ЦКТИ-ВИБРОСЕЙСМ

ИНЖИНИРИНГ ПРОЧНОСТЬ ВИБРОЗАЩИТА И СЕЙСМОСТОЙКОСТЬ

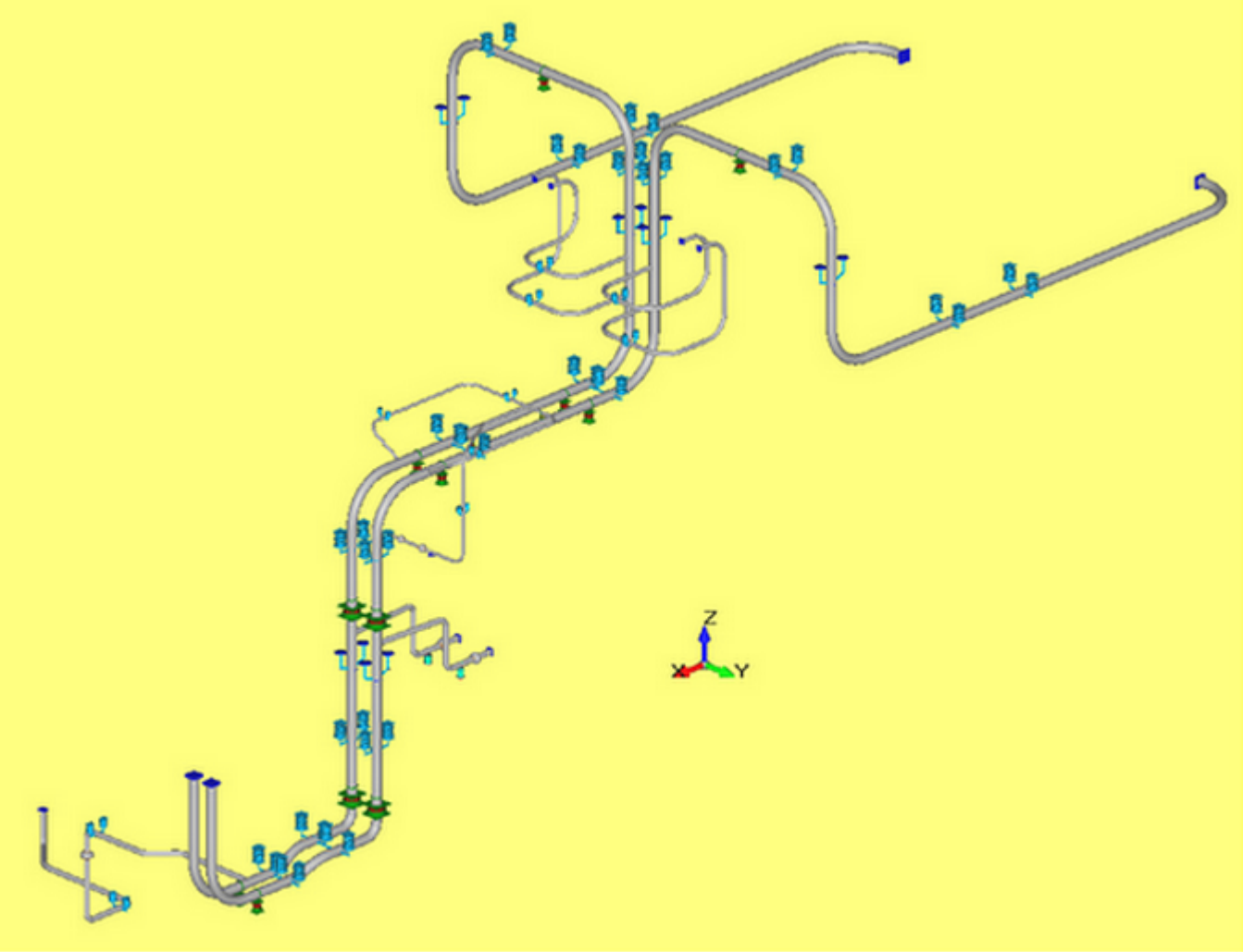
CKTI-VIBROSEISM

A STRUCTURAL-MECHANICAL CONSULTING ENGINEERING FIRM

Introduction to dPIPE 5.

Quick Start

Version 2.6.0



Saint Petersburg

Table of Contents

1 Introduction

The present document is intended for a quick introduction to dPIPE 5 and its basic operations. Below you can find a short description of its interface and hot keys. A test model exemplifies recommended work algorithms and methods of developing a calculation model, entering input data, establishing options for analysis and post-processing, performing analysis, viewing and assessing its results. The given model is restricted to only 30 nodes, so that it can be viewed within the demo version of the program.

1.1 Installing software

In order to install dPIPE 5 on your computer, launch installation program dp5_setup.exe and follow its instructions. You will be asked to choose the interface language for the installer, disk space to save it to, as well as the interface language for dPIPE.

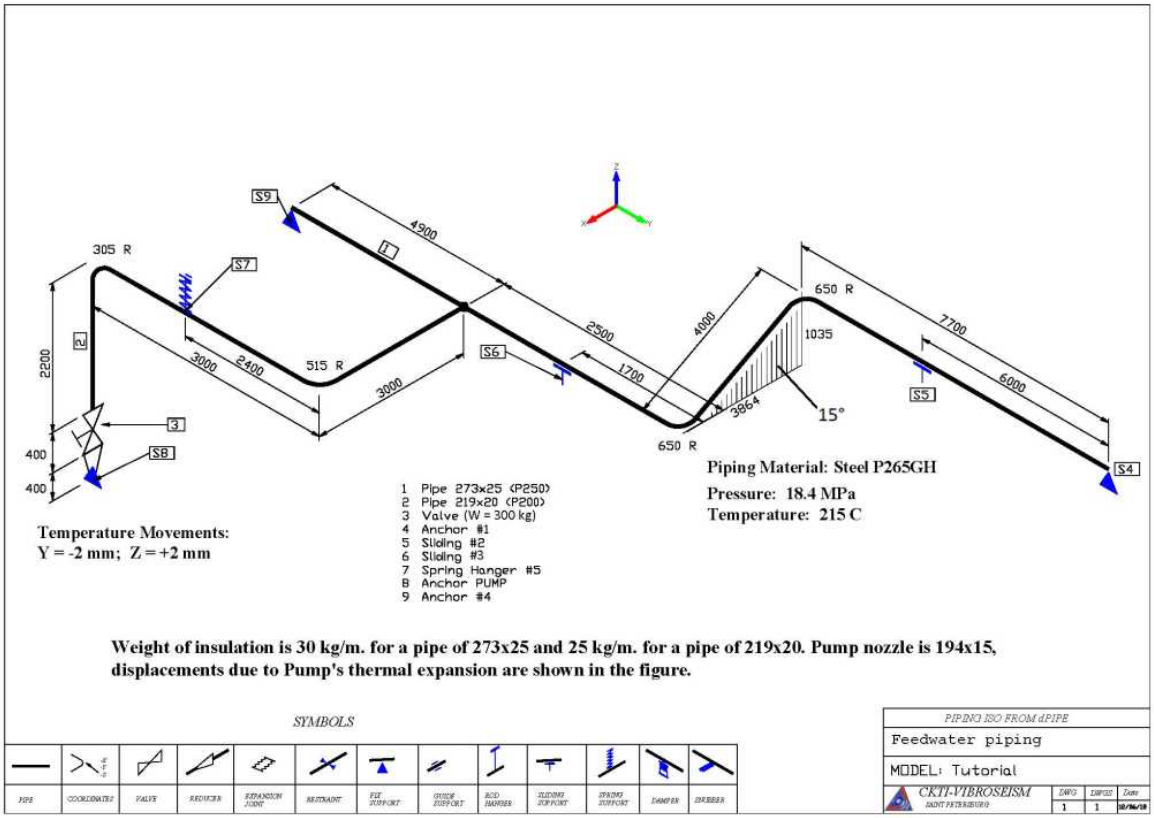
The installer can be either included in the installing package provided to the User upon purchasing dPIPE or downloaded from the website www.dpipe.ru (section "Files for download").

To access the full functionality of the program, one needs a security key (either hardware or software, either user or network).

Without security key, the program only operates in demo mode. In this case, its full functionality is limited to 30 nodes of calculation model.

1.2 Test model

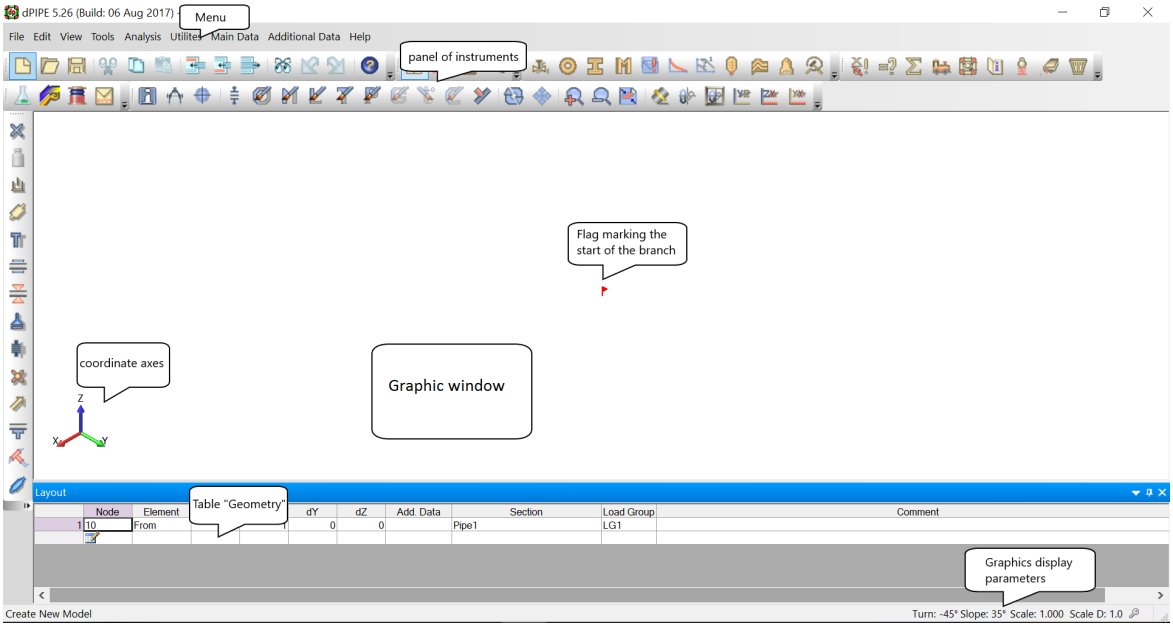
Let's take a look at the test model of Feedwater piping stretching from a pump into a header.



Open dPIPE and take a closer look at the main window of the program.

1.3 Main window

Menu and the toolbar are located in the upper part of the main window, while the graphics window and spreadsheet for entering piping geometry are located in its lower part. The spreadsheet could be switched on/off from the menu "Main Data/Layout"



In order to start, choose the Piping Code.

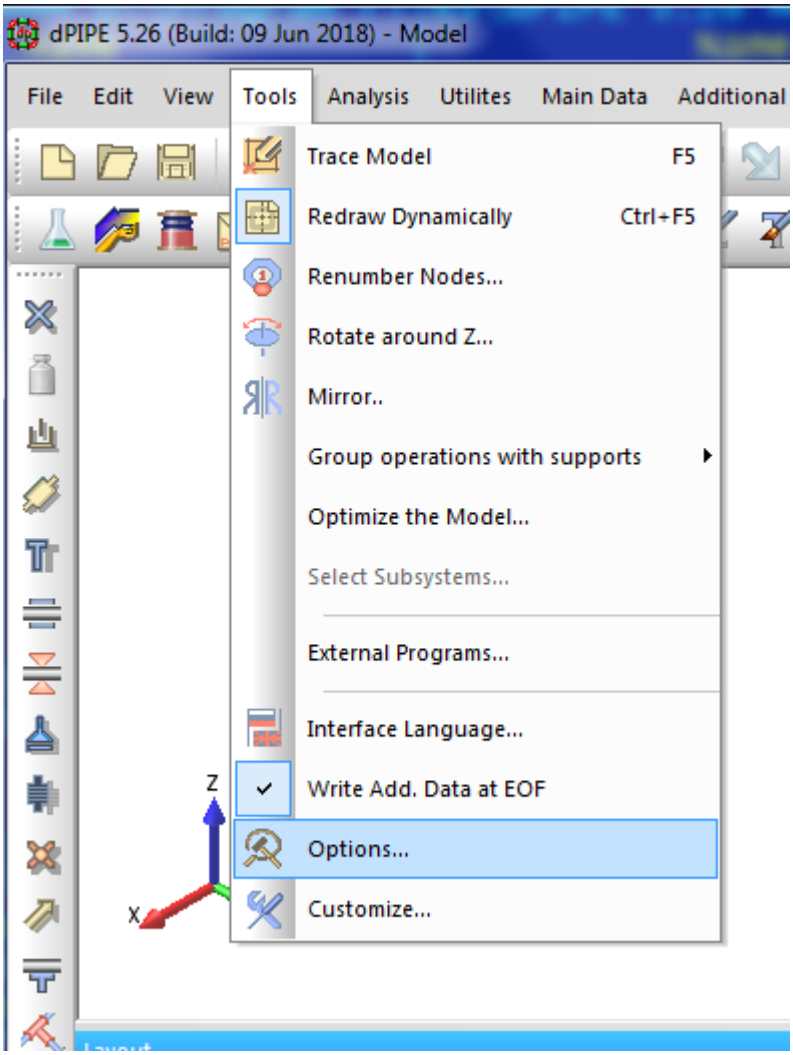
2 Quick start

This chapter provides information on how to create a simple model of the piping and run an analysis. The User is offered to build one branch of the test piping model between two anchors (from s4 to s9) following this guide

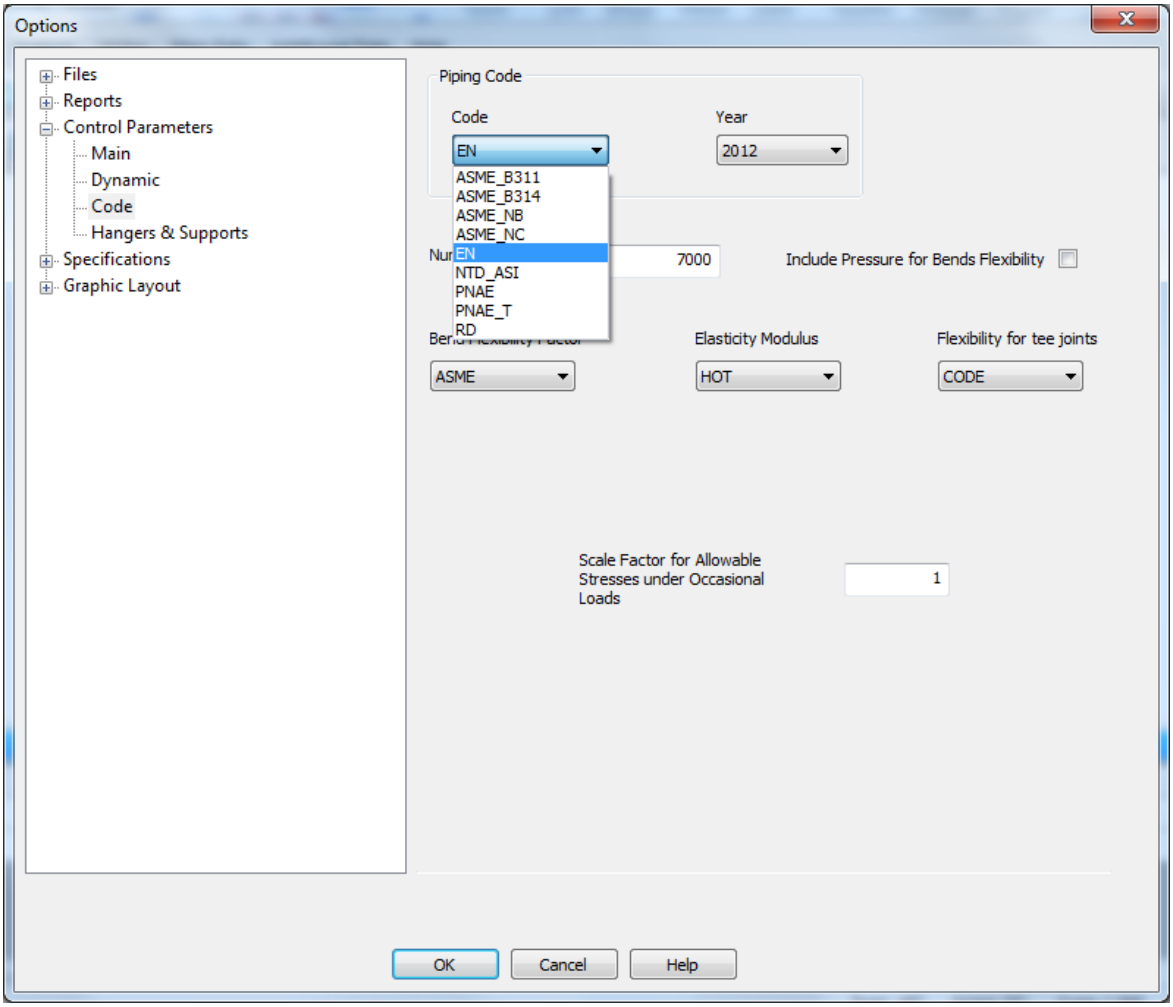
2.1 Piping Code

Since the data relevant to the piping model depend on the respective piping Design Code or Norms, it should be defined first of all.

Open menu "Tools" and choose "Options":



A dialogue window shall appear. In folder "Control parameters", choose "Code".

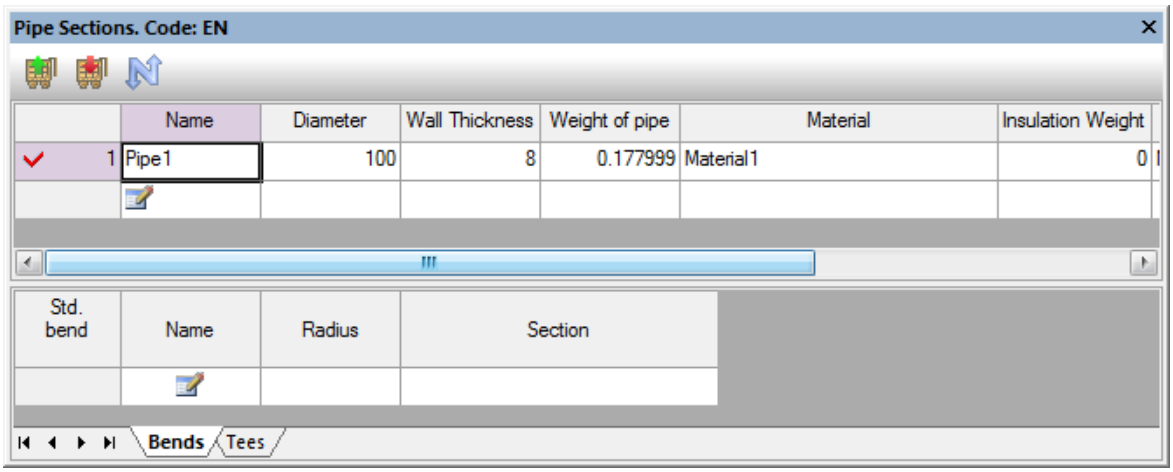


In the input field "Code", choose "EN" and click "OK".

Next step is to define an appropriate data for the piping sizes (cross-sections) used in the model.

2.2 Pipe cross-sections

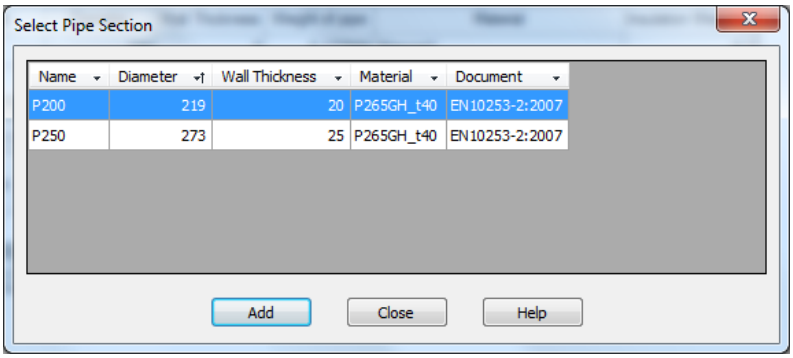
In the main window, use command "Pipe Sections" from the menu "Main Data". The following window shall open:



Press button  to import from database

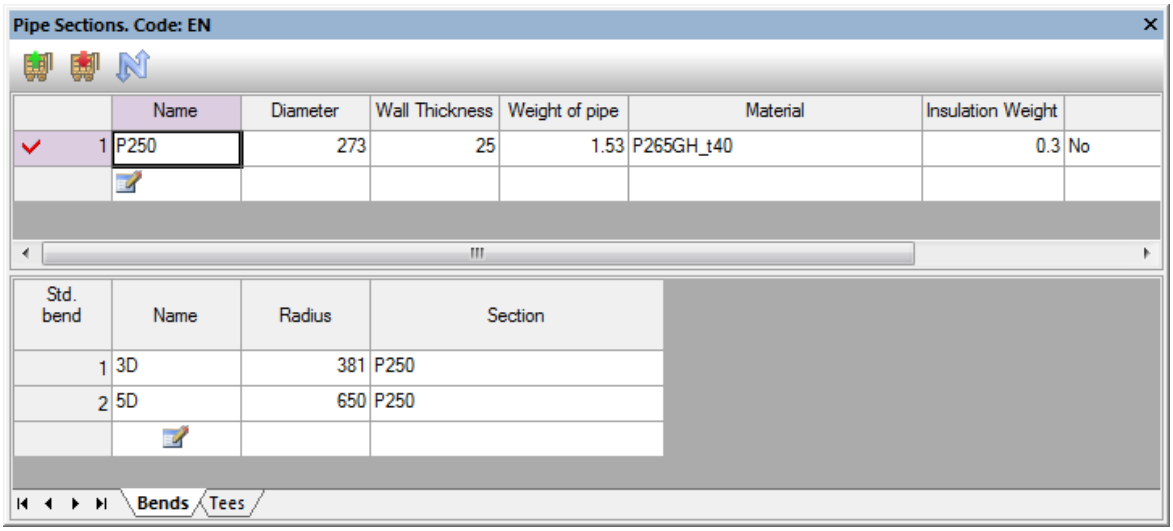
For Users' convenience, dPIPE is supplied with a set of databases containing different sizes of pipes and fittings along with materials used at nuclear and conventional power plants. These data are Code – dependent and are compiled in an editable text format (refer to files PIPE.dbs & MAT.dbs). These databases may be updated by the User.

The following dialogue window shall appear:



Choose sections P250 and click "Add":

The window "Pipe sections" now has a new entry. Expand the window to see all the lines:



The upper part of the table provides data on the pipe's sizes, while the lower - on the standard fittings matched above sizes.

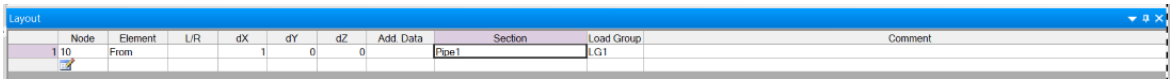
Note: the decimal symbol used in dPIPE is independent of Regional Settings and always is a dot (.)!!!

In the column "Insulation weight", enter weight for each section in N/mm: 30 kg/m ~ 0.3 N/mm.

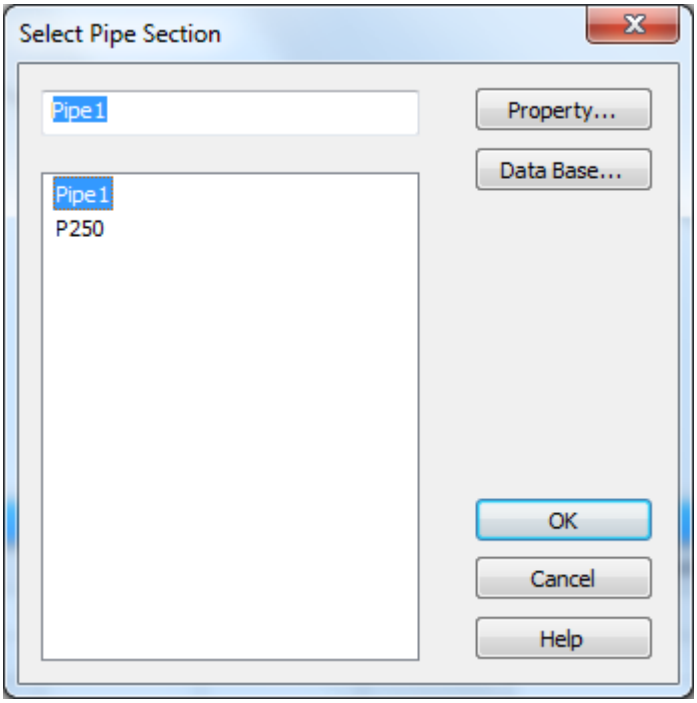
- Cross-section "Pipe1" cannot be deleted now

This data could not be deleted since this section is in use (it's referenced by piping layout). The same principle is used with other data: the program protects it from deleting if they are referenced somewhere…

In the lower window "Layout", double-click the left mouse button on field "Cross-section" (where "Pipe1" is written).



Window "Select Pipe section" shall appear:

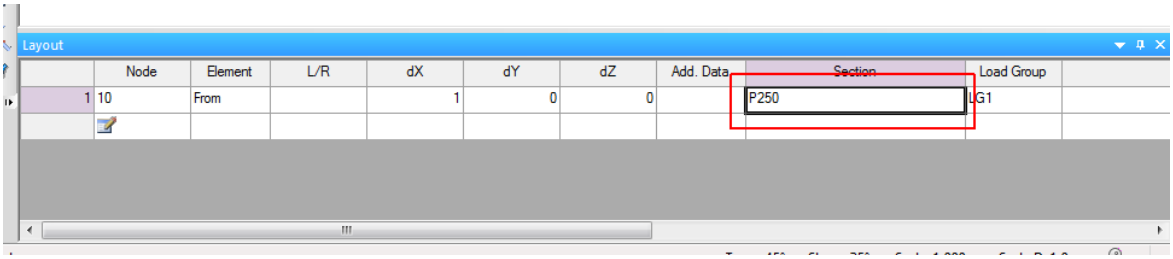


Choose P250 press OK.

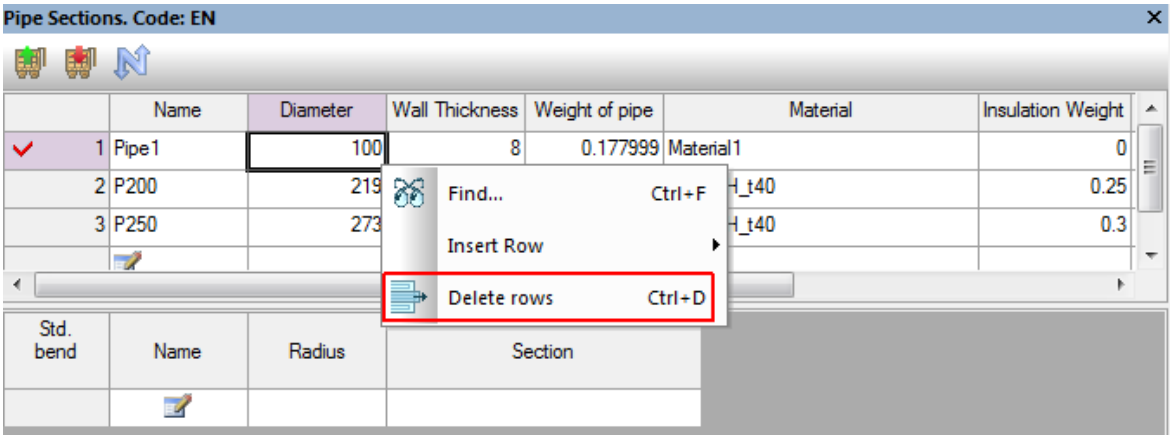
- Alternatively...

From the same dialogue there is a way to edit section's properties (select required section and press "Property" button) or adding the new one (just type new name and press OK, or select existing from the Data Base)

Input data can now be entered into table "Layout":



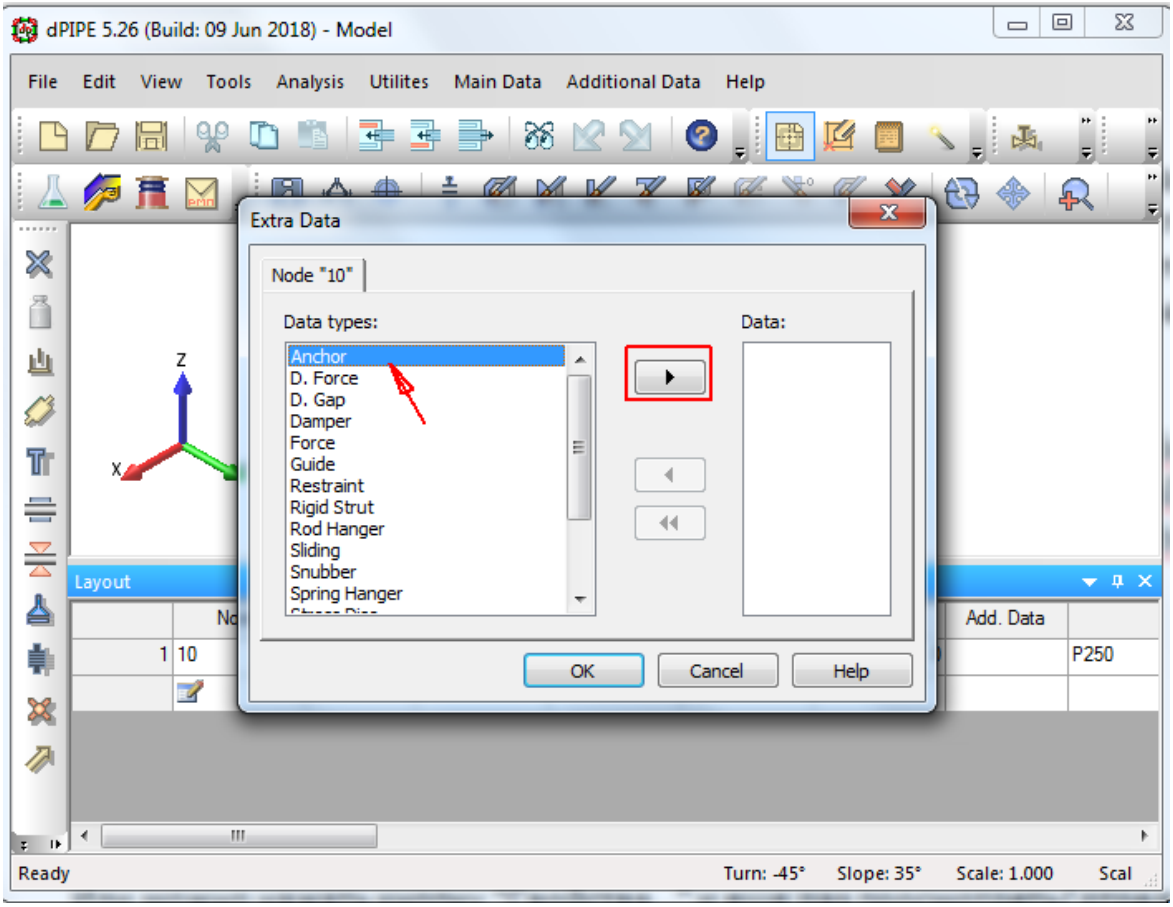
Now, section "Pipe1" can be deleted from the table "Pipe sections". Click the right mouse button on row "Pipe1" and choose "Delete rows" in the drop down list.



Close table "Pipe sections" and move on to table "Layout".

2.3 Piping layout

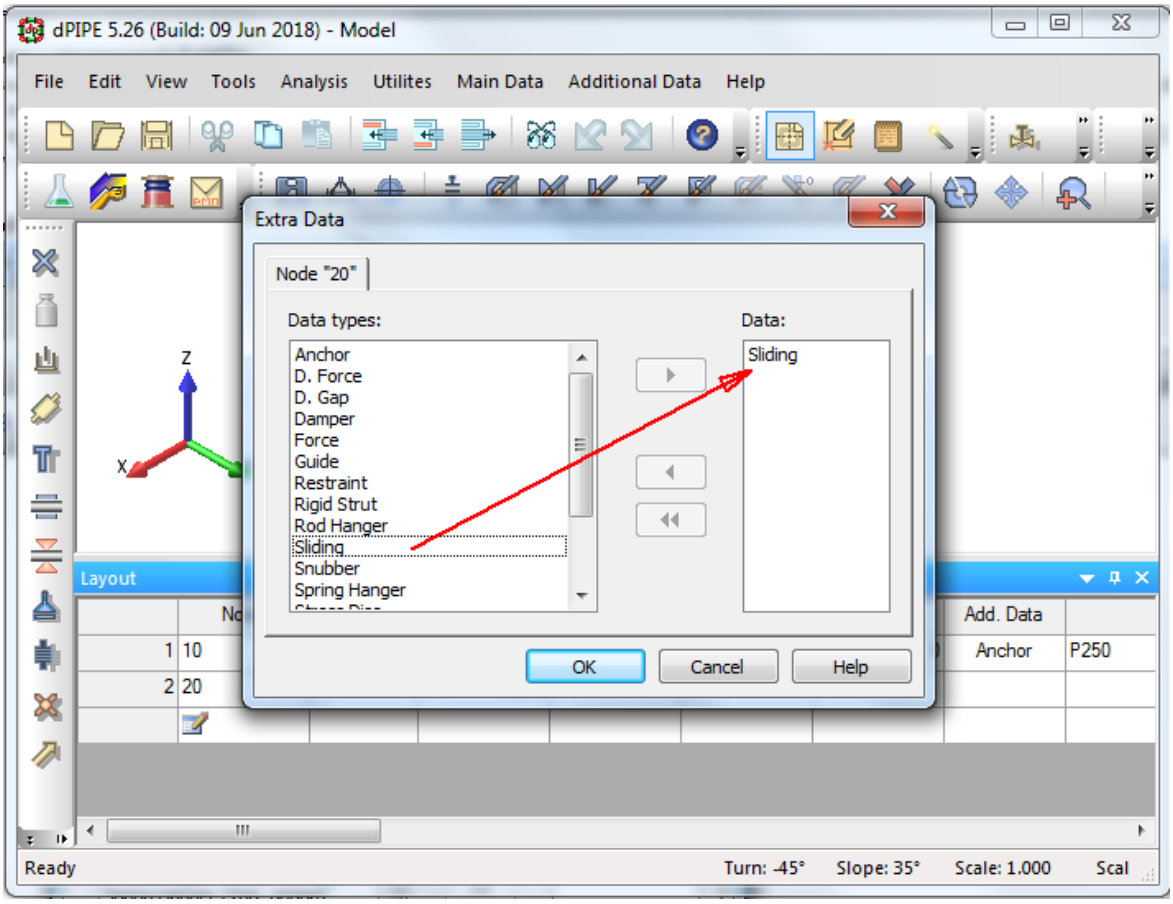
Enter 0, -1, 0 into fields dX, dY and dZ respectively. Then double-click the left mouse button on field "Add. data" (or use shortcut key “F2”). A dialogue window shall open. From the left list choose "Anchor" and either press the button with right arrow or double click the left mouse button on "Anchor" - an anchor shall appear in the node.



Press "OK", and the window shall close.

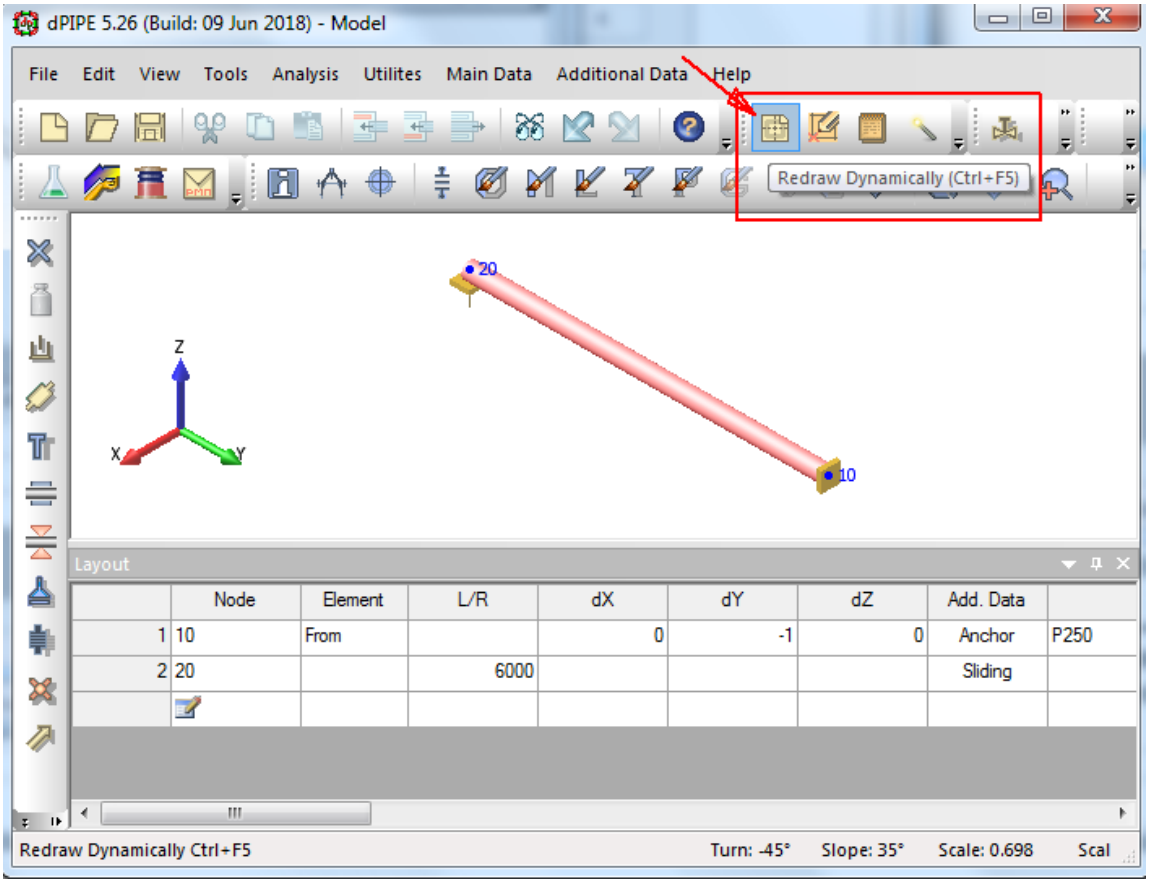
To continue entering input data, move on to the next row and enter **6000** in field **"L/R"**. This value represents the length of a piping section given in mm between anchor and sliding support. Upon pressing "ENTER", field "Node" shall have a new marking of node "20".

Enter data for a sliding support in a similar way to the anchor data in the same row in the field "Add. data":



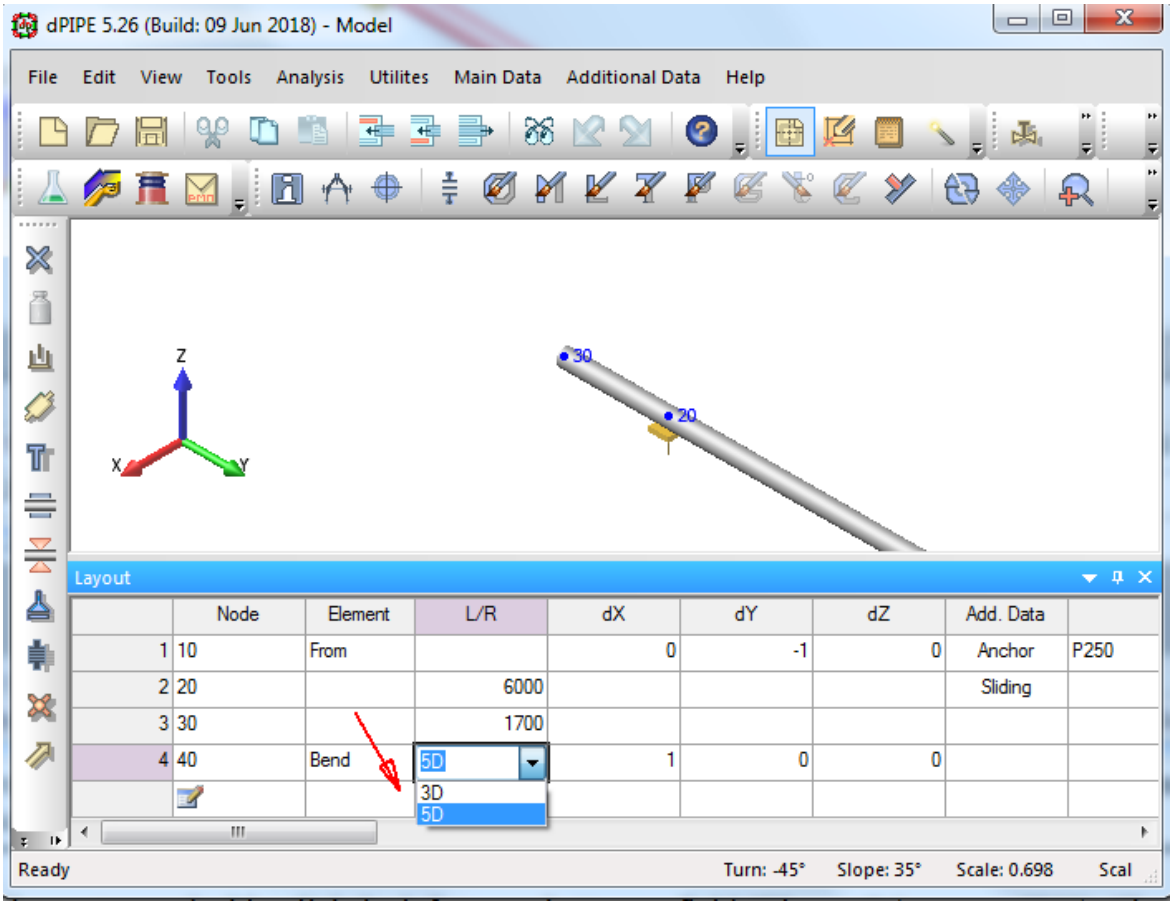
In field "L/R" of the third row enter 1700. An image of a pipe section should appear in graphic window between the toolbar and table "Layout".

The program working in two modes: the mode of dynamic drawing, when input data is checked simultaneously, and passive mode without checking and drawing. Switching between these two modes can be performed using either command "Tools" - "Redraw dynamically", or a hotkey combination "Ctrl+F5", or the button "Redraw dynamically"



In case of an error, a warning appears, and the program switches to the passive mode. To continue one should correct the mistake and press the "Redraw dynamically" button.

Then, move on to the next row, double-click the left mouse button on field "Element" (or press "F2"). Choose "Bend" in the dialogue window and press "OK". After that, double-click the left mouse button on the "L/R" box and choose type of bend "5D" from the drop-out list



Enter **1** in the "dX" box.

It is convenient to move between table cells using arrow keys (left - right and up - down arrows on the keyboard).

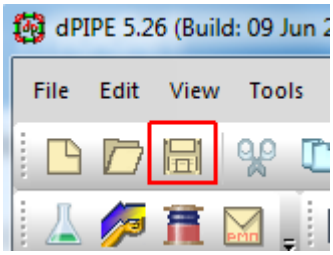
Move on to the next row, enter 4000 in the cell "L/R" and save the model.

Now let's consider how to set or redefine direction for a piping segment and build the rest of the model.

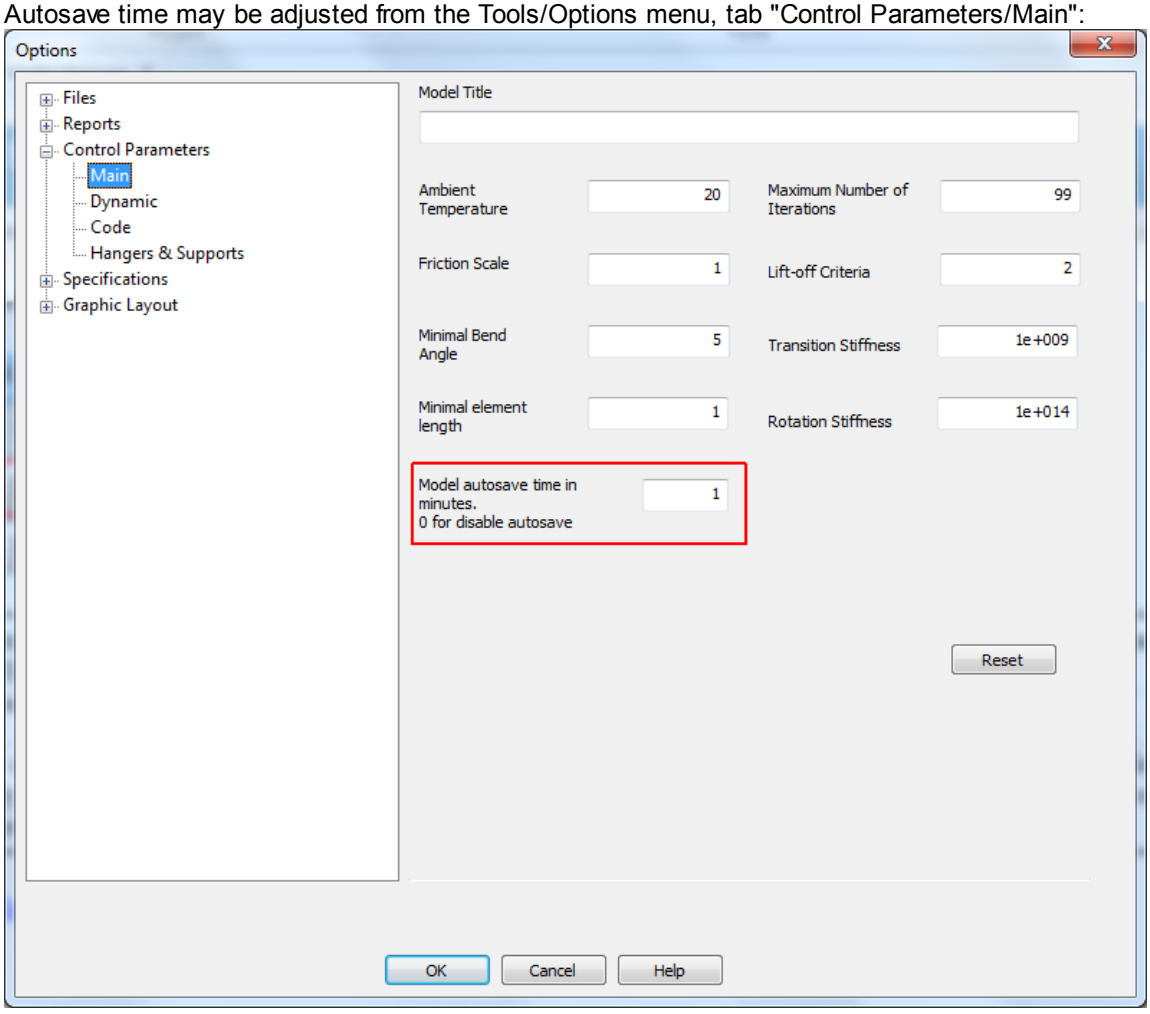
2.3.1 Saving the model

Choose the working directory, for instance, C:\work\ and save the model using a standard Windows dialogue. It is not recommended to save files in the dPIPE installation folder.

In menu "File", select "Save" or press the floppy disk button:



It's recommended to save the model time to time during the working session. By default, the program automatically saves the backup copy of the model every 5 minutes
Autosave time may be adjusted from the Tools/Options menu, tab "Control Parameters/Main":

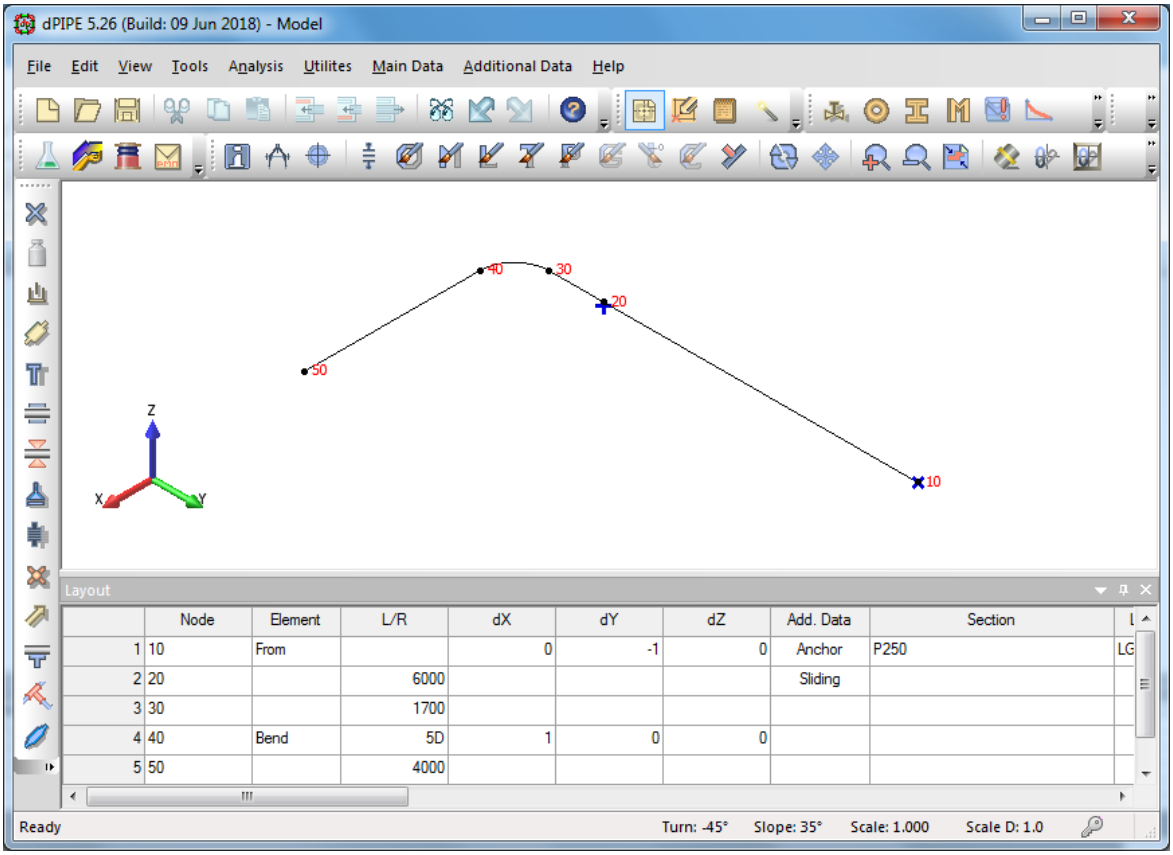


and in the case of program's failure may restore it. However, User should be aware, that there is no "undo" functionality in the working with layout's spreadsheet. To restore the last saved model, use "File" - "Last saved file" (or press "Ctrl-T").

Let's continue creating the model

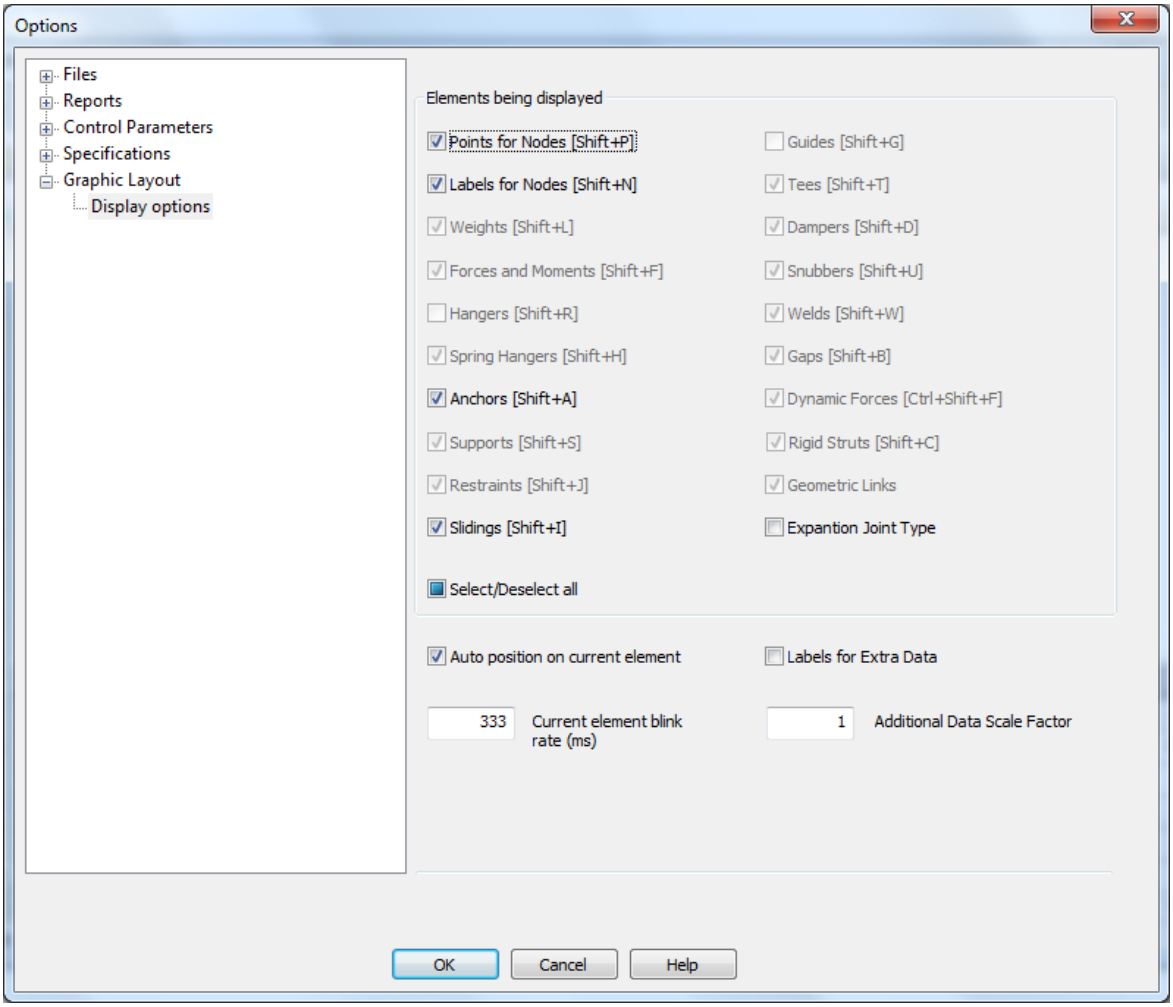
2.3.2 Assigning direction

Press "Ctrl+Shift+I" or "Ctrl+A" or respective buttons on the toolbar ( or ).

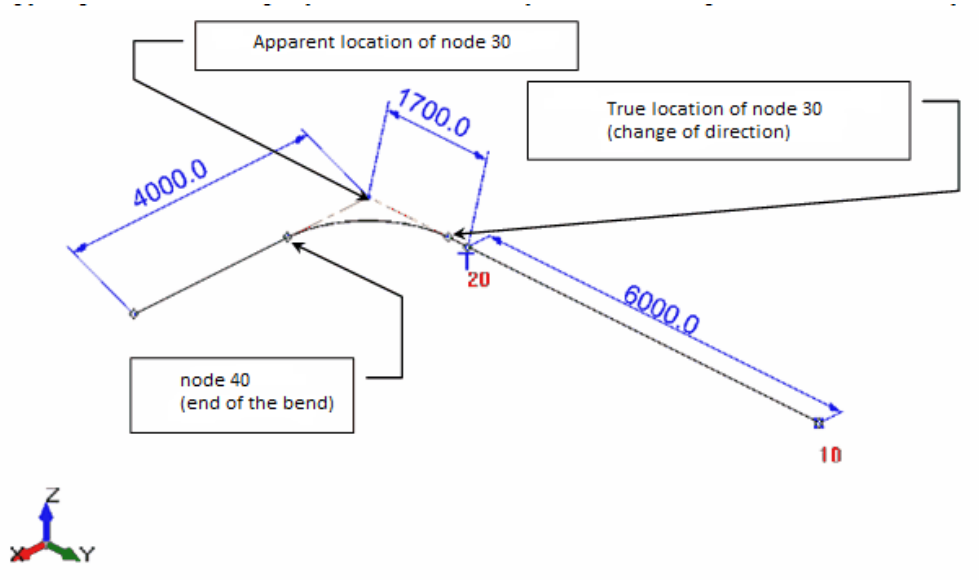


More about graphic window...

If some symbols of supports are missing in shown figure, press "F3" (or do it via menu items "Tools" -> "Options ..." -> "Graphic layout" -> "Display Options") and check the appropriate boxes:



Here is a piping segment, consisting of 4 elements and 5 nodes. The piping starts at node 10 ("From" command), where a fixed support (Anchor) is installed and is directed to -Y (-1 along the Y-axis). After 6000 mm, a sliding support (Sliding) is put in node 20. After another 1700 mm, the piping changes its direction. The new direction is + X (1 along the X-axis). The direction is changed at the bend (5D). The piping continues in the + X direction for 4000 mm to node 50. Let's look at the location of nodes 30 and 40:

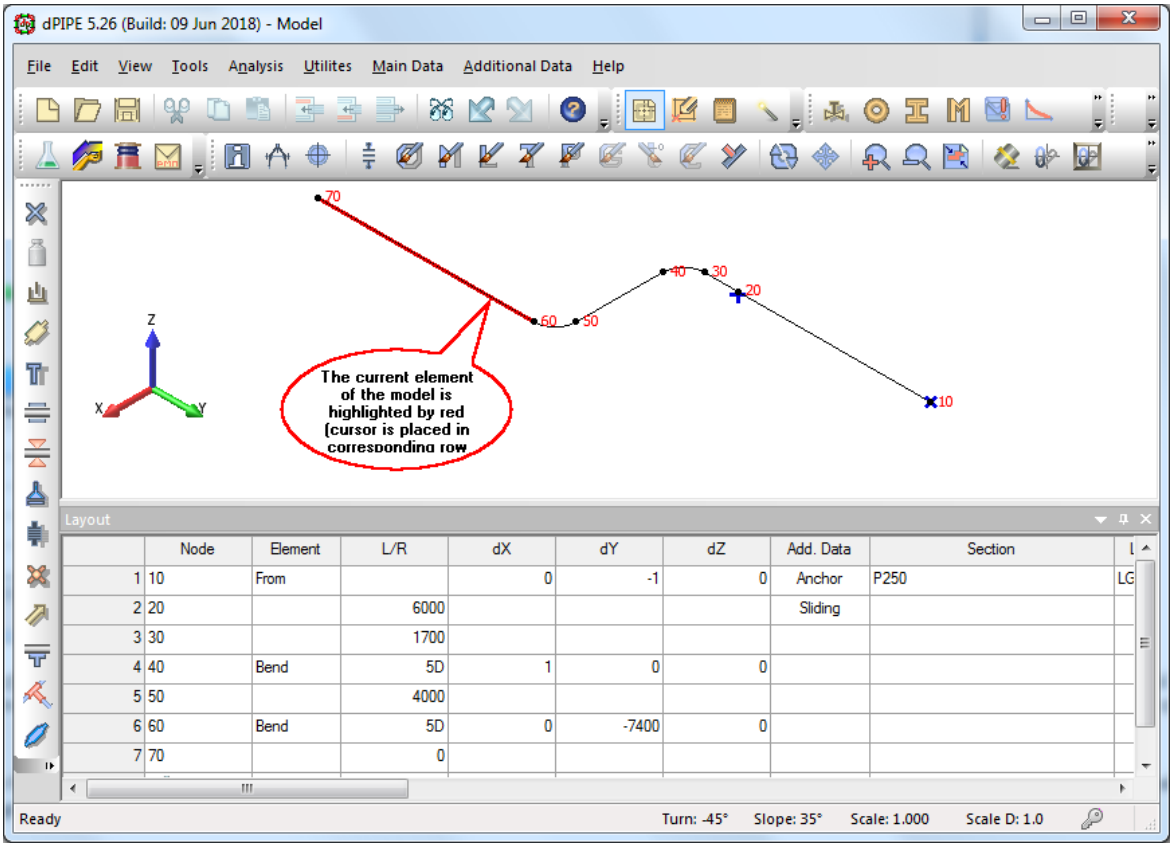


It should be noted: the length of any elements before or after BEND is measured as a distance between element's starting (or ending) node and the point of intersection of tangents to the bend's arc.

Let's continue entering the piping geometry up to the next fixed support.

2.3.3 Continuing to build the model

Move to the next row after node 50 and select "Bend" in the "Element" column. Select "5D" in the "L/R" column. In the dY field, type in "-7400" and press "Ctrl + Enter". Move the cursor to node 60. Press "Ctrl+Shift+I" or "Ctrl+A" or the appropriate buttons on the toolbar ( or ). The following image shall appear:

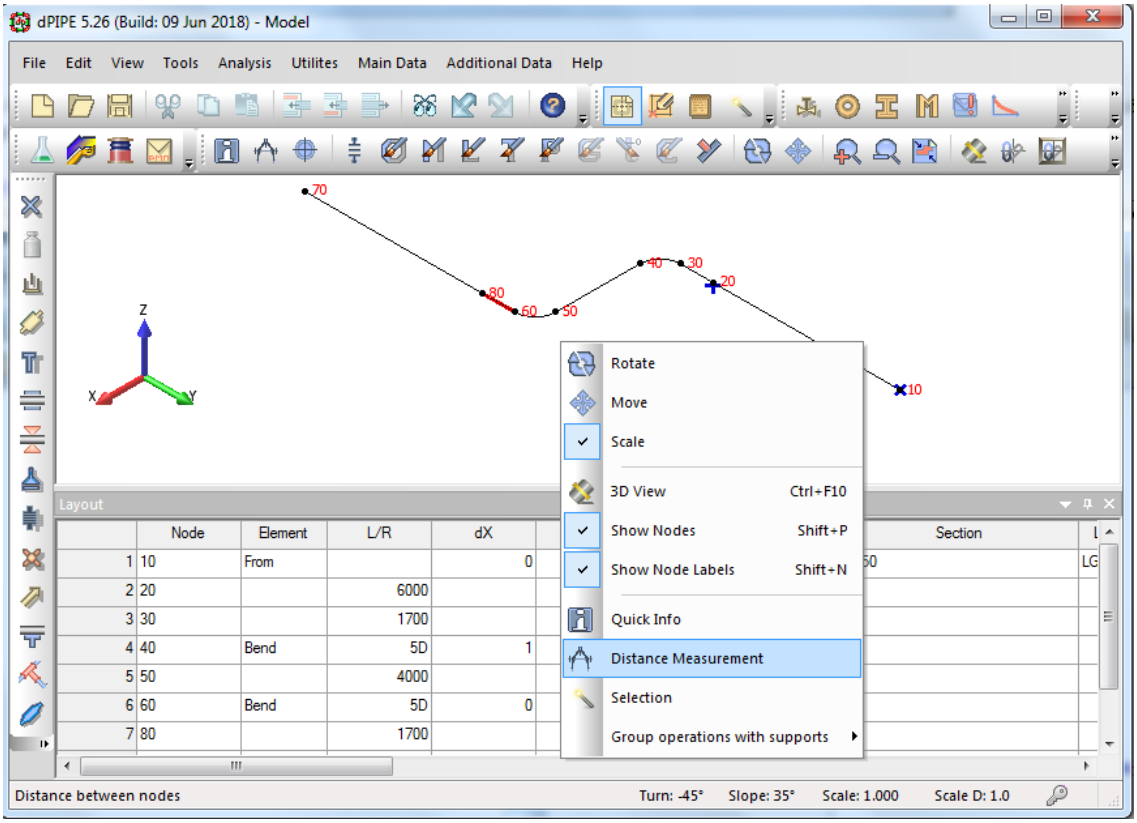


Note: the direction of piping is defined in DX/DY/DZ fields and assigned in the rows with nodes 10 and 40 by means of the direction cosines. The length of elements is specified at nodes 20, 30 and 50, respectively. At node 60, the length is specified in the direction field. But node 60 is the point, where the piping changes direction. Next element (Node 70) after the BEND was added with a length of zero. If one specifies a non-zero length, the program will ignore value and will normalize numbers to the directional cosines…

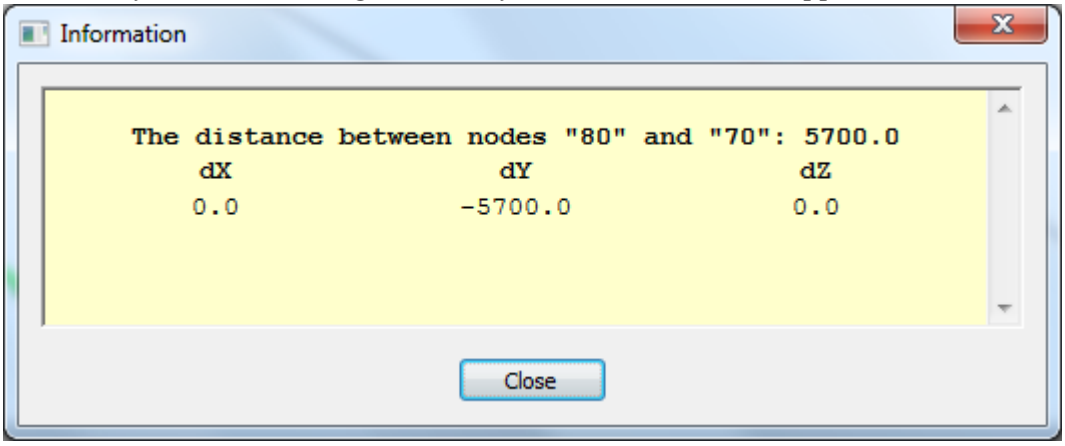
On the sixth row (node 60), press "Ctrl+Enter". Node 80 shall appear between nodes 60 and 70. Enter length "**1700**" in the "L / R" field. This will be the location of node 80 relative to node 60 along the -Y axis. A total distance of 7400 mm will remain unchanged. Therefore, the distance between node 80 and node 60 will be 7400-1700 = 5700 mm.

- Distance measurement

  In the graphics window, click the right mouse button and, if not selected, select items "Show nodes", "Show node labels" in the pop-up menu. Select "Distance measurement" from the pop-up menu:



Move the cursor to node 80 and click the left mouse button, then move it to node 70 and click the left mouse button again. The information window will appear:



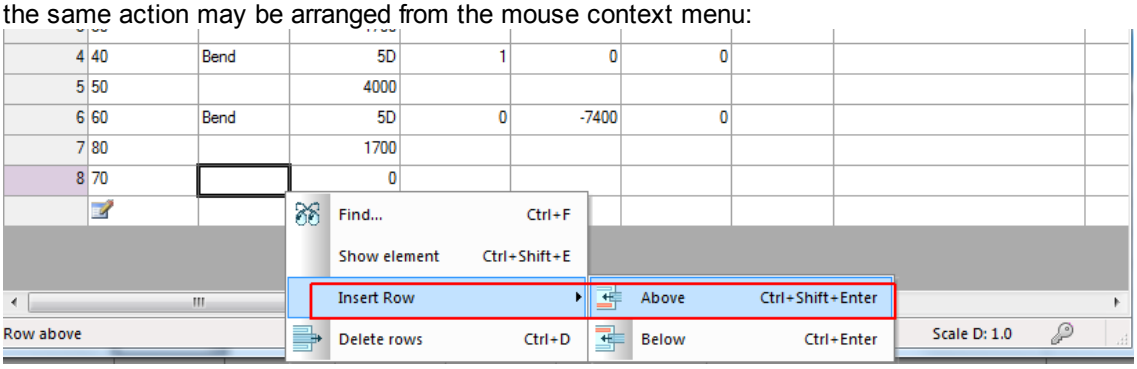
Close the window.

The same pop-up menu offers commands "Rotate", "Move", "Scale". To change the pivot point, press "Ctrl+E" or button  *. Changing scale can be done by moving the mouse in different directions or by scrolling. The same commands can be performed using the keyboard: rotation - by the cursor arrows, movement - by the cursor arrows with pressed button "CTRL", scaling by buttons "+"/"-".*

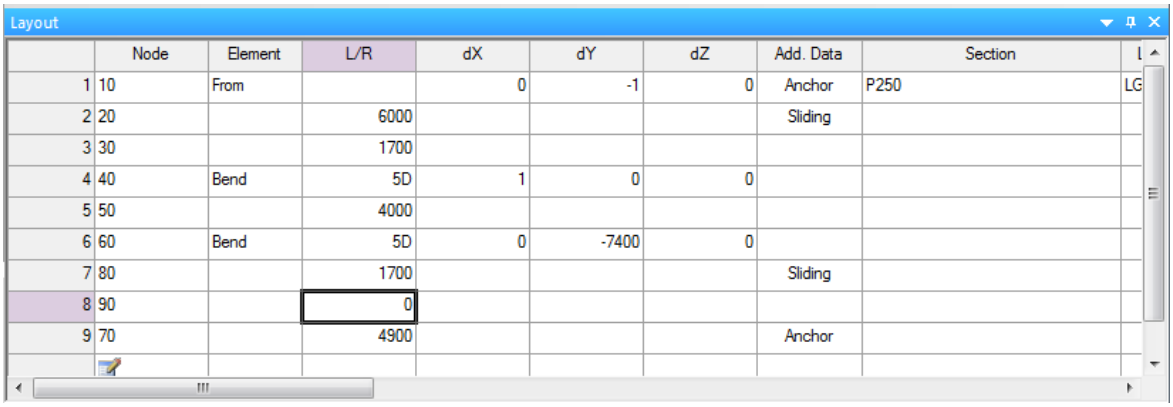
Add anchor to node 70 and sliding support to node 80.

Add node 90 before anchor at distance 4.9 m. To do that, move to the line with node 70, enter 4900 at "L/R" field and press "Ctrl+Shift+Enter"

the same action may be arranged from the mouse context menu:



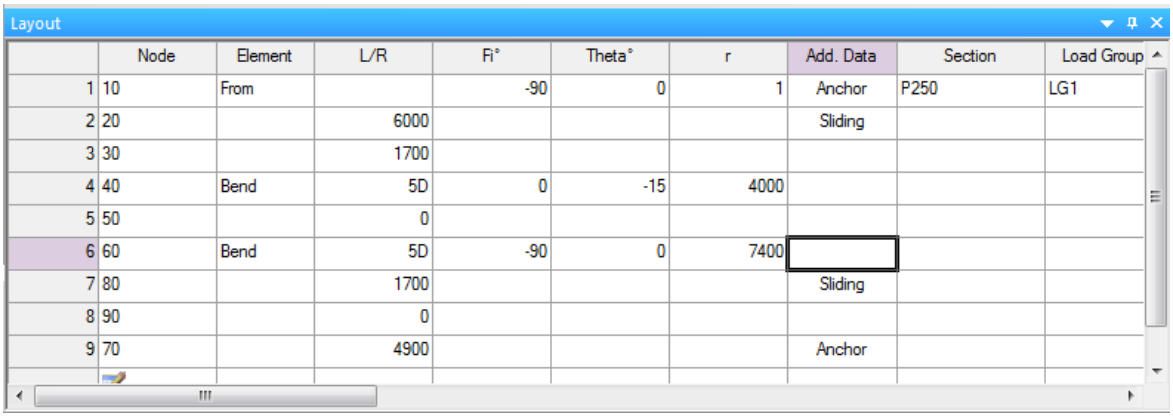
This action splits existing element "80-70" in two parts: "80-90" and "90-70":



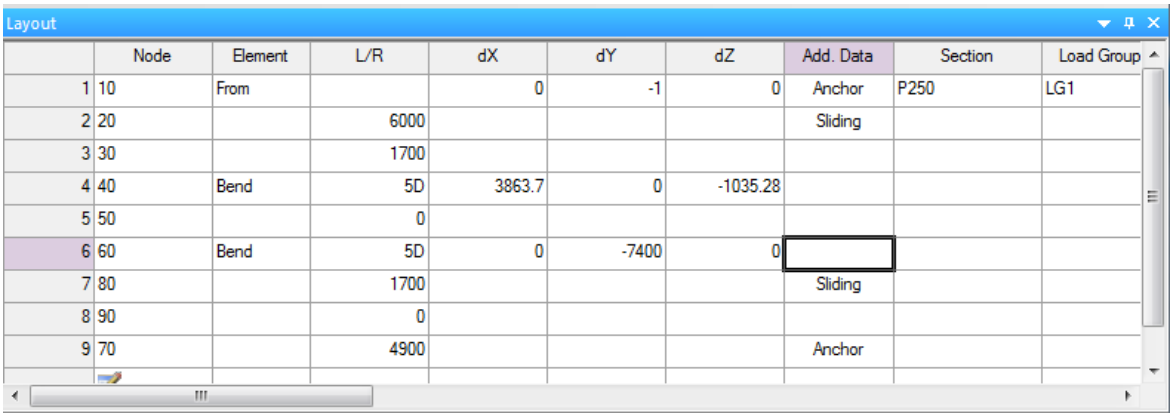
The distance between nodes 80 and 90 is 7400-1700-4900 = 800 mm. By inserting the intermediate nodes, we did not affect the overall length of the pipeline.

If there is a zero in the "L/R" column, the values in the directions fields "dX, dY, dZ" are considered as the a total length of the straight section, and the actual length of the element with L = 0 is calculated as difference between the total length and sum of lengths for all elements located between two points that change direction. Two zeros along straight run are not allowed

In conclusion, let's consider another useful way to enter the pipeline layout. Double-click on the table header in column "dX", "dY", or "dZ". The table will move from Cartesian to spherical coordinates. In node 40, enter the angle "Theta" = "**-15**" degrees, radius r = "**4000**", in node 50, enter "0".



Now the pipe section along the X axis is inclined on 15 degrees to the horizon. The length of the section is still 4.0 m. This option is especially useful for the modeling bends with an angle of 15, 30, 60 degrees. Double-click the right mouse button on the table header in the "Fi" or "Theta" or "r" column and return to the Cartesian coordinate system.


Layout

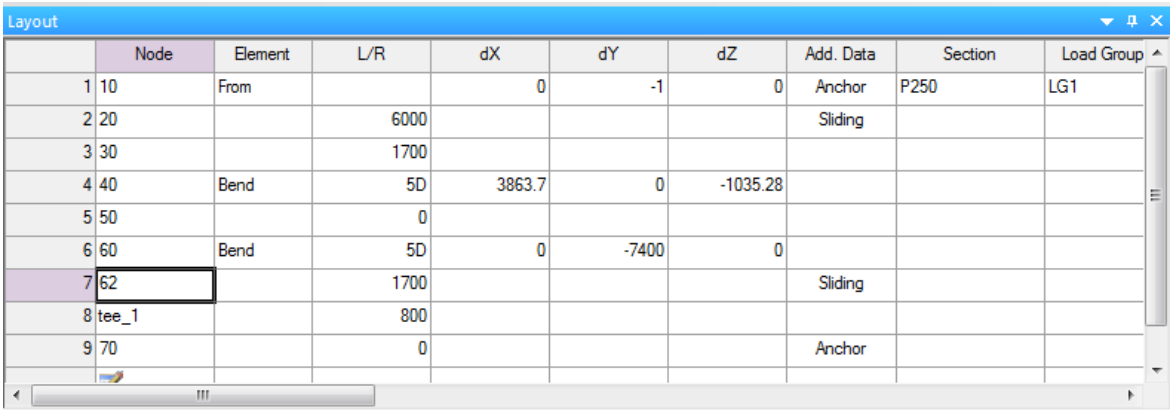
	Node	Element	L/R	dX	dY	dZ	Add. Data	Section	Load Group
1	10	From		0	-1	0	Anchor	P250	LG1
2	20		6000				Sliding		
3	30		1700						
4	40	Bend	5D	3863.7	0	-1035.28			
5	50		0						
6	60	Bend	5D	0	-7400	0			
7	80		1700				Sliding		
8	90		0						
9	70		4900				Anchor		

The program automatically calculates dimensions along X and Z axes.

In addition, a simple calculator is built into the table: enter "**7400-1700-4900**" in the "L/R" column in the 8th row (node **9**0) and press "Enter", in the 9th row, enter "0".

To rename node 80, place the cursor on this number (7th line, 1st column) and enter "62".

Rename node 90 to node "**tee_1**".


Layout

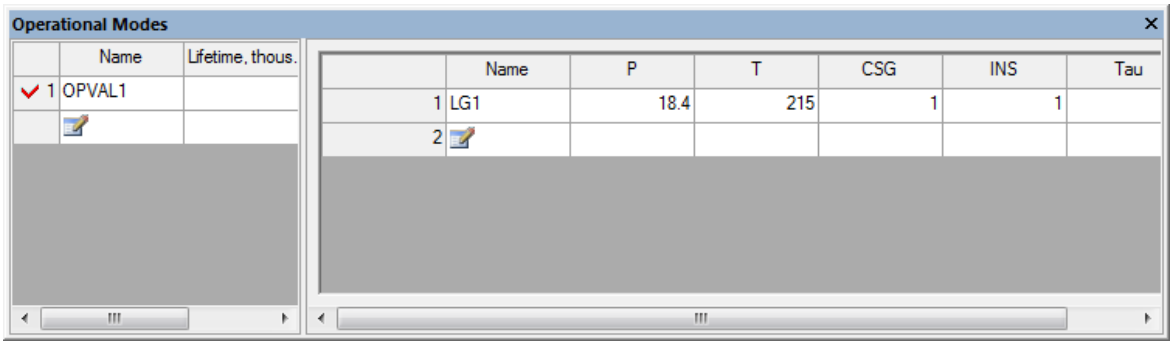
	Node	Element	L/R	dX	dY	dZ	Add. Data	Section	Load Group
1	10	From		0	-1	0	Anchor	P250	LG1
2	20		6000				Sliding		
3	30		1700						
4	40	Bend	5D	3863.7	0	-1035.28			
5	50		0						
6	60	Bend	5D	0	-7400	0			
7	62		1700				Sliding		
8	tee_1		800						
9	70		0				Anchor		

Now we will suspend entering data for the piping layout and will consider the setting of the operation modes, options for analysis and execution of analysis.

2.4 Operation modes

Choose command "Main data" - "Operation modes" (or press ) in order to assign medium operating parameters. In dPIPE 5, pressure is in MPa, temperature is in Celcius degrees, density of the medium is in fractions of water density.

Enter P =18.4 MPa, T=**215**°C, CSG=1.



Please, note:

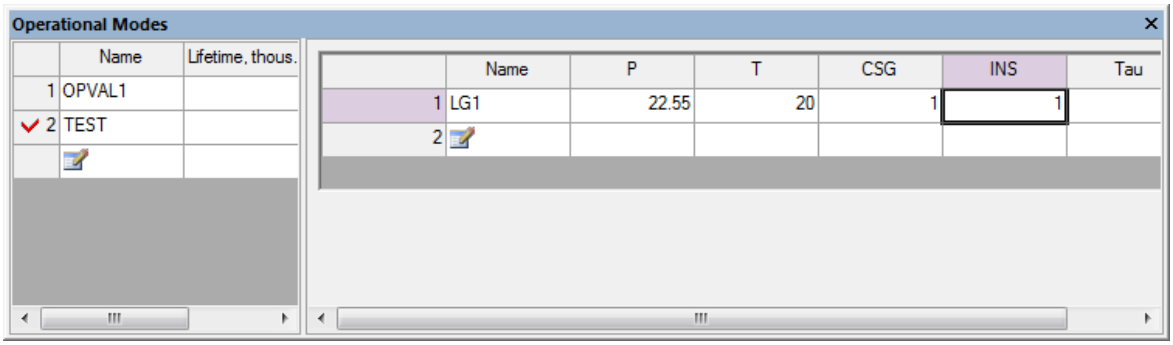
Marking LG1 (LG = Load Group) is used by default to assign piping sections with such parameters. Operation mode "OPVAL1" is a marking used by default to specify the operation mode.

Enter parameters for the hydro-testing mode. Place the cursor onto "OPVAL1" and press "Ctrl+Enter". Operating mode "OPVAL2" shall appear. Rename it as "TEST".

"TEST" is a predefined name for the hydro-testing mode. One can name it differently, but then has to change the reference in the standard set for analysis specification.

Enter P =**22.55** MPa, T=20°C, medium is water (CSG=**1**).

Hydro-testing temperature of 20°C corresponds to the default installation temperature.



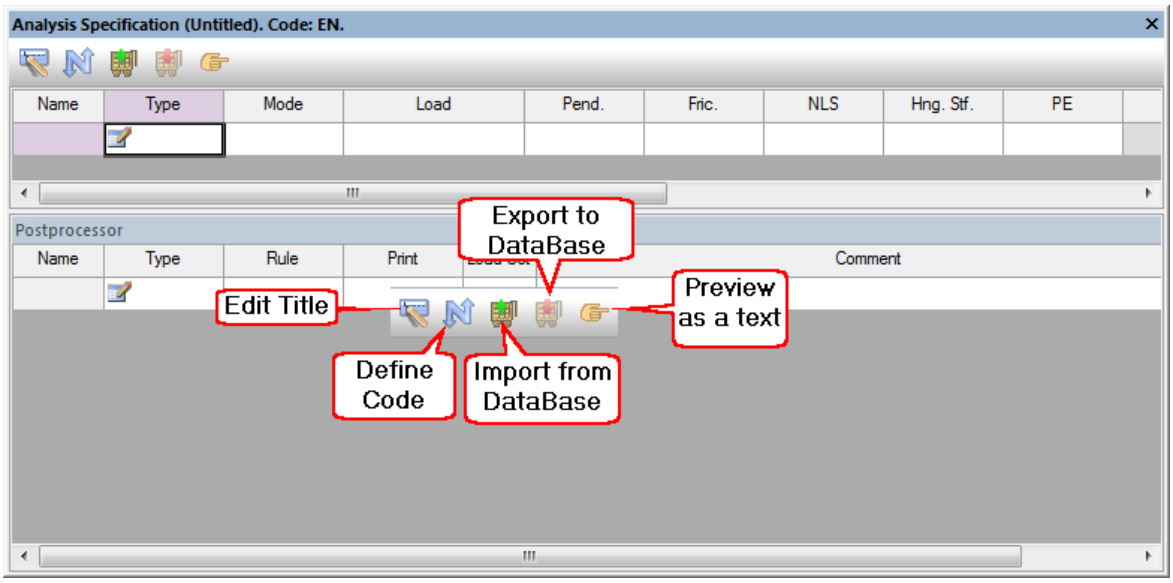
Close this table and determine the analysis specification and postprocessing of results.

2.5 Analysis specification

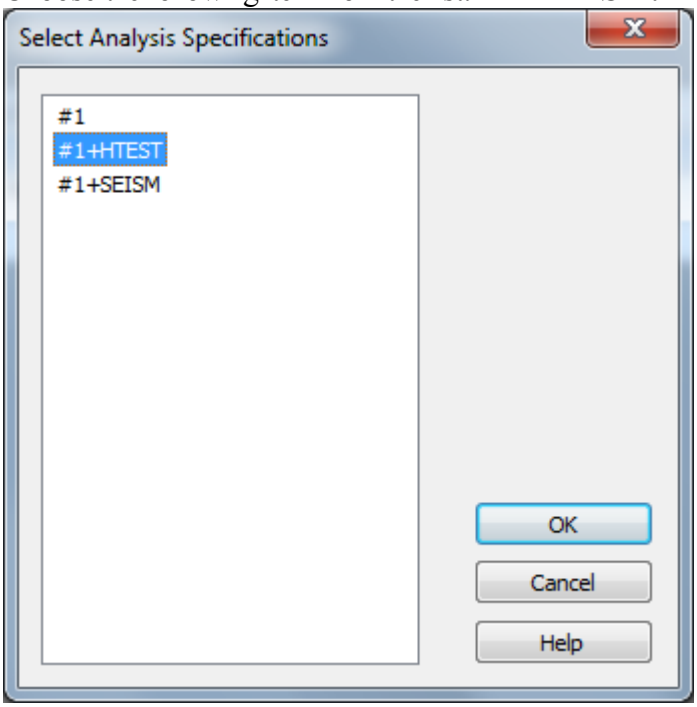
To check the Code compliance and define piping response under specified loading dPIPE performs set of analyses. Each analysis within the program is called a Load Case. In the post-processing stage, the program uses results of analyses to calculate the piping stresses, support's reactions, piping deflections, etc. Each post-processing directive is called as Load Set.

dPIPE provides a predefined set of the standard sequences of directives for analyses and post-processing. These options may be downloaded from the database (data is stored in solv.dbs file).

In the "Main Data" menu, choose "Analysis specification" and press the "Import from Database" button:



Choose the following item from the list: "***#1+HTEST***". Press OK**:**



In order to see the analysis specification commands, press "View as text":

Preview

```
SOLV "Spring Design + HTest. Stress Analysis (#1)"
& LC mod = '$OPER', type = 'DSGN', note = "Spring Hangers Design Loads" ; LC1
& LC mod = '$OPER', type = 'OPER_A', fric = 'No', pend = 'NO', note = "Hot Loads for spring design" ; LC2
& LC mod = '$COLD', type = 'OPER_B', fric = 'No', pend = 'NO', note = "Cold Load. Selection of springs" ; LC3
& LC mod = '$OPER', type = 'OPER_B', pend = 'YES', note = "Hot Load. Stage II" ; LC4
& LC mod = '$OPER', type = 'SUST_C', note = "Sustained Loads. Stage I. " ; LC5
& LC mod = '$COLD', type = 'OPER_B', pend = 'YES', note = "Cold Load. Stage IV" ; LC6
& LC mod = 'TEST', type = 'TEST', pend = 'YES', note = "Stage II (Hydro)" ; LC7
& LC mod = 'TEST', type = 'SUST_C', note = "Stage I (Hydro)" ; LC8

POST
& res = 'SGM1', ls = "LC5", note = "SGM1" ; LS1
& res = 'SGM1T', ls = "LC8", note = "SGM1T (HTEST)" ; LS2
& res = 'SGM3', ls = "LC4-LC6", note = "SGM3" ; LS3
& res = 'SGM4', ls = "LC5+LS03", note = "SGM4" ; LS4
& res = 'DISP', ls = "LC5", note = "Weight deflections" ; LS5
& res = 'DISP', ls = "LC4-LC6", note = "Thermal expansions" ; LS6
& res = 'DISP', ls = "LC8", note = "Hydro Test Displacements" ; LS7
& res = 'SUPP', ls = "LC4", note = "Hot Loads" ; LS8
& res = 'SUPP', ls = "LC6", note = "Cold Loads" ; LS9
& res = 'SUPP', ls = "LC7", note = "Hydro Test Loads" ; LS10

POST_REP load_hot = 'LC4', load_cold = 'LC6', load_des = 'LC1'
```

Steps..

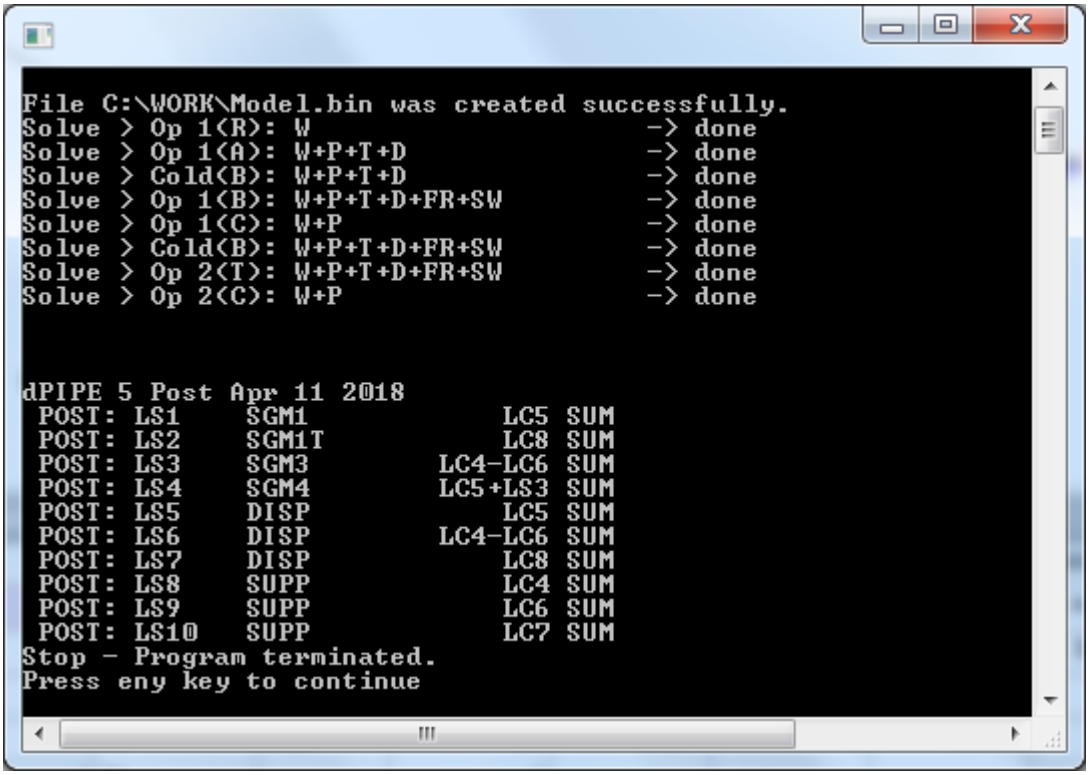
A first three Load Cases are intended for the selection of the springs used in the variable spring supports or hangers (if any). Next, LC4 calculates piping response in the hot state. LC5 defines internal forces used for the checking the code equation under sustained loads (SGM1 in the post-processing section). LC6 provides results for the cold state. LC7&LC8 are used for the hydro-test.

Close the "Analysis Specification" window and launch the analysis.

2.6 Run of analysis

Choose menu "Analysis" - "Batch mode" or press  on the toolbar. The program shall ask to save the model. Choose "Yes".

The analysis execution window shall appear:



The last line prompts the User to press any keyboard key to close the analysis execution window.

The analysis is complete, so one can view the results.

2.7 View the results

The results can be viewed either in a text or in a graphic format.

In order to view full text report, press  (menu "Analysis" - "View Results of Analysis"). To view spreadsheets with support loads, press  (menu "Analysis" - "View Support Loads").

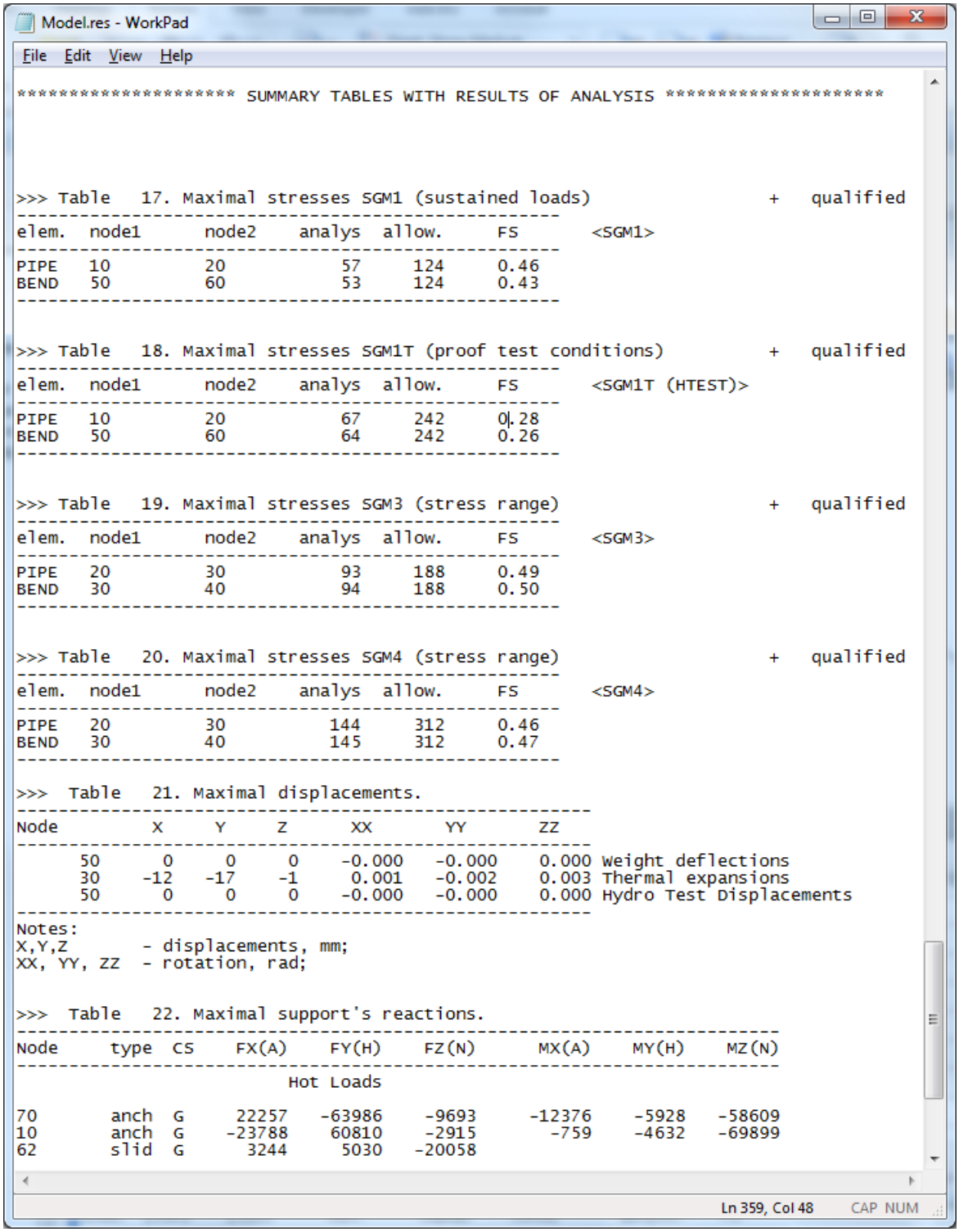
To view results in a graphic format, use the Pipe3DV software. To install it, press  (menu "Analysis" - "Pipe3DV").

2.7.1 Listing of results

Upon completing the analysis, dPIPE generates several text files, which are kept within a folder with the piping calculation model.

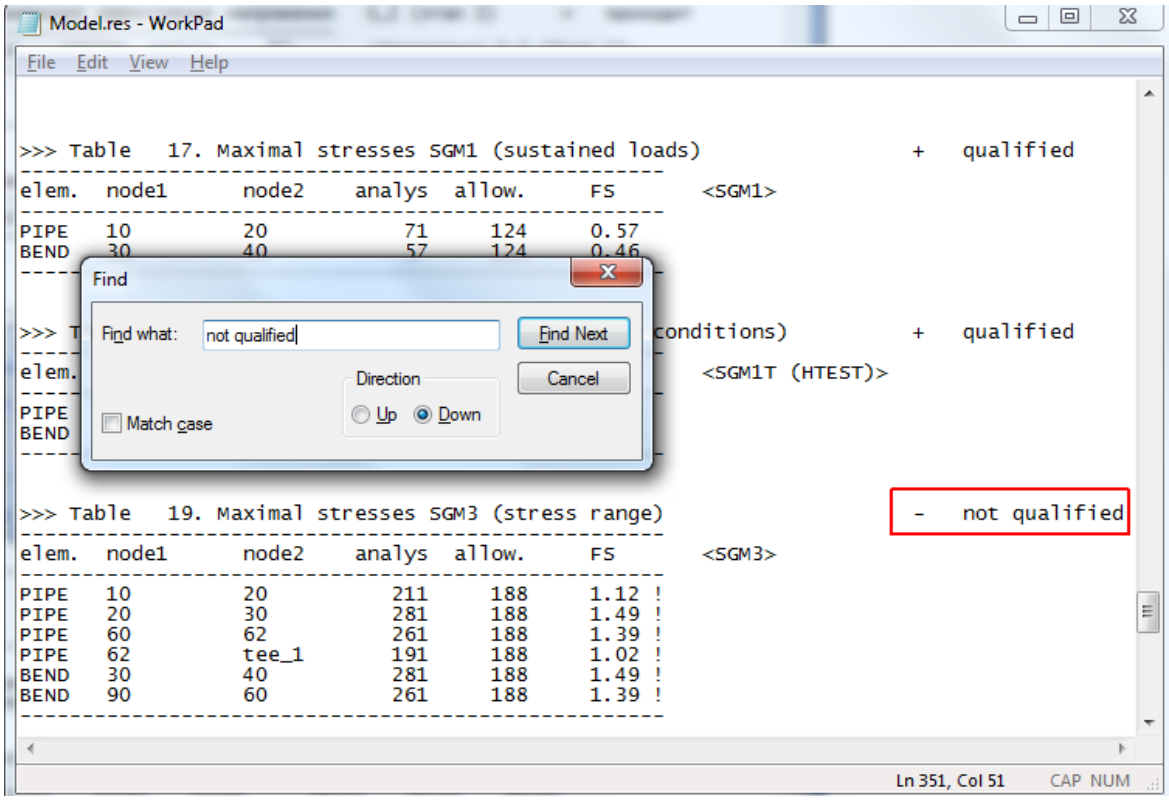
File with extension <model name>.res contains a full listing of the analysis results, based on the post-processing commands (POST). To open this file, press  (menu "Analysis" - "View Results of Analysis").

The opened window shall contain general information relation to the analysis options, as well as tables with results data. The last section of the file is named "Summary Tables with Results of Analysis", in which the values of the maximal displacements, support's reactions and stresses within piping elements are provided:

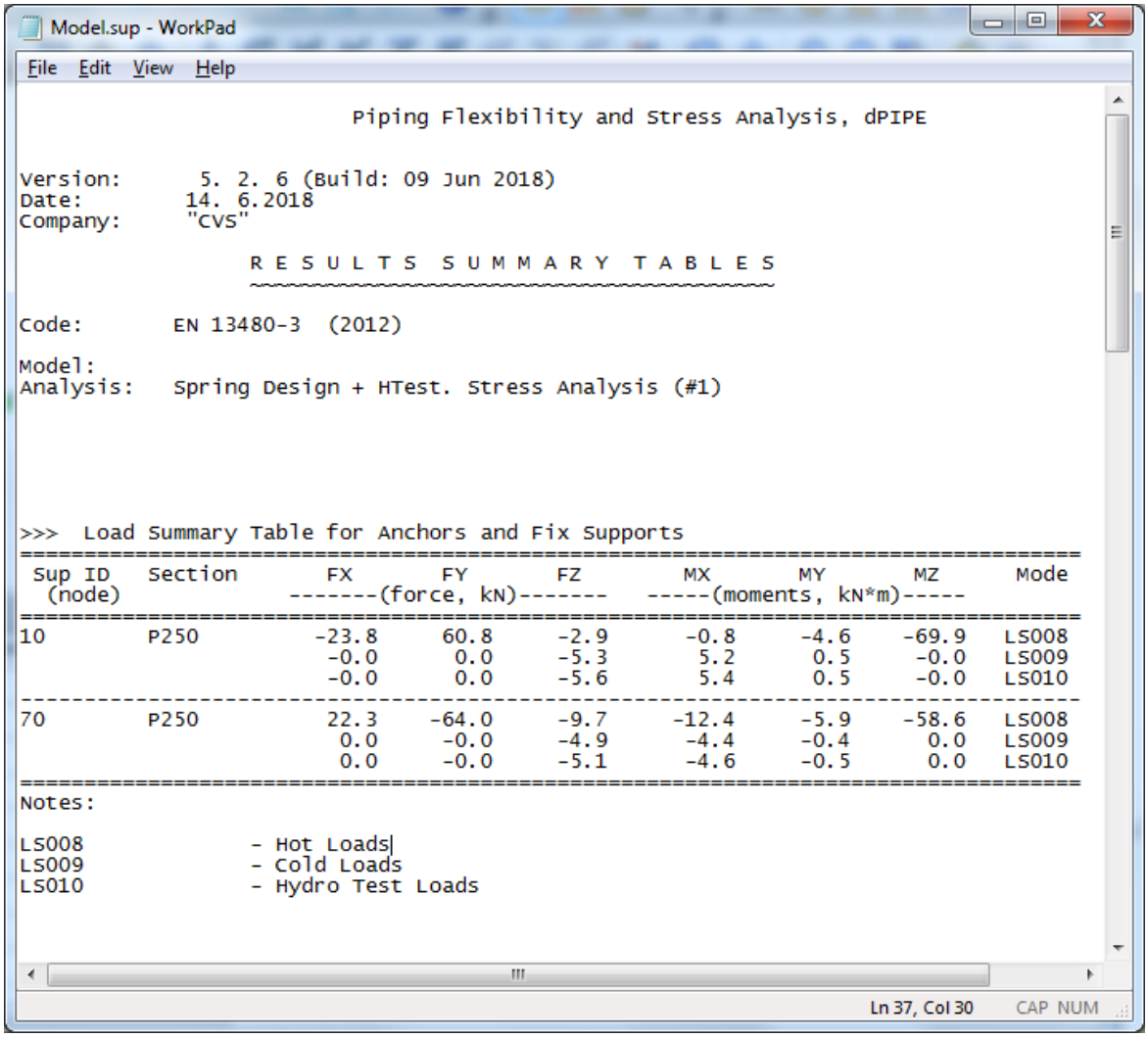


In case if calculated stresses exceed allowable ones, the respective line of the listing has an exclamation sign "!", and the spreadsheet title is marked "- not qualified". Using these indicators

and search command (Ctrl-F), it is easy to find instances, when the Code equations are not satisfied.



File with extension <model name>.sup contains spreadsheets with loads on spring hangers and supports. In order open this file, press  (menu "Analysis" - "View Support Loads"):



Model.sup - WorkPad

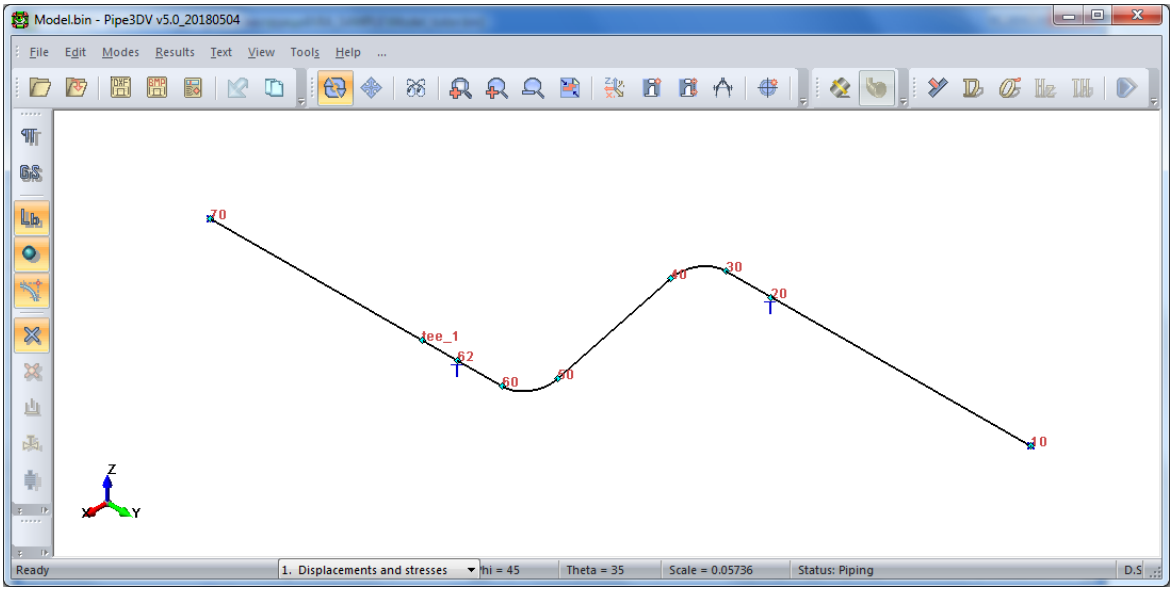
File Edit View Help

```
                        Piping Flexibility and Stress Analysis, dPIPE

Version:      5. 2. 6 (Build: 09 Jun 2018)
Date:        14. 6.2018
Company:     "CVS"

                  R E S U L T S  S U M M A R Y  T A B L E S

Code:        EN 13480-3  (2012)

Model:
Analysis:    Spring Design + HTest. Stress Analysis (#1)



>>>  Load Summary Table for Anchors and Fix Supports
=============================================================================================
 Sup ID    Section       FX       FY       FZ       MX       MY       MZ       Mode
  (node)             -------(force, kN)-------   -----(moments, kN*m)-----
=============================================================================================
10          P250        -23.8     60.8     -2.9     -0.8     -4.6    -69.9    LS008
                         -0.0      0.0     -5.3      5.2      0.5     -0.0    LS009
                         -0.0      0.0     -5.6      5.4      0.5     -0.0    LS010
---------------------------------------------------------------------------------------------
70          P250         22.3    -64.0     -9.7    -12.4     -5.9    -58.6    LS008
                          0.0     -0.0     -4.9     -4.4     -0.4      0.0    LS009
                          0.0     -0.0     -5.1     -4.6     -0.5      0.0    LS010
=============================================================================================
Notes:

LS008             - Hot Loads
LS009             - Cold Loads
LS010             - Hydro Test Loads
```

Ln 37, Col 30 CAP NUM

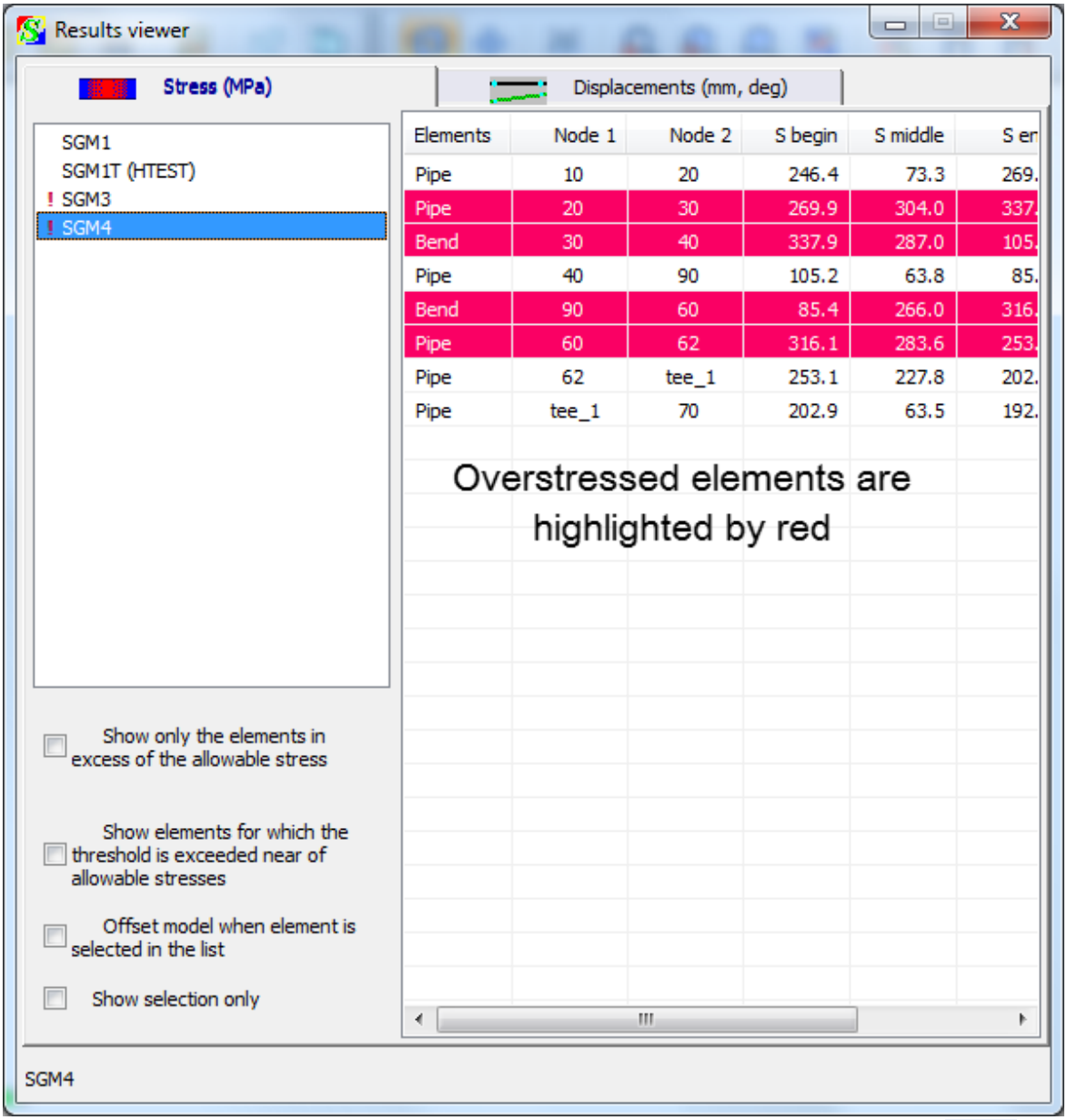
2.7.2 Pipe3DV

Pipe3DV is a browser for the dPIPE models and analysis results. To launch PIPE3DV use  button, as result a window with the analyzed model will be opened:

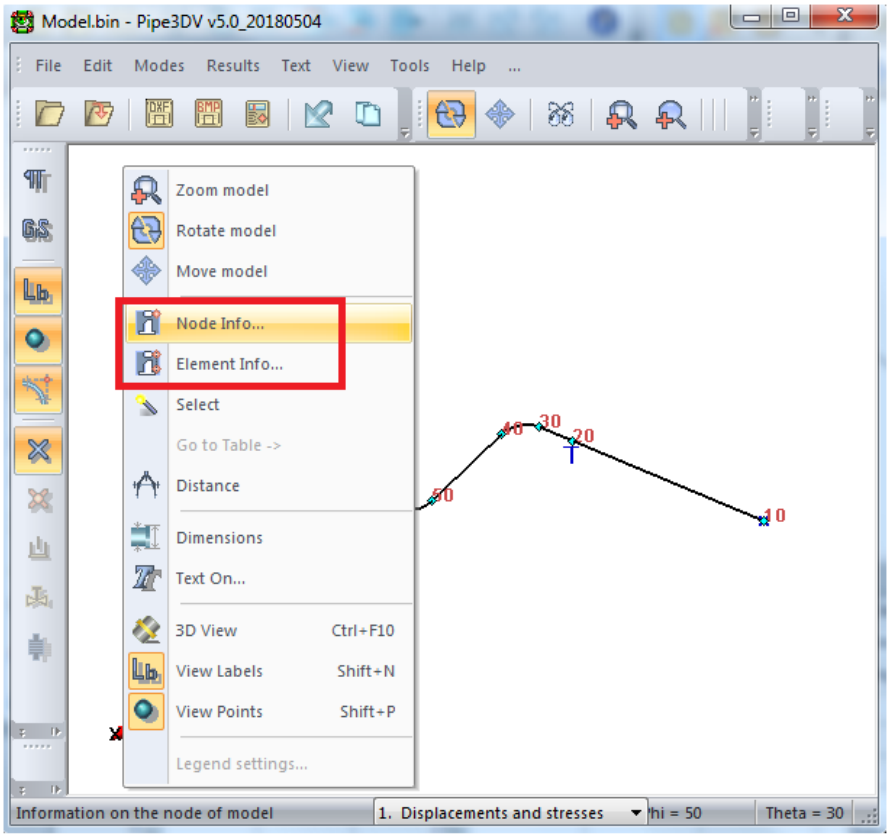


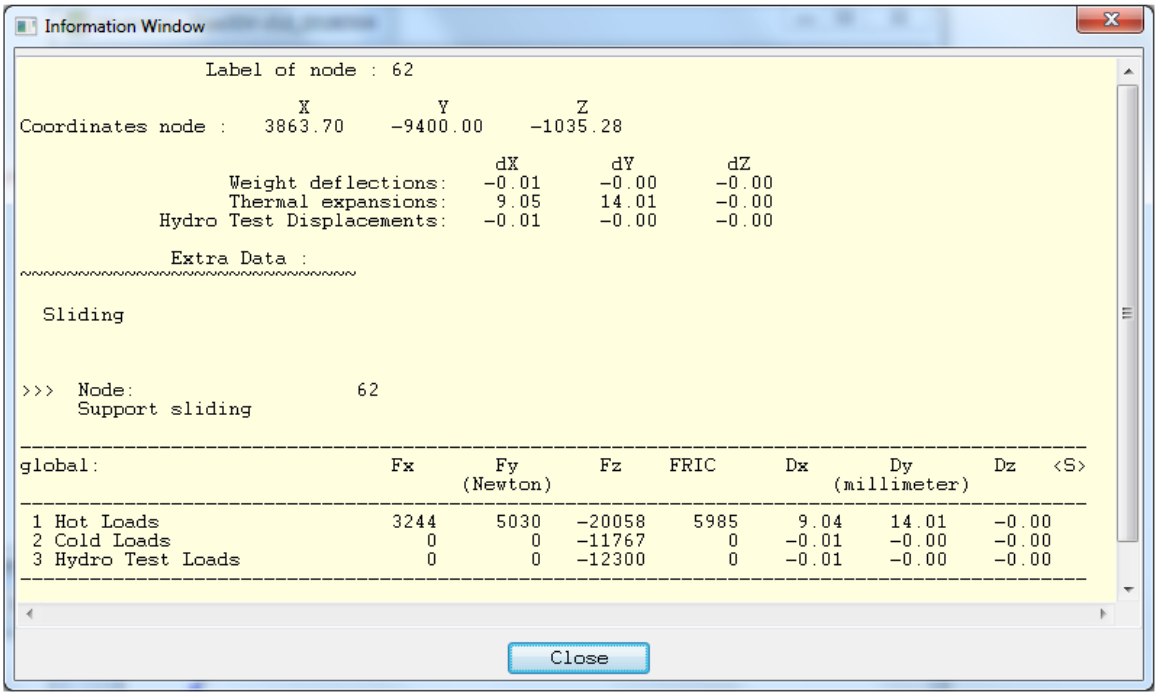
The left toolbar is used to switch on/off the different symbols on the picture: piping supports, nodal data, etc. The button  is utilized to turn on/off the picture from 1D to 3D mode. Picture's scaling may be accomplished with numpad keys "+/-". The model may be moved or rotated either by the mouse or with arrows keys (use CTRL to change the mode from the moving to the rotation).

The buttons  and  are used for the visualization of the stresses and piping deflections respectively. Results of analyses may be accessed also from the menu Results/Table (or use CTRL-R shortcut):



Mouse's context menu provides an access to the results associated with node () or element ():





Other PIPE3DV options: measuring of the distances between nodes (button ), dimensioning (), selection and showing of the different parts of the model basing on some features (piping materials, sections, logical names, etc.).

Pipe3DV is freeware, it does not require special installation, it may be used by the Users for the exchange of results with third parties, just sending file <model.bin> and giving the reference on PIPE3DV…