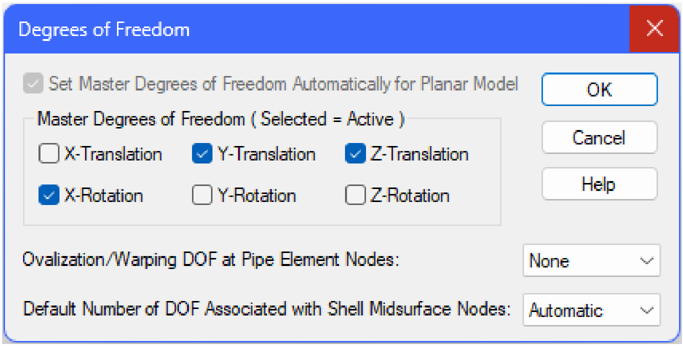
<u>Note</u>: The background color is not saved along with the model. This means that after each opening of the file, the background color will return to default – “black”.

STEP 4. Definition of global boundary conditions

Since the considered model is a planar model considered in the program default YZ plane, boundary conditions that do not participate in the analysis process should be excluded. To check which boundary conditions are active, go to “Control → Degrees of Freedom...”. In the newly opened window, all boundary conditions in the “Master Degrees of Freedom (Selected = Active)” option group should have their boxes checked. Make the following changes in the window:

X-Translation	Unchecked
Y-Translation	Checked
Z-Translation	Checked
X-Rotation	Checked
Y-Rotation	Unchecked
Z-Rotation	Unchecked

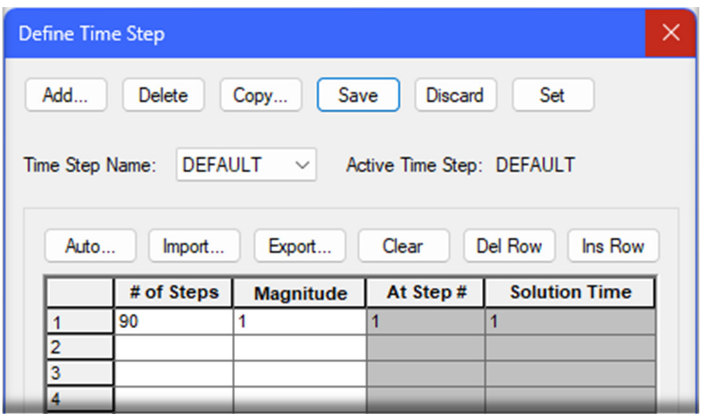
The figure below shows the window with selected boundary conditions:



The remaining data is left unchanged. After selecting the appropriate boundary conditions involved in the analysis, exit the window by pressing the “OK” button.

STEP 5. Definition of the number of time steps

To define time steps, go to “Control → Time Step...”. Then, in the newly opened window, enter the value of “90” in the first row of “# of Steps” column and in the same row under the “Magnitude” field, make sure that the value is “1.0”. The window view is shown in the figure below:



After entering the data, press the “Save” button and then leave the window with the “OK” button.

Note: In this example the number of steps means, that the analysis will take solved for 90 steps (each step of magnitude 1.0). In real problems it refers to the number of ‘time’ steps. In example, if we would like to take an analysis of 50 seconds with an interval of 2.5 s between consecutive steps, in the “# of steps” should be introduced “20” and in the same row under the “Magnitude” should be placed “2.5”. It means that 20 multiplied by 2.5 s would give the full time of 50 s, however in the post-processing one will be able to observe the model behavior in 2.5 s intervals.

STEP 6. Definition of load variability in subsequent time steps

In this step, three different types of load variability over time will be defined. The first type will determine a static load with a constant value along the whole analysis, the second type will determine the increase in force from 0 to 20 000 N using a load multiplier (in fact, a unit load will be used under the definition of load, due to the fact, that it will be multiplied by magnitude value from “Time Function”), while the third type will determine a sinusoidal variable load, in which the force multiplier at the highest points of sinusoid will be “1.0” and “–1.0” respectively. Hence, for this case the load magnitude will be defined using the load definition.

To declare load variability over time, go to “Control → Time Function...”. In the newly opened window, enter the following data:

Time Function Number:	1
Function Multiplier:	Constant (=1.0)

Values in the table

	Time	**Value**
1	0	1
2	1.00000000e+020	1

Note: It is possible to create time steps with a variable value of time. In such a case, declare a specified number of steps for the requested value of time in each row of the table. The time of the final step in the calculations equals the sum of:

$$\sum_{i=1}^{\infty} Steps_i \cdot Value\ of\ time$$

Then press the “Add...” button to add the load variation function no. 2 and enter the following data:

Time Function Number:	2
Function Multiplier:	Constant (=1.0)

Values in the table

	Time	**Value**
1	0	0
2	90	20 000

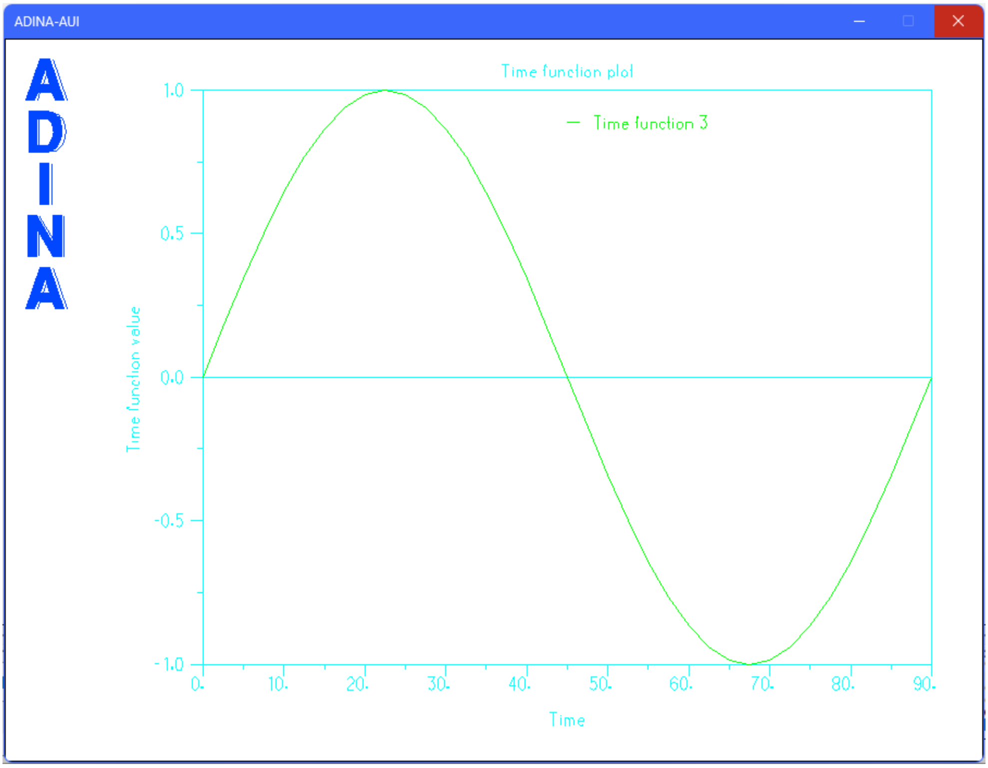
After entering the data, press the “Save” button and then “Add...” again to enter the sine function, and then enter the following data in the window:

Time Function Number:	3
Function Multiplier:	Sinusoidal
Function Parameters	
Angular Frequency (deg/time):	4
Phase Angle:	0

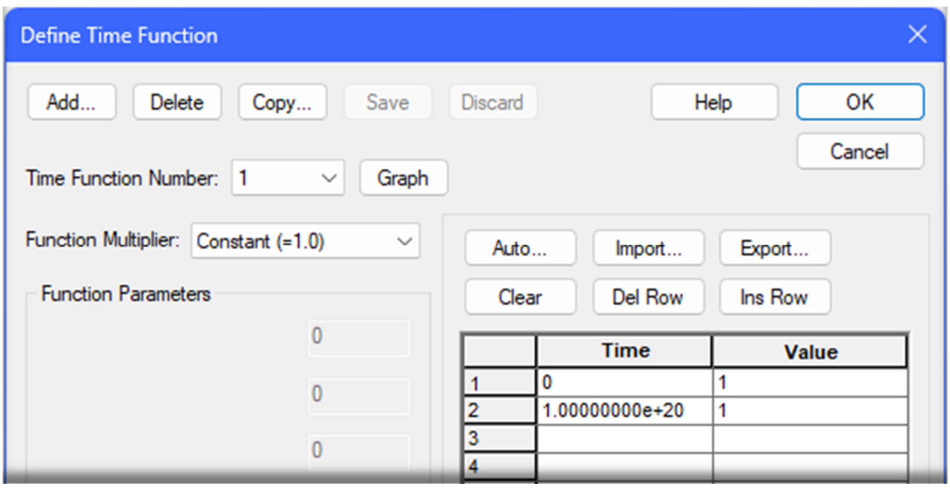
Values in the table:

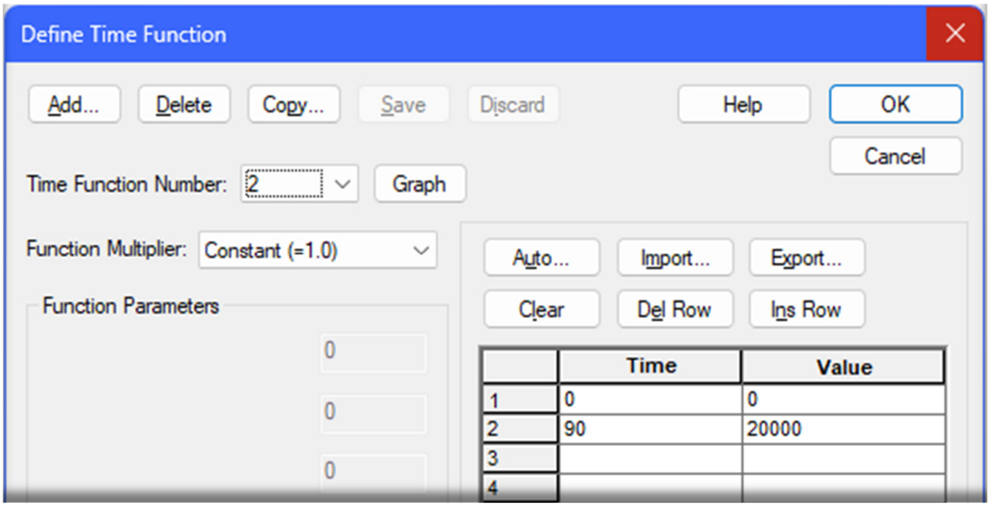
	Time	**Value**
1	0	1
2	90	1

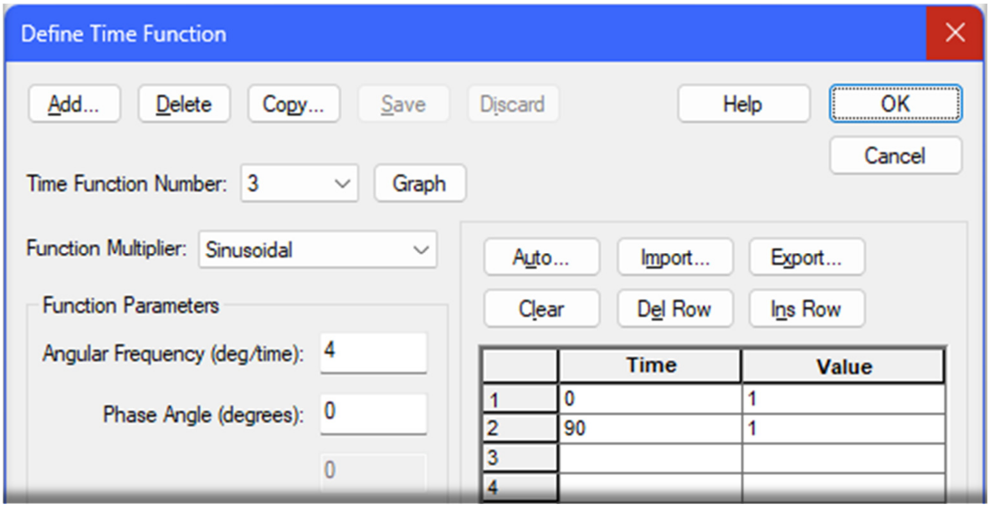
After entering the data for function number “3”, you can press the “Graph” button – the following graph should appear:



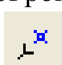
The chart window should then be closed by pressing the “X” button in the upper right corner. After entering all the data, press the “Save” button and then leave the window with the “OK” button. The window views showing the above provided entries are shown in the figures below:





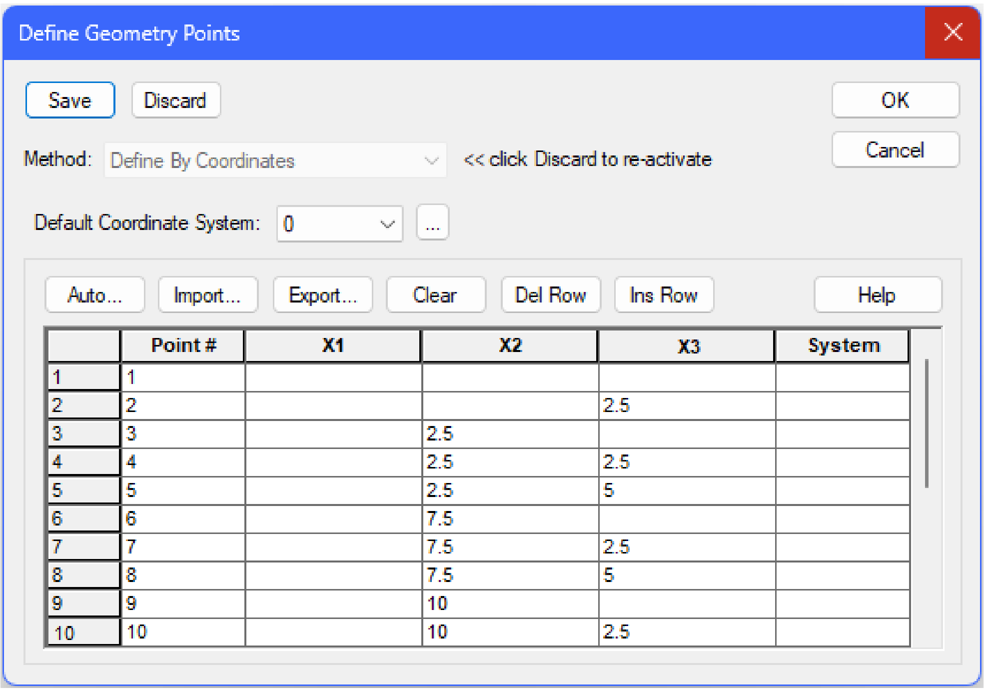


STEP 7. Definition of points

In this step, points describing the frame will be created. The points will be created on the program default YZ plane. To go to the definition of points, select "Geometry → Points → Define…" from the menu or press the button . In the newly opened window, add points according to the table below:

Point #	X1	X2	X3	System
1	0	0	0	0
2	0	0	2.5	0
3	0	2.5	0	0
4	0	2.5	2.5	0
5	0	2.5	5.0	0
6	0	7.5	0	0
7	0	7.5	2.5	0
8	0	7.5	5.0	0
9	0	10	0	0
10	0	10	2.5	0

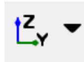
The window view with the entered data is shown in the figure below:

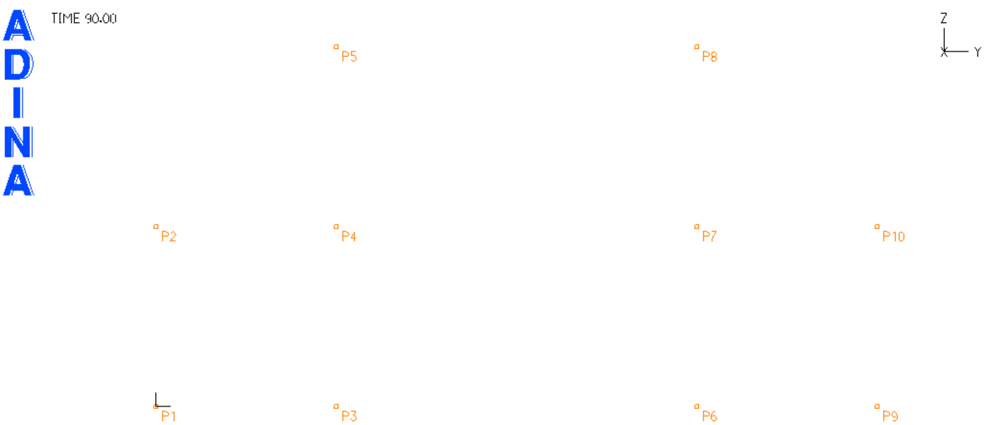


<u>**Note**</u>**:** There is no need to enter zero values in the table. After pressing the “Save” button, the table will be automatically completed with these values.

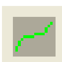
After entering the values from the table, press the “Save” button and then “OK”.

STEP 8. Displaying point ID numbers

To display point identifiers in the YZ plane, press the  button on the toolbar, and then press the  button, respectively. The model should look something like this:



STEP 9. Definition of lines

In this step, having the points already defined, lines describing the frame will be introduced. To add a new line in the model, go to "Geometry → Lines → Define..." or press the button , then in the newly opened window, press the "Add..." button and enter the data in accordance with the table below:

<u>**Note**</u>**:** After entering data from each table, press the "Save" button and then "Add..." to add a new line from the next table until data from the last table is entered.

Line Number:	1
Type:	Straight
Point 1:	1
Point 2:	2

Line Number:	2
Type:	Straight
Point 1:	3
Point 2:	4

Line Number:	3
Type:	Straight
Point 1:	4
Point 2:	5

Line Number:	4
Type:	Straight
Point 1:	6
Point 2:	7

Line Number:	5
Type:	Straight
Point 1:	7
Point 2:	8

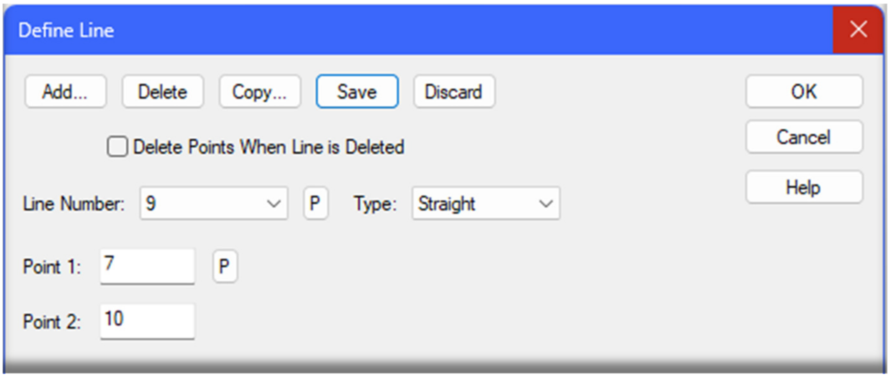
Line Number:	6
Type:	Straight
Point 1:	9
Point 2:	10

Line Number:	7
Type:	Straight
Point 1:	2
Point 2:	4

Line Number:	8
Type:	Straight
Point 1:	5
Point 2:	8

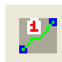
Line Number:	9
Type:	Straight
Point 1:	7
Point 2:	10

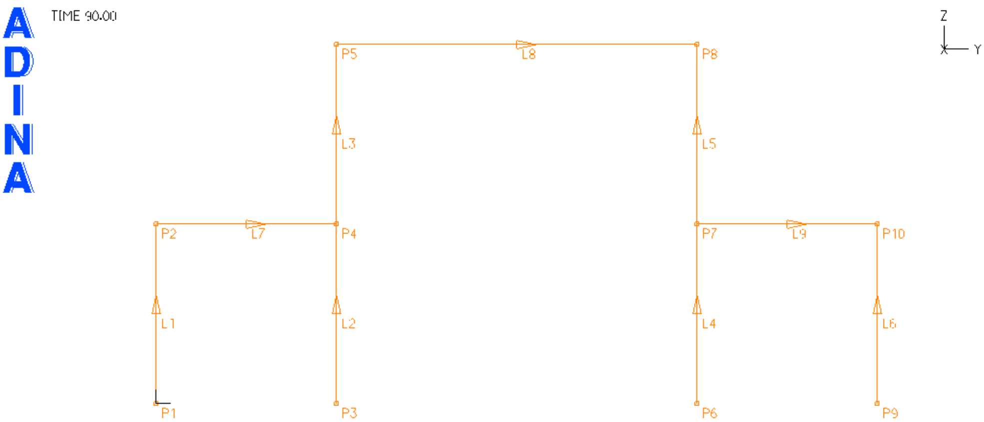
The window with data entered for the line with ID “9” is shown in the figure below:



After entering all the data, press the “Save” button and then leave the window with the “OK” button.

STEP 10. Displaying line ID numbers

In order to display the numbers of lines for their easier identification, click . The model should look like in the figure below:



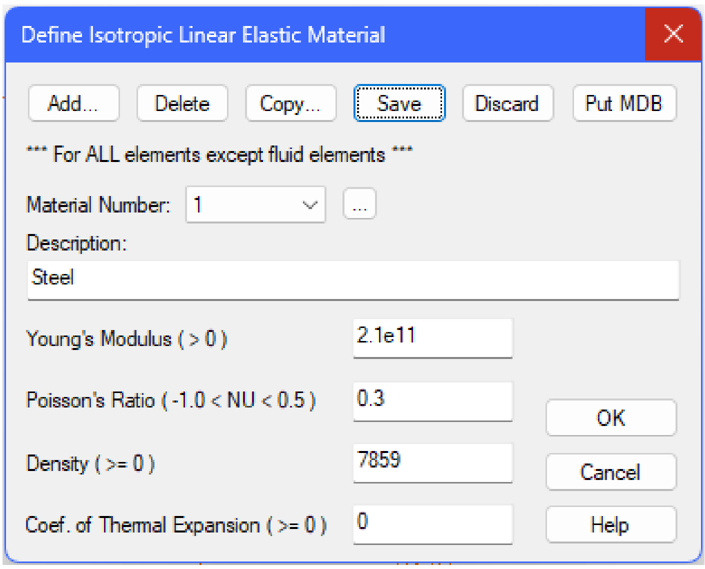
STEP 11. Definition of material constants

In order to go to the definition of material constants, choose the “Model → Materials → Manage Materials” tab, or click . In the newly opened window, find and click the “Isotropic” button in the “Elastic” group of materials. An alternative method of reaching the window with the characteristics of an isotropic material is to find “Model → Materials → Elastic → Isotropic…” in the upper tabs of the program.

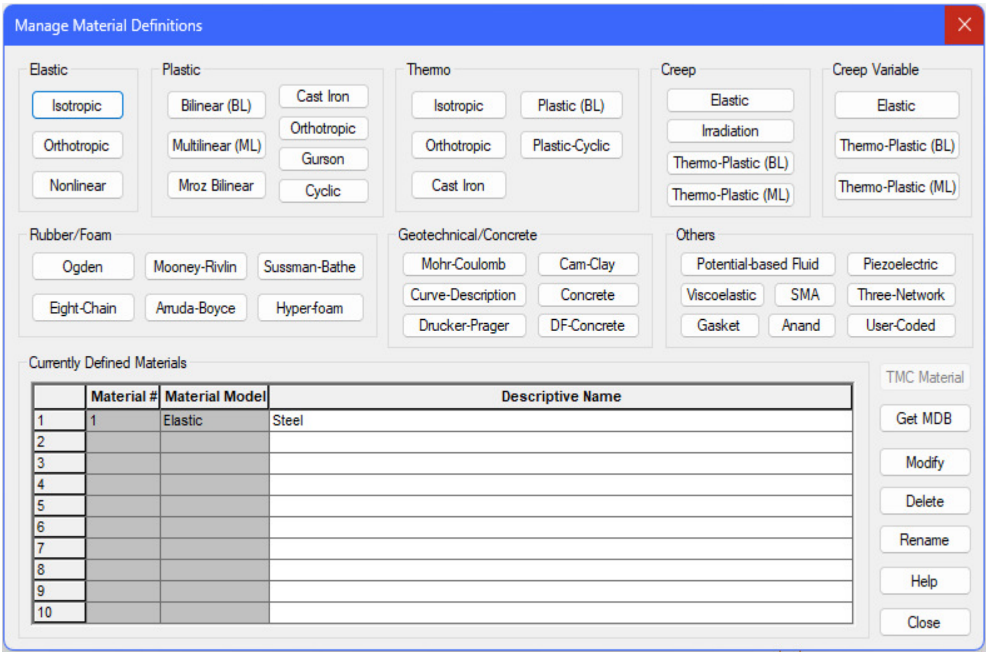
In this window, first click "Add…" in order to add a new material, and then enter the values of material constants as shown in table below:

Material Number:	1
Description:	Steel
Young's Modulus (> 0)	2.1e11
Poisson's Ratio (–1.0 < NU < 0.5)	0.30
Density	7859
Coef. of Thermal Expansion (>= 0)	0

The view of a window with the material constants introduced in the elastic material definition is shown in the figure below:



If the user wants to add material to the program's database, press the "Put MDB" button. After completing the operation, you can exit the window by pressing the "OK" button. Again, as in the previous case, after leaving the window, in the lower part of the "Manage Material Definitions" window, the table should contain the "Steel" material model with an assigned identification number of "1".



<u>Note</u>: After the program restart, in any new model user may retrieve the material added to the ADINA program database. To retrieve a material with all constants defined (previously by the user), go to "Model → Materials → Manage Materials..." or press the button , then in the newly opened window (if the material has been previously added to the program database), press the "Get MDB" button. A new window with a table should open, then select the material named "Steel" in the table and press the "Retrieve" button. No there is no need in defying the material constants every time the new model has been created.

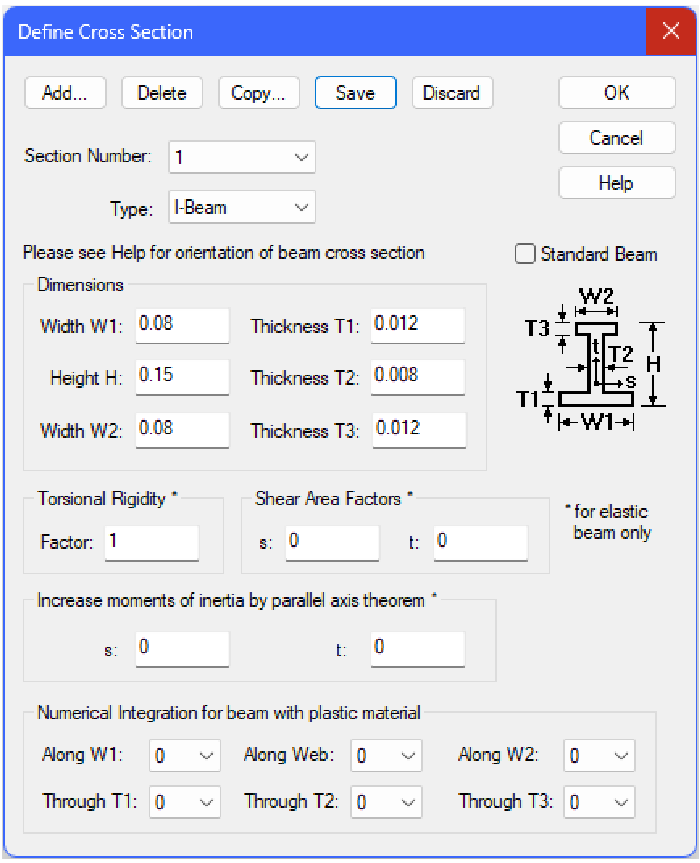
STEP 12. Definition of cross-section(s)

In order to define cross-section shapes, select "Model → Cross-Sections..." from the menu or press the button . After opening of a new window, press the "Add..." button and enter the following data:

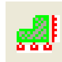
Section Number:	1
Type:	I-Beam
Dimensions	
Width W1:	0.08
Height H:	0.15
Width W2:	0.08
Thickness T1:	0.012
Thickness T2:	0.008
Thickness T3:	0.012

The remaining options are left unchanged. The window view with the entered data is shown in the figure below.

After entering the data, save them by pressing the "Save" button and then you can exit the window by pressing the "OK" button.



STEP 13. Definition of boundary conditions (fixity characteristics)

In this step, it is needed to define following supports: clamped, pinned, and pinned with ability to move against horizontal direction. To define supports and their type, go to “Model → Boundary Conditions → Apply Fixity...” or press the button . When a new window opens, enter the following data:

Apply to:	Points
Default Fixity:	All

The next step is to press the “Define...” button located next to the drop-down list for the “Fixity” option. After opening the next window, press the “Add...” button, then enter the following name of the support “Clamped”, confirm the fixity name with “OK” button and make following changes in the window:

Y-Translation	Checked
Z-Translation	Checked
X-Rotation	Checked
Fluid Potential	Unchecked
Pore Fluid Pressure	Unchecked
Beam Warp	Unchecked
Temperature	Unchecked

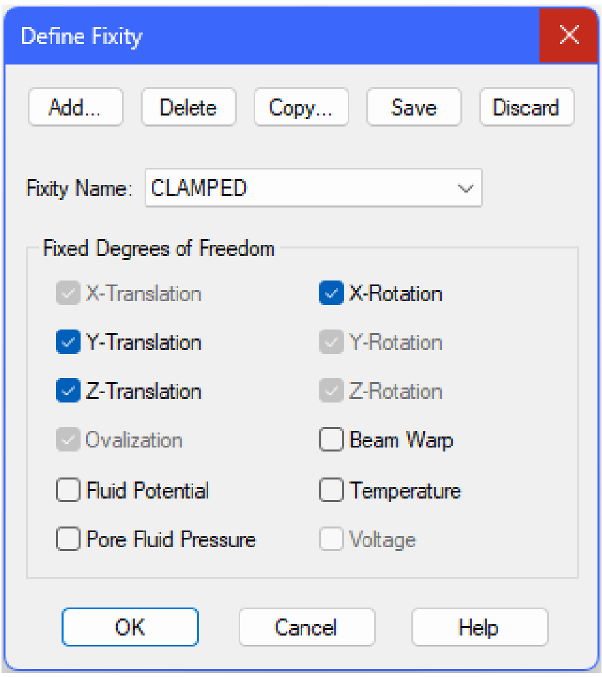
After entering the data, press the “Save” button and then “Add...” again to add the pinned support. When prompted to enter a name, you can enter “Pinned” and then press the “OK” button. Then enter the following data:

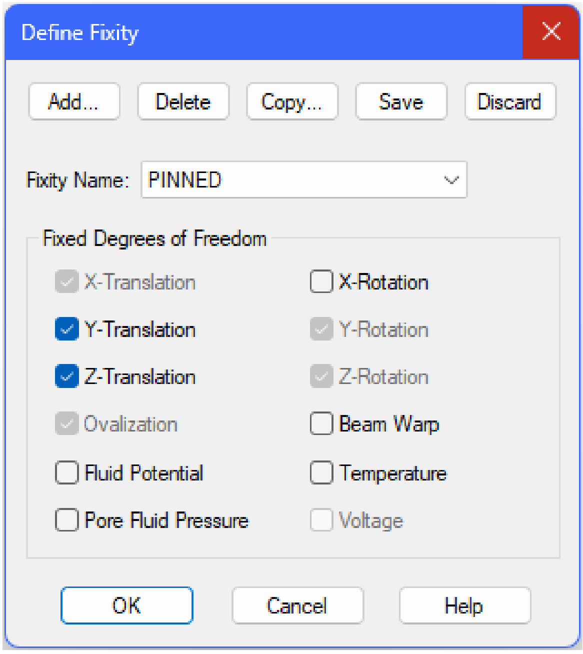
Y-Translation	Checked
Z-Translation	Checked
X-Rotation	Unchecked
Fluid Potential	Unchecked
Pore Fluid Pressure	Unchecked
Beam Warp	Unchecked
Temperature	Unchecked

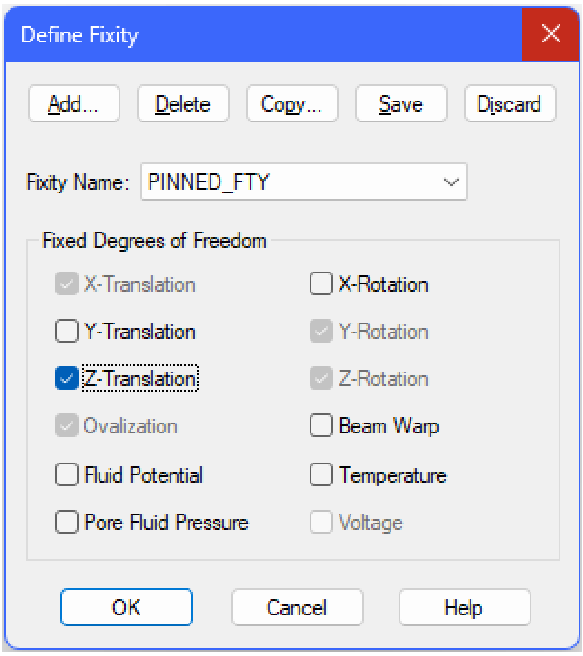
Once again press “Save”, then “Add…” to add the last support – pinned one with the ability to move in the horizontal direction. According to that, name the support as “Pinned_FTY” – the ‘FTY’ suffix is a abbreviated hint which means ‘free translation Y’ – and press “OK”. In the window make check/uncheck following options:

Y-Translation	Unchecked
Z-Translation	Checked
X-Rotation	Unchecked
Fluid Potential	Unchecked
Pore Fluid Pressure	Unchecked
Beam Warp	Unchecked
Temperature	Unchecked

The Define Fixity windows should look similar to that:







As soon as the fixities are defined, please press "OK" button in the "Define Fixity" window, and in the "Apply Fixity" window make following changes:

Fixity:	CLAMPED
Apply to:	Points
	Point {p}
1	1
2	9

Then, press "Apply" button. It applies the clamped support to point ID "1" and "9" in the model. Now let's focus on pinned support. Make the following changes in the "Apply Fixity" window:

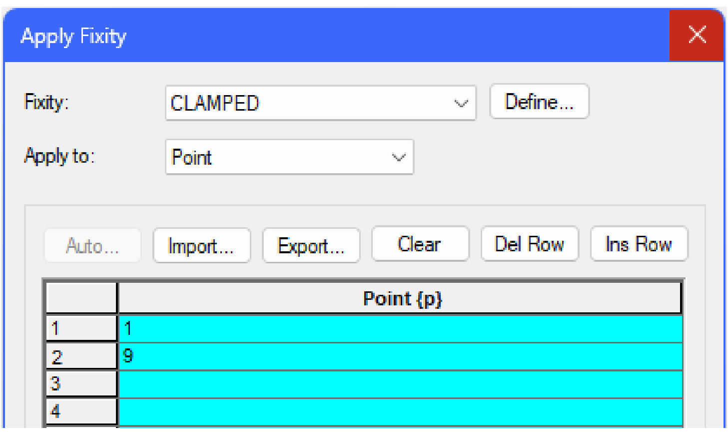
Fixity:	PINNED
Apply to:	Points
	Point {p}
1	3

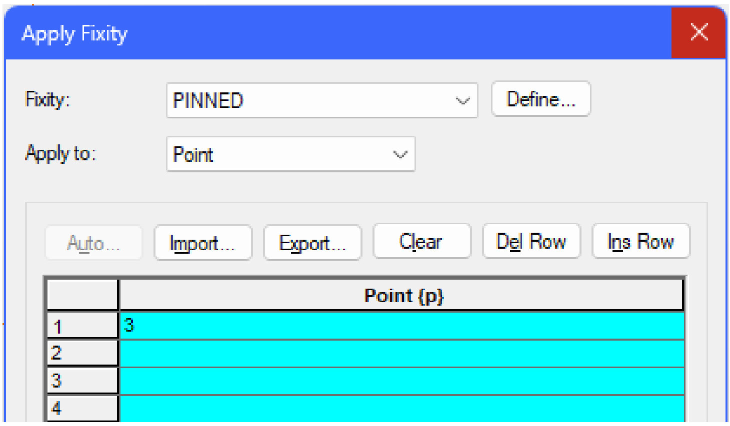
Once again press "Apply" button, and make the following changes:

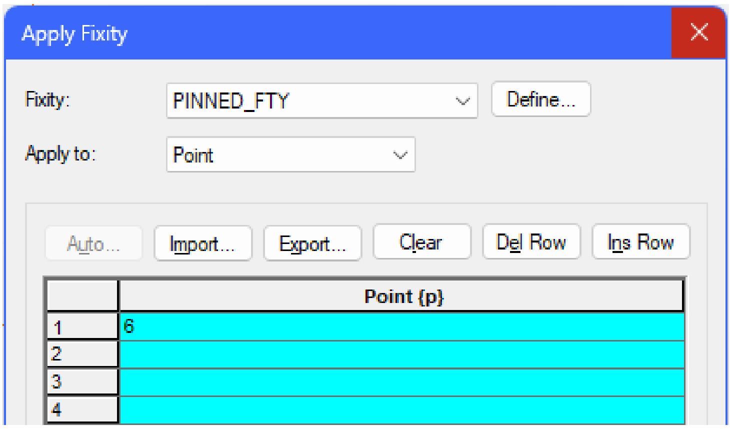
Fixity:	PINNED_FTY
Apply to:	Points
	Point {p}
1	6

After entering the data, press the "Apply" button and then leave the window with the "OK" button.

Windows presenting entered data are shown below:





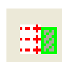


STEP 14. Displaying boundary conditions

To display the defined boundary conditions in the main window of the model, press the  button.

STEP 15. Definition of loads

Three types of loads will be defined in this model. Load by the structure's own weight, linear load and concentrated force.

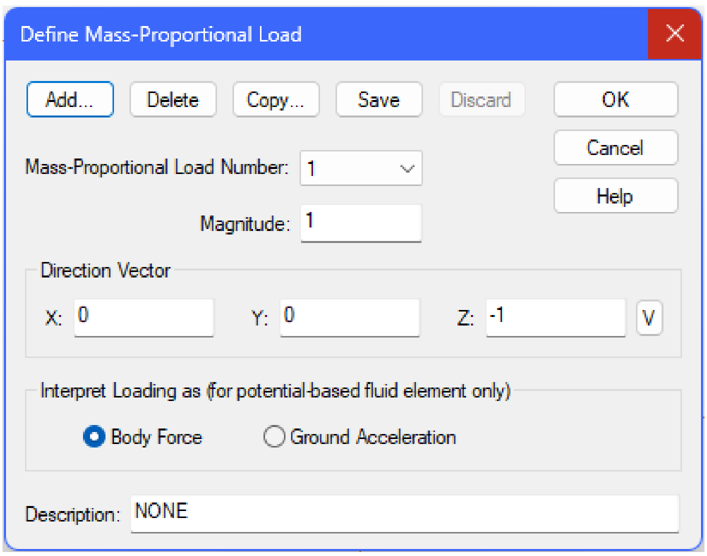
In order to define loads, go to "Model → Loading → Apply..." or press the  button. After opening a new window, the self-weight load of the structure will be declared first, so for the "Load Type:" option, select "Mass Proportional" from the drop-down list, then for the "Load Number:" option, press the "Define..." button. A new window will open, in which one should press "Add..." button to add a new load, then enter the data as shown in the table:

Mass Proportional Load Number:	1
Magnitude:	1
Body #:	1
Direction Vector	
X:	0
Y:	0
Z:	–1
Interpret Loading as (for potential-based fluid element only)	
Body Force	Checked
Description:	None

Note: As mentioned in previous examples entering a number greater than ±1 in the "Direction Vector", that value is still interpreted as 1 with the sign specified by the user. It is possible to enter also "–0.01" in the "Z:" field, which will also be treated as "–1", i.e. a load acting opposite to the "Z" axis.

Note: The load interpretation "Interpret Loading as (for potential based fluid element only)" as the English name suggests only works with potential based fluid elements.

After entering the data in the window, press the "Save" button and then "OK", thus returning to the "Apply Load" window. The window with the entered data is shown in the figure below:



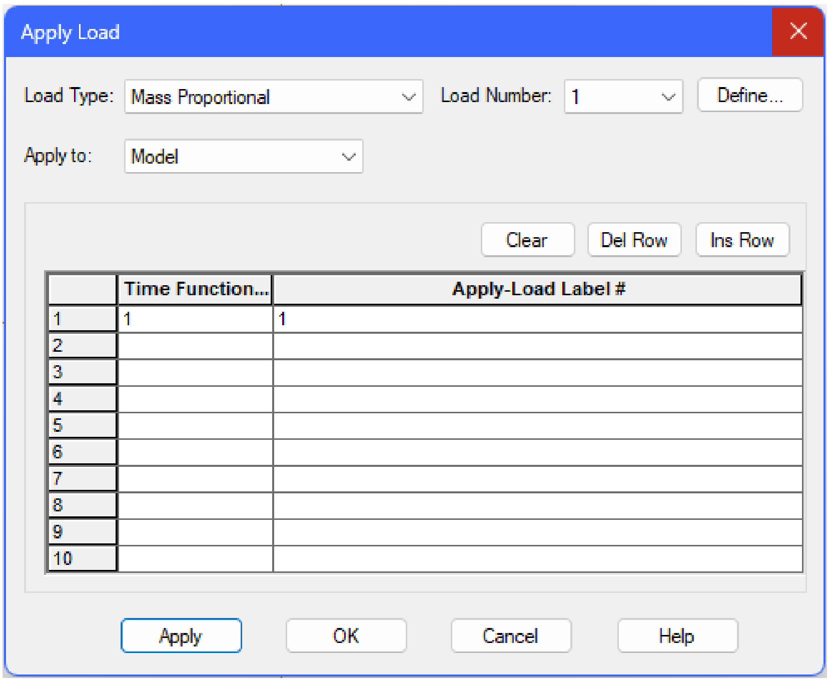
Since the declared load is constant throughout the entire analysis, in the main window – “Apply Load” in the table, this load should be assigned to the load variability function over time with id number “1”. The “Apply Load” window should therefore be completed as follows:

Load Type:	Mass Proportional
Load Number:	1
Apply to:	Model

In the table, the data should look like this:

	Time Function…	**Label #**
1	1	1

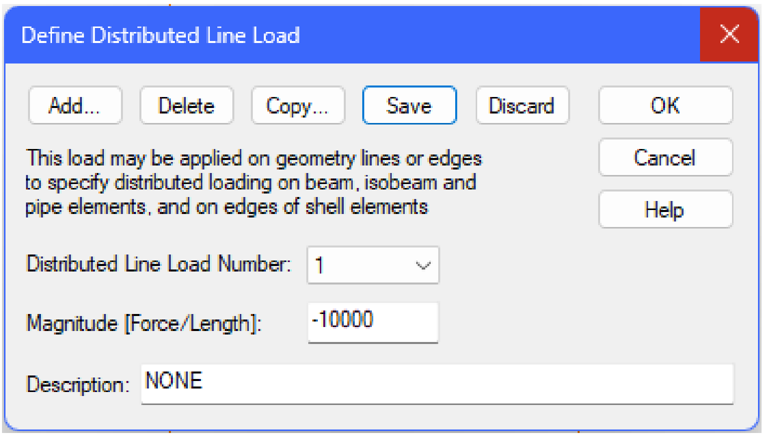
The window view with the entered data is shown in the figure below:



After entering the data, press the “Apply” button, and then, without leaving the “Apply Load” window, select “Distributed Line Load” from the drop-down list next to “Load Type:”, and then press the “Define...” button next to the drop-down list of the “Load Number:” field. In the newly opened window, press the “Add...” button and enter the following data:

Distributed Line Load Number:	1
Magnitude (Force/Length):	–10 000
Description:	None

After entering the data, the window should look like this:



After declaring the load value, press the "Save" button and then "OK" closing the "Define Distributed Line Load" window and return to the "Apply Load" window. In the "Apply Load" window following data should be entered:

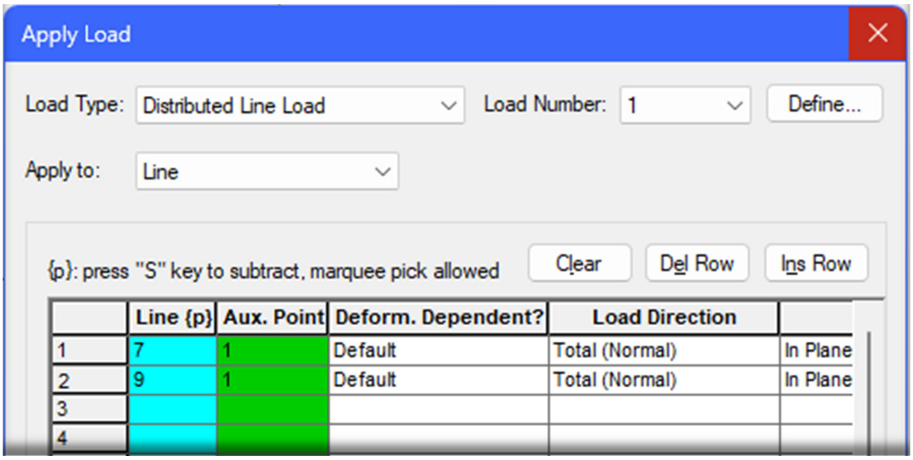
Load Type:	Distributed Line Load
Load Number:	1
Apply to:	Line

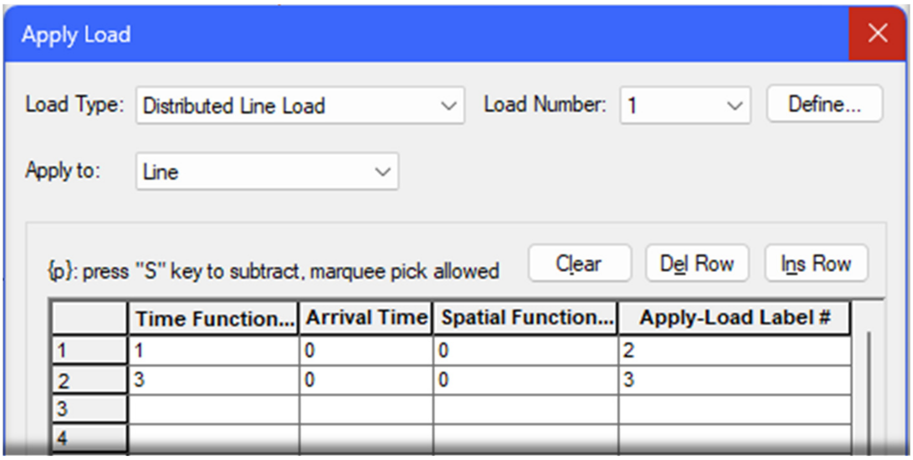
In the table, the data should look like this:

	Line {p}	**Aux. Point**	**…**	**Time Function…**	**…**	**Apply-Load Label #**
1	7	1	Empty	1	Empty	2
2	9	1	Empty	3	Empty	3

After entering the data, press the "Apply" button – this will cause the program to fill in the empty spaces with default values.

The view of the windows along with the entered data is shown in the figures below:



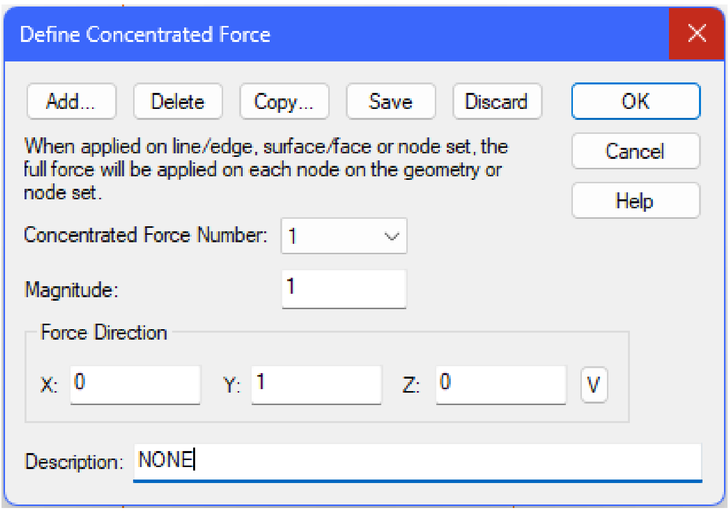


One can notice in the above introductions, the load for line no. "7" was assumed to be constant throughout the entire length of the analysis, while for the line with identification number "9" a sinusoidal load was applied.

Lastly, without closing the "Apply Load" window, change the load type "Load Type:" by selecting the "Force" option from the drop-down list – concentrated force. Declare the force again by pressing the "Define..." button located next to the drop-down list for the "Load Number:" option. When a new window opens, enter the following data:

Concentrated Force Number:	1
Magnitude:	1
Force Direction	
X:	0
Y:	1
Z:	0
Description:	None

After entering the data, the "Define Concentrated Force" window should look as shown in the figure below:



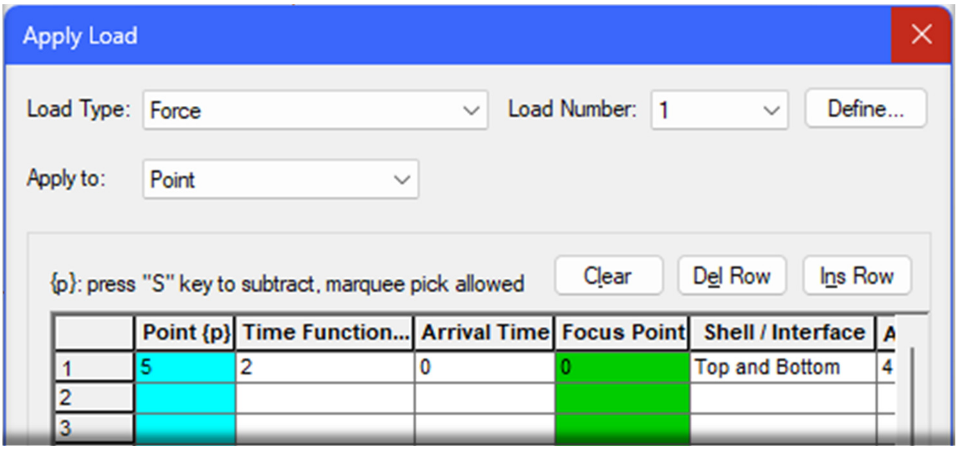
After entering the data, press the "Save" button and then close the window with the "OK" button. Returning to the "Apply Load" window, complete it as follows:

Load Type:	Force
Load Number:	1
Apply to:	Point

In the table, the data should look like this:

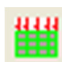
	Point {p}	**Time Function…**	**…**	**Label #**
1	5	2	Empty	4

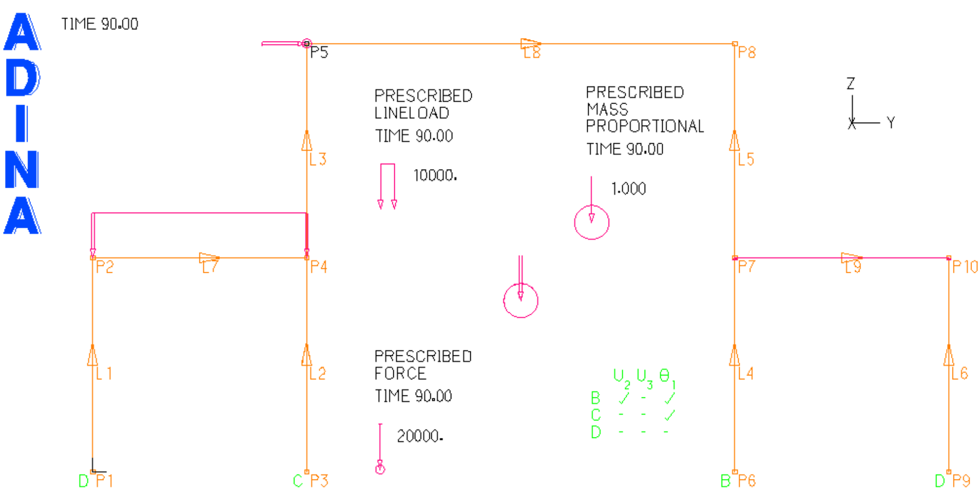
The view of the “Apply Load” window with the entered data is shown in the figure below:



After entering the data, press the “Apply” button – the program will automatically fill the empty table columns with default values. After performing all the above operations, you exit the window by pressing the “OK” button.

STEP 16. Displaying active model loads

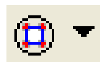
To display the loads acting in the model, press the button . The model should look like this:



Note: The load on the right part of the frame is 0 in the “TIME 90.00” step because the value of the load variation multiplier is “0”. To track the load behavior in individual time steps, use time manipulators – .

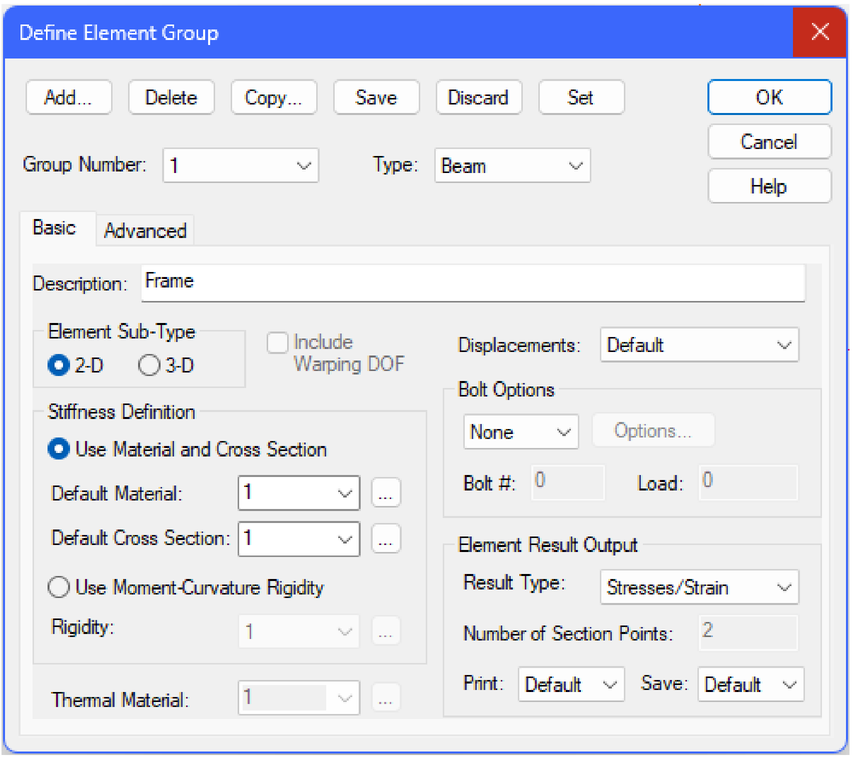
Note: The concentrated force load is equal to 20 kN in the “TIME 90.00” step because the load variation multiplier value is “20000”. As before, manipulators should be used to monitor the load behavior in individual time steps.

STEP 17. Specifying the type of analyzed construction

In this step, a group of elements with assigned steel material and a cross-section in the form of an I-beam will be defined. To go to the definition of a group of elements, select “Meshing → Element Groups...” from the menu or press the button . After opening a new window, press the “Add...” button and then enter the following data:

Group Number:	1
Type:	Beam
“Basic” tab	
Description:	Frame
Element Sub-Type	2-D
Displacements:	Default
Stiffness Definition	
Use Material and Cross Section	Checked
Default Material:	1
Default Cross Section:	1

The remaining data is left unchanged. The window with the entered data is shown in the figure below:



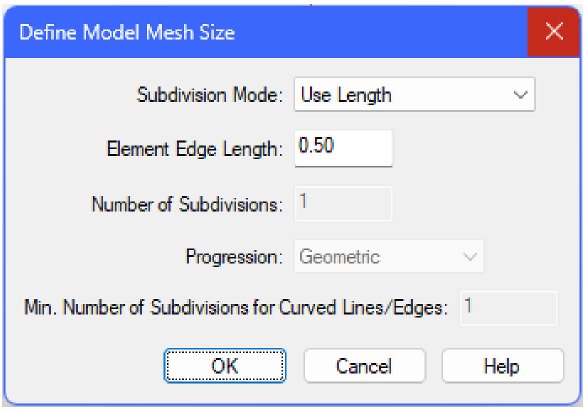
After entering the data, press the "Save" button and then close the window with the "OK" button.

STEP 18. Mesh subdivision

The mesh density will be the same for the entire model, i.e. a mesh with a size of 0.50 m. To define the mesh density in relation to the entire model, go to "Meshing → Mesh Density → Complete Model..." and then enter the following data in the new window:

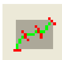
Subdivision Mode:	Use Length
Element Edge Length:	0.50

The window with the entered data is shown in the figure below:



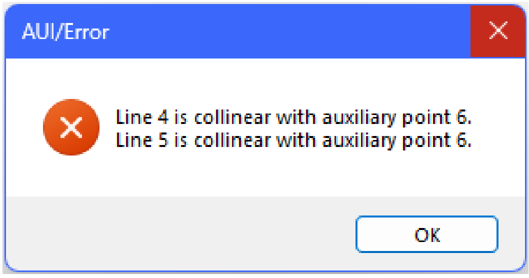
After entering the data, exit the window by pressing the “OK” button.

STEP 19. Definition of finite elements

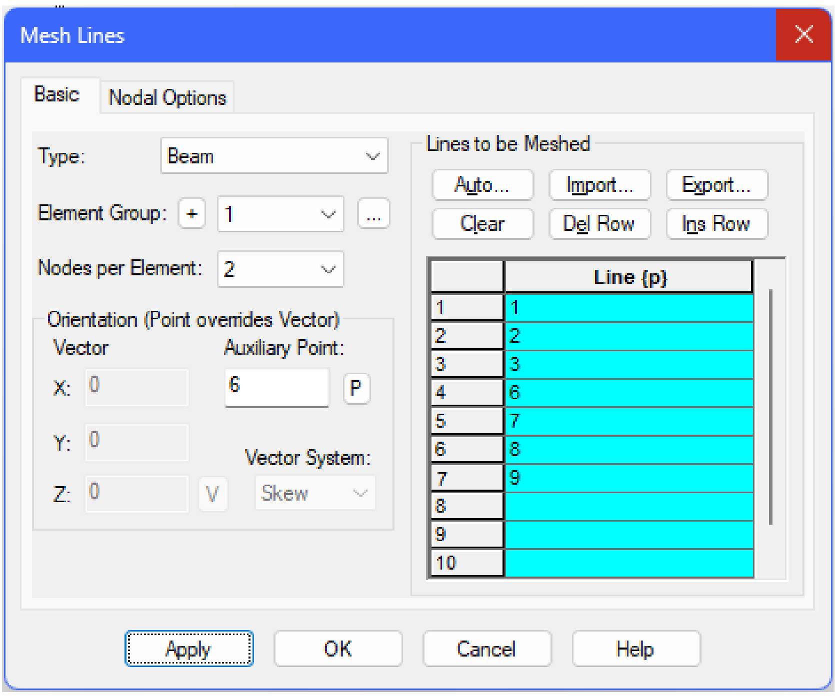
To define finite elements in this example, go to “Meshing → Create Mesh → Line...” or press the button , then enter the following options in the window:

Type:	Beam
Element Group:	1
Nodes per Element:	2
Orientation (Point overrides Vector)	
X:	0
Y:	0
Z:	0
Auxiliary Point:	6
Vector System:	Skew
Table “Lines to be Meshed”	
	Line {p}
1	1
2	2
3	3
4	6
5	7
6	8
7	9

Note: Line IDs “4” and “5” have been omitted due to the fact, that they would have been collinear with the chosen auxiliary point. The introduction of finite elements must have been divided into two stages, because if a point number “6” is used as an “Auxiliary Point” and lines numbered “4” and “5” are entered, the program will return an error that the line coincides with the point. Moreover, in the program it is not possible to indicate the direction of the local cross-section coordinate system. This option is particularly important when cross-sections other than bisymmetric are used. The content of the error message is shown in the figure below:



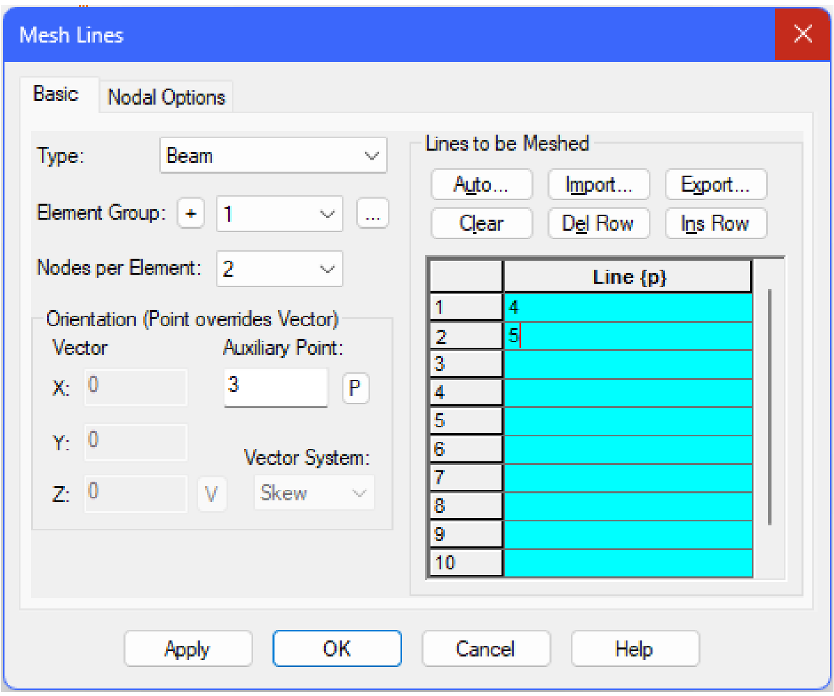
The window with the properly entered data is presented in the figure below:



After entering the data, press the “Apply” button and then, without leaving the window, enter the following data:

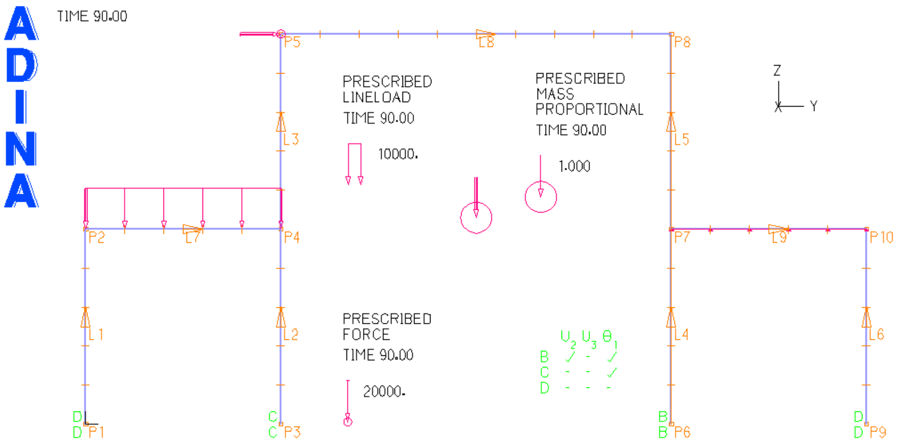
Type:	Beam
Element Group:	1
Nodes per Element:	2
Orientation (Point overrides Vector)	
X:	0
Y:	0
Z:	0
Auxiliary Point:	3
Vector System:	Skew
Table “Lines to be Meshed”	
	Line {p}
1	4
2	5

Window view with entered data:



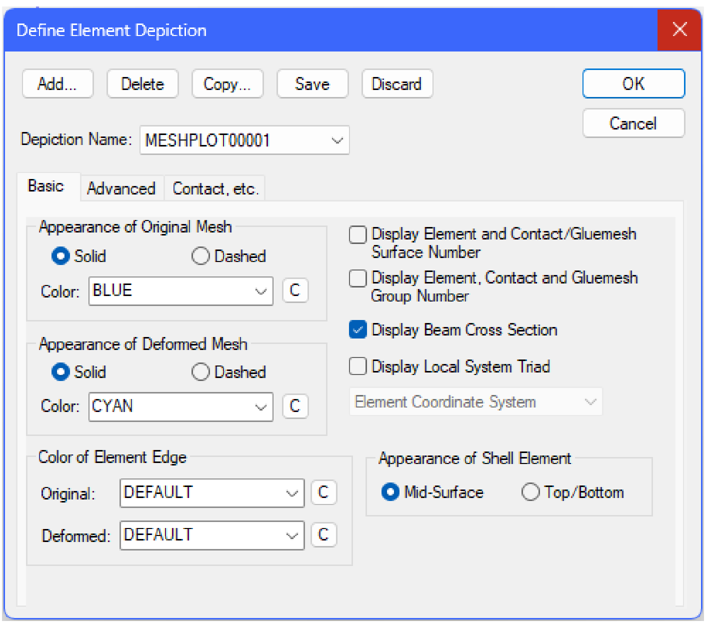
After entering the data, press the “Apply” button and then leave the window with the “OK” button.

After entering all the data, the model should look something like this:

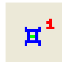
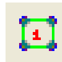


STEP 20. Definition of perfectly rigid connections

To define perfectly rigid connections and their graphical representation in the model, you must first enable the display of the cross-section. To do this, go to “Display → Geometry / Mesh Plot → Modify...” or press the button . Then, in the newly opened window, locate the “Element Depiction...” button in the “Mesh Attributes” group and press it. When the new window opens, select the “Display Beam Cross Section” checkbox. The window view with the introduced change is shown in the figure below:

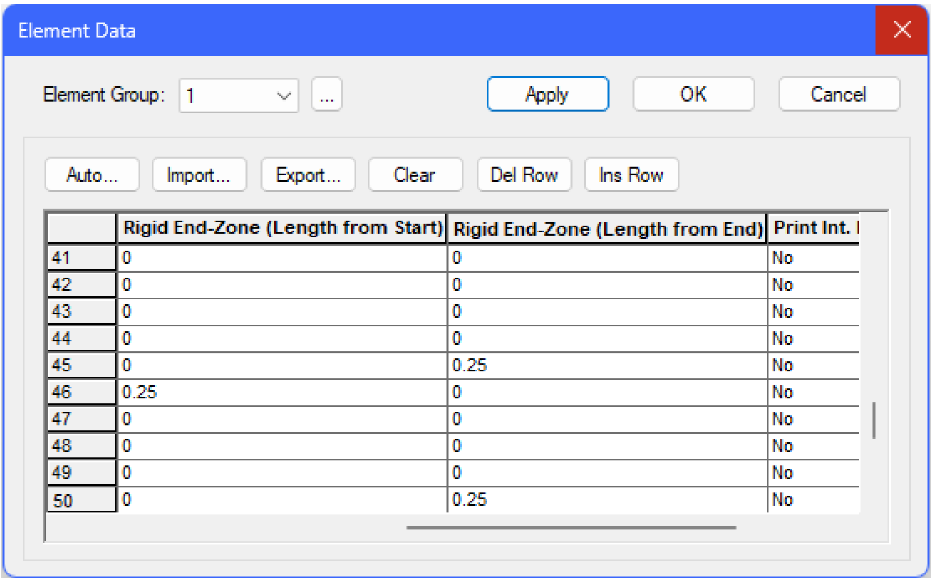


After making changes, press the "Save" button, then leave the window with the "OK" button. Then, in the "Modify Mesh Plot" window, press "Apply" and also leave the window with the "OK" button.

The next step is to enable the display of finite element identification numbers. First, you can disable the display of point ID numbers by turning off the button , then activate the display of finite element ID numbers by pressing the button . With the finite element ID numbers enabled, you should proceed to the definition of perfectly rigid connections. To do this, select "Meshing → Elements → Element Data..." from the menu and after opening a new window, enter the following data:

	Beam Element #	**…**	**Rigid End-Zone (Length from Start)**	**Rigid End-Zone (Length from End)**	**…**
…	…	…	…	…	…
5	5	…	0	0.25	…
…	…	…	…	…	…
10	10	…	0	0.25	…
11	11	…	0.25	0	…
…	…	…	…	…	…
15	15	…	0	0.25	…
…	…	…	…	…	…
20	20	…	0	0.25	…
21	21	…	0.25	0	…
…	…	…	…	…	…
25	25	…	0	0.25	…
26	26	…	0.25	0	…
…	…	…	…	…	…
35	35	…	…	0.25	…
36	36	…	0.25	0	…
…	…	…	…	…	…
40	50	…	0	0.25	…
…	…	…	…	…	…
45	55	…	0	0.25	…
46	56	…	0.25	0	…
…	…	…	…	…	…
50	60	…	0	0.25	…

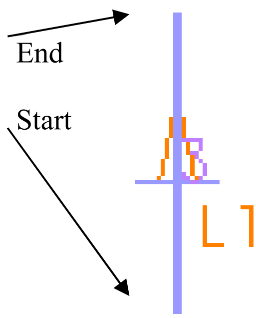
The window with example data is shown in the figure below:



	Rigid End-Zone (Length from Start)	Rigid End-Zone (Length from End)	Print Int. I
41	0	0	No
42	0	0	No
43	0	0	No
44	0	0	No
45	0	0.25	No
46	0.25	0	No
47	0	0	No
48	0	0	No
49	0	0	No
50	0	0.25	No

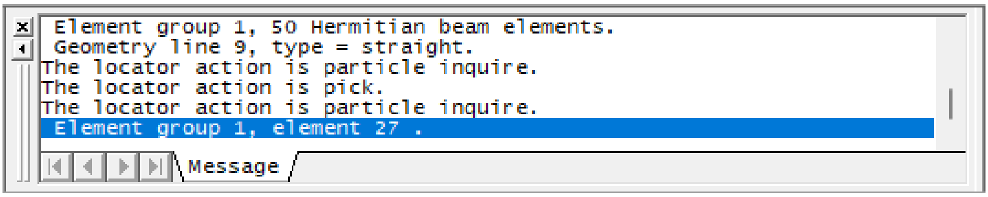
After entering all the data in the table, press the “Apply” button and then “OK”.

<u>Note</u>: The “Rigid End-Zone (Length from Start)” and “Rigid End Zone (Length from End)” options refers to the lines directions. In the case of the “Length from Start” option, the length of a perfectly rigid element is measured from the beginning of a given finite element in the same directions as an arrow indicating the direction of the line, while for “Length from End” works opposite – the distance is measured from the line end in the opposite direction of line direction arrow.

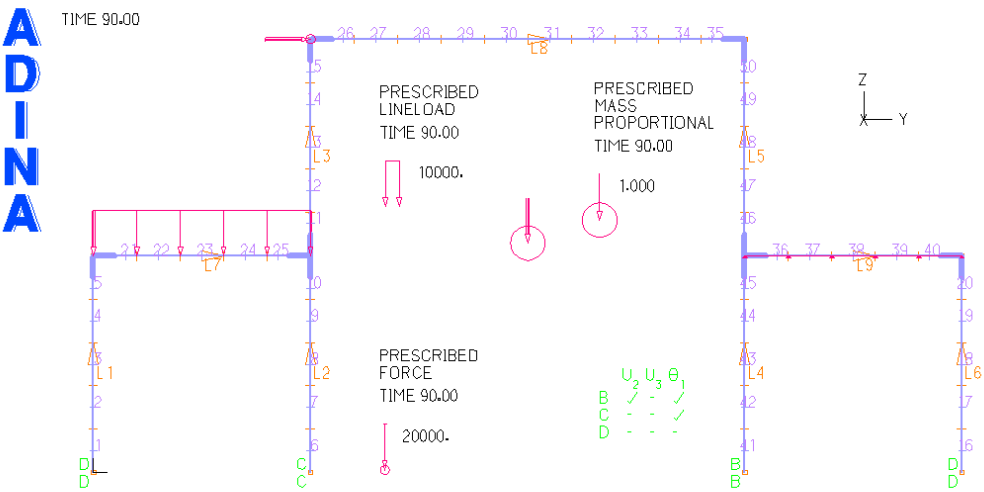


<u>Note</u>: The “Beam Elements” ID numbers do not have to match the table row numbers. Also, the numbers of finite elements may differ between those given in the example and those that the user sees in his model. Particular attention should be paid to this when introducing perfectly rigid elements. In case the user have different element numbers at the connections, please use them, instead of that given in the table.

Note: User can check the number of a given finite element by selecting the button  and then pressing the given finite element. The element ID number and element group are displayed in the "Message Window" at the bottom of the screen:



After returning to the main modeling window, press the button  causing the display of the current model to be refreshed. The current model should look like this:



One can notice in the drawing above, perfectly rigid elements in the frame nodes are marked with bold lines. If the lines meet at the nodes (as in the drawing above), it means that these elements have been defined as intended.

STEP 21. Saving existing model to a file

Each model should have been saved to a file between few steps taken in order to not lose the data. According to that select "File → Save as..." from the menu. When a new window opens, indicate the location of the saved file and its name.

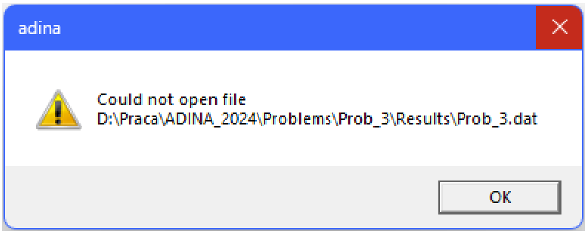
Note: Do not uses spaces in the file names, because it leads to an error! The space can be replaced with the underline character __.

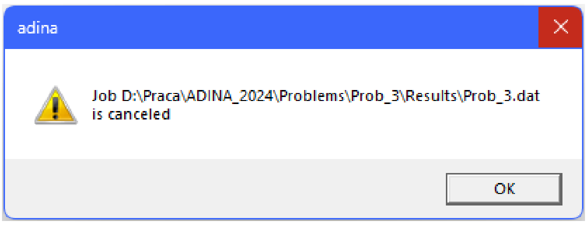
STEP 22. Starting calculations

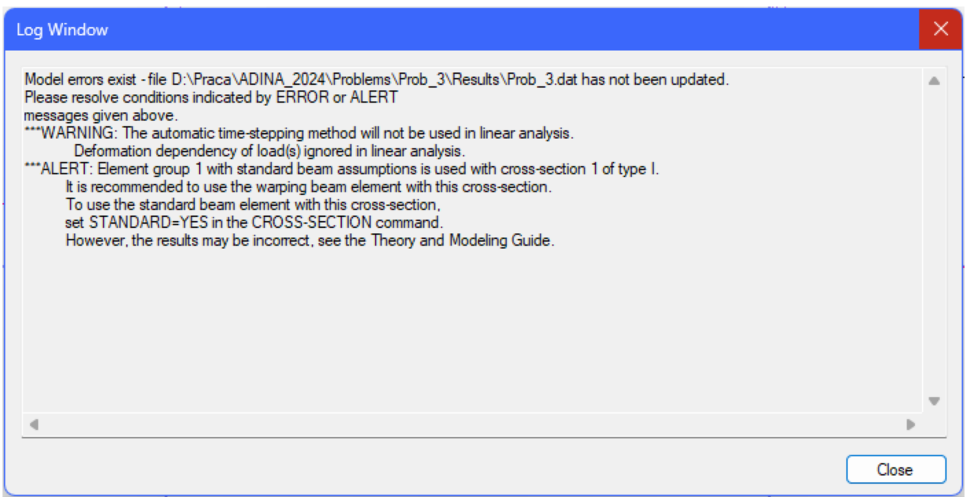
In order to start calculations, choose “Solution → Data File/Run” from the upper menu tabs, or choose  from the toolbars. Subsequently, input the path for saving the resultant file. However, before the user clicks the “Save” button, they should make sure that the “Run Solution” and “Automatic” function near the “Maximum Memory for Solution” options are selected in the window in the “Adina Structures Solution” group of options. After starting the calculations and recalculating the model, close all three dialogue windows which appeared during the calculations.

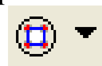
<u>Note</u>: Depending on the complexity of the model and the number and type of finite elements used, model calculations may take from a few seconds to even several hours.

<u>Note</u>: In new versions of ADINA software start of calculations may fail due to the following messages:



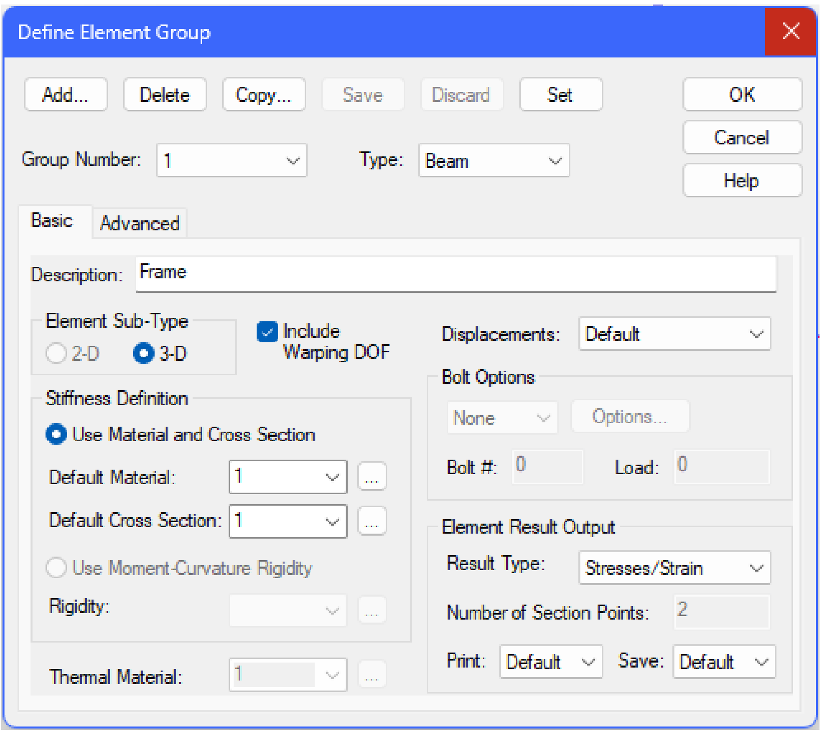




It means that the "warping DOF" – warping degree of freedom should have been introduced to the beam model, cause existing one may lead to incorrect results. According to that, one should go back to the definition of element groups. Please select "Meshing → Element Groups..." from the menu or press the button . After opening a new window, check if the "Element group:" drop-down list has number "1", if yes change the following data:

Group Number:	1
"Basic" tab	
Element Sub-Type	3-D
Include Warping DOF	Checked

The window with applied options is shown below:



The remaining data left unchanged, press "Save" and "OK" button to leave the window. Now, once again start the calculations by choosing "Solution → Data File/Run" from the upper menu tabs, or choose  from the toolbars.

STEP 23. Post-processing module (results)

In order to go to the results module, change “ADINA Structures” to “Post-Processing” in the drop-down list of the “Module” toolbar.

When the user is prompted that the changes in the drawing have not been saved, it is recommended to save the model by going to “File → Save” or using the button .

STEP 24. Opening the resultant file

In order to open the resultant file, choose  from the toolbar, or choose “File → Open” from the upper menu tabs.

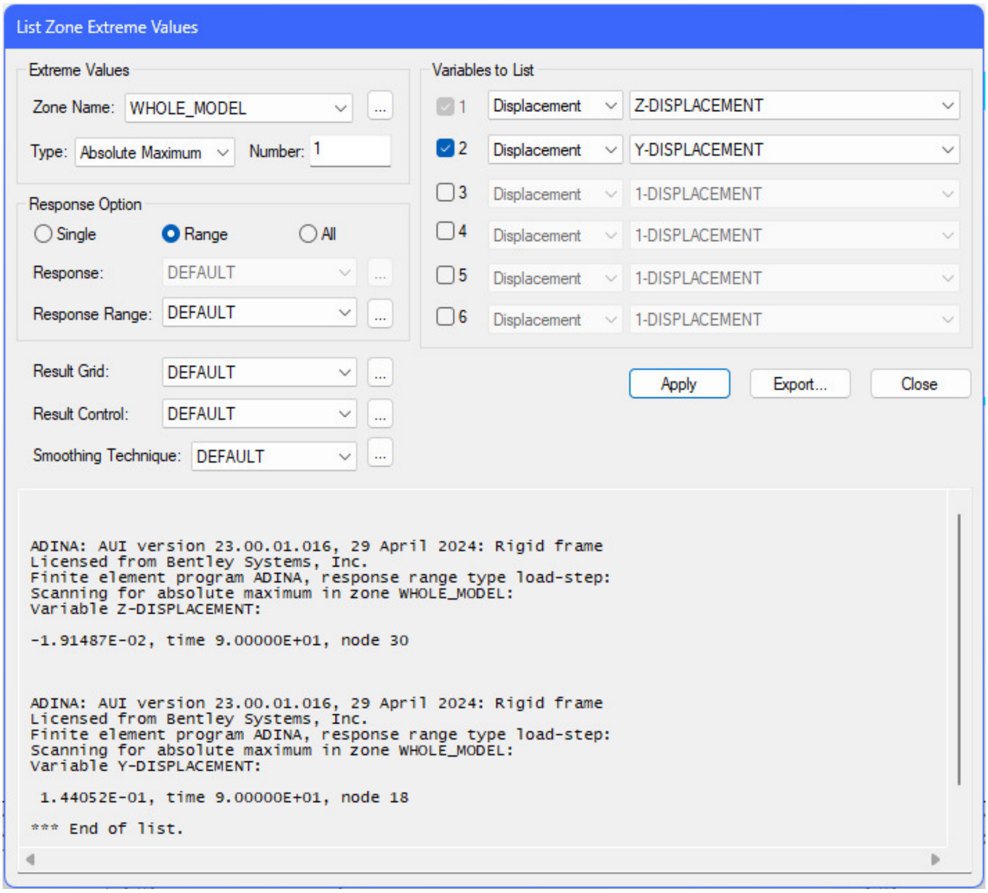
STEP 25. Determination of the maximum vertical and horizontal displacement and the maximum normal force, shear force and bending moment

To determine the maximum horizontal and vertical displacement in the resulting model, go to “List → Extreme Values → Zone...”, then select the following data in the newly opened window:

Zone Name:	WHOLE_MODEL
Type:	Absolute Maximum
Number:	1
Response Option	
Range:	Checked
Response Range:	DEFAULT
Result Grid:	DEFAULT
Response Control:	DEFAULT
Smoothing Technique:	DEFAULT

Variables to List		
1	Displacement	Z-DISPLACEMENT
2	Displacement	Y-DISPLACEMENT

After entering the data, press the “Apply” button. The window view with the entered data is shown in the figure below:



As can be seen in the figures above, the maximum vertical and horizontal displacements are:

Displacement Z	**Time**
–1.91487E-02	9.00000E+01
Displacement Y	**Time**
1.44052E-01	9.00000E+01

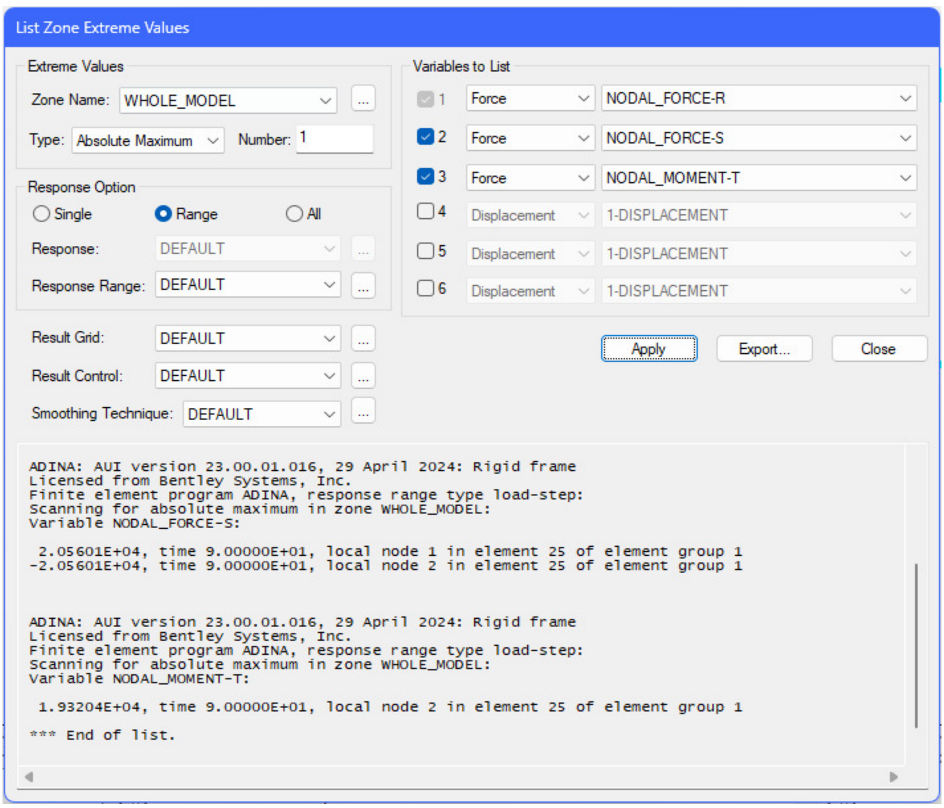
User can also display the maximum normal and shear forces and moments at finite element nodes. To do this, in the same window (List Zone Extreme Values), change the values to be displayed:

Variables to List		
1	Force	NODAL_FORCE-R
2	Force	NODAL_FORCE-S
3	Force	NODAL_MOMENT-T

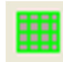
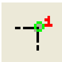
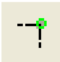
After entering the data, press the "Apply" button. The maximum values should be as follows:

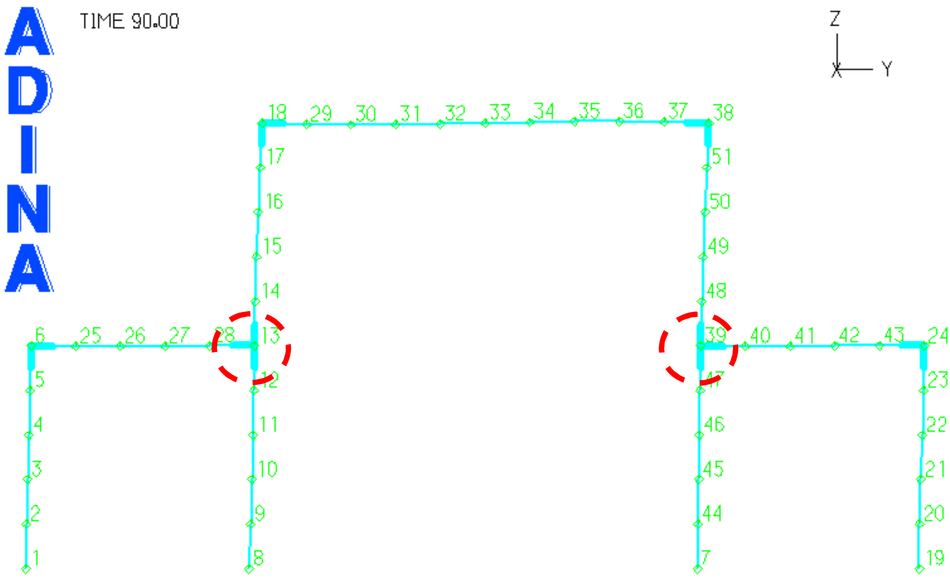
Normal force	**Time**	**Node**	**Element**
2.02108E+04	9.00000E+01	1	6
–2.02108E+04	9.00000E+01	2	6
Shear force	**Time**	**Node**	**Element**
2.05601+04	9.00000E+01	1	25
–2.05601E+04	9.00000E+01	2	25
Moment	**Time**	**Node**	**Element**
1.93204E+04	9.00000E+01	2	25

The window view with the entered data is presented in the figure below. After completing the operation, you can close the window with the "Close" button.



STEP 26. Preparing diagram of the global displacement of the connected nodes of the main aisle and the side ones

First, user need to reset the model view. To do this, press the  and  button, respectively. Then, turn on the display of finite element node numbers by pressing the button , you also turn on the graphical display of these nodes by pressing the button . Currently the model should look like this:



The nodes in relation to which the displacement graph will be drawn are marked with dashed line circles. In the author's case, the node number of the left aisle is "13" and the right one is "33".

Note: Node numbers may differ between the user's model and the author's model.

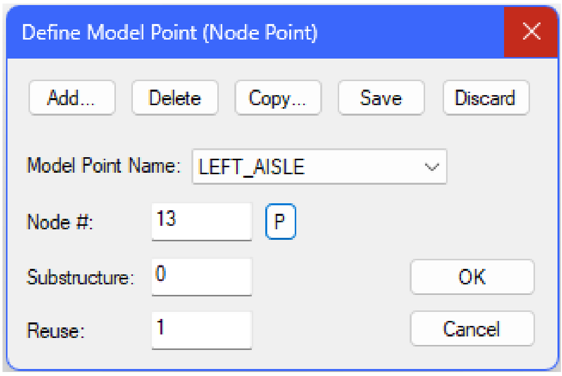
The next stage is to define the points of interest by the user that will be involved in the presentation of the diagram. To do this, go to "Definitions → Model Point → Node...". After opening a new window, press the "Add..." button and enter the title for the first point, e.g. "Left_aisle", and then use the "P" button to select the connecting node of the left aisle with the main one from the model. The data in the window should look like this:

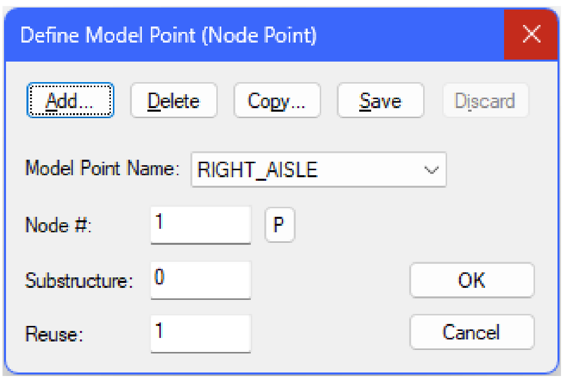
Model Point Name:	LEFT_AISLE
Node #:	13
Substructure:	0
Reuse:	1

After entering the data, press the “Save” button and then “Add...” again. Enter the title for the “Right_aisle” node again, press the “P” button in the “Define Model Point (Node Point)” window and select the connecting node of the right aisle with the main one. The data in the window should look like this:

Model Point Name:	RIGHT_AISLE
Node #:	39
Substructure:	0
Reuse:	1

The view of the windows along with the entered data is shown in the drawings below:



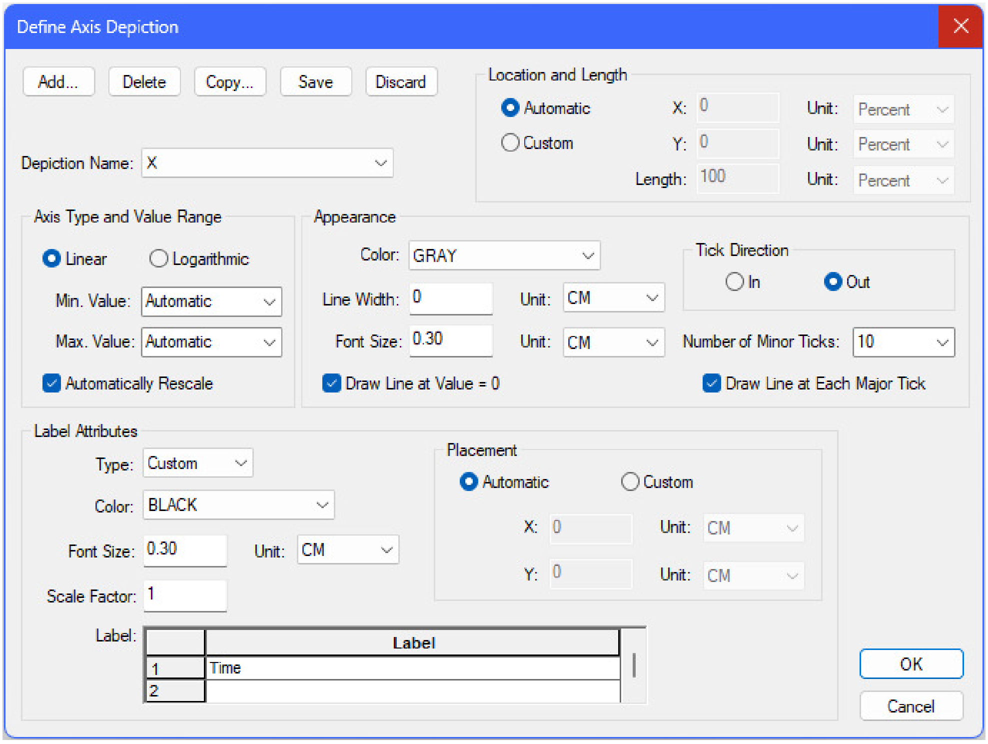


After entering the data, press the “Save” button and then leave the window with the “OK” button. One can also clear the results window by pressing the button .

To create a graph, go to “Graph → Response Curve (Model Point)”, then when a new window opens, press the “...” button next to the “X-Axis” option. In the newly opened window, press the “Add...” button and enter “X” as the title. Then change the following options in the window:

…	
Label Attributes	
Type:	Custom
Color:	BLACK
Font Size:	0.30
Unit:	CM
Scale Factor:	1
Table:	
	Label:
1	time
Appearance	
Color:	GRAY
Line Width:	0
Unit:	CM
Font Size:	0.30
Unit:	CM
Draw Line at Value = 0	Checked
Tick Direction	
Out	Checked
Number of Minor Ticks:	10
Draw Line at Each Major Tick	Checked

After entering the data, press the “Save” button and close the window with the “OK” button. The window view with the entered data is shown in the figure below:

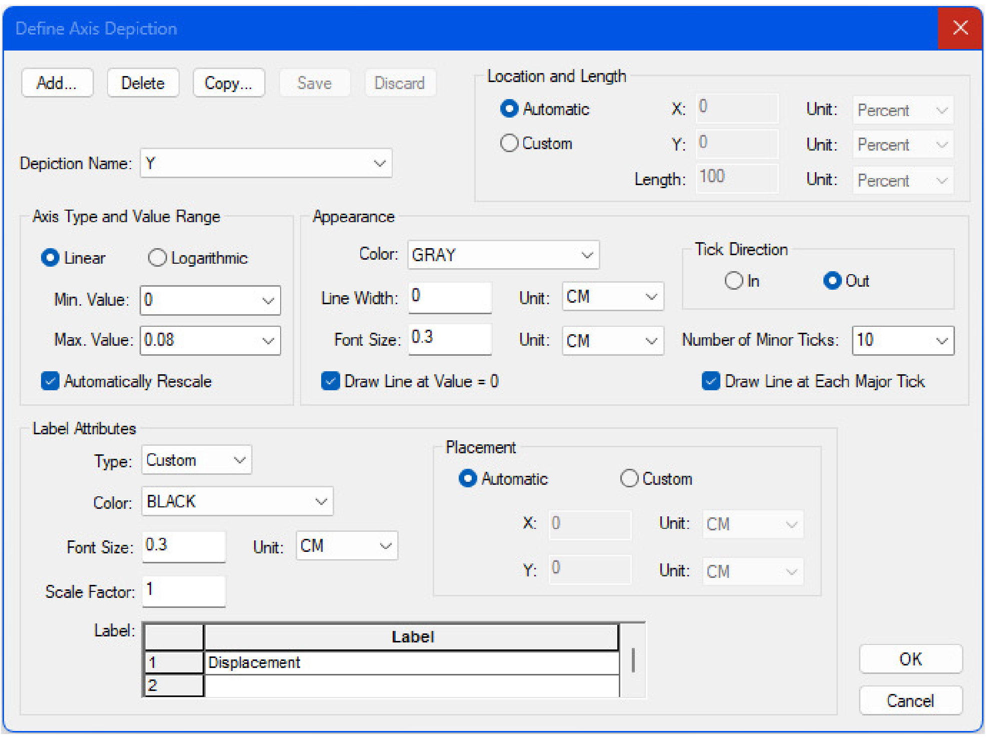


After returning to the “Display Response Curve (Model Point)” window, press the “...” button next to the “Y-Axis” option in “Graph Attributes” group. Once the window opens, press the “Add...” button and enter “Y” as the axis title. Then enter the following data in the window:

...	
Axis Type and Value Range	
Linear	Checked
Min. Value:	0
Max. Value:	0.08
Label Attributes	
Type:	Custom
Color:	BLACK
Font Size:	0.30
Unit:	CM
Scale Factor:	1

Table:	
	Label:
1	Displacement
Appearance	
Color:	GRAY
Line Width:	0
Unit:	CM
Font Size:	0.30
Unit:	CM
Draw Line at Value = 0	Checked
Tick Direction	
Out	Checked
Number of Minor Ticks:	10
Draw Line at Each Major Tick	Checked

Once again, after entering the data, press the "Save" button and then leave the window with the "OK" button. The window view with the entered data is shown in the figure below:



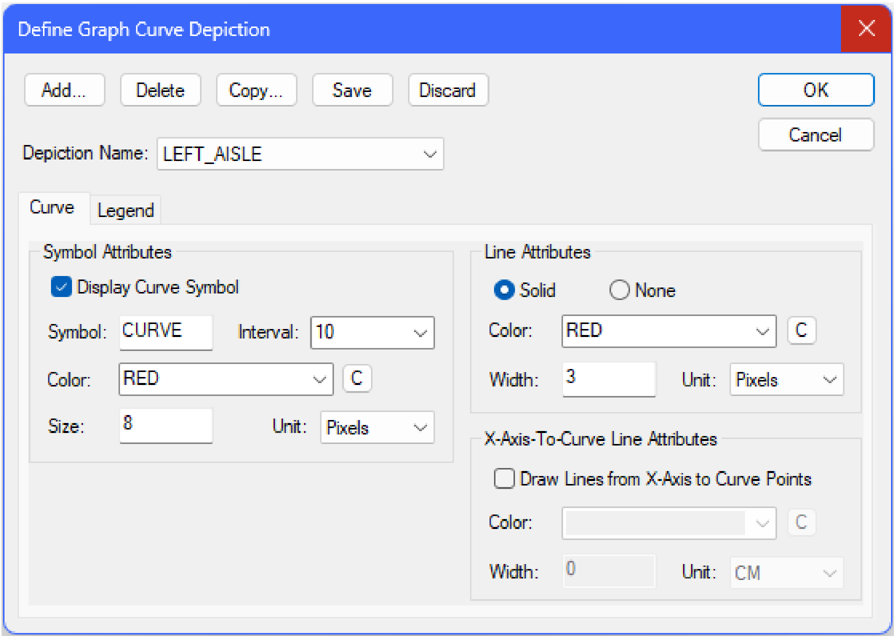
After returning to the “Display Response Curve (Model Point)” window, press the “...” button, this time next to the “Curve Depiction” option in “Graph Attributes” group. When the window opens, press the “Add...” button and enter “Left_aisle” as the line title. Then enter the following data in the window:

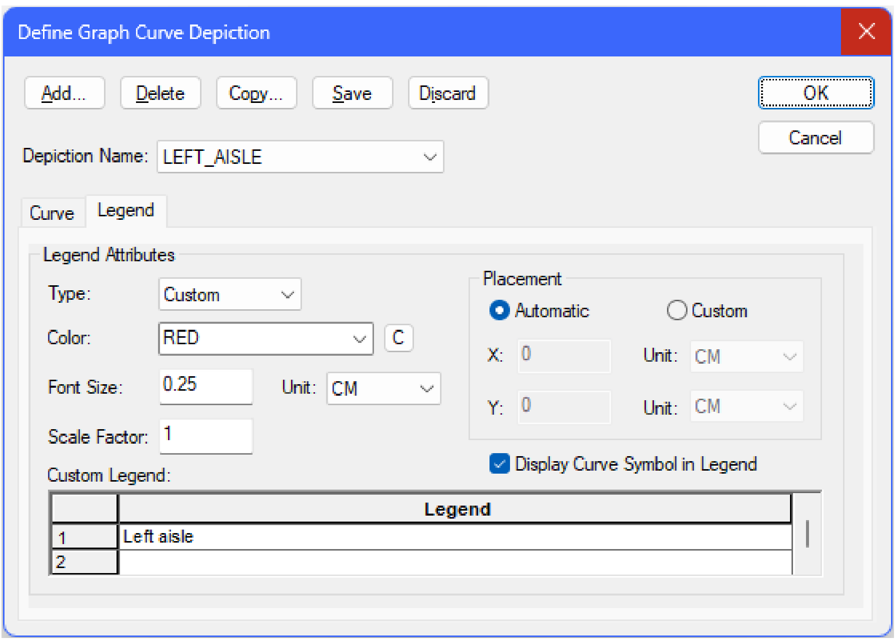
“Curve” tab	
Symbol Attributes	
Display Curve Symbol	Checked
Symbol:	CURVE
Interval:	10
Color:	RED
Size:	8
Unit:	Pixels
Line Attributes	
Solid	Checked
Color:	RED
Width:	3
Unit:	Pixels
X-Axis-To-Curve Line Attributes	
Draw Lines from X-Axis to Curve Points	Unchecked
“Legend” tab	
Legend Attributes	
Type:	Custom
Color:	RED
Font Size:	0.25
Unit:	CM
Scale Factor:	1
Placement	
Automatic	Checked
Display Curve Symbol in Legend	Checked
Custom Legend:	
	Legend:
1	Left aisle

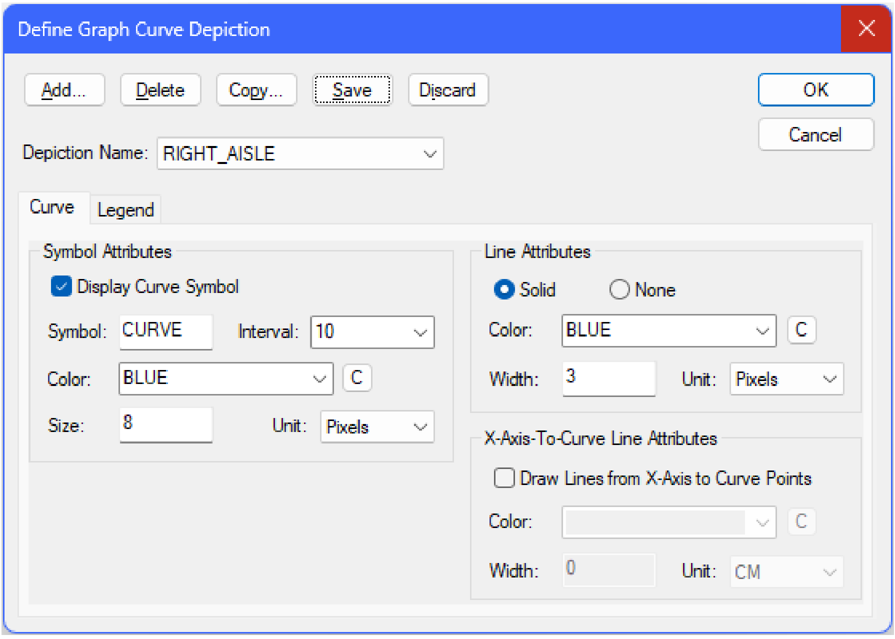
After entering the data, press the "Save" button, then press the "Add..." button again, enter the title "Right_aisle", press the "OK" button, and then enter the following data in the window:

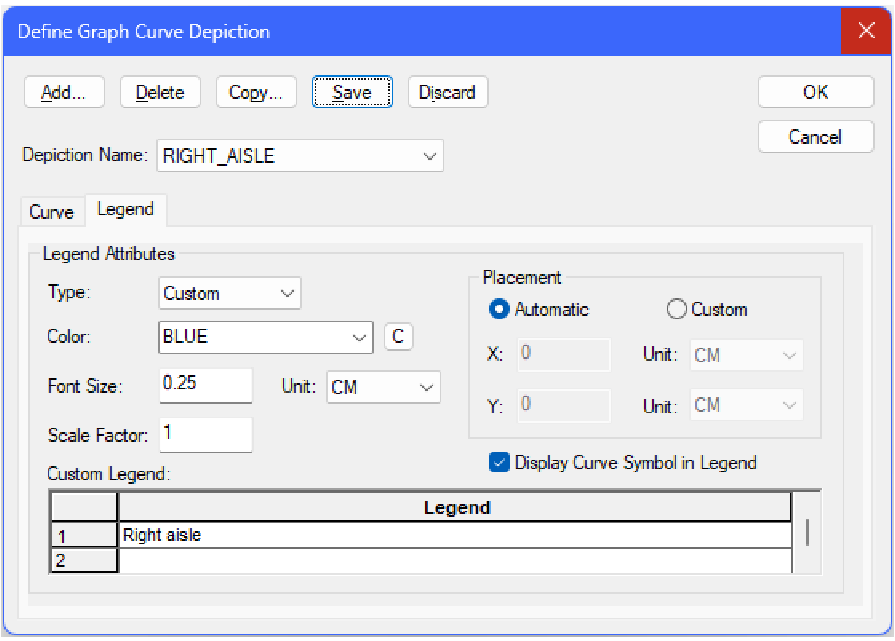
"Curve" tab	
Symbol Attributes	
Display Curve Symbol	Checked
Symbol:	CURVE
Interval:	10
Color:	BLUE
Size:	8
Unit:	Pixels
Line Attributes	
Solid	Checked
Color:	BLUE
Width:	3
Unit:	Pixels
X-Axis-To-Curve Line Attributes	
Draw Lines from X-Axis to Curve Points	Unchecked
"Legend" tab	
Legend Attributes	
Type:	Custom
Color:	BLUE
Font Size:	0.25
Unit:	CM
Scale Factor:	1
Placement	
Automatic	Checked
Display Curve Symbol in Legend	Checked
Custom Legend:	
	Legend:
1	Right aisle

After entering the data, press the “Save” button and then leave the window with the “OK” button. The view of the windows along with the entered data is shown in the drawings below:





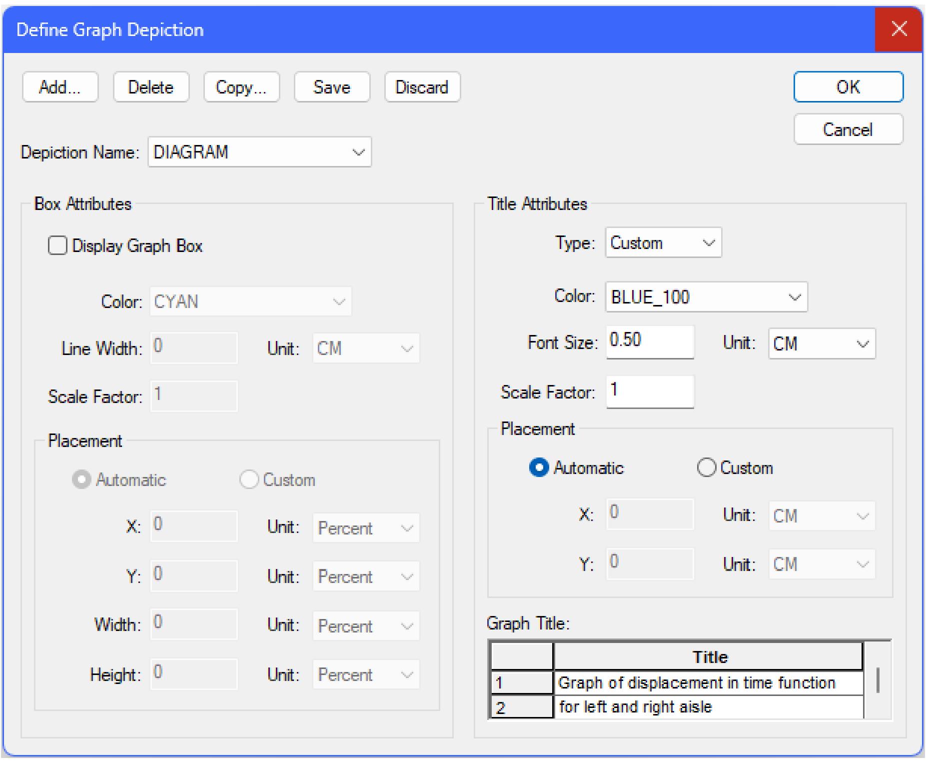




After returning to the "Display Response Curve (Model Point)" window, one need to enter the model title. To do this, press the "..." button next to the "Graph Depiction" option. In the newly opened window, press the "Add..." button, and when prompted for an identifying name for the data entered for the chart, enter "Diagram". Then enter the data presented in the table below:

Box Attributes	
Display Graph Box	Unchecked
Title Attributes	
Type:	Custom
Color:	BLACK
Font Size:	0.50
Unit:	CM
Scale Factor:	1
Placement	
Automatic	Checked
Graph Title:	
	Title
1	Graph of displacement in time function
2	for left and right aisle

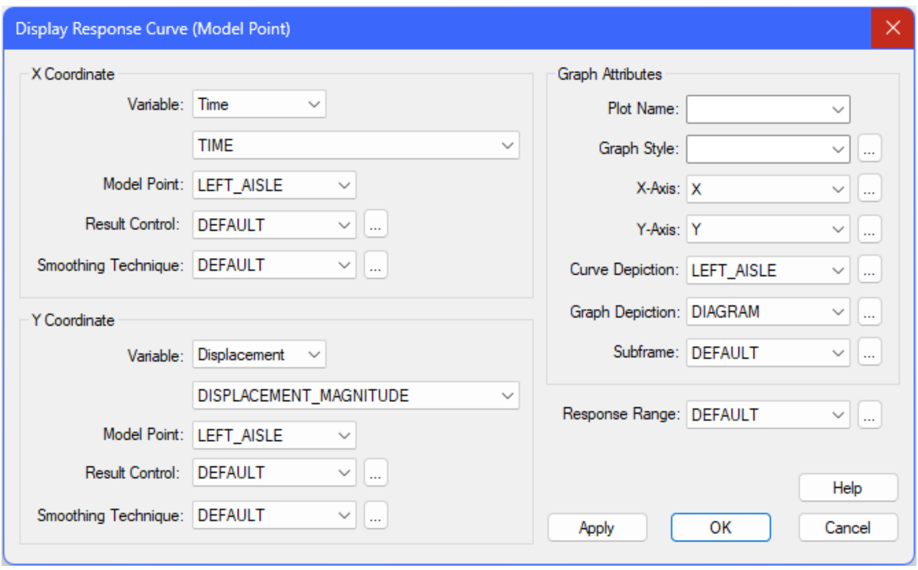
After entering the data, save the changes with the “Save” button and then exit the window with the “OK” button. The window view with the entered data relating to the appearance of the diagram is shown in the figure below:



In the “Display Response Curve (Model Point)” window, enter the following data:

X Coordinate	
Variable:	Time
	TIME
Model Point:	LEFT_AISLE
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Y Coordinate	
Variable:	Displacement
	DISPLACEMENT_MAGNITUDE
Model Point:	LEFT_AISLE
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Graph Attributes	
Plot Name:	Empty
Graph Style:	Empty
X-Axis:	X
Y-Axis:	Y
Curve Depiction:	LEFT_AISLE
Graph Depiction:	DIAGRAM
Subframe:	DEFAULT
Response Range:	DEFAULT

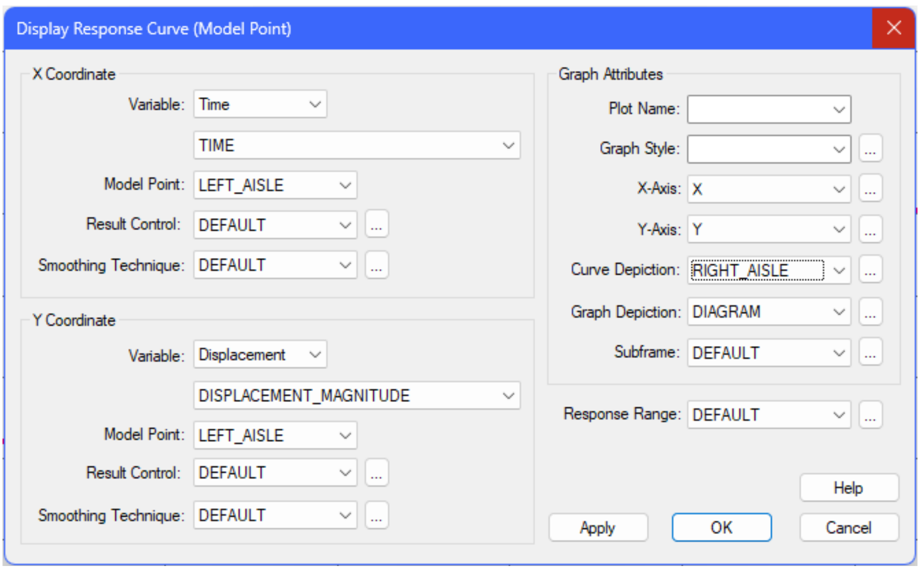
After entering the data, press the “Apply” button to apply the entered data. The window view with the entered data is shown in the figure below:



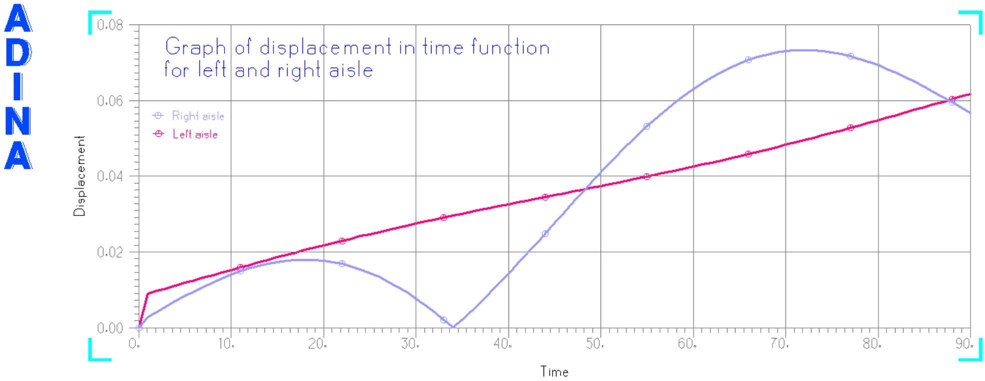
Then, in the “Display Response Curve (Model Point)” window, make the following changes:

X Coordinate	
Variable:	Time
	TIME
Model Point:	RIGHT_AISLE
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Y Coordinate	
Variable:	Displacement
	DISPLACEMENT_MAGNITUDE
Model Point:	RIGHT_AISLE
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Graph Attributes	
Plot Name:	Empty
Graph Style:	Empty
X-Axis:	X
Y-Axis:	Y
Curve Depiction:	RIGHT_AISLE
Graph Depiction:	DIAGRAM
Subframe:	DEFAULT
Response Range:	DEFAULT

After entering the data, press the “OK” button to apply the entered data and close the window. The window view with the entered data is shown in the figure below:



After performing all the above operations, the diagram in the main results window should look something like this:



<u>Note</u>: Title and lines depiction are rearranged with mouse click and drag-drop.

○ Three-dimensional problems

EXAMPLE 4
THE SPATIAL MODEL OF A SINGLE-CLAMPED ROUND BAR SUBJECTED TO STRETCHING

In this example a one-sided clamped bar with a circular cross-section subjected to stretching and mass-proportional load is modelled. In the analysis the spatial Cartesian coordinates system is used. It is recommended for the reader to get acquainted with the previous examples, since some of the functions were discussed earlier. A diagram of the analyzed model is presented in Figure 22.

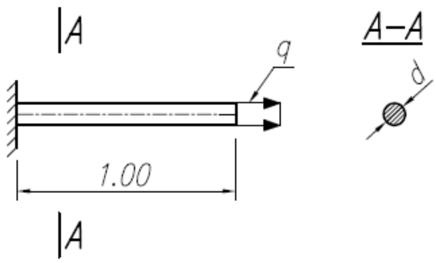
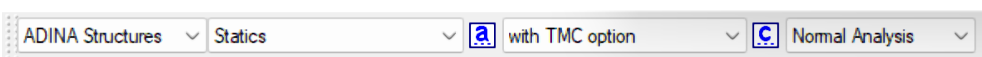


Fig. 22. Scheme of a bar with circular cross-section

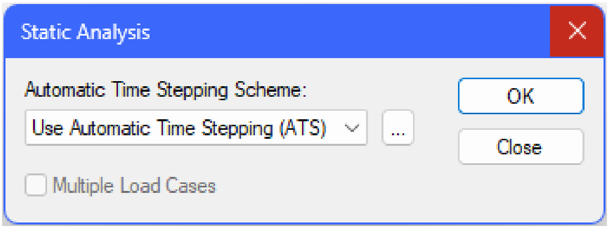
The following data is used in the analysis:
– bar cross-section:
  circular with $d = 0.10$ m
– load:
  tensile: $q = 10000\ \mathrm{N/m^2}$
– material constants:
  steel S235JR
  $E = 210$ GPa $= 2.1\mathrm{e}^{11}$ Pa
  $\upsilon = 0.30$
  $\rho = 7859\ \mathrm{kg/m^3}$

STEP 1. Definition of the type of analysis

Upon opening the ADINA software, choose “ADINA Structures” from the “Module Bar” in the “Program Module” section, and choose “Statics” from the drop-down list next to “Analysis Type”.



Moreover, click  , and in the newly opened window make sure that in the drop-down list a "Use Automatic Time Stepping (ATS)" option is chosen, if not please it should be chosen, then confirm the choice with the "OK" button.

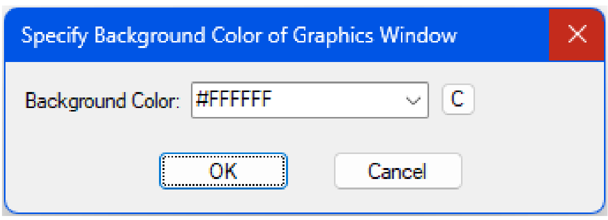


STEP 2. Entering the heading of the model

In order to specify a heading, go to "Control → Heading…". Subsequently, enter the project heading in the text box, e.g., "Stretched bar with circular cross-section". Upon entering a heading, click the "OK" button.

STEP 3. Definition of the background color of the main model window

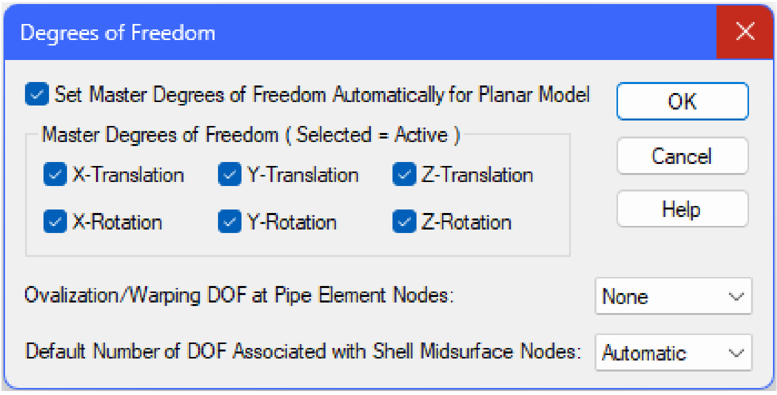
In order to define the background color, go to "Edit → Background Color…". Then, in the newly opened window choose the color white from the drop-down list. After choosing confirm the choice with the "OK" button.



<u>Note</u>: The background color is not saved along with the model. This means that after each opening of the file, the background color will return to default – "black".

STEP 4. Definition of global boundary conditions

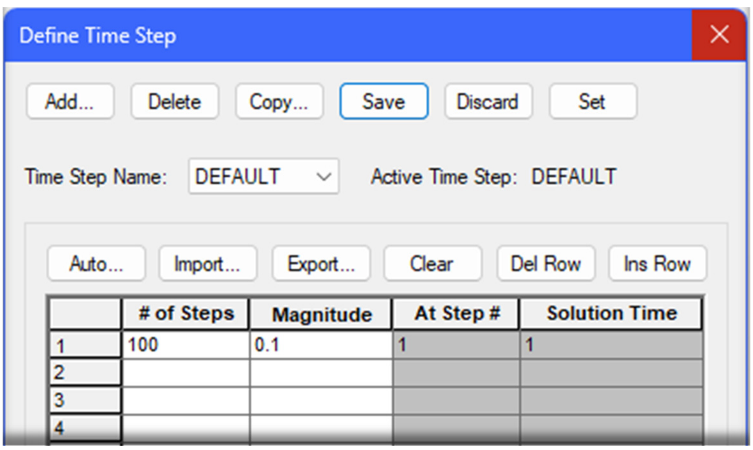
Since the model is a spatial model, all degrees of freedom should be included. Go to "Control → Degrees of Freedom…". In the newly opened window, select all boxes related to the degrees of freedom.



Note: Although the example uses a circular element (similar to 'pipe') and bending with respect to the Z axis and stretching along the X axis will be taken into considerations (similarly as for 2D type of analysis), the "Ovalization/Warping DOF at Pipe Element Nodes" option should remain as "None". It is connected with fact, that the model will be represented with volumetric finite elements, not via 1D (line) elements with assumed cross-section. The cross-section of a bar will be modeled directly in this example. Window with the input data is presented above.

STEP 5. Definition of the number of time steps

In order to define time steps, go to "Control → Time Step…". Then in the newly opened window enter a value of "100" in the first row of the "Number of Steps" column, and a value of "0.1" in the same row of the "Constant Magnitude" column. Upon the values are introduced, click the "Save" button, followed by "OK". The window view is shown in the figure below:



STEP 6. Definition of load variability in subsequent time steps

Linear variability of load over time will be used for the purpose of the present example. It is assumed that the load will increase from 0 to its maximum value over a time of 5 s, then the load will decrease linearly to reach a value of 2000 N at a time of 10 s from the start of the analysis.

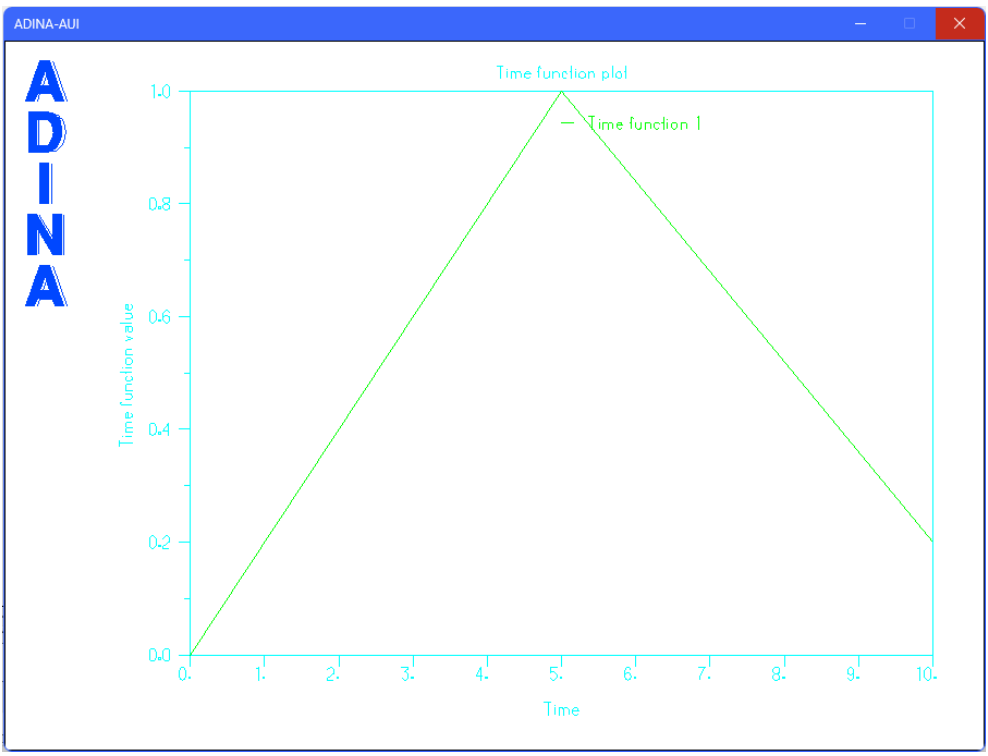
In order to declare the variability of load over time, go to "Control → Time Function…". In the newly opened window enter following data:

Time Function Number:	1
Function Multiplier:	Constant (=1.0)

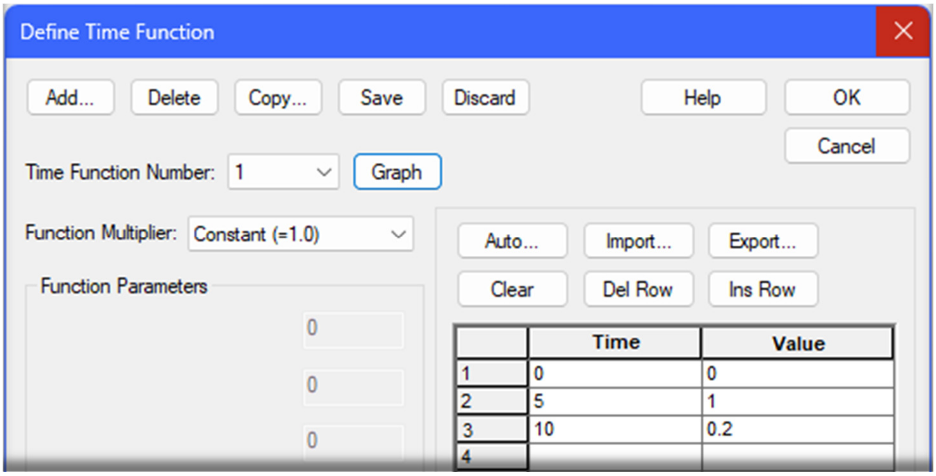
And in the table make the following changes:

	Time	**Value**
1	0	0
2	5	1
3	10	0.2

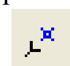
After entering the data for function number "1", user can press the "Graph" button and the following graph should appear:



The chart window should then be closed by pressing the "X" button in the upper right corner. After entering all the data, press the "Save" button and then leave the window with the "OK" button. The "Define Time Function" window with introduced data should look as:

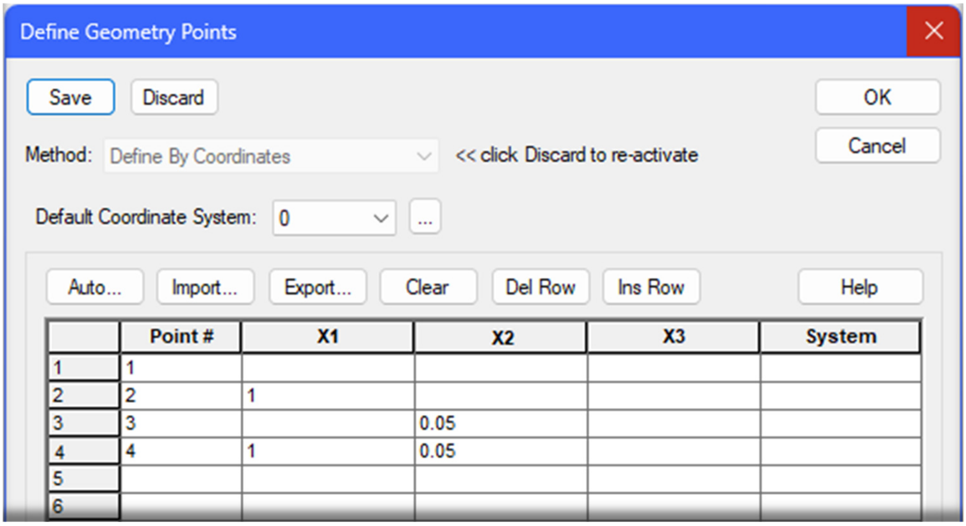


STEP 7. Definition of points

In order to define points, go to "Geometry → Points → Define…" in the main model window, or click . Upon opening a new window, input points in accordance with the table below:

Point #	X1	X2	X3	System
1	0	0	0	0
2	1	0	0	0
3	0	0.05	0	0
4	1	0.05	0	0

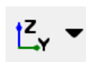
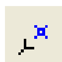
As soon as all the data is introduced, click the "Save" button, followed by "OK". The window view with the entered data is shown in the figure below:



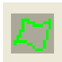
<u>Note</u>: There is no need to enter zero values into the table. The software will assume them by itself after clicking the "Save" button.

<u>Note</u>: In older ADINA software versions, once the points are introduced in the table and saved, the program will display an XY plane (2D); a proper view of the space may be achieved later. Currently, it is easier to work in a planar rather than in a spatial system. In newer ADINA versions program automatically shows points in the isometric view.

STEP 8. Displaying point ID numbers

To display point identifiers in the XY plane, press the arrow located on the  button and select "+XY view". Subsequently press the  button. The model should look something like this:

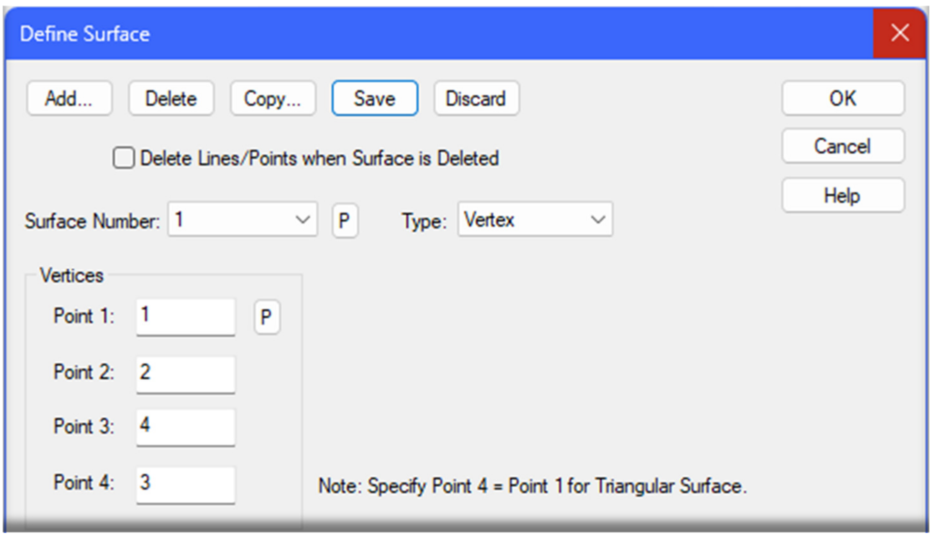
STEP 9. Definition of surface(s)

Due to the fact that points have already been created, it is possible to define surface on that points. In order to declare a surface, go to "Geometry → Surfaces → Define…", or click . Upon opening a new window, click the "Add…" button in order to add surface no. 1. Since the surface will be created on the basis of existing points, the surface type "Type:" should be set as "Vertex". Enter the data as provided in the table below:

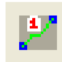
Surface Number:	1
Type:	Vertex
Point 1:	1
Point 2:	2
Point 3:	4
Point 4:	3

The points numbers forming the surface may be entered using the keyboard or by mouse – previously hitting the "P" button standing near the empty field of "Point 1:" option. It should be noted, that the order of points is important. Picking points which would form a planar surface with crossing lines should be avoided.

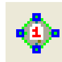
A window with properly input data is presented in the figure below:

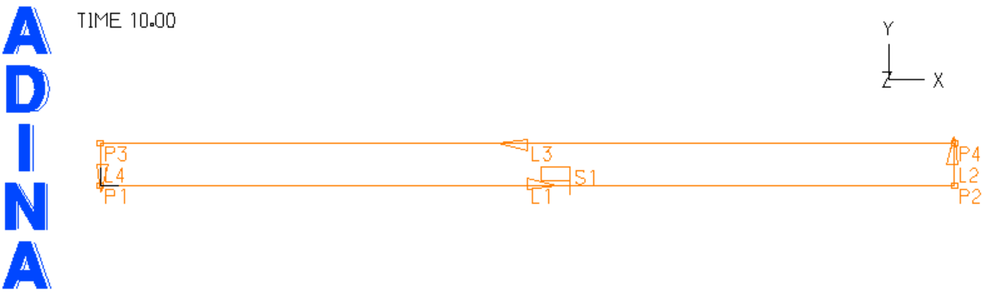


STEP 10. Displaying line ID numbers

In order to display the numbers of lines for their easier identification, click  button.

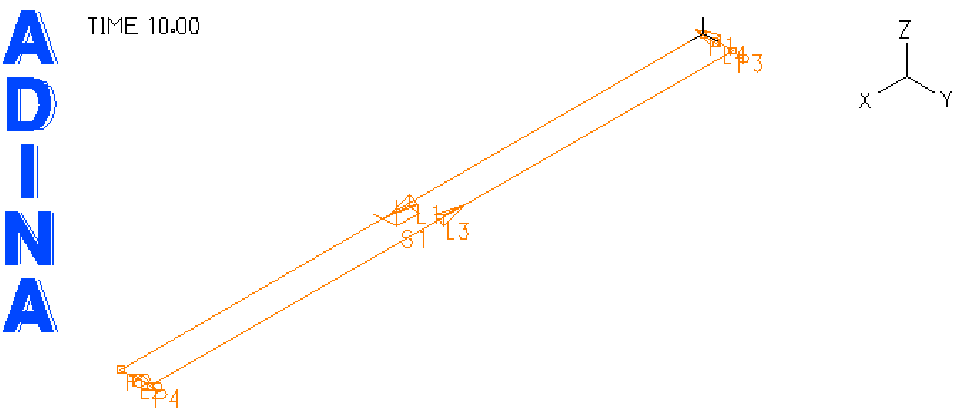
STEP 11. Displaying surfaces ID numbers

In order to display the numbers of surfaces for their easier identification, click  button. The model should look like in the figure below:



STEP 12. Displaying the model in a desired plane/space

In order to display the model on a proper plane or in space, click the arrow next to . Then choose a proper display option from the drop-down list. For the purpose of the present example, choose "IsoView 1". A view of the model is presented in the figure below:

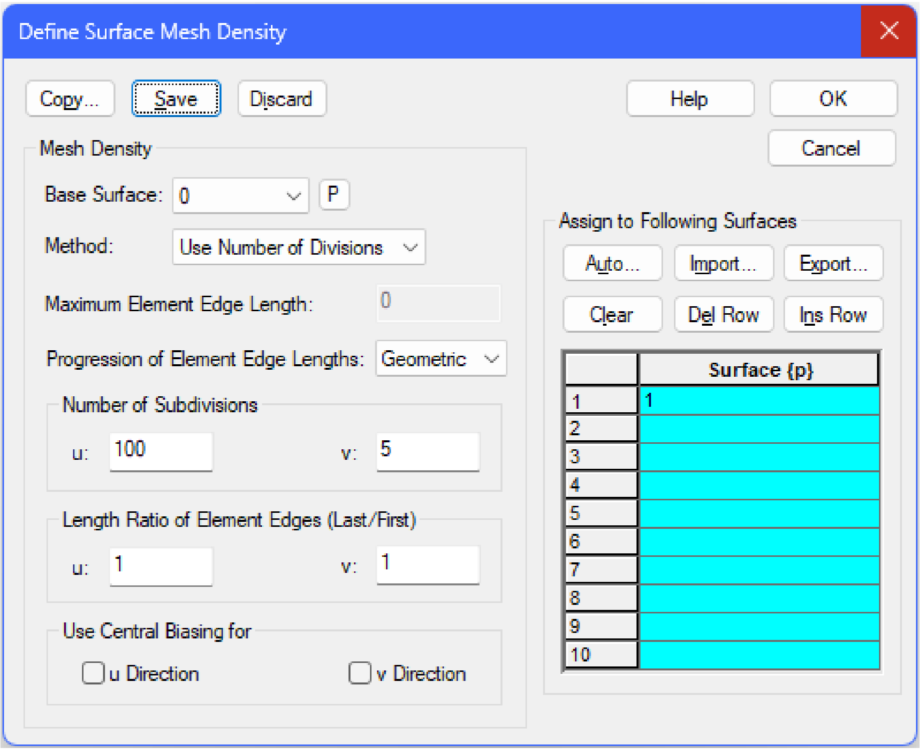


STEP 13. Mesh subdivision

Definition of the density of a finite element mesh before the creation of a volumetric element enables the generation of a radially propagating mesh for circular elements. In order to define the mesh density for the surface, go to “Meshing → Mesh Density → Surface…”. Then in the newly opened window make following changes:

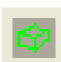
Base surface:	0
Method:	Use Number of Divisions
Progression of Element Edge Lengths:	Geometric
Number of Subdivisions	
u:	100
v:	5
Length Ratio of Element Edges (Last/First)	
u:	1
v:	1
Use Central Biasing for	
u Direction	Unchecked
v Direction	Unchecked
Assign to Following Surfaces (Table)	
	Surface {p}
1	1

Upon introducing all the data, click the “Save” button, followed by “OK”. A view of the data window is presented in the figure below:



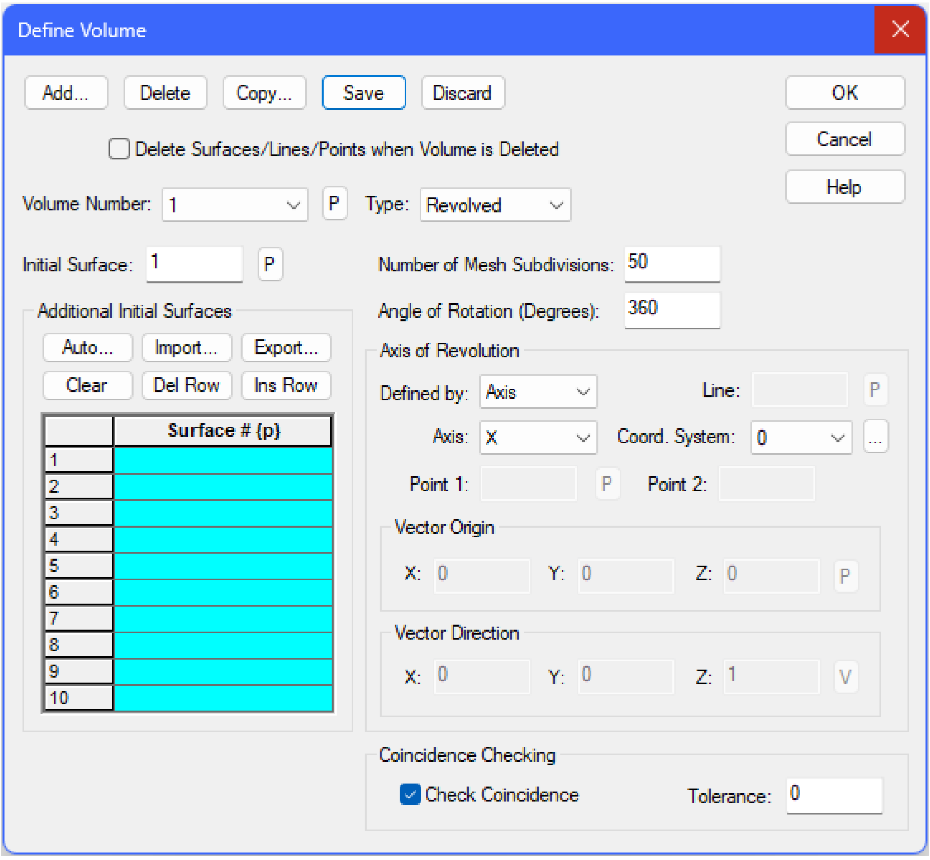
<u>Note</u>: Creating a mesh before creating a volumetric element has an additional advantage. Such an approach of specifying a mesh density before creating a volumetric element allows to save time – in case a volumetric element is created first, the density of meshing must be adjusted to the faces of a cylinder, and to the line forming the mantle of the cylinder. Moreover, the definition of finite elements in a model created in this manner may cause these elements to not be created radially.

STEP 14. Definition of a volumetric element

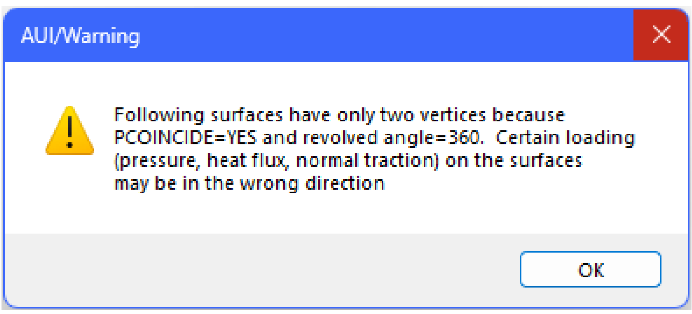
In order to define volumetric elements, go to "Geometry → Volumes → Define…", or click . When a new window opens, click the "Add…" button in order to add a volumetric element to the model, with the id number "1" and enter following data:

Volume number:	1
Type:	Revolved
Initial surface:	1
Number of Mesh Subdivisions	50
Angle of Rotation (Degrees)	360
Axis of Revolution	
Defined by:	Axis
Axis:	X
Coord. System:	0
Coincidence Checking	
Check Coincidence	Checked
Tolerance	0

Choosing the "Revolved" option from the drop-down list in "Type:" box will cause the creation of a volumetric element by revolving a surface around a specified axis of revolution. Enter a value of "1" in the "Initial Surface" box, or click the "P" button and indicate a surface for rotation in the main program window. In the "Angle of Rotation:" box, enter a value of "360" – full rotation, and enter a value of "50" for the "Number of Mesh Subdivisions:" option. It is a value of division of a finite element mesh for a circle created by rotation of the surface. All that remains is to make sure that rotation will proceed with respect to the "X" axis, meaning that the "Axis of Revolution" group of options in the "Defined by:" box should have the "Axis" option selected from the drop-down list, with the "X" option selected in the "Axis" box. In addition, the "Check Coincidence" option should be selected. Upon inputting all the data, click the "Save" button, and then "OK". The figure below presents the window along with the input data:

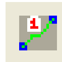
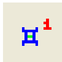


<u>Note</u>: Upon clicking the "Save" button, the following message will be displayed:



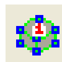
This message pays user attention to the fact that the created volumetric element has two vertices, and that some of the loads, such as pressure, heat flux or traction, may have a wrong direction compared to the definition.

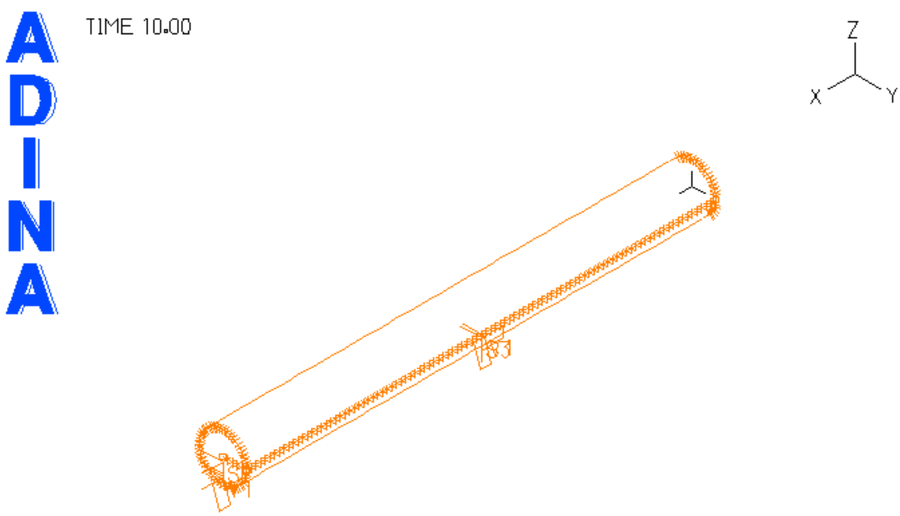
In such a case, when this message is displayed, it is recommended to take special care when defining loads. After getting acquainted with this message, close it using the “OK” button.

When a volumetric element has been input into the model via rotating a surface around the X axis, the view of the lines ID numbers can be deactivated with , and those of points with .

<u>Note</u>: Only one possible way of creating a volumetric element by rotating a surface is presented above. Rotation can be performed in 4 ways – by indicating an axis of the coordinate system as the axis of rotation (as it was done in the example), by choosing two points between which an invisible line forming the axis of rotation is generated, by selecting a line specifying the axis of rotation from the model, or by defining a rotation vector.

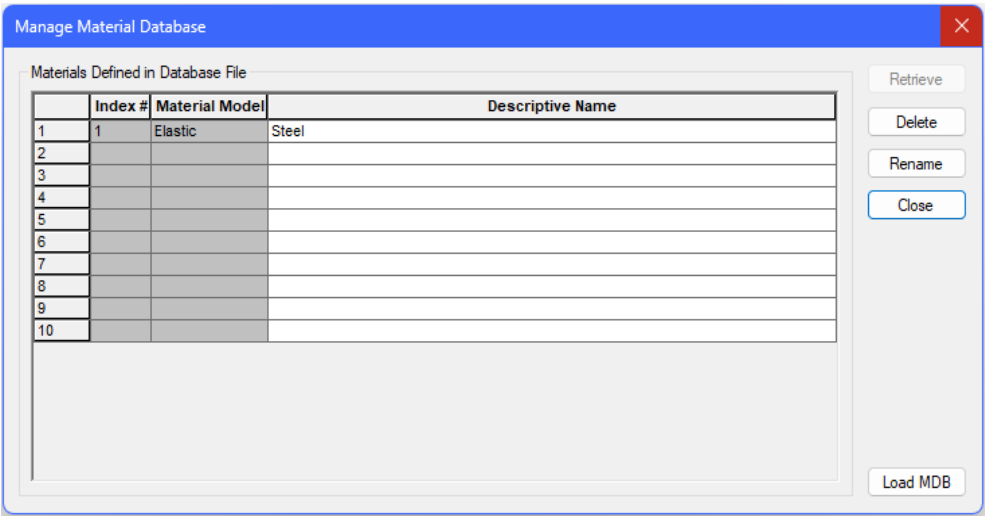
STEP 15. Displaying volumetric elements ID numbers

In order to display the ID numbers of volumetric elements, click  button. The model window should look like this:

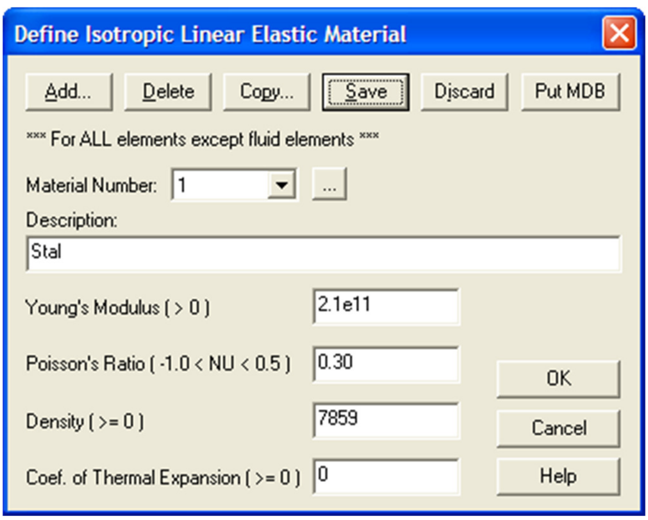


STEP 16. Definition of material constants

In order to go to the definition of material constants, choose the “Model → Materials → Manage Materials” tab, or click , then click the “Get MDB” button in the newly opened window (when materials have previously been added to the program database). Then select a material named “Steel” from the table, and click the “Retrieve” button. When the material has been loaded from the program database, click the “Close” button. In the lower part of the “Manage Materials” window, the Steel material returned from the database should be present in the table cell with the id number “1”. A view of the window for retrieving materials from the database is presented below:



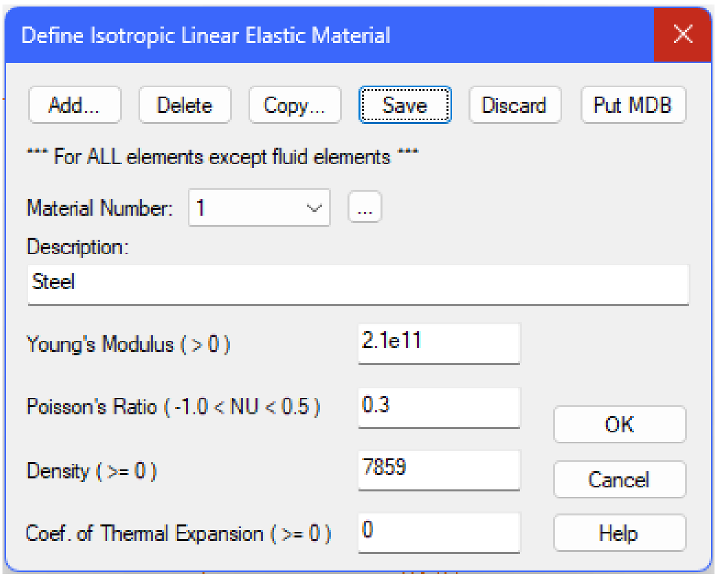
When the material required in the example is absent from the database, in the “Manage Materials” window find a group referring to elastic materials – “Elastic”, and then click the “Isotropic” button (isotropic material).



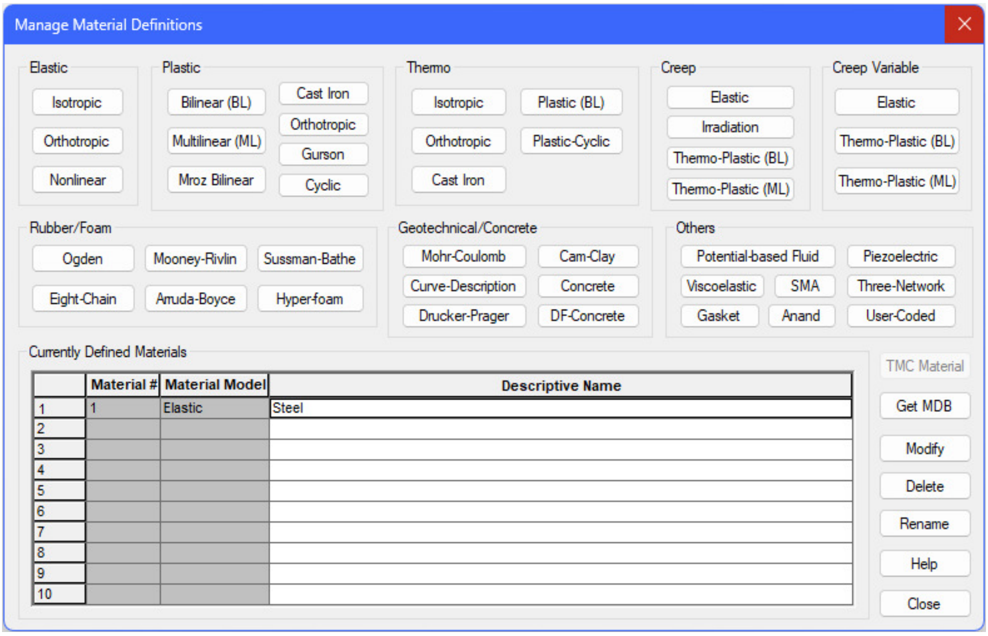
In the newly opened window, first click "Add…" in order to add a new material, and then enter the values of material constants as shown in table below:

Material Number:	1
Description:	Steel
Young's Modulus (> 0)	2.1e11
Poisson's Ratio (–1.0 < NU < 0.5)	0.30
Density	7859
Coef. of Thermal Expansion (>= 0)	0

The view of a window with the material constants introduced in the elastic material definition is shown in the figure below:

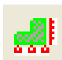


Upon inputting the data, click the "Save" and "OK" buttons. The main material manager window "Manage Materials" looks like in the figure below:



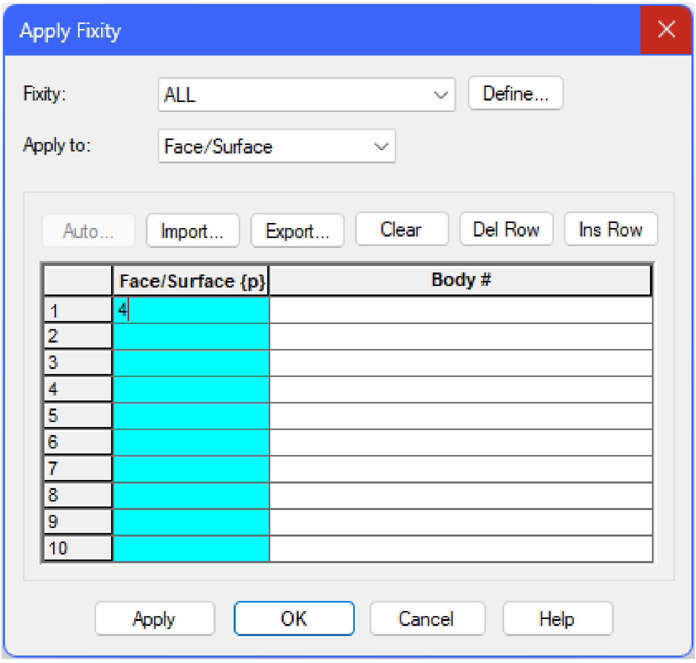
The last step is to close the "Manage Materials" window with the "Close" button.

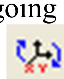
STEP 17. Definition of boundary conditions (fixity characteristics)

In order to define and apply a fixity in the system, go to "Model → Boundary Conditions → Apply Fixity…", or click . Since the example includes clamped support, and the program has a declared fixity with all degrees of freedom fixed by default (fixity name: "All"), there is no need to declare the fixity once again. In the "Apply Fixity" window, choose:

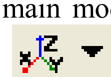
Fixity:	ALL
Apply to:	Face/Surface
	Face Surface {p}
1	4

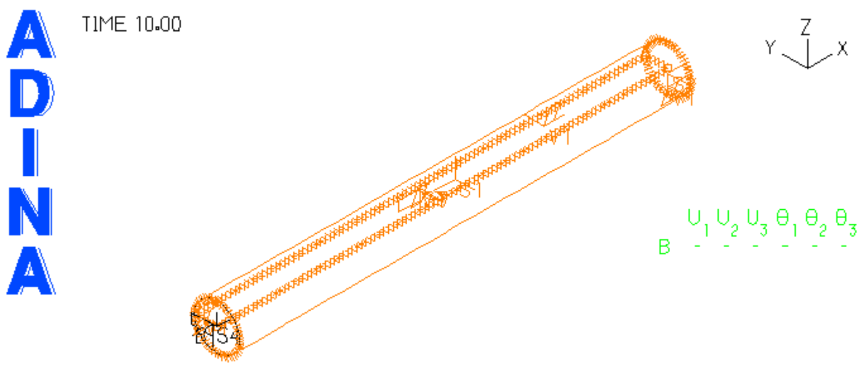
A surface having its center at the point 0;0;0 should have been chosen above. In case of selecting by mouse method when a proper surface has been selected from the model window, press the ESC key to end the selection, and return to the window for applying boundary conditions. Upon entering the surface number in the "Apply Fixity" window, click the "Save" button, and then "OK", therefore closing the window. A view of the window with the input data is presented below:



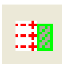
Note: Since double-clicking the table with the left mouse button results in moving the user to the model window, prior care should be taken to make the interesting surface or another element visible before performing this operation. It should be taken into considerations due to model cannot be rotated during the selection. Therefore (even with an open window for applying boundary conditions, loads, etc.), the model should be rotated, so that the part which is going to be selected should be visible. In order to activate the rotation option, click , and then hold the left mouse button over the model and move the mouse to rotate the model. Different method is to click the model once, to get the model in a dashed line box, then hold "SHIFT" button from the keyboard, click and hold left mouse button on the model, and then move the mouse, which will result in model rotation. In case of view the model from a different side/angle, it may prove helpful to use , which allows displaying the proper view plane. In this case, a perfect view of surface "S4" is achieved by clicking the arrow next to the button, and selecting "IsoView 3".

STEP 18. Displaying boundary conditions

In order to display the defined boundary conditions in the main model window, click  (after selecting the "IsoView 3" view type from ). Currently, the model should look like this:



STEP 19. Definition of loads

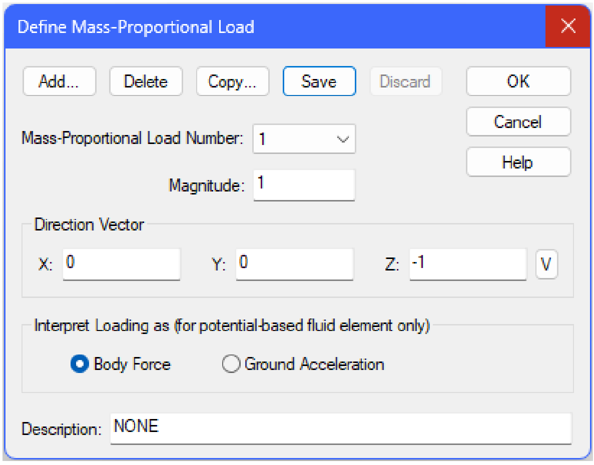
In order to define a load, choose “Model → Loading → Apply…” from the upper menu, or click . In the newly opened window, define both the mass-proportional load and an external load in the form of a surface load (pressure). After opening a new window, the self-weight load of the structure will be declared first, so for the “Load Type:” option, select “Mass Proportional” from the drop-down list, then for the “Load Number:” option.

Load Type:	Mass-proportional
Apply to:	Model

Then press the “Define...” button. A new window will open, in which one should press “Add...” button to add a new load, then enter the data as shown in the table:

Mass Proportional Load Number:	1
Magnitude:	1
Body #:	1
Direction Vector	
X:	0
Y:	0
Z:	–1
Interpret Loading as (for potential-based fluid element only)	
Body Force	Checked
Description:	None

Finally, save the defined load by clicking the “Save” and “OK” buttons.



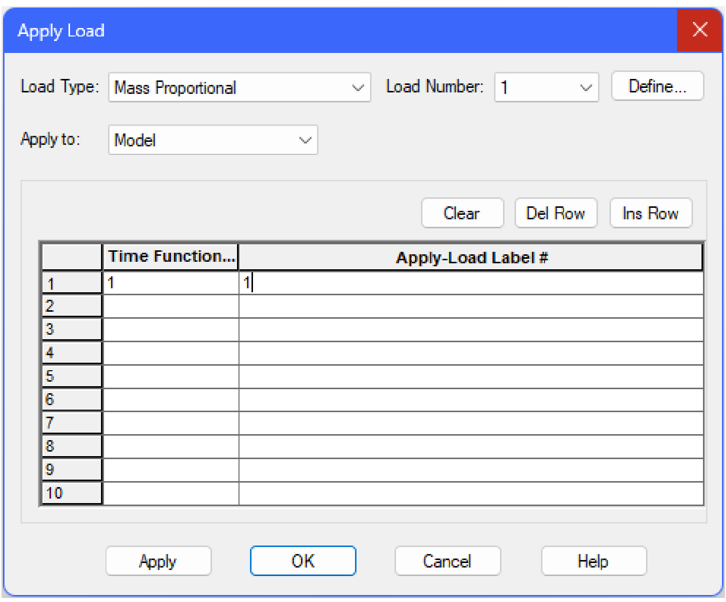
Upon returning to the "Apply Loads" window, insert following data:

Load Type:	Mass Proportional
Load Number:	1
Apply to:	Model

In the table, the data should look like this:

	Time Function…	**Label #**
1	1	1

The window view with the entered data is shown in the figure below:



When this operation is completed, click the “Apply” button to apply a load caused by the mass-proportional load in the model without leaving the window. A view of the window along with the input mass-proportional load is presented in the figure above.

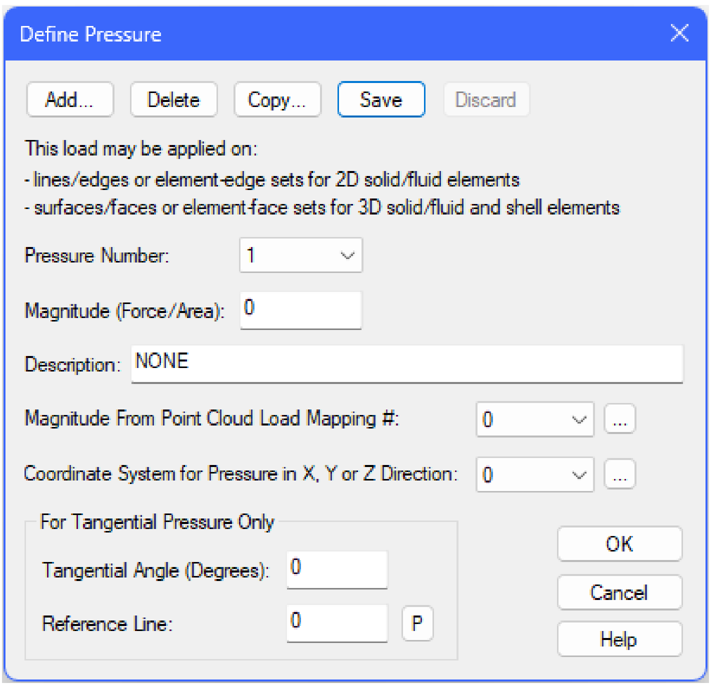
As soon as mass-proportional load is defined and assigned, choose “Pressure” from the “Load Type:” drop-down list and choose following data:

Load Type:	Pressure
Apply to:	Surface

After this operation, click the “Define…” button, and the “Add…” button in the newly opened window and enter following data:

Pressure Number:	1
Magnitude (Force/Area):	10000
Description:	None

Remaining options leave unchanged and then confirm the input data with the “Save” button, and subsequently close the window with the “OK” button. The figure below presents the define pressure window:



Upon leaving the window for defining pressure, in the “Apply Load” window enter the following data:

Load Type:	Pressure
Load Number:	1
Apply to:	Surface

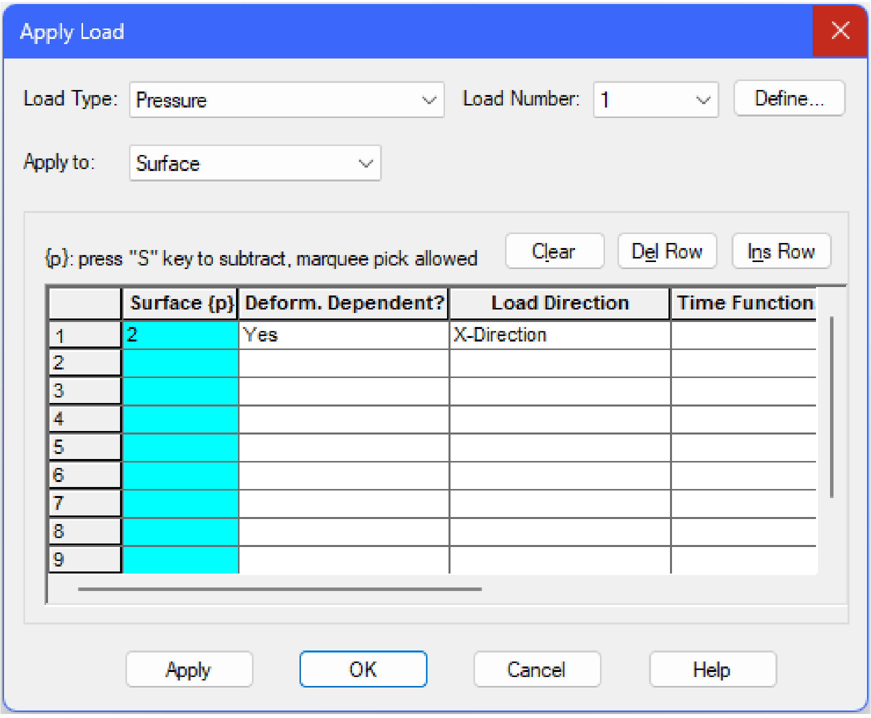
In the table, the data should look like this:

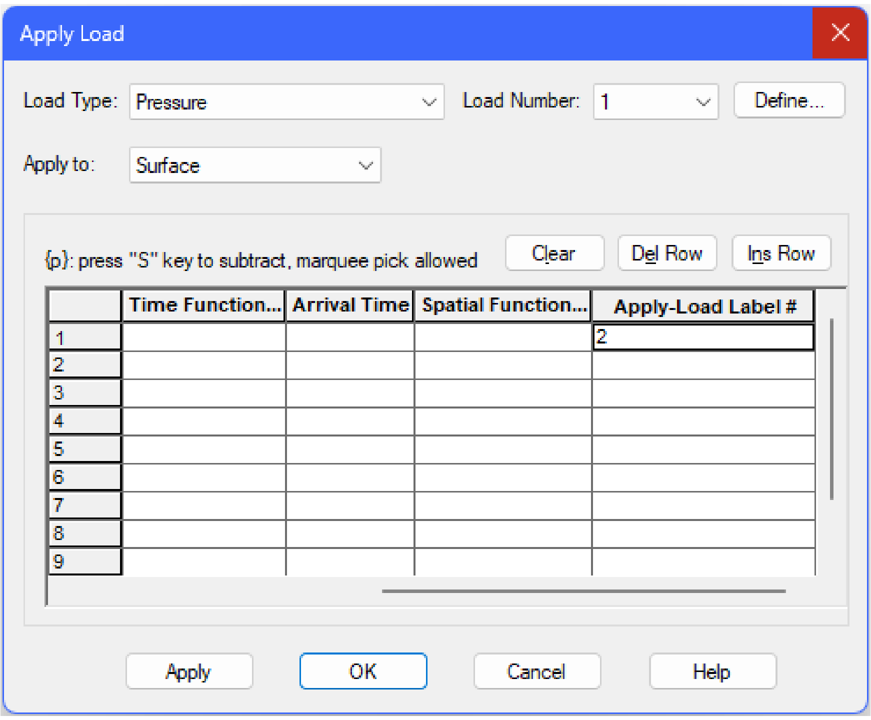
	Surface {p}	Deform. Dependent?	Load Direction	…	Apply-Load Label #
1	2	YES	X-direction	Empty	2

First of all the surface no. 2 should have been chosen.Therefore, enter a value of “2” in the first row. This can also be entered by double-clicking the “Surface {p}” column with the left mouse button, selecting the desired surface directly from the model, and completing the selection with the “ESC” key.

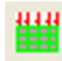
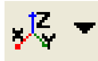
Note: As already previously mentioned, double-clicking the table with the left mouse button results in moving the user to the model window. Therefore, prior care should be taken for the surface or another element to be visible before performing this operation (see the previous Note). In this case, a perfect view of surface “S2” will be achieved by selecting “IsoView 1”.

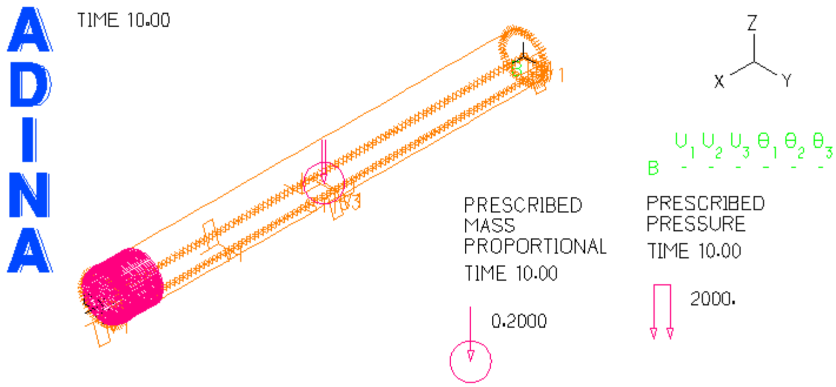
The “Deform. Dependent?” option set as “yes” means that the load will adjust its direction together with the rod deflection. In example, having a load perpendicular to the plane, as soon as the finite element starts to deform, the pressure is divided into appropriate concentrated forces acting on finite element nodes, and this load during the analysis will always be perpendicular to the tangent line/plane of finite element outline. Choosing the “Global X-Dir.” option specifies that the load will act in accordance with the X axis of the main coordinate system. Once these operations are completed, the horizontal scroll bar in the “Apply Load” window should be moved to the right, until the “Label #” column appears. In this column, enter a value of “2” in the first row, which becomes the identification number of the pressure load. As soon as all the values are introduced, click the “Apply” button and “OK” closing the window for defining and assigning loads. The following figures present the application of pressure load in the model:


Apply Load
Load Type: Pressure
Load Number: 1
Define...
Apply to: Surface
{p}: press "S" key to subtract, marquee pick allowed
Clear
Del Row
Ins Row
Surface {p}
Deform. Dependent?
Load Direction
Time Function
1 2 Yes X-Direction
2
3
4
5
6
7
8
9
Apply
OK
Cancel
Help


Apply Load
Load Type: Pressure
Load Number: 1
Define...
Apply to: Surface
{p}: press "S" key to subtract, marquee pick allowed
Clear
Del Row
Ins Row
Time Function...
Arrival Time
Spatial Function...
Apply-Load Label #
1 2
2
3
4
5
6
7
8
9
Apply
OK
Cancel
Help

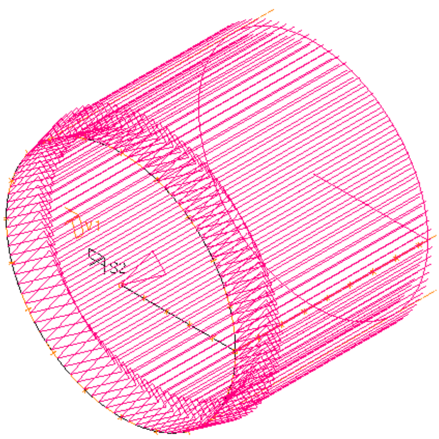
STEP 20. Displaying active model loads

In order to display defined loads in the main model window, click  (a view for the "IsoView 1" option selected by means of the arrow next to ). The model window looks like this:



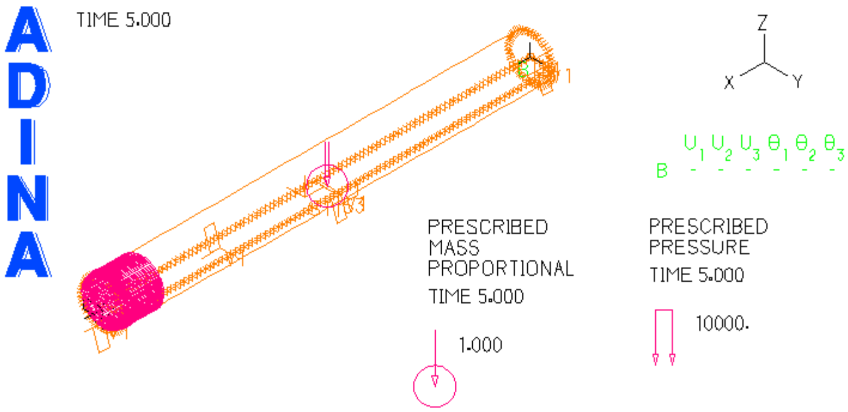
<u>Note</u>: The figure above may be unclear. Since during earlier declaration of the volumetric element, there was a message that some of the loads such as pressure, heat flux and traction may have a different direction compared to the definition, it is necessary to check whether the existing load is consistent with our declaration.

To this end, magnify the model in the area of load by means of  (selecting the area of interest). The arrows of load should be directed toward surface S2, as shown in the figure below.



If everything is correct, shrink the element to the default view by means of .

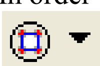
Since the default choice of the program is to display the last time step (in the present example 10 s – "Time 10.000" on the screen), the load is also adjusted to this time step. It was previously declared that at the 10 s the load is supposed to reach 0.2 of the maximum value, meaning 2000 N.



Changing the time in the model by means of  until "Time 5.000" is displayed (for a maximum load), and then double-clicking  will cause the displayed load value to amount to "10000". The figure below presents the model with the time set as "Time 5.000". It should be noted that changing the time step via  buttons does not update the values of applied loads.

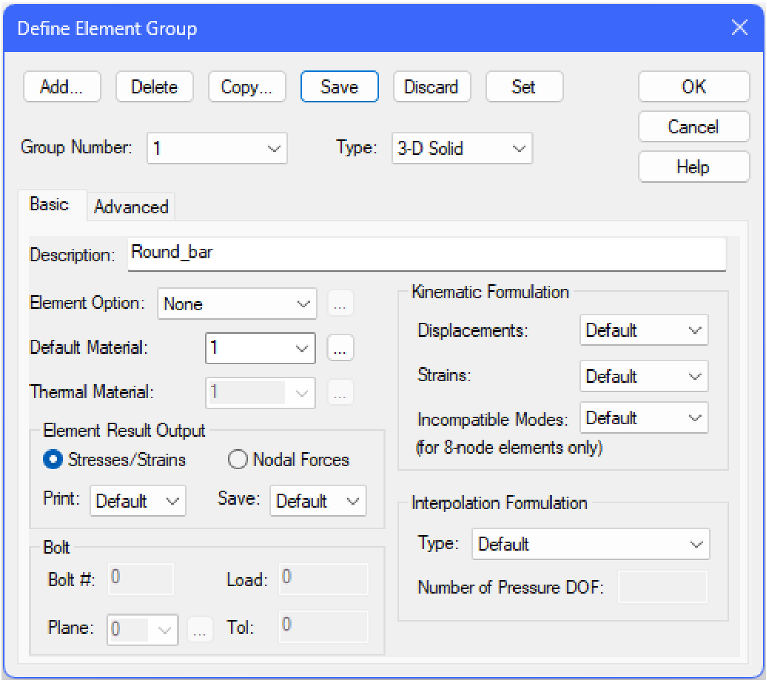
STEP 21. Specifying the type of analyzed construction

The present example requires the creation of only one element group. It is connected with the analyzed bar is made of a single type of material only.

In order to create an element group, go to "Meshing → Element Groups…", or click . After opening a new window, press the "Add..." button and then enter the following data:

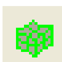
Group Number:	1
Type:	3D-Solid
"Basic" tab	
Description:	Round_bar
Element Option:	None
Default Material:	1

The remaining options remain unchanged. The window with the entered data is shown in the figure below:



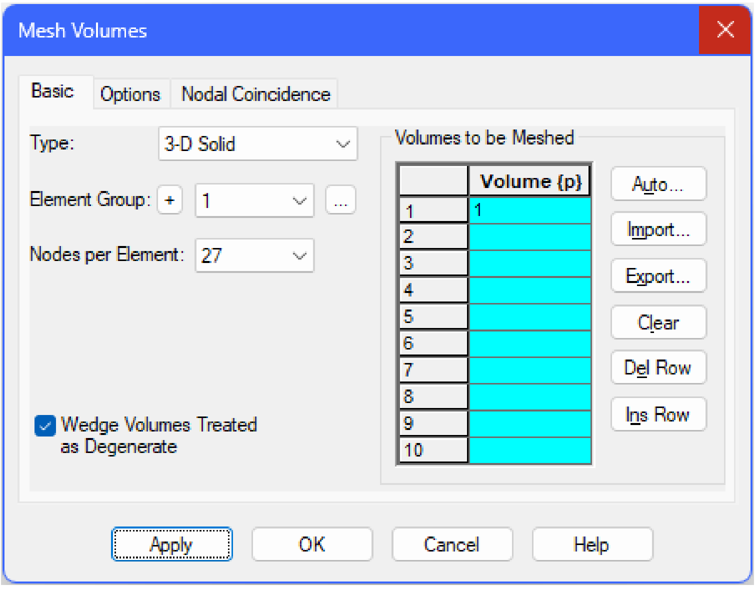
Upon inputting all the data, click the "Save" button, and then "OK".

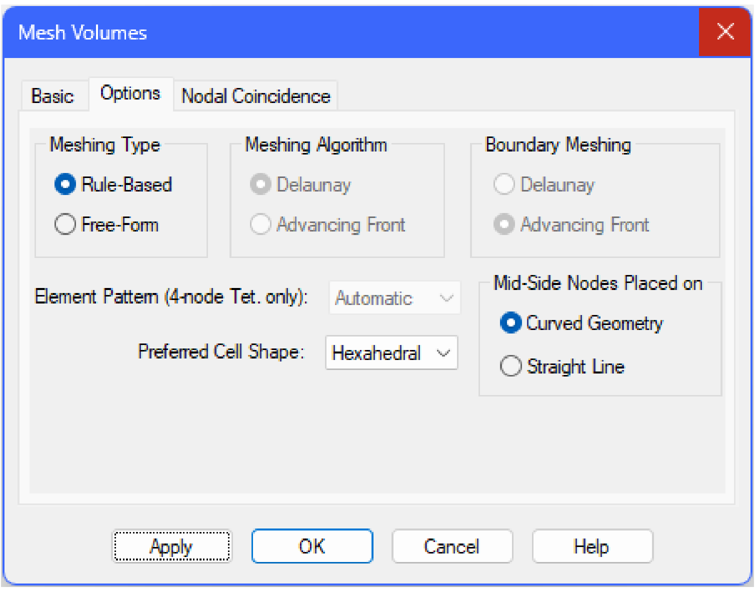
STEP 22. Definition of finite elements

In order to define finite elements in the model, go to "Meshing → Create Mesh → Volume…", or click . Once the new window has opened enter following data:

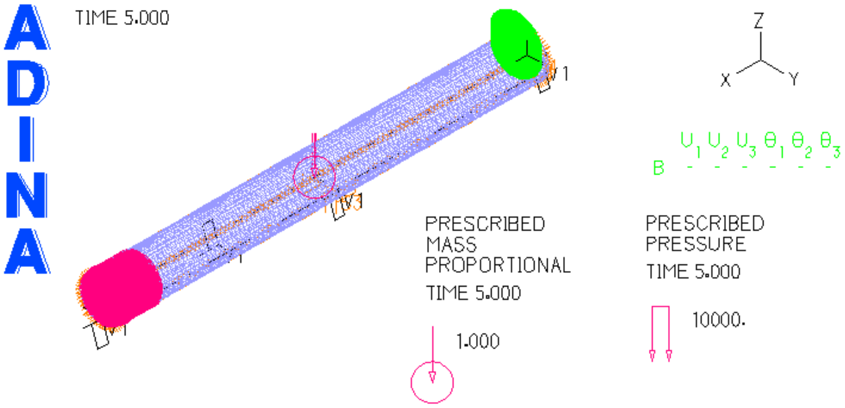
"Basic" tab	
Type:	3-D Solid
Element Group:	1
Nodes per Element:	8
Volumes to be Meshed	
	Volume {p}
1	1
"Options" tab	
Rule-Based	Checked
Preferred Cell Shape:	Hexahedral

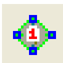
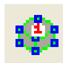
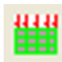
Remaining options leave unchanged. Window with entered data are presented below:

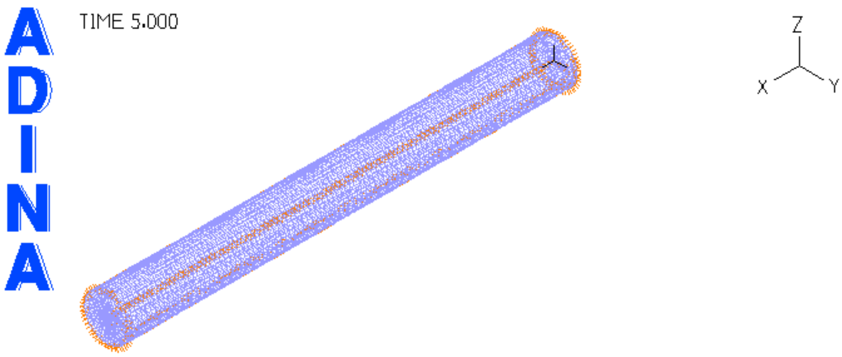




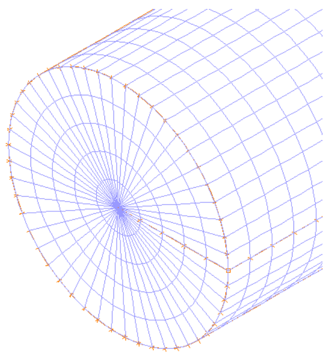
Upon inputting all the above mentioned data, click the “Apply” button, and then “OK”. When all the settings have been applied, the model should look like in the figure below (the view is set as “IsoView 1”).



Once the view of the identification numbers of surfaces has been deactivated with , and those of volumetric elements with , and the view of boundary conditions  and loads  has been deactivated, the model should display the finite element mesh only.



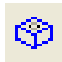
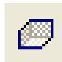
Magnification of the cylinder base can be achieved by means of ; the view looks like this:



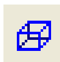
As can be seen in the figure above, finite elements propagate radially, as was previously assumed.

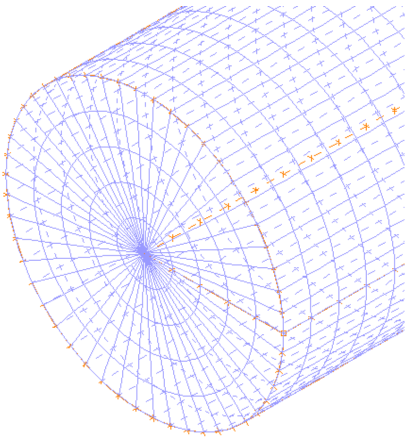
<u>Note</u>: For a different way of creating an element with a cylindrical shape, and when willing to prepare a finite element mesh, it may be helpful to select the "Wedge Volumes Treated as Degenerate" function in the "Mesh Volume" window, which in the present example was active by default.

STEP 23. Model presentation types

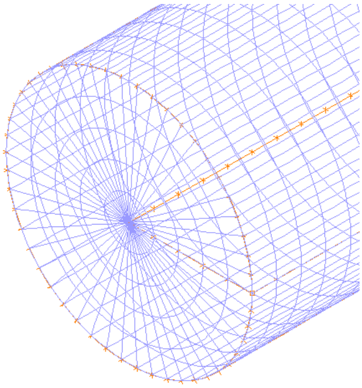
By keeping the magnification of the analyzed bar, it is possible to demonstrate the action of the following buttons: , , , and .

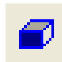
In the program, the button named "Hidden Surfaces Removed"  is active by default, which means that all things located inside the model are not visible.

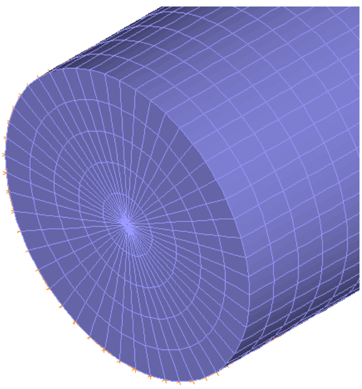
Activation of the button named "Wire Frame"  will display all the lines (the invisible ones are shown as dashed lines), like in the figure below.

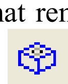


When the button named "Cull Front Faces"  is activated, all the lines of the outer surfaces will be displayed (including those which are on the other side of the model). All the lines will be solid.



Deactivation of , and activation of the button named “Shading”  will display a shaded model. Work is made much easier when numerous various notches have been made in the model. When this button has been activated, the model looks like this:



After viewing the possible presentations of the model, return to the default settings by clicking . Subsequently, the view of the load and the boundary conditions can be reactivated. When  has been selected, move the legend of both the loads and the boundary conditions to a convenient place. All that remains is to activate the view of the finite element mesh without shading, using .

STEP 24. Save existing model to a file

Each model should have been saved to a file between few steps taken in order to not lose the data. According to that select “File → Save as...” from the menu. When a new window opens, indicate the location of the saved file and its name.

Note: Do not uses spaces in the file names, because it leads to an error! The space can be replaced with the underline character __.

STEP 25. Starting calculations

In order to start calculations, choose "Solution → Data File/Run" from the upper menu tabs, or choose  from the toolbars. Subsequently, input the path for saving the resultant file. However, before the user clicks the "Save" button, they should make sure that the "Run Solution" and "Automatic" function near the "Maximum Memory for Solution" options are selected in the window in the "Adina Structures Solution" group of options. After starting the calculations and recalculating the model, close all three dialogue windows which appeared during the calculations.

Note: Depending on the complexity of the model and the number and type of finite elements used, model calculations may take from a few seconds to even several hours.

STEP 26. Post-processing module (results)

In order to go to the results module, change "ADINA Structures" to "Post-Processing" in the drop-down list of the "Module" toolbar.

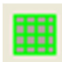
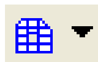
When the user is prompted that the changes in the drawing have not been saved, it is recommended to save the model by going to "File → Save" or using the button .

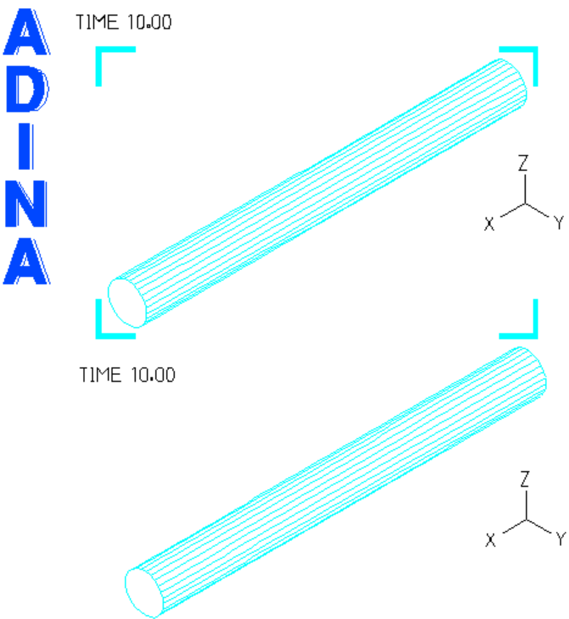
STEP 27. Opening the resultant file

In order to open the resultant file, choose  from the toolbar, or choose "File → Open" from the upper menu tabs.

Note: Depending on the specifications of the computer, the loading of a file in which a spatial analysis has been performed may take between about a dozen seconds up to even several minutes. The number of the applied finite elements and the number of nodes used have the greatest impact on the loading time of the file.

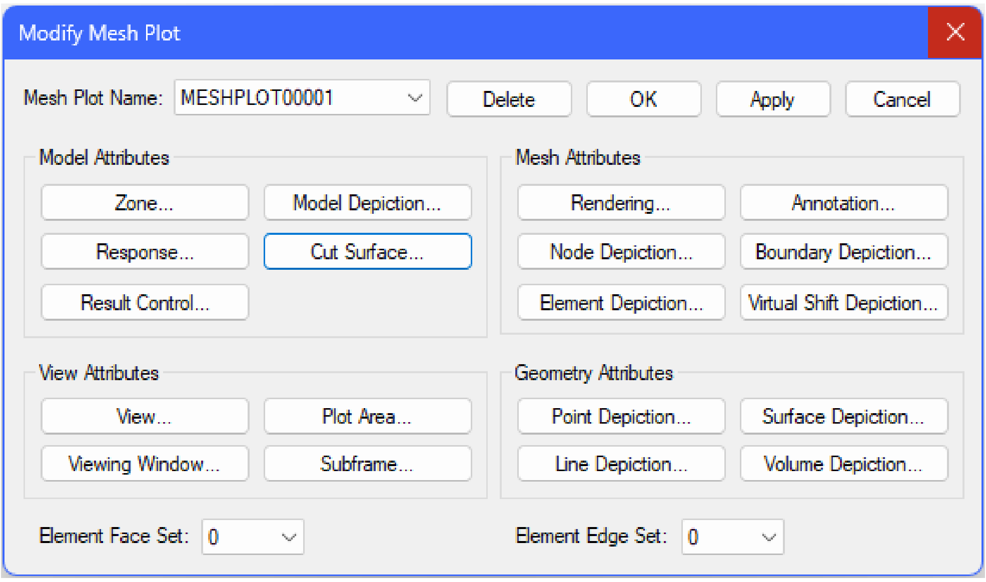
STEP 28. Creating a longitudinal section and a cross-section along with a map of stresses

Initially, two models should be created in a single window. Therefore, the existing model should be shrunk after opening, so as to occupy less than half of the results window. The model is shrunk by activating , then clicking the displayed model with the left mouse button and holding it while making vertical movements with the mouse. Zooming out/in can also be done in another way – by holding the CTRL key along with the left mouse button over the model and moving the mouse vertically. Having properly zoomed out, activate the moving button , and move the model to the upper part of the window. Once these actions are completed, add another model. To do this, click  and also shrink the model using the method described above. Then move the shrunk model to the lower part of the window. Elements with data related to a given model such as the time step and the coordinate system should also be moved closer to a given model. It is also possible to deactivate the view of the finite element mesh, in order to improve the visibility of the maps which will be displayed in the future. According to that, click the arrow next to , and choose the "Group Outline" option. The operation should be performed for both models. A model active at the present moment is indicated by a frame. The post-processor window should look like in the figure below.

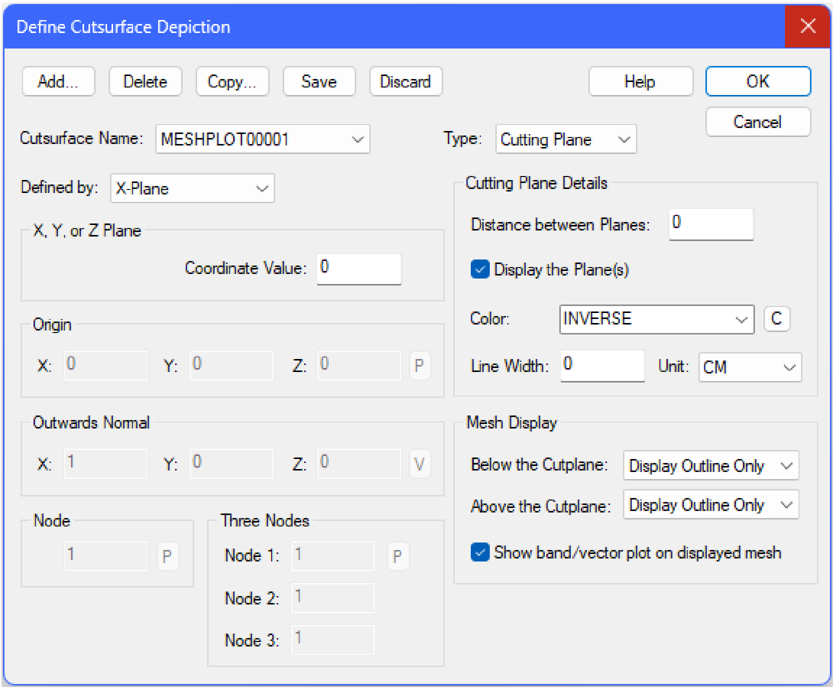


<u>Note</u>: The lines visible along the side surface of the cylinder are lines specifying the discretization of the arc. Although the surface of the ring/circle is displayed as round, in reality, a smooth arc is discretized into straight lines, and the density of dividing a finite element mesh for a given element of a circle/ring determines the number of straight lines making up the arc.

Start with selecting the model present in the upper part of the window. In order to create a longitudinal section, go to "Display → Geometry / Mesh Plot → Modify…", or click , and in the newly opened window check whether the "Mesh Plot Name:" box displays "MESHPLOT00001", which in this case specifies the first default displayed model in the post-processor window, shrunk and moved to the upper part of the window. Upon checking, or selecting "MESHPLOT00001" from the drop-down list, click the "Cut Surface" button from the "Mesh Plot Name:" box. A view of the "Modify Mesh Plot" window is presented below.



Upon clicking the "Cut Surface" button, a new window will appear. Check whether the "Cutsurface Name:" box displays "MESHPLOT00001", then choose the "Cutting Plane" option from the drop-down list in the "Type:" box. A view of the window is presented in the figure below:



In the same window, there are options related to the settings of a cross-section.

Description of specific options

"Cutsurface Name:" – a setting specifying the displayed model to which the cross--section refers.

"Defined by:" – specifies the manner in which a cross-section is created. The possible choices are planes: "X-Plane"; "Y-Plane"; "Z-Plane", a plane created by means of vectors – "Origin and Normal", a plane created by means of three nodes – "Three Nodes", and a plane defined by means of node and vector – "Node and Normal". Depending on the selected option, the following boxes located below the "Defined by:" function are activated.

"Type:" – allow to specify one of three options – "None" means that none cutting plane will be used, "Cutting Plane" allow to adjust options regarding cutting plane, "Isosurface" allow to set cutting plane with options to display/not display a given range of results in a cut plane.

The following settings are available in the "Cutting Plane Details" group of options (details related to the cross-section):

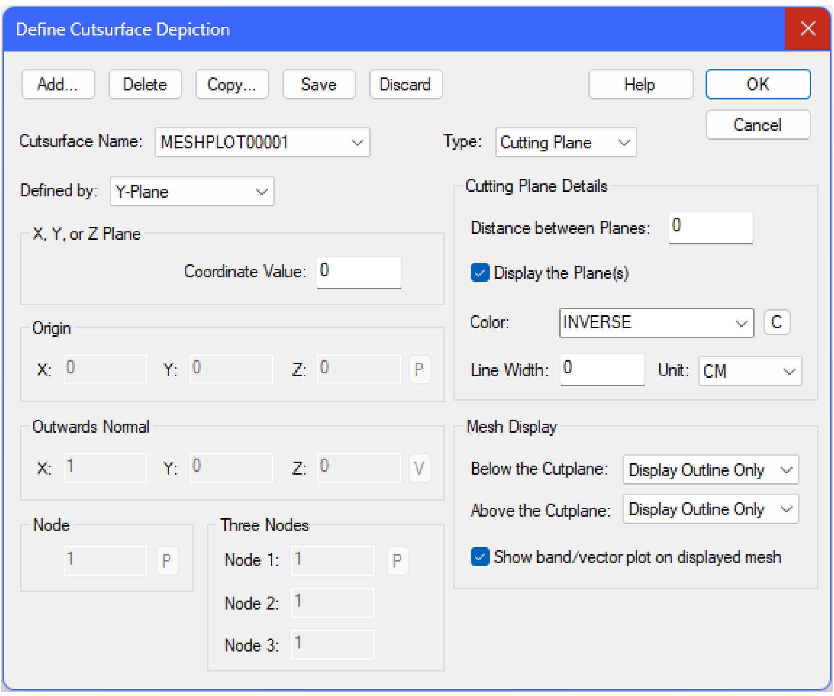
– "Distance between Planes" – an option specifying the distance between the consecutive cross-sections. Pay attention to the value adopted while declaring this function, since, for example, for a bar with a length of 1.00 m when a value of 0.25 is entered in the box, cross-sections will be displayed in the 0.00 m, 0.25 m, 0.50 m (the basal cross-section), 0.75 m, and 1.00 m points.

Note: In order to be able to create two independent cross-sections for one model and being able to move them across window separately, first take care to display two identical models, then declare a cross-section for each of them separately.

– “Display the Plane(s)” – an option determining the ability to display cross-sections.
– “Color” – allows for changing the color of the displayed lines, surfaces, etc., for a given cross-section.
– “Line width:” – defines the line thickness.
– “Unit:” – specifies the units in which the line thickness is displayed.

The “Mesh Display” group of options displays the finite element mesh:
– “Below the Cutplane:” – specifies the ability to display the finite element mesh below the cutplane.
– “Above the Cutplane:” – specifies the ability to display the finite element mesh above the cutplane.
– “Show band/vector plot on displayed mesh” – specifies whether to show the resultant map or a vector map on the displayed mesh.

Since a longitudinal section should be prepared in the present example, the model should be cut by the XZ plane. All it takes to do is to change the “Defined by:” option from “X-Plane” to “Y-Plane” in the window, leaving “Coordinate Value:” as “0”, since the beginning of the coordinate system is located at the beginning of the bar axis. A view of the window along with the input data is presented below.



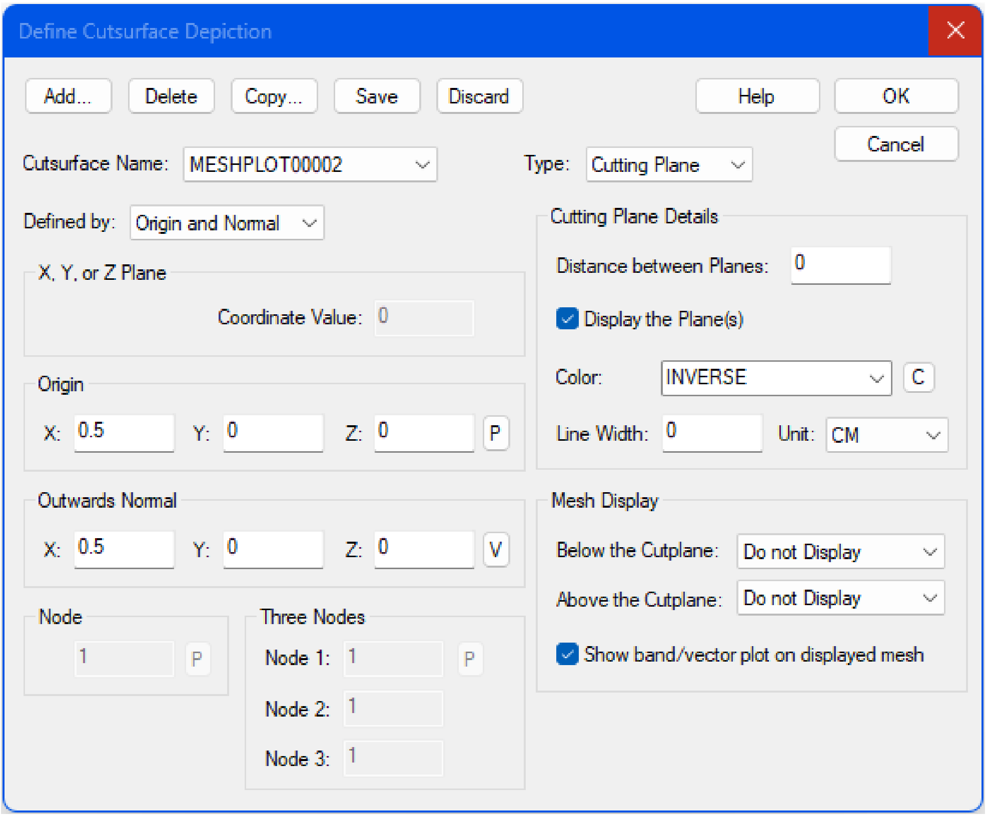
Then click the “Save” and “OK” buttons. Upon returning to the “Modify Mesh Plot” window, click the “Apply” button.

Without closing the “Modify Mesh Plot” window, in the “Mesh Plot Name:” option choose “MESHPLOT00002” from the drop-down list, and click the “Cut Surface” button again. In the newly opened window, make sure that “MESHPLOT00002” is shown in the “Cutsurface Name:” box, and choose “Cutting Plane” from the drop-down list in the “Type:” box. Once editable options are displayed in the same window, create a cross-section located in the middle of the presented bar, meaning at 0.50 m. A method of creating a section on the basis of a plane defined by vectors will be discussed for the purpose of the present example (of course, a method defining it by a plane at specified coordinate value would be faster and easier; however, other solutions should also be demonstrated). The definition of a plane by vectors provides the ability to arrange it in any direction in space.

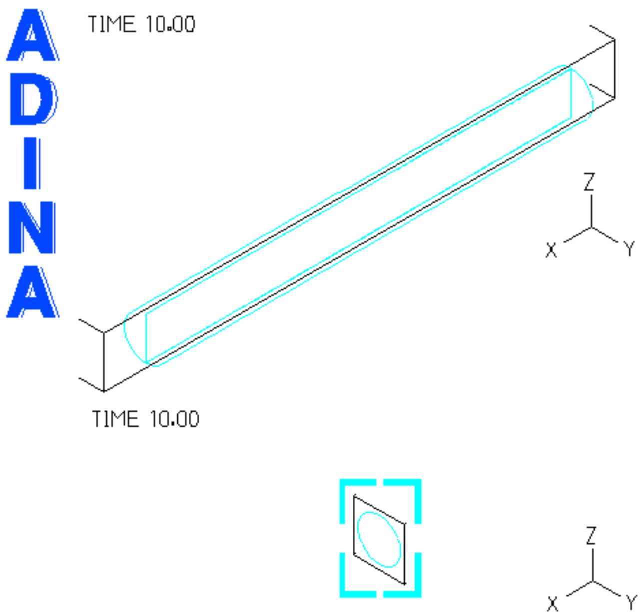
Once editable options have been displayed enter the following data:

Cutsurface Name:	MESHPLOT00002
Type:	Cutting Plane
Defined by:	Origin and Normal
Origin	
X:	0.50
Y:	0
Z:	0
Outwards Normal	
X:	0.50
Y:	0
Z:	0
Cutting Plane Details	
Distance between Planes:	0
Display the Plane(s)	Checked
Color:	Inverse
Line Width:	0
Unit:	CM
Mesh Display	
Below the Cutplane:	Do not display
Above the Cutplane:	Do not display
Show band/vector plot on displayed mesh	Checked

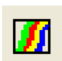
This means that the plane will be created in accordance with the Y and Z axes in a point distant by 0.50 m from the beginning of the global coordinate system, in accordance with the X axis. Also the mesh plot will not be displayed both before and after the cutting plane. A view of the window along with the input data is presented in the figure below.

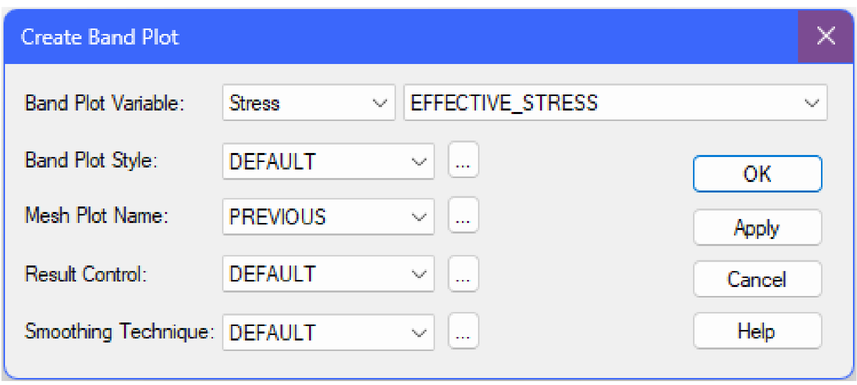


Upon inputting all the data, click the “Save” and “OK” buttons, and upon returning to the “Modify Mesh Plot:” window click “Apply” and “OK”. The main results window should look like this:

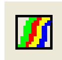


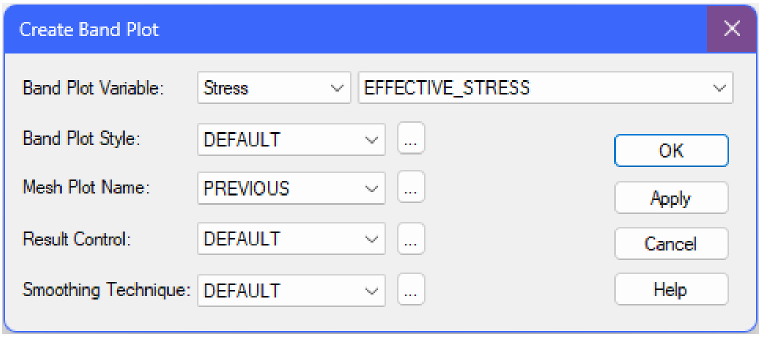
Subsequently, change the view for the upper model. Select it by clicking it once with the left mouse button, and then choose "XZ+ View" from the options next to  (by clicking the arrow). The XZ plane of the cross-section should be presented.

The next step is to create the resultant map. In this case, a map of effective stresses will be created. In order to create the map, go to "Display → Band Plot → Create…", or click . In the newly opened window, for the "Band Plot Variable:" option choose "Stress" from the first drop-down list and "EFFECTIVE_STRESS" from the second one, and click the "OK" button. A view of the window along with the selected maps is presented below.

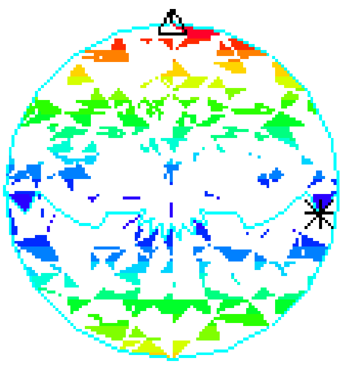


Once a map of stresses has been displayed for the longitudinal section, it is possible to move the legends related to this model to a more convenient place.

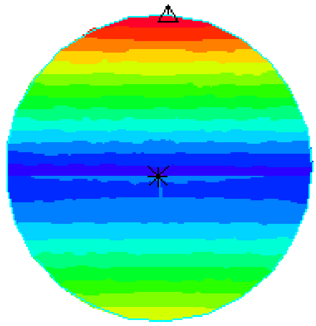
The next step is to select the model located in the lower part of the window with the left mouse button. Subsequently, click the arrow next to , and choose "YZ View" from the options to directly show the generated cross-section of the bar. Once again, create a map of effective stresses. In order to create the map, go to "Display → Band Plot → Create…", or click . In the newly opened window, for the "Band Plot Variable:" option choose "Stress" from the first drop-down list and "EFFECTIVE_STRESS" from the second one, and click the "OK" button. A view of the window along with the selected maps is presented below:



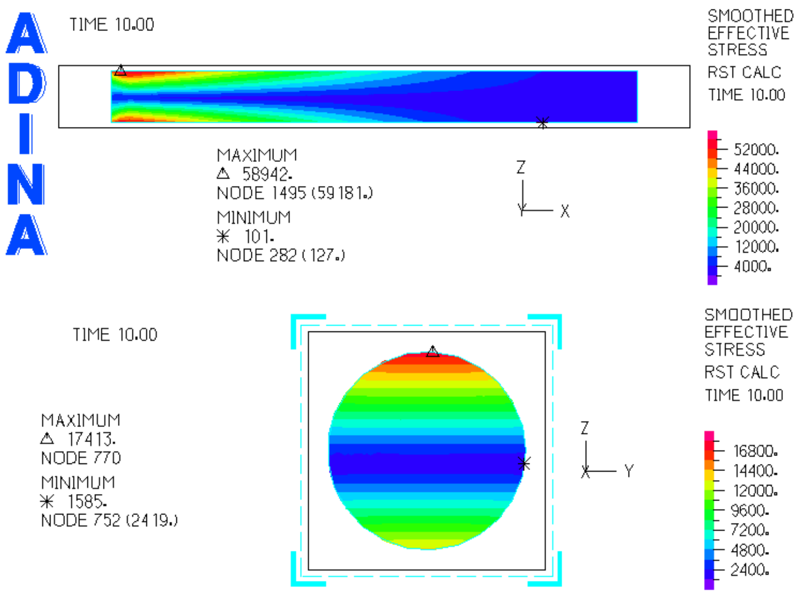
<u>Note</u>: Once effective stresses have been confirmed for a cross-section, a map of these stresses (in older ADINA versions) may appear in the form presented in the figure below:



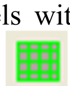
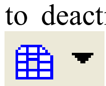
This is a display error in the program; in order to display the map properly, rotate this cross-section slightly by means of . After the rotation, the map on the cross-section should look similar to this:



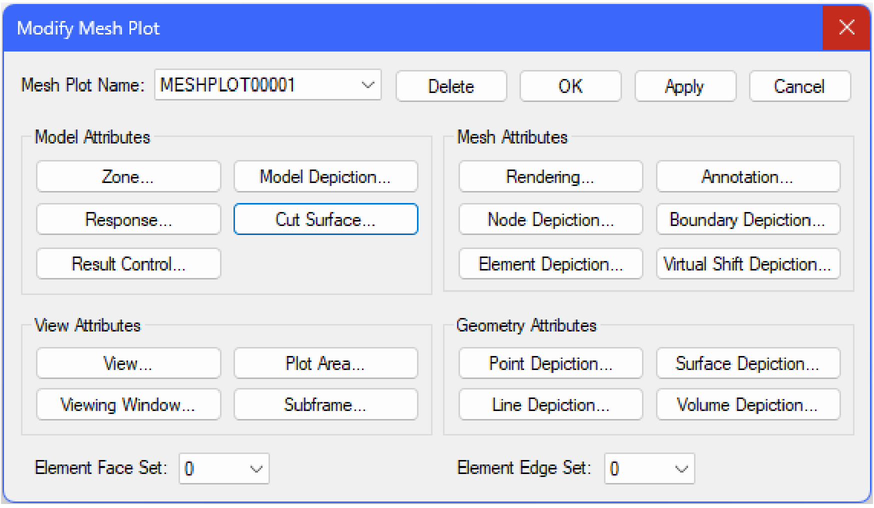
The displayed maps of stresses should still be smoothened by activating  for both presented cross-sections. When all the descriptions have been cleaned up, the main results window should look like this:



STEP 29. Creating several cross-sections in a single model

In order to present the importance of the option which creates several cross-sections within a single resulting model, it is recommended to clear all the previously created models with . Subsequently, activate the view of a single resulting model with . The next step is to deactivate the view of the finite element mesh by clicking the arrow next to  and selecting the "Model Outline" option. Once these operations are completed, set the time as "Time 5.000" by means of

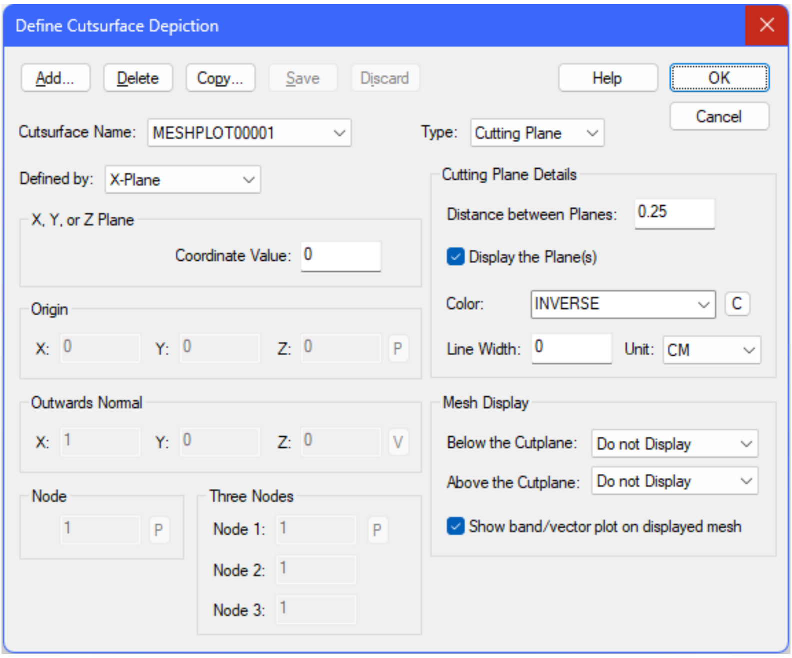
. Then go to “Display → Geometry / Mesh Plot → Modify…”, or click . A window will open like the one presented in the figure below.



In the newly opened window cut-planes created in the present example will be spaced apart by 0.25 m along X-axis; therefore, a value of “0.25” should be entered in the “Distance Between Planes:” option and the “X-Plane” should be chosen for “Defined by:” option. Following data should have been introduced in the “Define Cutsurface Depiction” window:

Cutsurface Name:	MESHPLOT00001
Type:	Cutting Plane
Defined by:	X-Plane
X, Y, or Z Plane	
Coordinate Value:	0
Cutting Plane Details	
Distance between Planes:	0.25
Display the Plane(s)	Checked
Color:	Inverse
Line Width:	0
Unit:	CM
Mesh Display	
Below the Cutplane:	Do not display
Above the Cutplane:	Do not display
Show band/vector plot on displayed mesh	Checked

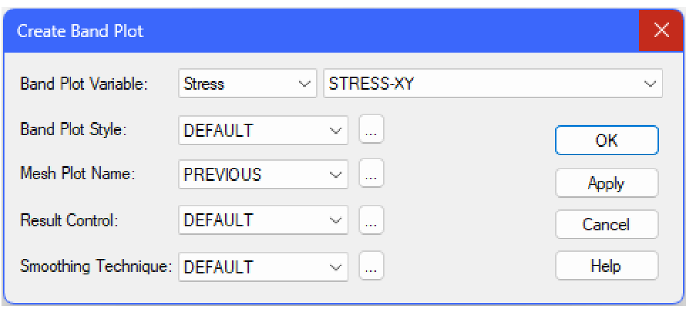
The window with introduced data is presented below:



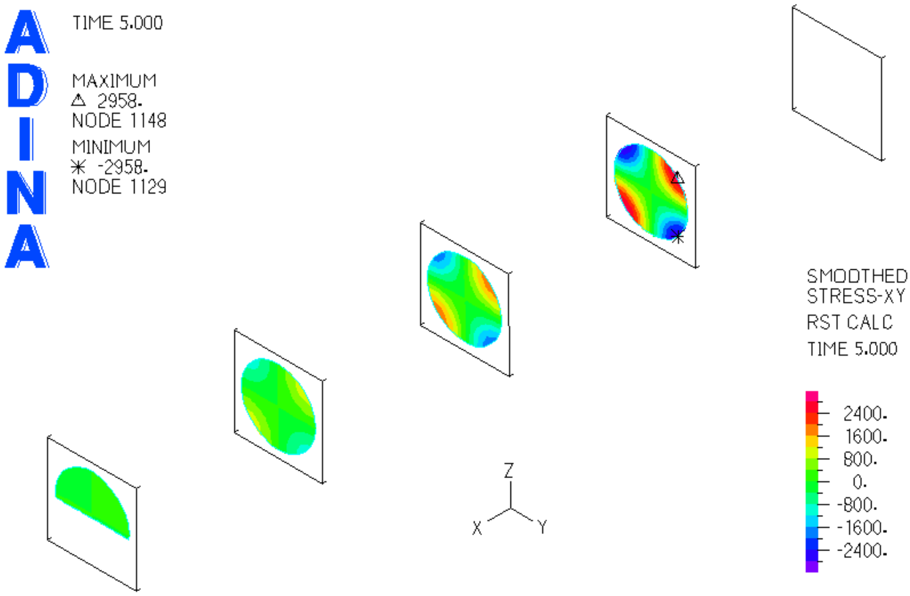
Then confirm the input data with the “Save” button, and click “OK” to close the window.

Upon returning to the “Modify Mesh Plot” window, click the “Apply” button, and then “OK”.

In the results window, one more map will be created for tangential XY stresses. To this end, go to “Geometry → Band Plot → Create…”, or click . Subsequently, in the newly opened window, the first drop-down list for the “Band Plot Variable” option must display “Stress”, with “STRESS-XY” chosen from the drop-down list to the right. Upon inputting the data, click the “OK” button. A view of the window along with the introduced changes is presented in the figure below.

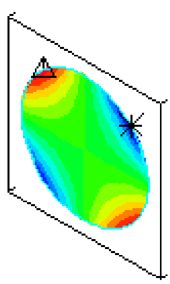


When the maps have been displayed, it is still necessary to activate smoothing of the presented maps by activating . As a result of the performed actions, the model should look like this:

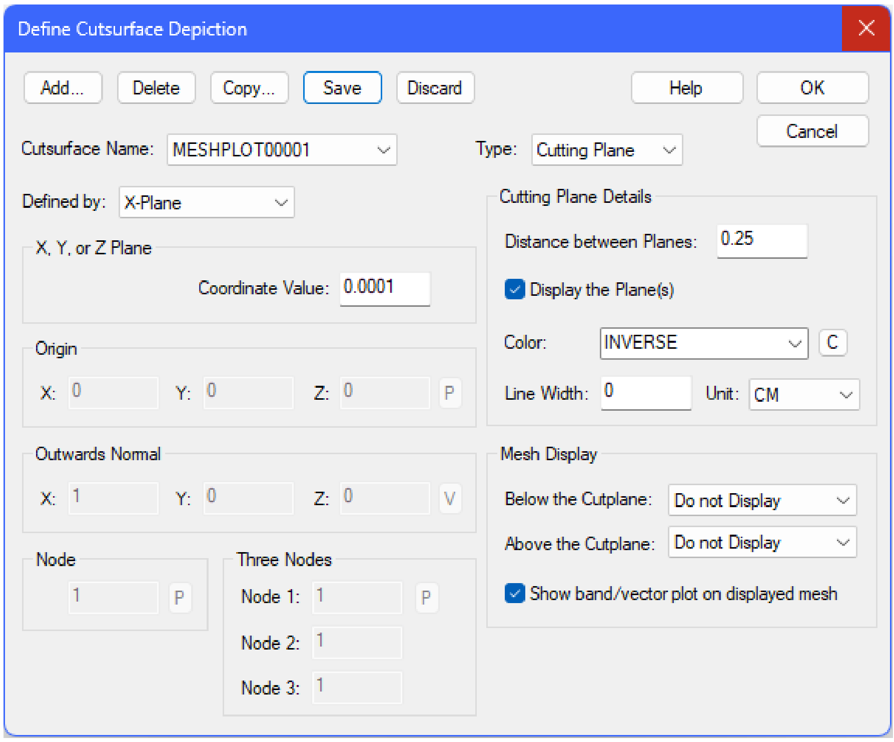


Note: The “abrupt end” of the results for the surface located at the bottom of the figure above, and the lack of results for the surface located at the top, are related to the improper displaying of the outermost cutting-planes by the program. This should also be kept in mind when defining a single cross-section prepared on one end of the bar. In such cases, when the cross-section has not been displayed, enter a value close to 0 in the “Coordinate Value” box.

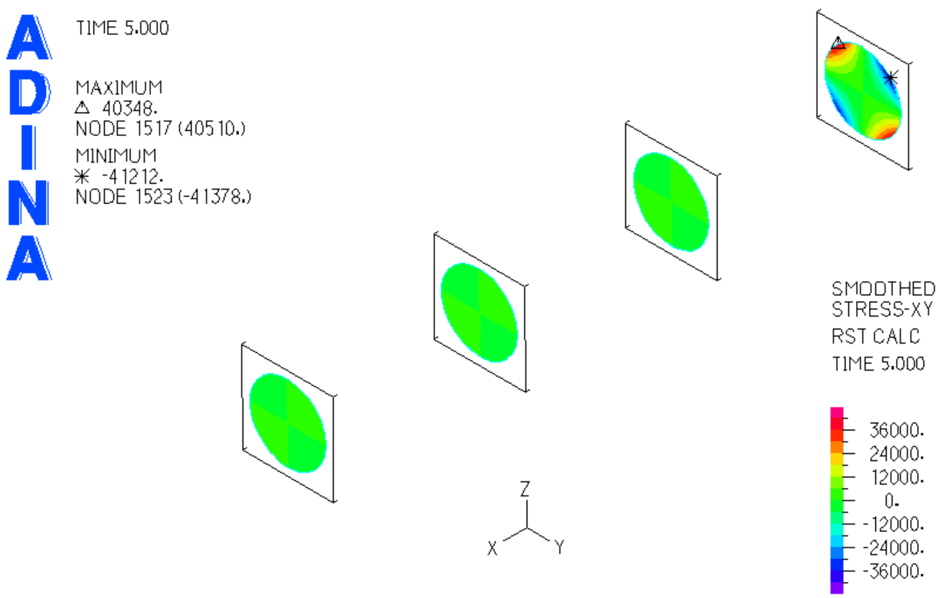
The figure below presents a cross-section which was not displayed earlier (the top right cross-section).



The baseline cross-section was set to be placed 0.0001 m from the beginning of the coordinate system. A window with the defined values is presented below.



As can be seen, in the figure below there are already only 4 cross-sections. This is due to the fact that the fifth cross-section would be placed at a distance of 1.001 m, meaning beyond the assumed length of the bar.



EXAMPLE 5
THE SPATIAL SHELL CONSTRUCTION. CALCULATIONS INCLUDING THE IRREGULAR SHAPE OF SURFACE LOAD

In this example activities related to modeling of bending of a shell fixed to the ground in a three-dimensional Cartesian system (3D) are demonstrated. The reader is advised to refer to the previous examples as some of the features have been discussed earlier. The diagram of the analyzed model is shown in Figure 23.

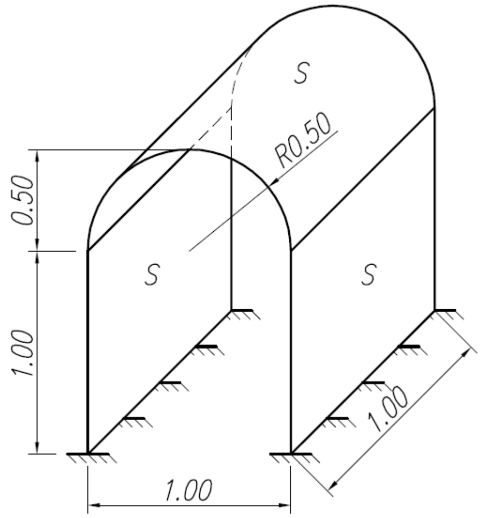


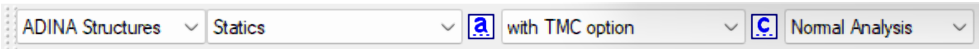
Fig. 23. Scheme of the three dimensional shell

The following data is used in the analysis:

– load:
  pressure $q = 1000\ \mathrm{N/m}$
– material constants:
  steel S235JR:
  $E = 210\ \mathrm{GPa} = 2.1\mathrm{e}^{11}\mathrm{Pa}$
  $\upsilon = 0.30$
  $\rho = 7859\ \mathrm{kg/m^3}$
– boundary conditions:
  full fixity (6 degrees of freedom blocked) along the entire length at the line of contact between shell wall and the ground

STEP 1. Definition of the type of analysis

Upon opening the ADINA software, choose “ADINA Structures” from the “Module Bar” in the “Program Module” section, and choose “Statics” from the drop-down list next to “Analysis Type”.

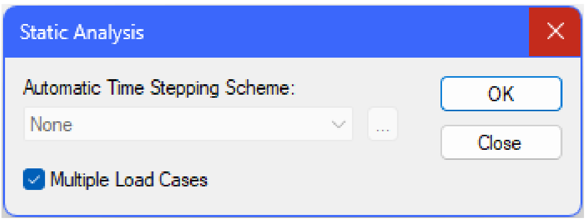


STEP 2. Entering the heading of the model

In order to specify a heading, go to “Control → Heading…”. Subsequently, enter the project heading in the text box, e.g., “Shell bending”. Upon entering a heading, click the “OK” button.

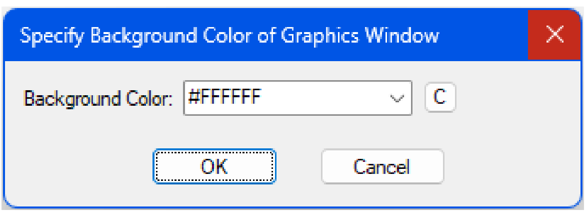
STEP 3. Definition of load combinations

To define a load combination in the model, press the button , then in the newly opened window for the option “Automatic Time Stepping Scheme” select “None” from drop-down list, and after that locate the “Multiple Load Cases” option and check it. After making changes, exit the window by pressing the “OK” button. The window view is shown in the figure below:



STEP 4. Definition of the background color of the main model window

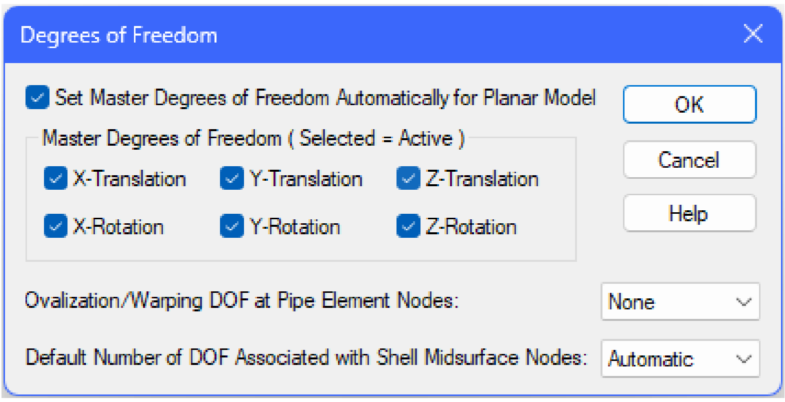
In order to define the background color, go to “Edit → Background Color…”. Then, in the newly opened window choose the color white from the drop-down list. After choosing confirm the choice with the “OK” button.



Note: The background color is not saved along with the model. This means that after each opening of the file, the background color will return to default – “black”.

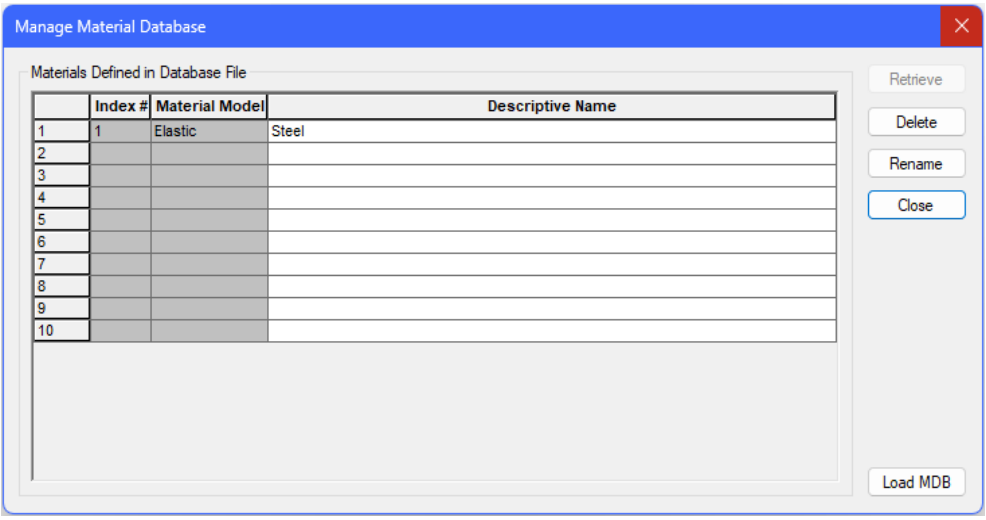
STEP 5. Definition of global boundary conditions

Since the model under consideration is a spatial, it is recommended that all global boundary conditions involved in the modeling and calculation process are active. To check which boundary conditions are active, go to “Control → Degrees of Freedom...”. In the newly opened window, all boundary conditions in the “Master Degrees of Freedom (Selected = Active)” option group should have their boxes checked. The figure below shows the window with selected boundary conditions:



STEP 6. Definition of material constants

In order to define materials constants, go to “Model → Materials → Manage Materials...” or press the button , then in the newly opened window (if the material has been previously added to the program database), press the “Get MDB” button. A new window with a table should open, then select the material called “Steel” in the table and press the “Retrieve” button. After loading the material from the program database, press the “Close” button. In the “Manage Materials” window, in its lower part, in place of the table, there should be a material – “Steel” with the id number “1”. The retrieving material window is presented below:

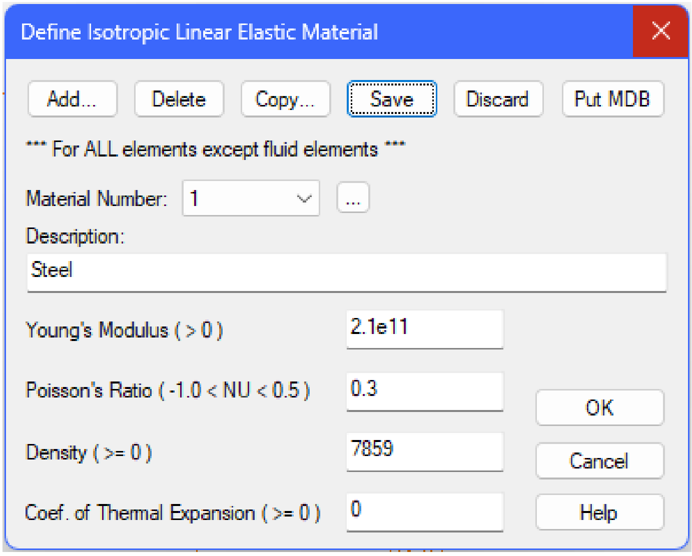


If the database does not contain the material required in the example, in the "Manage Materials" window, find the group relating to elastic materials – "Elastic", and then press the "Isotropic" button – isotropic material. In the newly opened window, press the "Add..." button and enter the data in accordance with the table below:

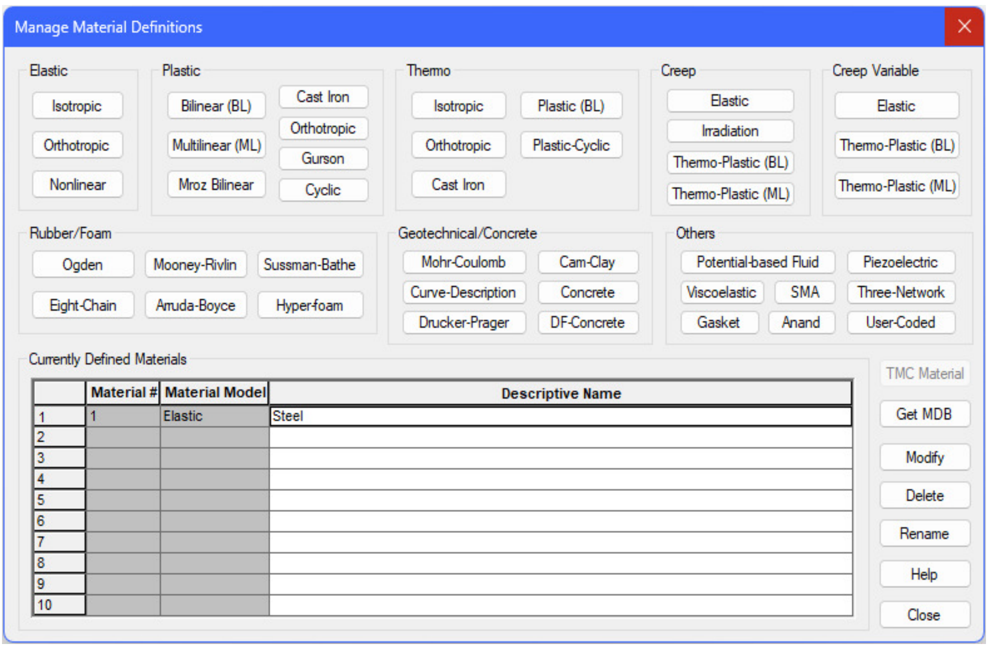
Material Number:	1
Description:	Steel
Young's Modulus (> 0)	2.1e11
Poisson's Ratio (–1.0 < NU < 0.5)	0.30
Density	7859
Coef. of Thermal Expansion (>= 0)	0

After entering the values, press the "Save" button. If the user wants to add material to the program's database, press the "Put MDB" button. After completing the operation, you can exit the window by pressing the "OK" button. Again, as in the previous case, after leaving the window, in the lower part of the "Manage Material Definitions" window, the table should contain the "Steel" material model with an assigned identification number of "1".

The window view with the entered material in the elastic material definition window is shown in the figure below:

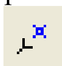


The view of the material manager window should look similarly as:



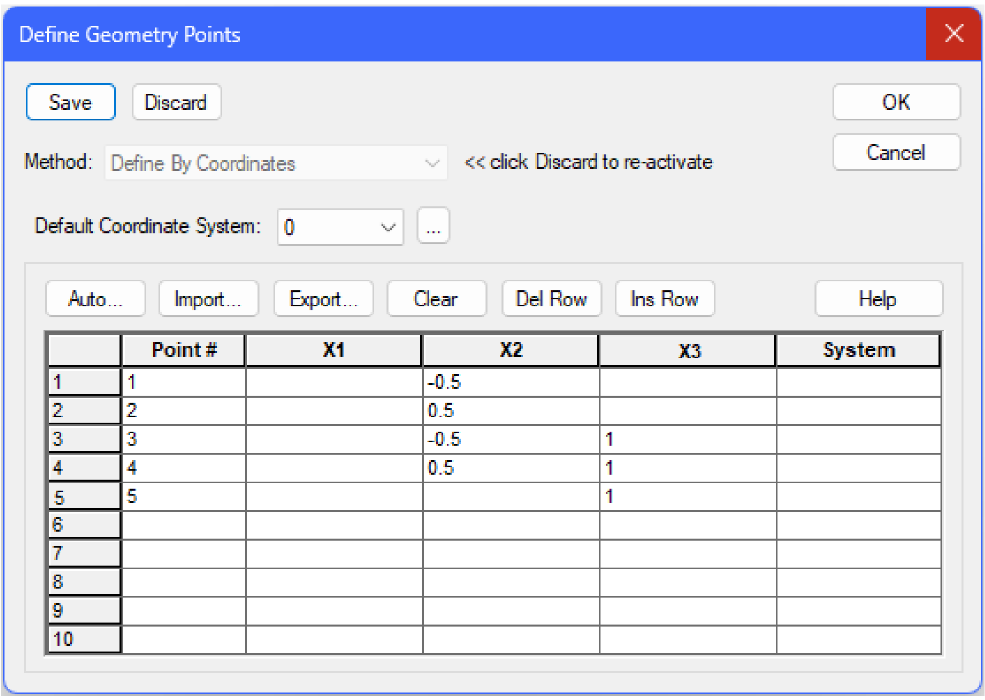
After performing all operations, exit the “Manage Material Definitions” window by pressing the “Close” button.

STEP 7. Definition of points

In this step, auxiliary points will be created to model shell elements describing the given scheme. For this purpose, points will be created on the YZ plane, which will then be extruded. In order to define points, go to "Geometry → Points → Define…" in the main model window, or click . Upon opening a new window, input points in accordance with the table below:

Point #	X1	X2	X3	System
1	0	–0.5	0	0
2	0	0.5	0	0
3	0	–0.5	1	0
4	0	0.5	1	0
5	0	0	1	0

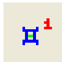
The window view with the entered data is shown in the figure below:



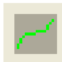
__**Note:**__ There is no need to enter zero values in the table. After pressing the "Apply" button, the table will be automatically completed with these values.

After entering the values from the table, press the “Apply” button and then “OK”.

STEP 8. Displaying point ID numbers

To display point identifiers, press the button  on the toolbar.

STEP 9. Definition of auxiliary lines. Part I

In order to define auxiliary lines for later extrusion to surfaces, go to “Geometry → Lines → Define...” or press the button . When a new window opens, press the “Add...” button to add a new line.

For this step the user needs three lines. Two defining walls and one showing a roof in the form of an arch. At the moment, it is only possible to enter straight lines, but the line representing the arc will be created in a non-standard way by rotating the point around the axis of the local coordinate system later in the study.

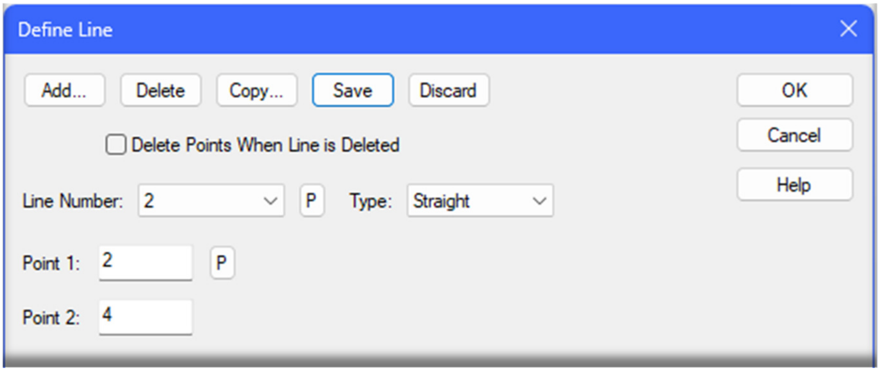
To create the mentioned lines, enter data in accordance with the following tables, taking into account to save the entered data after the first table with the “Save” button, and then add a new line with the “Add...” button. The data to be entered is presented in the tables below:

Line Number:	1
Type:	Straight
Point 1:	1
Point 2:	3

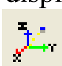
Line Number:	2
Type:	Straight
Point 1:	2
Point 2:	4

After completing the entry, press the “Save” button and exit the window with the “OK” button.

The view of the window defining the line with id number “2” is shown in the figure below:

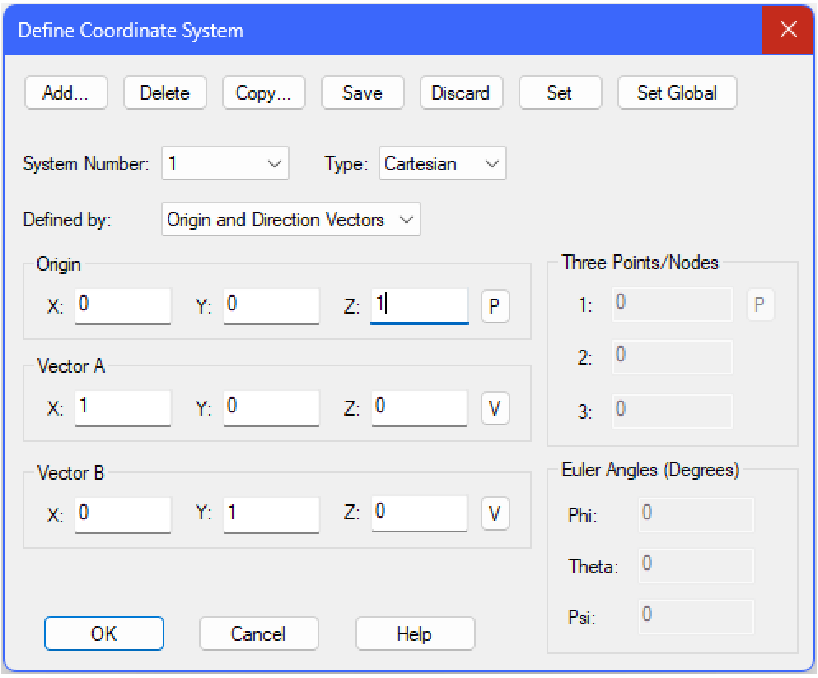


STEP 10. Definition of local coordinate system

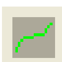
The local coordinate system will be created at a point (0; 0; 1.00), which corresponds to the center point of the arc. This definition will be needed when entering an arc created by rotating a point in regard to the X axis of the defined local coordinate system. To define a local Cartesian coordinate system with the X axis perpendicular to the plane of the displayed model, go to “Geometry → Coordinate Systems...” or press the button . After opening a new window, press the “Add...” button and enter the following data:

System Number:	1
Type:	Cartesian
Defined by:	Origin and Direction Vectors
Origin	
X:	0
Y:	0
Z:	1
Vector A:	
X:	1
Y:	0
Z:	0
Vector B:	
X:	0
Y:	1
Z:	0

After entering the data, press the “Save” button and leave the window with the “OK” button. The window view with the entered data is presented in the figure below:

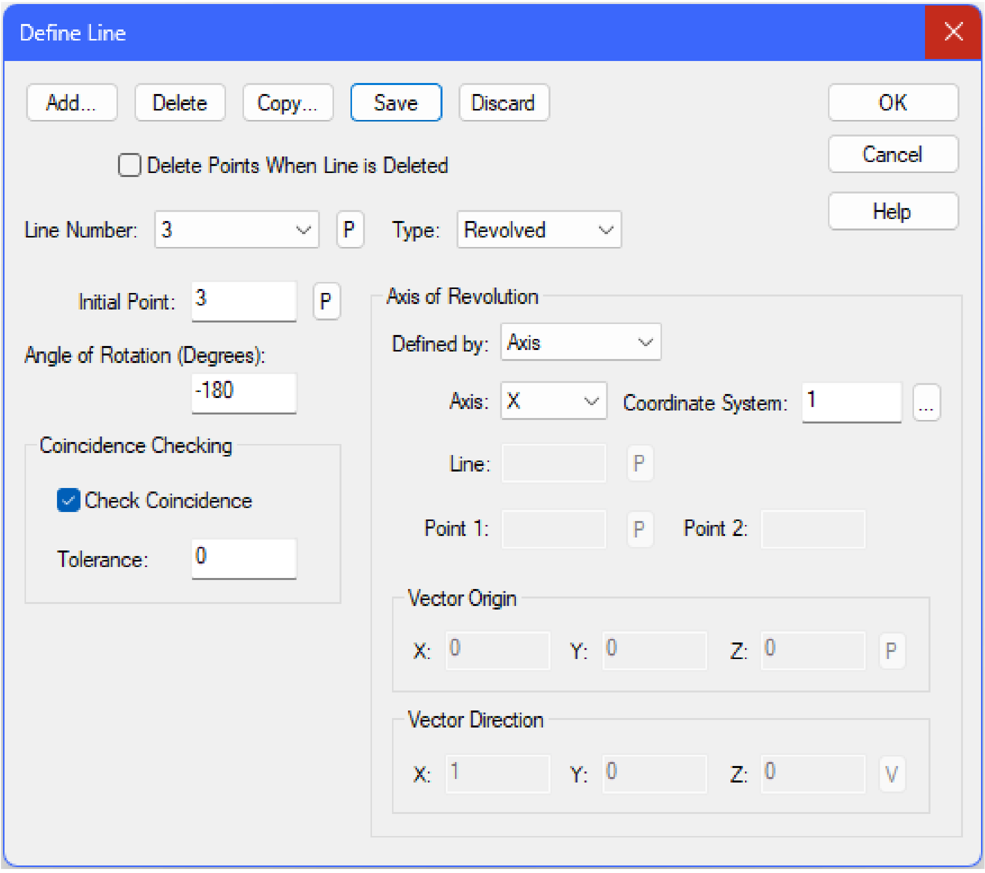


STEP 11. Definition of auxiliary lines. Part II

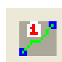
Since a local coordinate system has been defined, it is possible to rotate point “P3” around the X axis of that system. To define an arc, go back to the line definition window by selecting “Geometry → Lines → Define...” from the menu or pressing the button . In the newly opened window, press the “Add...” button and enter the following data:

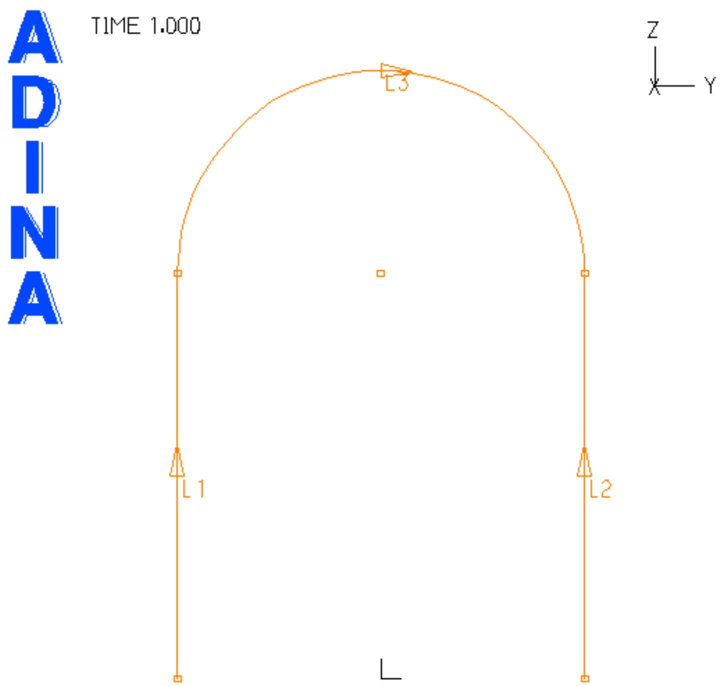
Line Number:	3
Type:	Revolved
Initial Point:	3
Angle of Rotation:	–180
Axis of Revolution	
Defined by:	Axis
Axis:	X
Coord. System:	1

After entering the data, press the “Save” button and then leave the window with the “OK” button. The window view with the entered data is shown in the figure:

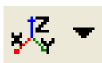


STEP 12. Displaying line ID numbers

To display the line identification numbers, locate and press the button  on the toolbar. If the model does not fit entirely in the main program window, press the button . Currently the model should look like this:



STEP 13. Model view adjustment

To change the view (where the model is observed), left-click on the arrow next to the button  and then select "Iso View 1" from the available options.

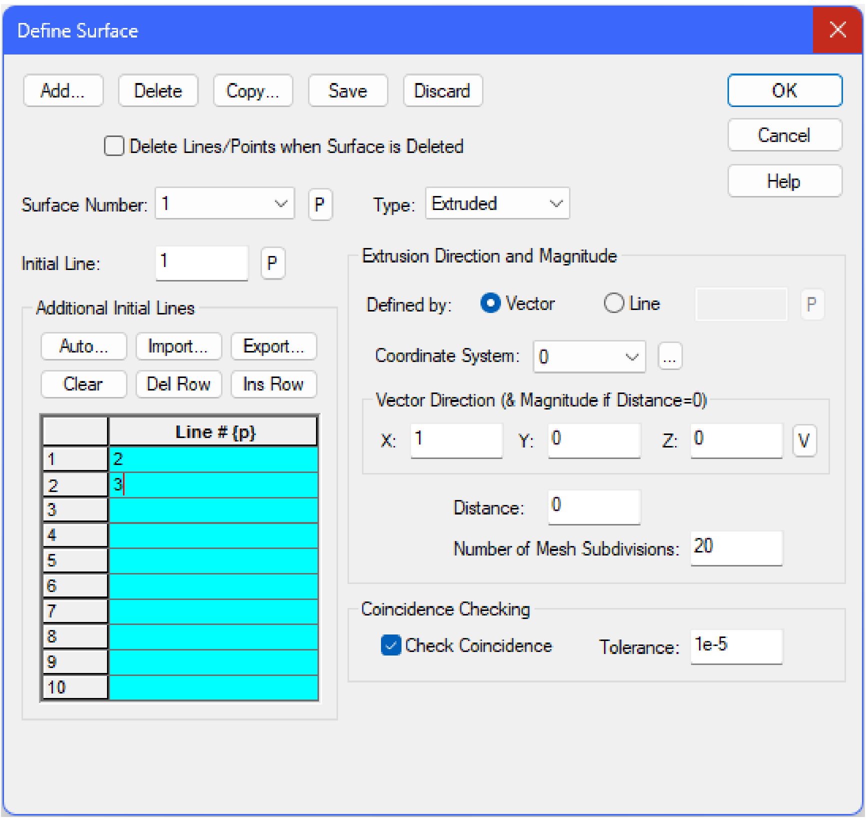
STEP 14. Definition of surface(s)

In this step, all three defined lines will be extruded by 1.00 m in the direction of the X axis. To go to the surface definition, select "Geometry → Surfaces → Define..." from the menu or press the button . In the newly opened window, press the "Add..." button and enter the following data:

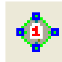
Surface Number:	1
Type:	Extruded
Initial Line:	1
Extrusion Direction	
Defined by:	Vector
Coordinate System:	0

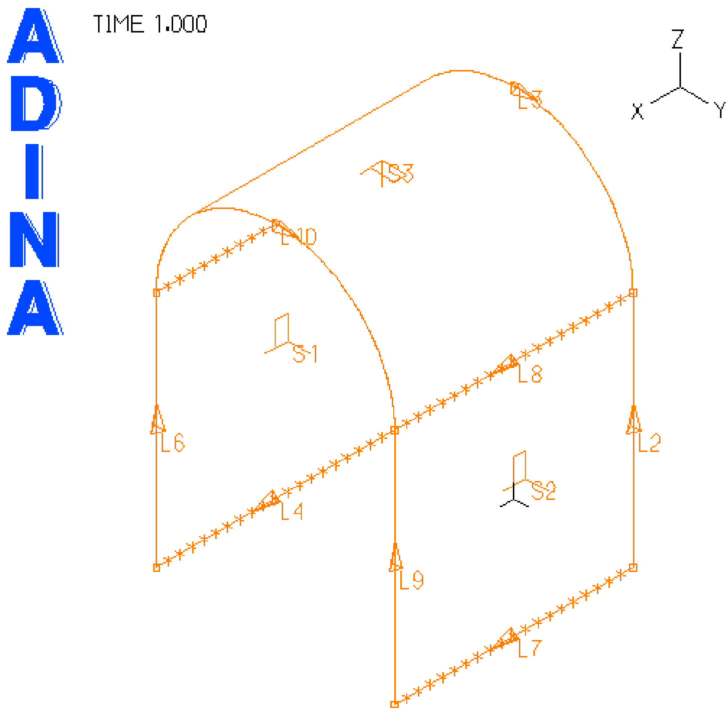
Vector	
X:	1
Y:	0
Z:	0
Number of Mesh Subdivisions:	20
Coincidence Checking	
Check Coincidence	checked
Tolerance	1e-005
Additional Initial Lines	
	Line {p}
1	2
2	3

After entering the data, press the "Save" button and then exit it with the "OK" button. The window view with the entered data is shown in the figure below:



STEP 15. Displaying surface(s) ID numbers

To display surface identification numbers, locate and press the button  on the toolbar. Currently the model should look like this:



STEP 16. Mesh subdivision

In this step, subdivision parameters for lines that have not yet been subdivided for finite elements are defined. First, vertical lines will be divided, then arcs. To define the mesh subdivion under finite elements, go to “Meshing → Mesh Density → Line...” or press the arrow next to the button  and select “Subdivide Lines”. In the newly opened window, enter the following data:

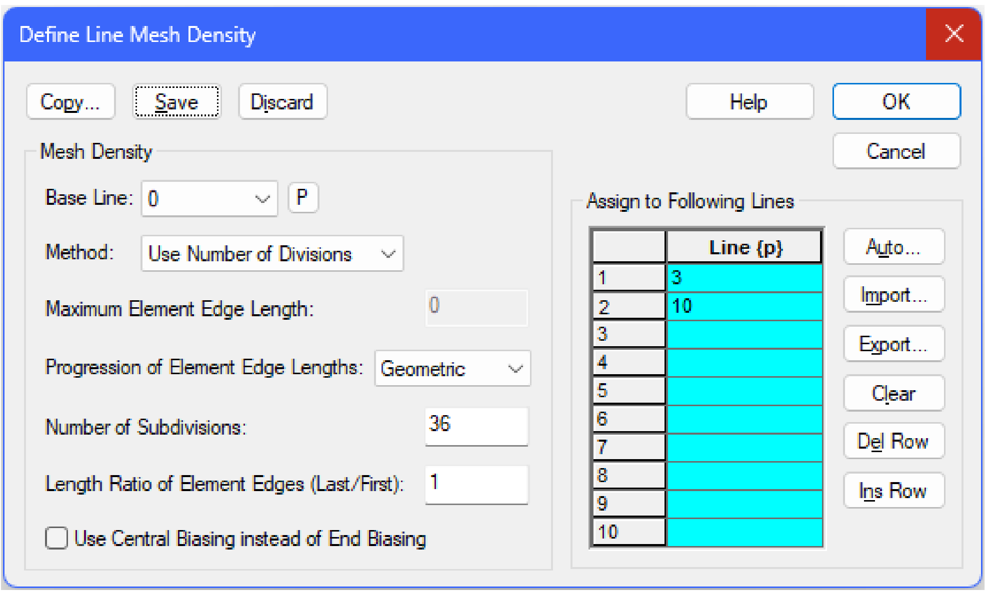
Mesh Density	
Base Line:	0
Method:	Use Number of Divisions
Progression of Element Edge Lengths:	Geometric
Number of Subdivisions:	20
Length Ratio of Element Edges (Last/First):	1
Use Central Biasing instead of End Biasing	Unchecked
Assign to Following Lines	
	Line {p}
1	1
2	2
3	6
4	9

The window view with the entered data is shown in the figure below:

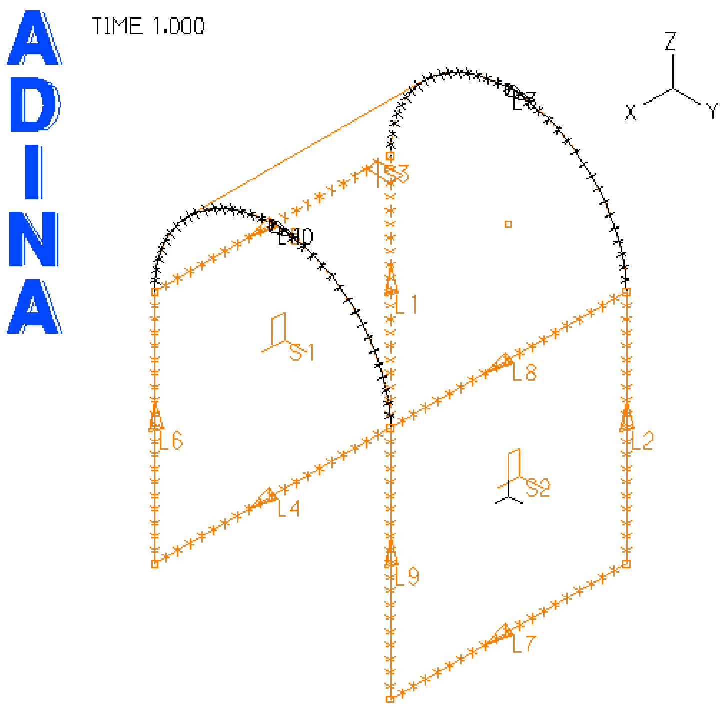
After entering the data, press the "Save" button, then define the mesh subdivision for the lines forming the arcs. To do this, enter the following values in the "Define Line Mesh Density" window:

Mesh Density	
Base Line:	0
Method:	Use Number of Divisions
Progression of Element Edge Lengths:	Geometric
Number of Subdivisions:	36
Length Ratio of Element Edges (Last/First):	1
Use Central Biasing instead of End Biasing	Unchecked
Assign to Following Lines	
	Line {p}
1	3
2	10

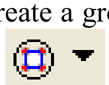
After entering the data, press the "Save" button and then leave the window with the "OK" button. The window view with the entered data is shown in the figure below:



Currently, the model should look similar to that:

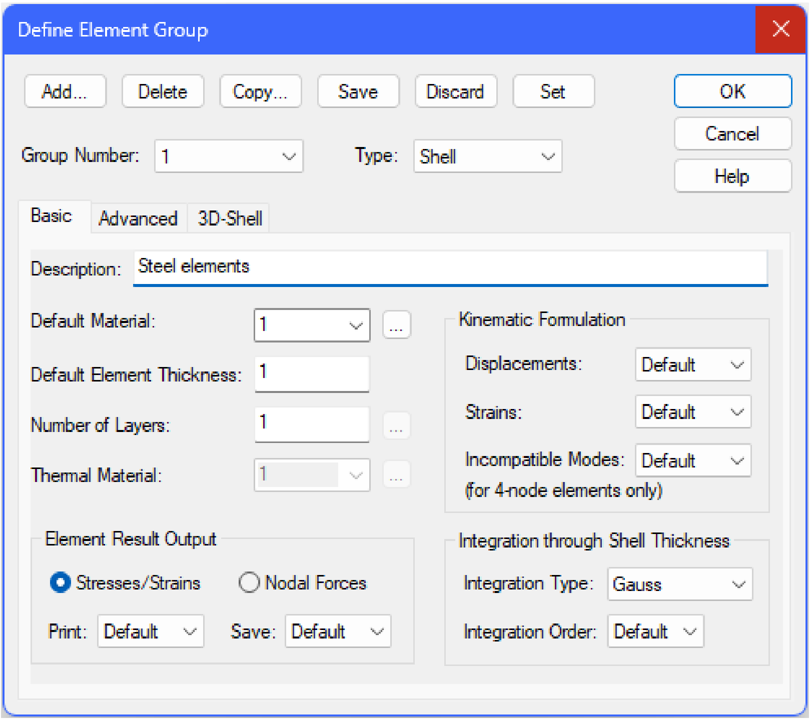


STEP 17. Specifying the type of analyzed construction

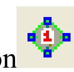
In this example, there is only one material model, so user can define a single group of elements describing the shell. To create a group of elements, go to “Meshing → Element Groups...” or press the button . In the newly opened window, press the “Add...” button and then enter the following data:

Group Number:	1
Type:	Shell
“Basic” tab	
Description:	Steel elements
Default Material:	1
Default Element Thickness:	1
Number of Layers:	1

The remaining data is left unchanged. Save the entered data by pressing the “Save” button and then exit the window by pressing the “OK” button. The window view with the entered data is shown in the figure below:



STEP 18. Displaying surfaces ID numbers

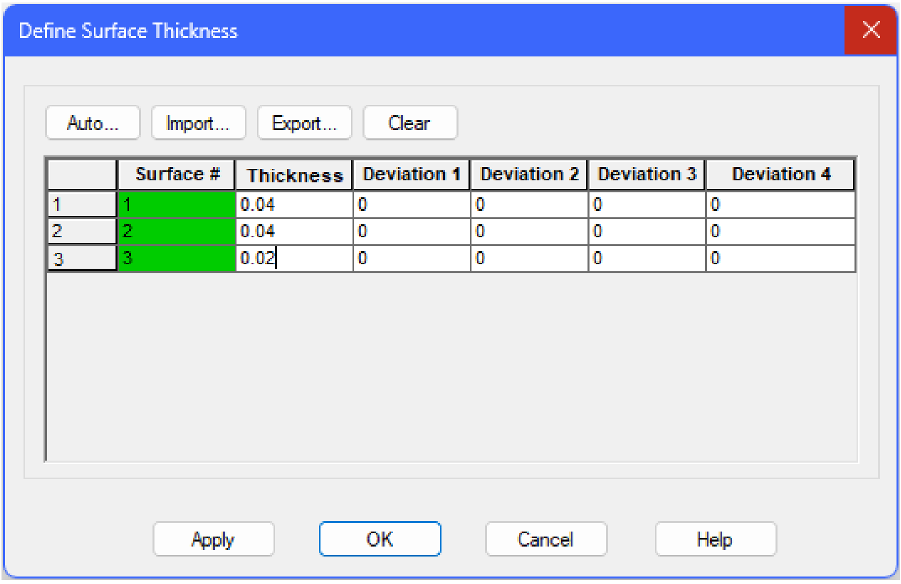
In order to display the surface identification numbers, press the button  on the toolbar.

STEP 19. Defintion of surface(s) thickness

To define the thickness for each surface separately, go to “Geometry → Surfaces → Thickness...”. In the newly opened window, enter the following data:

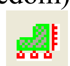
	Surface #	Thickness	Deviation 1	Deviation 2	Deviation 3	Deviation 4
1	1	0.04	0.0	0.0	0.0	0.0
2	2	0.04	0.0	0.0	0.0	0.0
3	3	0.02	0.0	0.0	0.0	0.0

After entering the above data, press the “Apply” button and then leave the window with the “OK” button. The window view with the entered data is shown in the figure below:



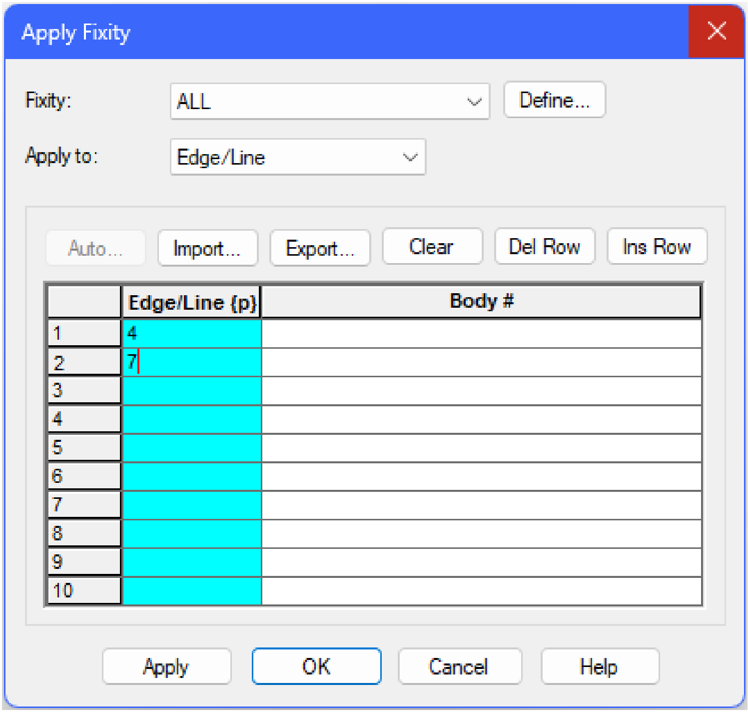
<u>Note</u>: Although in the window regarding the definition of a group of elements, the default thickness of elements was set as "1", defining the thickness via "Geometry → Surfaces → Thickness..." is the master option. This means that after changing the default values in the "Define Surface Thickness" window, the entered values will be overwritten onto the "Deafult Shell Element Thickness" entered in the "Define Element Group" window.

STEP 20. Definition of boundary conditions (fixity characteristics)

To define the line supports (clamped line – blocking 6 degrees of freedom), go to "Model → Boundary Conditions → Apply Fixity..." or press the button . In the newly opened window, in the "Apply to:" option, select "Lines" from the drop-down list. Since the example includes clamped support, and the program has a declared fixity with all degrees of freedom fixed by default (fixity name: "All"), there is no need to declare the fixity once again.. In the "Apply Fixity" window open, enter the following data:

Fixity:	ALL
Apply to:	Edge/Line
	Edge/Line {p}
1	4
2	7

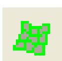
After entering the data, press the "Save" button and then leave the window with the "OK" button. The window view with the entries is shown in the figure below:



STEP 21. Displaying boundary conditions

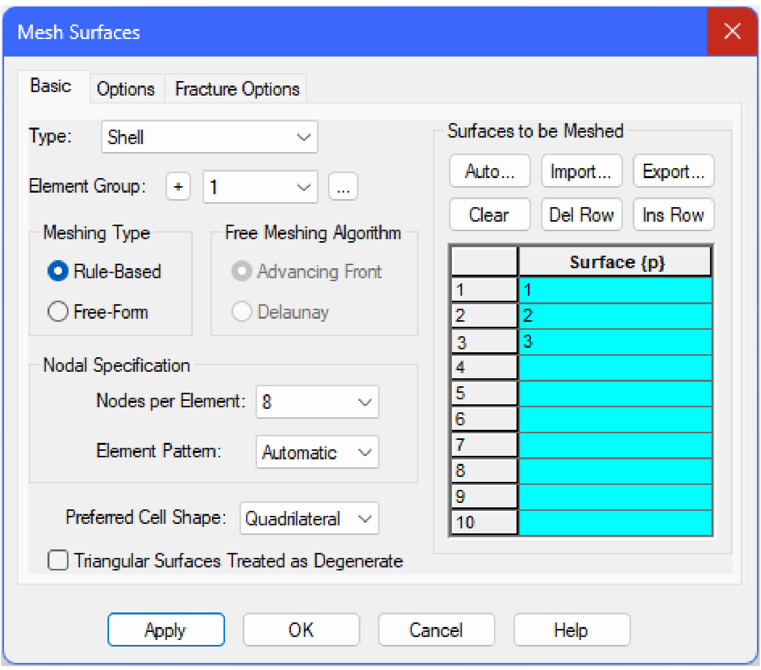
In order to display the defined boundary conditions in the main model window, click  button.

STEP 22. Definition of finite elements

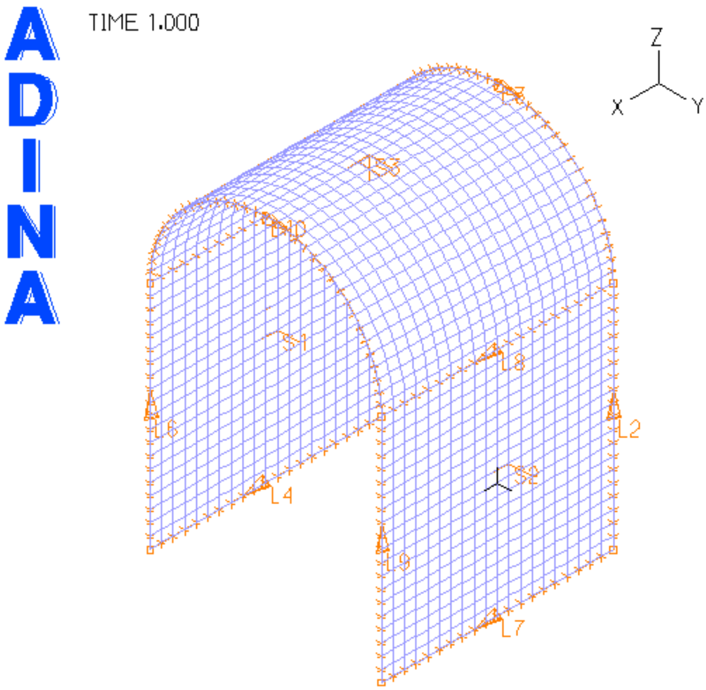
To define finite elements, go to "Meshing → Create Mesh → Surface..." or press the button . When a new window opens, enter the following data:

“Basic” tab	
Type:	Shell
Element Group:	1
Meshing Type	
Rule-Based	Checked
Nodal Specification	
Nodes per Element:	8
Element Pattern	Automatic
Preferred Cell Shape	Quadrilateral
Triangular Surfaces Treated as Degenerate	Unchecked
Surfaces to be Meshed	
	Surface {p}
1	1
2	2
3	3

The remaining cards, i.e. ”Advanced” and “Fracture Options”, are left unchanged. The window view with the entered data is shown in the figure below:

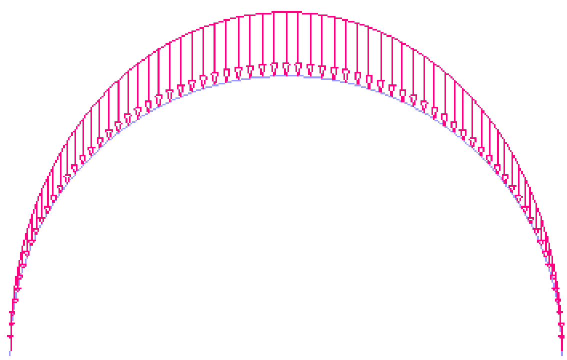


After entering the data, press the “Apply” button and then leave the window with the “OK” button. Currently the model should look like this:



STEP 23. Definition of load variability

In this step, a spatial function will be created to describe the variability of the load value distributed on the plane. To define the load function, go to “Geometry → Spatial Functions → Surface...”, then press the “Add...” button in the newly opened window. The purpose of this introduction is to obtain a load that will have the maximum value in the center of the arch's symmetry, while at the edges of the arch the load value is to be 0. This relationship is shown in the figure below:



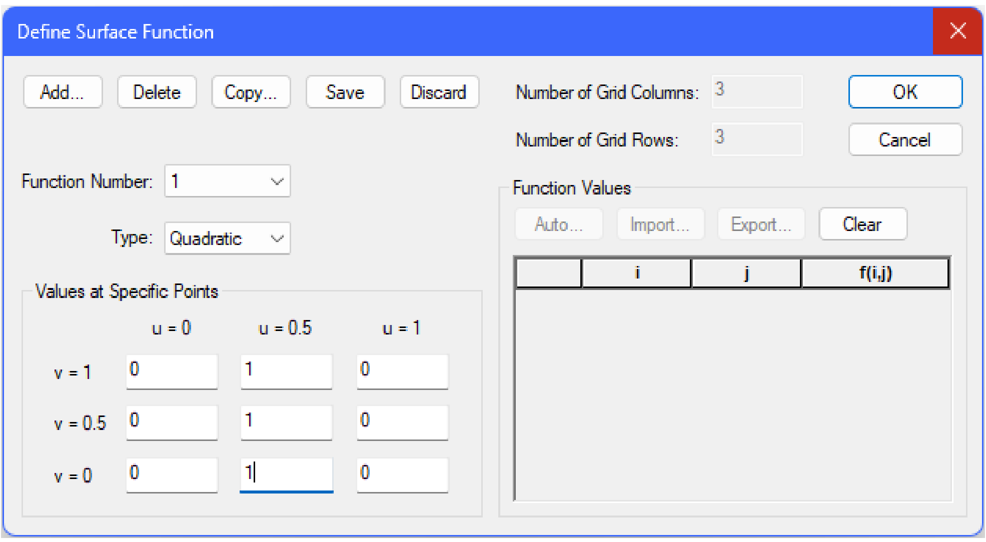
To obtain such a load curve, enter the following data in the "Define Surface Function" window:

Function Number:	1
Type:	Quadratic

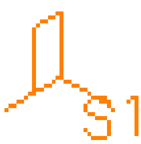
The table in the window should be completed with the following data:

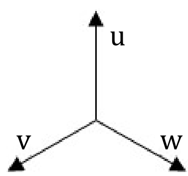
Values at Specific Points			
	u = 0	**u = 0.5**	**u = 1**
v = 0	0	1	0
v = 0.5	0	1	0
v = 1	0	1	0

After entering the data, press the "Save" button, then you can close the window with the "OK" button. The window view with the entered data is shown in the figure below:



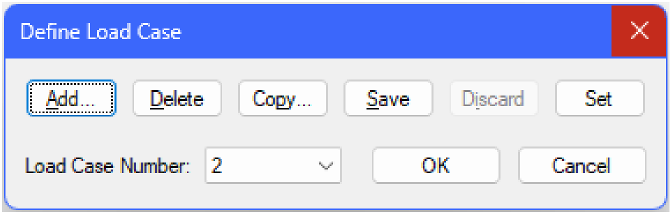
If the user is not sure which direction is referred to as "u" and which is referred to as "v", the figure below should clear up any doubts:





STEP 24. Definition of load cases

In this example, there will only be two load cases. To go to the case definitions, select “Model → Loading → Load Case...” from the menu. Then, in the newly opened window, press the “Add...” button twice. The window view is shown in the figure below:



After entering the data, press the “Save” button and close the window with the “OK” button.

STEP 25. Definition of load(s) combinations

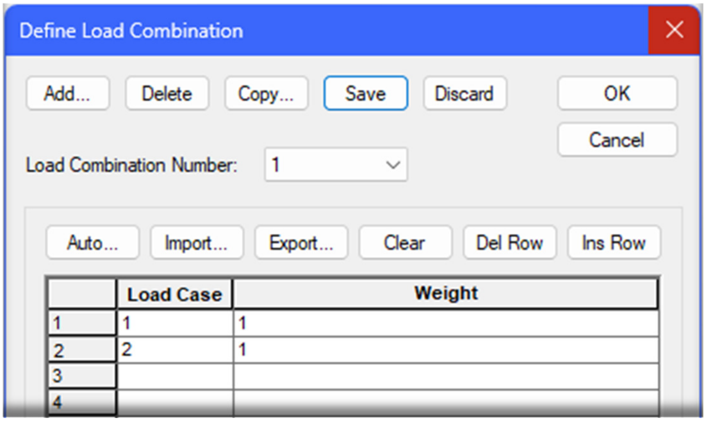
To create a load combination, go to “Model → Loading → Load Combination...”. Then, in the newly opened window, two load combinations will be created – one relating to the Serviceability Limit State (SLS), the other to the Ultimate Limit State (ULS). At this stage, we assume that the first case refers to the self-weight of the structure, while the second case concerns the snow loading on the upper surface – the arch. (referring to previously defined load cases). To add load combination No. 1, press the “Add...” button and enter the following data:

Load Combination Number:	1

Data in the table:

	Load Case #	**Weight**
1	1	1
2	2	1

The window view with the entered data is presented in the figure below:



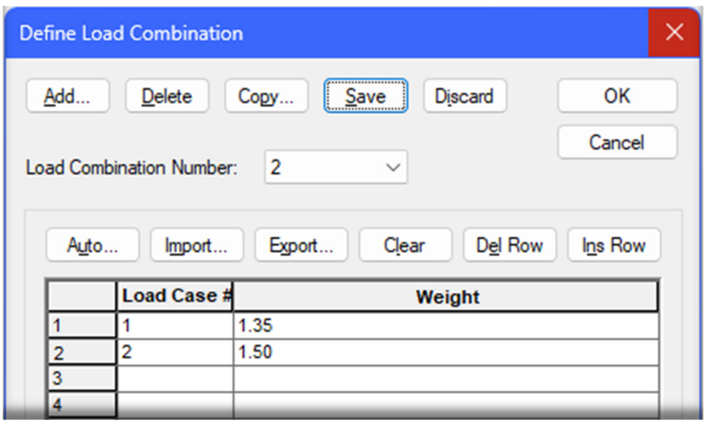
After entering the data, press the "Save" button and then "Add..." again. Currently, the ULS case will be added, so please enter the following data:

Load Combination Number:	2

Data in the table:

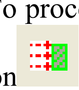
	Load Case #	Weight
1	1	1.35
2	2	1.50

After entering the data, press the "Save" button and then leave the window with the "OK" button. The window view with the entered data is presented in the figure below:



Note: Weights corresponds to the Eurocode(s) partial safety factors.

STEP 26. Definition of loads

Two types of loads will be defined in this model. The load of the structure's own weight and the distributed load of the arch shell, e.g. snow. To proceed to defining loads, go to "Model → Loading → Apply..." or press the button . After opening a new window, the self-weight load of the structure will be declared first, so for the "Load Type:" option, select "Mass Proportional" from the drop-down list, then for the "Load Number:" option, press the "Define..." button. Another new window will open, in which you should press "Add..." to add a new load, then enter the data as shown in the table:

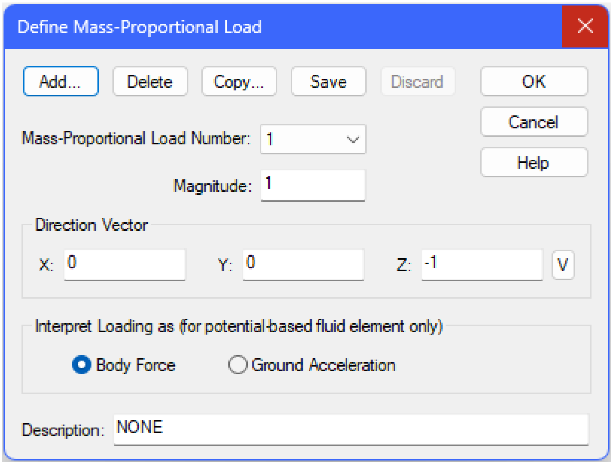
Mass Proportional Load Number:	1
Magnitude:	1
Body #:	1
Direction Vector	
X:	0
Y:	0
Z:	–1
Interpret Loading as (for potential-based fluid element only)	
Body Force	Checked
Description:	None

Note: If user enter a number greater than ±1 in the "Direction Vector", this value is still interpreted as 1 with the sign specified by the user. It can also be entered "–0.01" in the "Z:" field, which will also be treated as "–1", i.e. a load acting opposite to the "Z" axis.

Note: The load interpretation "Interpret Loading as (for potential based fluid element only)" as the English name suggests only works with potential based fluid elements.

Note: The "Magnitude" load value is a coefficient by which the volumetric weight of a given element is multiplied. The volumetric weight of the material was entered together with the definition of the material model. There is no need to enter a special coefficient here, because the coefficient will be used from the load combination, so "1" characterizes that the volumetric weight is to be assumed without any changes.

After entering the data in the window, press the "Save" button and then "OK", thus returning to the "Apply Load" window. The window with the entered data is shown in the figure below:



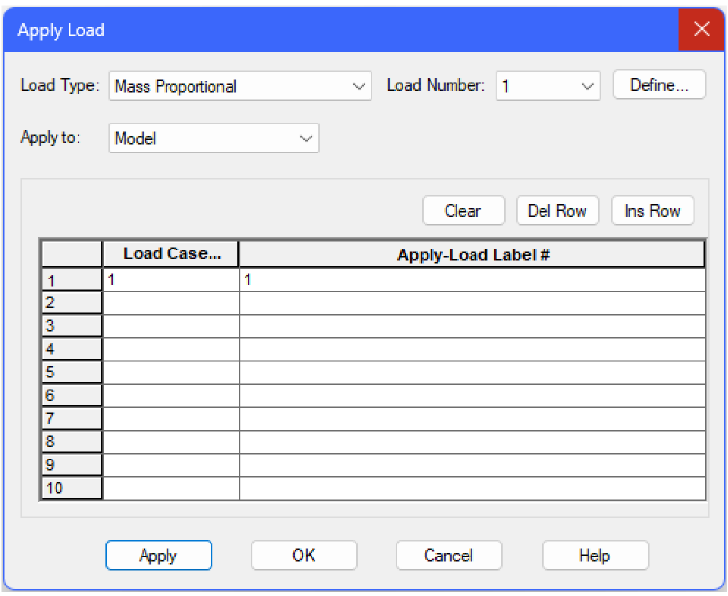
After returning to the "Apply Load" window, enter the following values:

Load Type:	Mass Proportional
Load Number:	1
Apply to:	Model

In the table, the data should look like this:

	Load Case…	**Apply-Load Label #**
1	1	1

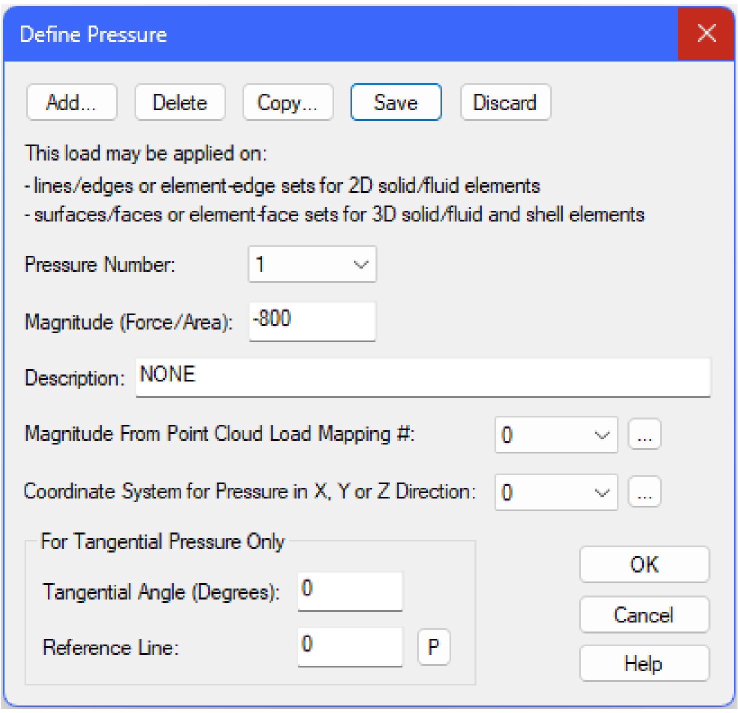
The window view with the entered data is shown in the figure below:



After entering the data, press the "Apply" button, and then, without leaving the window, select "Pressure" from the drop-down list next to "Load Type:", then press the "Define..." button next to the drop-down list for the "Load Number:" field. In the newly opened window, press the "Add..." button and enter the following data:

Pressure Number:	1
Magnitude (Force/Area):	–800
Description:	None
Magnitude From Point Cloud Load Mapping #	0
Coordinate System for Pressure in X, Y or Z Direction:	0
For Tangential Pressure Only	
Tangential Angle:	0
Reference Line:	0

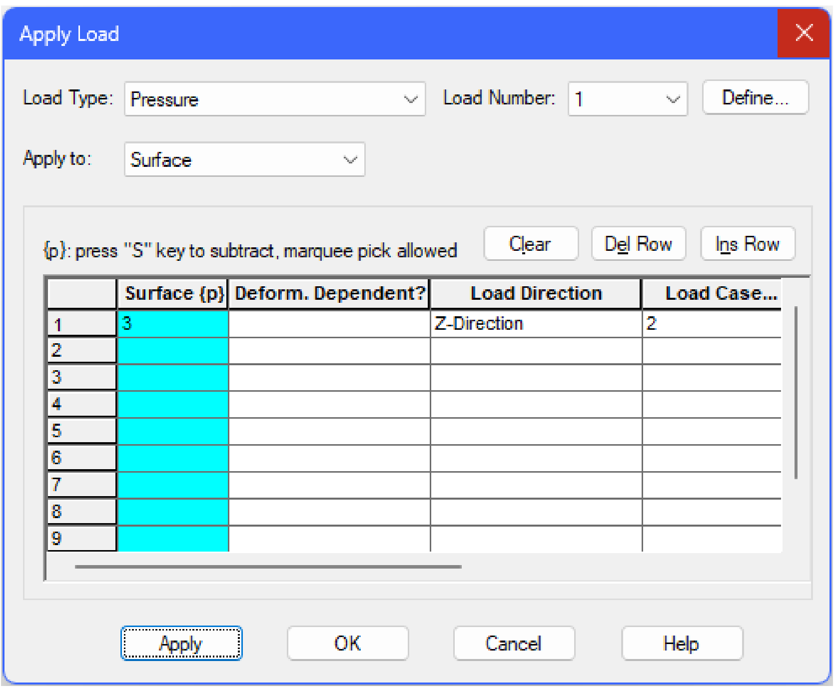
After entering the data, the window should look like this:

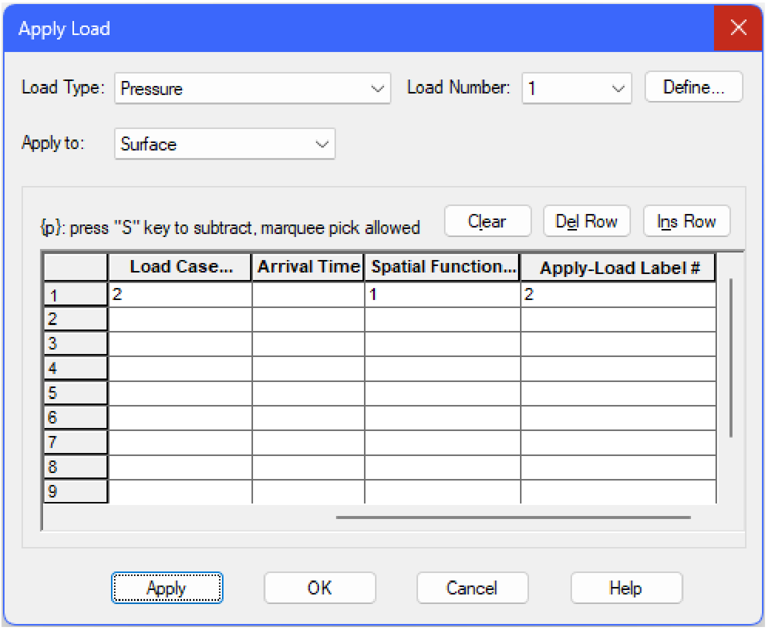


After declaring the load value, press the "Save" button and then "OK." closing the "Define Pressure" window and returning to the "Apply Load" window. In the "Apply Load" window, enter the following data:

Load Type:	Pressure
Load Number:	1
Apply to:	Surface
Values in a horizontal table	
Surface {p}	3
Deform. Dependent?	omit
Load Direction	Z-Direction
Load Case	2
Arrival Time	omit
Spatial Function	1
Label #	2

After entering the data, press the "Apply" button and then "OK". The window view with the entered data is shown in the drawings below:

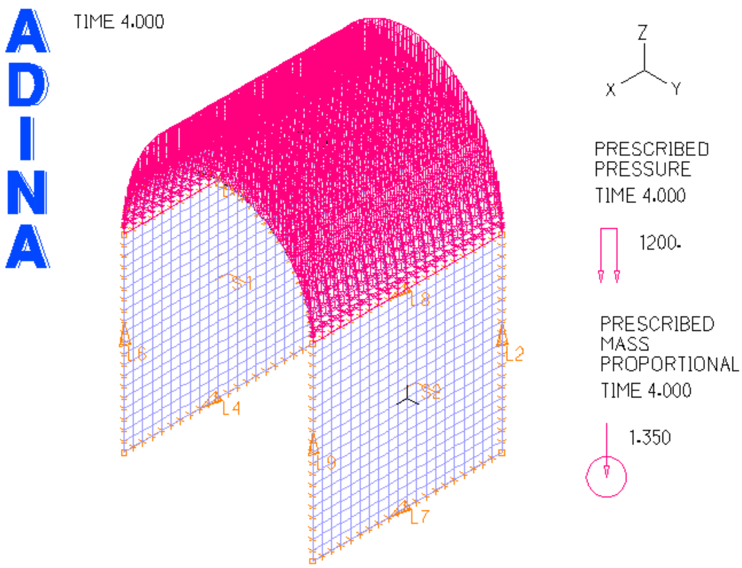




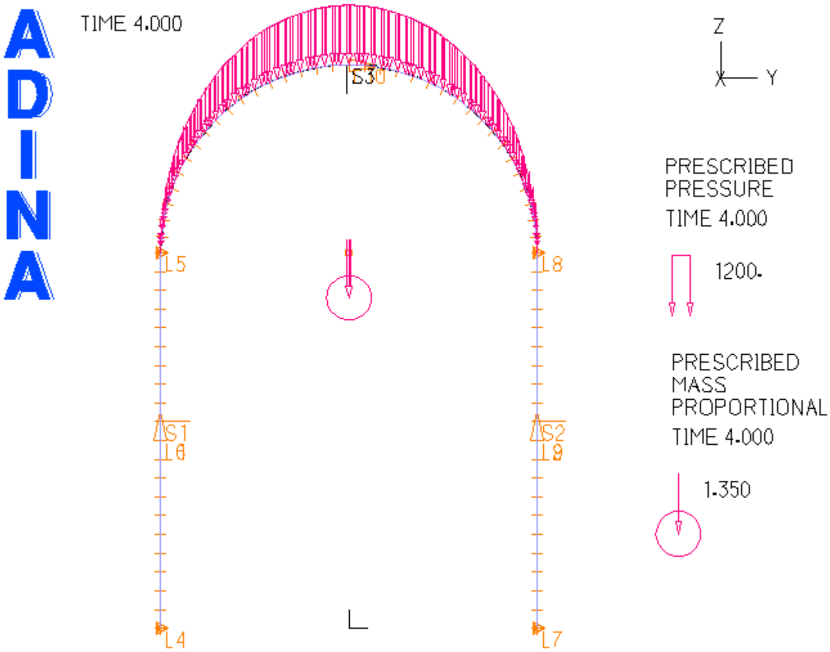
<u>Note</u>: After pressing the "Apply" button, any items that were omitted will be filled in with default values.

STEP 27. Displaying active model loads

In order to display defined loads in the main model window, click  button on the toolbar. The model window should look similar to that:



Since the above drawing does not accurately represent the load distribution, the user should switch view to the YZ plane. To do this, select the arrow next to the button  and select the "+YZ View" option. The model view should look like this:



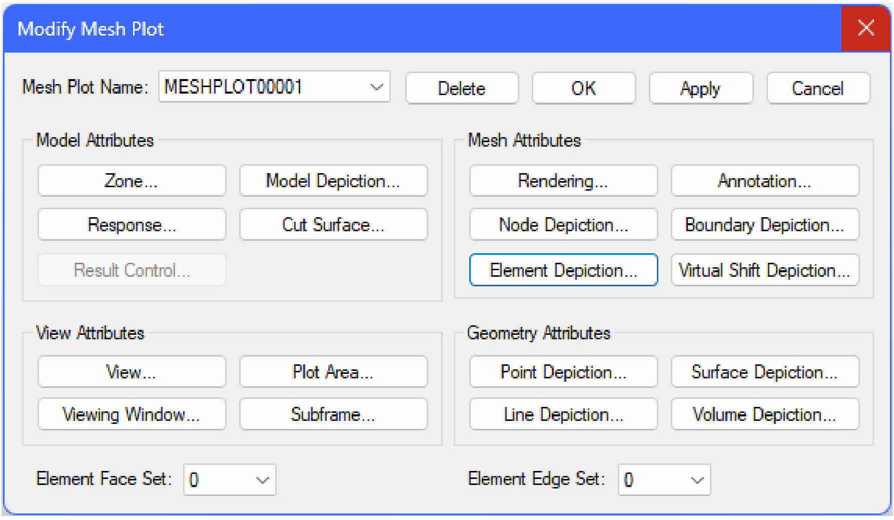
As shown in the figure above, the load was declared correctly.

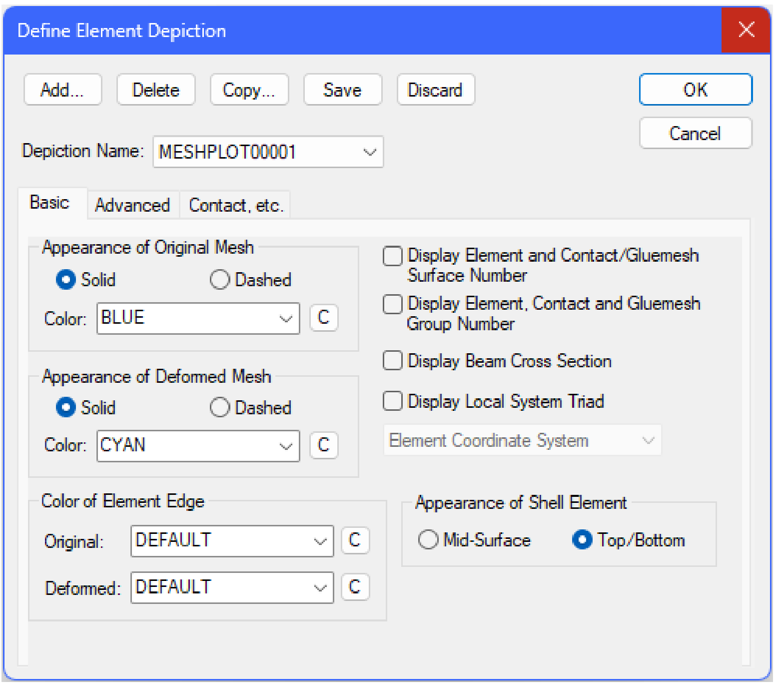
STEP 28. Displaying shell elements thinckess

To display the thickness of shell elements, go to "Display → Geometry / Mesh Plot → Modify..." or press the button . In the newly opened window, locate and press the "Element Depiction" button in the "Mesh Attributes" option group. After pressing the button, a new window will open in which the following data should be changed in the "Basic" tab:

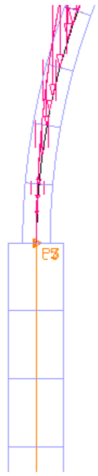
"Basic" tab	
...	
Appearance of Shell Element	
Top/Bottom	Checked

After entering the data, press the “Save” button and then exit both open windows with the “OK” button. Both windows are presented in the figures below:





To precisely see the difference in thickness between the arch and the wall, select the button  and then use the left mouse button to select the area where the arch and the wall connect. The model should look something like this:



As you can see in the drawing above, the wall is twice as thick as the arch element, so the element thicknesses have been entered correctly. To return to the view of the entire model, press the button  several times and activate the button .

STEP 29. Save existing model to a file

Each model should have been saved to a file between few steps taken in order to not lose the data. According to that select "File → Save as..." from the menu. When a new window opens, indicate the location of the saved file and its name.

Note: Do not uses spaces in the file names, because it leads to an error! The space can be replaced with the underline character __.

STEP 30. Starting calculations

In order to start calculations, choose "Solution → Data File/Run" from the upper menu tabs, or choose  from the toolbars. Subsequently, input the path for saving the resultant file. However, before the user clicks the "Save" button, they should make sure that the "Run Solution" and "Automatic" function near the "Maximum Memory for Solution" options are selected in the window in the "Adina Structures Solution" group of options. After starting the calculations and recalculating the model, close all three dialogue windows which appeared during the calculations.

Note: Depending on the complexity of the model and the number and type of finite elements used, model calculations may take from a few seconds to even several hours.

STEP 31. Post-processing module (results)

In order to go to the results module, change “ADINA Structures” to “Post-Processing” in the drop-down list of the “Module” toolbar.

When the user is prompted that the changes in the drawing have not been saved, it is recommended to save the model by going to “File → Save” or using the button .

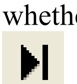
STEP 32. Opening the resultant file

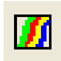
In order to open the resultant file, choose  from the toolbar, or choose “File → Open” from the upper menu tabs.

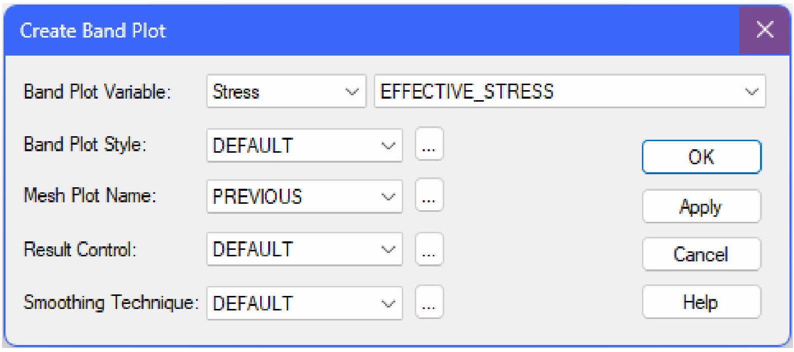
Note: Depending on the specifications of the computer, the loading of a file in which a spatial analysis has been performed may take between about a dozen seconds up to even several minutes. The number of the applied finite elements and the number of nodes used have the greatest impact on the loading time of the file.

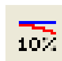
STEP 33. Effective stress results

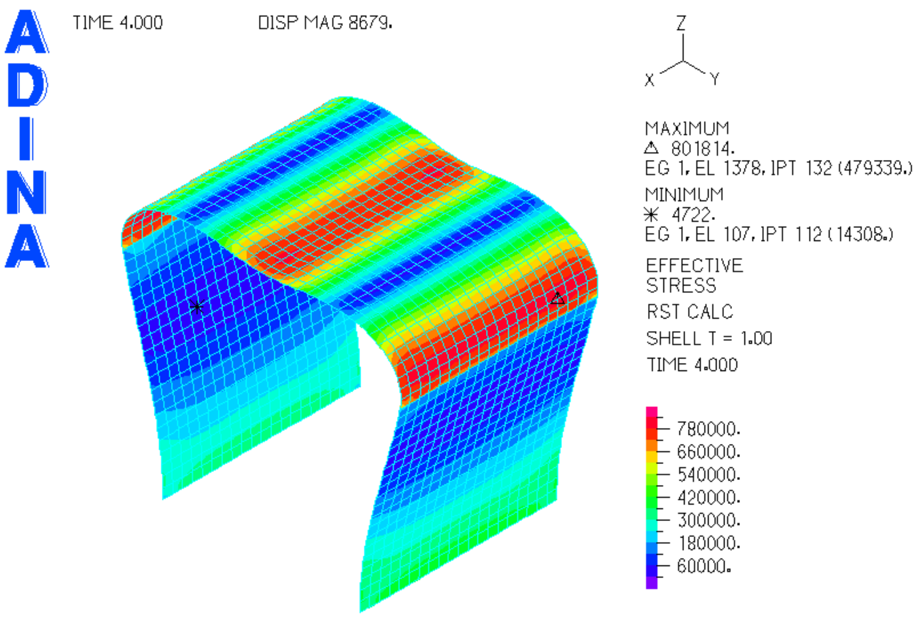
In this step, it is shown how to present stresses so that the distinction between the two external surfaces of the finite element and the stresses averaged inside the element are visible. The results will be displayed for the ULS combination, but it is worth classifying what a given time constant – “TIME” in the upper left corner of the window means. The load cases and combinations adopted for calculations are assigned unique identification numbers. According to how the loads were defined, the following classification is adopted at default:
“TIME 0.000” – no loads
“TIME 1.000” – self-weight (load case)
“TIME 2.000” – snow load (load case)
“TIME 3.000” – Serviceability limit state (own weight + snow load, both cases with safety factor 1.00)
“TIME 4.000” – Ultimate limit state (self-weight with a factor of 1.35 and snow load with a safety factor of 1.50)

Before the stresses, which are averaged to the center of the finite element are displayed, make sure that the results will be presented for the ULS state – “TIME 4.000”. To do this, look at the upper left corner of the result window and check whether the “TIME 4.000” value is there, and if it is not, one should use the button .

For now one can view the stresses. First, stress maps will be displayed for the planes passing through the center of the finite elements. To do this, go to “Display → Band Plot → Create...” or press the button . After opening a new window, for the “Band Plot Variable” option, select “Stress” in the first drop-down list, and “EFFECTIVE_STRESS” in the second drop-down list next to it. After entering the data, press the “OK” button. The window view with the entered data is shown in the figure below:

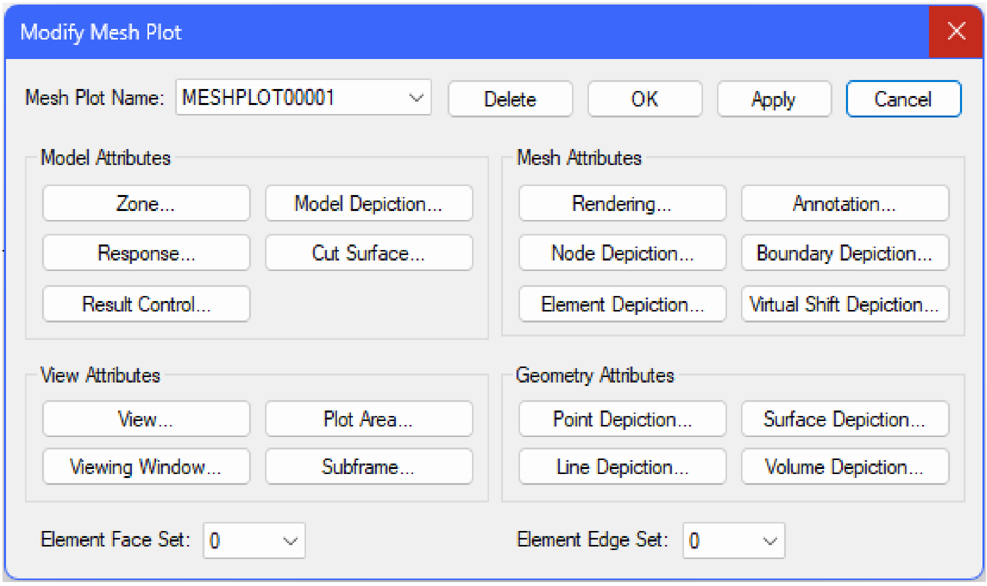


In order to better visualize what is happening to the model, user can turn on scaled displacements to see how the given structure deforms under the influence of applied loads. To display the scaled deformations, press the button . Currently the model should look like this:



Maintaining the current display, the finite element thickness will be displayed, with the stresses for each finite element displayed for each face separately.

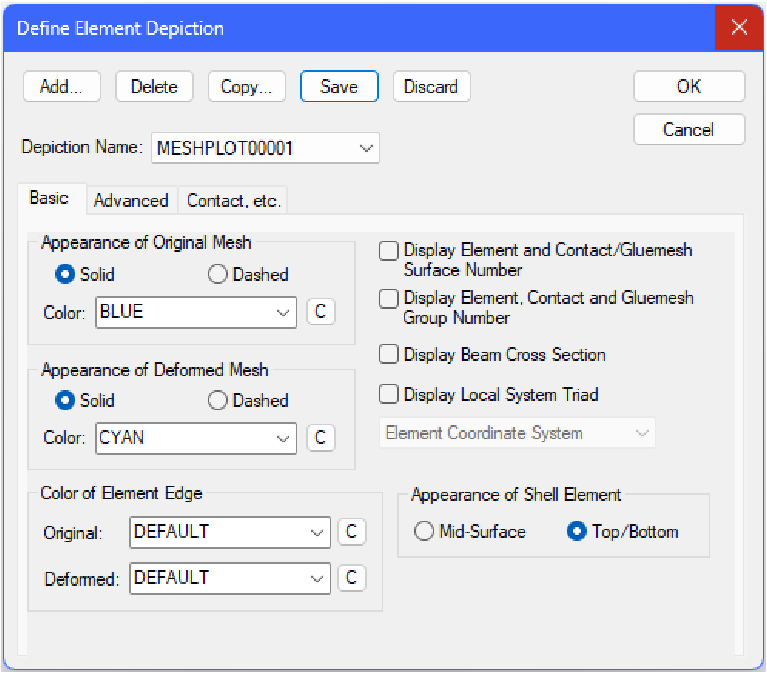
To enable the display of stresses on opposite faces of each finite element, go to "Display → Geometry Mesh / Plot → Modify..." or press the button . "Modify Mesh Plot" window is presented below:



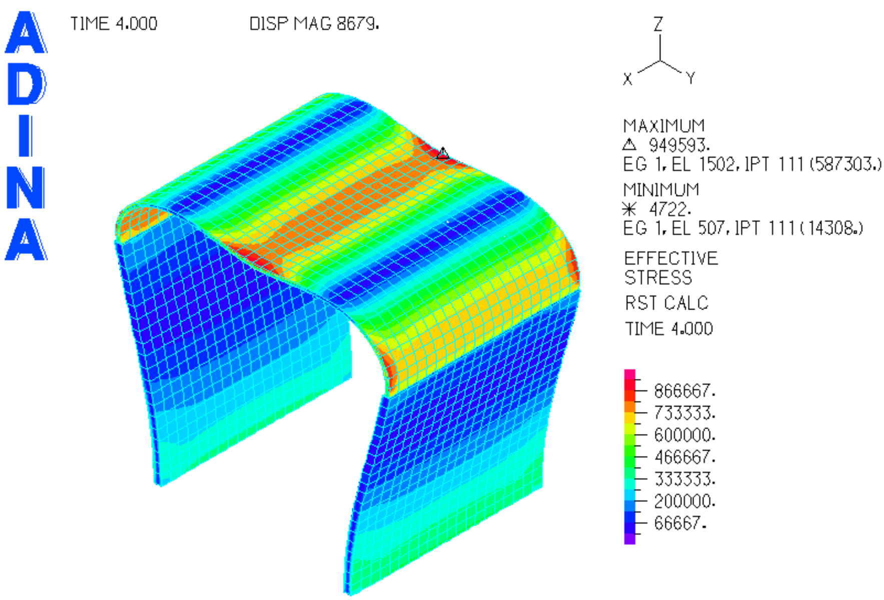
In the newly opened window press the "Element Depiction" button. When the next window opens, change the following data:

"Basic" tab	
…	
Appearance of Shell Element	
Top/Bottom	Checked

Remaining options are left unchanged. After entering the data, press the "Save" button and then exit both open windows with the "OK" button. The view of the "Define Element Depiction" window is shown in the following figure:

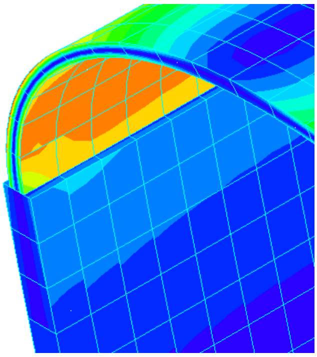


Additionally, the model after lowering the window should look like:

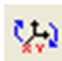


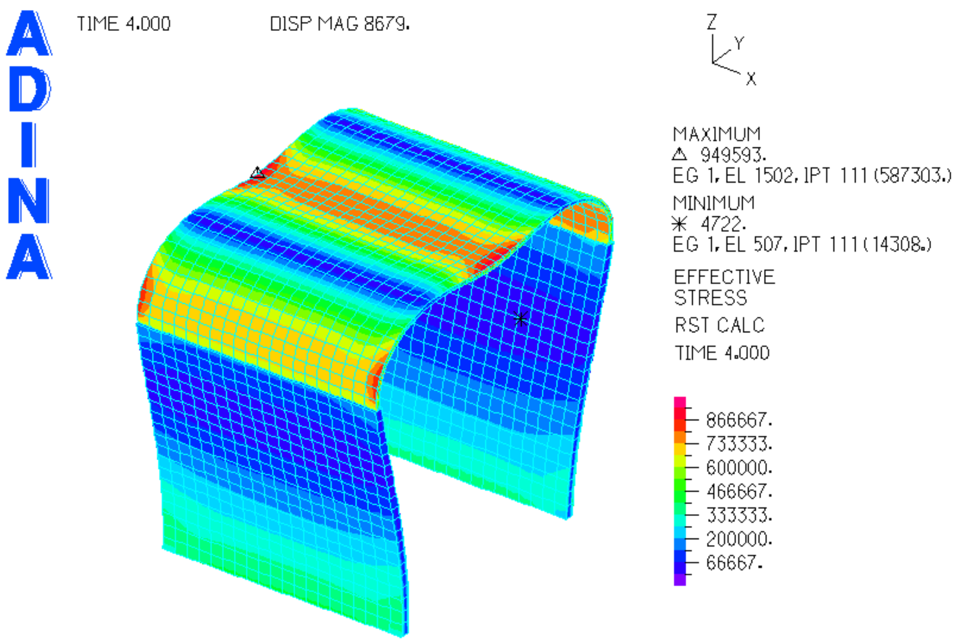
To show the essence of the applied function, user needs to zoom in on the model at the connection of the wall and the arch. To zoom in, activate the button 

and then select the desired area to be enlarged. In the author's version, the model looks something like this:



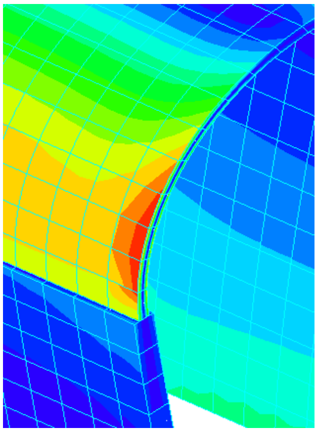
Note: When the display of stresses for opposite faces of a finite element is turned on, it is not possible to turn on stress smoothing using the button . Also the plot may disappear after the use of mentioned button.

Currently, the model should be moved to the starting position by pressing the button  several times. Then activate the button  and rotate the model so that the outer part of the previously approximated place where the wall and the arch connect is visible. The model should look something like this:

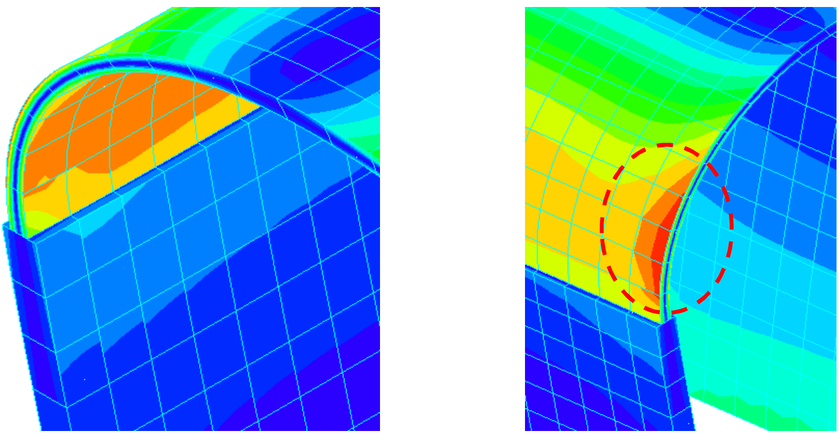


Note: It is possible to rotate the model in a second way. After activating the button , select the model with the left mouse button and hold it down, and also press and hold the SHIFT key on the keyboard. With this combination of buttons pressed, you move the mouse and rotate the model to the desired location.

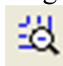
Select the button  again and select the area at the wall-arch connection for the same part of the model as before. The author's model view looks something like this:

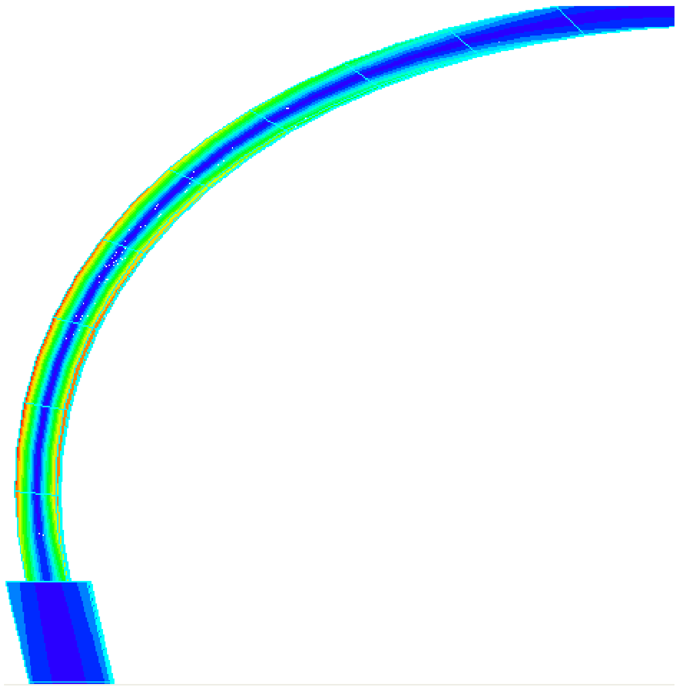


Comparing both obtained drawings:



It is clearly visible that the stress values on the inner and outer surfaces of the shell are different. Stress concentration occurs on the outer shell of the arc near its beginning (marked with a dashed line circle), whereas in the inner arch shell side the stress values are more or less the same – there are no concentrations.

Moreover, stresses can be considered "at the thickness" of the shell. To see how the stresses are distributed along the thickness of the shell near the connection of the arch with the wall, first the observation plane of the model must be changed. First, move the model back to its starting position by pressing the button  several times. To change the model viewing plane, press the arrow next to the button  and select the "+YZ View" option. Activate the button  again and mark the area near the connection of the arch with the wall as shown in the figure below:

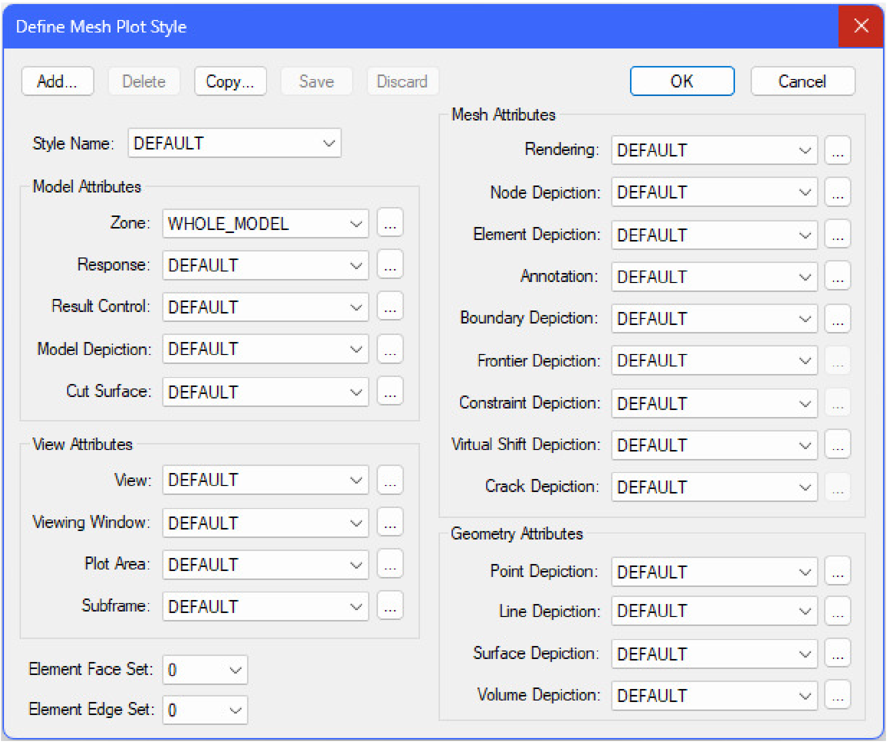


According to the above shown figure, both compressive and tensile stresses are shown across the thickness of the arch.

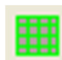
Note: In the ADINA program, compressive and tensile stresses are displayed in the same colors in the case of the "Effective Stress" option, the stress values presented in the legend and in the drawing are the absolute value of the obtained stresses.

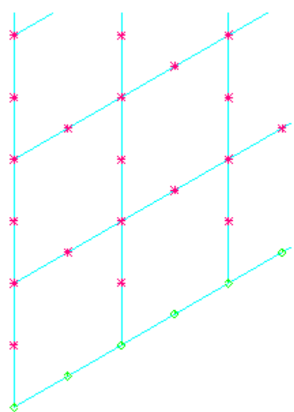
STEP 34. Degrees of freedom in finite element nodes

To graphically present which nodes have freedom of movement and rotation, and which nodes have no rotation ability, go to "Display → Geometry Mesh/Plot → Define Style...". After that the new window appears:



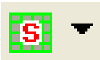
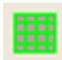
locate the "Node Depiction" option in the "Mesh Attributes" option group, and then select "ROTATIONAL_DOF" from the drop-down list.

After making changes, press the "Save" button and then leave the window with the "OK" button. After returning to the main results window, press the  and  button. Then activate the zoom button  and zoom in on the model at the support, as shown in the figure below:



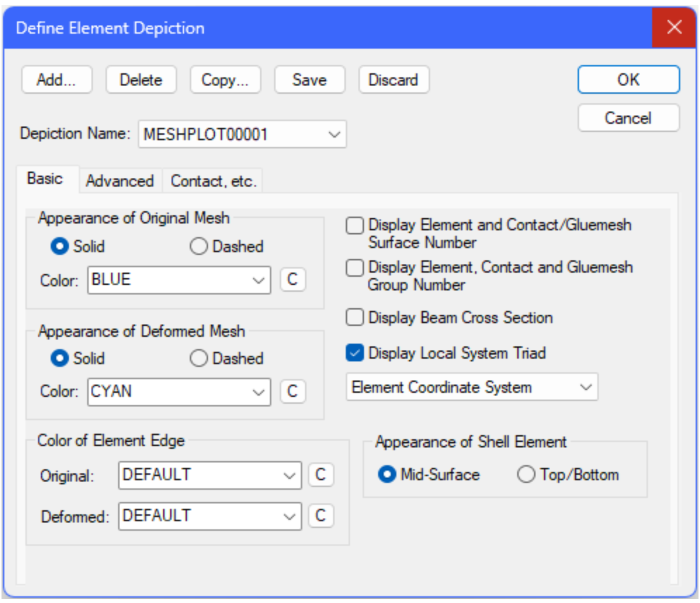
The presented nodes should differ in colors for the user. In the case of nodes located at the support, they will differ in color from the other nodes in the structure. Additionally, the support nodes have 0 degrees of freedom, while the remaining nodes of the structure have rotational degrees of freedom free, respectively.

STEP 35. Displaying local coordinate systems on finite elements

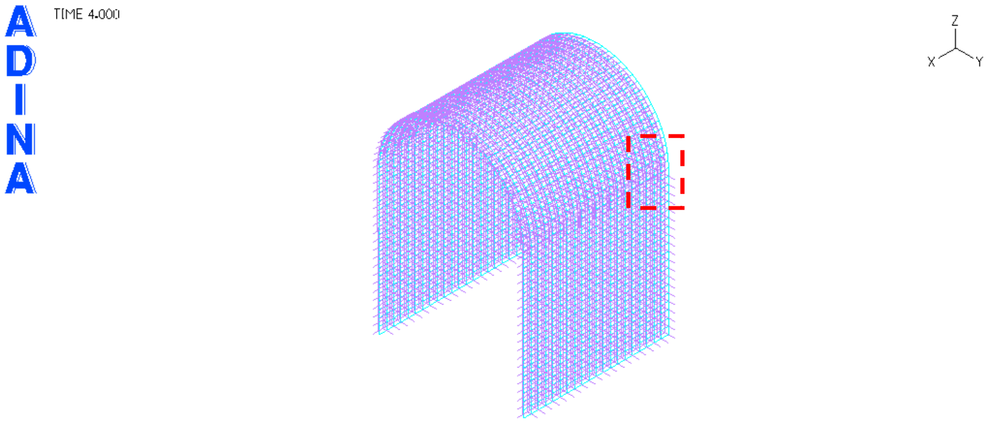
To display the local coordinate system of each finite element, first the model have to be cleaned from previous entries. To do this, select the arrow next to the button  and select the “Reset Mesh Plot Style” option. Then press the following buttons  and . After displaying a “clean” model consisting of finite elements, go to “Display → Geometry Mesh/Plot → Modify...” or press the button . Once a new window opens, locate the “Element Depiction” button in the “Mesh Attributes” button group. After another new window opens, enter the following data:

“Basic” tab	
…	
Display Local System Triad	Checked

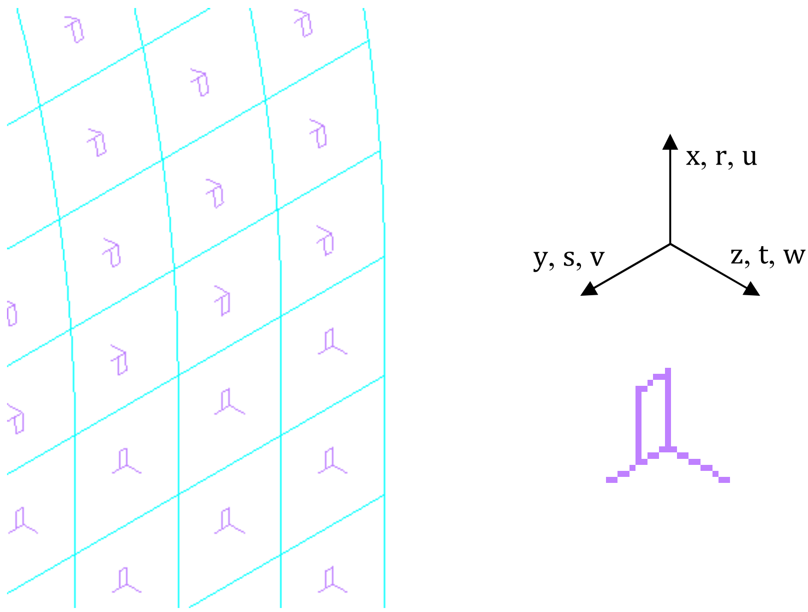
Select the “Element Coordinate System” option from the drop-down list. After entering the data, press the “Save” button and then you can exit both windows with the “OK” button. The view of the “Define Element Depiction” window with the entered data is shown in the figure below:



After returning to the main results window, activate the zoom button  and then indicate the area to be zoomed in, e.g. the connection of the arc with the wall for the rightmost area in this window:



When zoomed in, the drawing should look something like this:



According to the figure above, the local coordinate systems differ in both the wall and the arc. In the arch, the "u" and "w" axes are in completely opposite directions in comparison to the same axes of the local coordinate systems assigned to the wall.

- **Dynamics**

EXAMPLE 6
VIBRATION FREQUENCY AND VIBRATION MODES OF A T-SHAPE CANTILEVER BEAM

In this example activities related to modeling of a one end clamped steel bar with one-degree of freedom concentrated mass attached to the free end. Modelling is limited to the planar Cartesian system (2D). The diagram of the analyzed model is shown in Figure 24.

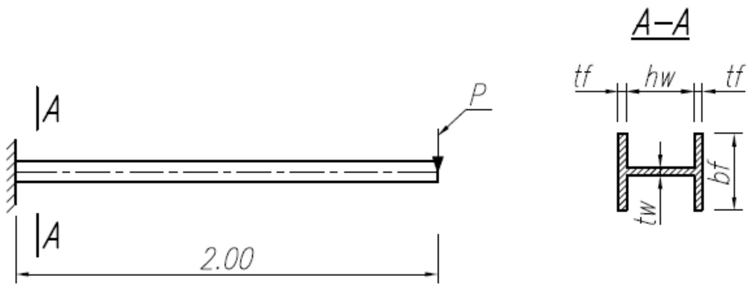


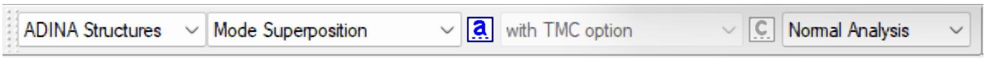
Fig. 24. Schematic of a concentrated mass beam with one degree of freedom

The following data is used in the analysis:

– cross-section dimensions:
  $h_w = 0.484$ m
  $t_w = 0.010$ m
  $b_f = 0.300$ m
  $t_f = 0.008$ m
– load:
  concentrated force $P = 2000$ N
– material constants:
  steel S235JR:
  $E = 210$ GPa $= 2.1\mathrm{e}^{11}$Pa
  $\upsilon = 0.30$
  $\rho = 7859$ kg/m^3
– boundary conditions:
  one beam's end fixed (blocked all degrees of freedom)

STEP 1. Definition of the type of analysis

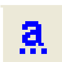
Upon opening the ADINA software, choose “ADINA Structures” from the “Module Bar” in the “Program Module” section, and choose “Mode Superposition” from the drop-down list next to “Analysis Type”.

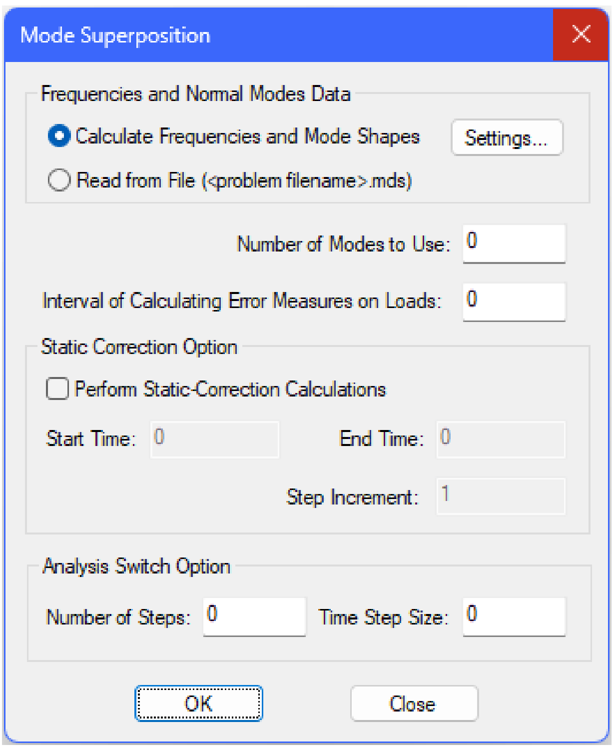


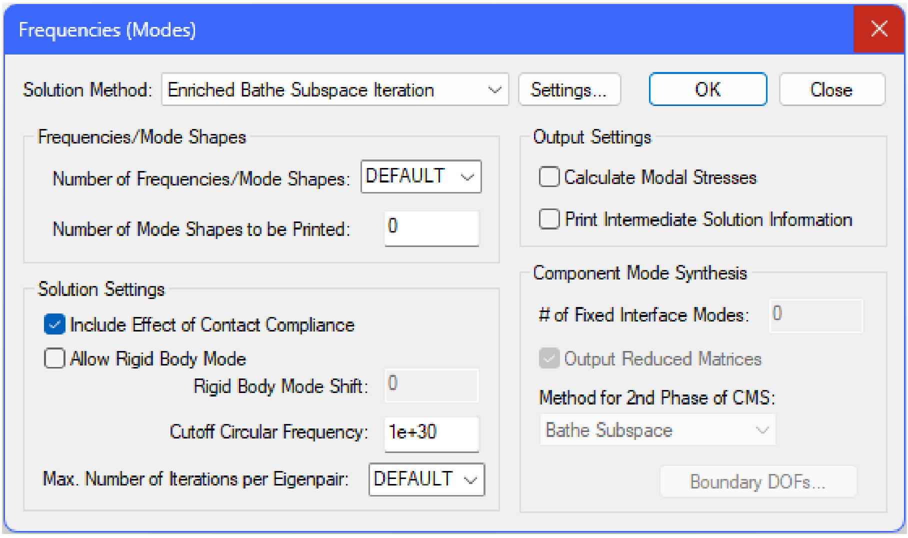
STEP 2. Entering the heading of the model

In order to specify a heading, go to “Control → Heading…”. Subsequently, enter the project heading in the text box, e.g., “Beam – free and forced vibrations”. Upon entering a heading, click the “OK” button.

STEP 3. Define number of frequencies to be calculated

Press the  button located on the toolbar and after a new window opens press the “Settings...” button, another window should open. Both windows are presented below:





Description of individual functions in the window:

"Calculate Frequencies and Mode Shapes" – force the program to calculate frequencies and mode shapes of a given model.

"Read from File (<problem file>.mds)" – frequencies and mode shapes are read from the file.

"Number of Modes to Use" – the number of modes which should be taken into consideration by the software.

"Interval of Calculating Error Measures on Loads" – specifies the 'time' interval in which the error measures are calculated.

"Perform Static-Correction Calculations" – enables/disables calculations of static--correction under dynamic calculations.

"Start Time / End Time" – specifies in which "TIME" range the static-correction is used.

"Time interval" – specifies the interval between defined "Start/End Time" range.

"Analysis Switch Option – Number of Steps" – options regarding analysis switch, specifies the number of steps used in switch analysis.

"Analysis Switch Option – Time Step Size" – time step size used in switch analysis.

In the "Frequencies (Modes)" window:

"Solution Method" – allow the user to choose the solution method of frequencies/ mode shapes.

"Number of Frequencies/Mode Shapes" – specifies the number of frequencies/mode shapes to be calculated in the model.

"Number of Mode Shapes to be Printed:" – number of mode shapes and their frequencies to be printed in the *.out file.

"Solution Settings" – solution options

"Include Effect of Contact Compliance" – allow to include the effect of contact between different parts of model (activated by default).

"Allow Rigid Body Mode" – an option that allows user to activate elements that are perfectly rigid. Specifies whether or not rigid-body modes are allowed. Should be used when the lowest frequency may be zero, or any part of the model would be insufficiently supported if all contact, mesh glueing and generalized constraints are removed.

"Rigid Body Mode Shift" – mode shifting option.

"Cutoff Circular Frequency:" – a field specifying the interruption of the program action when the specified or higher circular frequency is reached

"Max. Number of Iterations per Eigenpair:" – maximum number of iterations per eigenpair (frequency/mode shape).

"Output Settings" – options regarding the output file

"Calculate Modal Stresses" – option indicating whether or not to calculate modal stresses for post-processing.

"Print Intermediate Solution Information" – an option that allows user to print intermediate solution results in the *.out file.

"Component Mode Synthesis" – the component mode synthesis (CMS) method is used to calculate the lowest frequencies/modes of the original finite

"# of Fixed Interface Modes" – specifies the number of the fixed interface dynamic vibration modes to be calculated in the first phase of CMS.

"Output Reduced Matrices" – specifies whether the reduced or full matrices are written in the output file.

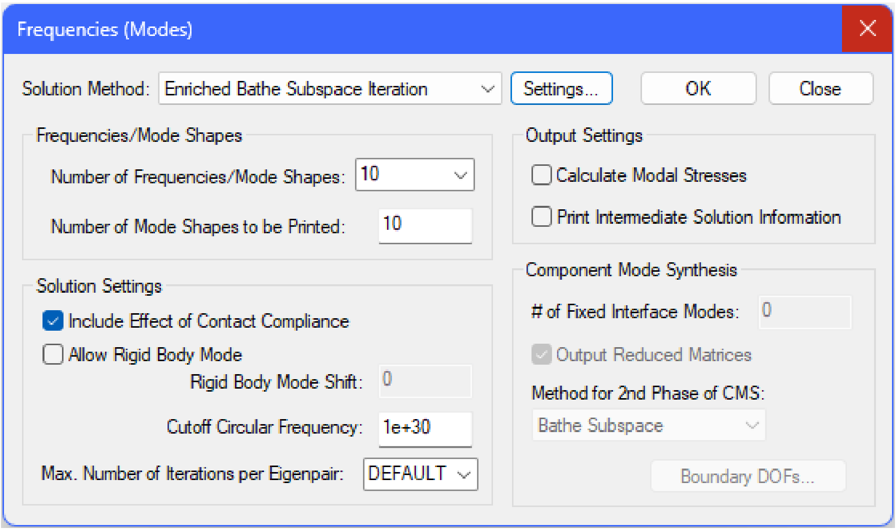
"Method for 2nd Phase of CMS" – specifies the number of frequencies/modes of the original finite element system to be calculated in

the second phase of CMS. Can only be used when "… subspace iteration" method is choosen.

Since for the purposes of this example it will be necessary to display both 10 frequencies results and vibration modes, the following data should be entered in the "Frequencies (Modes)" window:

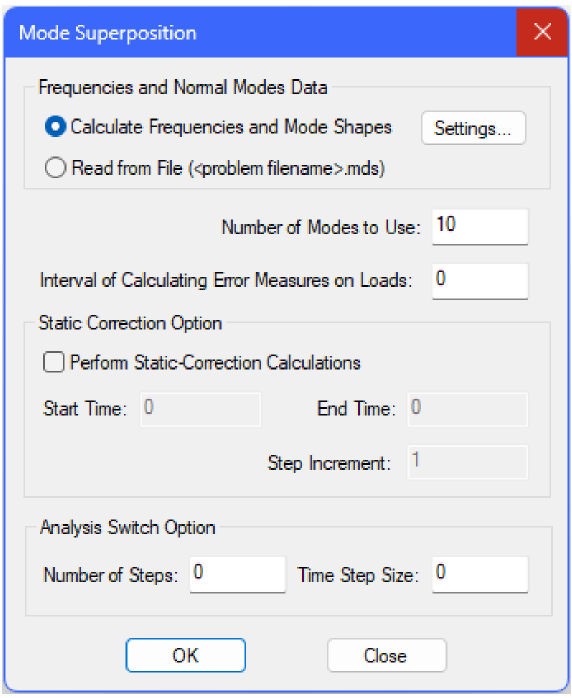
Frequencies/Mode Shapes	
Number of Frequencies/Mode Shapes:	10
Number of Mode Shapes to be Printed:	10
Leave the remaining options unchanged	

The window view with the entered data is shown in the figure below:



After entering the data, press the “OK” button to save changes and to exit the window.

After returning to the “Mode Superposition” window, enter “10” for the “Number of Modes to Use:” option. The figure below shows the window with the changes introduced.



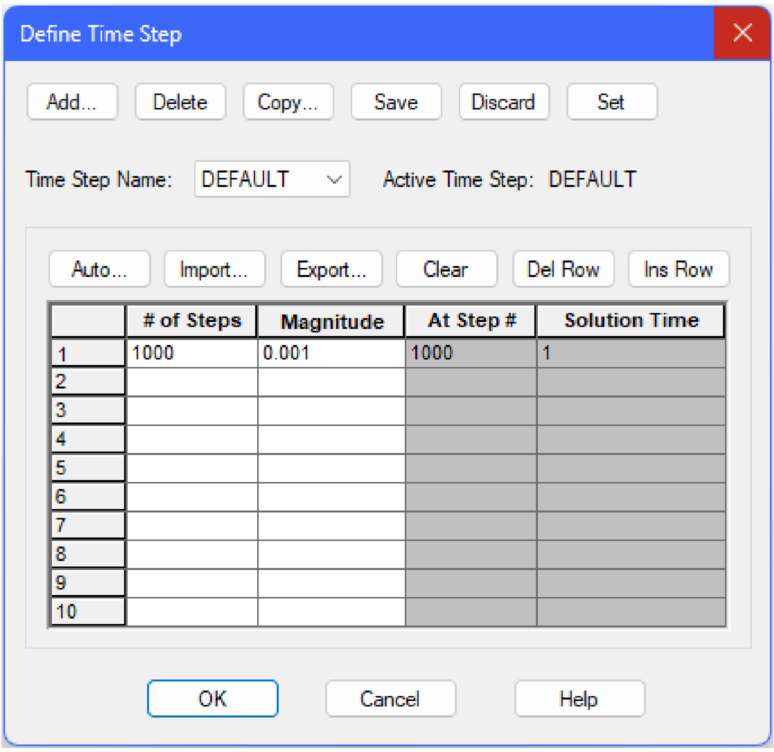
After making changes, press the “OK” button to leave the window.

STEP 4. Definition of the number of time steps

To define time steps, go to “Control → Time Step...”. Then, in the newly opened window, enter following data:

	# of Steps	**Magnitude**
1	1000	0.001

After entering the values, press the “Save” button and then “OK.” The window with the entered data is presented below:

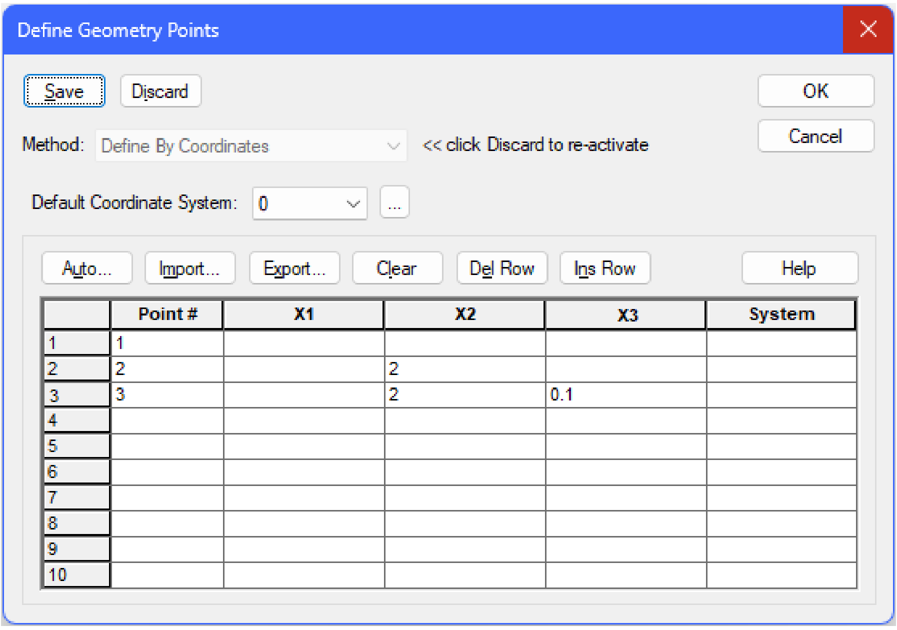


STEP 5. Definition of points

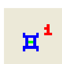
In order to define points, go to “Geometry → Points → Define…” in the main model window, or click . Upon opening a new window, input points in accordance with the table below:

Point #	X1	X2	X3	System
1	0	0	0	0
2	0	2	0	0
3	0	2	0.1	0

<u>Note</u>: There is no need to enter zero values in the table. After pressing the "Apply" button, the table will be automatically completed with these values. Then, in the window for defining points, press "Apply" and "OK". The window with entered data is presented below:



STEP 6. Displaying point ID numbers

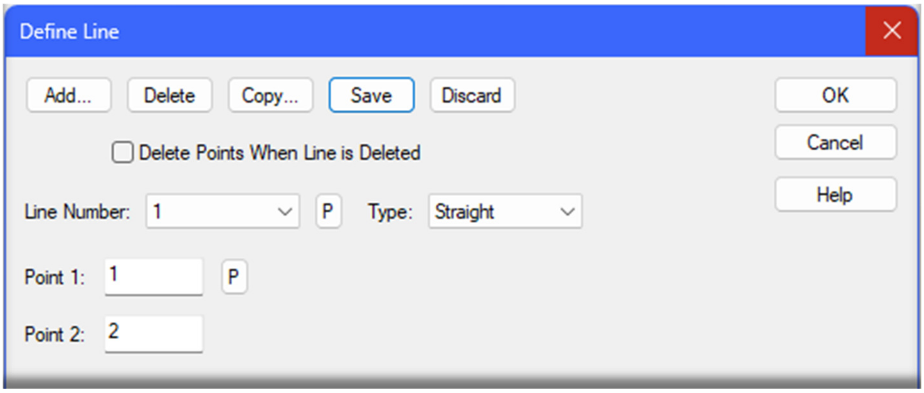
To display point identifiers, press the button  on the toolbar.

STEP 7. Definition of lines

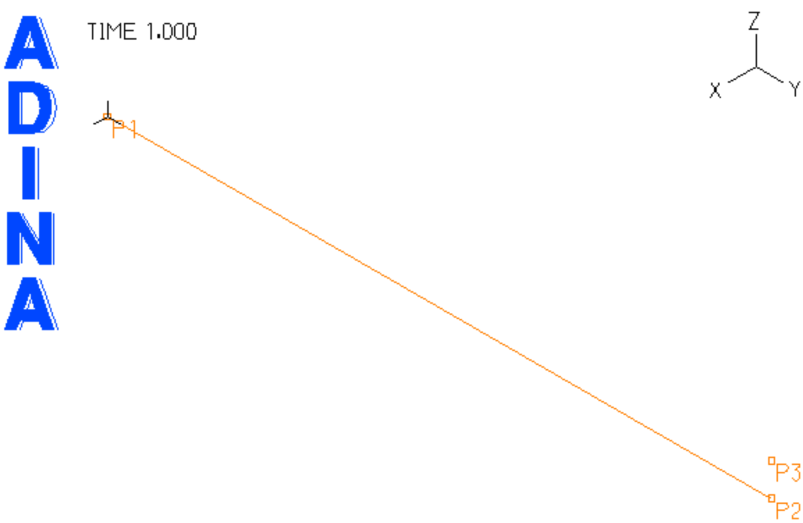
In this example, user needs to create only one straight line, according to that go to "Geometry → Lines → Define..." or press the button . After opening a new window, press the "Add..." button and then, in place of the "Type:" line type, select the option from the drop-down list "Straight". After performing the operation, enter the data according to the table below:

Line Number:	Type:	Point 1:	Point 2:
1	Straight	1	2

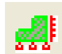
Points can also be entered using the "P" button located next to the text field belonging to the "Point 1:" option. After selecting the button, left-click the mouse in the modeling window to indicate 2 points of interest, and then the program will return to the point definition window. The window view with the entered data is shown in the figure below:



After entering the data, press the "Save" button and then "OK". The main model window should look like this:



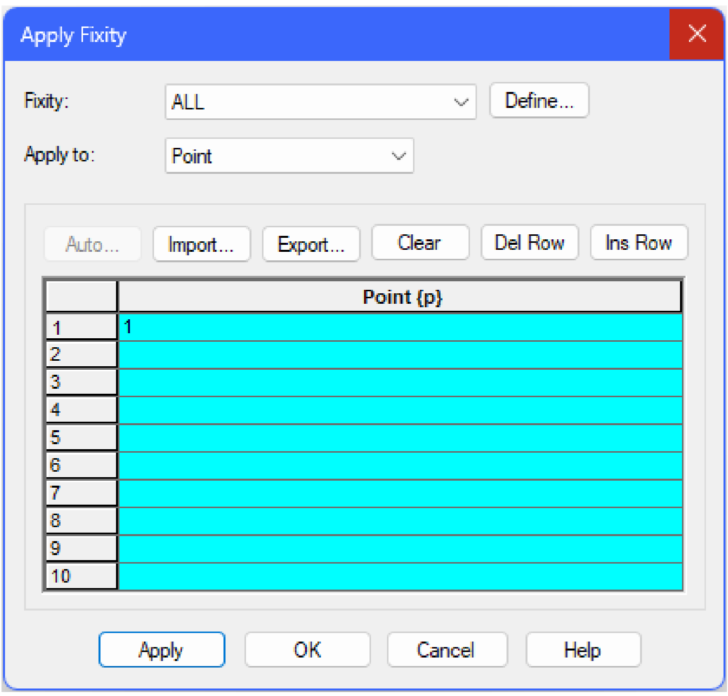
STEP 8. Definition of boundary conditions (fixity characteristics)

To define clamped support in the considered system, go to "Model → Boundary Conditions → Apply Fixity..." or press the button . Since the example only includes a clamped support, and the program by default has a declared such a support with all degrees of freedom blocked (support name "All"), there is no need to declare the support again. In the "Apply Fixity" enter following data:

Fixity:	ALL
Apply to:	Edge/Line
	Point {p}
1	1

<u>Note</u>: In older ADINA software versions if the "Fixity" column remains empty for the declared nodes/lines/areas, after pressing the "Save" button it will be completed based on the option selected from the drop-down list for "Default Fixity".

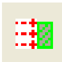
After performing the above operations, press the "Save" button and then "OK". The window view with the entered data is shown in the figure below:



STEP 9. Displaying boundary conditions

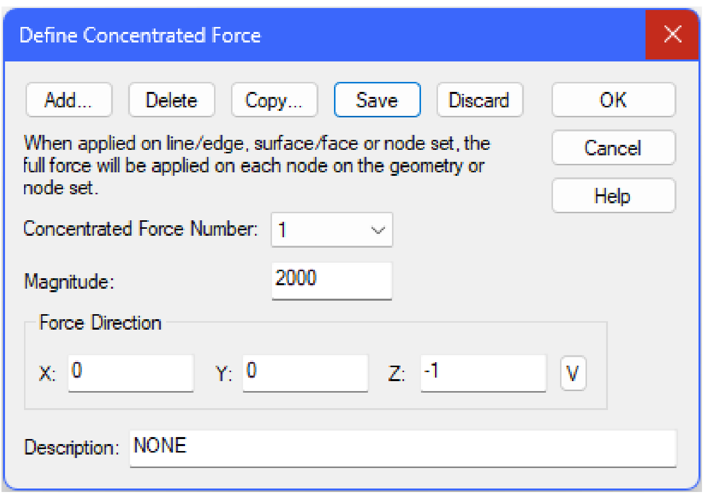
In order to display the defined boundary conditions in the main model window, click  button located on the toolbar.

STEP 10. Definition of loads

In order to define a load, choose “Model → Loading → Apply…” from the upper menu, or click . In the newly opened window, in the load type – “Type:”, select “Force” from the drop-down list – the same as in the case of defining concentrated force. In place of “Apply to:” the “Point” option should be selected. After entering this data, press the “Define” button next to the text field for the “Load Number:” option. A new window will open, in which press the “Add...” button and the further data should be entered in accordance with the table below:

Concentrated Force Number:	1
Magnitude:	2000
Force Direction	
X:	0
Y:	0
Z:	–1
Description:	None

After entering the data, the window should look like this:



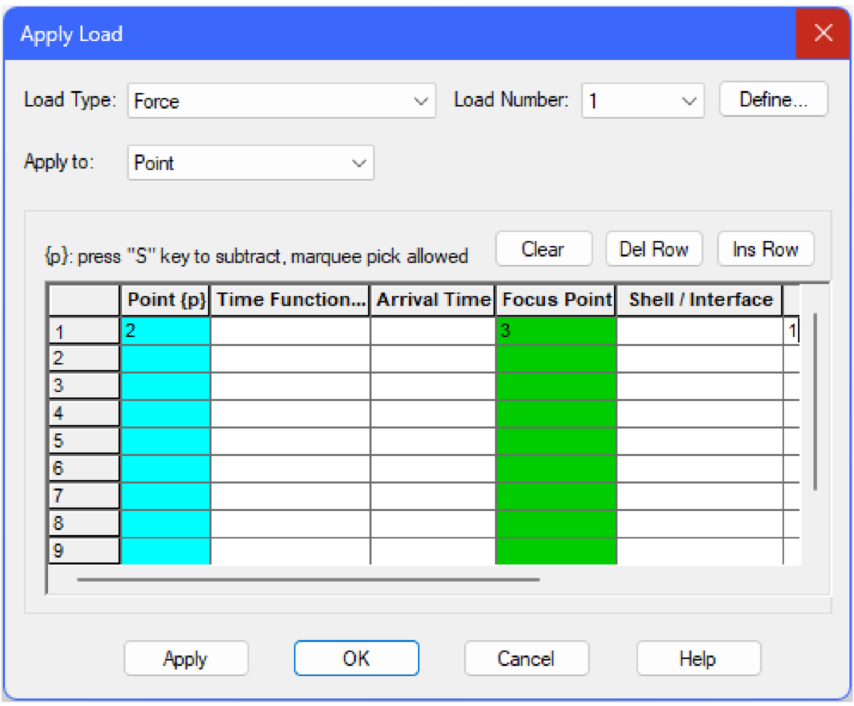
Then press the "Save" and "OK" buttons. After returning to the "Apply load" window enter following data:

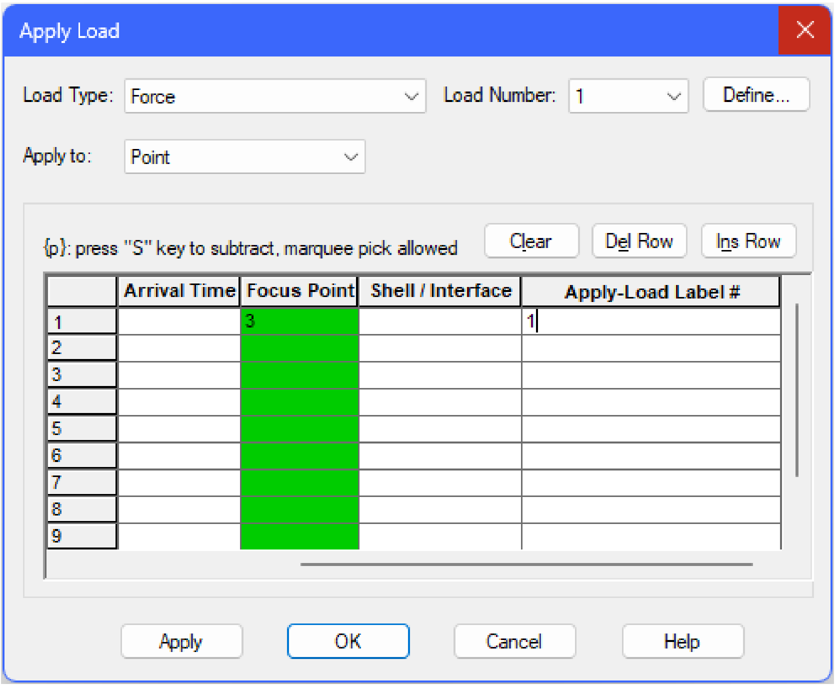
Load Type:	Force
Load Number:	1
Apply to:	Point

In the table, the data should look like this:

	Point {p}	**…**	**Focus Point**	**…**	**Apply-Load Label #**
1	2	Empty	3	Empty	1

After entering the above data, the window should look like this:

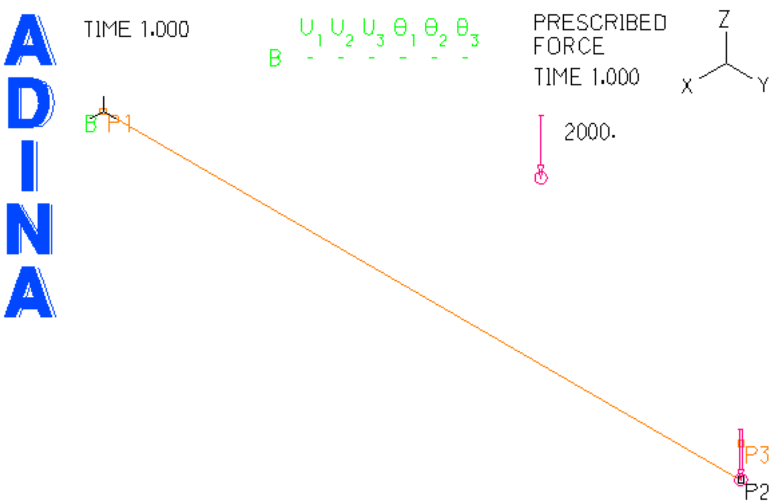




As soon as the data is entered press the “Apply” button and then leave the window with the “OK” button.

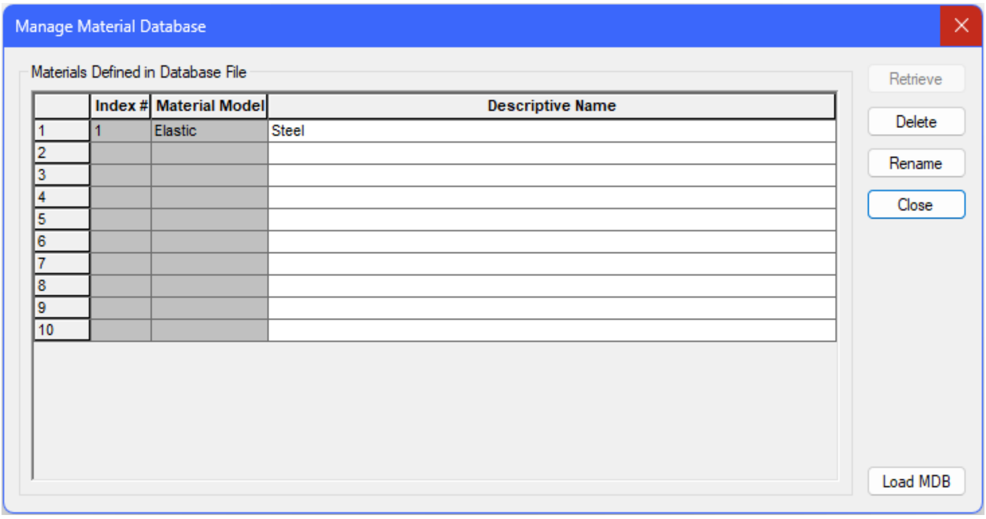
STEP 11. Displaying active model loads

In order to display defined loads in the main model window, click  button on the toolbar. The model window should look similar to that:



STEP 12. Definition of material constants

In order to define materials constants, go to "Model → Materials → Manage Materials..." or press the button , then in the newly opened window (if the material has been previously added to the program database), press the "Get MDB" button. A new window with a table should open, then select the material called "Steel" in the table and press the "Retrieve" button. After loading the material from the program database, press the "Close" button. In the "Manage Materials" window, in its lower part, in place of the table, there should be a material – "Steel" with the id number "1". The retrieving material window is presented below:

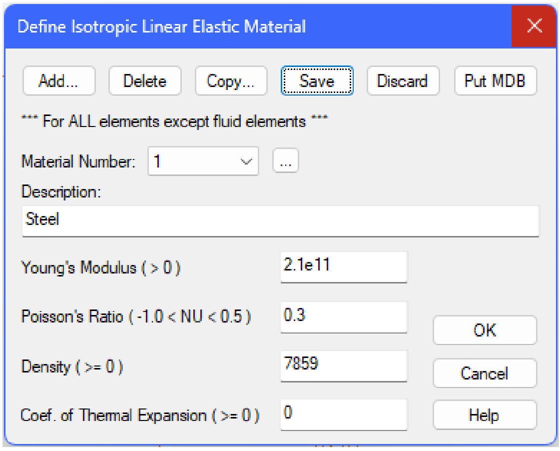


If the database does not contain the material required in the example, in the "Manage Materials" window, find the group relating to elastic materials – "Elastic", and then press the "Isotropic" button – isotropic material. In the newly opened window, press the "Add..." button and enter the data in accordance with the table below:

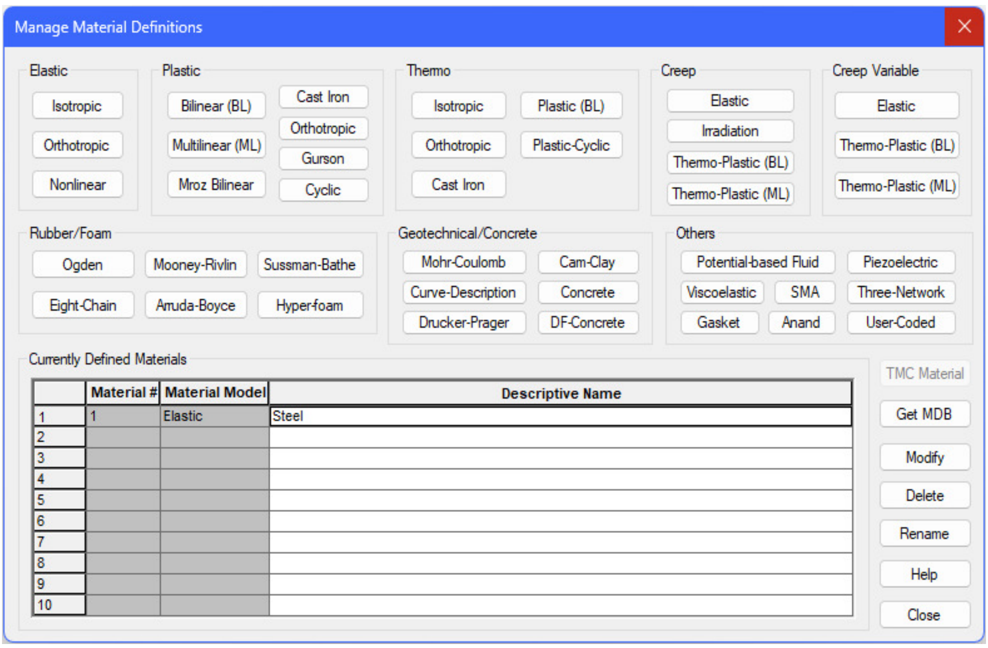
Material Number:	1
Description:	Steel
Young's Modulus (> 0)	2.1e11
Poisson's Ratio (–1.0 < NU < 0.5)	0.30
Density	7859
Coef. of Thermal Expansion (>= 0)	0

After entering the values, press the “Save” button. If the user wants to add material to the program's database, press the “Put MDB” button. After completing the operation, you can exit the window by pressing the “OK” button. Again, as in the previous case, after leaving the window, in the lower part of the “Manage Material Definitions” window, the table should contain the “Steel” material model with an assigned identification number of “1”.

The window view with the entered material in the elastic material definition window is shown in the figure below:

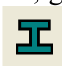


The view of the material manager window should look similarly as:



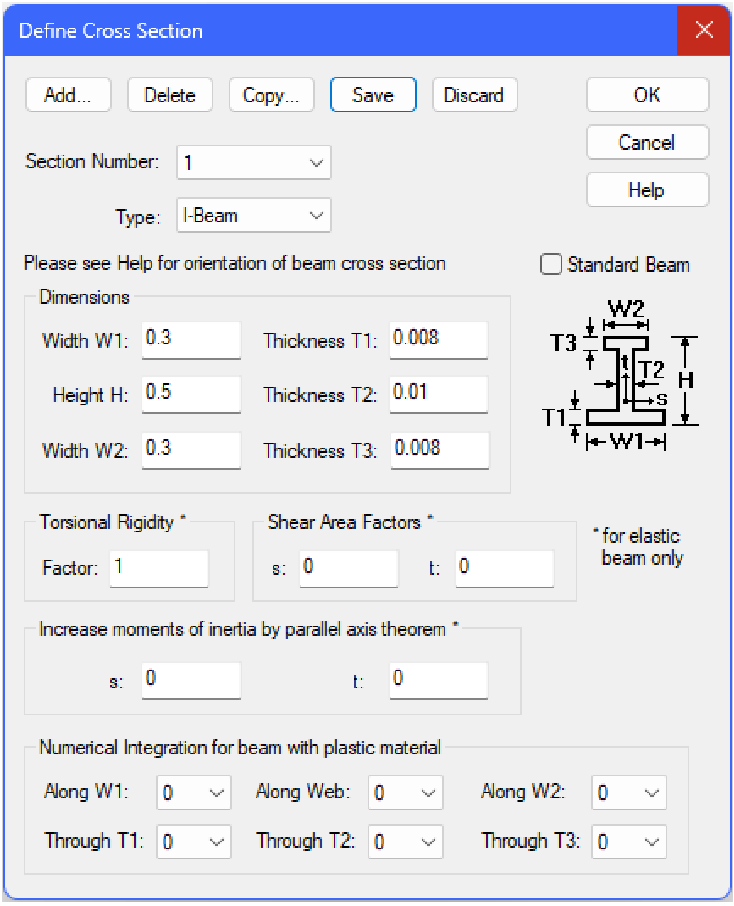
After performing all operations, exit the “Manage Material Definitions” window by pressing the “Close” button.

STEP 13. Definition of the cross-section of the beam

In order to define a cross-section, go to “Model → Cross-Sections…” in the upper tabs of the program, or click  in the program toolbars. In the newly opened window, click the “Add…” button. Enter following data:

Section Number:	1
Type:	I-Beam
Standard Beam	Unchecked
Dimensions	
Width W1:	0.300
Height H:	0.500
Width W2:	0.300
Thickness T1:	0.008
Thickness T2:	0.010
Thickness T3:	0.008
Torsional Rigidity *	
Factor:	1
Shear Area Factors *	
s:	0
t:	0
Increase moments of inertia by parallel axis theorem	
s:	0
t:	0
Numerical Integration for warping beam with nonlinear material	
Along W1:	0
Through T1:	0
Along Web:	0
Through T2:	0
Along W2:	0
Through T3:	0

After entering the above data, the window should look like this:



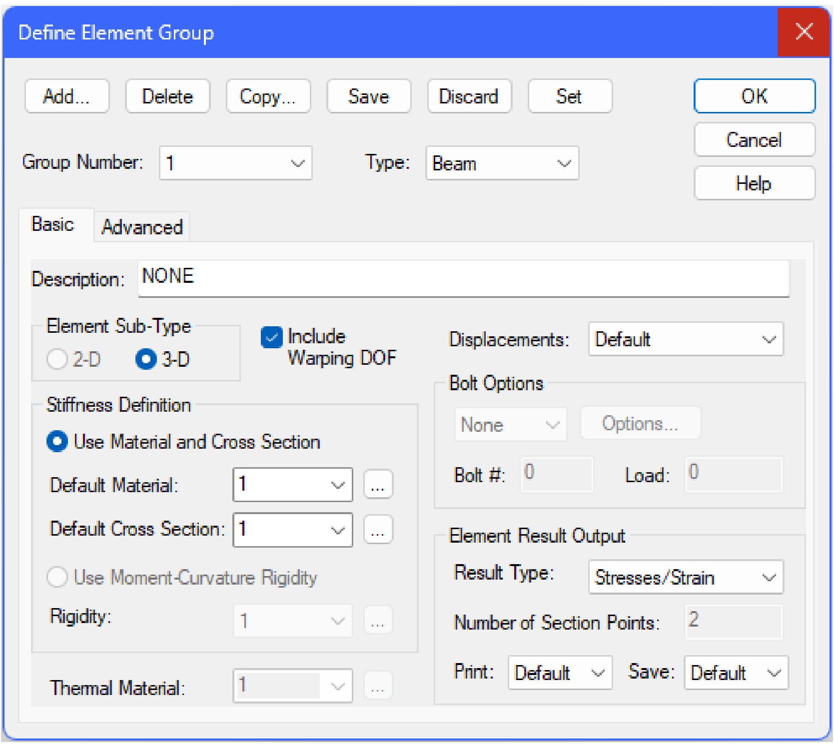
After entering all the data, press the "Save" button and then "OK."

STEP 14. Specifying the type of analyzed construction

In this example, there is only one group of elements. This group is determined by the type of material and the cross-section of the model. To create a group of elements, go to "Meshing → Element Groups..." or press the button . After pressing the button, a new window will open, press the "Add..." button and then enter the following data:

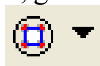
Group Number:	1
Type:	Beam
"Basic" tab	
Description:	Beam
Element Sub-Type	3-D
Include Warping DOF	Checked
Displacement:	Default
Stiffness Definition	
Use Material and Cross Section	Checked
Default Material:	1
Default Cross Section:	1

Remaining options should be left unchanged. The window view with the entered data is shown in the figure below:



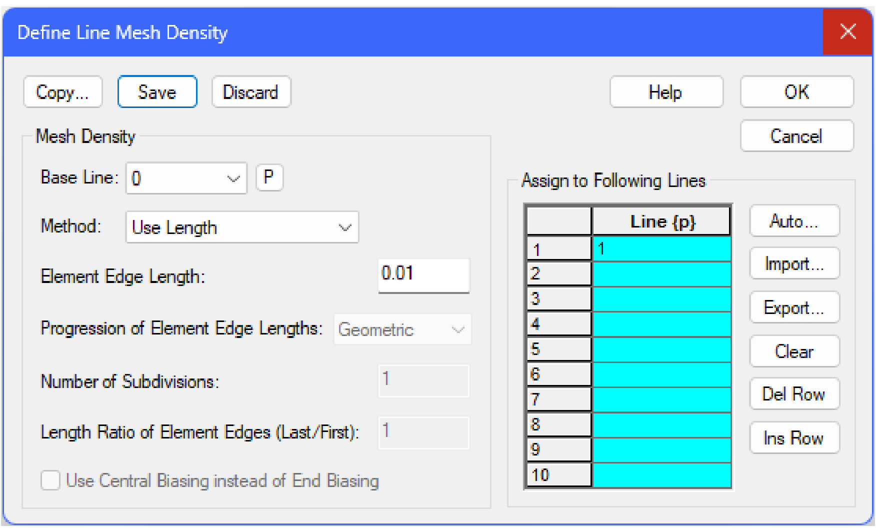
After entering the above data, press the "Save" button and then leave the window with the "OK" button.

STEP 15. Mesh subdivision

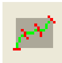
In this example, line subdivision will be used. To define the division, go to “Meshing → Mesh Density → Line...” or press the arrow next to the button  and then select “Subdivide Lines” from the available options. When a new window opens, enter the options according to the table below:

Mesh Density	
Base Line:	0
Method:	Use Length
Element Edge Length	0.01
Progression of Element Edge Lengths:	Geometric
Number of Subdivisions:	20
Length Ratio of Element Edges (Last/First):	1
Use Central Biasing instead of End Biasing	Unchecked
Assign to Following Lines	
	Line {p}
1	1

The remaining options are left unchanged. Then press the “Save” and “OK” buttons to leave the window. The window view with the entered data is shown in the figure below:

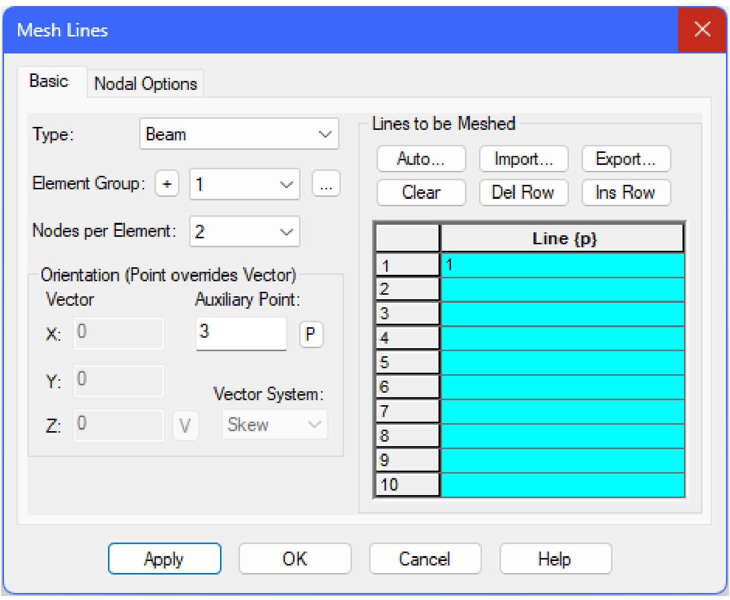


STEP 16. Definition of finite elements

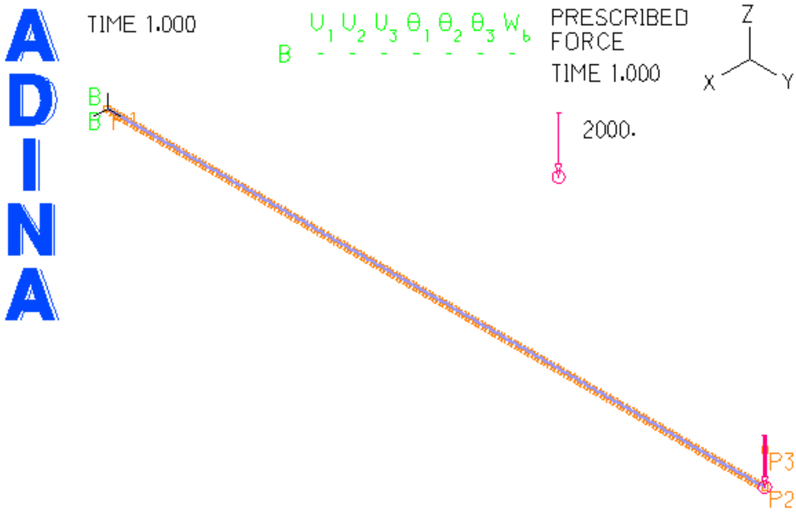
To define finite elements in the model, go to “Meshing → Create Mesh → Line...” or press the button . In the newly opened window, make sure that the type of elements used “Type:” refers to the “Beam”. The remaining data should be entered in accordance with the table below:

Type:	Beam
Element Group:	1
Nodes per Element:	2
Orientation (Point overrides Vector)	
X:	0
Y:	0
Z:	0
Auxiliary Point:	3
Vector System:	Skew
Table “Lines to be Meshed”	
	Line {p}
1	1

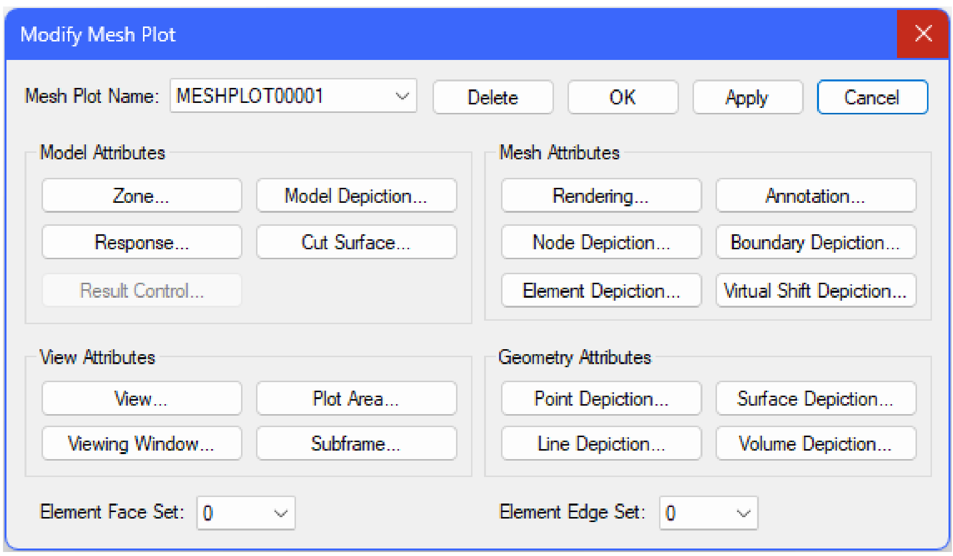
The remaining options are left unchanged. After performing these operations, press the “Apply” button and then “OK”. The window view with the entered data is shown in the figure below:



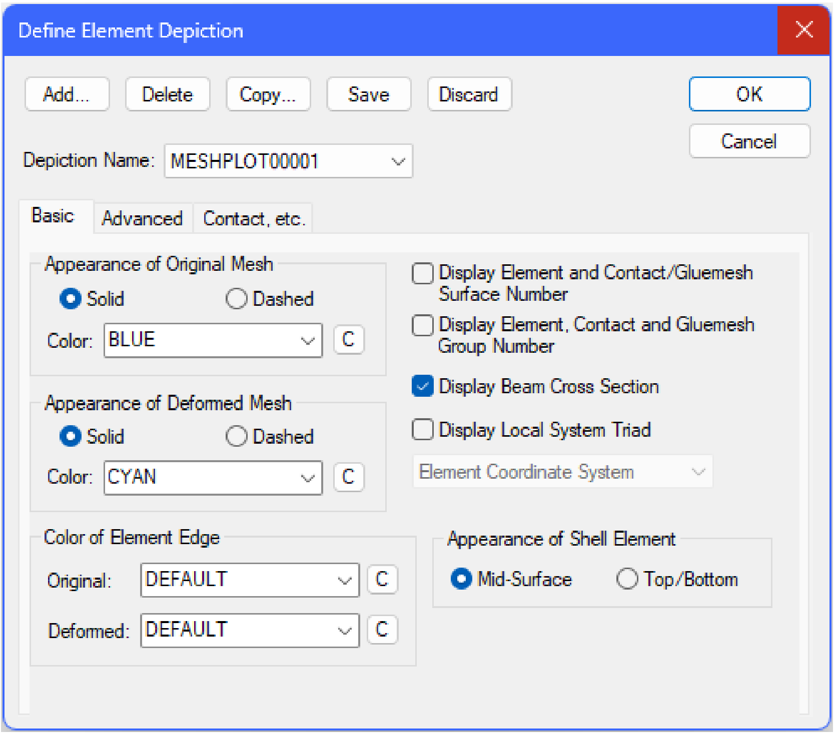
After introducing finite elements, the model should look like this:



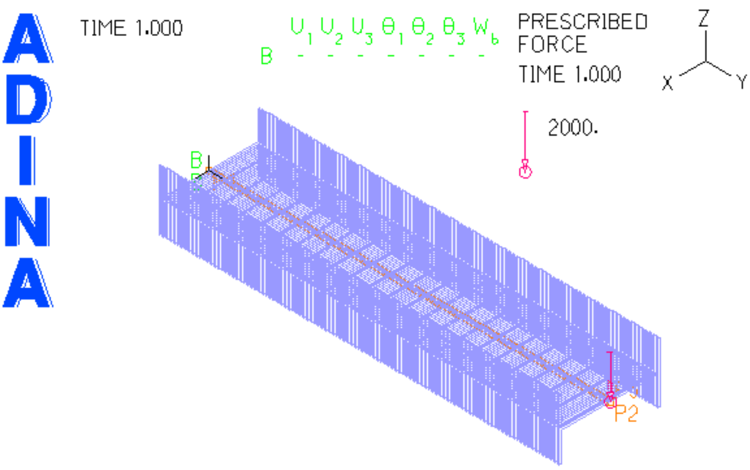
<u>Note</u>: To check whether the cross-section has been entered correctly, perform the operations described below. To do this, go to "Display → Geometry / Mesh Plot → Modify..." or press the button . After the window appears:



press the "Element Depiction" button. Another window will open, in which user must check the box for the "Display Beam Cross Section" option:



After performing the above operations, press the "Save" button and then "OK" thus leaving the "Define Element Depiction" window and then pressing "Apply" and then "OK" in the "Modify Mesh Plot" window. Currently, the main modeling window should look like this:



One can notice in the drawing above, that the height of the I-beam is consistent with the X axis, which was intended, thus one can proceed to the next step.

<u>Note</u>: In older ADINA software versions if the same cross-section is defined so that the height of the I-beam coincides with the Z axis, leaving the load axis unchanged, errors may occur during calculations. The calculations may be interrupted or the model may be recalculated – but without taking into account the external load. It is recommended to pay attention to the messages from the “Log Window” each time.

STEP 17. Save existing model to a file

Each model should have been saved to a file between few steps taken in order to not lose the data. According to that select “File → Save as...” from the menu. When a new window opens, indicate the location of the saved file and its name.

<u>Note</u>: Do not uses spaces in the file names, because it leads to an error! The space can be replaced with the underline character __.

STEP 18. Starting calculations

In order to start calculations, choose “Solution → Data File/Run” from the upper menu tabs, or choose  from the toolbars. Subsequently, input the path for saving the resultant file. However, before the user clicks the “Save” button, they should make sure that the “Run Solution” and “Automatic” function near the “Maximum Memory for Solution” options are selected in the window in the “Adina Structures Solution” group of options. After starting the calculations and recalculating the model, close all three dialogue windows which appeared during the calculations.

<u>Note</u>: Depending on the complexity of the model and the number and type of finite elements used, model calculations may take from a few seconds to even several hours.

STEP 19. Post-processing module (results)

In order to go to the results module, change “ADINA Structures” to “Post-Processing” in the drop-down list of the “Module” toolbar.

When the user is prompted that the changes in the drawing have not been saved, it is recommended to save the model by going to “File → Save” or using the button .

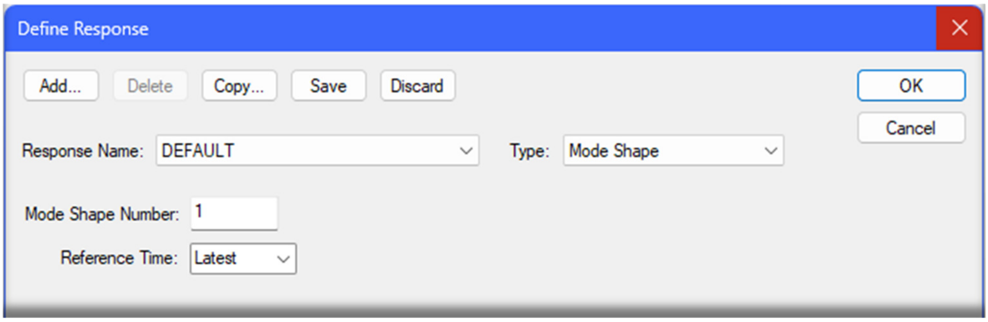
STEP 20. Opening the resultant file

In order to open the resultant file, choose  from the toolbar, or choose “File → Open” from the upper menu tabs.

<u>Note</u>: Depending on the specifications of the computer, the loading of a file in which a spatial analysis has been performed may take between about a dozen seconds up to even several minutes. The number of the applied finite elements and the number of nodes used have the greatest impact on the loading time of the file.

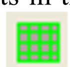
STEP 21. Displaying the vibration modes

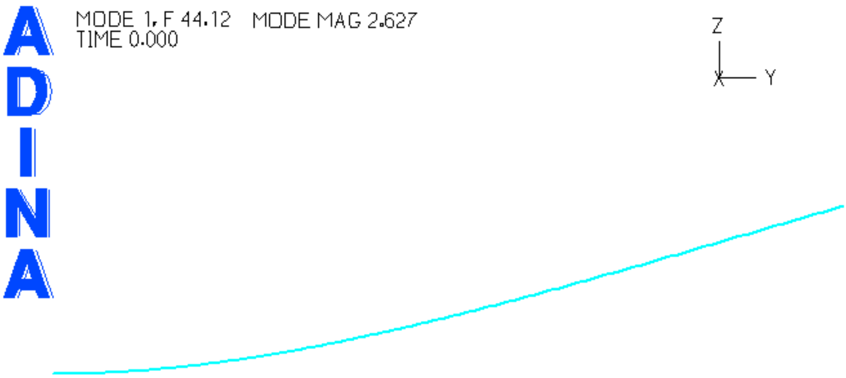
In order to display the vibration modes, first go to “Definitions → Response...”, then in the newly opened window, in the “Type:” place, select the vibration modes – “Mode Shape” from the drop-down list. The figure below shows the window with the introduced change:



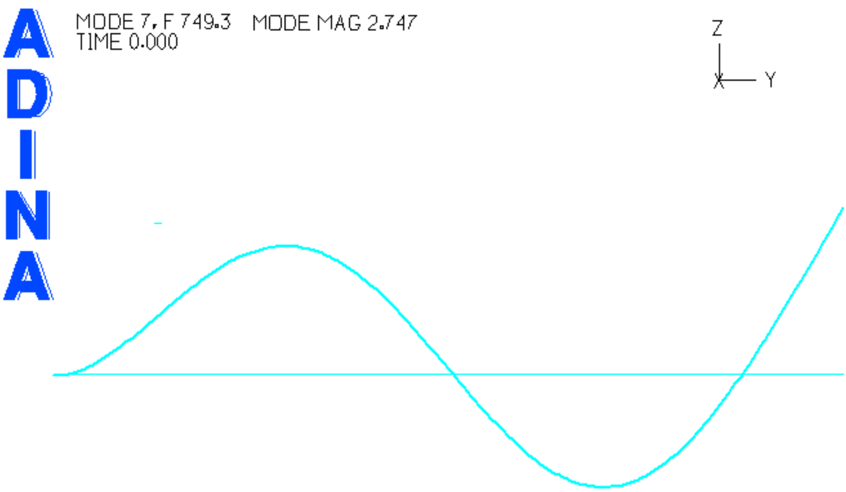
After changing the result display type, press the “Save” button and then “OK” to exit the window.

<u>Note</u>: The program will not automatically create a view of vibration mode. The main results window will contain the same model that was loaded earlier, but if the user try to change the results using the time manipulation arrows , the program will assume that the user wants to view the results concerning deflections, stresses, etc. It should be noted that pressing the manipulation arrows leads to revert changed options in the “Define Response” window back to the “Load Step”. According to that, one should once again change the “Type” option in the “Define Response” window to “Mode Shape”.

In order to display the vibration mode results in the main window, first delete the current model using the button  and then . The main results window should look like this:



Currently, subsequent vibration modes can be displayed using manipulators , e.g. vibration mode 7 looks as follows:



<u>Note</u>: “Reference Time” in the “Define Response” cannot be changed to other than “latest”. Thus, there are no vibration modes and frequency for consecutive time steps taken into analysis.

<u>Note</u>: The example uses a concentrated force on purpose to make users aware that concentrated force and any other type of force have no effect on the vibration modes and vibration frequencies obtained in the program.

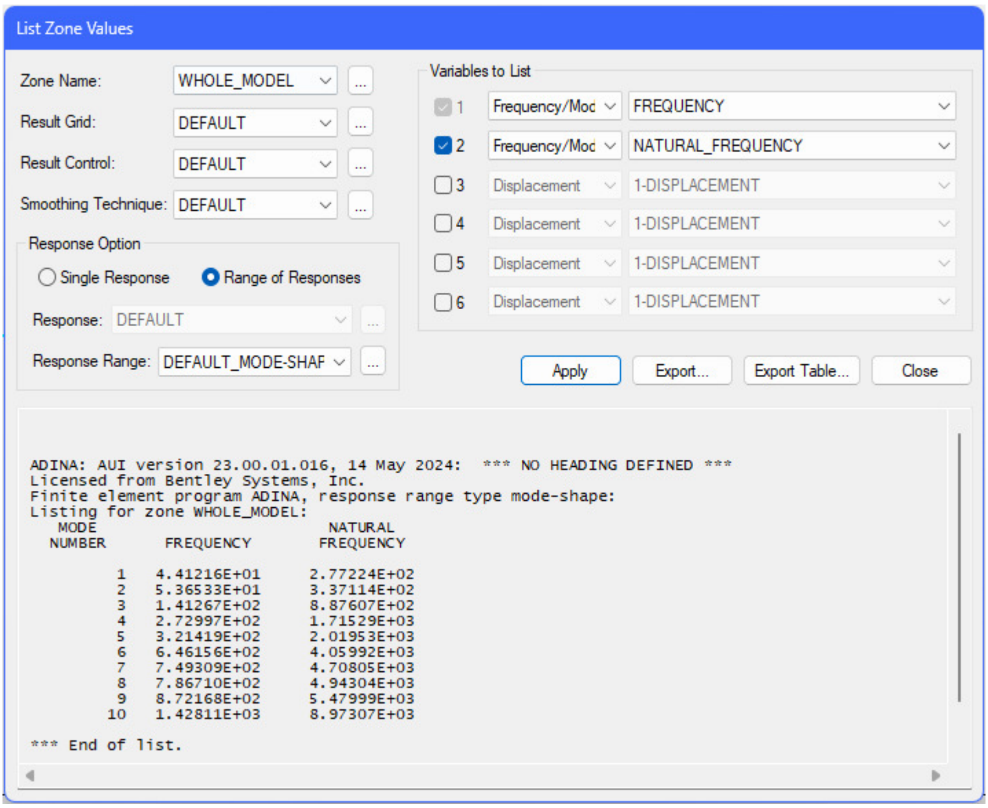
STEP 22. Displaying the vibration frequency in list form

To display vibration frequencies for a beam in the form of a list, go to “List → Value List → Zone...”. In the newly opened window make following changes:

Zone Name:	WHOLE_MODEL
Result Grid:	DEFAULT
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Response Option	
Range of Responses	Checked
Response:	DEFAULT
Response Range:	DEFAULT_MODE-SHAPE

Variables to list		
1	Frequency/Mode	FREQUENCY
2	Frequency/Mode	NATURAL_FREQUENCY

After entering all the data, press the “Apply” button. The window view with the entered data is shown in the figure below:

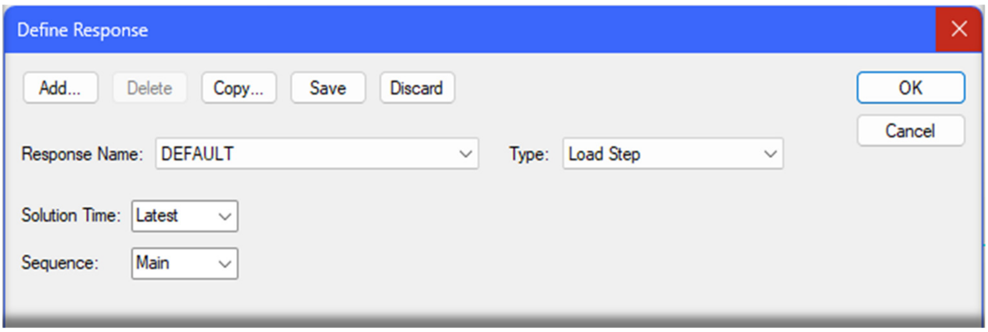


The results can also be exported to a text file using the “Export...” button indicating the file name and its location. Such files are saved with the *.txt extension.

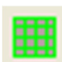
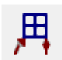
STEP 23. Displaying the reaction and support moment

Note: Since this example uses a concentrated force, the user can display maps of displacements, stresses, internal forces, etc.

First, it is needed to switch back from vibration mode results to static calculations. To do this, go to “Definitions → Response...”. Then, in the newly opened window, select the “Load Step” option in the type of results to be displayed – “Type:” from the drop-down list. The remaining data in the window is left unchanged. Then press the “Save” button and then close the window with the “OK” button. The window view with the entered data is shown in the figure below:



After performing the above operation, delete the current model 

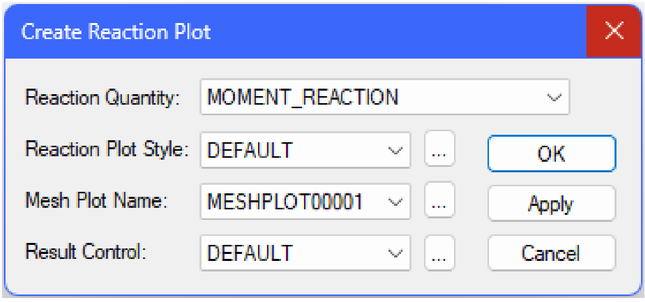
and reload the finite element mesh  . Then, go to displaying the reaction (moment in the support) in the model – from the menu, select “Display → Reaction Plot → Create...” or press the  button located on the toolbar and when a new window opens, enter the following data:

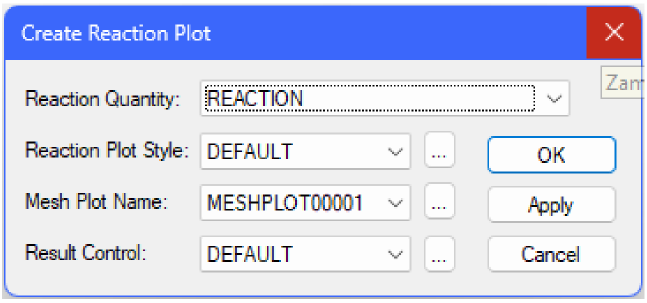
Reaction Quantity:	MOMENT_REACTION
Reaction Plot Style:	DEFAULT
Mesh Plot Name:	MESHPLOT00001
Result Control:	DEFAULT

After entering the data, press the “Apply” button, then enter the following data in the “Create Reaction Plot” window (reactions to normal/cutting forces):

Reaction Quantity:	REACTION
Reaction Plot Style:	DEFAULT
Mesh Plot Name:	MESHPLOT00001
Result Control:	DEFAULT

The views of the windows and the entered data are shown in the drawings below:



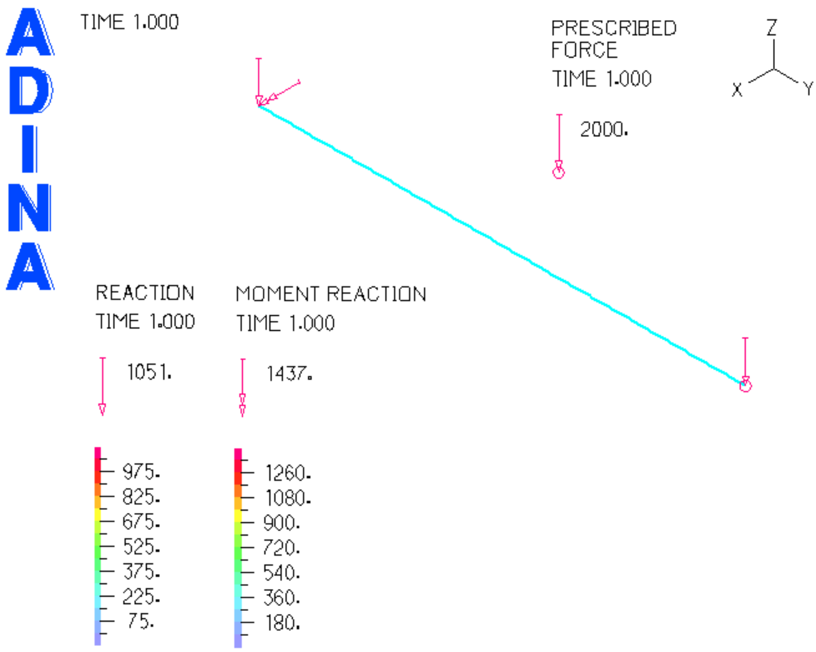


After entering above mentioned data, press the "OK" button, thus leaving the window.

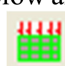
In the main results window you can also display the acting force by pressing the button .

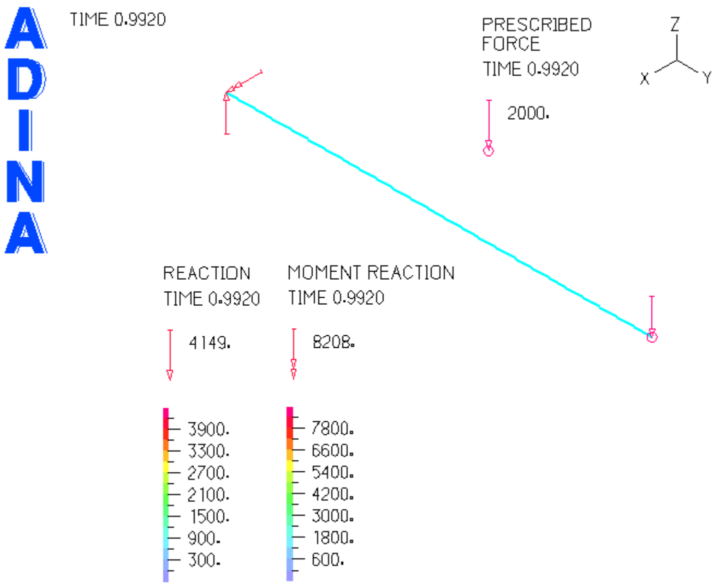
To have a good view of the support reaction – moment, you need to change the plane from which the model is visible. To do this, select the arrow next to the button  and then select the "Iso View 1" option.

Next, activate the button , select the model with the left mouse button and use mouse movements to zoom out so that it fits in the result window. Then activate the button  again and arrange the model and legends in the window so that nothing overlaps. The resulting window is shown in the figure below:



<u>Note</u>: When using time step manipulators  – it turns out that the arrows representing the moment in the support and the support reactions change. Displaying the legend again for the support moment and reaction at a different time step, it turns out that different values are obtained, even though the concentrated force was assumed to be constant over the entire length of the analysis. For example, for the time step “TIME 0.99200”, the results shown in the figure below are obtained.

The impact value has been refreshed here by pressing the button  twice and, as you can see, it is constant – 2000 N. The legend at the beginning shows the reaction and support moment values in the “TIME 1.0000” step and even though the “TIME 0.99200” step is displayed there, this is an error in program display. Please also remember that the existing reaction and support moment legend cannot be refreshed, i.e. the results are displayed in the time step in which these legends were defined.



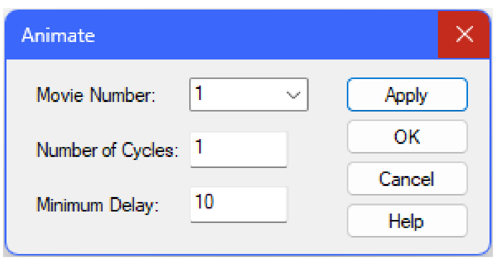
<u>**Note**</u>**:** The variability of the support reactions is related to the vibrations of the rod. The greater the amplitude in the deflection of the free end of the bar, the greater the reactions obtained in the place of clamped support. This should also be taken into account when displaying graphical and text maps of displacements, stresses, etc.

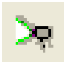
To better illustrate the essence of the problem, enable the display of scaled bar displacements by pressing the button 

, and then create an animation by pressing the button . After the animation is completed, go to "Display → Animate..." and then set the following data in the newly opened window:

Movie Number:	1
Number of Cycles:	1
Minimum Delay:	10

The window view with the entered data is shown in the figure below:



After entering the data, press the "OK" button. The animation will restart, but based on the entered data. To restart the animation defined in this way, press the button . The "Apply" button also starts the animation, however leaves the "Animate" window open, thus pressing the "OK" button will start the animation once again with predefined data.

Note: In order to obtain the influence of an external force on the vibration frequency, use the "Restart Run" option. The use of this option is thoroughly discussed for problems 8 and 26 in the ADINA program manual, which can be found by going to "Help → ADINA Primer (pdf)..." in the program window.

EXAMPLE 7
A CLAMPED-CLAMPED BEAM WITH A STEPPED CROSS-SECTION. DEFINITION OF OBJECTS WITH A SPECIFIC LIFE CYCLE IN THE COMPUTATIONAL MODEL

In this example activities related to modeling of stepped, a both ends' clamped beam subjected to different dynamic loads. It is recommended for the reader to get acquainted with the previous examples, since some of the functions were discussed earlier. A diagram of the analyzed model is presented in Figure 25.

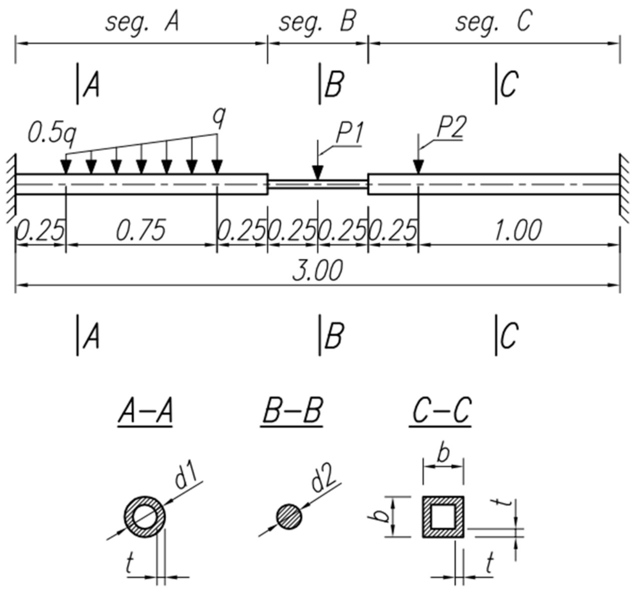


Fig. 25. Scheme of an analyzed beam

The following data is used in the analysis:

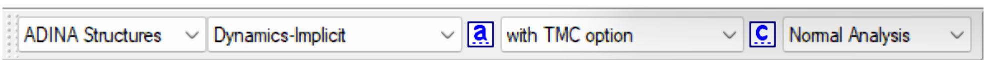
– loads:
  $P1 = 2\ \text{kN} = 2000\ \text{N}$
  $P2 = 4\ \text{kN} = 4000\ \text{N}$
  $q = 1\ \text{kN/m} = 1000\ \text{N/m}$
– material constants:
  segment A and B – steel S235JR
  $E = 210\ \text{GPa} = 2.1\text{e}^{11}\ \text{Pa}$

$\upsilon = 0.30$
$\rho = 7860 \text{ kg/m}^3$
segment C – aluminum alloy
$E = 70 \text{ GPa} = 7\text{e}^{10} \text{ Pa}$
$\upsilon = 0.33$
$\rho = 2700 \text{ kg/m}^3$
The following dimensions have been assumed for the cross-sections:
the A-A cross-section:
$d1 = 0.10 \text{ m}$
$t = 0.008 \text{ m}$
the B-B cross-section:
$d2 = 0.06 \text{ m}$
the C-C cross-section:
$b = 0.10 \text{ m}$
$t = 0.02 \text{ m}$

Note: For training purposes, the present example assumes that segment B is a segment with a specified life cycle. For technological reasons, segment B becomes demolished at a time step of $t = 0.5$ s, thus leaving behind two independent beams (segments A and C). In the SI unit system used across this example, it is recommended to assume time in seconds (not in time steps).

STEP 1. Definition of the type of analysis

Upon opening the ADINA software, choose "ADINA Structures" from the "Module Bar" in the "Program Module" section, and choose "Dynamics-Implicit" from the drop-down list next to "Analysis Type".



Note: Since there are two types of possible to choose dynamic analyzes: "Dynamic--Implicit" and "Dynamic-Explicit", it is recommended to consider time steps with a values higher than $1 \cdot 10^{-4}$ as "Dynamic-Implicit", and lower as "Dynamic-Explicit".

STEP 2. Entering the heading of the model

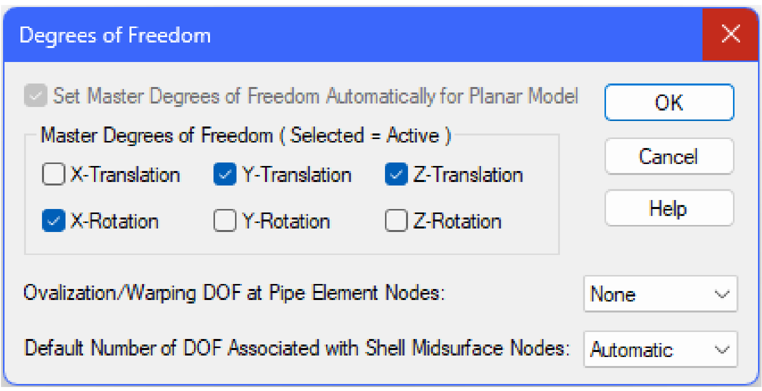
In order to specify a heading, go to "Control → Heading…". Subsequently, enter the project heading in the text box, e.g., "Stepped beam dynamics". Upon entering a heading, click the "OK" button.

STEP 3. Definition of global boundary conditions

Since the considered model is a planar model considered in the program default YZ plane, boundary conditions that do not participate in the analysis process should be excluded. To check which boundary conditions are active, go to “Control → Degrees of Freedom...”. In the newly opened window, all boundary conditions in the “Master Degrees of Freedom (Selected = Active)” option group should have their boxes checked. Make the following changes in the window:

X-Translation	Unchecked
Y-Translation	Checked
Z-Translation	Checked
X-Rotation	Checked
Y-Rotation	Unchecked
Z-Rotation	Unchecked

The figure below shows the window with selected boundary conditions:



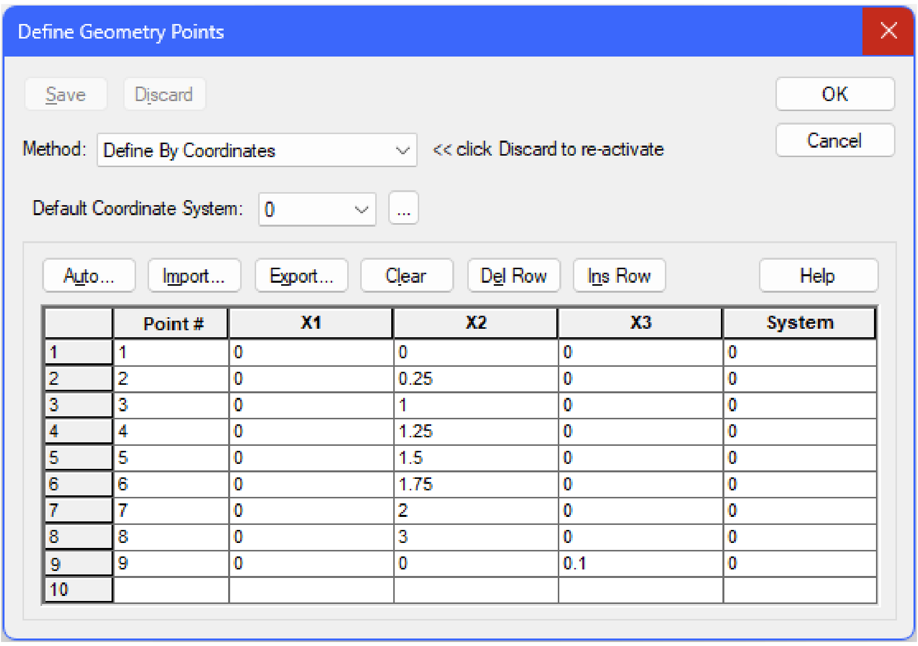
The remaining data is left unchanged. After selecting the appropriate boundary conditions involved in the analysis, exit the window by pressing the “OK” button.

STEP 4. Definition of points

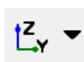
In order to define points in the model, go to “Geometry → Points → Define… “ from the menu or press the button . In the newly opened window, add points according to the table below:

Point #	X1	X2	X3	System
1	0	0	0	0
2	0	0.25	0	0
3	0	1.00	0	0
4	0	1.25	0	0
5	0	1.50	0	0
6	0	1.75	0	0
7	0	2.00	0	0
8	0	3.00	0	0
9	0	0	0.1	0

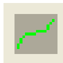
Subsequently, click "Apply" and "OK" in the main window for defining points. The window view with the entered data is shown in the figure below:



STEP 5. Displaying point ID numbers

To display point identifiers in the YZ plane, press the  button on the toolbar, and then press the  button, respectively.

STEP 6. Definition of lines

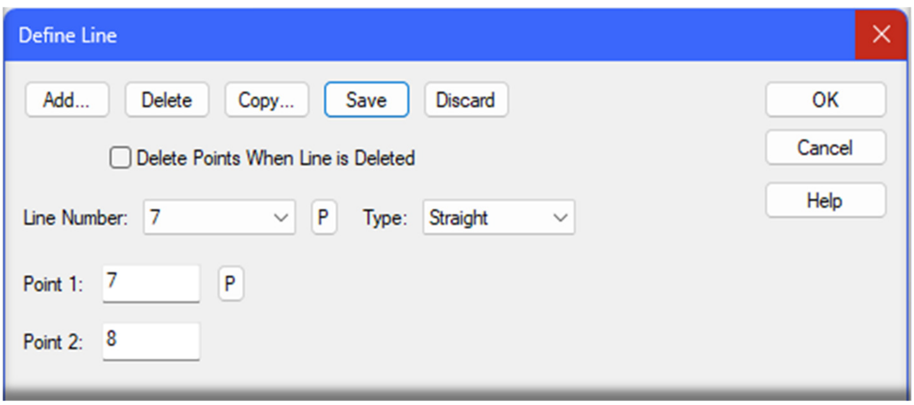
All lines in the present example are straight lines, so the only one type of will be used – "Straight". In order to define a line, choose "Geometry → Lines → Define…" from the upper menu, or click . Then click the "Add…" button in the newly opened window in order to add the first line.

<u>Note</u>: After entering data from each table row, press the "Save" button and then "Add..." to add a new line from the next row until data from the last row is entered.

Enter the data according to the table below:

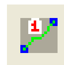
Line Number	Point 1	Point 2	Type
1	1	2	Straight
2	2	3	Straight
3	3	4	Straight
4	4	5	Straight
5	5	6	Straight
6	6	7	Straight
7	7	8	Straight

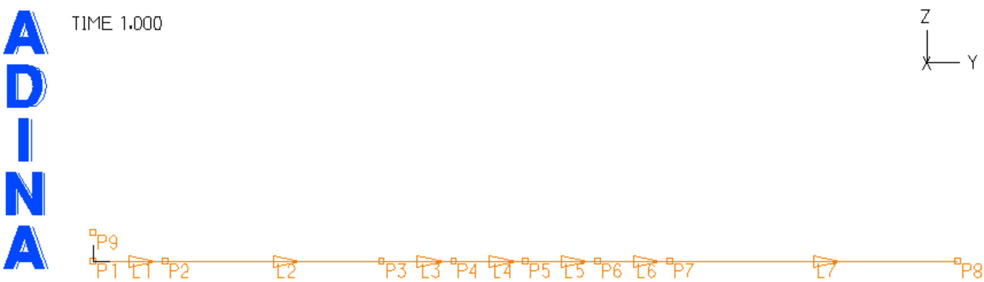
When all the lines have been defined, click the "OK" button in order to confirm the input values. The window with data entered for the line with ID "7" is shown in the figure below:



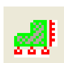
Note: When adding lines one by one, there is no need to click the “Save” button. Just click the “Add…” button after inputting the identification numbers of points. The line will be created, and the user will go to the generation of a new line at once. However, clicking the “Save” button enables avoiding a situation in which, after closing the window with the X or the “Cancel” button, the last created line is not created.

STEP 7. Displaying line ID numbers

In order to display the numbers of lines for their easier identification, click . The model should look like in the figure below:

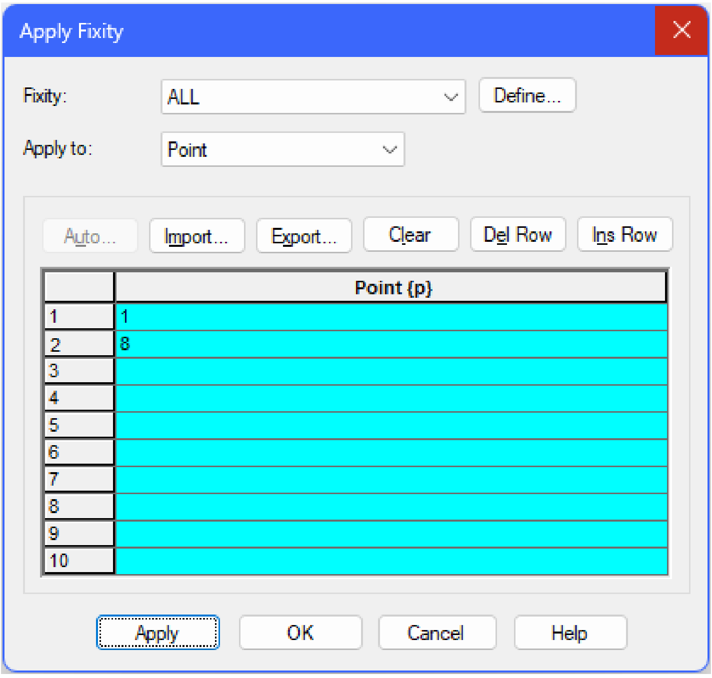


STEP 8. Definition of boundary conditions (fixity characteristics)

In order to define a fixity in the system, go to “Model → Boundary Conditions → Apply Fixity…”, or click . Since there are two clamped supports in the model, there is no need to define a new fixity, since a fixity named “ALL” has all the possible directions of translation and rotation blocked. When a new window opens, enter the following data:

Fixity:	ALL
Apply to:	Points
	Point {p}
1	1
2	8

After entering the data, press the “Apply” button and then leave the window with the “OK” button. Window presenting entered data is shown below:



<u>Note</u>: In the previous ADINA versions if no value is declared for a given point in the "Fixity" column (the box remains blank), upon clicking the "Save" button the program will automatically fill out the blank boxes with the boundary condition specified in the "Default Fixity" drop-down list.

STEP 9. Displaying boundary conditions

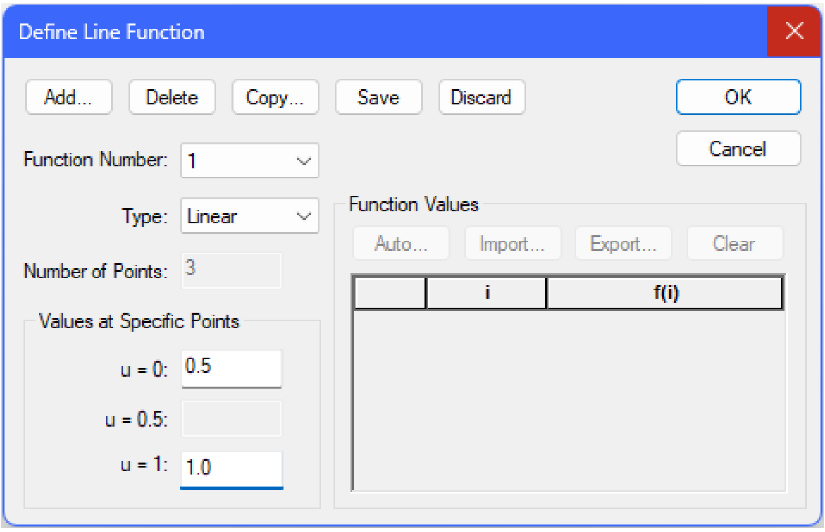
In order to display the defined boundary conditions in the main model window, click .

STEP 10. Definition of load variability

Since in the present example a "trapezoidal" load is used, it is necessary to create an additional function specifying the increment of the force multiplier in selected points. In order to define such a function, go to "Geometry → Spatial Functions → Line…", then in the newly opened window click the "Add…" button. As shown in the analyzed scheme figure one can notice, that from the left side the "trapezoidal" load has 0.5q, and on the right side 1.0q. To obtain such an increment in line load enter the following data in the "Define Line Function":

Function Number:	1
Type:	Linear
Values at Specific Points	
u = 0:	0.5
u = 0.5:	
u = 1.0:	1.0

A view of properly declared values in the window is presented in the figure below:



After entering the data, press the "Save" button, then you can close the window with the "OK" button.

STEP 11. Definition of load variability (additional informations)

It is possible to apply a load to a construction in three various ways: a variable load (2 points), a variable load described by a parabolic function, or a load declared by means of a table (linear variability).

Three various types of loads will be examined for the present example, which exists in a single plane (2D). For the below presented examples it is assumed that the bar length is equal 1.00 m, and a linear distributed force is equal 1000 N.

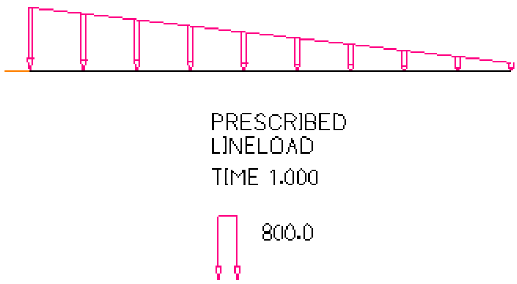
Load patterns along with the input data being possible to be declared in the software are presented below.

A linearly variable load

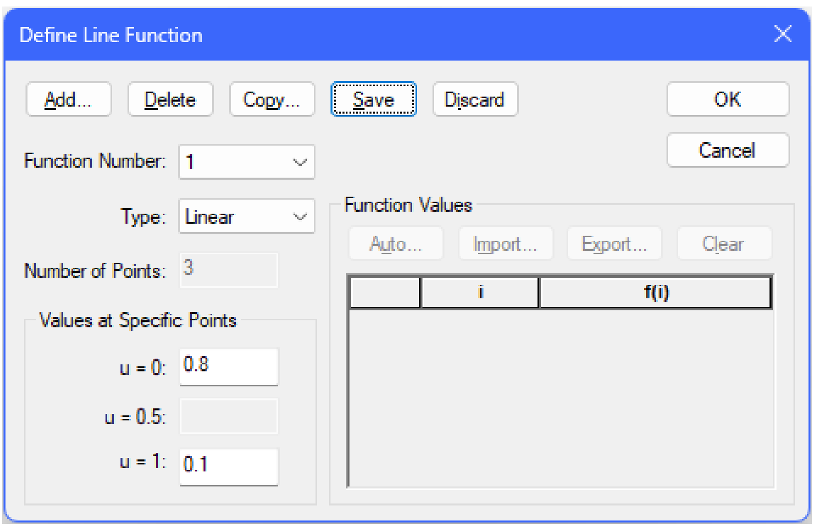
The value of load is determined by the values used in the “u = 0” and “u = 1” points. Since a value of “0.8” is assumed for “u = 0” and “0” for the “u = 1.0” as shown in the table below:

Function Number:	…
Type:	Linear
Values at Specific Points	
u = 0:	0.8
u = 0.5:	
u = 1.0:	0.1

the value of force will reach value of $1000 \cdot 0.80 = 800$ N in the left node and 100 N in the right node. The load scheme would look similar to that:



The window with entered data would be:

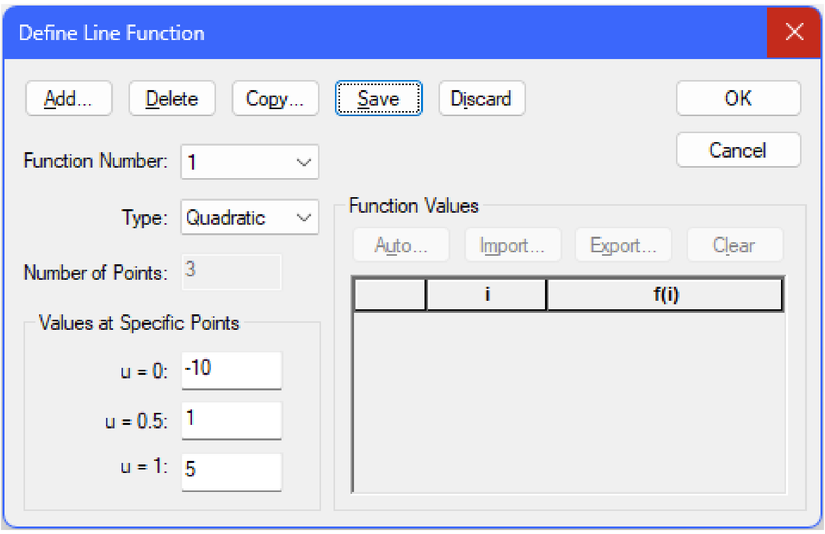


Determining a load by means of a parabolic function

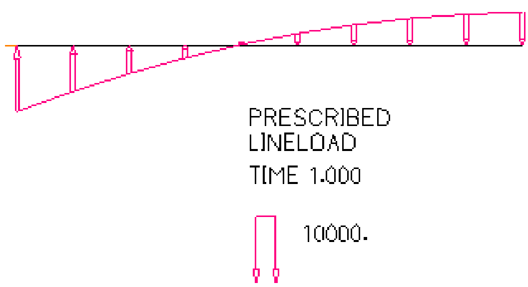
It should be noted that in case of declaring a load, it is also possible to use negative values, which means that the load will have an opposite direction. In this case following data are used to obtain parabolic function of load pattern:

Function Number:	…
Type:	Quadratic
Values at Specific Points	
u = 0:	–10
u = 0.5:	1
u = 1.0:	5

The window with introduced data would be:



thus the load pattern would be:



Definition of any function for a continuous load

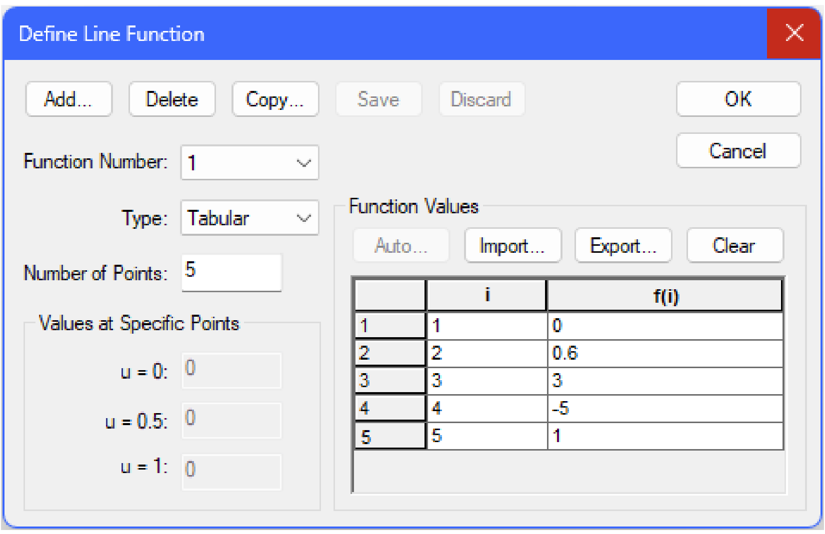
It is possible to declare nearly any function for a continuous load. For example, when following data are introduced:

Function Number:	…
Type:	Tabular
Number of points:	… (i)

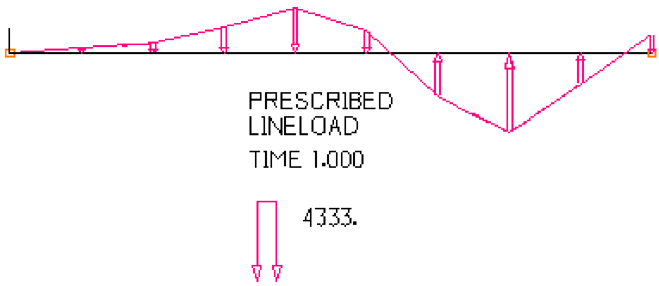
In the table, are introduced "i" numbers characterizing virtual nodes created on a beam span in which the load will be applied. It means that if a user inputs 5 different rows, the beam will be divided into "i – 1" spans, and between these spans the data from "f(i)" column will be used. Moreover, it should be noted that between the consecutive "f(i)" a linear interpolation is used. Only in external virtual nodes (first/last) the value of force is the same as in the introduced in the table. For example, by introducing following values in the table:

	i	**F(i)**
1	1	0
2	2	0.6
3	3	3
4	4	–5
5	5	1

the window of "Define Line Function" would look like:

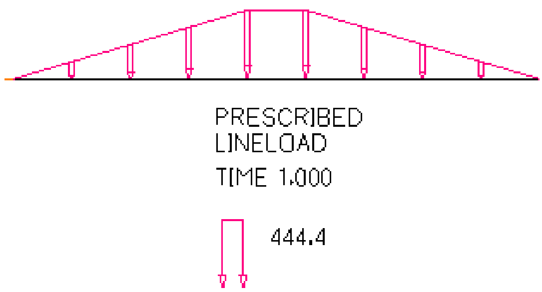


and the continuous line load would reach following variability:



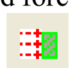
One can notice, that in the point "i" = 4, the value for the function used was "–5" thus one could expect the value of 5000 N, however, the value there is 4333 N. That software behaviour is connected with the use of linear interpolation between points.

Note: When declaring the abovementioned function, with division into, e.g., three segments, where the first and last values are 0, while the middle one the value is equal 0.5, a proper result will not be achieved (see the figure below).



It is clearly visible on the abovementioned scheme, the segment along which the load is distributed was not divided precisely into three sections, with the highest value of load occurring in the middle. On the other hand, a flattening was observed in the upper part of the graph, along with a value not corresponding to the adopted assumptions.

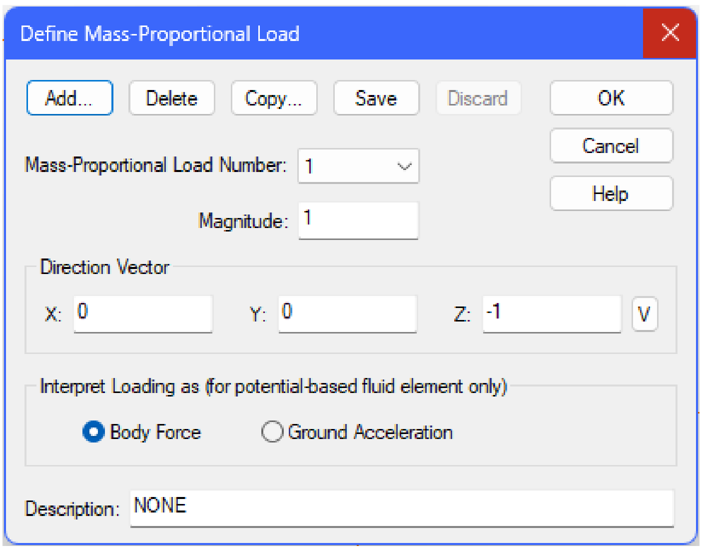
STEP 12. Definition of loads

Three different types of loads will be defined in present model – own structure's weight, linear load and concentrated force. In order to define loads, go to "Model → Loading → Apply..." or press the  button. After opening a new window, the self-weight load of the structure will be declared first, so for the "Load Type:" option, select "Mass Proportional" from the drop-down list, then for the "Load Number:"

option, press the “Define...” button. A new window will open, in which one should press “Add...” button to add a new load, then enter the data as shown in the table:

Mass Proportional Load Number:	1
Magnitude:	1
Body #:	1
Direction Vector	
X:	0
Y:	0
Z:	–1
Interpret Loading as (for potential-based fluid element only)	
Body Force	Checked
Description:	None

After entering the data in the window, press the “Save” button and then “OK”, thus returning to the “Apply Load” window. The window with the entered data is shown in the figure below:



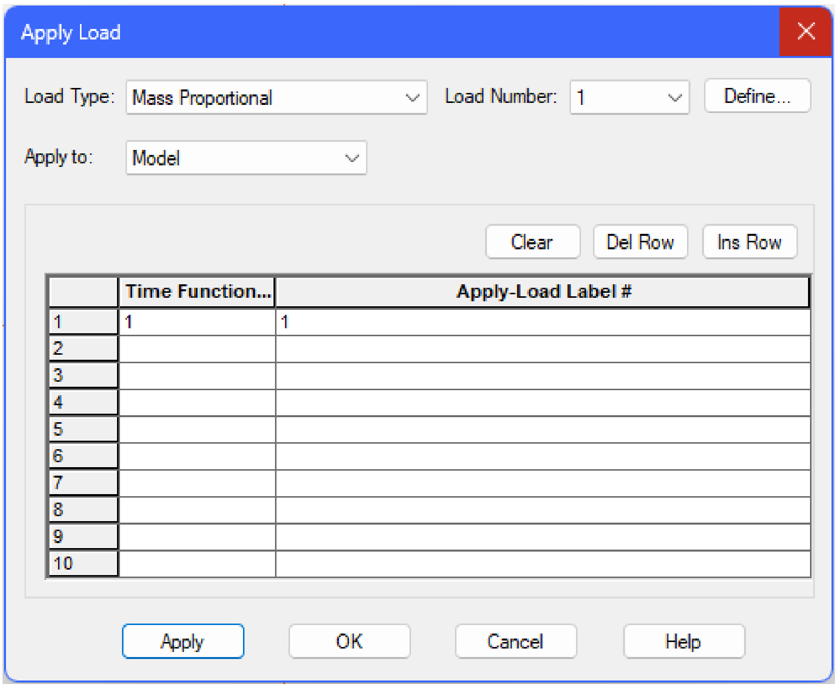
The “Apply Load” window should therefore be completed as follows:

Load Type:	Mass Proportional
Load Number:	1
Apply to:	Model

In the table, the data should look like this:

	Time Function…	**Label #**
1	1	1

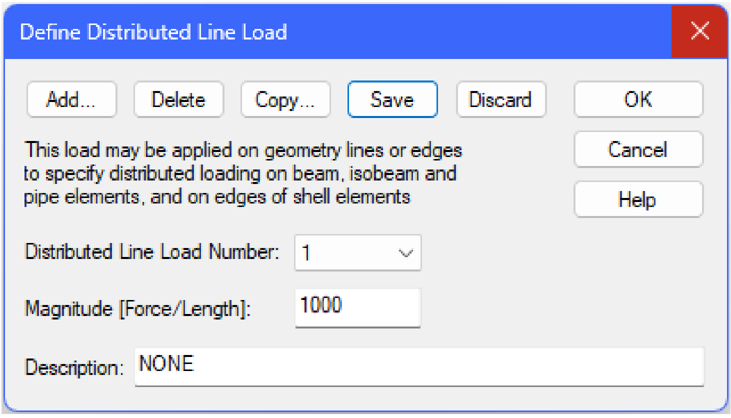
The window view with the entered data is shown in the figure below:



After entering the data, press the “Apply” button, and then, without leaving the “Apply Load” window, select “Distributed Line Load” from the drop-down list next to “Load Type:”, and then press the “Define...” button next to the drop-down list of the “Load Number:” field. In the newly opened window, press the “Add...” button and enter the following data:

Distributed Line Load Number:	1
Magnitude (Force/Length):	1000
Description:	None

After entering the data, the window should look like this:



After declaring the load value, press the "Save" button and then "OK" closing the "Define Distributed Line Load" window and return to the "Apply Load" window. In the "Apply Load" window following data should be entered:

Load Type:	Distributed Line Load
Load Number:	1
Apply to:	Line

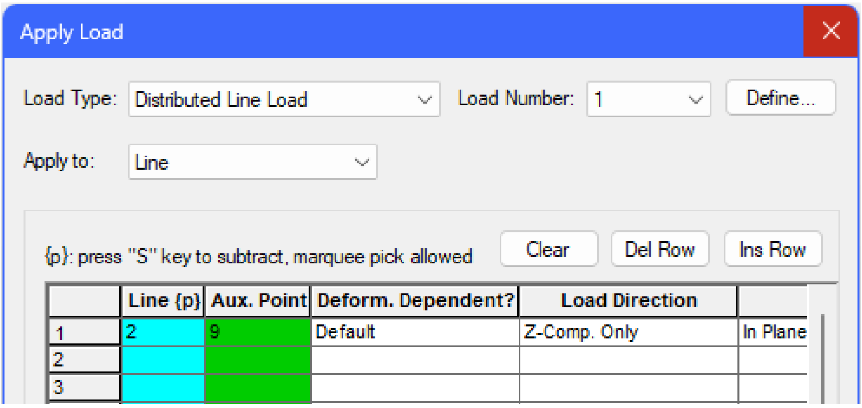
In the table, the data should look like this:

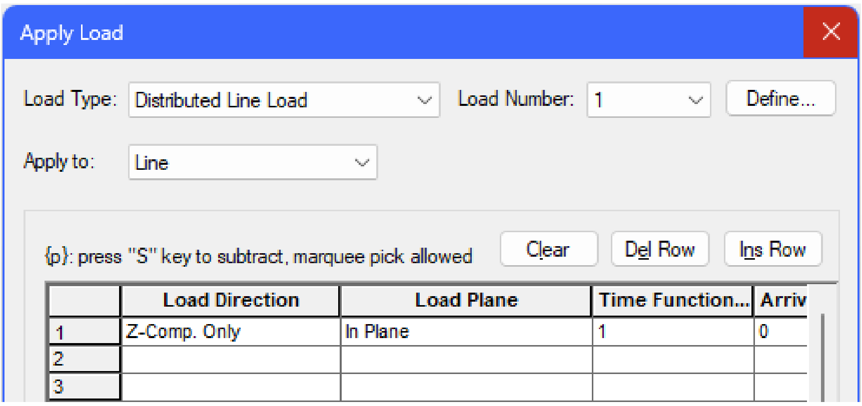
	Line {p}	**Aux. Point**	**…**	**Load Direction**	**…**	**Time Function**
1	2	9	Empty	Z-Comp. Only	Empty	1

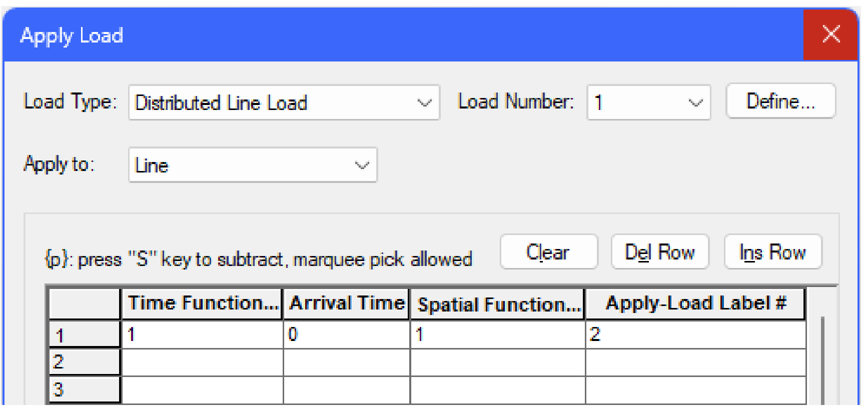
	Line {p}	**Aux. Point**	**…**	**Spatial Function…**	**Apply-Load Label #**
1	2	9	Empty	1	2

After entering the data, press the "Apply" button – this will cause the program to fill in the empty spaces with default values.

The view of the windows along with the entered data is shown in the figures below:







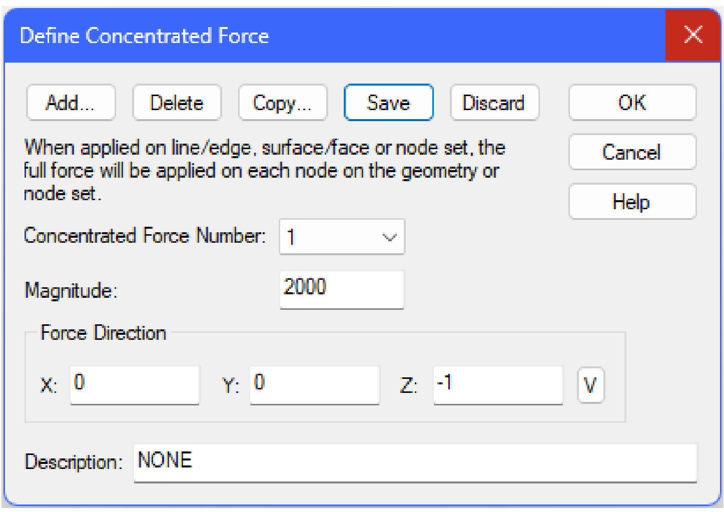
Lastly, without closing the "Apply Load" window, change the load type "Load Type:" by selecting the "Force" option from the drop-down list – concentrated force. Declare the force again by pressing the "Define..." button located next to the drop-down list for the "Load Number:" option. When a new window opens, enter the following data:

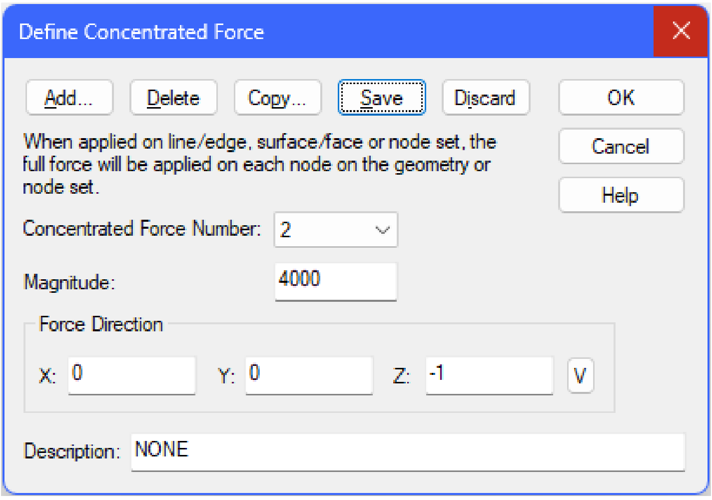
Concentrated Force Number:	1
Magnitude:	2000
Force Direction	
X:	0
Y:	0
Z:	–1
Description:	None

Press "Add…" button once again and enter following data:

Concentrated Force Number:	2
Magnitude:	4000
Force Direction	
X:	0
Y:	0
Z:	–1
Description:	None

After entering the data, for both forces, the definition windows should look similar to that presented below:





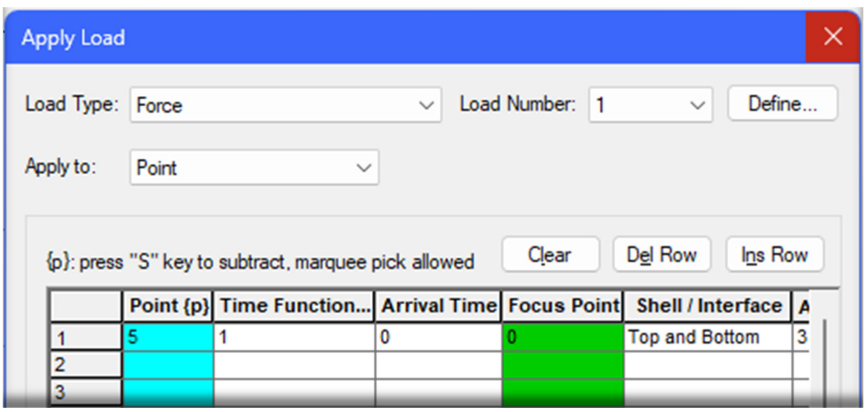
After entering the data, press the "Save" button and then close the window with the "OK" button. Returning to the "Apply Load" window, complete it as follows:

Load Type:	Force
Load Number:	1
Apply to:	Point

In the table, the data should look like this:

	Point {p}	**Time Function…**	**…**	**Label #**
1	5	1	Empty	3

The view of the "Apply Load" window with the entered data is shown in the figure below:



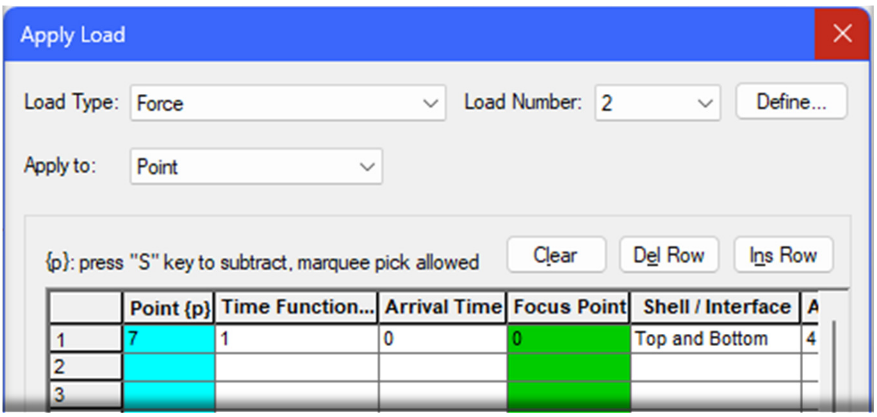
After entering the data, press the "Apply" button – the program will automatically fill the empty table columns with default values. Now in the same window make following changes:

Load Type:	Force
Load Number:	2
Apply to:	Point

In the table, the data should look like this:

	Point {p}	**Time Function…**	**…**	**Label #**
1	7	1	Empty	4

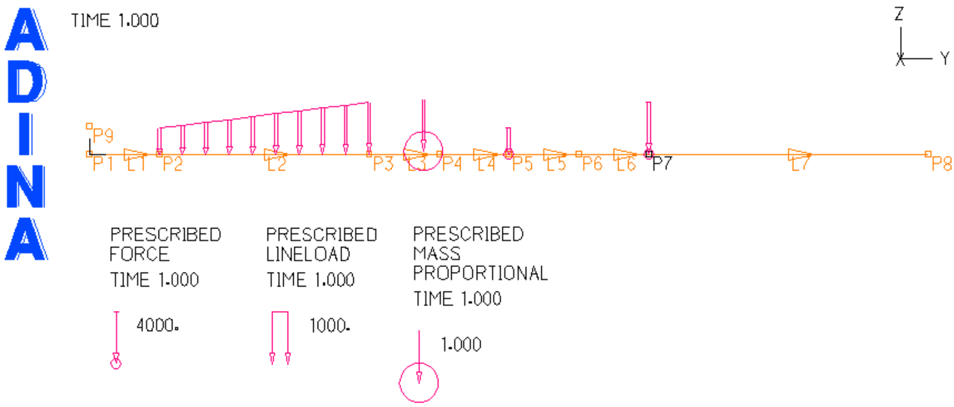
The view of the "Apply Load" window with the entered data is shown in the figure below:



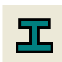
After entering all the data, press the "Apply" button – the program will automatically fill the empty table columns with default values. After performing all the above operations, you exit the window by pressing the "OK" button.

STEP 13. Displaying active model loads

In order to display the defined loads in the main model window, click . The model window should look like this:



STEP 14. Definition of cross-section(s)

In order to define cross-section shapes, select “Model → Cross-Sections...” from the menu or press the button . After opening of a new window, press the “Add...” button and enter the following data:

Section Number:	1
Type:	Pipe
Dimensions	
Diameter D:	0.10
Thickness T:	0.008

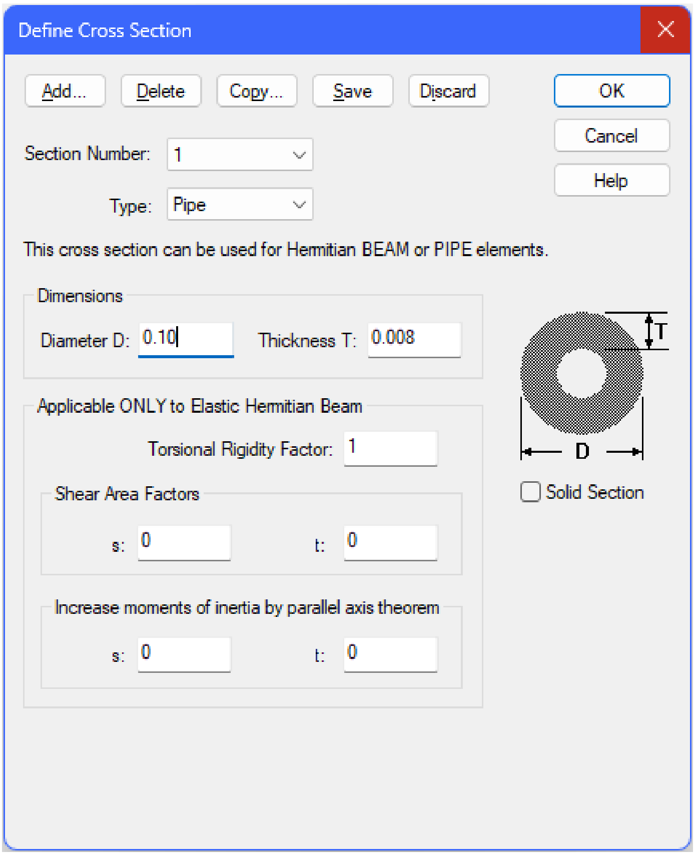
Remaining options left unchanged. Press the “Save” button, and once again click “Add…”. The second cross-section is a circular solid bar. Thus, for the second cross-section input following data:

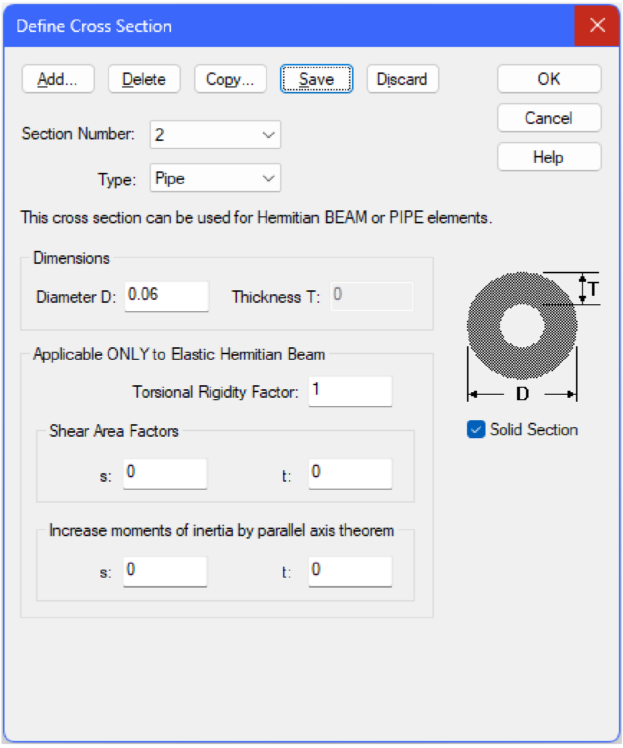
Section Number:	2
Type:	Pipe
Dimensions	
Diameter D:	0.06
Solid Section	Checked

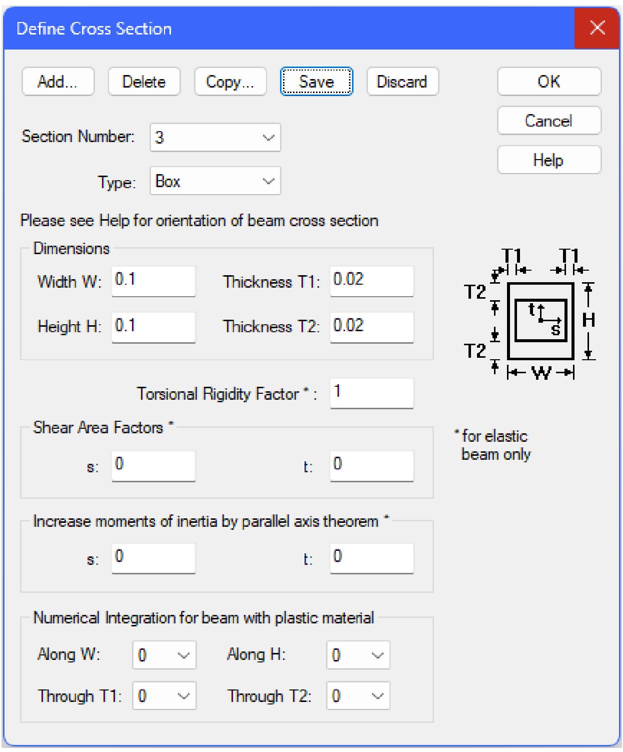
Once again, the remaining options in the window remain unchanged; save the changes with the “Save” button, and add another cross-section with the “Add…” button. The third cross-sections corresponds to the box-like profile. According to that enter the following data in the “Define Cross Section” window:

Section Number:	3
Type:	Box
Dimensions	
Width W:	0.10
Height H:	0.10
Thickness T1:	0.02
Thickness T2:	0.02

The remaining options are left unchanged. Finally, save the input values with the "Save" and "OK" buttons. Windows with introduced data are presented below:

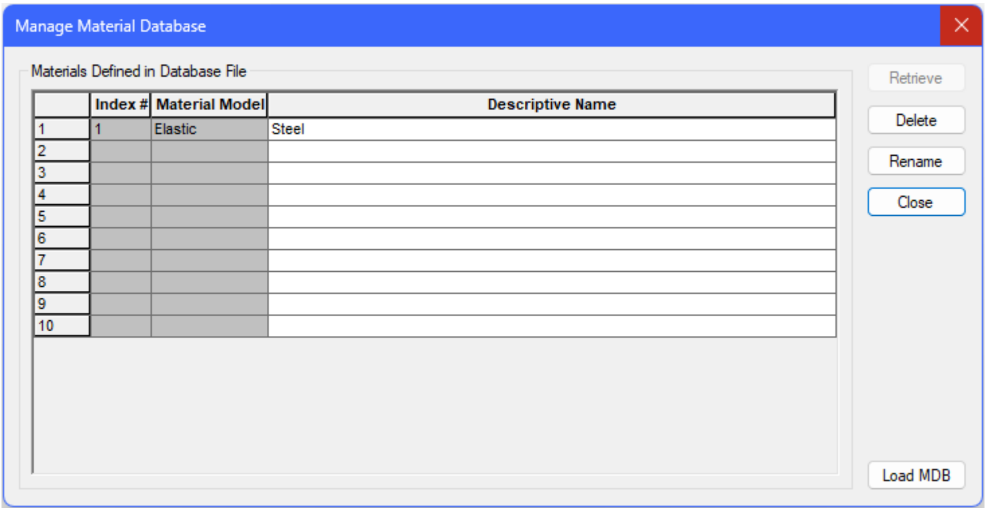



Define Cross Section
Add...
Delete
Copy...
Save
Discard
OK
Cancel
Help
Section Number: 2
Type: Pipe
This cross section can be used for Hermitian BEAM or PIPE elements.
Dimensions
Diameter D: 0.06
Thickness T: 0
T
D
Solid Section
Applicable ONLY to Elastic Hermitian Beam
Torsional Rigidity Factor: 1
Shear Area Factors
s: 0
t: 0
Increase moments of inertia by parallel axis theorem
s: 0
t: 0


Define Cross Section
Add...
Delete
Copy...
Save
Discard
OK
Cancel
Help
Section Number: 3
Type: Box
Please see Help for orientation of beam cross section
Dimensions
Width W: 0.1
Thickness T1: 0.02
Height H: 0.1
Thickness T2: 0.02
T1
T1
T2
T2
t
s
H
W
Torsional Rigidity Factor * : 1
Shear Area Factors *
s: 0
t: 0
* for elastic beam only
Increase moments of inertia by parallel axis theorem *
s: 0
t: 0
Numerical Integration for beam with plastic material
Along W: 0
Along H: 0
Through T1: 0
Through T2: 0

STEP 15. Definition of material constants

In the present example there are two different materials which need to be defined. In order to define materials constants, go to "Model → Materials → Manage Materials..." or press the button , then in the newly opened window (if the material has been previously added to the program database), press the "Get MDB" button. A new window with a table should open, then select the material called "Steel" in the table and press the "Retrieve" button. After loading the material from the program database, press the "Close" button. In the "Manage Materials" window, in its lower part, in place of the table, there should be a material – "Steel" with the id number "1". The retrieving material window is presented below:



If the database does not contain the material required in the example, in the "Manage Materials" window, find the group relating to elastic materials – "Elastic", and then press the "Isotropic" button – isotropic material. In the newly opened window, press the "Add..." button and enter the data in accordance with the table below:

Material Number:	1
Description:	Steel
Young's Modulus (> 0)	2.1e11
Poisson's Ratio (–1.0 < NU < 0.5)	0.30
Density	7859
Coef. of Thermal Expansion (>= 0)	0

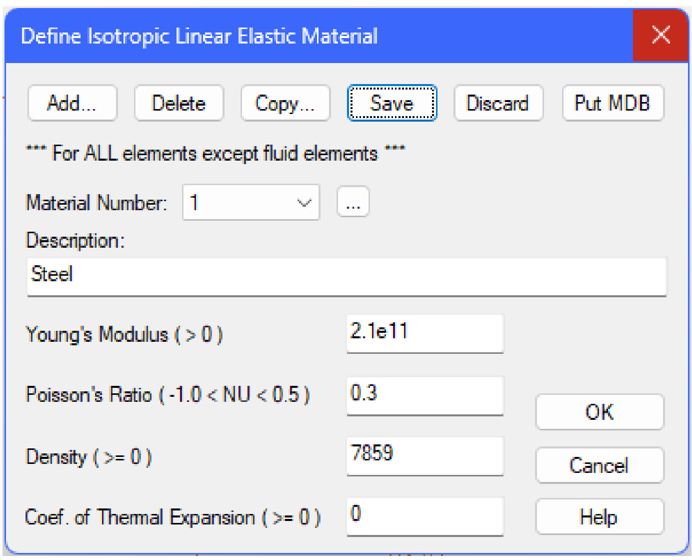
After entering the values, press the “Save” button. If the user wants to add material to the program's database, press the “Put MDB” button. Close the window the “OK” button.

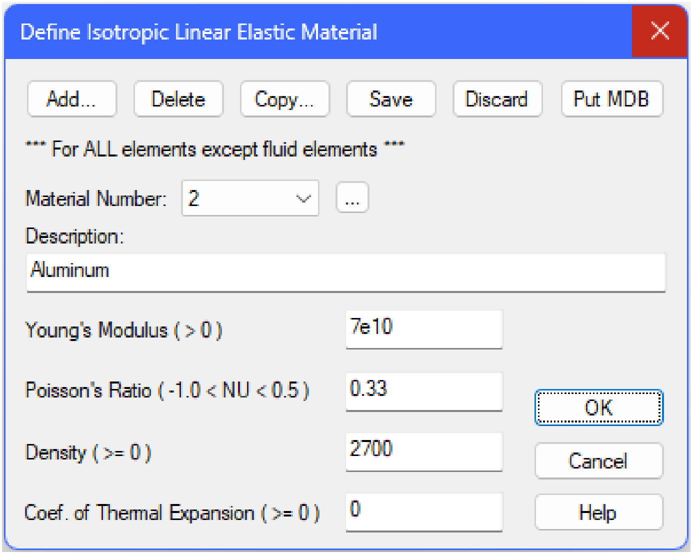
For now, the second material will be added. From the “Elastic” group of materials press the “Isotropic” button. In the newly opened window hit “Add...” and enter following data:

Material Number:	2
Description:	Aluminum
Young’s Modulus (> 0)	7e10
Poisson’s Ratio (–1.0 < NU < 0.5)	0.33
Density	2700
Coef. of Thermal Expansion (>= 0)	0

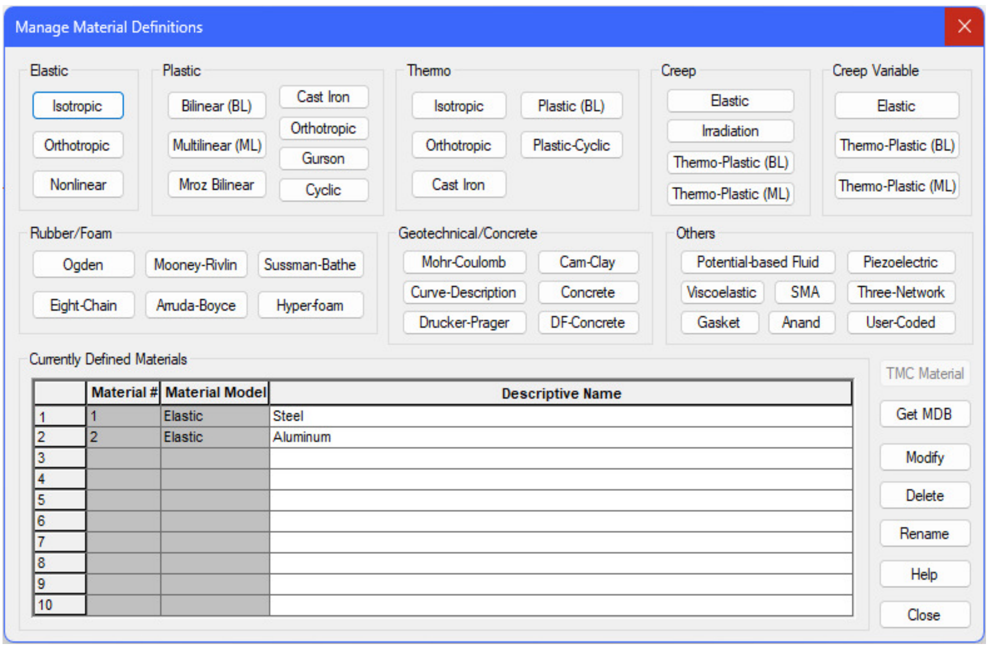
After completing the operation, exit the window by pressing the “OK” button. Again, as in the previous case, after leaving the window, in the lower part of the “Manage Material Definitions” window, the table should contain the “Aluminum” material model with an assigned identification number of “2”.

The window views with the entered materials in the elastic material definition window are shown in the figures below:





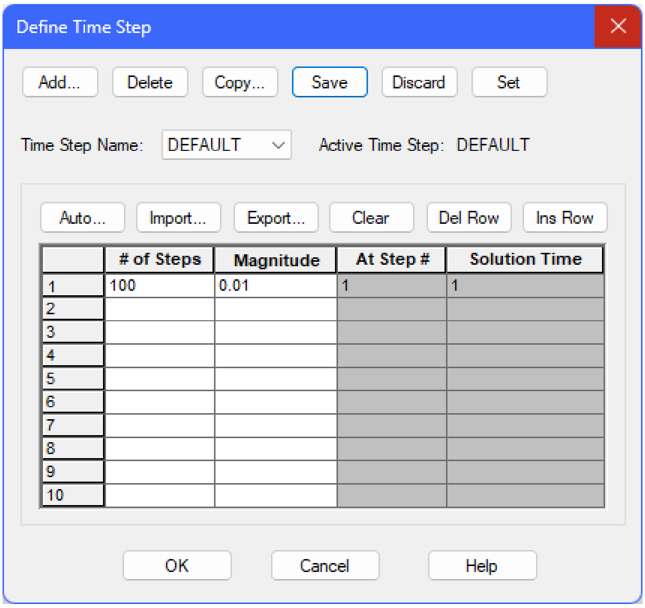
The view of the material manager window should look similarly as:



After performing all operations, exit the "Manage Material Definitions" window by pressing the "Close" button.

STEP 16. Definition of the number of time steps

100 time steps will be used for the purpose of the present example, each with a value of 0.01 s. Therefore, the total duration of the analysis will fill the range of 0 up to 1 s. In order to declare the number of time steps, go to "Control → Time Step…". Then, in the newly opened window, enter the value of "100" in the first row of "# of Steps" column and in the same row under the "Magnitude" field, make sure to set the value to "0.01". The window view is shown in the figure below:

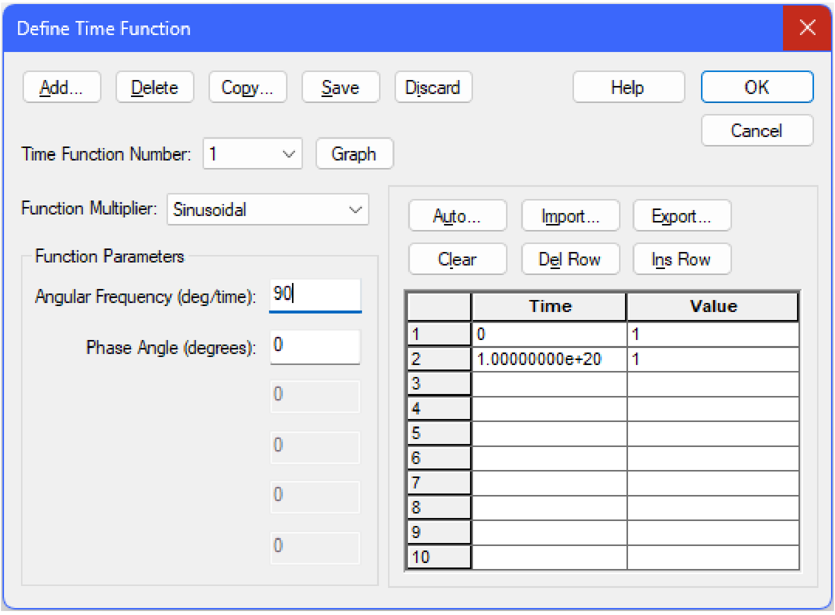


STEP 17. Definition of load variability in subsequent time steps

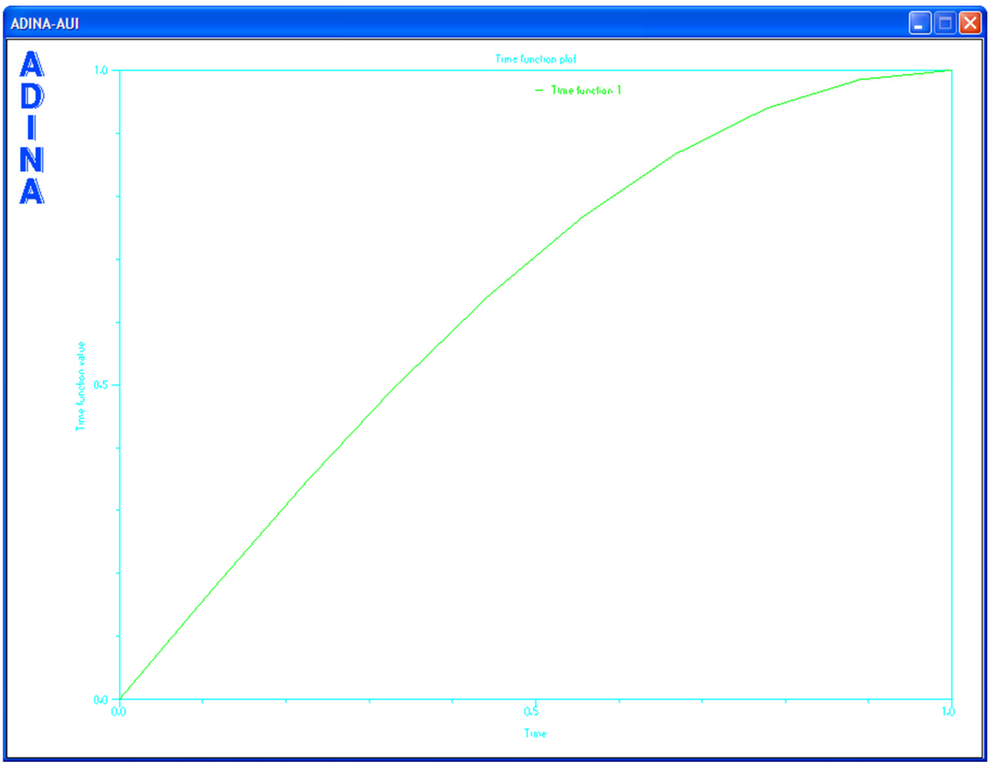
In the present example, a sine function is used to declare a nonlinear increase in load over time. In order to declare the variability of load over time, go to "Control → Time Function…", and then enter the following data:

Time Function Number:	1
Function Multiplier:	Sinusoidal
Angular Frequency (deg/time):	90
Phase Angle (degrees):	0

The remaining options are left unchanged. The value of 90 in "Angular Frequency (deg/time):" means that the load will reach its maximum value over a time of 1 s). As soon as changes are completed save the data via "Save" button and close the window with the "OK" button. A view of the values input in the window is presented in the figure below.

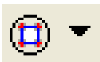


A graph of the variability of load over time is in turn presented in the figure below:



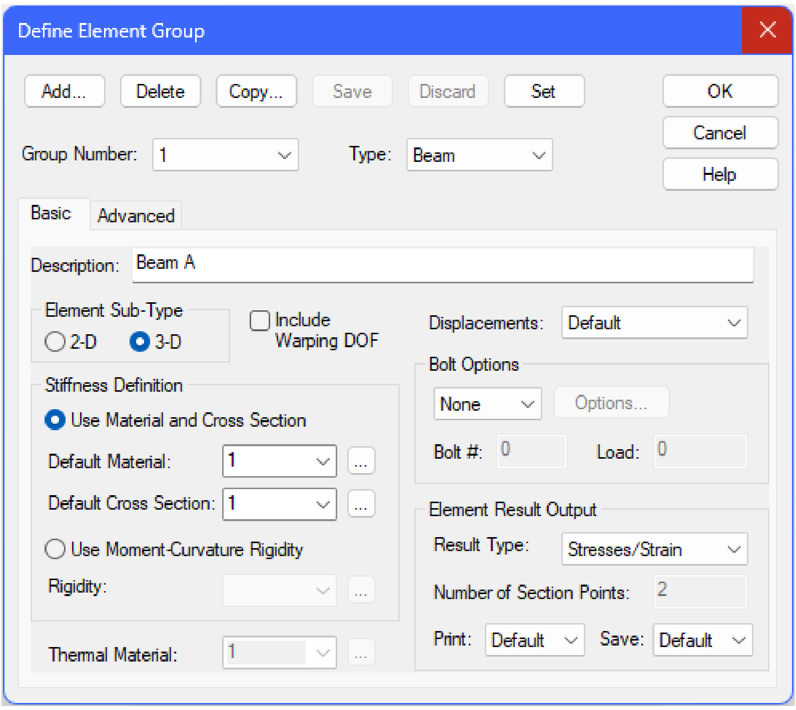
STEP 18. Specifying the type of analyzed construction

The present example requires the creation of 3 various element groups. It is connected with three bar segments, each with various cross-sectional area. The first and third segment of the bar (cross-sections A-A and C-C) are single-clamped (blocked 6 degrees of freedom). For the purpose of learning, segment B has a specified life cycle of 0.50 s. According to the above, segment B of the analyzed model will take part in the redistribution of forces in the entire beam from the beginning of the analysis, up to its 50th step (a time of 0.50 s); subsequently, the middle segment will become demolished. In the following time steps, segment B becomes isolated from the model, meaning that the calculations are performed only with respect to segments A and C.

In order to create an element group, go to “Meshing → Element Groups…”, or click . Clicking the button will open a new window, in which the “Add…” button should be clicked to add a new element group. The first created element group will be related to segment A, therefore input following data:

Group Number:	1
Type:	Beam
“Basic” tab	
Description:	Beam A
Element Sub-Type	3-D
Include Warping DOF	Unchecked
Displacements:	Default
Stiffness Definition	
Use Material and Cross Section	Checked
Default Material:	1
Default Cross Section:	1

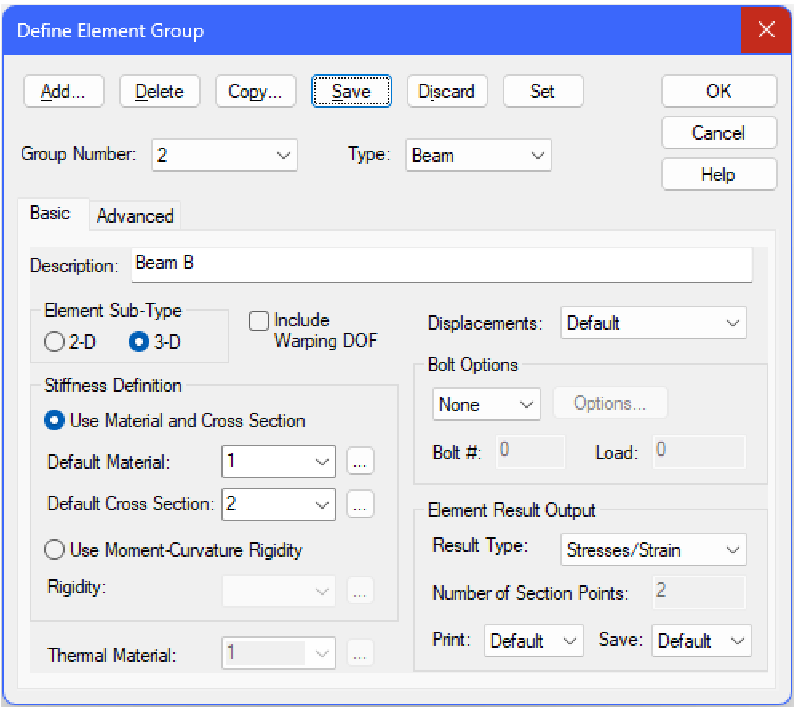
Since the numerical values symbolizing the cross-section and the type of material (if the user has not remembered/written down the id numbers for proper materials and cross-sections) are quite confusing in the form of numbers, in both of these cases use the “… “ button to check the id numbers of the requested cross-section and type of material. In the present example, choose the "… " button next to the "Default Material" option, then upon confirmation that steel is a material with the number id = 1, the material manager window can be closed with the "Close" button. Make sure that a value of "1" is displayed as "Default Material" in the "Define Element Groups" window. Similar actions are taken in order to specify a cross-section – "Default Cross-Section". In here, it is also worth it to use the "… " button, and then in the new window find the cross-section requested by the user by selecting the individual id numbers from the drop-down list in the "Section Number" box. The present element group needs a cross-section in the form of a pipe; therefore, its assigned value of "1" in the "Define Cross-Section" window. Subsequently, close the "Define Cross-Section" window by means of the "OK" or "Cancel" button, making sure that a value of "1" is displayed in the "Define Element Groups" window next to the "Default Cross-Section" option. The remaining data in the window remain unchanged. Window view with introduced changes is presented below:

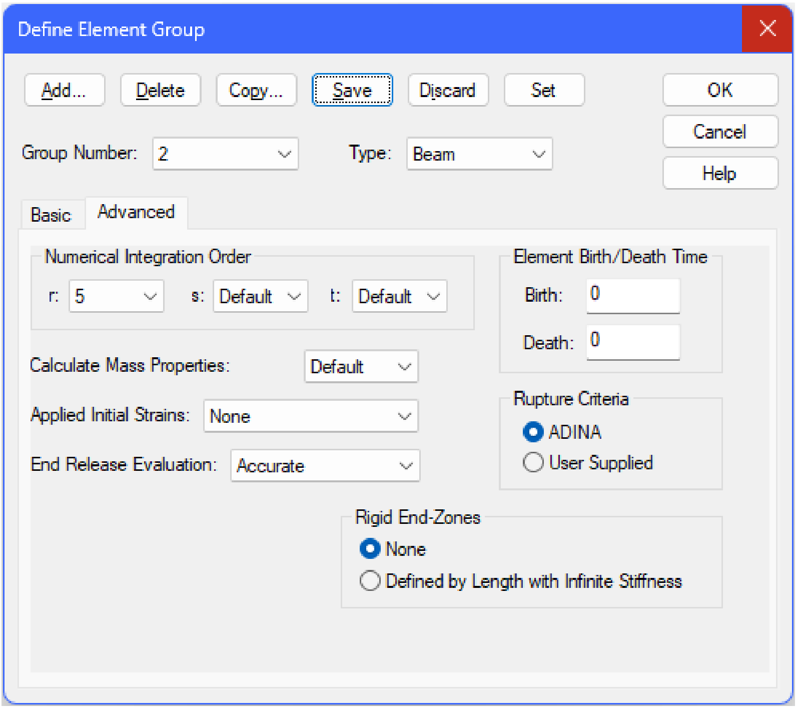


As soon as changes are introduced in the "Define Element Group" window, press "Save" and once again "Add…" button. The second group will be segment B, which is made of steel, and has a cross-section in the form of a circular solid bar, also it is assumed that this segment has specified life cycle. According to that enter following data for the segment B in the "Define Element Group" window:

Group Number:	2
Type:	Beam
“Basic” tab	
Description:	Beam B
Element Sub-Type	3-D
Include Warping DOF	Unchecked
Displacements:	Default
Stiffness Definition	
Use Material and Cross Section	Checked
Default Material:	1
Default Cross Section:	2
Advanced tab	
Element Birth/Death Time	
Birth:	0
Death:	0.5

The remaining options in the window remain unchanged. Finally, save the element group with an id = 2 with the “Save” button. The window with introduced data is presented below:

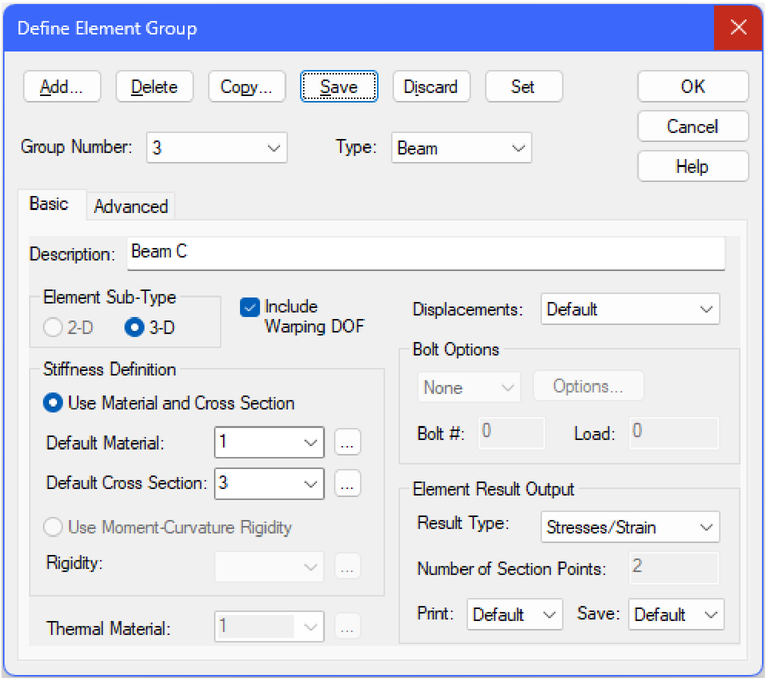




At last add the final element group by means of the "Add…" button. The third group covers a bar with a box-like cross-section, made of aluminum. Return to the "Basic" tab in the "Define Element Group" window. Similarly to the previous two groups enter following data:

Group Number:	3
Type:	Beam
"Basic" tab	
Description:	Beam C
Element Sub-Type	3-D
Include Warping DOF	Checked
Displacements:	Default
Stiffness Definition	
Use Material and Cross Section	Checked
Default Material:	2
Default Cross Section:	3

Once the operation is completed, the times of the appearance/disappearance of the element in the "Advanced" tab will be automatically set as 0 – an element will take part in the analysis over its entire duration. When all the operations are completed, click the “Save” button in order to save the edited values, and the “OK” button. Window with entered data is presented below:



STEP 19. Mesh subdivision

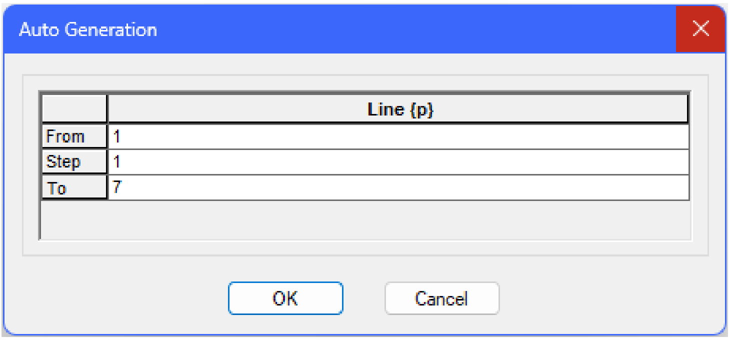
In the present example, the same length of a finite element segment equal to 1.00 cm will be used for each beam. In order to specify the length value of a finite element, go to “Meshing → Mesh Density → Line…”. In the newly opened window, choose:

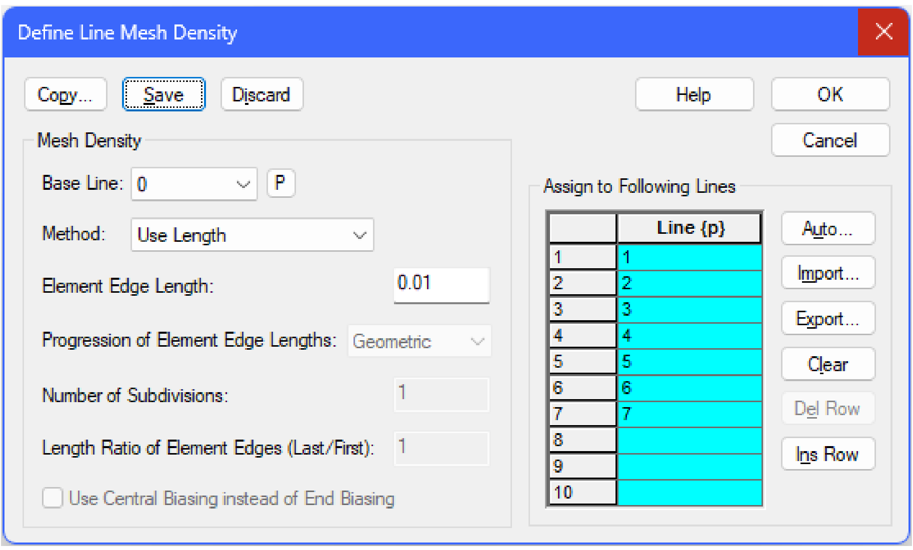
Mesh Density	
Base Line:	0
Method:	Use Length
Element Edge Length:	0.01
Assign to Following Lines	
	Line {p}
1	1
2	2
3	3
4	4
5	5
6	6
7	7

For filling the table it is possible to use the “Auto…” button. As soon as this button is clicked, a new window appears in which following data should be introduced:

	Line {p}
From	1
Step	1
To	7

then confirm the data with the “OK” button. Properly input data are presented below:

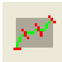




When the abovementioned operations are completed, save the data with the “Save” button, and then leave the window with the “OK” button.

It should be noted that faster method to create a mesh subdivision with identical size for each element is to go to “Meshing → Mesh Density → Complete Model…”. That method was used across previous examples.

STEP 20. Definition of finite elements

In order to define finite elements in the model, go to "Meshing → Create Mesh → Line…", or click  button, then enter the following options in the window:

Type:	Beam
Element Group:	1
Nodes per Element:	2
Orientation (Point overrides Vector)	
X:	0
Y:	0
Z:	0
Auxiliary Point:	9
Vector System:	Skew
Table "Lines to be Meshed"	
	Line {p}
1	1
2	2
3	3

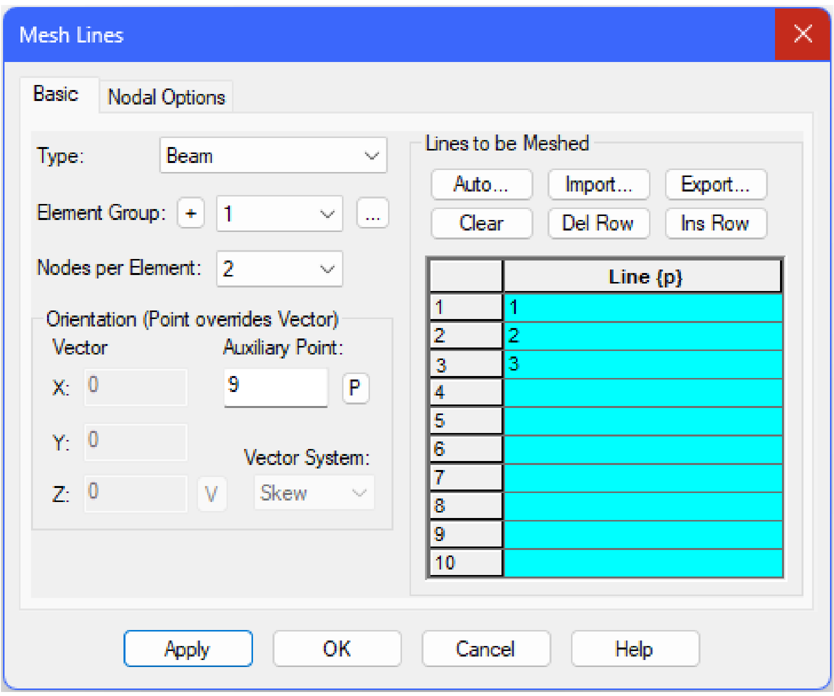
Remaining options are left unchanged. Press the "Apply" button, do not leave the window yet, and input following data once again:

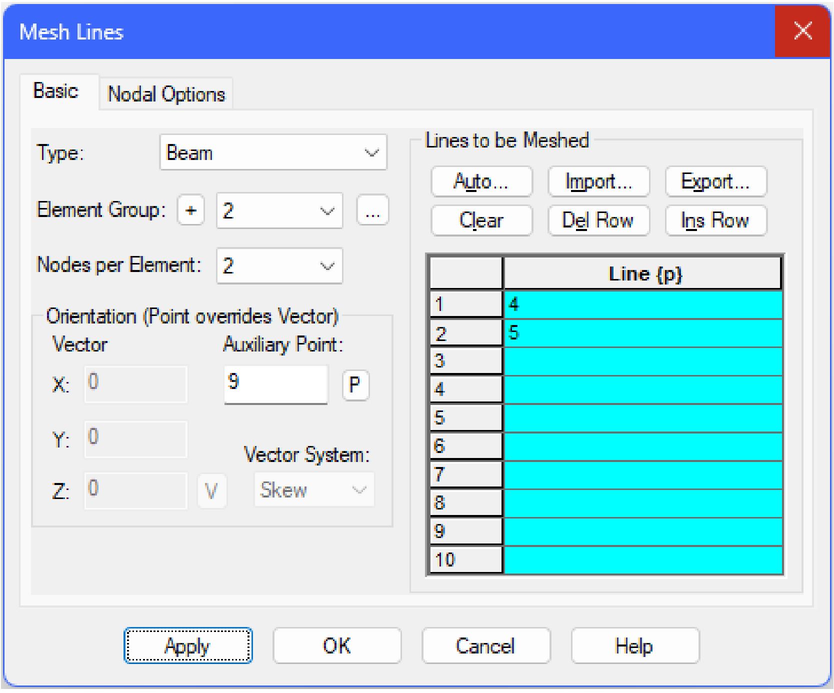
Type:	Beam
Element Group:	2
Nodes per Element:	2
Orientation (Point overrides Vector)	
X:	0
Y:	0
Z:	0
Auxiliary Point:	9
Vector System:	Skew
Table "Lines to be Meshed"	
	Line {p}
1	4
2	5

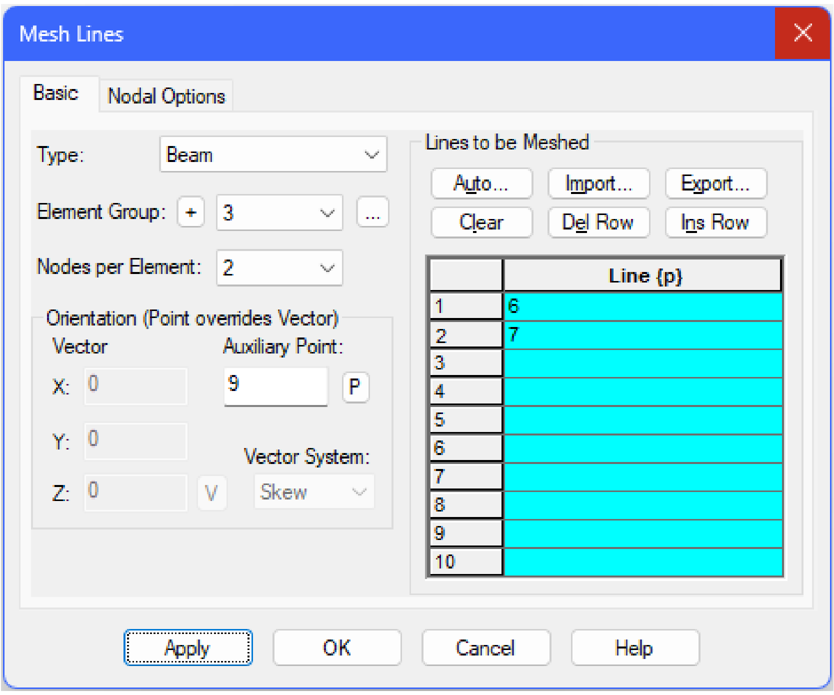
Remaining options are left unchanged. Once again press the "Apply" button, do not leave the window, and enter following data:

Type:	Beam
Element Group:	3
Nodes per Element:	2
Orientation (Point overrides Vector)	
X:	0
Y:	0
Z:	0
Auxiliary Point:	9
Vector System:	Skew
Table "Lines to be Meshed"	
	Line {p}
1	6
2	7

Remaining options are left unchanged. Hit the "Apply" button and leave the window by pressing "OK". A properly filled out windows are presented in the figures below:




Mesh Lines
Basic
Nodal Options
Type:
Beam
Element Group:
2
Nodes per Element:
2
Orientation (Point overrides Vector)
Vector
Auxiliary Point:
X: 0
9
P
Y: 0
Vector System:
Z: 0
V
Skew
Lines to be Meshed
Auto...
Import...
Export...
Clear
Del Row
Ins Row
Line {p}
1 4
2 5
3
4
5
6
7
8
9
10
Apply
OK
Cancel
Help


Mesh Lines
Basic
Nodal Options
Type:
Beam
Element Group:
3
Nodes per Element:
2
Orientation (Point overrides Vector)
Vector
Auxiliary Point:
X: 0
9
P
Y: 0
Vector System:
Z: 0
V
Skew
Lines to be Meshed
Auto...
Import...
Export...
Clear
Del Row
Ins Row
Line {p}
1 6
2 7
3
4
5
6
7
8
9
10
Apply
OK
Cancel
Help

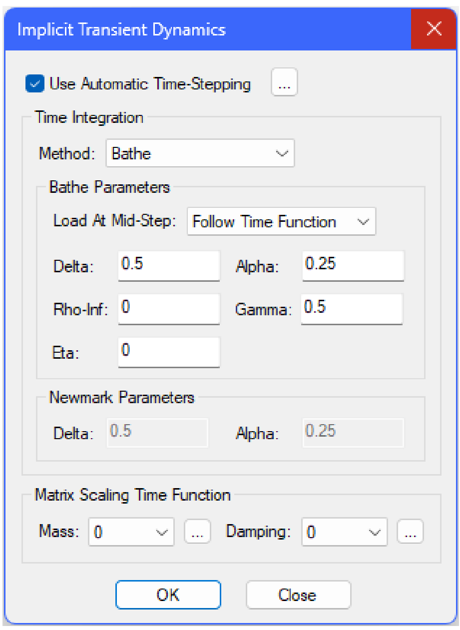
Note: Since the cross-sections used in the model are bisymmetric, the declared point 9 specifying the orientation of the cross-section can have any position on the YZ, XY or XZ plane. In the remaining cases, in which the cross-section is not bisymmetric, that point specifies the line in which the width of the declared profile extends.

Note: The window for defining finite elements can be entered before modeling the element groups, since the element groups can be created directly by means of the "Mesh Lines" or "Mesh Surfaces" window, etc. New element groups can be defined by means of the " + " button placed to the left of the text box belonging to the "Element Group" option.

Note: It is possible to check which element group has its assigned proper cross-section and material by clicking the "…" button in the "Mesh Lines" or "Mesh Surfaces" window, etc., placed to the right of the text box belonging to the "Element Group" option. Faster and easier identification is performed when a given group of elements has an entry specified by the user in the "Description" box (typically, this box can be left blank with a default entry of "NONE"; however, such an entry does not make work easier when there is a large number of elements in the model).

STEP 21. Definition of an automatic time stepping analysis

In case the convergence is not met, the automatic time stepping analysis greatly help to achieve that convergence. In order to define an automatic time step in the absence of convergence, click the button related to analysis options: , and select the "Use Automatic Time-Stepping" box in the newly opened window. The window is presented in the figure below:



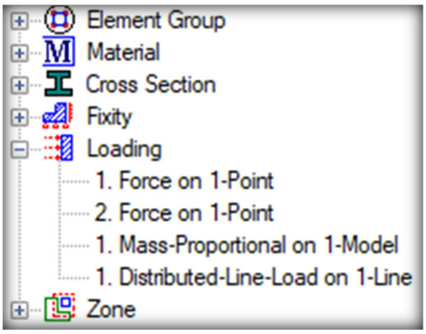
The remaining values remain unchanged.

An automatic time stepping in the absence of convergence allows for obtaining a solution between user defined time steps via time division, e.g., for a step size of 0.01 s and the existing absence of convergence in a time of 0.33 s, the program will start dividing the time interval between 0.33 s and 0.34 s in half; if this produces no results, the interval between 0.33 s and 0.335 s will be divided, etc. In order to better understand how this function works, it is recommended to click the "…" button, and then the "Help" button in the newly opened window.

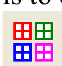
Once the automatic time step has been activated, the window can be closed with the "OK" button.

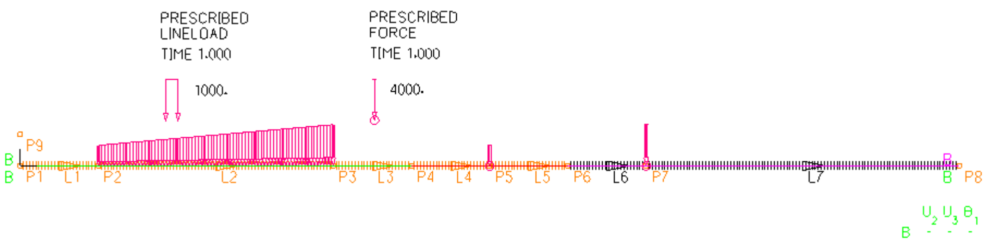
STEP 22. Checking the correctness of the adopted model

The first things to be checked are the loads, in particular by checking whether the mass-proportional load has been declared, since it is missing from the printout of loads in older ADINA software versions. In order to check the loads quantitatively, unfold "Loading" in the "Model Tree" window with the " + " button.



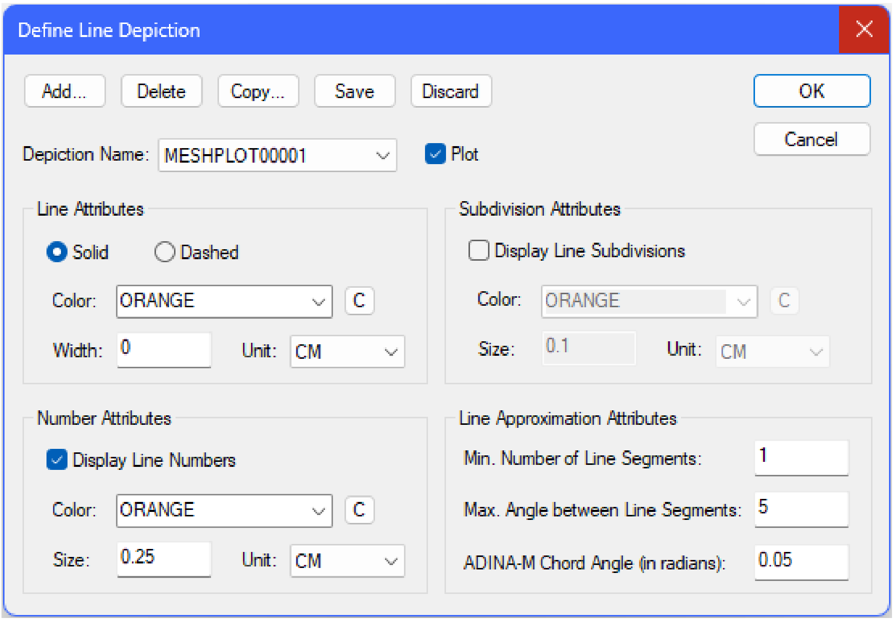
As can be seen in the figure above, all the loads have been introduced.

The next step is to check the declaration of element groups in the model. In order to do this, click . Unfortunately, the existing example is obstructed by segments showing the division of the element, like in the figure below:

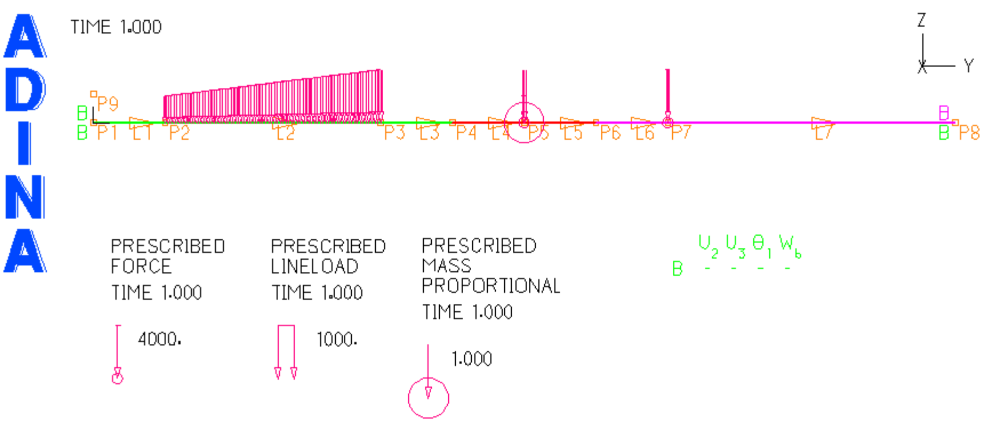


In order to deactivate presentation of the division of the segment, go to "Display → Geometry / Mesh Plot → Modify…" or click  button, make sure that the "Mesh Plot Name:" option refers to "MESHPLOT00001", and then click the "Line Depiction" button in the newly opened window.

Once a new window has opened in the "Subdivision Attributes" group of options, unselect the "Display Line Subdivision" box, and then click the "Save" button and "OK". The window with unchecked option is presented below:



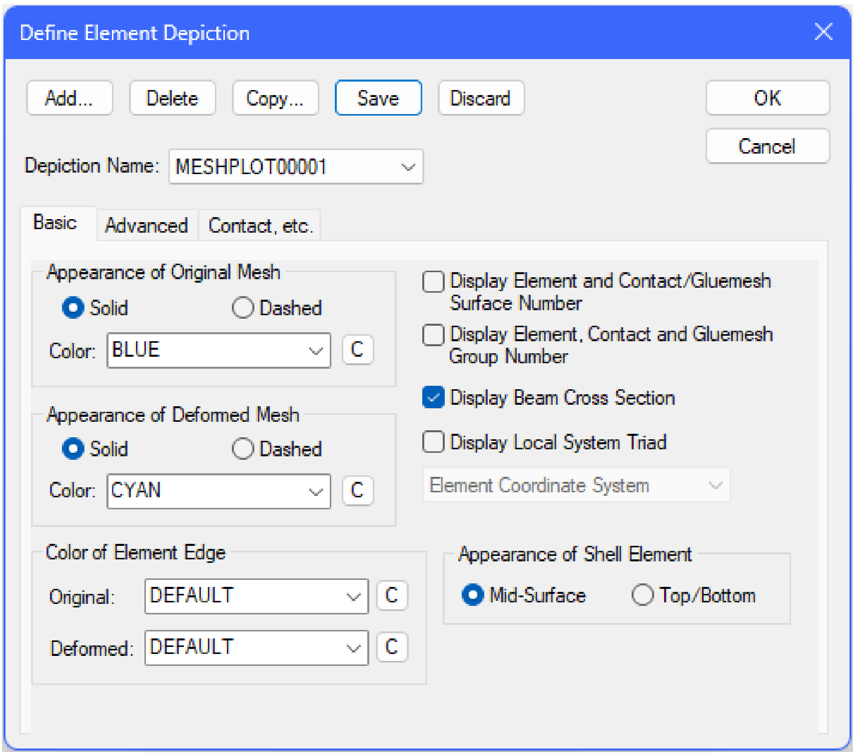
Upon returning to the "Modify Mesh Plot" window, also click the "Apply" button in it, and then "OK". The model should look as follows:

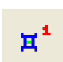
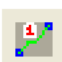


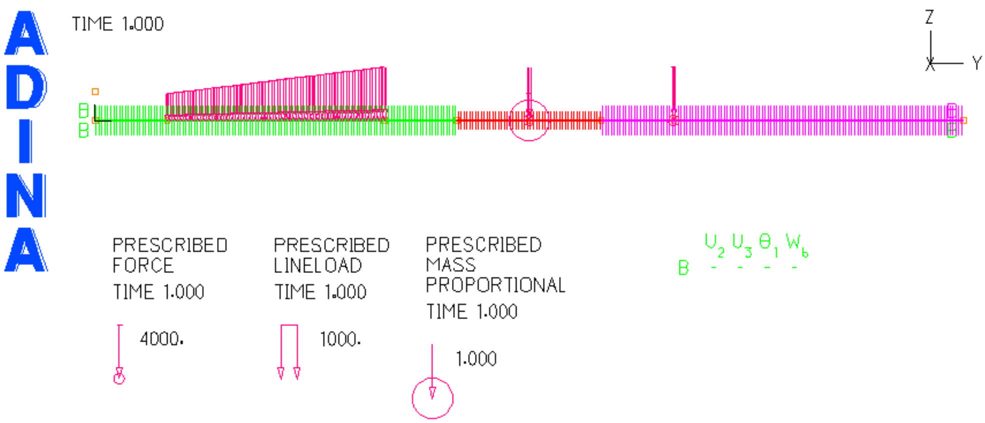
In the model made by the author, segment A is indicated with the green color, segment B with orange, and segment C with pink, respectively.

<u>Note</u>: The colors of beam segments listed above may differ between the author's model and the user's model.

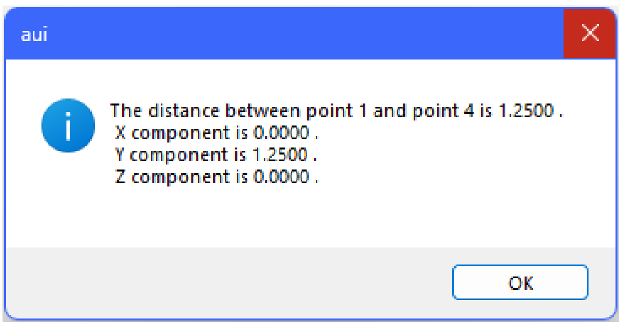
If it is still insufficient, return to the "Modify Mesh Plot" window, and by going to "Display → Geometry / Mesh Plot → Modify…", click the "Element Depiction" button. In the newly opened window, select the box belonging to the "Display Beam Cross Section" options. Then save the changes with the "Save" button, and leave the "Define Element Depiction" window with the "OK" button. A view of the window is presented below.



Upon returning to the "Modify Mesh Plot" window, click the "Apply" button and "OK". Once the window has been closed, in order to increase legibility of the figure, it is also recommended to unselect the view of line and point numbers by clicking  and , in this order. At this time, the model should look like this:



All that remains is to check the length of the declared beam segments with specific cross-sections. In order to check segment length for cross-section A-A, click , and then click the first node, in which the bar begins, and then the second node, in which the bar ends. Once the operation is completed, a window with information about the distance between the two nodes along proper coordinate axes will appear.



Of course, it is also possible to check other information about the model; however, for the needs of the present example, this is enough.

STEP 23. Starting calculations

In order to start calculations, choose "Solution → Data File / Run" from the upper menu tabs, or choose  from the toolbars. Subsequently, input a path for saving the resultant file. However, before the user clicks the "Save" button, they should make sure that the "Run Solution" and "Automatic Memory Allocation" options are selected in the window in the "Adina Structures Solution" group of options.

After starting the calculations and recalculating the model, close all three dialogue windows which appeared during the calculations.

STEP 24. Saving existing model to a file

Each model should have been saved to a file between few steps taken in order to not lose the data. According to that select "File → Save as..." from the menu. When a new window opens, indicate the location of the saved file and its name.

<u>Note</u>: Do not uses spaces in the file names, because it leads to an error! The space can be replaced with the underline character __.

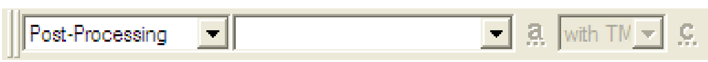
STEP 25. Starting calculations

In order to start calculations, choose "Solution → Data File/Run" from the upper menu tabs, or choose  from the toolbars. Subsequently, input the path for saving the resultant file. However, before the user clicks the "Save" button, they should make sure that the "Run Solution" and "Automatic" function near the "Maximum Memory for Solution" options are selected in the window in the "Adina Structures Solution" group of options. After starting the calculations and recalculating the model, close all three dialogue windows which appeared during the calculations.

<u>Note</u>: Depending on the complexity of the model and the number and type of finite elements used, model calculations may take from a few seconds to even several hours.

STEP 26. Post-processing module (results)

In order to go to the results module, change "ADINA Structures" to "Post-Processing" in the drop-down list of the "Module" toolbar.



When the user is prompted that the changes in the drawing have not been saved, it is recommended to save the model by going to "File → Save" or using the button .

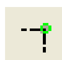
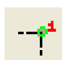
STEP 27. Opening the resultant file

In order to open the resultant file, choose  from the toolbar, or choose "File → Open" from the upper menu tabs.

<u>Note</u>: Depending on the specifications of the computer, the loading of a file in which a spatial analysis has been performed may take between about a dozen seconds up to even several minutes. The number of the applied finite elements and the number of nodes used have the greatest impact on the loading time of the file.

STEP 28. Creating a graph

A graph showing the displacement of the end of segment A bar (in the place where segment A is connected to segment B) and at the end of segment C bar (at the connection of segments B and C, respectively) will be created for the purpose of the present example. As can be seen after opening the file, the model has a gap between segments A and C. This is due to the fact that the analysis assumed a specific life cycle of segment B, and the model is displayed for the last time step adopted in the analysis. Therefore, when the life cycle has been exceeded, i.e., after the demolition of segment B, two independent systems will be created in the form of supports. By means of the arrow buttons  and the scalable displacement function activated with , it can be observed that segment B becomes demolished at a time point of 0.50 s, while segments A and C begin vibrating along with the time passing by. Since in such a case the maps created for the bars are quite confusing when presenting the results of displacements, it is possible to create a graph of the displacement of two different points.

In order to create a graph, start with declaring the points for which the displacements will be examined. In order to do that, go to "Definitions → Model Point → Node…". Click the "Add…" button in the new window The user will be asked to enter a name; the free end of segment A is the first one to be entered; therefore, the name has been defined as "SEG_A_FREE_END"; subsequently, click the "OK" button, upon which the program will go the node selection window. Having the "Define Model Point (Node Point)" window opened, scalable displacements should be deactivated by means of . Subsequently, activate the view of nodes with ; it is also possible to activate the view of node numbers by means of . The next step is the best possible magnification of the area of the free end of segment A. To this end, click , and then, while holding the left mouse button, select an area comprising the free end of segment A by means of a frame. The model should look like in the figure below.

Since the zoom-in button is still active, it should be turned into a selection cursor by activating . Now it is possible to return to the “Define Model Point (Model Point)” window. Following data should be entered:

Model Point Name:	SEG_A_FREE_END
Node #:	127
Substructure:	0
Reuse:	1

Subsequently, save changes in the window using the “Save” button, and add another node by clicking the “Add…” button. Once again, the user will be asked to enter a name; in this case, the name is “SEG_C_FREE_END”; afterwards, click the “OK” button. Once again, without closing the “Define Model Point (Node Point)” window, start with shrinking the resulting model to the default value by clicking . Subsequently, the model should be magnified; however, this time in the area of the free end of segment C. Click the zoom in button  again, and select a small area comprising the free end of segment C. The model should look like in the figure below.

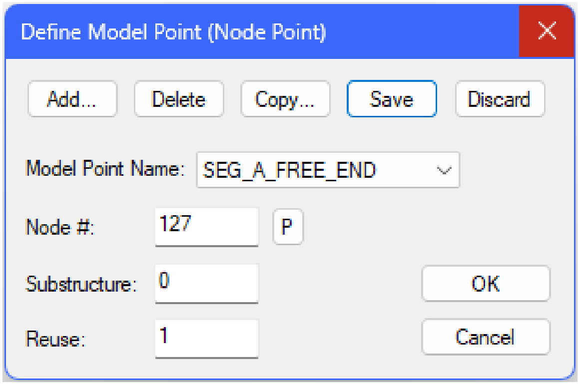
Now the node definition window can be activated again. Afterwards, enter following data:

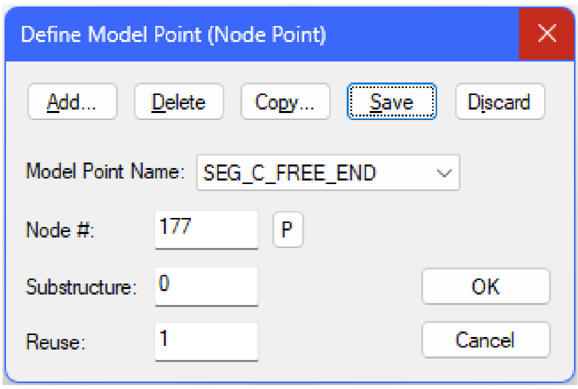
Model Point Name:	SEG_C_FREE_END
Node #:	177
Substructure:	0
Reuse:	1

or choose the desired node from the model by means of the “P” button.

The remaining options in the window remain unchanged. Finally, click the “Save” button in order to save changes, and then “OK” to close the window.

The node definition windows are presented below:

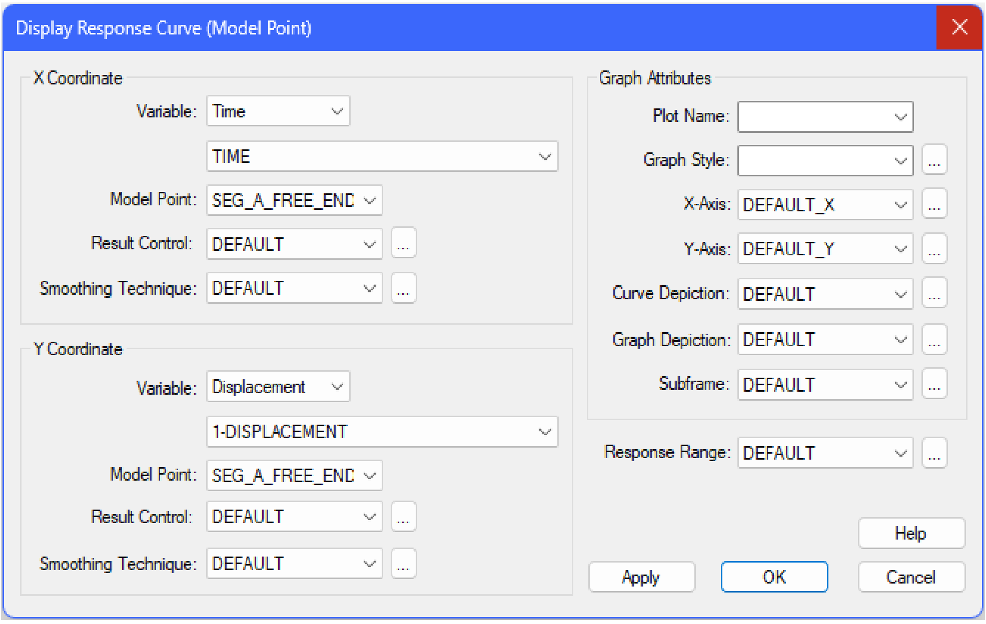




<u>Note</u>: It may happen that the node numbers in the user’s model are different from those in the presented example; therefore, it is recommended select proper nodes for the purpose of the present example.

With both nodes defined, it is possible to clear the resulting model window with 

. The selection cursor should also be chosen by clicking . After the above-mentioned operations, it is possible to start creating a graph. To this end, go to “Graph → Response Curve (Model Point)…”. Open a new window, which looks as follows.



Functions available in the left part of the window refer to the X and Y axes of the graph coordinate system.

– “Variable” – in this drop-down list it is possible to choose the time, displacement, stress, etc.; the second drop-down list helps to choose proper functions, i.e., if displacement is selected, the second drop-down list will allow for specifying the axis along which the displacement should be examined.
– “Model Point” refers to the node which has been declared by the user.
– “Result Control” enables changing the view of values on the graph, i.e., specifying the type of interpolation between points on the graph, etc.
– “Smoothing Technique” enables specifying the manner of smoothing the maps – the function is useful for “2-D Solid” – and “3-D Solid”-type models.

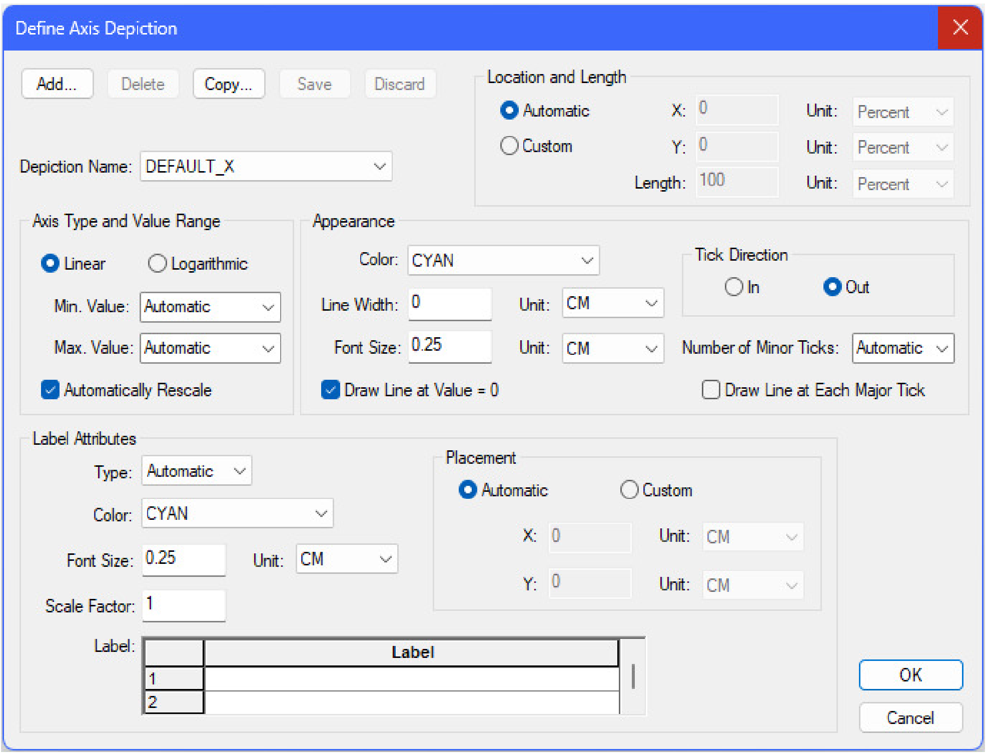
In the right part of the window there are options related to the graph itself – the line type of the X and Y axes, the curve type of the presented function, etc.

– “Plot Name” – enables specifying whether a new graph would be created, or whether additional curves would be added to an already existing graph.
– “Graph Style” – enables defining a single style including the options presented below, so that each option does not have to be defined separately. This function includes the following options: “X-Axis”, “Y-Axis”, “Curve Depiction”, “Graph Depiction” and “Subframe”.

- "X-Axis" and "Y-Axis" – options related to displaying the X and Y axes on the graph.
- "Curve Depiction" – options related to displaying a curve for specific variables on the graph.
- "Graph Depiction" – options for displaying the graph window.
- "Subframe" – options related to the graph frame.
- "Response Range" – options related to the time step adopted on the graph.

The analyzed example will use all the options related to the view of the graph.

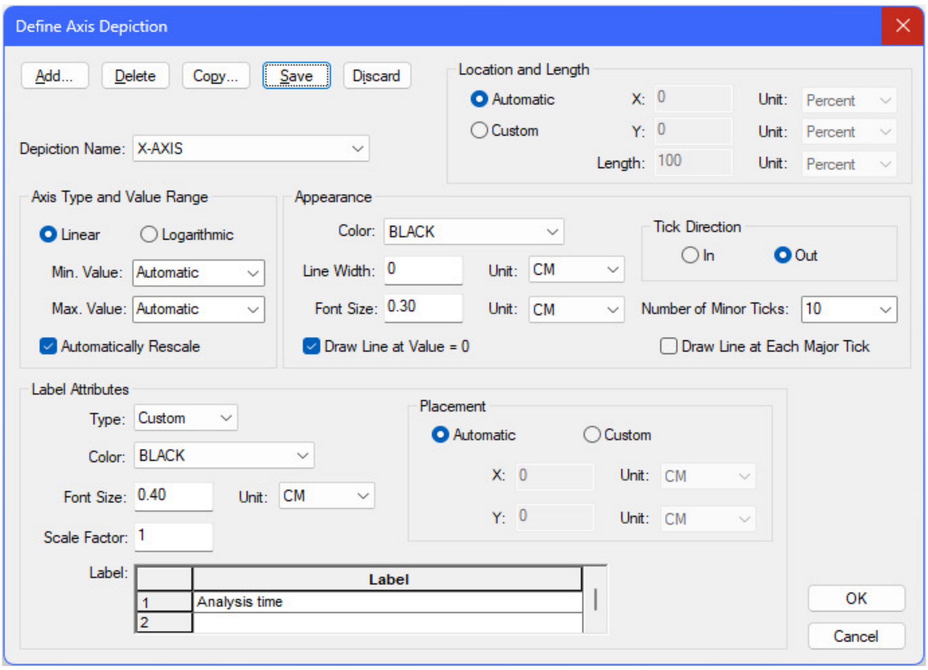
Start with declaring options related to displaying the graph and the curve directly; therefore, starting from the beginning, click the "…" button next to "X-Axis". A new window will appear, which is presented below:



In the present window, click the "Add…" button, enter a new name, e.g., "X_axis", and click "OK". Then enter following data:

...	
Label Attributes	
Type:	Custom
Color:	BLACK
Font Size:	0.40
Unit:	CM
Scale Factor:	1
Table:	
	Label:
1	Analysis time
Appearance	
Color:	BLACK
Line Width:	0
Unit:	CM
Font Size:	0.30
Unit:	CM
Draw Line at Value = 0	Checked
Tick Direction	
Out	Checked
Number of Minor Ticks:	10
Draw Line at Each Major Tick	Unchecked

Remaining options are left unchanged. The window with changed data is presented below:



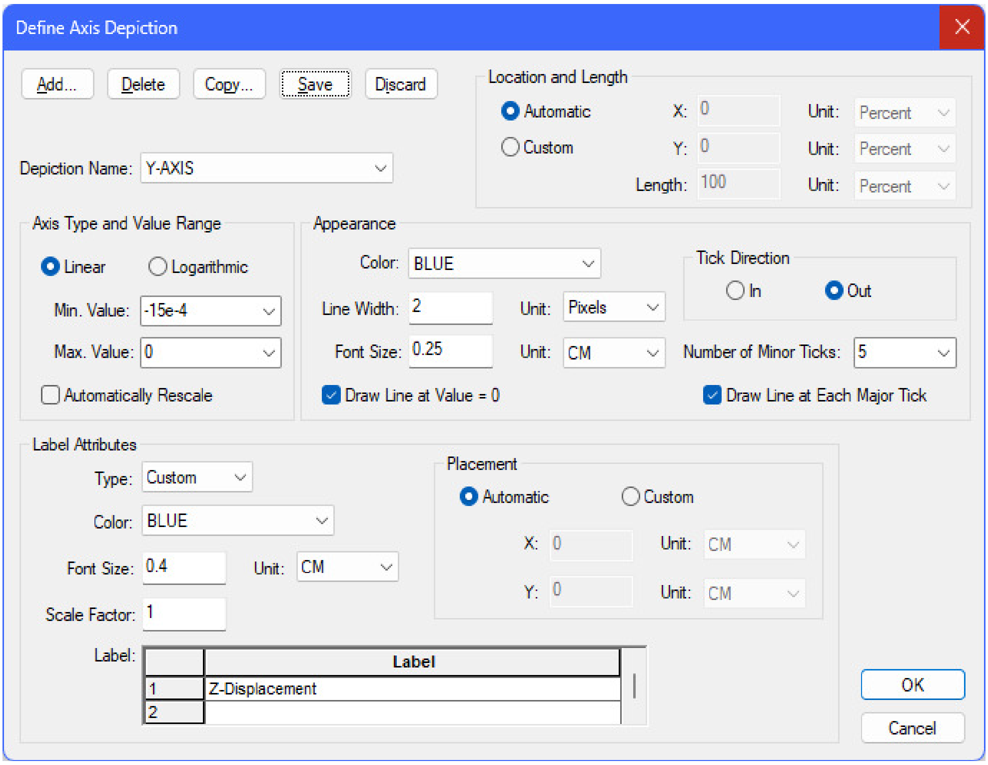
As soon as the settings for the X axis have been specified, click the "Save" button.

<u>Note</u>: Since the settings for the Y axis are exactly the same as for the X axis, their input can begin directly from the window in which the values of X were defined, or it is possible to close the window with the "OK" button, and then click the "…" button in the "Display Response Curve (Model Point)" window next to the "Y-Axis" option in the right part of the window.

The present example uses a faster method, which means inputting the settings for the Y axis directly from the window in which values for the X axis were defined. According to that, click the "Add…" button, enter a name, e.g., "Y_Axis", and click the "OK" button. Similarly, as before, enter following data:

…	
Axis Type and Value Range	
Linear	Checked
Min. Value:	–15e-4
Max. Value:	0
Automatically Rescale	Unchecked
Label Attributes	
Type:	Custom
Color:	BLUE
Font Size:	0.40
Unit:	CM
Scale Factor:	1
Table:	
	Label:
1	Z-Displacement
Appearance	
Color:	BLUE
Line Width:	2
Unit:	PIXELS
Draw Line at Value = 0	Checked
Tick Direction	
Out	Checked
Number of Minor Ticks:	5
Draw Line at Each Major Tick	Checked

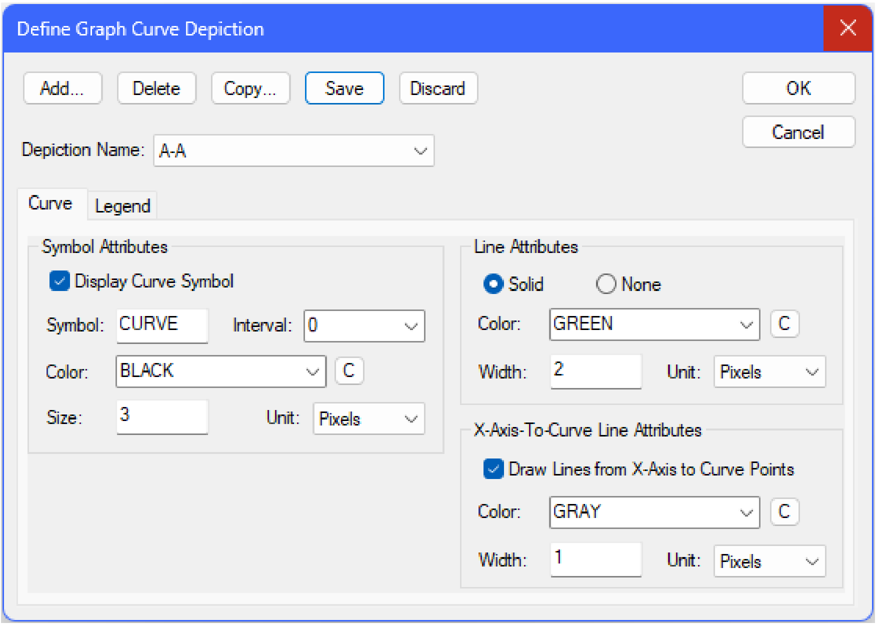
Finally, save the input data with the "Save" button, and leave the window with the "OK" button. The window with introduced data should look similar to that:

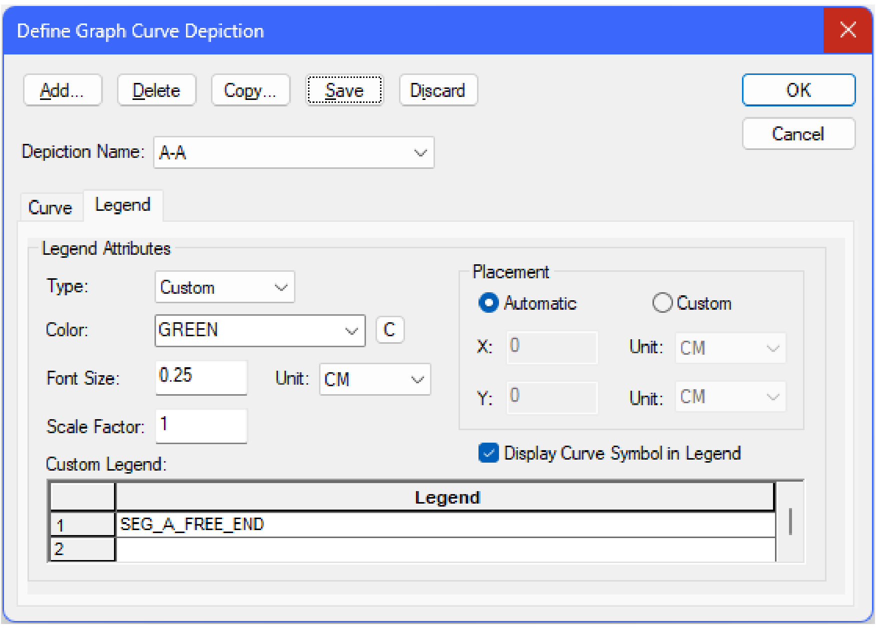


In the "Display Response Curve (Model Point)" window, choose the "…" button from its right part, belonging to the "Curve Depiction" option. A new window will open. For the purpose of the present example, two line types will be created, one specifying the free end of segment A, the other referring to the free end of segment C. Click the "Add…" button in the "Define Graph Curve Depiction" window, and then enter the name "A-A" and click "OK". Enter following data:

“Curve” tab	
Symbol Attributes	
Display Curve Symbol	Checked
Symbol:	CURVE
Interval:	0
Color:	BLACK
Size:	3
Unit:	Pixels
Line Attributes	
Solid	Checked
Color:	GREEN
Width:	2
Unit:	Pixels
X-Axis-To-Curve Line Attributes	
Draw Lines from X-Axis to Curve Points	Checked
Color:	GRAY
Width:	1
Unit:	Pixels
“Legend” tab	
Legend Attributes	
Type:	Custom
Color:	GREEN
Font Size:	0.25
Unit:	CM
Scale Factor:	1
Placement	
Automatic	Checked
Display Curve Symbol in Legend	Checked
Custom Legend:	
	Legend:
1	SEG_A_FREE_END

The “Save” button can be clicked upon inputting all the data. A view of the window with the input values is presented below:

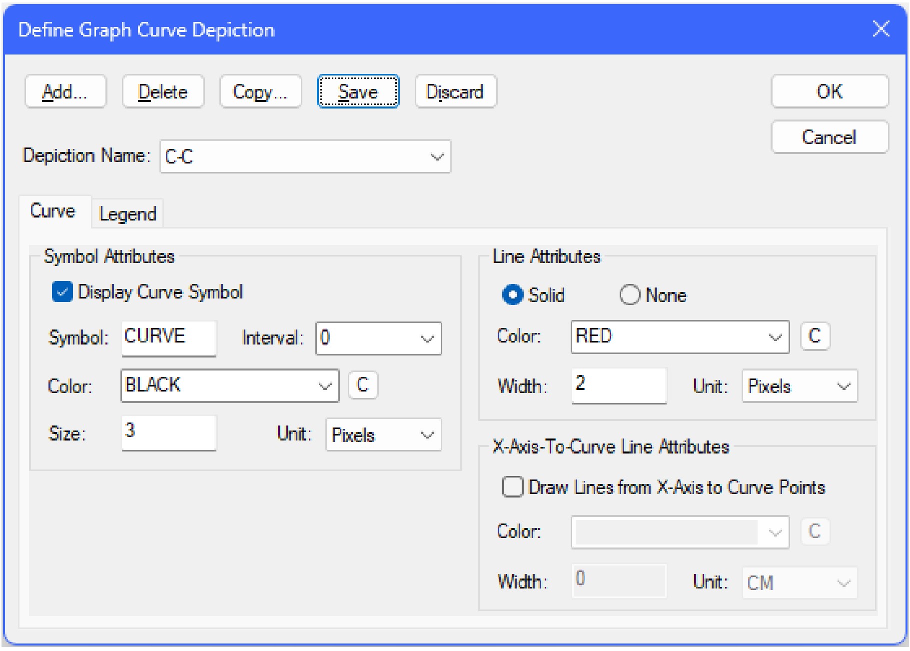


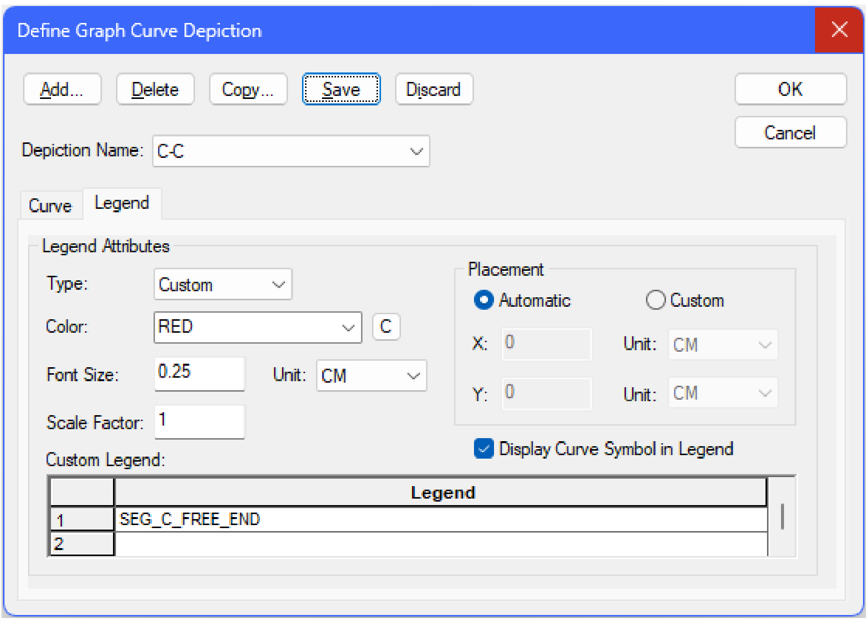


All that remains is to create parameters of the curve specifying the displacement along the Z axis of the free end of segment C; therefore, click the “Add…” button in the same window (“Define Graph Curve Depiction”), enter the name “C-C”, and click the “OK” button. Then, choose following options and data:

“Curve” tab	
Symbol Attributes	
Display Curve Symbol	Checked
Symbol:	CURVE
Interval:	0
Color:	BLACK
Size:	3
Unit:	Pixels
Line Attributes	
Solid	Checked
Color:	RED
Width:	2
Unit:	Pixels
X-Axis-To-Curve Line Attributes	
Draw Lines from X-Axis to Curve Points	Unchecked
Color:	GRAY
Width:	1
Unit:	CM
“Legend” tab	
Legend Attributes	
Type:	Custom
Color:	RED
Font Size:	0.25
Unit:	CM
Scale Factor:	1
Placement	
Automatic	Checked
Display Curve Symbol in Legend	Checked
Custom Legend:	
	Legend:
1	SEG_C_FREE_END

The remaining options remain unchanged, and the input changes should be confirmed by the “Save” button and “OK”, thus closing the “Define Graph Curve Depiction” window. Properly filled out windows are presented below:

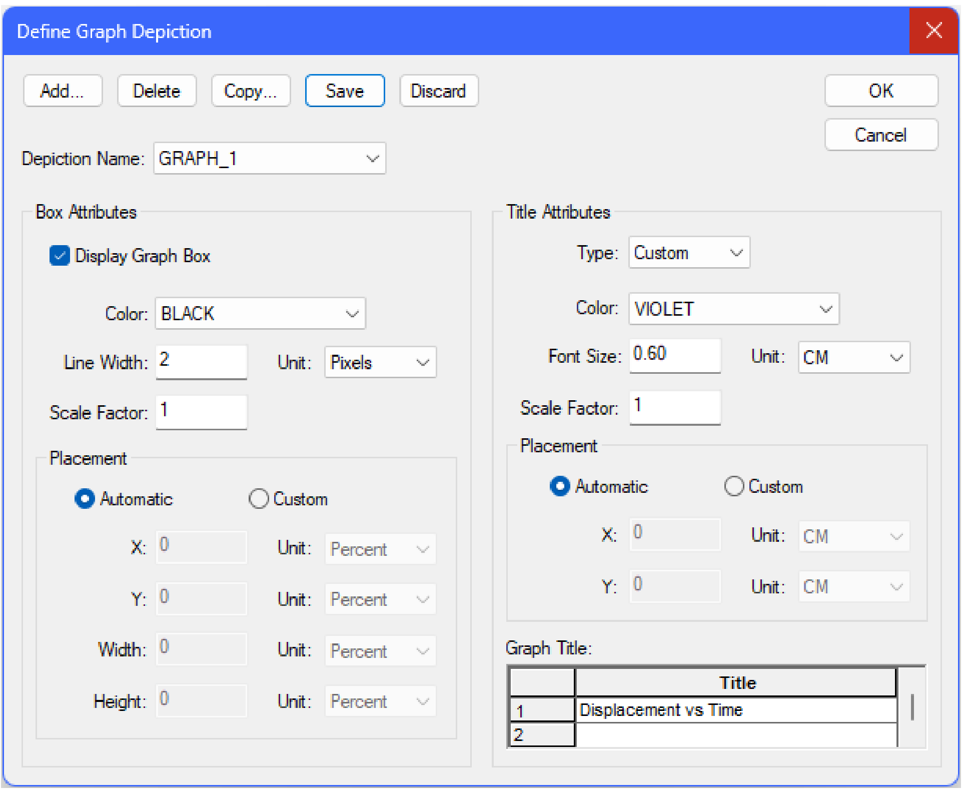




Upon returning to the "Display Response Curve (Model Point)" window, choose the "…" button in the right part of the window for the "Graph Depiction" option, then click the "Add…" button in the new window, enter a name, e.g., "Graph_1", and click "OK". Make following changes in the window:

Box Attributes	
Display Graph Box	Checked
Color:	Black
Line Width:	2
Unit:	Pixels
Scale Factor:	1
Title Attributes	
Type:	Custom
Color:	VIOLET
Font Size:	0.60
Unit:	CM
Scale Factor:	1
Placement	
Automatic	Checked
Graph Title:	
	Title
1	Displacement vs Time

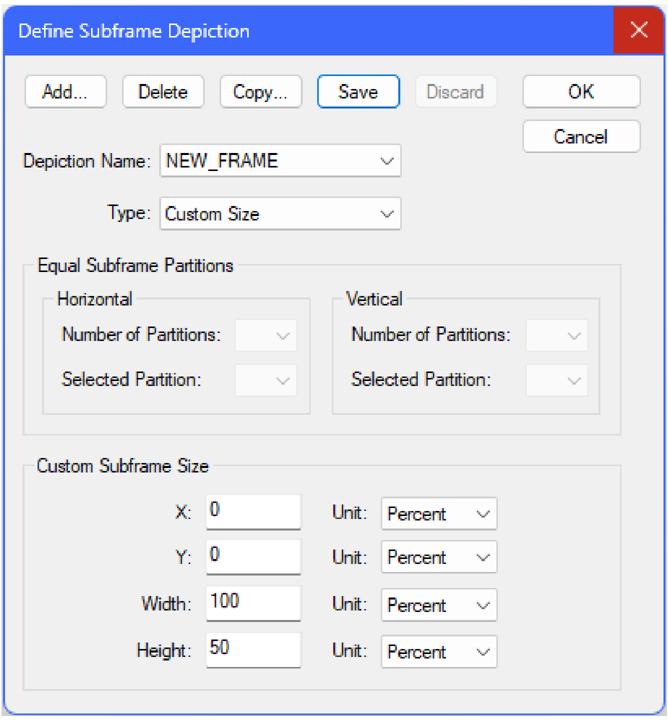
Upon inputting all the changes, click the “Save” and “OK” buttons. A window with the input data is presented below:



Upon returning to the "Display Response Curve (Model Point)" window, all that remains is the "Subframe" option related to the size of the graph; click the "…" button belonging to this option, and the "Add…" button in the new window, enter the name "New_frame", and confirm it with the "OK" button. In the window enter following data:

Depiction Name:	NEW_FRAME
Type:	CUSTOM_SIZE
Custom Subframe Size	
X:	0
Unit:	Percent
Y:	0
Unit:	Percent
Width:	100
Unit:	Percent
Height:	50
Unit:	Percent

Leave the remaining options unchanged. Confirm the input changes with the "Save" button, and leave the window with the "OK" button. A view of the window is presented below:



In the main "Display Response Curve" window, the "Response Range" option will not be changed, since the graph will present the displacement of points from the beginning to the very end of the analysis (100 time steps, 0.01 s each).

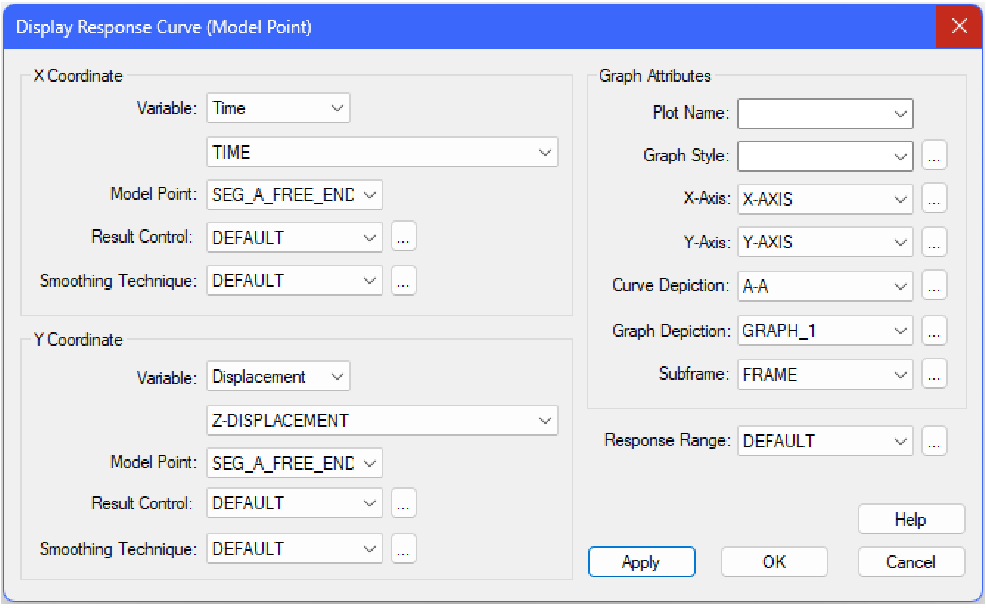
Note: In the present example, there is no need to create a "Graph Style", since the graph will ultimately have 2 curves, which causes certain complications when using "Graph Style". This option should be mainly used when it is certain that the graph consists of only one curve.

Note: The graph defined in this example is adjusted to being displayed against a white background. Against a black background, many of the applied functions may not be visible; therefore, if the user uses a black background, it is recommended to change them to white by means of "Edit → Background Color…", then choose the color "WHITE" from the drop-down list and confirm it with the "OK" button.

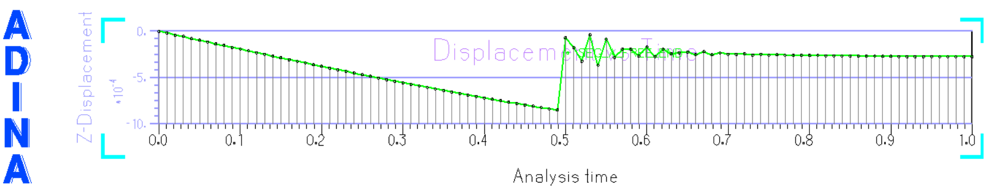
Once the abovementioned data have been input, a graph can already be created. In the "Display Response Curve (Model Point)" window make following changes:

X Coordinate	
Variable:	Time
	TIME
Model Point:	SEG_A_FREE_END
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Y Coordinate	
Variable:	Displacement
	Z_DISPLACEMENT
Model Point:	SEG_A_FREE_END
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Graph Attributes	
Plot Name:	Empty
Graph Style:	Empty
X-Axis:	X-AXIS
Y-Axis:	Y_AXIS
Curve Depiction:	A-A
Graph Depiction:	GRAPH_1
Subframe:	FRAME
Response Range:	DEFAULT

The values defined in the whole window in the form of a drawing are presented below:



Subsequently, click the “Apply” and “OK” buttons. The model window should look like in the figure below.

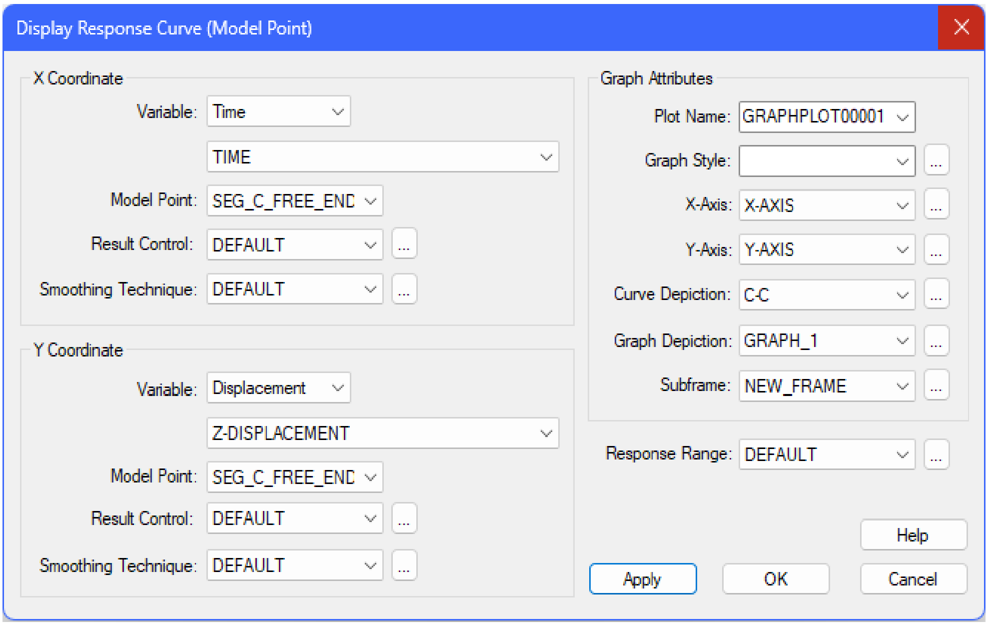


In the figure above, some of the texts overlap the graph. They can be moved by means of , by holding a given text with the left mouse button and moving it in the model window.

A line referring to the free end of segment C should still be added to the graph. Again, go to the graph definition window “Graph → Response Curve (Model Point)”. In the left part of the window related to the view of variables on the graph, act similarly like when defining a graph for a point from segment A. Make following changes:

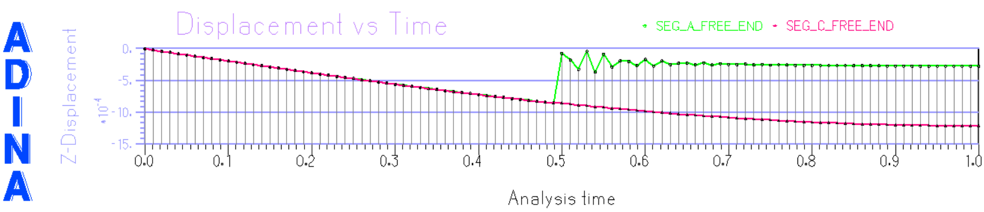
X Coordinate	
Variable:	Time
	TIME
Model Point:	SEG_C_FREE_END
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Y Coordinate	
Variable:	Displacement
	Z_DISPLACEMENT
Model Point:	SEG_C_FREE_END
Result Control:	DEFAULT
Smoothing Technique:	DEFAULT
Graph Attributes	
Plot Name:	GRAPHPLOT00001
Graph Style:	Empty
X-Axis:	X-AXIS
Y-Axis:	Y-AXIS
Curve Depiction:	C-C
Graph Depiction:	GRAPH_1
Subframe:	FRAME
Response Range:	DEFAULT

It is critical to change the blank box to "GRAPHPLOT00001" in the drop-down list of the "Plot Name:" box. If this option is not set, a separate graph will be created; the curve referring to the free end of the bar located in the left part of segment C will not be added to the previously created graph referring to the free end of the bar in the right part of segment A. A properly filled out window is presented below.



Upon inputting all the data, click the "OK" button.

The model should look as follows:

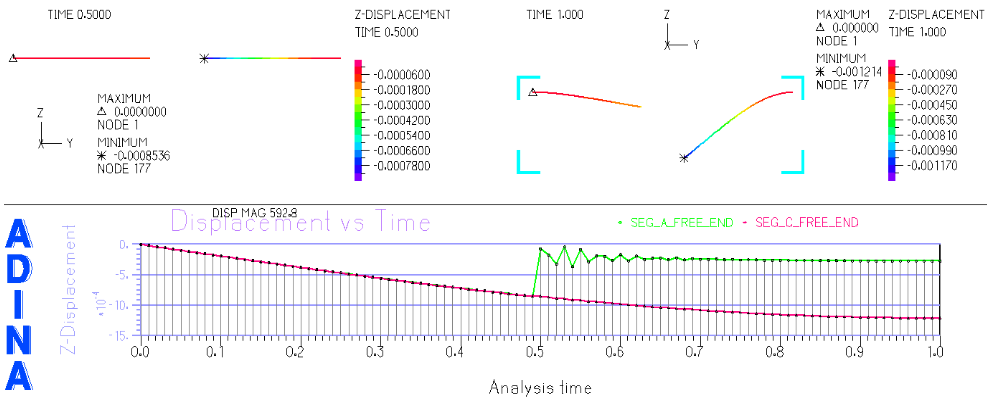


Note: The text overlapping the graph was moved by means of .

The figure above presents a graph created in a frame in the lower part of the window, with dimensions of 100% of its width and 50% of its height. These settings have been input in the "Define Subframe Depiction" window, upon selecting the "…" button belonging to the "Subframe" option from the "Display Response Curve (Model Point)" window. Such a manner of displaying the graph is a desirable procedure when, apart from the graph, the resultant maps are to be displayed for the model.

Note: It is also possible to zoom in and out of the graph by using , or by moving the mouse while holding the CTRL key.

A sample window with a graph and a map of displacements is shown in the figure below:



SUMMARY

Dear reader, if you have completed all the examples described in this book, then you already have sufficient knowledge on the ADINA software, enabling you to examine the technical subjects related to building structures by yourself. The ability to complete a given numerical model also makes it easier to operate on other programs based on the finite element method, like ABAQUS, ANSYS, CATIA, etc. In order to broaden theoretical knowledge about the finite element method, go to [1-10] from the list of references. Information regarding the ADINA software as well as its available functions and their application are described in [11-14]. Some design aspects are also discussed on the website of the software, in the Support/Tech Briefs tab [15]. Moreover, operation by means of commands entered in the command line of the main menu of the ADINA software is presented in publications [16-19]. Various types of publications constitute a valuable source of information regarding the shaping of numerical models as well as their calculation and application. Selected publications related to numerical calculations by means of the ADINA software can be found in scientific journals available in printed and electronic version i.e. [20-22]. A broader review of publications from various authors indirectly related to the ADINA software and its application are presented on the website of the software, in the Support/Publications tab [23]. It should be mentioned that the ADINA software has an additional guidebook [24] presenting the modeling process step by step. Apart from the present book, it is also worth it to get acquainted with the examples presented. This guidebook can be found directly in the ADINA program, by going to the "Help/ADINA Primer (pdf)…" tab.

It should be pointed out that the tools based on the finite element method and the acquired knowledge from a given field of science can be used to perform numerical calculations of a model, for which calculations using the traditional method would be too time-consuming, and thus uneconomical, as well there is a risk of making an error due to applied simplifications. Therefore, the programs provide the ability to produce correct results in a short time, assuming that all the input data have been introduced properly. According to the above, it can be noticed that programs based on the FEM are the future of the pursued projects and research, allowing for better understanding of the behavior of a given model or phenomenon, with no need to perform expensive laboratory tests.

REFERENCES

1. Bathe K.J., Finite Element Procedures. 1st edition, Klaus Jürgen Bathe, Cambridge, MA 2006.
2. Bathe K.J., Nonlinear Finite Element Analysis in ADINA, Proceeding of the 9^{th} ADINA Conference, Cambridge, MA, 1993, 511-891.
3. Huebner K.H., Dewhirst D.L., Smith D.E., Byrom T.G., The Finite Element Method for Engineers, J. Wiley & Sons Inc., New Delhi 2001.
4. Rakowski G., Kacprzyk Z., Metoda elementów skończonych w mechanice konstrukcji, Oficyna Wydawnicza Politechniki Warszawskiej, Warszawa 2005.
5. Rakowski G., Metoda elementów skończonych. Wybrane problemy, Oficyna Wydawnicza Politechniki Warszawskiej, Warszawa 1996.
6. Zienkiewicz O.C., Metoda elementów skończonych, Arkady, Warszawa 1972.
7. Zienkiewicz O.C., Taylor R. L., The Finite Element Method. Its Basis & Fundamentals, Elsevier 2005.
8. Zienkiewicz O.C., Taylor R.L., The Finite Element Method. Volume I: The Basis, Butterworth-Heinemann, Oxford 2000.
9. Zienkiewicz O.C., Taylor R.L., The Finite Element Method. Volume II: Solid Mechanics, Butterworth-Heinemann, Oxford 2000.
10. Zienkiewicz O.C., Taylor R.L., The Finite Element Method. Volume III: Fluid Dynamics, Butterworth-Heinemann, Oxford 2000.
11. ADINA R&D, ADINA. Theory and Modeling Guide. Volume I: ADINA Solids & Structures, ADINA R&D Inc., Watertown 2011.
12. ADINA R&D, ADINA. Theory and Modeling Guide. Volume II: ADINA Heat Transfer, ADINA R&D Inc., Watertown 2011.
13. ADINA R&D, ADINA. Theory and Modeling Guide. Volume III: ADINA CFD & FSI, ADINA R&D Inc., Watertown 2011.
14. ADINA R&D, ADINA. Theory and Modeling Guide. Volume IV-EM: ADINA EM, ADINA R&D Inc., Watertown 2011.
15. www.adina.com/newsgrp.shtml
16. ADINA R&D, ADINA User Interface. Command Reference Manual. Volume I: ADINA Solids & Structures Model Definition, ADINA R&D Inc., Watertown 2011.
17. ADINA R&D, ADINA User Interface. Command Reference Manual. Volume II: ADINA Heat Transfer Model Definition, ADINA R&D Inc., Watertown 2011.
18. ADINA R&D, ADINA User Interface. Command Reference Manual. Volume III: ADINA CFD & FSI Model Definition, ADINA R&D Inc., Watertown 2011.
19. ADINA R&D, ADINA User Interface. Command Reference Manual. Volume IV: ADINA EM Model Definition, ADINA R&D Inc., Watertown 2011.

20. Major M., Kuliński K., Major I., Dynamic analysis of an impact load applied to the composite wall structure, MATEC Web of Conferences 2017, 107, 1-6.
21. Major M., Kuliński K., Major I., Innovative concrete-polyurethane composite blocks located in three-layer wall – thermal numerical analysis, Engineering Mechanics, ed. V. Fuis, 2017, 602-605.
22. Major M., Major I., Kuliński K., Analysis of the Mechanical Wave in the Composite Made of Concrete and Rubber – Numerical Analysis, Transactions of the VSB – Technical 2019, 19, 2, 25-28. DOI: 10.35181/tces-2019-0015.
23. www.adina.com/pubs/publications1.shtml
24. ADINA R&D, ADINA Primer, ADINA R&D Inc., Watertown 2011.