ЦКТИ–ВИБРОСЕЙСМ

ИНЖИНИРИНГ ПРОЧНОСТЬ ВИБРОЗАЩИТА И СЕЙСМОСТОЙКОСТЬ

CKTI–VIBROSEISM

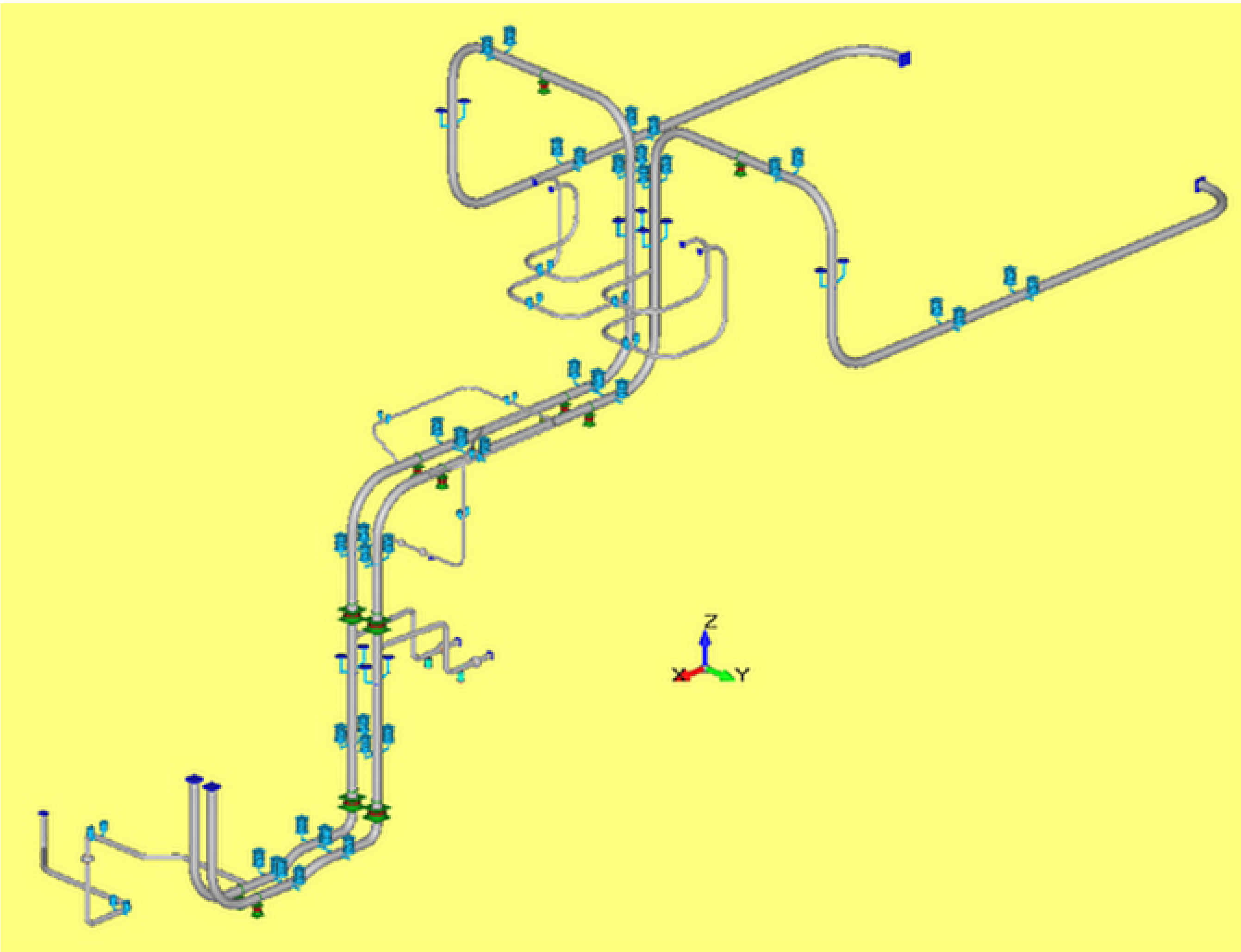
A STRUCTURAL-MECHANICAL CONSULTING ENGINEERING FIRM

Piping Stress and Flexibility Analysis Software

dPIPE 5

User Manual

Version 5.2.8



Saint Petersburg, Russia

Table of Contents

Part I Abbreviations 6

Part II Introduction 6

Part III Software structure. File system 7

Part IV PC configuration requirements 9

Part V Software installation 10

Part VI Protection keys and License Management 10

1 User hardware Key Sentinel HL Pro DL ... 11
2 Network Hardware Key HL-NET ... 11
3 User Software Key SL-UserMode ... 12
4 Network Software Key SL-AdminMode ... 12
5 dPIPE in Sub-Network ... 13
6 Sentinel SL License Transfer between two PCs ... 14

Part VII DDE spreadsheet 16

1 Spreadsheet interface ... 16
2 Spreadsheet navigation ... 22
3 Partial visualization of the model ... 24
4 Copy, paste, modify and delete piping segments ... 27
5 Changing the number display format ... 30
6 Toolbars ... 30
- Main toolbar ... 32
- "Piping" toolbar ... 33
- Spreadsheets with basic data ... 33
- Analysis ... 34
- Additional data ... 34
- Operations with the graphic window ... 35
- Graphic input of the model ... 36

7 Hot key summary ... 37
8 Menu commands ... 39
- File ... 39
  - Create PCF ... 40
  - Archive Project ... 41
  - dP5 Backup ... 42
- Edit ... 42
- View ... 42
- Tools ... 43
  - Group operations with supports ... 45
  - External programs ... 47
  - Interface Language ... 49
  - Options and settings ... 49

Files ... 50
Reports ... 51
Templates ... 52
Supports & Spring Hangers ... 52
Output Listing ... 53
Valves ... 54
Small Bore Pipes ... 55
Control parameters ... 56
Main parameters ... 56
Dynamic parameters ... 58
Code ... 59
Hangers and Supports ... 61
Specifications ... 62
Fatigue Analysis ... 62
HELB ... 64
Export to LICAD ... 65
Display options ... 66
Analysis ... **67**
Utilites ... **69**
Main Data ... **70**
Additional data ... **77**
Help ... **79**

Part VIII Input data language 80

1 General commands ... 82

Insert data from another file (INCLUDE) ... **83**
Model title (TITLE) ... **83**
Control parameters (CTRL) ... **84**
Fatigue Data (FAT) ... **97**
Creep Data (CREEP) ... **99**
Materials (MAT) ... **100**
Pipe cross-section (PIPE) ... **105**
Beam cross-section (BEAM) ... **112**
Operating modes (OPVAL) ... **115**
Discont. stresses due to temp. gradient and stratification (GRAD) ... **117**
Water oxidation (ENVFAT) ... **119**
Data for spring hangers and supports (SDEF) ... **120**
Seismic response spectra (SPEC) ... **121**
Seismic accelerograms (ACCE) ... **123**
Specification for analysis (SOLV) ... **125**
Specification for Dynamic Analyses (DCASE) ... **129**
Specification for postprocessor (POST) ... **134**
Specification for fatigue analysis (FATG) ... **139**
Specification for HELB analysis (POST_HELB) ... **142**
Specification for report generation (POST_REP) ... **144**
Specification for Support's Allowable Loads (SUP_LOADS) ... **149**
Export of piping Supports Loads to LICAD (DP2LCD) ... **150**
Managing the databases (DBF) ... **152**
Suppressing results by headers ($NOHEAD) ... **154**
End of input data (END_OF_DATA) ... **154**

2 Local Commands ... 154

Initialization parameters ... **159**
From (F) ... **160**
Straight pipe (P) ... **161**
Bend-1 (B) ... **162**
Bend-2 (B) ... **165**
Miter bend (MTR) ... **166**

Reducer (R) ... 169
Valve (V) ... 171
"Half-valve" (V1, V2 commands) ... 175
Angle valve (VA) ... 177
Valve with offset (VO) ... 183
3-way valve (V3W) ... 189
Expansion Joint (EJ) ... 194
Axial Expansion Joint (EA) ... 198
Tied Expansion Joint (ET) ... 201
Hinge Expansion Joint (EH) ... 203
Gimbal Expansion Joint (EG) ... 205
Rigid Link (RX/RP) ... 207
Flexible Joint (FJ) ... 209
Cold Spring (CS) ... 211
Beam (S) ... 212
Set Position (POS) ... 214
Tee/Branch Connection (TEE) ... 217
"Standard" (TEE) ... 223
Stress Indexes ... 224
Welds (WLD) ... 226
Concentrated weight (CW) ... 228
Concentrated Loads (FOR) ... 229
Anchor (ANC) ... 231
Support (SUP) ... 237
Translational Restraint (STS) ... 241
One-way restraint (STS+, STS- commands) ... 245
Skewed Restraint (SRS) ... 248
Rigid Strut (STRT) ... 252
Guide Support (STG, STG-) ... 255
Spring Hanger/Support (SPR) ... 259
Rod Hanger (ROD) ... 264
Viscous Damper (DMP) ... 267
Snubber (SNUB) ... 273
Dynamic Limit Stop (DGAP) ... 278
Concentrated Transient Dynamic Load (DFRC) ... 281
Time History output /Travel indicator (TH_OUT) ... 283
Discontinuity stresses (STR_DISC) ... 286

Part IX References 288

Part X Appendix I 288

Part XI Appendix II 289

Part XII Appendix III 293

Part XIII Appendix IV 295

Part XIV Appendix V 297

1 Analysis according to PNAE Code ... 297
2 Analysis according to the Russian Boiler Code (RD) ... 304

Part XV Appendix VI 306

1 Basic definitions ... 306
2 Run of analysis, errors and warnings ... 308

3 Results of analysis ... 314
4 SH.DBS file structure ... 317
5 Selection of springs from LISEGA Catalog ... 318

Part XVI Appendix VII 321

Part XVII Appendix VIII 322

Part XVIII Appendix IX 324

Part XIX Appendix X 325

Part XX Appendix XI 329

1 NC_3600 ... 329
2 NB_3600 ... 332

Part XXI Appendix XII 335

Part XXII Appendix XIII 343

Part XXIII Appendix XIV 345

Part XXIV Appendix XV 346

Part XXV Appendix XVI 348

Part XXVI Appendix XVII 350

Part XXVII Appendix XVIII 354

Part XXVIII Appendix XIX 358

Part XXIX Sample of analysis 369

1 Piping Data ... 370
2 Seismic Load ... 371
3 Input Data and Run of analysis ... 371

1 Abbreviations

Abnorm.	Abnormal Conditions
Fault.	Faulted Conditions (eg. LOCA)
HE	High Energy (pipes)
HELB	High Energy Lines Break
HTEST	Hydraulic Test
LOCA	Loss of Coolant Accident
NOC	Normal Operation Conditions
OBE	Operating Basis Earthquake
PNAE	Rules and Regulations in the Nuclear Power Industry (Russian Nuclear Code)
SSE	Safe Shutdown Earthquake

2 Introduction

dPIPE 5 is Piping Stress and Flexibility Analysis Software Program used to analyze piping and structural systems subjected to static and dynamic loads.

The following types of analyses are performed under the set of specified loads:

static analysis:

- analysis of displacements, support reactions, internal forces and stresses in the piping system under the action of weight loads (distributed and concentrated) as well as internal pressure;
- analysis of displacements, support reactions, internal forces and stresses in the piping system under the action of thermal expansion loads;

dynamic analysis:

- analysis of response displacements, accelerations, support reactions, internal forces and stresses in the piping system under the action of seismic loads specified in the form of response spectra (analysis according to Response Spectrum Method, **RSM**). In order to determine the seismic response of the system, the Independent Support's Motion Analysis has been implemented in the software, i.e. the possibility of setting specific response spectra for various elevations of piping system anchorage to building structures;

- analysis of response displacements, accelerations, support reactions, internal forces and stresses in the piping system under the action of transient dynamic loads specified in the form of accelerograms (seismic loads) or time varying loads concentrated in the nodes of FE model (Time History Analysis, **THA**).

Stress evaluation of piping components is performed according to any of the following piping codes:

For Nuclear Piping:

- Russian Code PNAE G-7-002-86;
- ASME BPVC Code, Section III, Subsections NB, NC, ND
- NTD A.S.I. Sekce III, Příloha A, Hodnocení pevnosti zařízení a potrubí jaderných elektráren typu VVER, 2016 (Czech Code)

For Industrial Piping:

- Russian Boiler Code RD 10-249-98;

- ASME B31.1;
- ASME B31.4;
- EN 13480 "Metallic Industrial Piping"

A specially developed input data entry language is used for describing the piping system geometry and specifying its physical and mechanical characteristics. The software has a comprehensive error diagnostic system, a graphic interface for entering input data and viewing them as well as calculation data.

3 Software structure. File system

The dPIPE 5 software consists of individual program modules performing various analysis process stages. All program modules are interconnected via the common file system. Program modules of the dPIPE 5 software are listed below along with brief description of their purpose.

Designation	Description
DPC.EXE	Preprocessor. Input data entry and verification
LIST_BIN.EXE	Input Data listing generation program
DP5S.EXE	Static analysis module. It generates the Piping Calculation Model stiffness matrix and load vectors, solves the system of equations, defines the internal forces and deformations in elements, support reactions,as well as node displacement
DP5D.EXE	Dynamic analysis module. It calculates Piping Eigen Frequencies and Mode Shapes
POST.EXE	Post-processor. Printout of the results and check of the stresses to the Code compliance
Pipe3DV.EXE	View of Piping Calculation Model and the results in graphical form
WORKPAD.EXE	Text editor for Input Data editing and printout preview
Pipe3DV.CHM	Help file for the Pipe3DV program
DDE.EXE	Spreadsheet for Input Data entry
dPIPE_5.CHM	Help file for the dPIPE 5 software
Runtime libraries and auxiliary programs to the DDE spreadsheet:	
DATALIB.DLL	
DATALIB.NET.DLL	
DATALIB.NET.RESOURCES.DLL	
GYMFCEXT.DLL	
HASP.DLL	
PL.DLL	
PLDRAW.DLL	
REGISTRY.DLL	
RES.DLL	

SERIALIZE.DLL	
SETUP_RES.DLL	
SPAWNER.EXE	
Files containing databases and auxiliary information:	
FMT_PRE.DBS	File with settings for printout of Input Data listing tables (in Russian)
FMT_PST.DBS	File with settings for printout of tables with the results (in Russian)
FMT_PRE_E.DBS	File with settings for printout of Input Data listing tables (in English)
FMT_PST_E.DBS	File with settings for printout of tables with the results (in English)
MAT.DBS	File with the materials and fatigue curve database
SH.DBS	File with the database for the spring hanger characteristics (see also Appendix VI)
DMP.DBS	File with the database for High Viscous Damper (HVD) characteristics
BEAM.DBS	File with the database for beam element cross-section properties
PIPE.DBS[1)]	File with the database for standard pipe and bend properties
SOLV.DBS	File with the set of pre-defined specifications for analysis and post-processing
VLV_OTT.DBS	File containing Database with allowable loads on valve's nozzles in accordance with General Technical Requirements NP-068-05, *[REF 14]*
SUP_LDS.MDB (located in DB folder)	File with allowable loads on piping supports (in Microsoft Access format), see. Appendix XII for description
DP.BAT	Run complete analysis in batch mode
SOLV.BAT	Run solution module in batch mode
CLEAR.BAT	Clearing the working directory
UTILS folder:	
R2DP_N.EXE, R90_2_DP5.EXE, R93_2_DP5.EXE[2)]	Program for Input Data conversion from RAMPA format into dPIPE 5 format
DP2DP5.EXE[3)]	Program for Input Data conversion from the previous versions of dPIPE formats into dPIPE 5 format
MAT_DP_OLD.DBS	File with the database for materials (for previous versions of dPIPE)
SH.DBS	File with the database for spring hanger characteristics (for previous versions of dPIPE)
DMP.DBS	File with the database for HVD characteristics
EN-US and RU-RU folders contain runtime libraries and help files for the English and Russian program interfaces respectively	

1) If the content of the database depends on the CODE used for analysis, this data should be preceded by ***$SET CODE*** instruction. File pipe.dbs could also contain the instruction ***DOC*** for the certain document. This information is used for the following filtering of the data. Sample: $SET CODE ='PNAE' DOC="OST24 AUSTENIT"

2) For the conversion data from the RAMPA program into dPIPE 5, it is possible to use the icon located in the right corner of the upper toolbar. In doing so, the RAMPA-93 and RAMPA-90 files with Input Data should have the extension ".dat" and ".nml" respectively;

3) For the conversion data from the previous versions of dPIPE into dPIPE 5, it is possible to use the icon located in the right corner of the upper toolbar. In doing so, the files with Input Data corresponding to the previous versions of dPIPE should have the extension ".dat".

The following table contains a description of the file system to be used in the dPIPE 5 software. The file type is indicated in the "Note" column. ASCII corresponds to text files; BIN corresponds to internal binary files.

File name	Description	Originator	Use	Note
NAME.DP5	Input data (piping system geometry, properties of materials, specification for analysis and post-processing of the results)	User, DDE, conversion programs	DPC, DDE	ASCII
NAME.BIN	File with the database for Piping Calculation Model and the results	DPC	DP5S, DP5D, LIST_BIN, POST, PIPE3DV	BIN
NAME.BIN2	Intermediate file with the static analysis results	DP5S	POST, DP5D	BIN
NAME.BIN3	Intermediate file with the dynamic analysis results	DP5D	POST	BIN
NAME.OUT 1)	Input data printout	DP_LIST	User	ASCII
NAME.RES 1)	Printout of the results	POST	User	ASCII
NAME.SUP 1)	Summary tables of loads on supports, equipment and valves	POST	User	ASCII
NAME.INF	Summary tables of loads (in raw format) on supports for all load cases	POST	User	ASCII

1)Content of files with input data listing (*.OUT) and analysis results (*.RES and *.SUP) may be adjusted to specific formats for numbers and text in Tables headers/footers through the use of files-templates: pre_fmt.dbs for input listing and pst_frmt.dbs for analysis results. Default locations of these files are “ru-RU” and “en-US” folders (depending on interface language). Location could be changed either through the interface Options/Reports/Templates or by command DBF.

4 PC configuration requirements

The following PC configuration is needed for running the dPIPE 5 software:

- Operating system: Windows 7or higher;
- .NET Framework software development and execution environment, Version 4.7.2 or higher

- Graphics Card with OPEN GL and DirectX support

5 Software installation

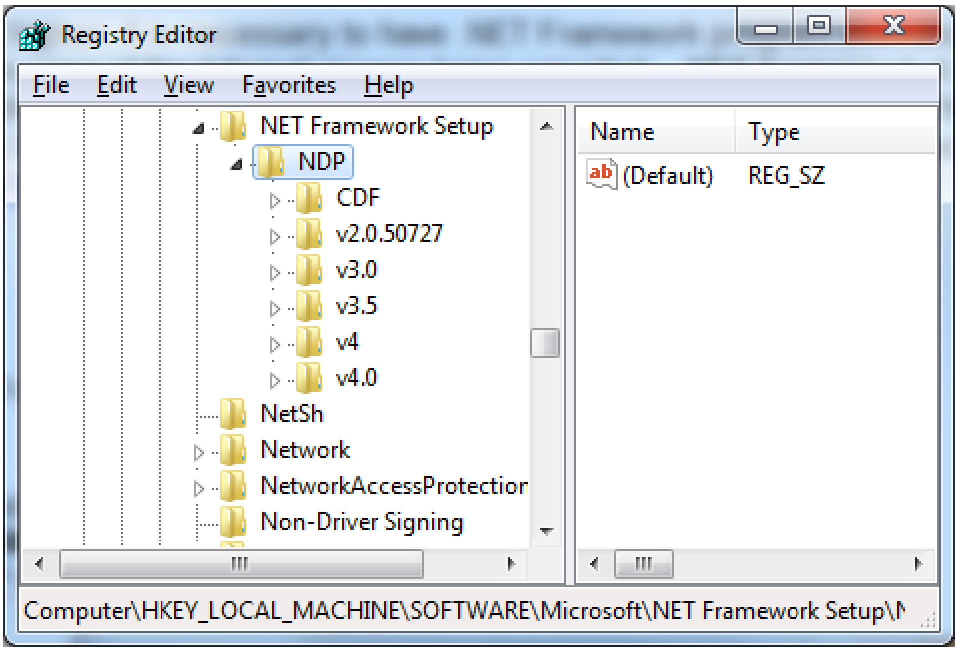
In order to install the dPIPE 5 software, it is necessary to run the installation program DP5_XXSETUP.EXE (XX - is the actual number of dPIPE Release) and then follow the installation program instructions.

For the proper work of dPIPE, version 5.23 and higher, it is necessary to have .NET Framework program execution environment, Version 4 or a later version[1)]. if this software is not installed on the computer, then, in case of the Internet access being provided, .NET Framework 4 will be installed automatically from the Microsoft.com web site during installation of the dPIPE software. If the Internet access is not provided, then before proceeding to dPIPE installation, it is necessary to install .NET Framework 4 independently. .NET Framework 4 software is free. The installation program can be downloaded from the following web site address: http://www.microsoft.colnm/ru-ru/download/details.aspx?id=24872.

Note:

1) You can find out the actual version of .NET Framework installed on your computer as follows:

1. Open the Start menu, select Run.
2. In the text field, Open enter regedit.exe.
3. In the Register Editor, open the following subsection:
   HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\NET Framework Setup\NDP
   The versions installed are listed in the NDP subsection:



6 Protection keys and License Management

dPIPE employs Software Licensing and Protection system SENTINEL™. The program can work with hardware keys HL (inserted into the computer's USB port), as well as with software SL keys. Type of Protection key may be a Single User License or Network License. Depending on the type of key different procedures for installation/upgrade are described below.

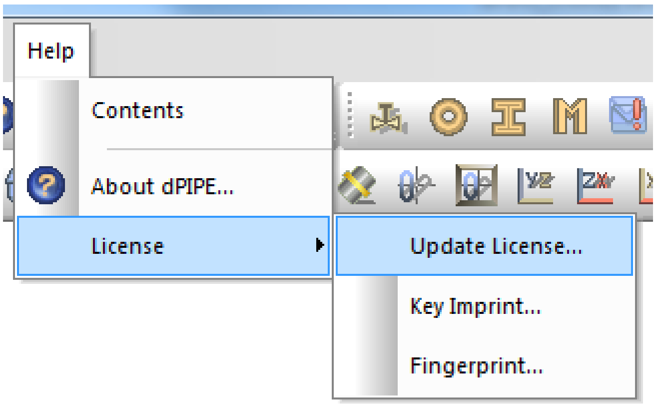
To the attention of system administrators:

After installing the license manager you can get information and access to management to the Sentinel keys that are currently present on the network, including the locally connected Sentinel keys with a help of the Sentinel Admin Control Center (http://localhost:1947).

User hardware Key HL
Network Hardware Key HL-NET
User Software Key SL-UserMode
Network Software Key SL-AdminMode
dPIPE in Sub-Network
Sentinel SL License Transfer between two PCs

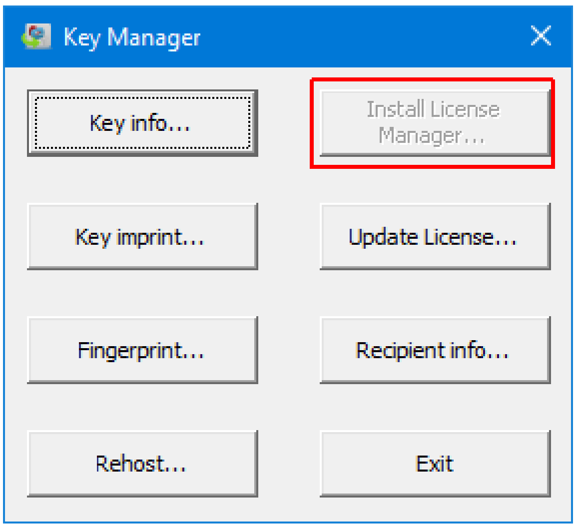
User hardware Key Sentinel HL Pro DL

User hardware Key Sentinel HL Pro DL: no special installation procedure is required. The Key is supplied to the User with an embedded License. To update the License file "*.v2c" is sent from Developer to the User .This file should be pointed for the command "Update License":



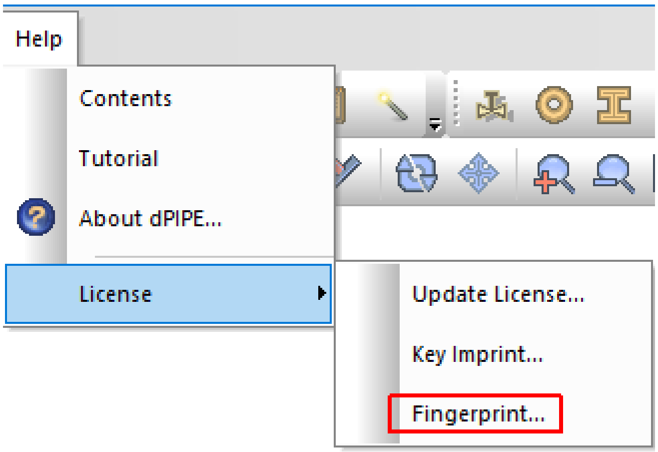
Network Hardware Key HL-NET

Network Hardware Key HL-NET. This Key is supplied to the User with an predefined number of the Licenses. The license manager can be installed using the KeyMan program, which is located in the Sentinel sub-folder under the dPIPE installation root directory. The Sentinel sub-folder is created if the "Server utilities" option was selected when installing the program. Server Utilities is a stand-alone option and can be installed separately from dPIPE, for example on a server.



User Software Key SL-UserMode

User Software Key SL-UserMode. Use “Help-License-Fingerprint” command to generate “Client to Vendor” file *.c2v. This file should be sent to the Developer. File *.v2c obtained in response allows activate license through “Help-License-Update License” command.

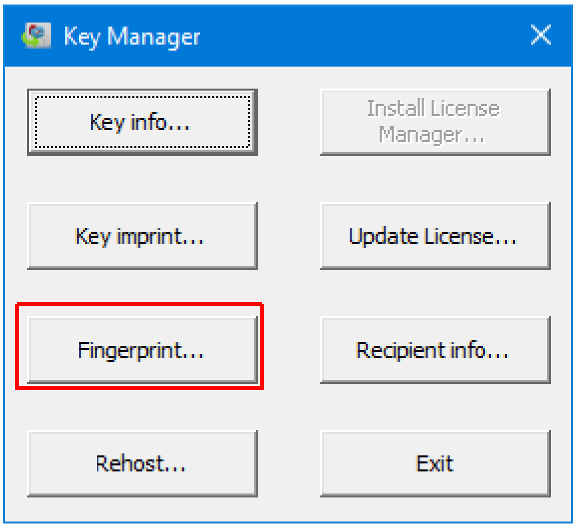


In response, the Developer sends a * .v2c file, which allows you to activate the license through the item "Update license". When changing/updating a license, the procedure is the same as for a hardware key.

Attention: before performing any operations related to the upgrade of operating systems or hardware on computers/servers on which user or network software licenses are installed, these licenses should be transferred to another computer, see chapter "Sentinel SL License Transfer between two PCs"

Network Software Key SL-AdminMode

Network Software Key SL-AdminMode. License Manager should be installed on the Server. Then use Key Manager software and run command “Fingerprint”:



Generated *.c2V file should be sent to the Developer. File *.v2c obtained in response allows activate license through “Update License” command of Key Manager.

Attention: before performing any operations related to the upgrade of operating systems or hardware on computers/servers on which user or network software licenses are installed, these licenses should be transferred to another computer, see chapter "Sentinel SL License Transfer between two PCs"

dPIPE in Sub-Network

Work with dPIPE in Sub-Network:

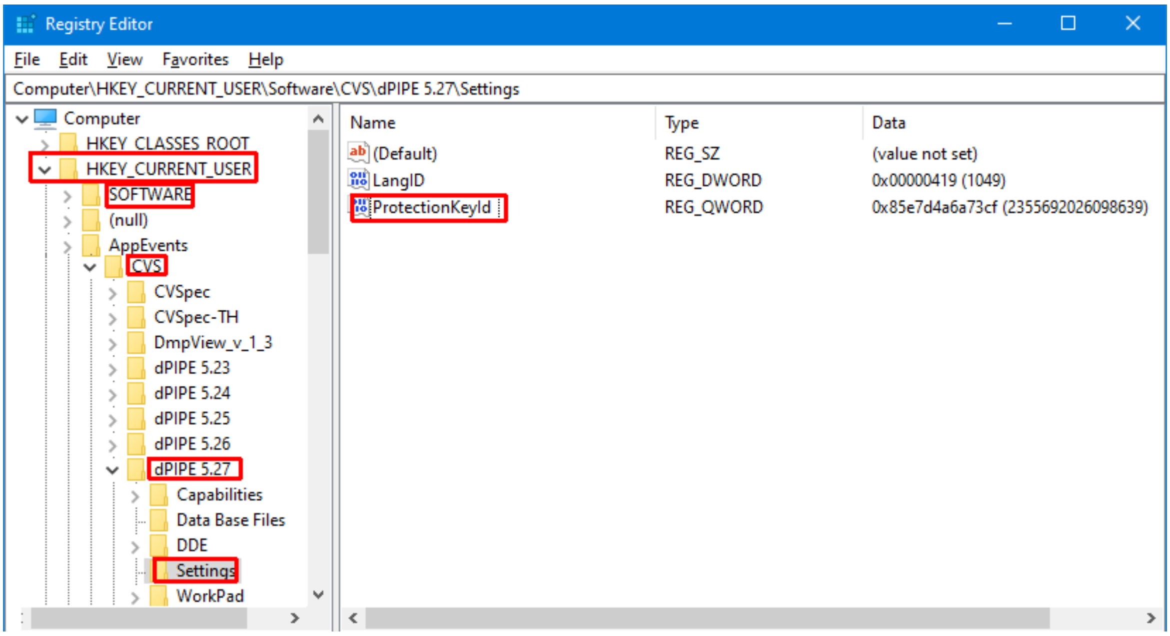
To use SENTINEL keys across subnets one has to create a text file «hasp_82556.ini» with the following content:

[NETWORK] ***; Section name, mandatory string***
BROADCASTSEARCH = 0 ***; switching off broadcast search of the key***
SERVERADDR = <Server Name> ***; The IP address or name of the server where the license manager is installed***
DISABLE_IPV6 = 1 ***; switching off the use of IPv6***

This file should be placed on the Client PC in the following folder:

Type of application	Directory
Desktop (Windows Vista/7 or later)	%LocalAppData%/SafeNet Sentinel/Sentinel LDK/
Desktop (Windows XP)	%UserProfile%/Local Settings/Application Data/SafeNet Sentinel/Sentinel LDK/
Service (LocalSystem) x64 operating system	%windir%\SysWOW64\config\systemprofile\AppData\Local\SafeNet Sentinel\Sentinel LDK\
Service (LocalSystem) x86 operating system	%windir%\System32\config\systemprofile\AppData\Local\SafeNet Sentinel\Sentinel LDK\
Service (Network)	%windir%\ServiceProfiles\NetworkService\AppData\Local\SafeNet Sentinel\Sentinel LDK\

To connect a specific application with certain protection key, one has to enter in the Register the record ProtectionKeyId (QWORD type) with the value equal to the number of KEY. Address of the Registry Entry is:
HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\CVS**ApplicationName**\Settings, where ApplicationName depends on the addressed application: “dPIPE 5.XX” (5.XX - version number, i.e. 5.27), “Tcalc”, «CVSpec-TH», "G-FRC 2.0":



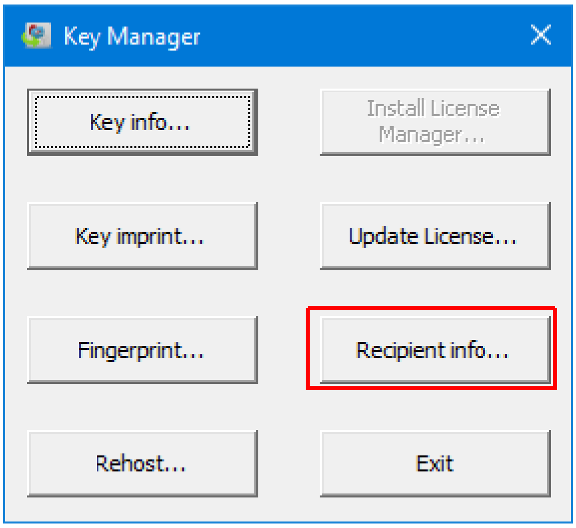
Sentinel SL License Transfer between two PCs

Sentinel SL License Transfer between two PCs

The software license key may be transferred between two PCs if the corresponding option is activated. If the option is not activated, then before transferring it is necessary to activate it by updating the key. Let's denote **S**(Source) the computer from which the key is transferred, and **R**(Recipient) the computer to which the key is transferred.

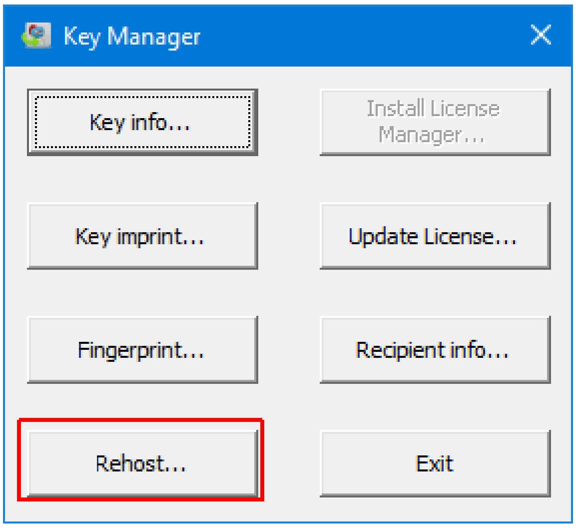
The procedure:

1. Launch the KeyMan program on computer R. By clicking the "Recipient Info" button, you will get a file with information about computer R:



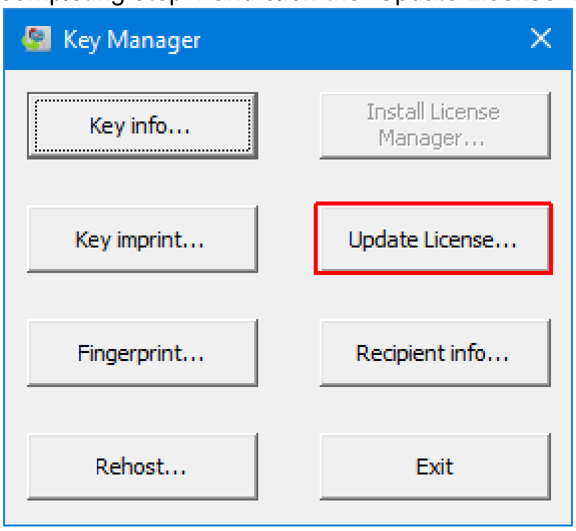
If Sentinel License Manager is not installed on the computer, the program will prompt you to install it, since if you transfer the network key, it must be installed before receiving the information file.

2. The file received in step 1 should be transferred to computer **S**. Run the KeyMan program on PC “**S**” and click the “Rehost” button

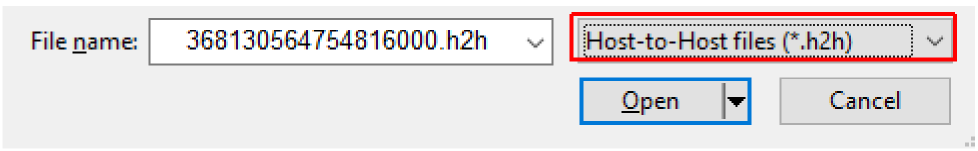


If the computer does not have keys that can be transferred, a message will be displayed. If there are several keys, the program will prompt you to select one from the list. In the dialog that appears, open the file transferred from the “R” computer. A file will be created to transfer the key with the name “KeyNumber.h2h” and the key will be deleted from computer S. The file must be saved using the dialog that appears.

3. Copy “KeyNumber.h2h” file to computer R. Run the KeyMan program if it was closed after completing step 1 and click the “Update License” button.



In the appeared dialog select the file type “Host-to-Host” and open the file. If successful, a corresponding message or error message will be displayed if it occurred during the transfer process.



IMPORTANT!

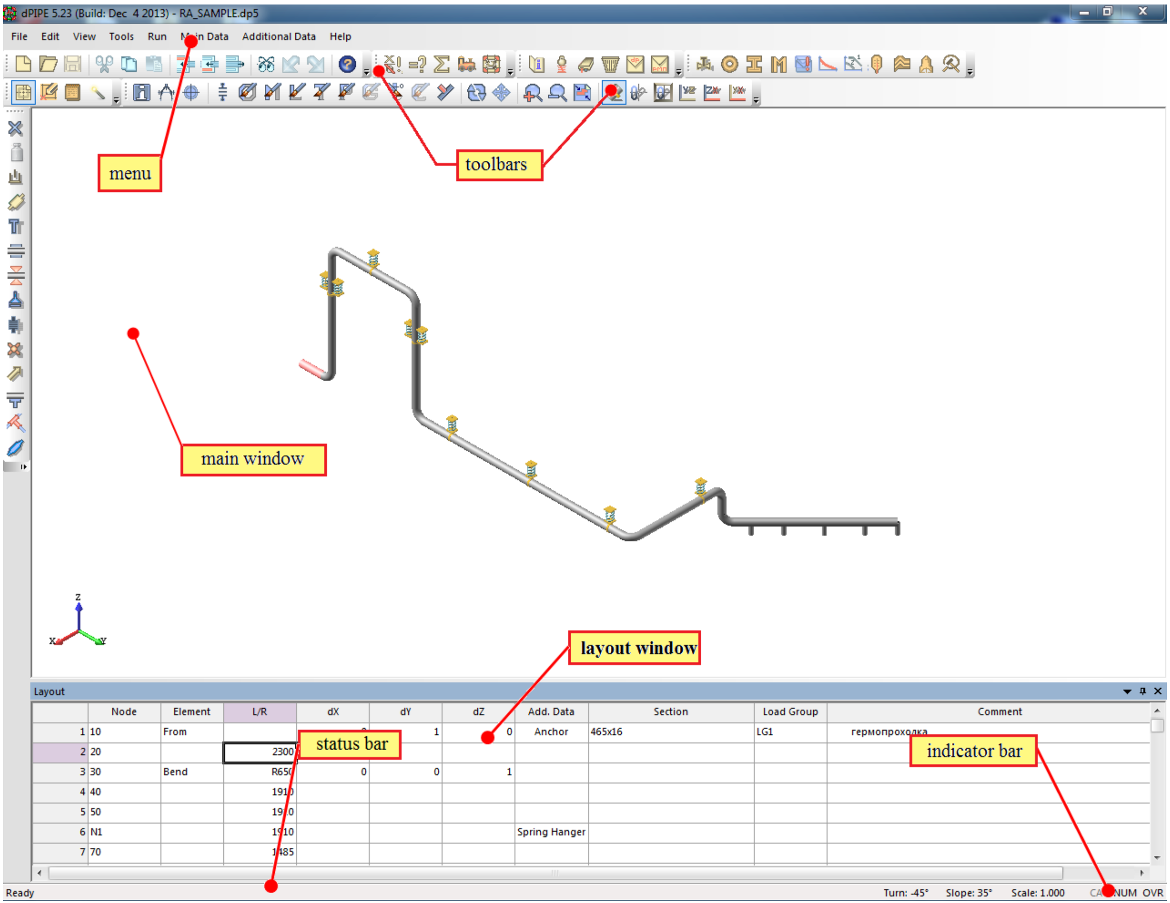
Before performing any actions to transfer the key, you must make sure that the same version of the License Manager is installed on both PCs and, if necessary, update the driver. This can be done using the KeyMan program included in the dPIPE distribution (Sentinel folder).

During the process of borrowing licenses, starting from the creation of the ID file on the acceptor and until the moment the license is applied from H2H, it is highly recommended not to update or reinstall the drivers on both PCs.

7 DDE spreadsheet

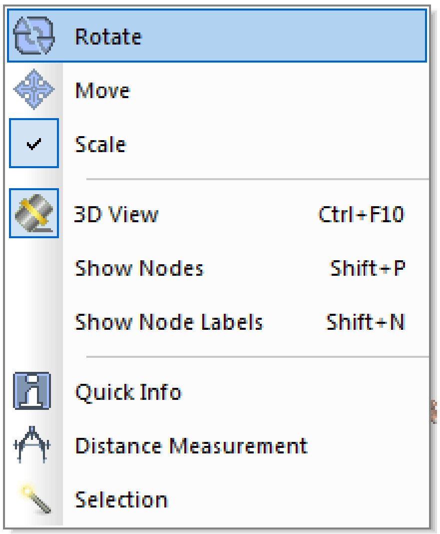
DDE spreadsheet serves as the interface between entering the Input Data, running the analysis and viewing the results of the dPIPE 5 software. Input Data can be entered in two modes: *active mode* – with dynamic tracing and displaying the piping system being composed and *passive mode* – without tracing and input syntax check.

Spreadsheet interface



The ***main window*** of the program displays either the graphic model the piping system, or the model description in the form of a set of commands of the dPIPE software [input data entry language](#).

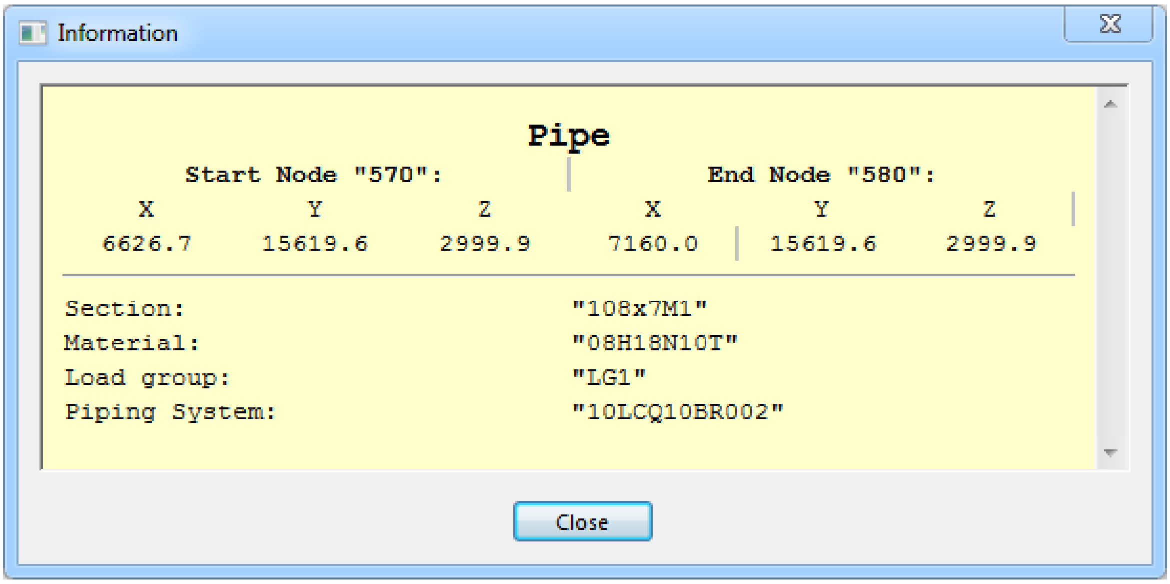
In case of right click in the main graphic window area, the ***context menu*** appears, which allows to execute the following set of operations:



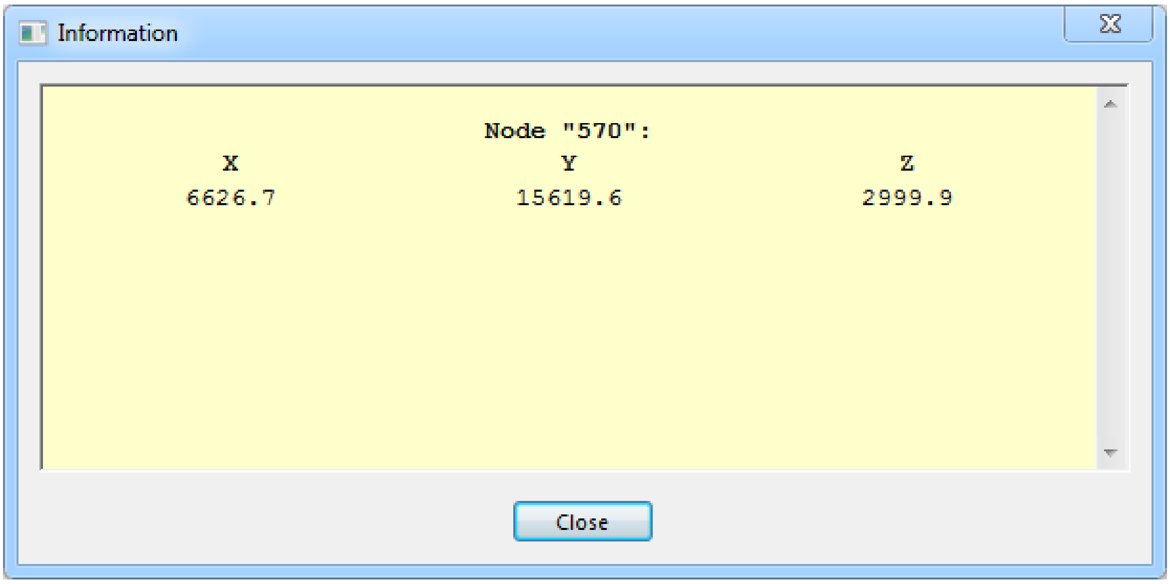
Rotate (): rotation of the model by means of the mouse with the left button being pressed;
Move (): movement of the model over the screen by means of the mouse with the left button being pressed;
Scale: mode for selecting a window by the mouse in order to zoom in (select from top downward) or zoom out (select from bottom upward) the model part being viewed. Zooming can also be performed by means of the mouse wheel. In addition, the following icons can be used for zooming:  - zoom in with centering,  - zoom out with centering (centering is carried out with respect to the part of the screen, at which the left click is performed);
3D View (): toggle between 3D Rendered View and non rendered view (CTRL-F10);
Show nodes: show node's symbols (Shft-P);
Show Node Labels: toggle to show/hide node's labels (Shft-N);

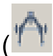
Quick Info (): mode, at which a click on the highlighted element/node calls the information window:

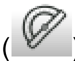
Information about an element:



Information about a node:



Distance Measurement (): mode allowing to measure the distance between any two nodes of the model. *Hold «CTRL» key for temporary switching between this or "quick info" mode to navigation*

Angle measurement (): measuring of the angle between two straight elements

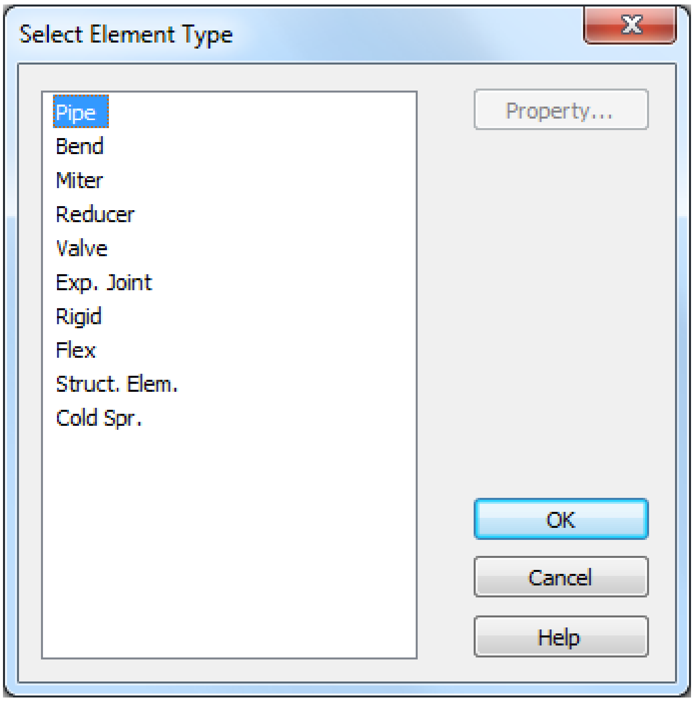
Measuring angles () between the centerline of a straight element and its projections on the X0Y, Y0Z, X0Z planes

Selection ():mode allowing to select a part of the model for subsequent operations to copy, paste, modify and delete the piping segments selected (see "Copy, paste, modify and delete piping segments").

Layout input window containing the table for entering Input Data has the following fields:

Node – identification name of the model node;

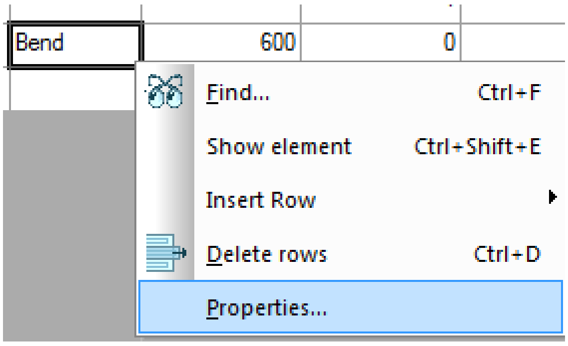
Element – type of the element; the element type selection is carried out either by double click in the corresponding cell, or by pressing F2 key:



The element types in the menu are connected with dPIPE commands as follows:

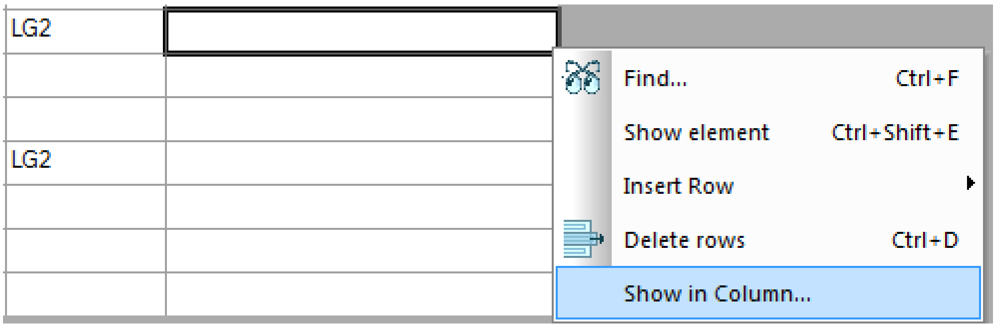
Menu	Description	Commands
Arc	command for bend breakdown	BEND (2)
Bend	bend-1	BEND (1)
Cold Spr.	cold spring	CS
Exp. Joint	expansion joint	EJ, EA, ET, EH, EG
Flex	flexible element	FJ
From	beginning of branch	FROM
Miter	sector elbow	MTR
Pipe	straight pipe	PIPE
Reducer	reducer	REDU
Rigid	rigid link	RX, RP
Struct. Elem.	beam	S
Valve	valve	VALV, V1, V2

Additional editing of the element's properties is carried out from the context menu with selection of the "Properties" item:

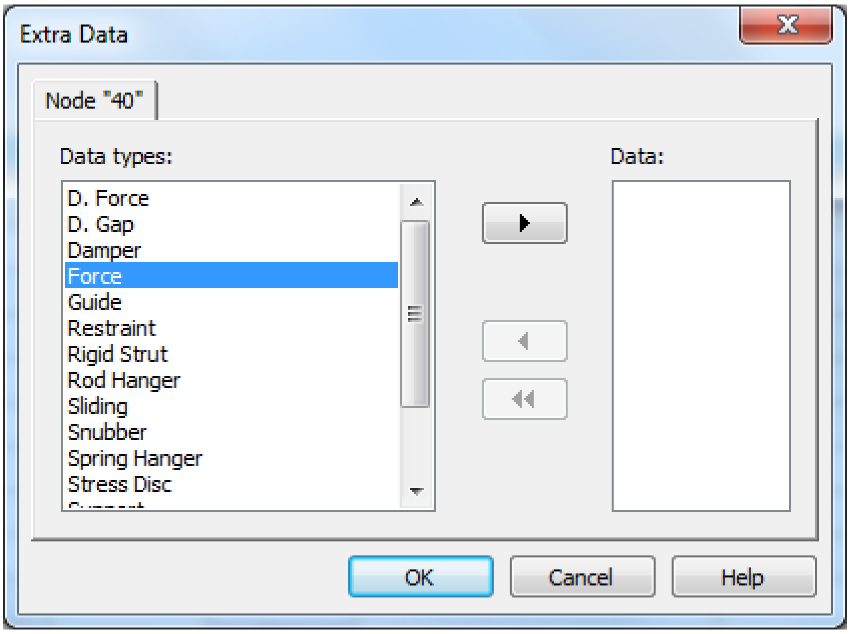


L/R – length of element/bend radius;

dX, dY, dZ (Fi, Teta, R) – direction of the element (to be specified in the Cartesian coordinate system or in the spherical coordinate system). Toggle between these modes is performed either by double clicking on the corresponding table headings or from the context menu (right click) with selection of the"Show in Column..." item:



Additional data – data related to the model nodes (supports, concentrated parameters, tees, etc.). The data type selection is performed either by the click in the corresponding cell, or by pressing ***F2*** key:

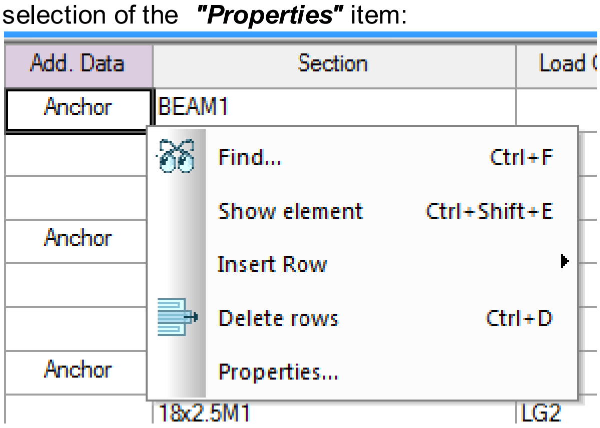


The types of additional data in the dialogue appeared are connected with the dPIPE commands as follows:

Dialogue	Description	Commands

Anchor	anchor support	ANC
D. Force	nodal dynamic force	DFRC
D. Gap	dynamic stop with gap	DGAP
Damper	damper	DMP
Force	nodal static loads	FOR
Guide	guide support	STG, STG-
Restraint	unidirectional transverse or rotational support	STS, SRS, STS+/-
Rigid Strut	rigid strut	STRT
Rod Hanger	rigid hanger	ROD
Sliding	sliding support	STZ,STZ-
Snubber	dynamic snubber	SNUB
Spring Hanger	spring hanger/support	SPR
Stress Disc.	additional temperature stresses	STR_DISC
Support	6-component support	SUP
TEE	tee	TEE
THA Out	output of the transient response/displacement indicator	TH_OUT
Weight	nodal weight	CW
Weld	weld	WLD

Editing the characteristics of the additional data is carried out either from the context menu with selection of the ***"Properties"*** item:



or by double click in the additional data window.

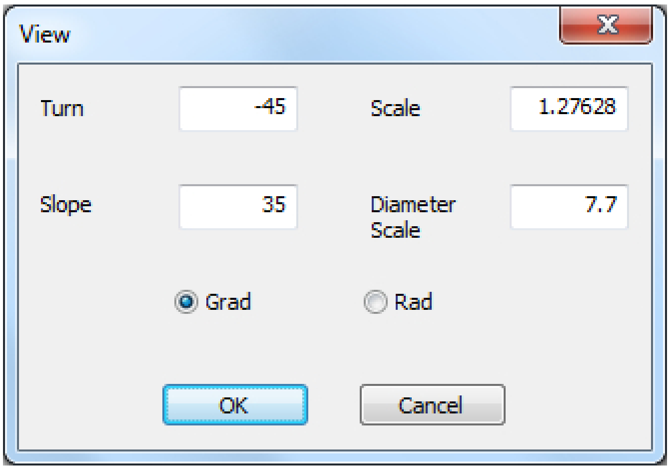
Section – name of the piping system cross-section (PIPE command) or name of the cross-section for the beam element (BEAM command); the selection of the cross-section is performed by pressing on ***F2*** key in the corresponding cell or by double left click.

Load Group – name of the load group of the piping segments; the selection of the piping system group is carried out by pressing on ***F2*** key in the corresponding cell or by double left click (see also OPVAL command).

Comment/Name – user's comments (any data following the "semicolon" character in the Input Data file line. In addition, this column serves for determination of the identification name of the piping segment. To toggle between the display modes is performed similar to the element direction.

Status bar contains information relating to the current line on possible errors during the entry of input data.

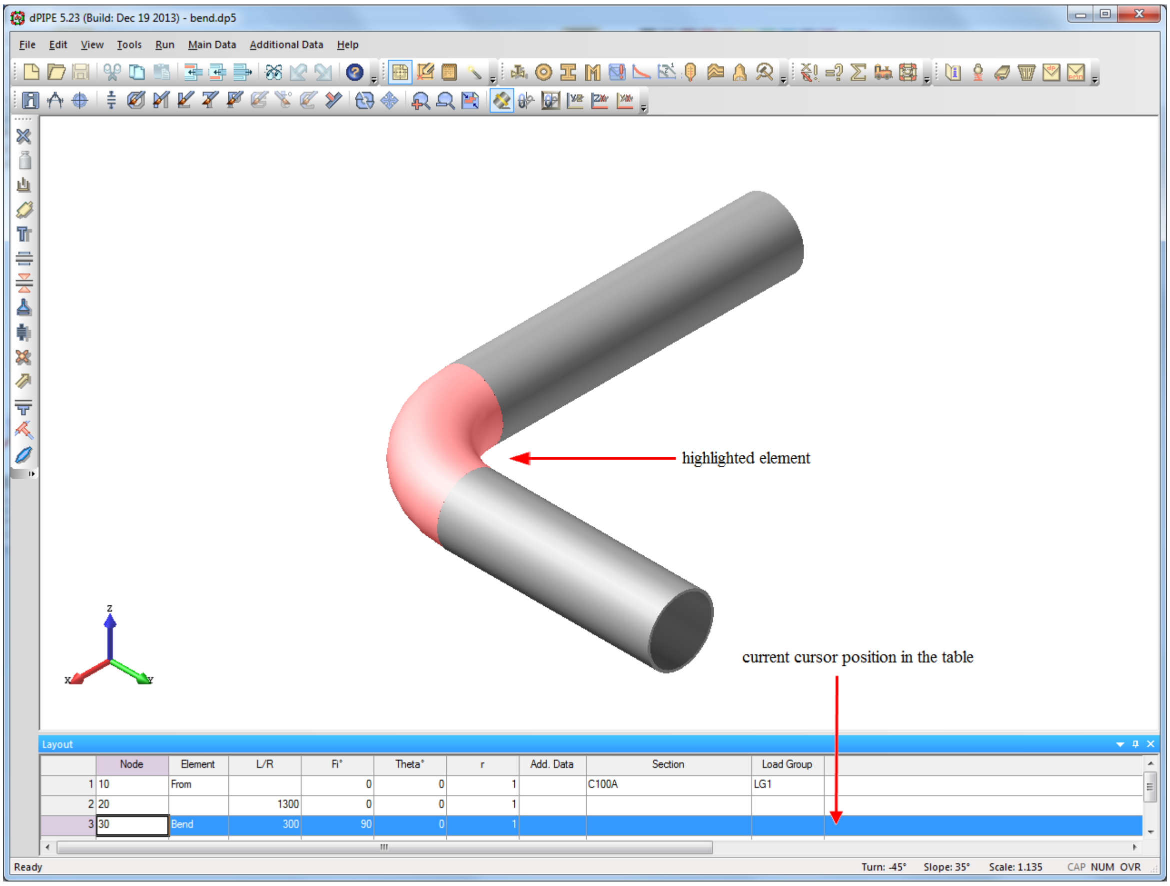
Indicator bar displays the status of keyboard keys: capital letters (CAPS), insert mode (INS), numbers input mode (NUM), as well as the parameters of the current view in the main window. By double clicking on the bar a dialog appears that provides to change/specify the desired settings + change the relative scale of the piping diameter (option is useful when displaying a long lines of small bore tubes; shortcuts: CTRL + GREY "+" / CTRL + GREY "-"):



Spreadsheet navigation

The connection between the main program window and the layout input window is performed in the "active" mode as follows:

1. During navigation over the spreadsheet lines the corresponding elements are highlighted and blink:

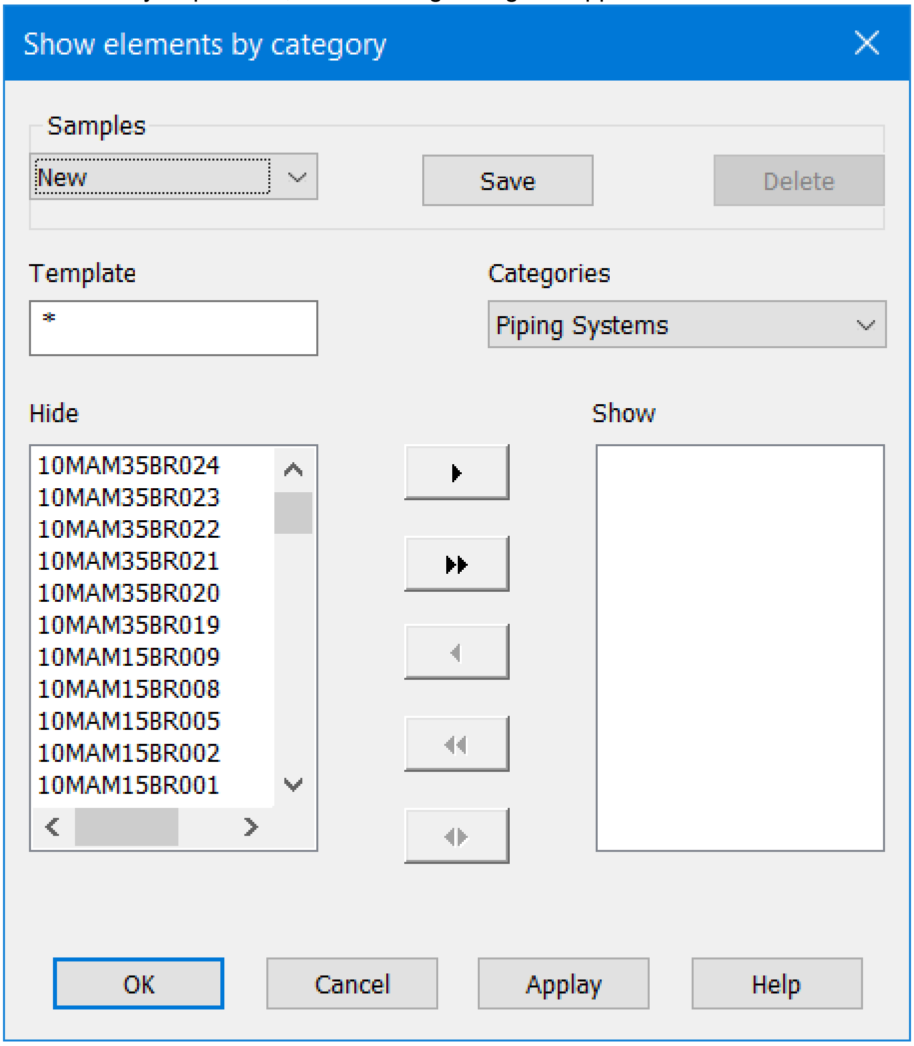


The blink rate could be adjusted on the "display options" page, in the field "Current element blink rate".

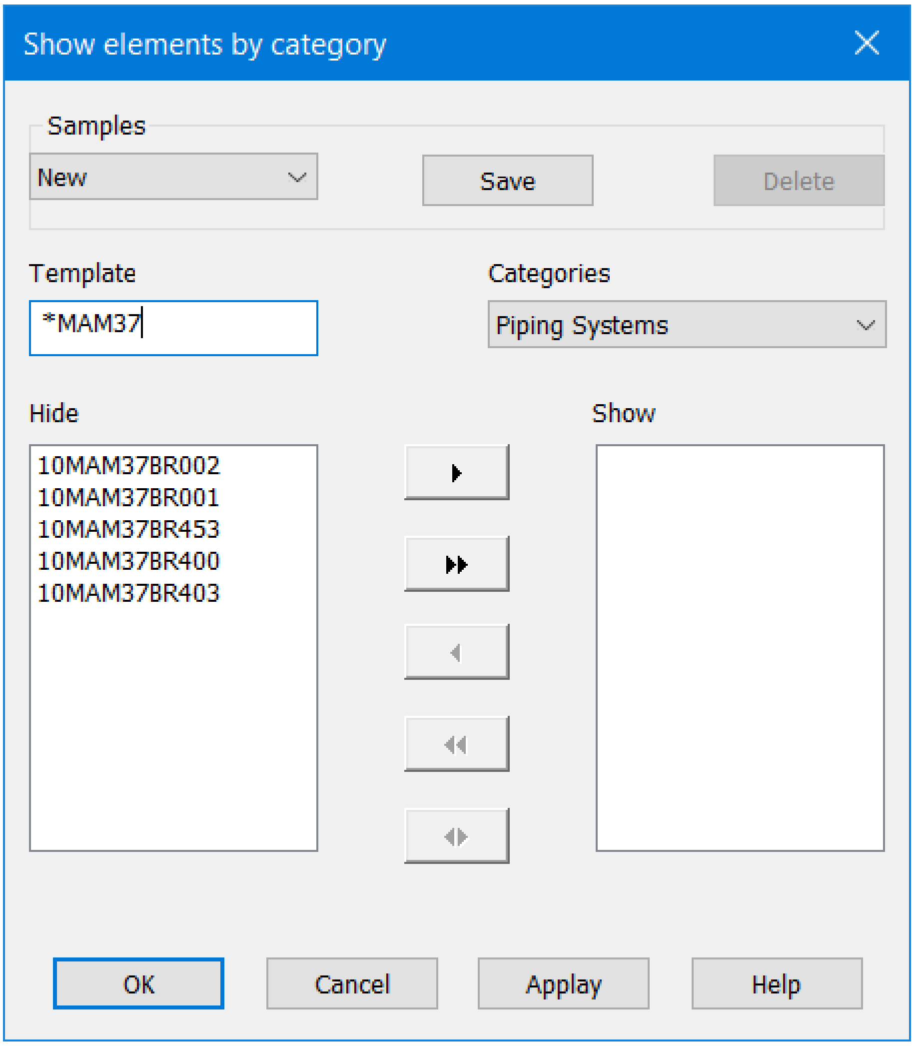
2. If only a part of the model is shown on the screen, then during navigation over the spreadsheet lines the synchronization occurs between the current position of the cursor in the spreadsheet and the model area being displayed. This option can be disabled by removing the corresponding check mark on the "display options" page (flag "Auto position on current element");
3. By simultaneously pressing the "Shift" key and double-clicking the left mouse button on an element in the graphic window, there will be either shifting from the graphic window to the corresponding row of the table, or calling up the corresponding dialog of the properties of the element or support. A convenient key combination could be set up in the "display options" window (group box "*Shift + double-click on the element*").

Partial visualization of the model

When F4 key is pressed, the following dialog will appear:

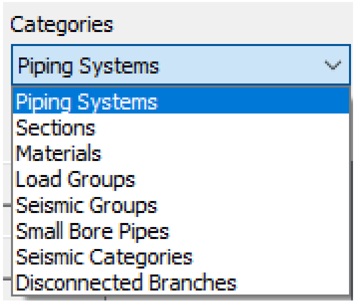


The "Filter" field is designed for displaying some elements out of the whole list according to the mask specified. For example, if *MAM37 mask is entered, then the dialog will get the following view:

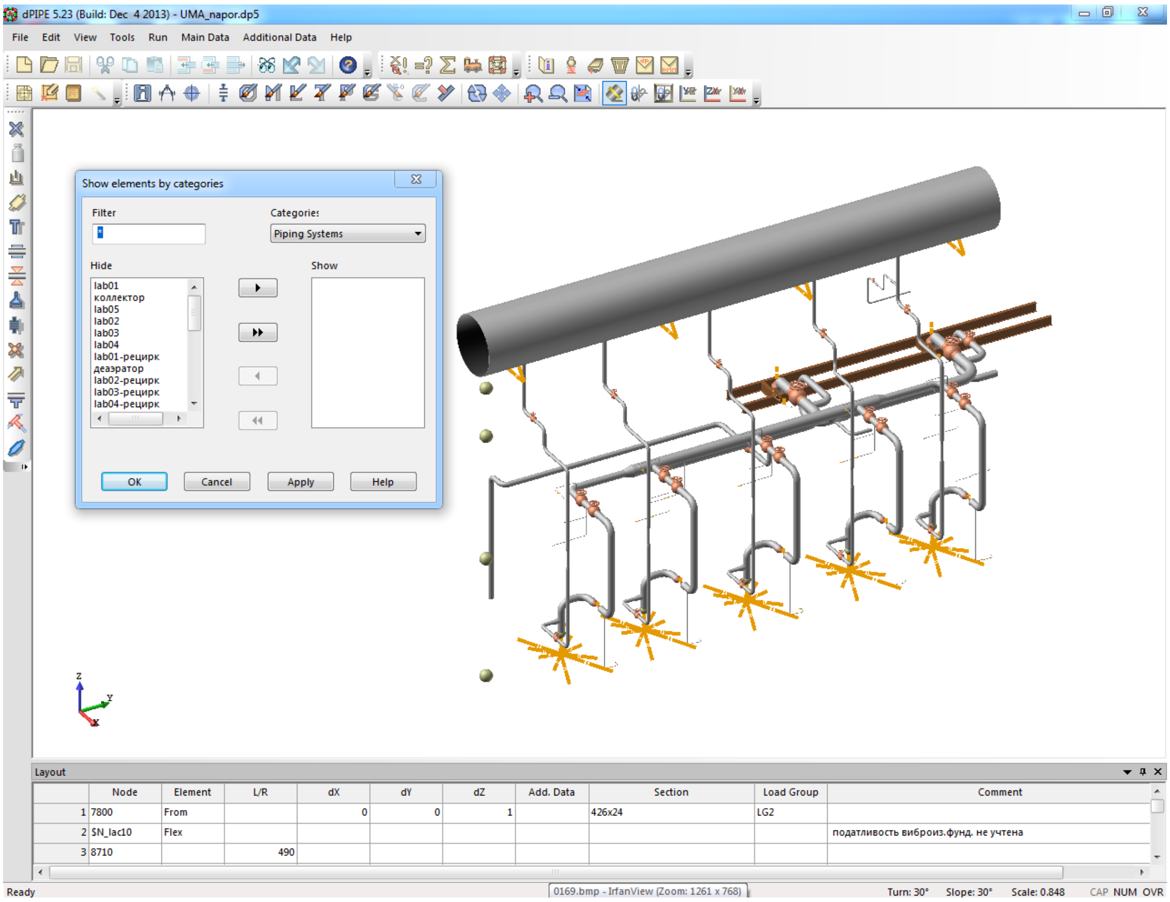


The "Categories" list contains typical features for grouping the piping segments:

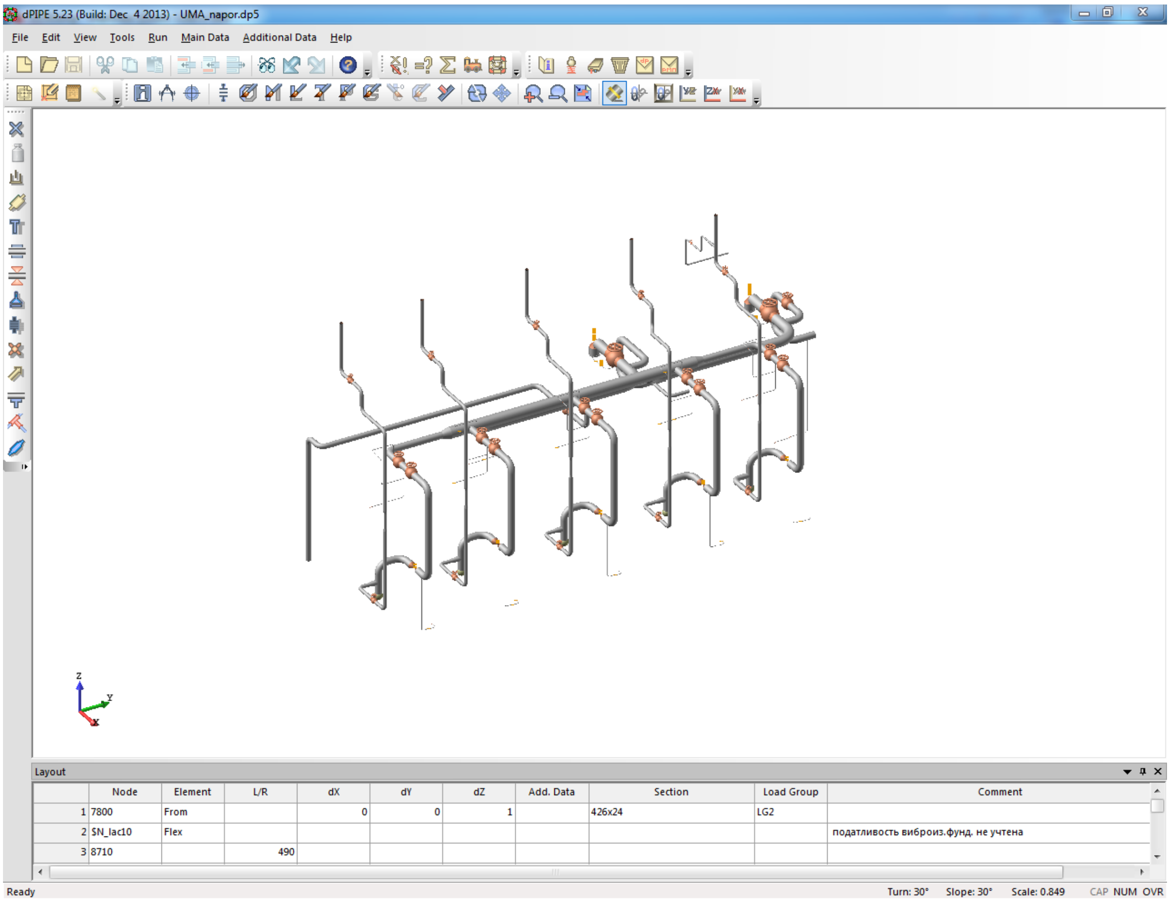
- by piping segment name (name parameter);
- by cross-sections;
- by materials;
- by load groups;
- by seismic support groups;
- by branches not connected by a common node or parameter CNODE:



Groups of the elements belonging to some category are displayed in the left or right part of the dialog ("Hide" and "Show" fields). Depending upon the groups of elements selected the model can be displayed partially:



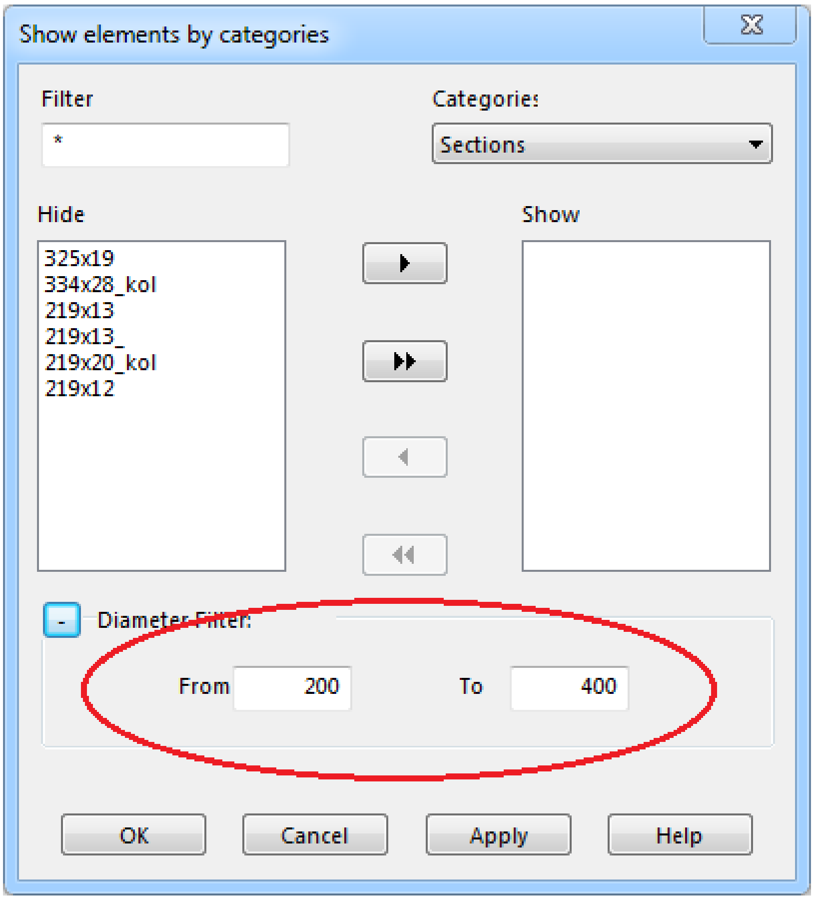
before selection



after selection

The toggle between the partial and the whole model is performed by pressing the SHFT-F4 key combination.

For "***Cross-sections***" category, apart from the filter adjusted to the cross-section names, it is possible to use a filter by the value of outside piping diameter :



The selected part of the calculation model can be saved under a custom name and accessed both during the current session and in subsequent sessions of working with the model. Information about the selected sections is added to the file *.dp5 after the command END_OF_DATA.

Watch Video

Copy, paste, modify and delete piping segments

For operations with a portion of the model, it is necessary to select the required segment by means of the tool  located on the "Piping" toolbar, which is also accessible from the mouse context menu. It can be done by means of computer mouse, or by selecting the segment by frame, or by left click on the required element. During the selection operation the following rules are valid:

- single left click - only one elements will be marked. With repeated click the selection will be canceled;

- left click with simultaneous holding the ***SHFT*** key - the whole branch will be marked;
- when the ***CTRL*** key is held, the elements will be added/excluded from the set that was marked before.

The selection operations can be interrupted for any other operations related to spreadsheet navigation and resumed after repeated pressing the  key.

The following operations can be performed with the model part selected:

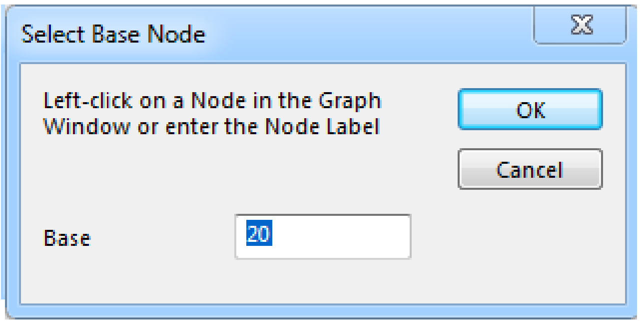
➢ Modify:
  - name of the segment;
  - cross-section;
  - material;
  - load group;
  - Safety Class;

➢ Delete the elements selected as spreadsheet lines (at that the program will interpret such operation as the deletion of lines in the text file, i.e. if the beginning of the branch line falls into the sample, then the current branch line will "adhere"to the previous one, which involves modification of the whole model geometry)

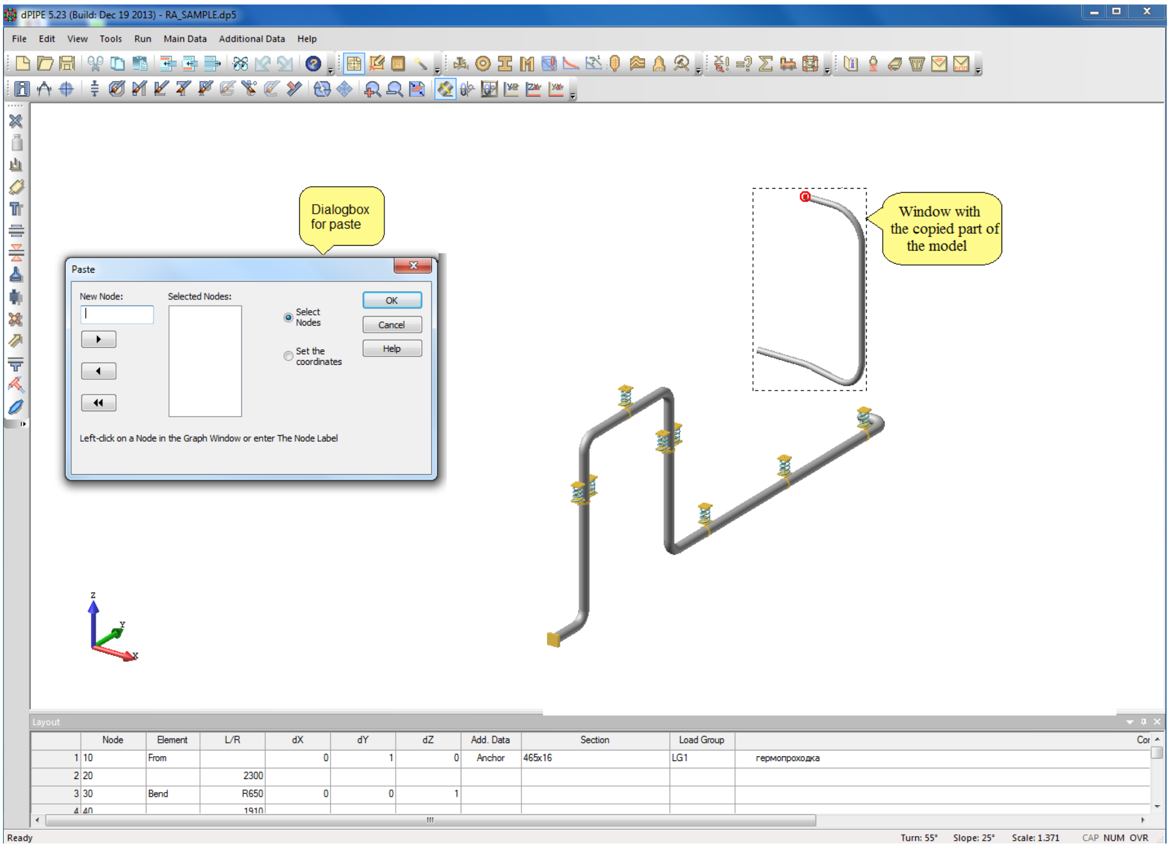
➢ Delete the elements selected and fix the model; in doing so, the whole geometry remains unchanged
➢ Renumber the nodes of the model part selected
➢ Turn the model part about the global Z axis
➢ Mirror the model part selected with respect to the XZ or YZ plane

This list of operations is accessible either from the mouse context menu or via the "Edit/Modify" menu item

The model part selected can be copied for subsequent pasting either into the current model or into any other model. Copying is performed by means of the ***CTRL-C*** key combination or from the "Edit/Copy" menu. When copying it is necessary to specify the base node, in respect of which the subsequent paste of the segment:

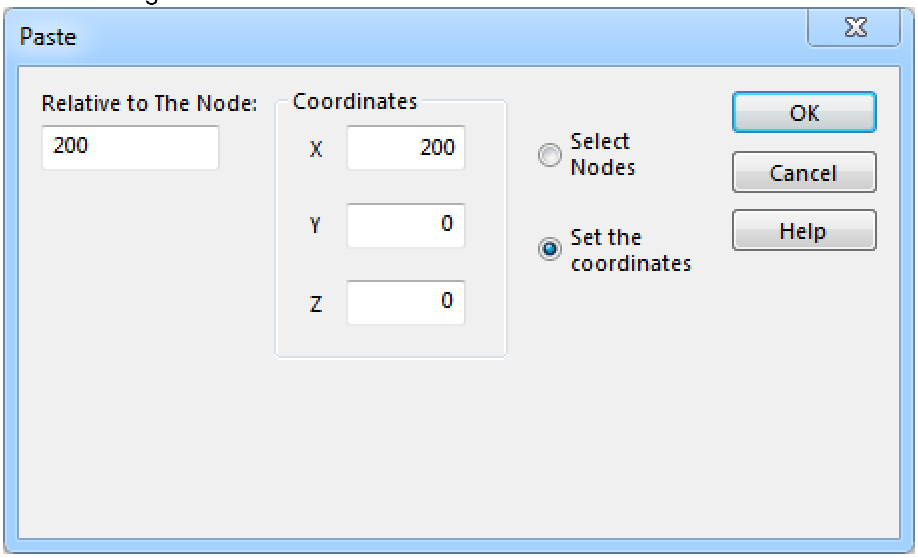


The segment copied can be pasted by means of the CTRL-V key combination or from the Edit/Paste menu. In doing so, a window with the image of the model part copied and the paste dialog box will appear :



There are several methods for pasting the segment copied:

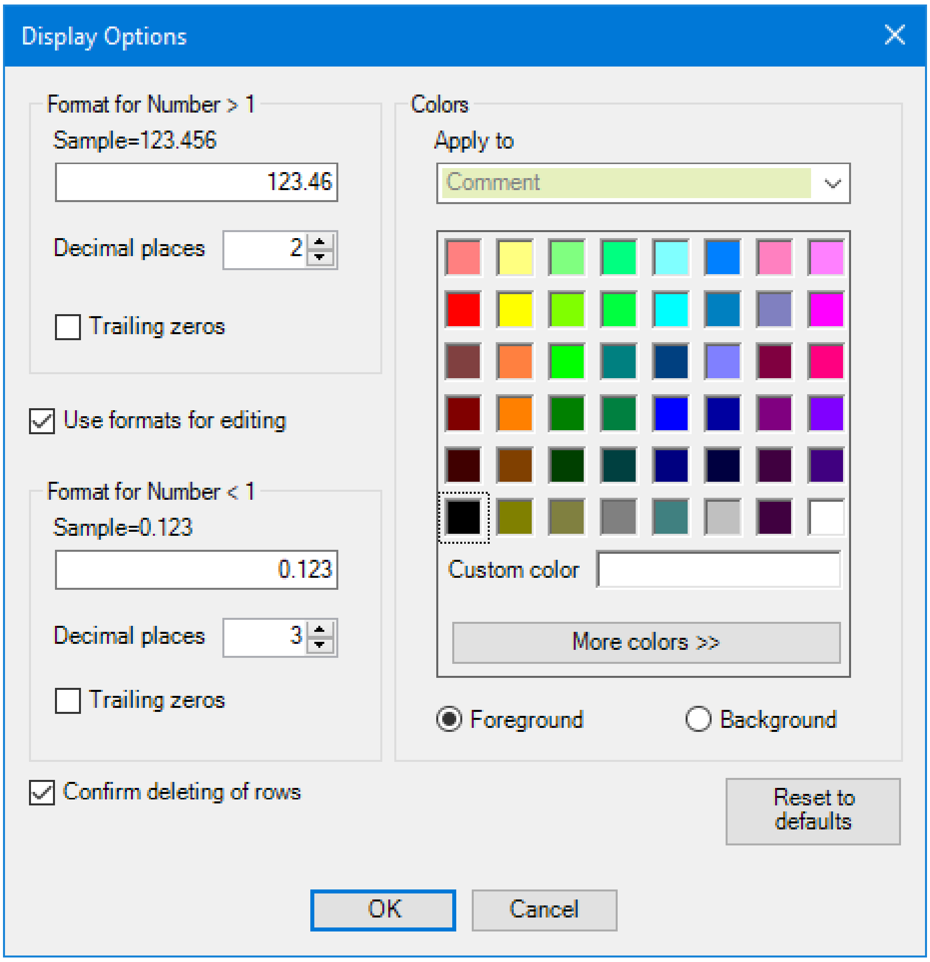
- paste the segment into the existing node (nodes) of the model; to do this it is necessary to perform left click on the required node or type the node label in the "New Node" field. In order to enter several nodes it is necessary to press and hold the ***CTRL*** key;
- paste the section using either the absolute coordinates for the base point or the coordinates with respect to the existing node of the model:



It should be noted that only those nodal additional data that are displayed at the moment of copying will fall into the segment copied. In doing so, the seismic groups of supports and the data related to the piping system operating modes (for example, pre-defined displacements of supports and concentrated loads) will not be copied.

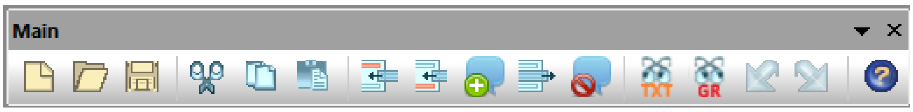
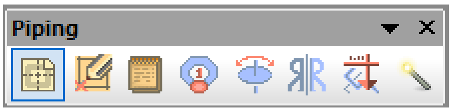
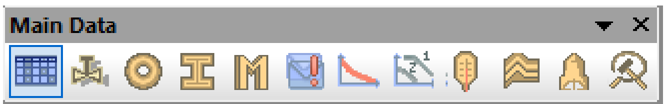
Changing the number display format

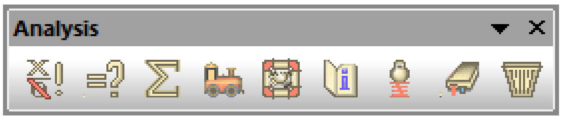
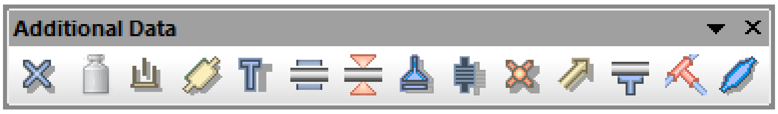
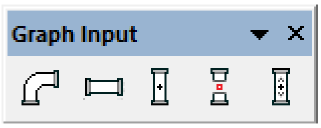
Option to change the format for displaying numbers in the "Geometry" table and for data associated with nodes of the analysis model for which direction must be specified. The dialog is called from the "Layout" table by simultaneously pressing the CTRL key on the keyboard and the right mouse button. In the context menu that appears, select the “Additional” item. In the dialog that appears, you can configure the number format (number of digits after the decimal point):



Toolbars

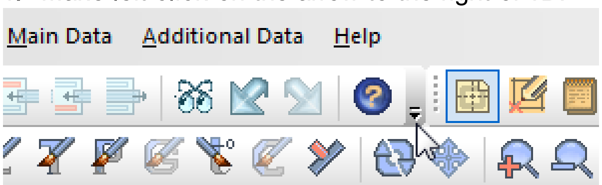
In order to provide easy operations with dPIPE, the DDE input data entry spreadsheet is equipped with the following set of toolbars, which can be customized by the User:

Toolbar	Display image
Main	
Piping	
Main data	

Analysis	
Graphics	
Additional Data	
Utils	
Graphic input	

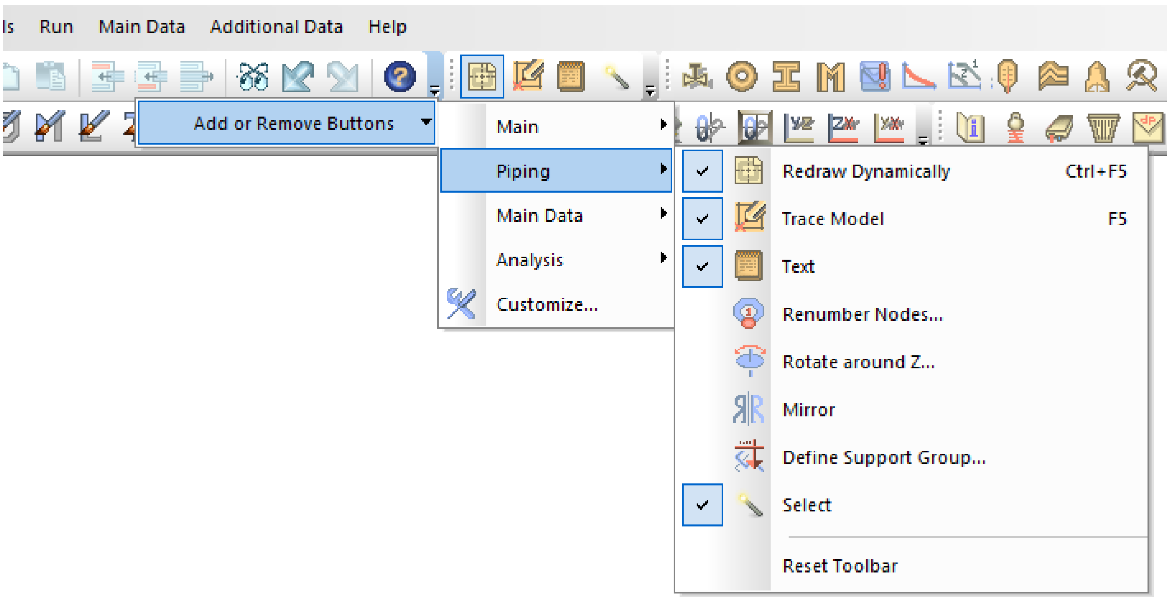
The toolbars (TB) allow hiding or showing individual buttons as required by the user. For example, it is possible to hide those button that are used rarely. For this purpose, it is necessary to do the following:

1. Make left click on the arrow to the right of TB:



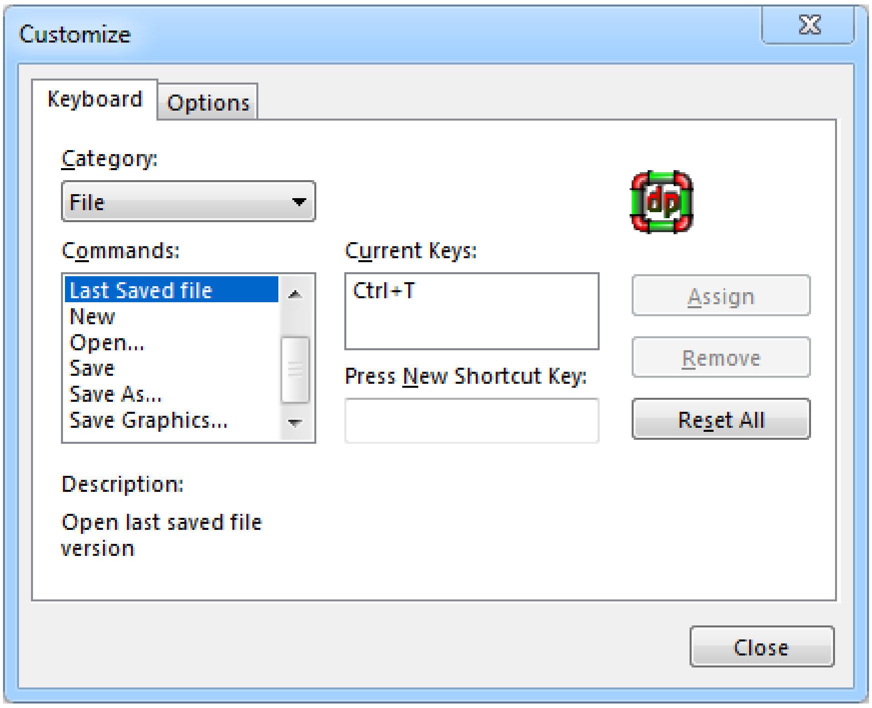
2. Place the mouse cursor over the "Add or Delete buttons" pop-down menu appeared. The menu with TB names will appear.

3. Place the cursor over the name of TB, which is to be changed, and, by right clicking on the button, mark (select) the buttons, which are to be displayed, or remove the marks (deselect) from the buttons, which are to be hidden:



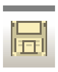
If on step 3 the «***Reset Toolbar***» menu item is selected, then a set of buttons defined for the TB by default will be displayed on it.

On step 2 the "Customize" menu item can be selected. (The same menu item is also contained in the "Tools" submenu of the program's main menu.) The following dialog box will appear:



The "Keyboard" tab of this dialog box allows to assign "hot keys". The "Options" tab allows selecting additional TB display options.

Main toolbar

Button	Action	Hot keys
	Create a new model	CTRL+N
	Open a file with existing model	CTRL+O
	Save a model	CTRL+S
	"Cut" data (effective in the text window)	CTRL+X, SHFT+DEL
	"Copy" data (effective in the text window)	CTRL+C
	"Paste" data from the clipboard (effective in the text window)	CTRL+V
	Insert a new line under the current line (effective in the "Geometry" window)	CTRL+ENTER
	Insert a new line above the current line (effective in the "Geometry" window)	CTRL+SHFT+ENTE R
	Delete a line (effective in all windows with spreadsheets)	CTRL+D

	Insert lines with comments in the geometry spreadsheet	-
	Show/hide comments in the geometry spreadsheet	-
	Find text content in the geometry spreadsheet	CTRL+ALT-F
	Find nodes and components ID in the graphic window	CTRL+F
	Cancel the previous action (effective only in the graphic window)	CTRL+X, ALT+Backspace
	Return to the previous action (effective only in the graphic window)	CTRL+Y
	Call "About program" information (also shows the licensing information and the number of connections for the network version of the program)	-

"Piping" toolbar

Button	Action	Hot keys
	Enable/disable the dynamic mode of rendering	CTRL+F5
	Redraw the model once (tracing)	F5
	Enable/disable the text window with input data	F7
	Renumber the nodes	-
	Turn the model at an arbitrary angle about the global Z axis	-
	Mirror the whole model	-
	Select and assign the support groups	-
	A tool for selecting a part of the model	-

Spreadsheets with basic data

Button	Action	Hot keys
	Enable/disable the window with the piping system geometry spreadsheet	-
	Enable/disable the window with the valves spreadsheet	
	Enable/disable the window with the pipe cross-section spreadsheet	-

	Enable/disable the window with the beam cross-section spreadsheet	-
	Enable/disable the window with materials spreadsheet	-
	Enable/disable the window with warnings	-
	Enable/disable the window with the fatigue strength spreadsheet	-
	Enable/disable the window with the specific data for high temperature piping	-
	Enable/disable the window with the piping system operation mode spreadsheet	-
	Enable/disable the window with seismic response spectrum spreadsheet	-
	Enable/disable the window with the specification for analysis and post-processing of the results	-
	Enable/disable the window with the analysis options	-

Analysis

Button	Action	Hot keys
	Check of Input Data and launch of pre-processor	-
	Analysis	-
	Post-processing of the results	-
	Analysis in the batch mode	-
	Show the results in the graphic form (PIPE3DV)	-
	View the input data listing	-
	View the summarized tables with loads on supports	-
	View the results listing	-
	Clear the working folder	-

Additional data

Button	Action
	Show/hide the anchor supports spreadsheet
	Show/hide the concentrated weight loads spreadsheet
	Show/hide the damper supports spreadsheet

	Show/hide the snubbers spreadsheet
	Show/hide the rod hangers spreadsheet
	Show/hide the guide supports spreadsheet
	Show/hide the restrains spreadsheet
	Show/hide the six-component supports spreadsheet
	Show/hide the spring support/hangers spreadsheet
	Show/hide the weld joints spreadsheet
	Show/hide the nodal force/moment spreadsheet
	Show/hide the sliding supports spreadsheet
	Show/hide the tee spreadsheet
	Show/hide the rigid struts spreadsheet
	Show/hide the spreadsheet with dynamic forces

Operations with the graphic window

Button	Action	Hot keys
	Quick Information	-
	measurement of distances between nodes	-
	angle measurement	-
	angle measurement(2)	
	center the model in the window	CTRL+E
	call of "Display Options" page	F3
	coloring of cross-sections	-
	coloring of materials	-
	coloring of load groups	-
	coloring of the temperatures for operational modes	-

	coloring of the pressures for operational modes	-
	coloring of the segment names	-
	coloring of the seismic support groups	-
	call of "Show elements by categories" dialog	F4
	toggle into the model rotation mode	-
	toggle into the model movement mode	-
	zoom in the model with respect of the mouse pointer	-
	zoom out the model with respect of the mouse pointer	-
	show the entire model	CTRL+A
	enable/disable 3D view rendering	CTRL+F10
	toggle the model into isometry keeping an actual scale	CTRL+I
	toggle the model into isometry and show it on the whole	CTRL+SHFT-I
	show the model on the X axis side	CTRL+SHFT+Z
	show the model on the Y axis side	CTRL+SHFT+Y
	show the model on the Z axis side	CTRL+SHFT+X

Note: In the "coloring" mode, it is possible to change colors. To do this, double-click on the legend with the mouse

Graphic input of the model

Button	Action	Samples
	Input of the straight piping section	Watch Video
	Insert bends along branch	Watch Video
	Insert intermediate node	Watch Video
	Split branch	Watch Video

	Move Node	Watch Video

Hot key summary

General

Action	Key combination
Open a new file	Ctrl-N
Open a file	Ctrl-O
Save a file	Ctrl-S
Return to the file saved	Ctrl-T
Toggle between active windows	F6, Shft-F6

Graphics window

Action	Key combination
Zoom	+/-
Rotate	→, ←,↑, ↓
Move	Ctrl →, ←,↑, ↓
Copy the graphic image of the model into the clipboard	Ctrl-C, Ctrl-Ins
Center the model in the window	Ctrl-E
Show the entire model	Ctrl-A
Show the model in isometry without zooming	Ctrl-I
Show the entire model in isometry	Ctrl-Shft-I
Show the entire model on the X axis side	Ctrl-Shft-X
Show the entire model on the Y axis side	Ctrl-Shft-Y
Show the entire model on the Z axis side	Ctrl-Shft-Z
Call a dialog for additional data	F3
Toggle between the graphics window and text window	F7
Return to the previous view	Backspace
Find a node	Ctrl-F
Search for text in the Spreadsheet or dp5 file	Ctrl-ALT-F
Show node labels	Shft-N
Show nodes	Shft-P
Show anchors	Shft-A
Show spring hangers/supports	Shft-H

Show stiff hangers	Shft-R
Show 6-component supports	Shft-S
Show nodal forces	Shft-F
Show sliding supports	Shft-I
Show guide supports	Shft-G
Show snubbers	Shft-U
Show restraints	Shft-J
Show dampers	Shft-D
Show weld joints	Shft-W
Show nodal weights	Shft-L
Show tees	Shft-T
Show dynamic stops with gaps	Shft-B
Show rigid struts	Shft-C
Enable/disable 3D rendering view	Ctrl-F10
Select elements by categories	F4
Show/hide elements selected by categories	Shft-F4
Zoom in/Zoom out proportionally the image of piping system diameters in the whole model	Ctrl - "Grey+" ("Grey-")
Return to к unscaled diameter	Ctrl-1

Windows with Input Data entry spreadsheets

Action	**Key combination**
Edit a cell	F2
Move between adjacent cells	→, ←,↑, ↓
Go to the next cell with data entry	Enter
Go to the 1st column	Home
Go to the last column	End
Paging	Page Up, Page Down
Go to the beginning of table	Ctrl-Home
Go to the end of table	Ctrl-End
Delete the current row of table	Ctrl-D
Insert a row before the current row	Ctrl-Shift-Enter
Insert a row after the current row	Ctrl-Enter
Find a Node	Ctrl-Alt-F
Trace the model	F5

Enable/disable dynamic tracing	Ctrl-F5

Main window in the text mode

Action	**Key combination**
Find a text	Ctrl-Alt-F
Find and replace a text	Ctrl-H
Copy a text into the clipboard	Ctrl-C
Paste a text from the clipboard	Ctrl-V

Menu commands

File menu

Edit menu

View menu

Service menu

Analysis menu

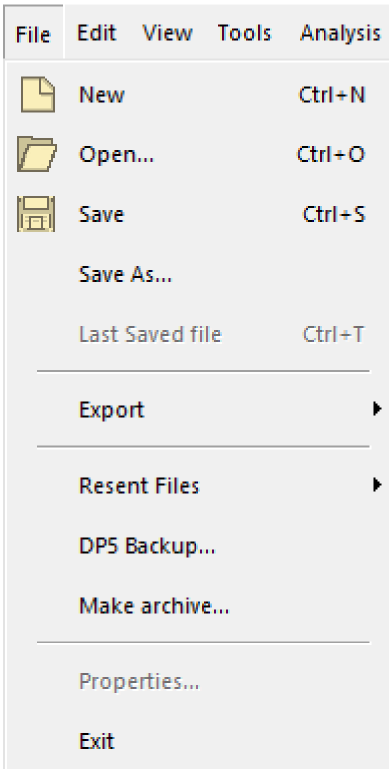
Main data

Additional data

Help menu

File

File – a conventional set of commands for operations with Input Data files (files with Input Data should have the extension *.DP5). The commands allow creating a new model, open the current model, save the current model under another name, save the calculation model image as well as return to the last saved version of the file with Input Data.

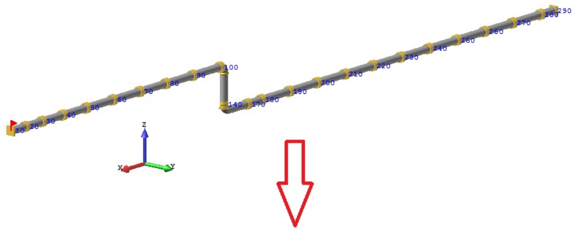


The “Export” command allows saving the image of the model into a file (Save Graphics), export a visible part of the model to PCF format. The item “Make Archive” allows to create a ZIP archive of the project, the item “Properties” contains a link to the full path of the input data file * .dp5.

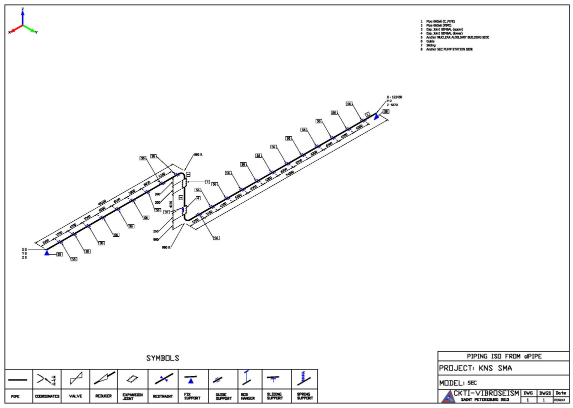
See also DP5 Backup

Create PCF

"Create PCF" allows converting the calculation model of the piping system into a file with the extension *.PCF, which can be used for plotting isometric drawings of piping systems by means of software developed by INTERGRAPH: ISOGEN SOLO EDITION or ISOGEN TEAM EDITION:



calculation model

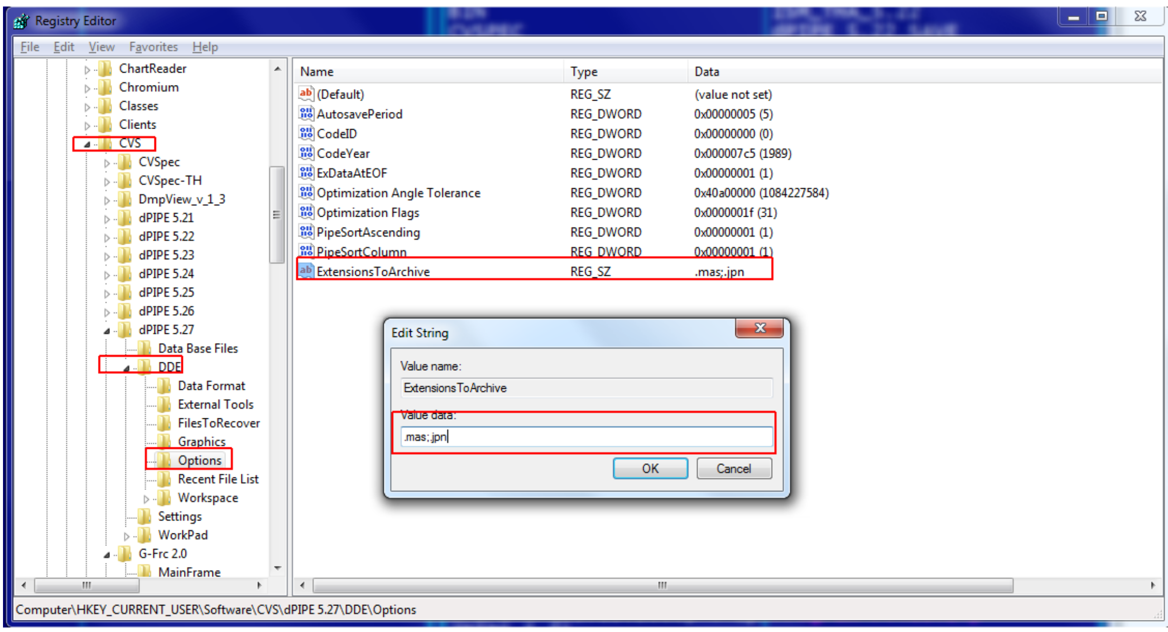


Isometric drawing of the calculation model, showing dimensions and notations

Archive Project

The command allows creating a ZIP archive of the project, which includes a *.dp5 file with input data, all relevant files referenced in the * .dp5 file (spectra, accelerograms and files with dynamic forces are recorded in separate folders), the calculation results (files with * .OUT, * .SUP; * .RES, * .BIN), the file release_info.txt with information about the current version of the program. Optionally, the archive includes a folder with databases. The archive file is named as <model name> _YYYYMMDD.zip (year-month-day)

User can add user-defined rules for archiving files with arbitrary extensions. To do that one should record in the registry the key HKEY_CURRENT_USER\Software\CVS\dPIPE5.27\DDE\Options\ExtensionsToArchive:



The key data ExtensionsToArchive is formed by the rule: .EXT, where EXT is the extension of the files to be added to the archive. When listing several types of extensions, a semicolon ";" is used as a separator.

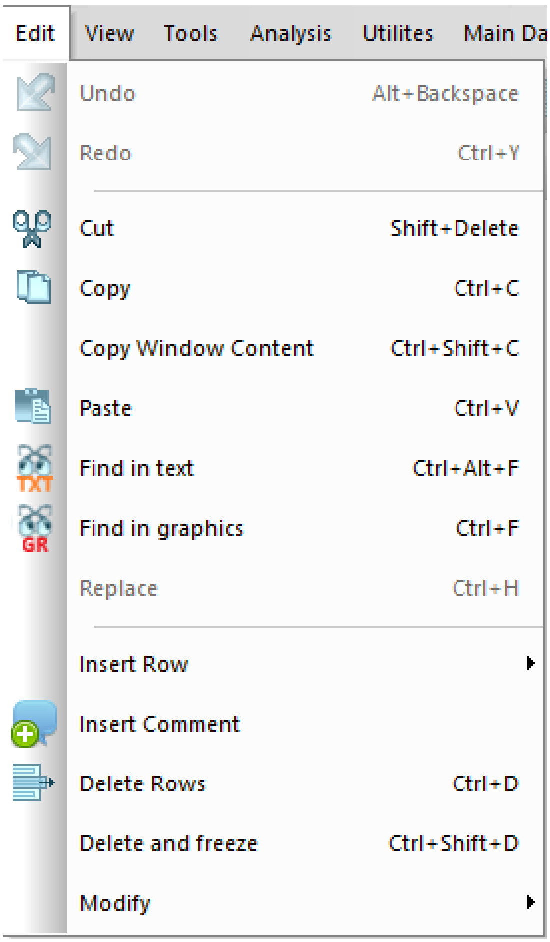
Warning: archiving option works only with the installed .NET Framework version 4.5 and higher

dP5 Backup

Depending on the "Model Autosave Time" parameter, dPIPE saves the calculation model every N minutes as specified in this parameter. Typically, in the event of a crash, the file with the calculation model automatically opens upon relaunching the program. If this does not occur, the User can use this command to open the folder where backup files are stored. Upon normal program termination, the backup file is deleted. Calculation model files saved in this folder are named using a code + the filename of the calculation model and the extension *.dP5.

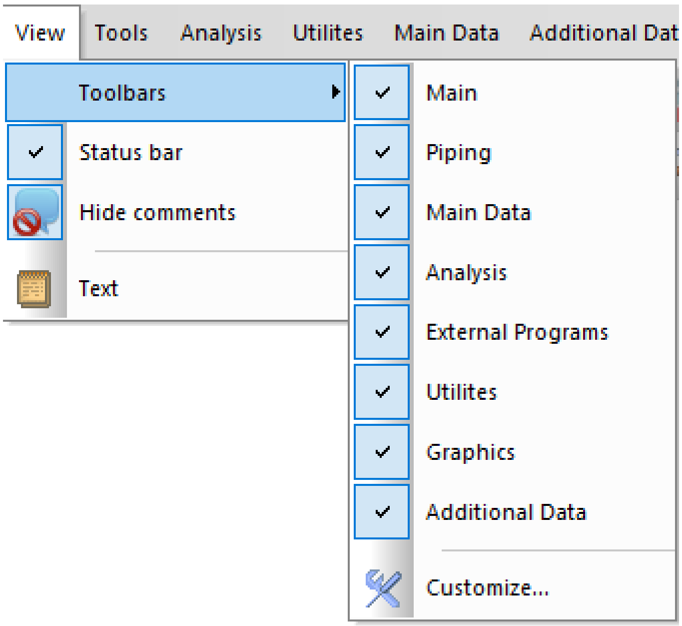
Edit

Edit – a set of commands for Input Data editing. "***Find in Text***" and "***Find in Graphic***" commands let you search for node names, identification names of supports and piping elements, and other text information in various tables and in the graphic window of the program. The "***Insert/Delete row***" commands are active in the program dialogs containing spreadsheets with input data. The "***Modify***" command is active in the mode of selection of the elements ( button on the "Piping" toolbar)



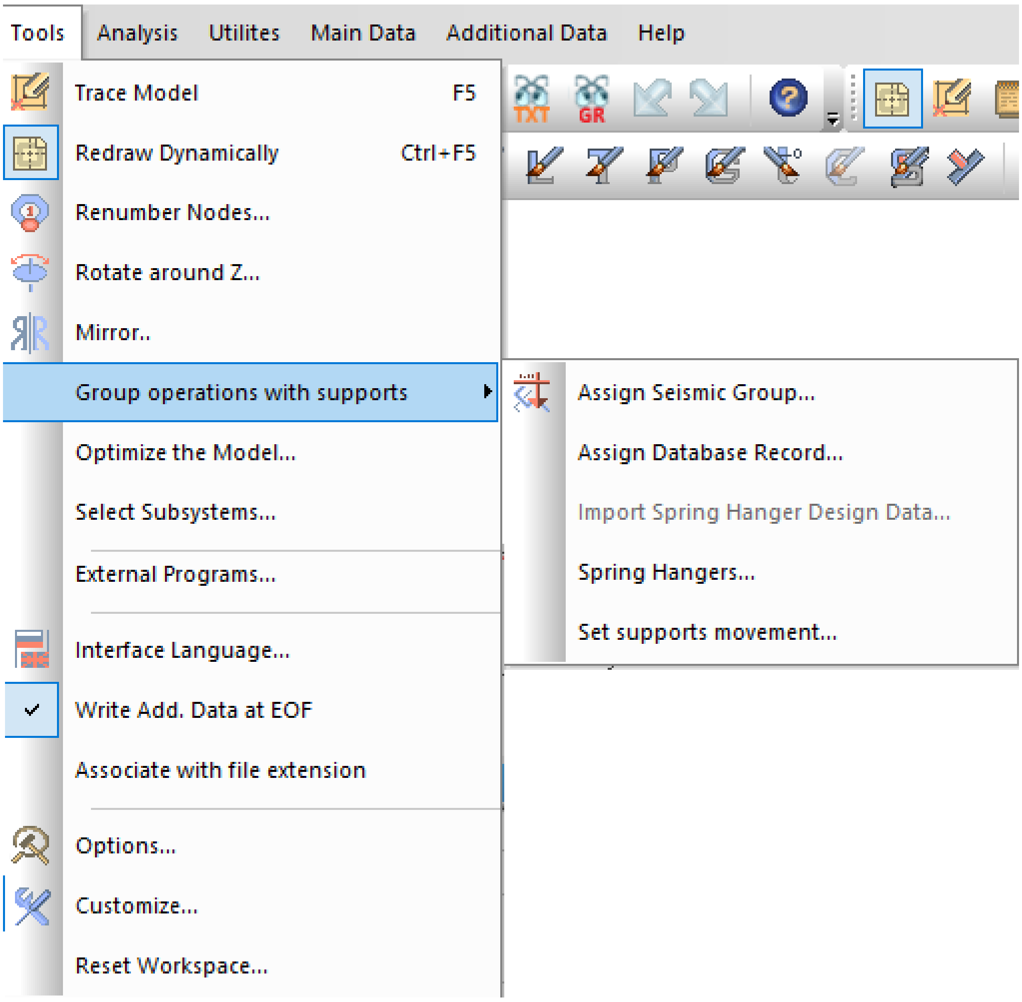
View

View – a set of commands for activating certain elements of the spreadsheet interface. The "Text" command switched the program window into the graphic or text mode.



Tools

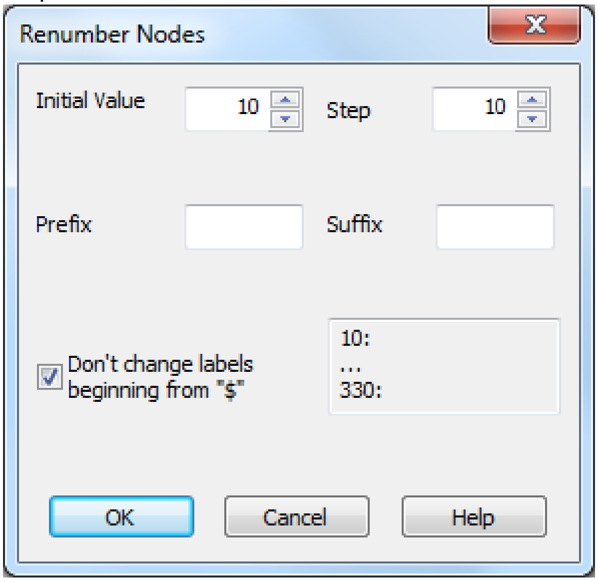
Tools – a set of tools for work with DDE.



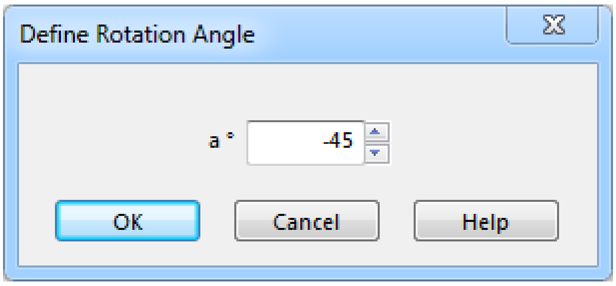
The ***"Trace Model"*** command performs a single check of the input syntax and redraw the piping system model without reaching the active mode.

In selecting the ***"Redraw Dynamically"*** mode, the Input Data entry is carried out in the active mode.

The "***Renumber nodes***" command allows renumbering the nodes of the current design mode according to the template specified:

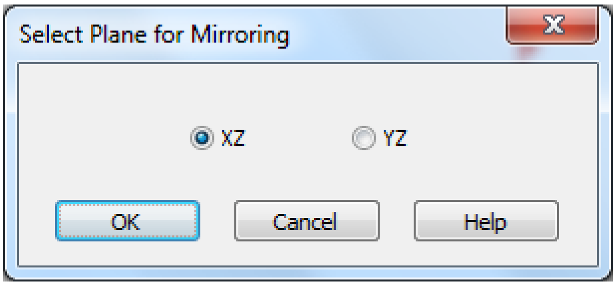


The "***Rotate around Z...***" command allows rotating the image of the whole model about the OZ axis at angle a. In doing so, the positive direction of the turn shall be counted from the OX axis to the OY axis:



In case of model rotation, the automatic recalculation of the pre-defined displacements of supports takes place.

The "***Mirror...***" command allows mirroring the image of the whole model with respect to the XOZ or YOZ plane:

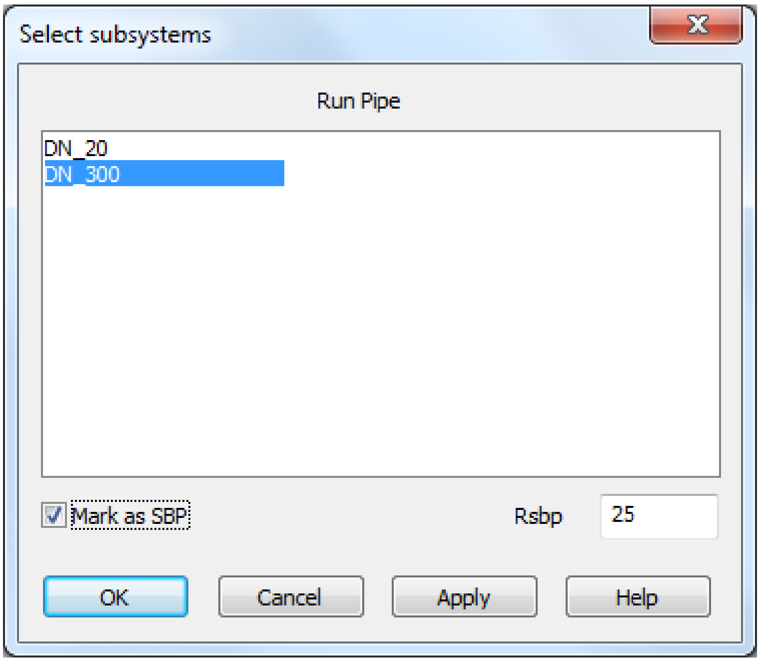


The command does not apply to the display of the cross sections of beam elements (Beam(S))

The command ***"Group operations with supports"*** contains a number of operations that are applied to various supports of the model selected with the mouse.

Upon the "***Optimize the model***" command, the model topology optimization (merge of individual branches) and the deletion of the intermediate nodes take place, which are not connected to any additional data. The use of this command is convenient for models, which were created by converting data.

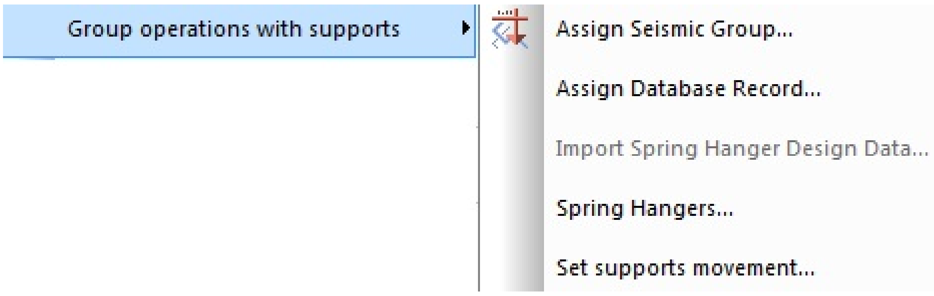
"Select Subsystems" command is used to define those portion of piping that may be marked as SBP(small bore pipes), see Appendix XV. The RSBP value is the ratio of run to branch pipe moment of inertia



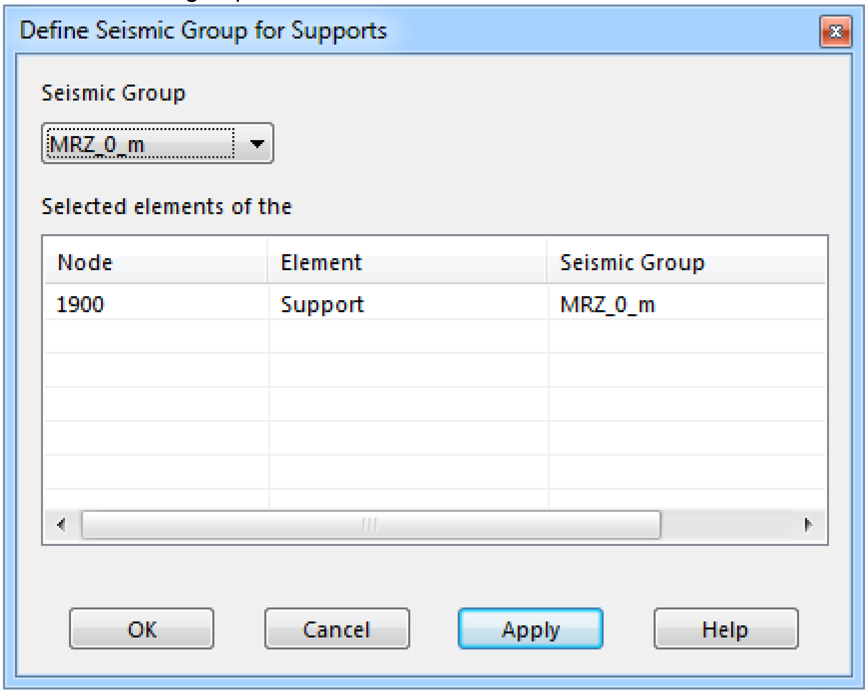
An option "***Write Add. Data at EOF***" allows put all the data associated with the nodes of the model (mainly this is piping supports) at the end of the * .dp5 file, which allows them to be efficiently edited in the text mode

Group operations with supports

The set of the following commands is contained under the title "***Group operations with supports***":



The "***Assign Seismic Group***" command serves for assigning of the certain supports to specific seismic group associated with those defined in the "***Response spectra***" command. The command is used in seismic analysis by the RSM taking into account independent support motions (multi-support excitation effect). When the button is pressed, the mouse cursor in the graphics window takes the form of frame with arrow tool ( ) and the dialog appears displaying the groups assigned and allowing, in the graphics window, to mark those areas , in which the supports attributed to a certain seismic group are located:



The "***Assign Database Record***" command allows selecting all one-type supports in the graphics window and assign them one and the same record from the database containing permissible loads on piping supports.

Watch Video

The ***"Import Spring Hanger Design Data"*** command allows the importing of the relevant data resulting from the calculations for spring's design (spring's types, sizes and operation load) to DDE table. This operation must be carried out immediately after the completion of the relevant calculations.

Watch Video

The "***Spring hangers...***" command is used for various operations under a selected group of the spring hangers/supports: change of the spring's size/type or table (catalogue), locking for hydraulic test, etc.

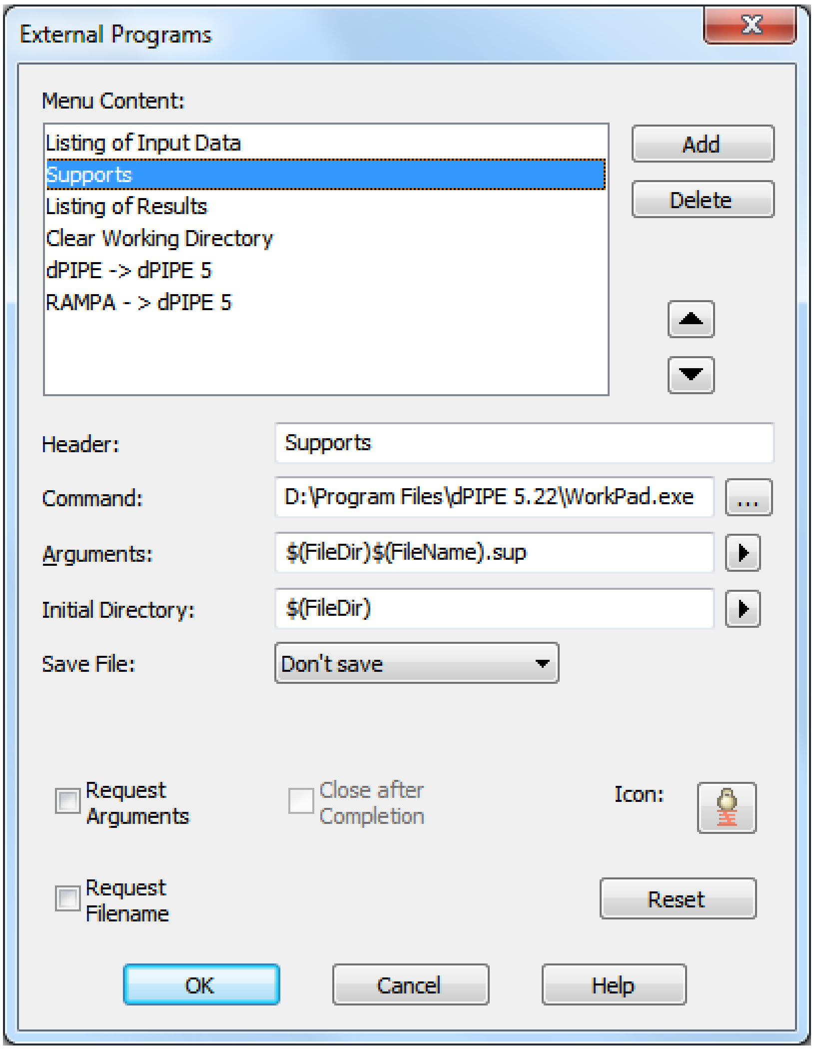
Watch Video

The "***Set supports movements***" command assigns predefined movements for the selected group of supports according to the operational mode consistent with this assignment.

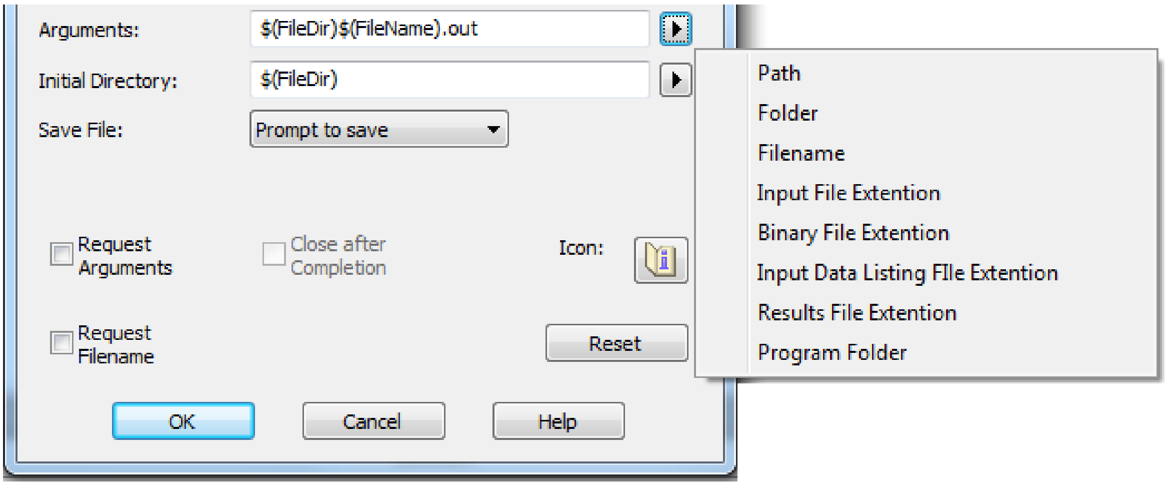
Watch Video

External programs

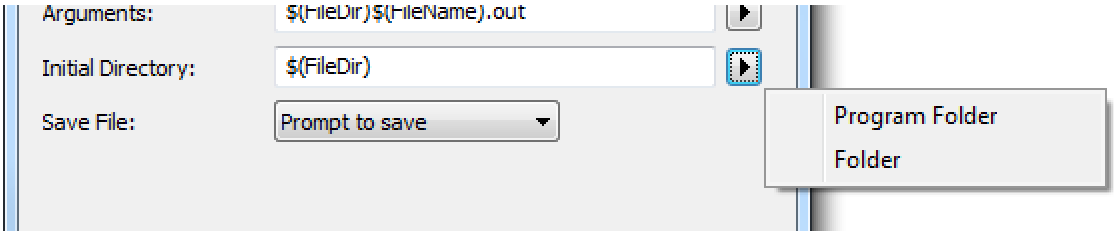
The **"External programs..."** command allows configuring the external modules, which could be connected and called from DDE:



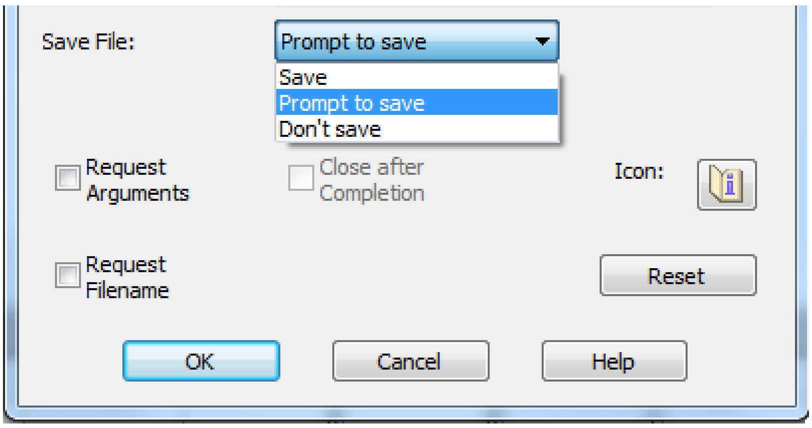
The dialog shown above allows to ***add/delete*** the programs connected, move them ***up/down*** in the ***"Menu content"*** field, give the title for executable programs (it is displayed both in the ***"Tools"*** menu and in pop-up tips to the corresponding buttons). The ***"Command"*** field contains the full part of the executable file, ***"Arguments"*** allow to transfer certain values to the command line:



The ***"Initial directory"*** field specifies the place, from which a certain program shall be started:

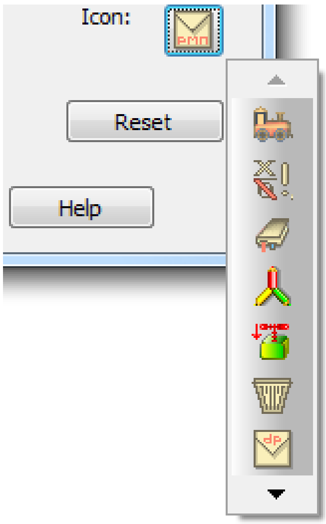


The ***"Save a file"*** option established the rules for saving the current Input Data file before execution of the external program.



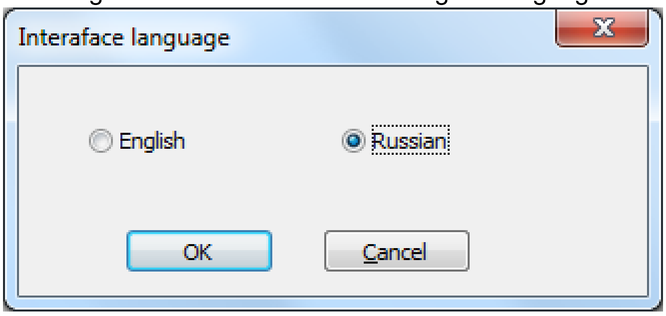
The "***Request the arguments"*** , ***"Close on completion"*** and ***"Request the filename"*** fields can take the "Yes" and "No" values depending upon their activation by users.

The ***"Image on the button"*** field allows associating a certain picture on the button with the executable module:



Interface Language...

The ***"Interface Language..."*** command is used for changing the program interface language from Russian into English and vice versa. The change of language is carried out in the following dialog:



and shall be effective after restart of the DDE spreadsheet. With the change of language, not only the program interface but also the language of printouts of the input data and the results of analysis will change.

Options and settings

The "Options" () command allows to make settings of the connected database files, control parameters and specifications for analysis and post-processing of the results.

"Files"

"Control parameters"

"Specifications"

"Reports"

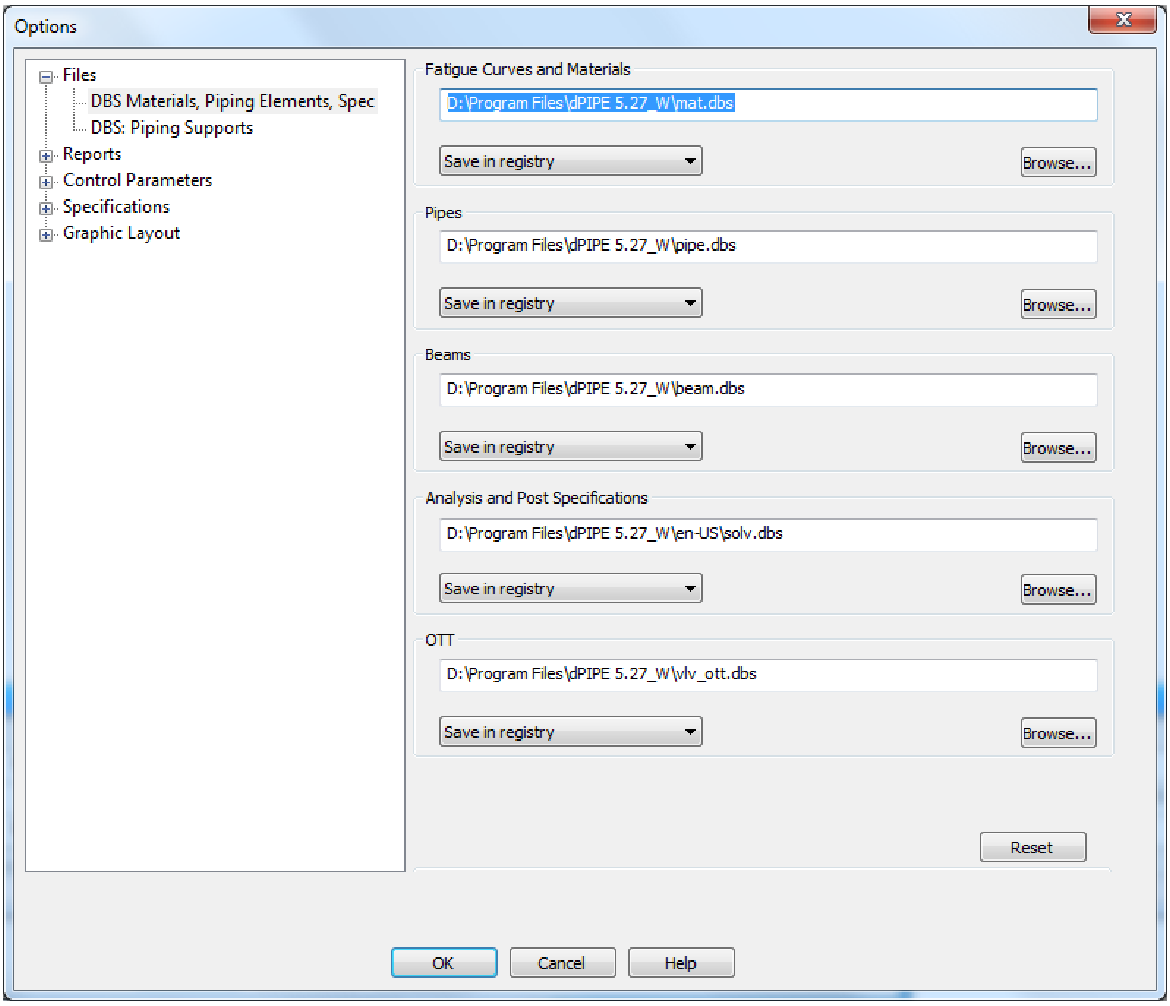
"Graphic Layout"

The "Settings" command () is used for establishing the user's set of toolbar "hot" keys and settings.

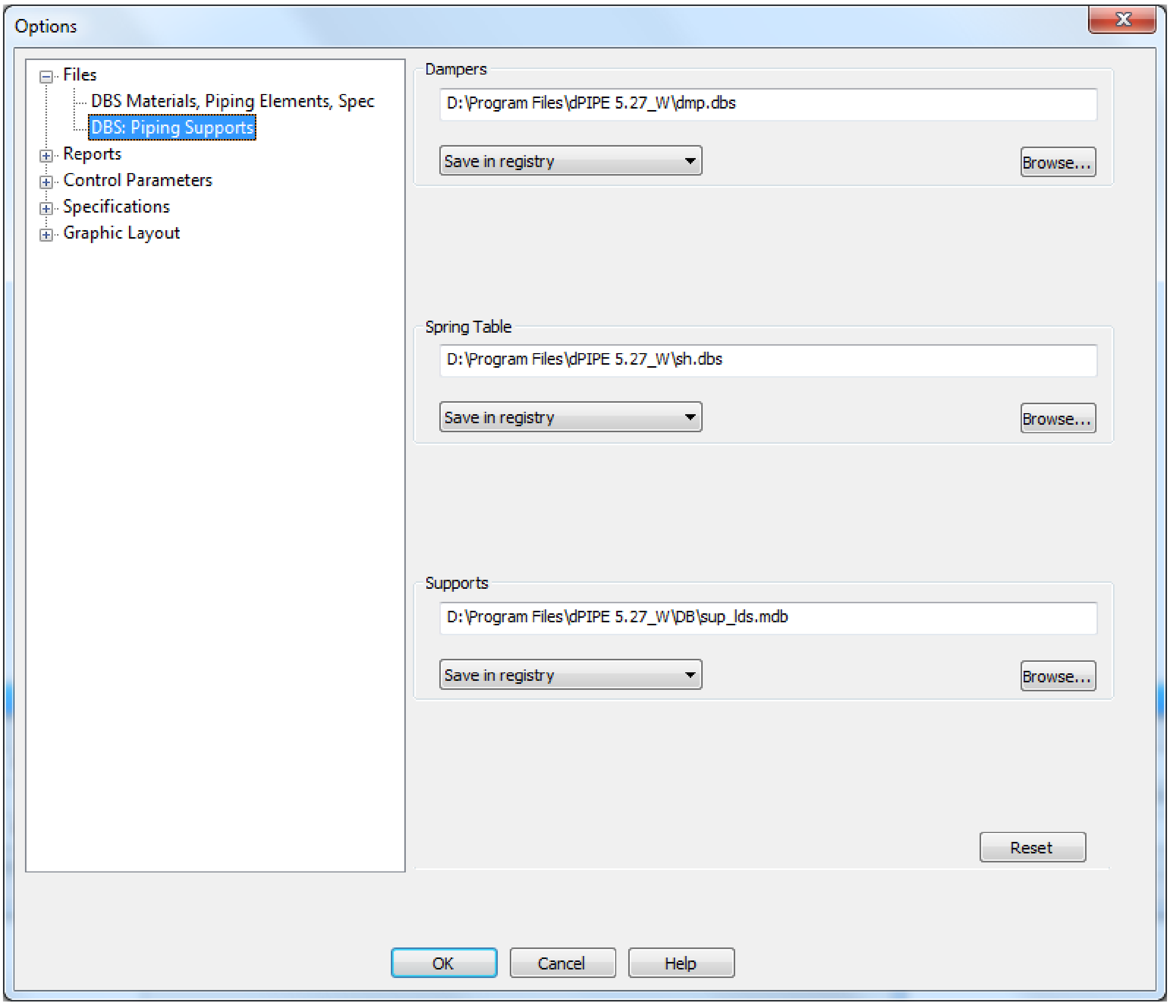
Files

The ***"Files"*** folder allows to assign the location of files to be used at creating the calculation model of the piping system.

The following bookmarks are located inside the folder: "**DBS: Materials, Piping Elements, Spec"**



"**DBS: Piping supports"**:



The tabs with databases contain paths to the corresponding files. Depending upon the options specified in the dialog these paths can be saved both in the system register ("save in registry") and can be written into a file with the model ("save in a file", see also the DBF command). With the "save relative paths" options being enabled, the program writes down the paths to files in respect to the folder, in which the model of the piping is located. This method is convenient for transferring the model from one computer to another.

The "Reset" button cancels all changes made by the User and returns all changes to the default parameters.

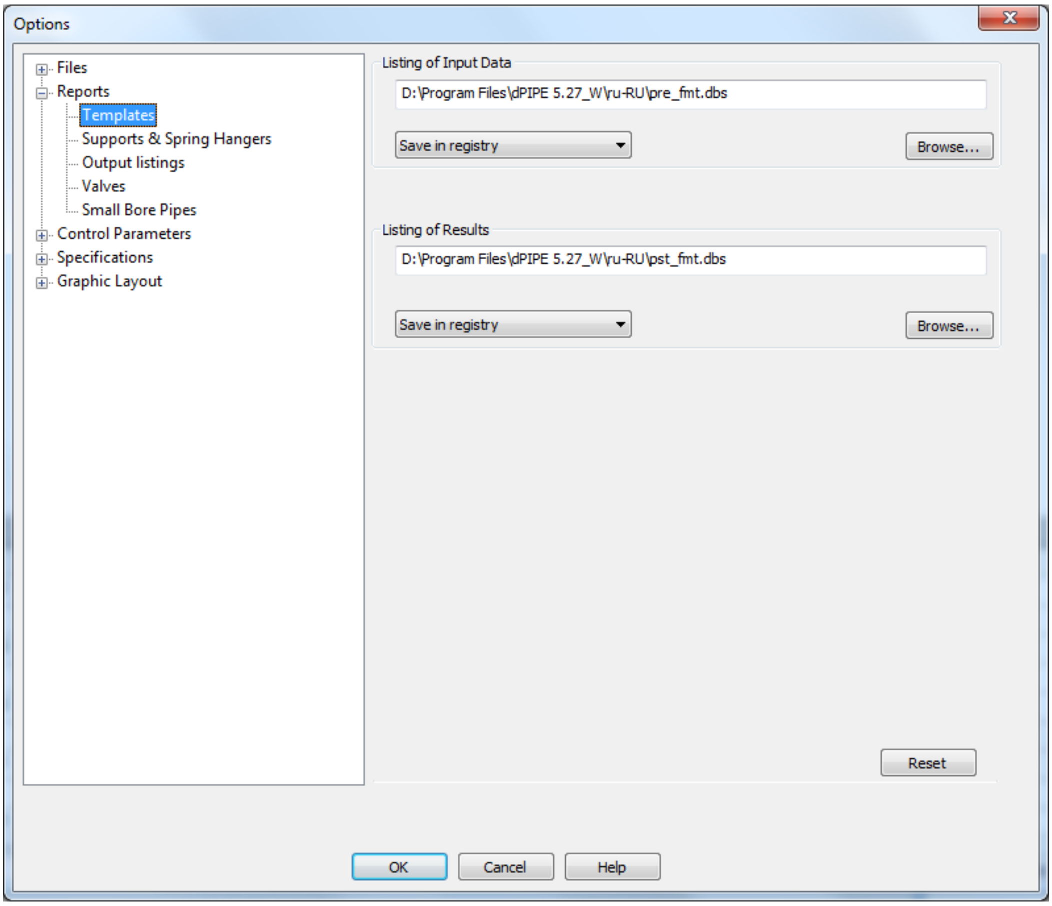
Reports

"Reports" tab determines the data to be used for generating of the summary tables with results (POST_REP):

Templates

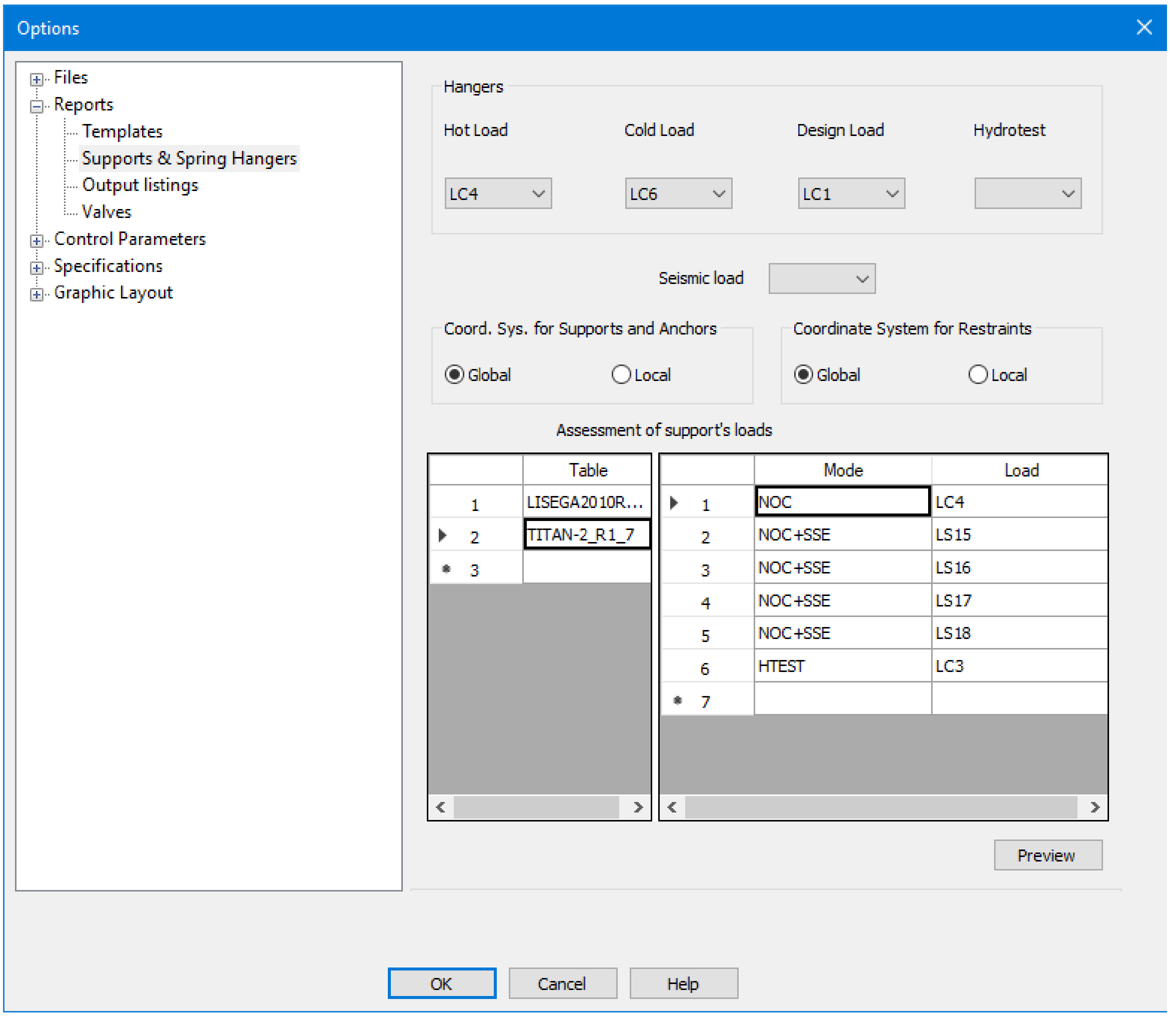
"Templates":

It defines location of templates that are used for output listing files with input data (*.out) and results (*.RES and *.SUP). See command DBF as well.



Supports & Spring Hangers

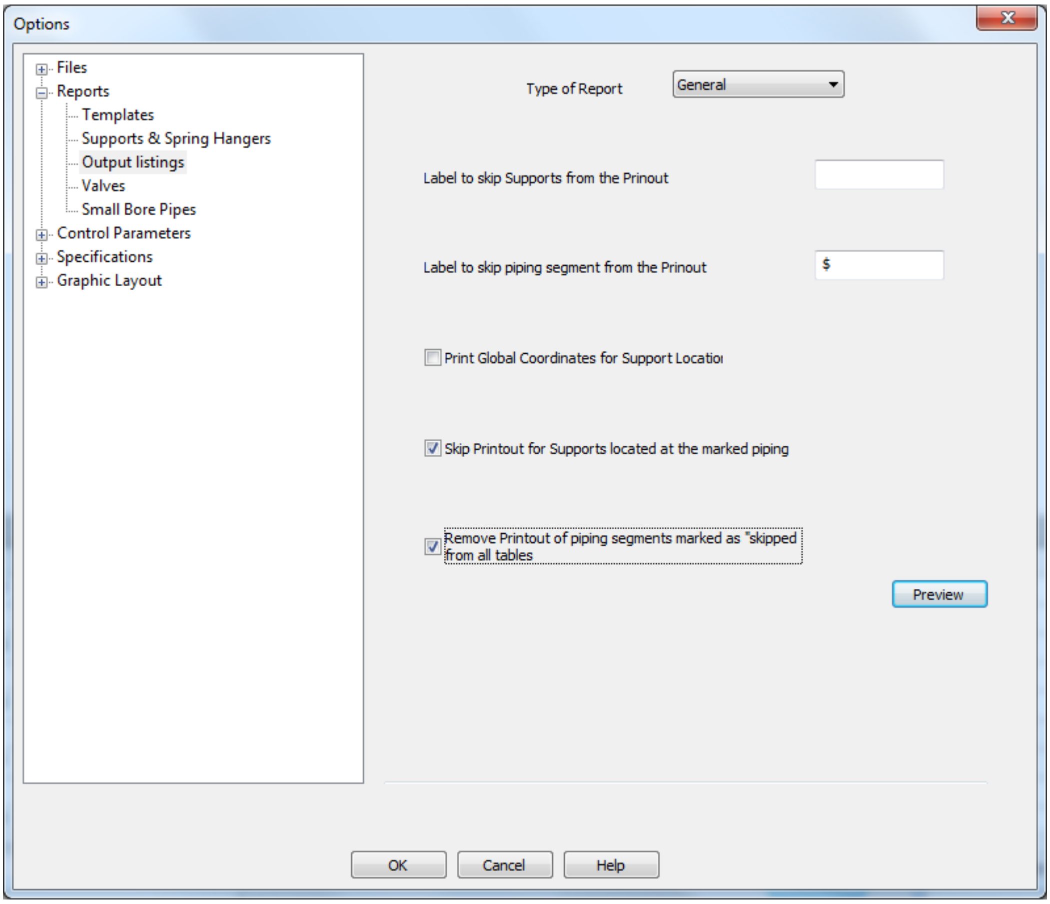
Dialog field	Parameter
Hot Load (hangers)	LOAD_HOT
Cold load (hangers)	LOAD_CLD
Design load (hangers)	LOAD_DES
Hydrotest	LOAD_HT
Seismic Load	LOAD_SEISM
Coordinate system for 6-component supports and Anchors	ANC_CS
Coordinate system for Restraints	RSTR_CS
Assessment of support's loads	SUP_LOADS



Output Listing

"Output Listing":

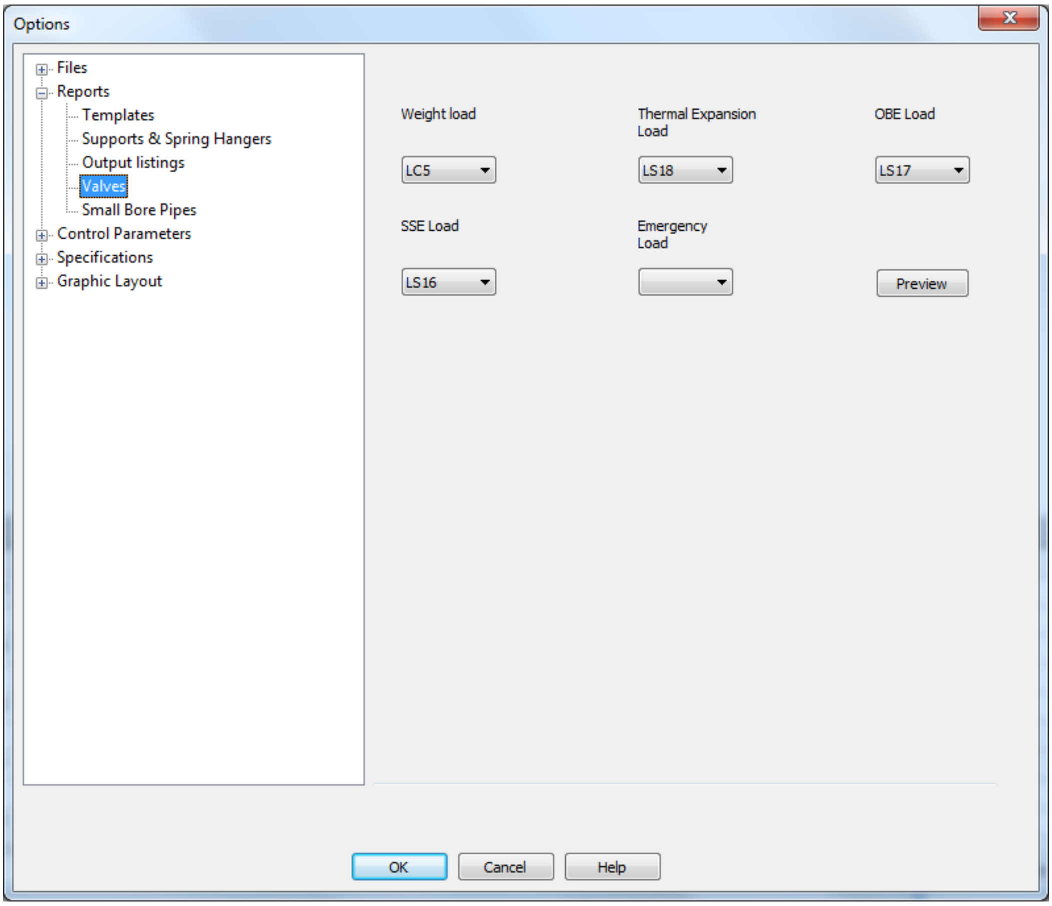
Dialog field	Parameter (POST_REP command)
Type of report	REP_TYPE
Label to skip Supports from the Printout	SKIP_SUP
Label to skip piping segment from the Printout	SKIP_STR
Print Global Coordinates for Support's Locations	SUP_CRD
Skip Printout for supports located at the marked piping segments (all Tables)	SUP_SKIP
Remove Printout of piping segments marked as "skipped" from all tables	SKIP_OUT



Valves

"Valves":

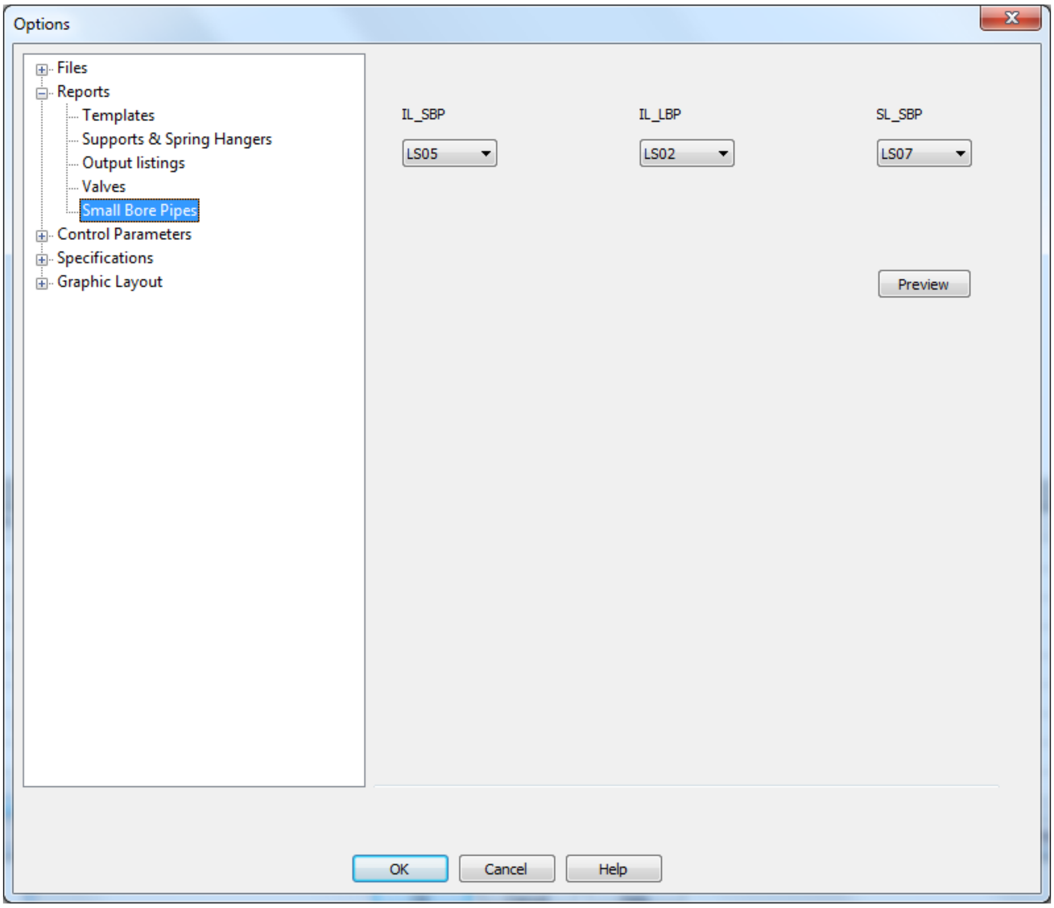
Dialog filed	Parameter (POST_REP command)
Weight load	OTT_W
Thermal Expansion Load	OTT_T
OBE Load	OTT_PZ
SSE Load	OTT_MRZ
Emergency Load	OTT_AS



Small Bore Pipes

The tab "Small-bore pipes" becomes available for entering data when the model contains at least one segment of pipes marked as "SBP", see Appendix XV.

Dialog filed	Parameter (POST_REP command)
Reference on seismic stresses accounting the inertial loads for ***small bore pipes***	IL_SBP
Reference on seismic stresses accounting the inertial loads for ***"big" bore pipes***	IL_LBP
Reference on stresses accounting the secondary seismic (i.e. SAM) loads for ***small bore pipes***	SL_SBP



Control parameters

The ***"Control parameters"*** tab contains the parameters being described by the CTRL command and has four levels:

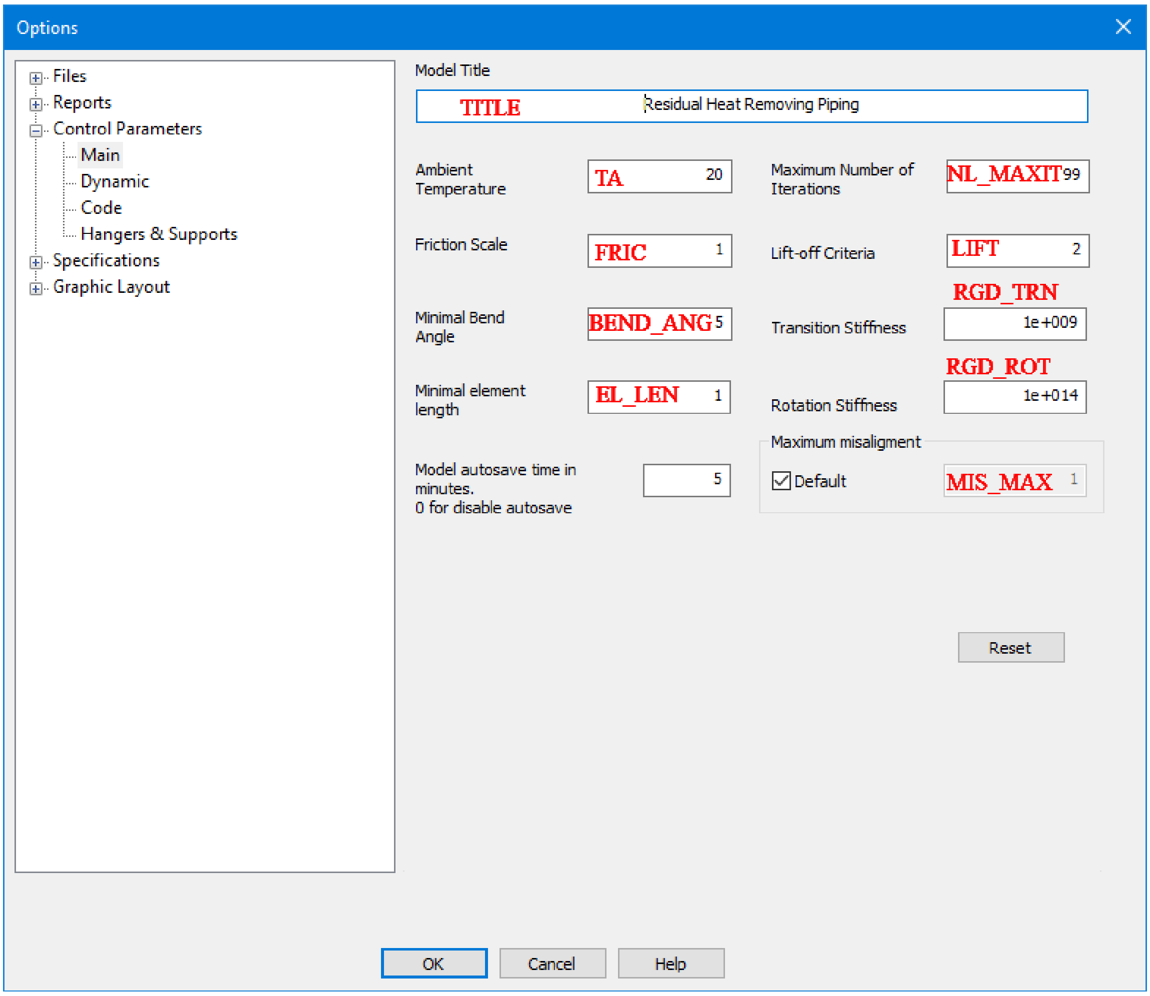
⇨ *Main* control parameters;
⇨ *Dynamic parameters*;
⇨ parameters depending upon Strength analysis codes (*Code*) as well as
⇨ parameters for the springs of flexible hangers and supports of piping systems (*Hangers & Supports*).

Main parameters

"Main" tab

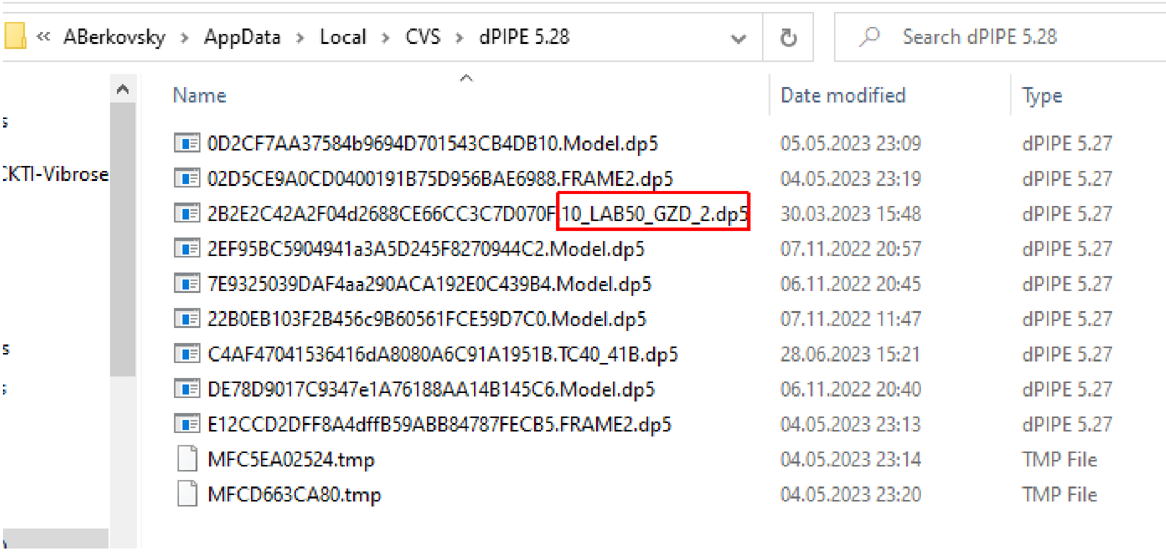
Dialog field	Command	Parameter
Model Title	TITLE	-
Ambient (Installation) temperature	CTRL	TA
Friction Scale	CTRL	FRIC
Minimal bend angle	CTRL	BEND_ANG
Minimal element length	CTRL	EL_LEN
Maximum number of iterations	CTRL	NL_MAXIT

Lift-off Criteria	CTRL	LIFT
Maximal mismatch value	CTRL	MIS_MAX
Transition Stiffness (default for "rigid")	CTRL	RGD_TRN
Rotational stiffness (default for "rigid")	CTRL	RGD_ROT
Model's autosave time	see: NOTE	



NOTE:

The 'Model autosave time' feature is utilized by the program to periodically save the *.dp5 file in a folder located at the address "%LocalAppData%/CVS/dPIPE 5.XX" (where XX represents the current version number of the program). In case of an abnormal termination of a session with DDE, the program attempts to recover data from this folder. Throughout the session, the program writes a *.dp5 file to this folder under a name that includes the model's name:



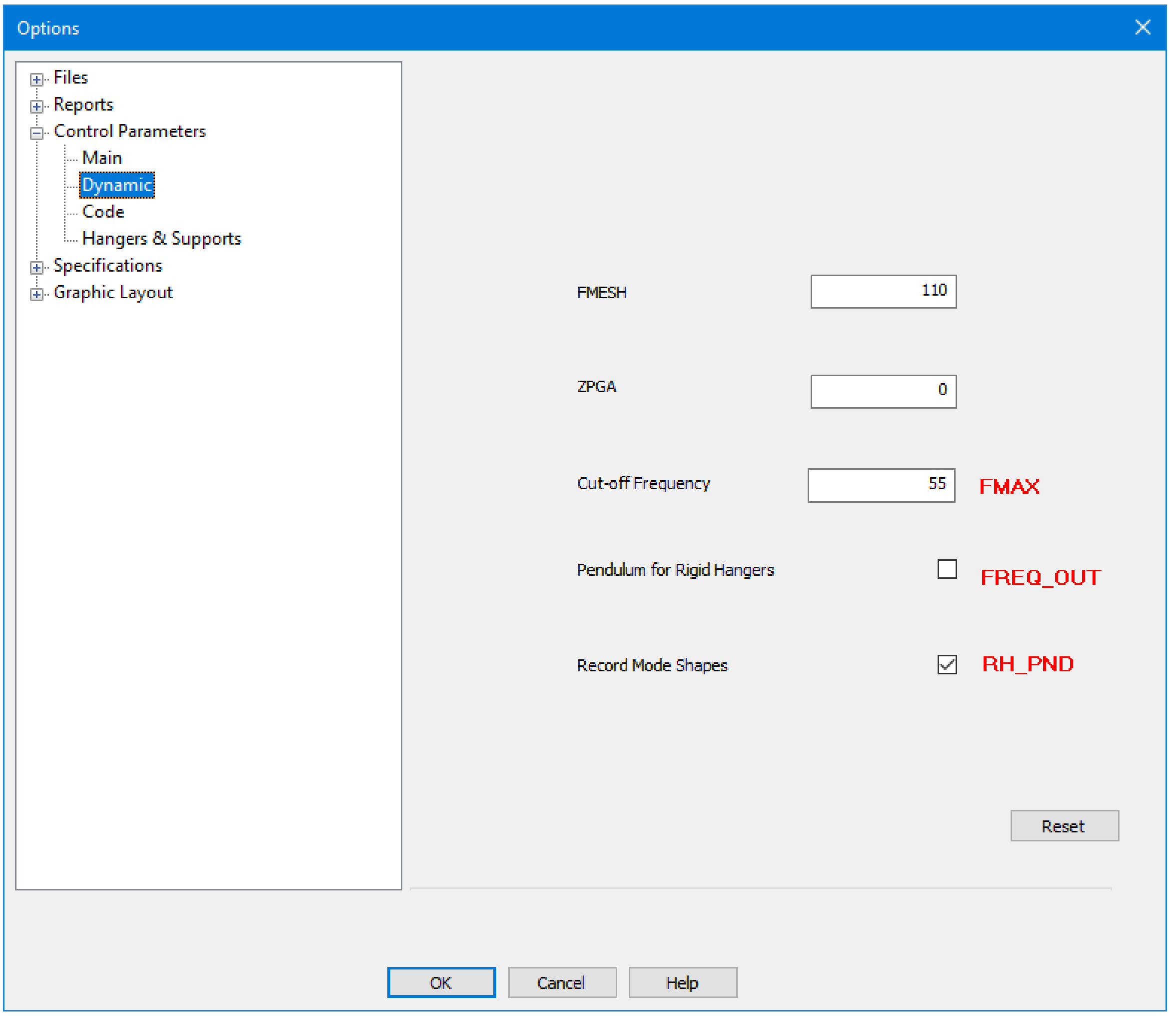
When the session terminates normally, the temporary file is deleted.

This folder can be accessed using the 'dP5 Backup' command.

Dynamic parameters

"Dynamic parameters"

Dialog field	Command	Parameter
Cut-off Frequency	CTRL	FMAX
Partial frequency for automatic meshing of the piping FE model (FMESH)	CTRL	FMESH
Zero period ground acceleration (ZPGA)	CTRL	ZPGA
Record Mode Shapes	CTRL	FREQ_OUT
Swing or pendulum effect for rod hangers	CTRL	RH_PND

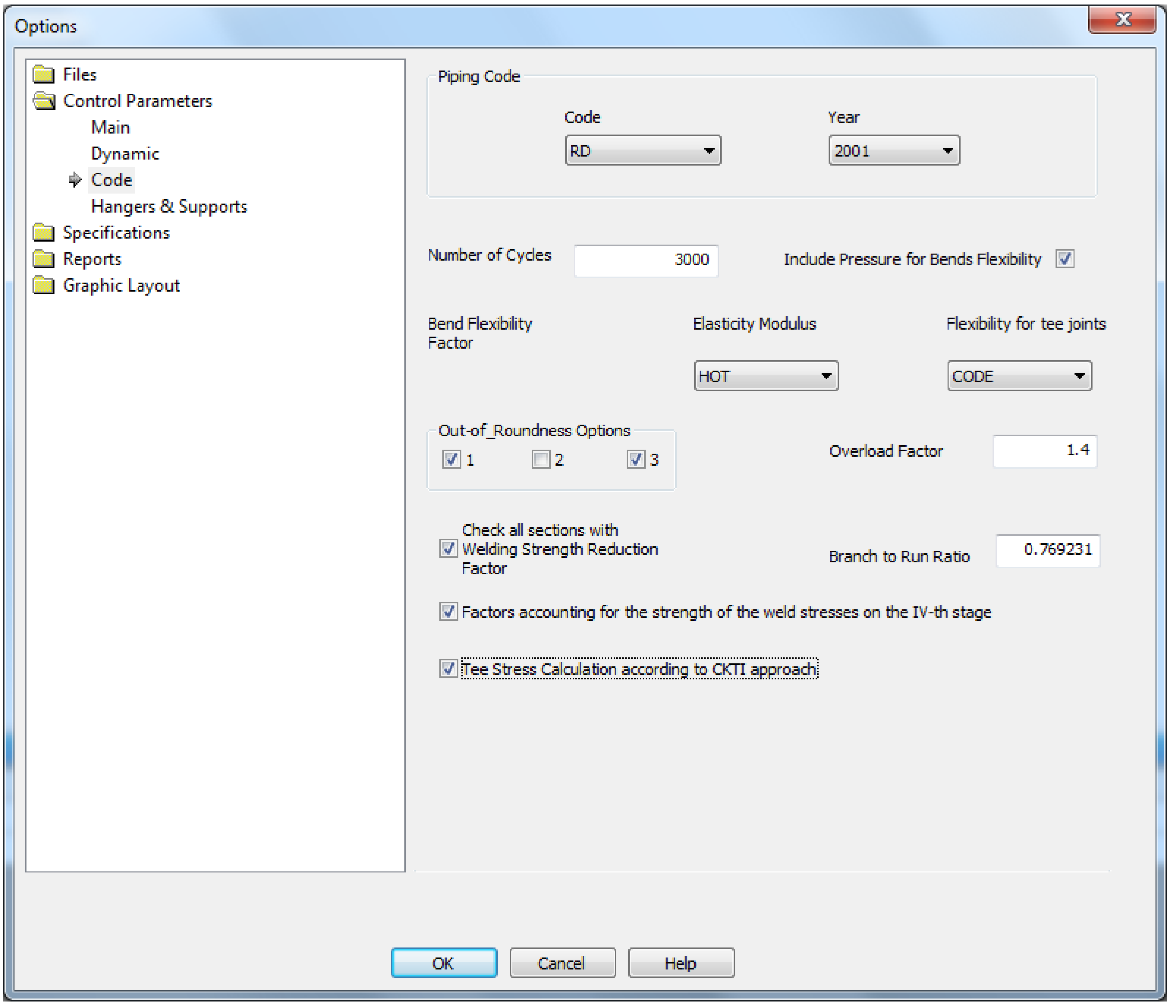


Code

Strength Analysis Code

Dialog field	Command	Parameter
Codes	CTRL	CODE
Year	CTRL	CODE_YEAR
Out-of-Roundness options	CTRL	OVAL
Overload Factor	CTRL	KS
Check of all cross-sections with Welding Strength Reduction Factor	CTRL	WLD_CHK
Stress averaging factor (used for hight temperature piping to address section's out-of roundness of the curved pipes)	CTRL	HI_E
Bend Flexibility Factor	CTRL	BEND_CODE
SN_T (criterion for the computation of allowable stresses for occasional and emergency loads)	CTRL	SN_T
Number of cycles	CTRL	NC

Option for accounting of the pressure for the bend flexibility factor	CTRL	BEND_PRES
Modulus of elasticity to be used for stiffness matrix generation (Hot or Cold)	CTRL	E_MOD
Simplified elastic-plastic analysis (for PNAE Code only)	CTRL	PNAE_KE
Occasional loads amplification factor for allowable stresses	CTRL	K_OL
Option for accounting the pressure for the bend stress index calculations (ASME B31.1 only)	CTRL	BEND_PSTR
Welding Stress Reduction Factor for Sustained Loads (ASME B31.1 only)	CTRL	WLD_SUST
Method for the computation of stresses from pressure at the SL category stress analysis	CTRL	SL_PRES
Method for the computation of SA permissible stresses for SE category stresses (ASME B31.1)	CTRL	SA_LBRL
Branch to Run Ratio (TEE assessment)	CTRL	BRN_RUN
Option for limitation of allowable stresses Sh and/or Sc with the value of 140 MPa	CTRL	SH_140
Option to account the weld strength factor for stresses at stage IV (RD Code)	CTRL	RD_WLD_IV
Use CKTI Engineering methodology for Stress Analysis of Tees and Branch Connections (only within 'RD' Code), See Appendix XIII.	CTRL	TEE_RD
Option for tee/branch connections flexibility factors. Applicable for "standard" tees. See Appendix XIV.	CTRL	TEE_FLEX

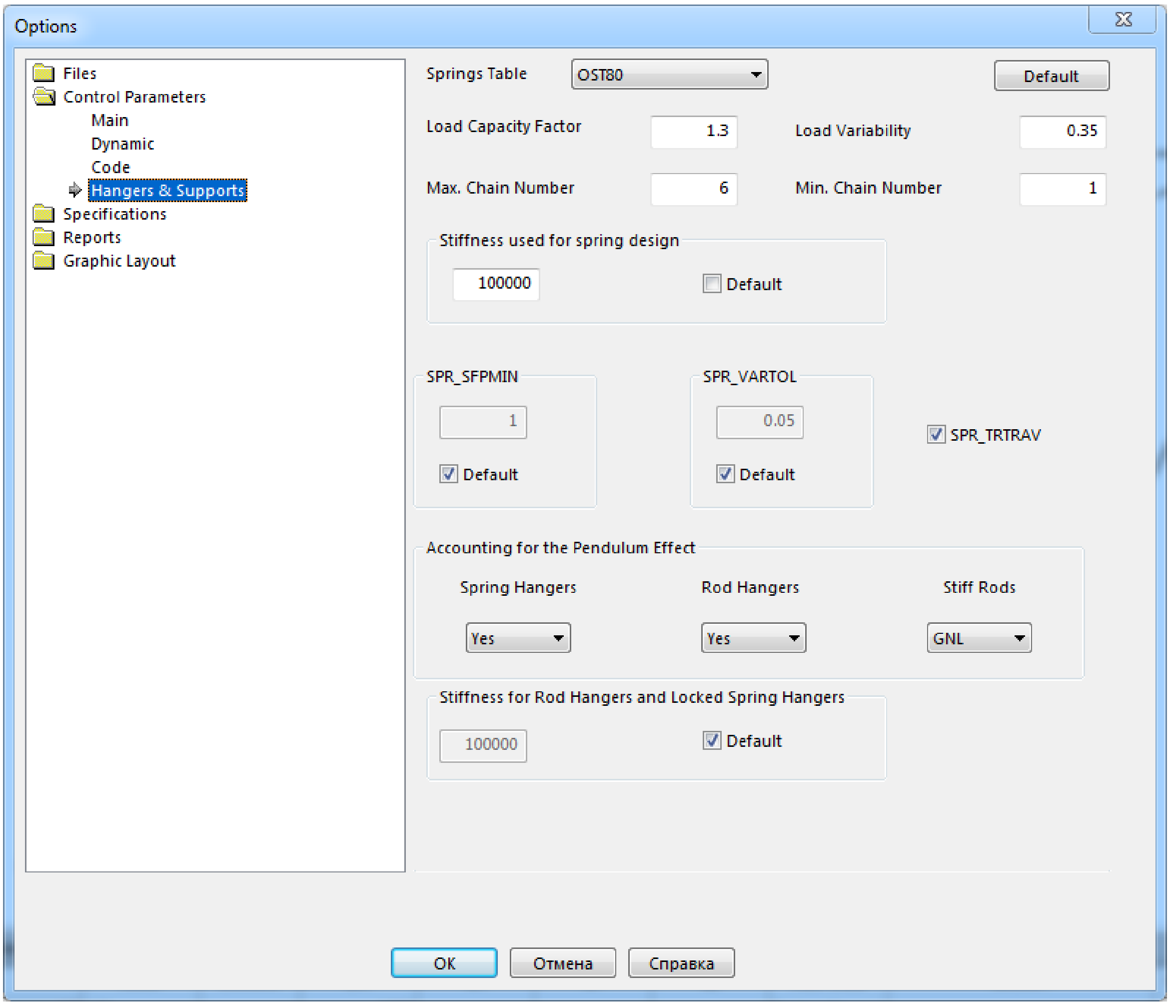


Hangers and Supports

Hangers and Supports

Dialog field	Command	Parameter
Spring's Table (Standard)	SDEF	STAB
Load Variability	SDEF	PVAR
Load Capacity Factor	SDEF	PFAC
Max. Chain Number	SDEF	ZMAX
Min. Chain Number	SDEF	ZMIN
Stiffness used for spring's design (stiffness of the "rigid" support when spring size is selected to balance weight load)	CTRL	RGD_SPR
SPR_SFPMIN (*minimum load safety margin coefficient*)	CTRL	SPR_SFPMIN
SPR_VARTOL (*reference variability value to be used in selecting the springs*)	CTRL	SPR_VARTOL
SPR_TRTRAV (method to compute *spring travel*)	CTRL	SPR_TRTRAV
Stiffness of rod hangers and locked spring hangers	CTRL	RH_STF
Swing (Pendulum) Effect		

Spring hangers	CTRL	SWING_SH
Stiff hangers	CTRL	SWING_RH
Stiff tie rods	CTRL	SWING_ST



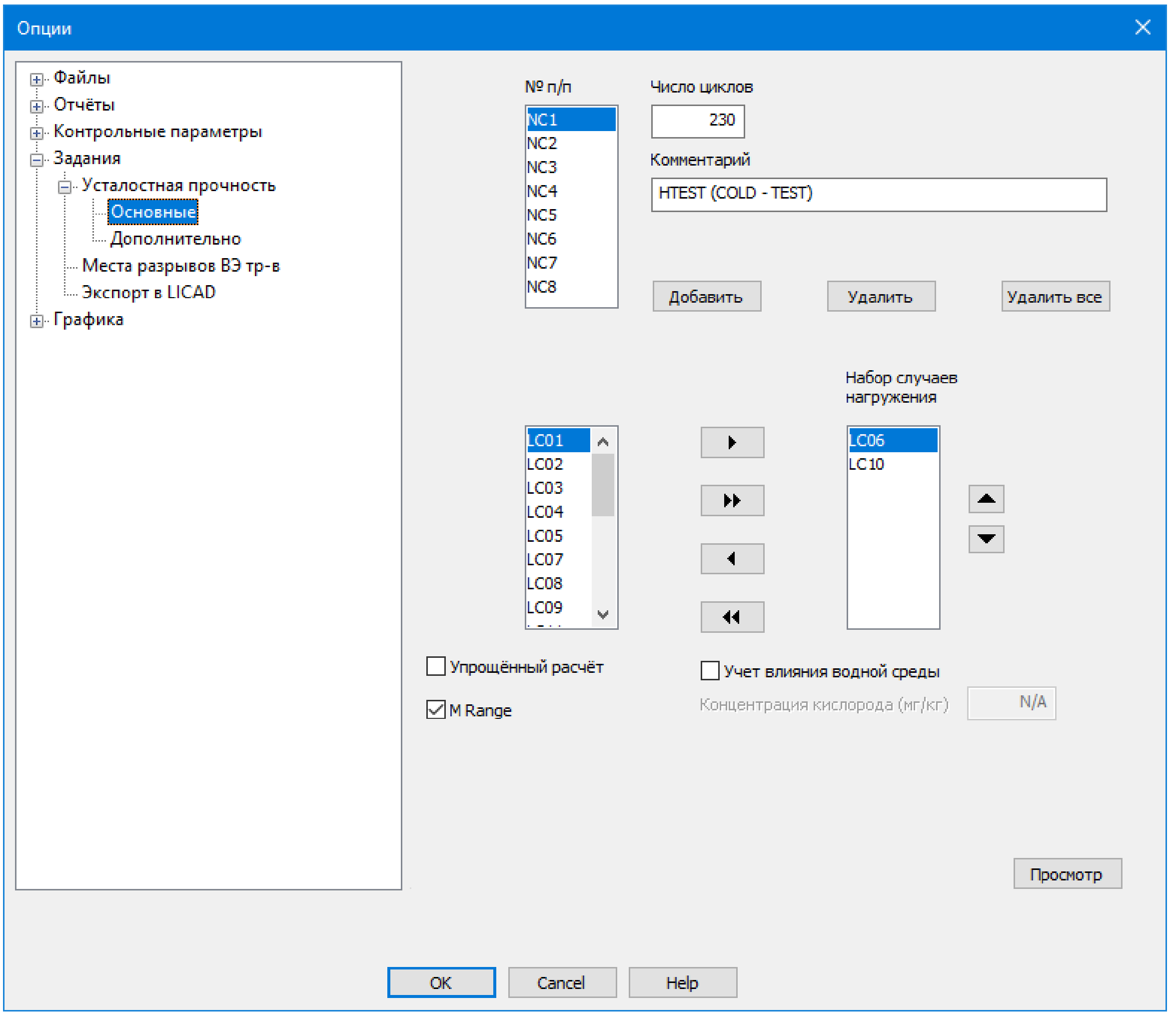
Specifications

This tab defines data required for FATIGUE analysis ("Fatigue analysis", FATG command) and High Energy Lines Break (HELB) analysis (POST_HELB command).

Fatigue Analysis

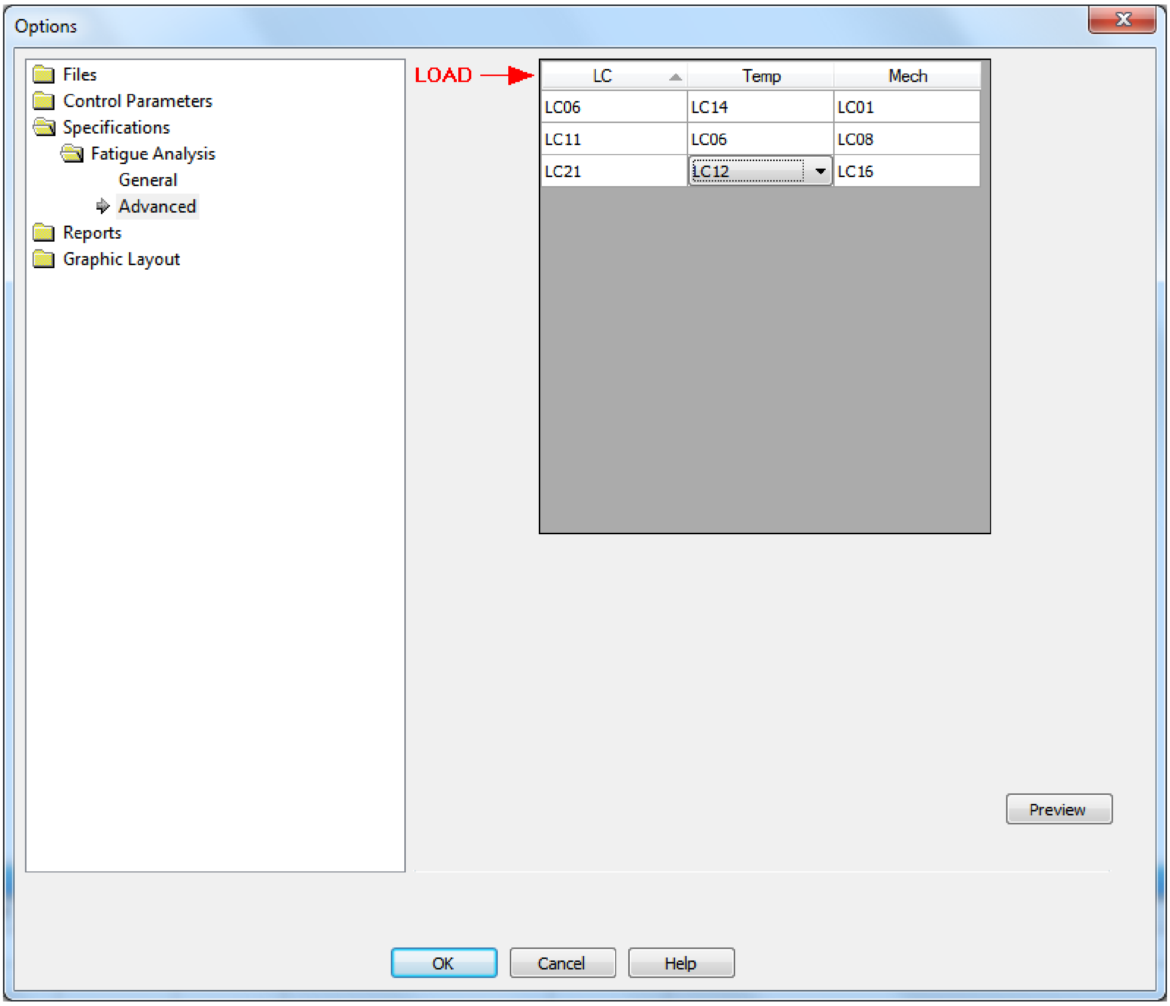
"Fatigue Analysis":

Dialog field	Command	Parameter
Load Cases/Load Sets	SOLV	LC
Number of cycles	FATG	NC
Loading history	FATG	SEQ
Simplified analysis	FATG	FATG_SAF
Option for the Range of Mechanical Loads	FATG	M_RANGE



Additional data for fatigue strength analysis:

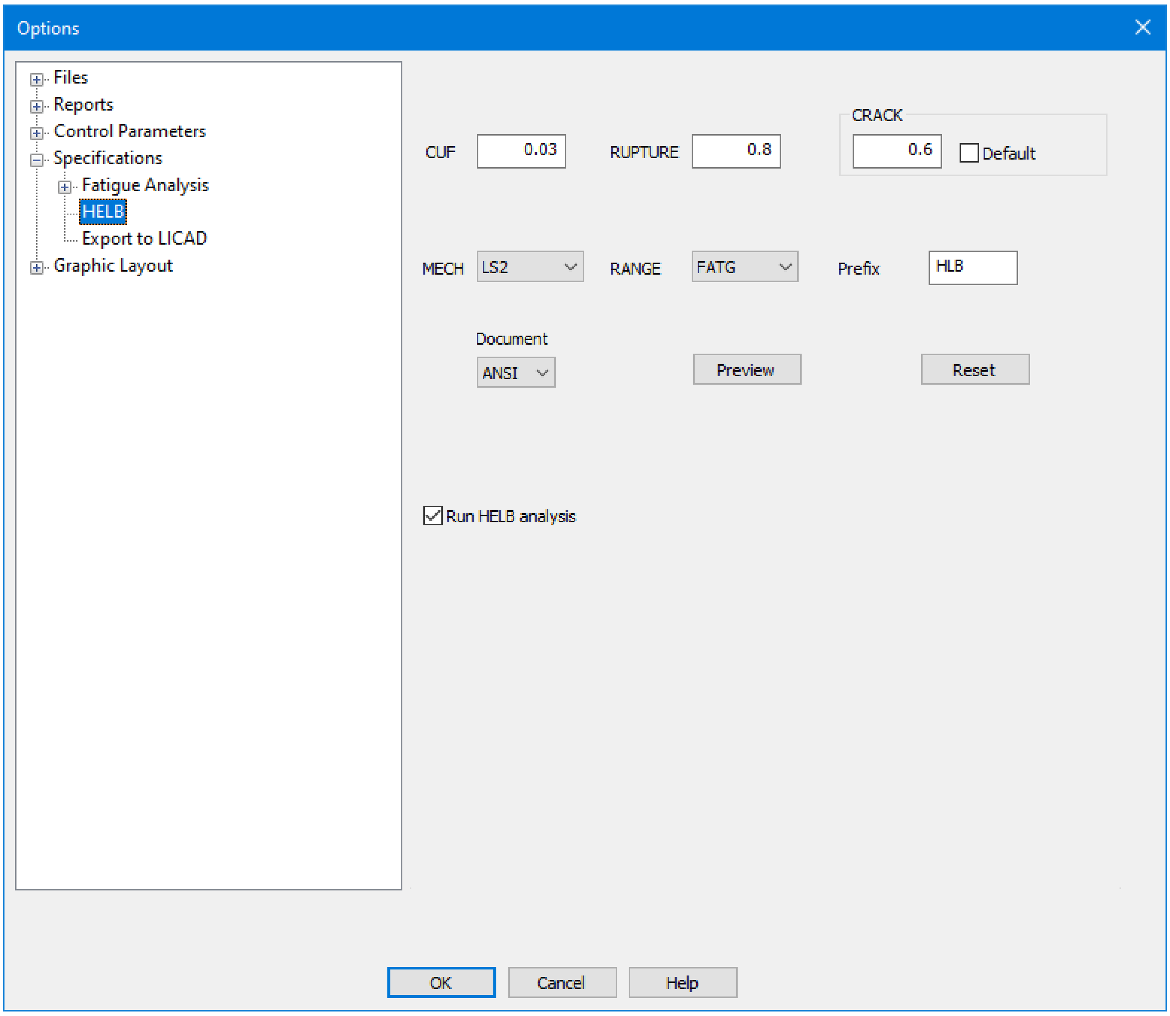
Dialog field	Command	Parameter
LC	FATG	LOAD
TEMP	FATG	TEMP
MECH	FATG	MECH



HELB

HELB tab specifies data for High Energy Line Breaks (HELB) analysis to define postulated ruptures and through-wall cracks.

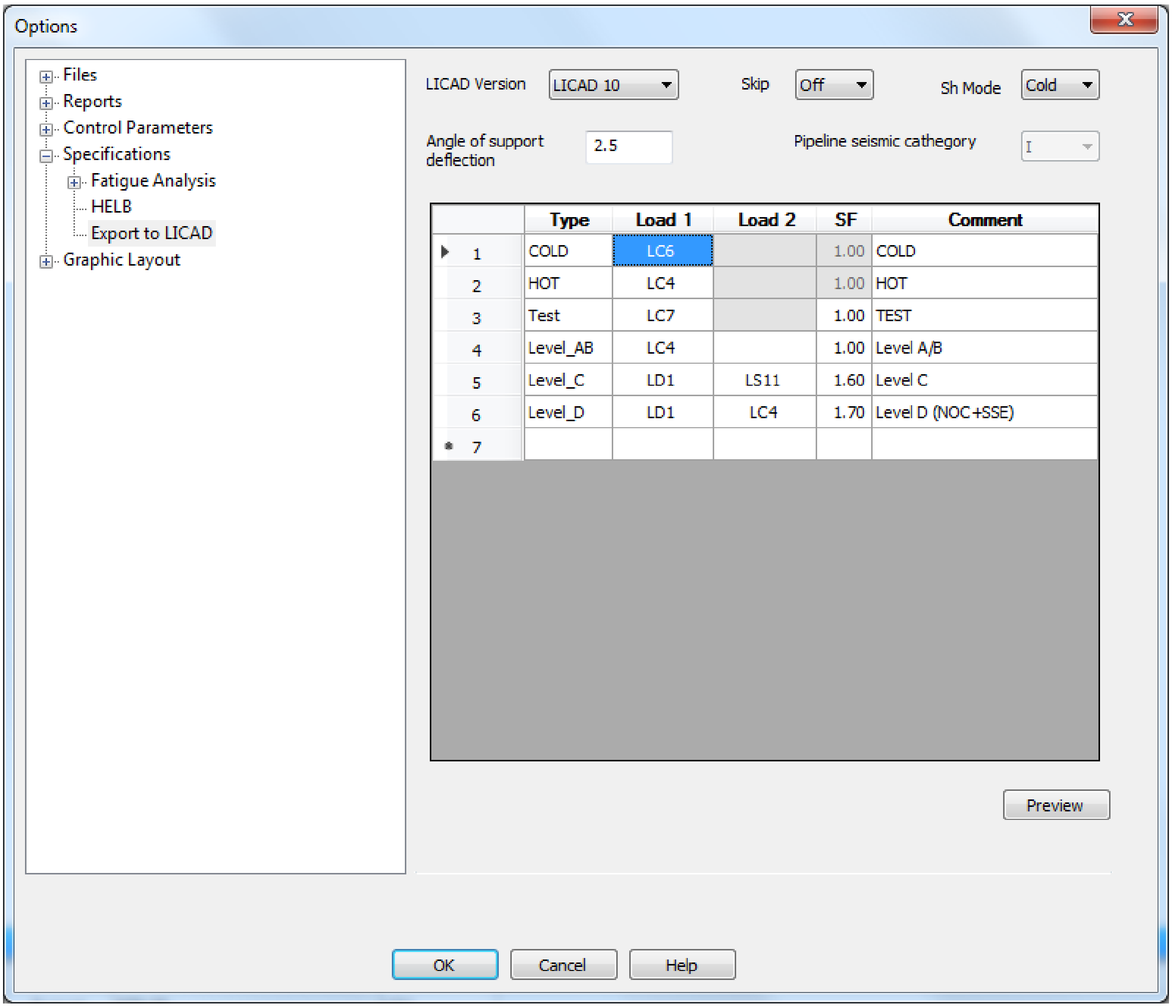
Dialog field	Command	Parameter
CUF	POST_HELB	CUF
RUPTURE	POST_HELB	RUPTURE
CRACK	POST_HELB	CRACK
MECH	POST_HELB	MECH_LS
RANGE	POST_HELB	RANGE_LS
Prefix	POST_HELB	HELB_STR
Document	POST_HELB	DOC



Export to LICAD

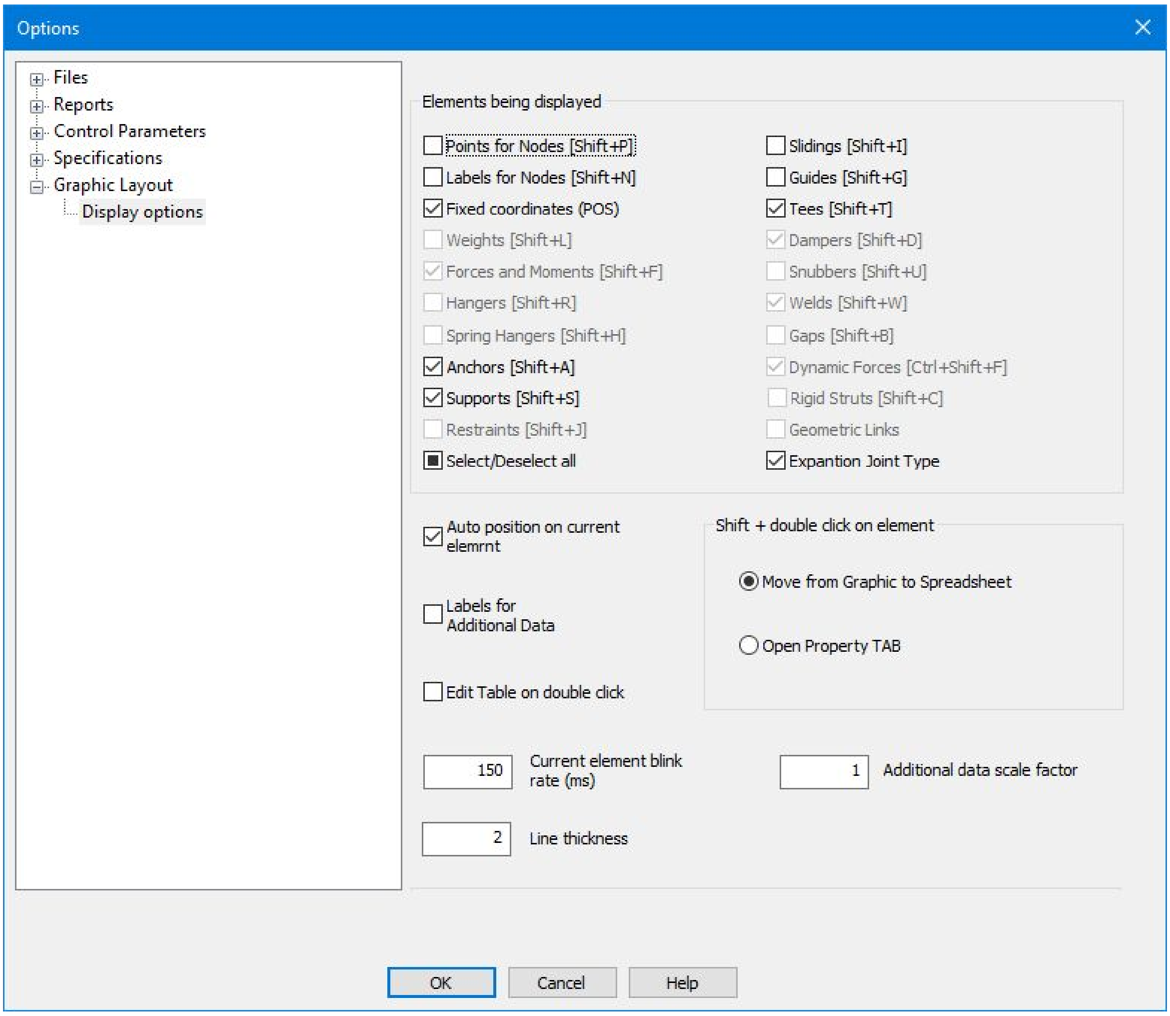
Export to LICAD tab: specification for the export of loads acting on piping supports from dPIPE to LICAD. See [Appendix XVI]

Dialog field	Command	Parameter
LICAD Version	DP2LCD	LCD_VER
Skip		SKIP
Sh Mode		SH_MODE
Angle of support deflection		ANGLE
Seismic caregory		S_CAT
Type		TYPE
Load 1		LOAD
Load 2		
SF		SF
Comment		NOTE



Display options

The "Display options" page is used to control the graphical representation of the model on Graphic Layout folder:



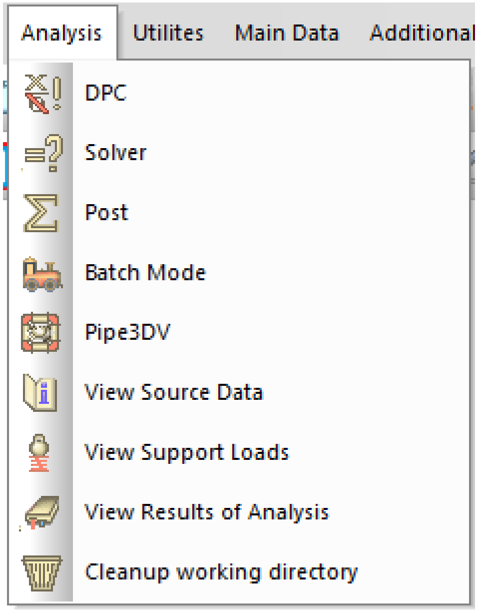
The list of "*Elements being displayed*" corresponds to the "additional data" set. By enabling/disabling the "***Labels for Extra Data***" flag it is possible to show/hide the labels of nodes of the elements displayed, which are related to nodes of the piping model. The flag of "***Auto position on current element***" during navigation over the spreadsheet allows synchronizing the movement over the spreadsheet and display of the model's segment being viewed. The "***Current element blink rate***" field allows regulating the blinking frequency of the element, on which the cursor is placed in the spreadsheet. In case when 0 is entered, the blinking will stop. "**Additional data scale factor**" allows entering the scaling coefficient in order to zoom in/out the additional data in the program window.

Watch Video

Activating the "*Edit tables on double-click*" option allows you to avoid the system "beep" signal when double-clicking on fields in the "Add. data", "Section" and "Load Group" columns of the "Layout" table. The "*Line Thickness*" field allows you to adjust the line thickness when displaying the piping model in line.

Analysis

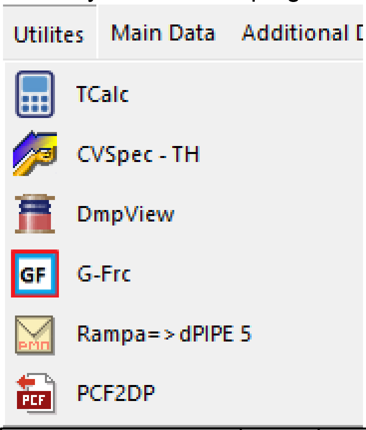
The "Analysis" menu item is a set of commands for the execution of analysis:



Menu Item	Icon on toolbar	Operation
DPC		start pre-processor and generate the input data listing
Solver		start the analysis module;
Post		start post-processor and generate the results;
Batch Mode		execute all the above-mentioned commanded in batch mode;
Pipe3DV		call the PIPE3DV program for viewing the piping model and the result;
View the input data listing		view file with input data listing (*.OUT)
Loads on supports		view file with summary tables of loads on supports, equipment and valves (*.SUP)
View the results listings		view file with printout of the results(*.RES)
Clear the working directory		Clear the working directory from temporary files to be created by the program by running the *clear.bat* file

Utilites

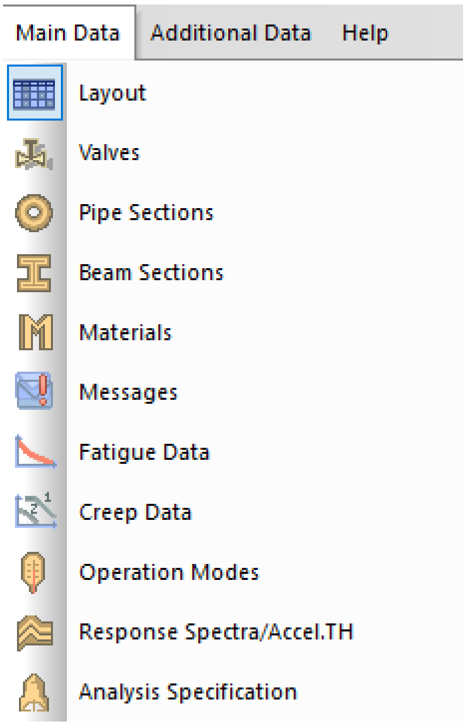
Auxiliary stand alone programs:



TCalc		Pressure Design Calculator (supports Russian PNAE and RD codes). May be used as a stand-alone application. Requires licensing for the full functionality mode.
CVSpec-TH		software for developing and processing the Seismic Response Spectra, as well for generation of artificial acceleration Time Histories, see: https://www.dpipe.ru/en/dpipe_utilities_en/cvspecth_en.html. May be used as a stand-alone application. Requires licensing for the full functionality mode.
DmpView		view and manage data and characteristics of viscoelastic dampers;
G-Frc		dPIPE utility for evaluation of the fluid forces arisen due to sudden pipe break according to the requirements of REF 19, 20. The program allows to evaluate thrust forces and Jet Impingement Effects. May be used as a stand-alone application. Requires licensing for the full functionality mode. See: https://www.dpipe.ru/en/dpipe_utilities_en/g-frc_en.html
RAMPA => dPIPE 5		Conversion of Input Data from the RAMPA program into the dPIPE5 data format (run the R2DP_N.exe program). In case of Input Data conversion from RAMPA-93 and RAMPA-90, the initial file shall have the extension of ".dat" and ".nml" respectively. In case of successful execution of the program, a file with the same name as the converted file will be created in the working directory but with the ".dp5" extension. For further operation this file shall be loaded into the DDE spreadsheet.
PCF2dP		The preprocessor module for converting data from PCF (Piping Component Files) into dPIPE models, see Appendix XIX.

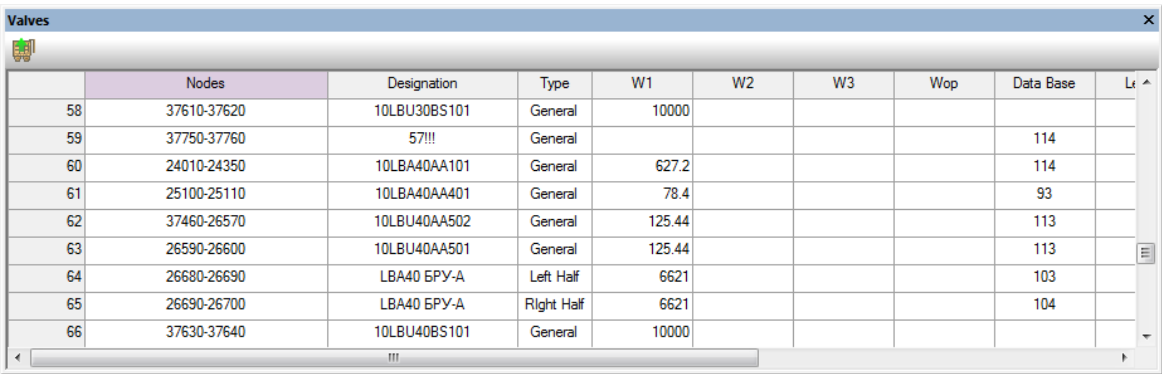
Main Data

The "**Main data**" menu allows to switch over between main active program windows:



Layout - geometry input window

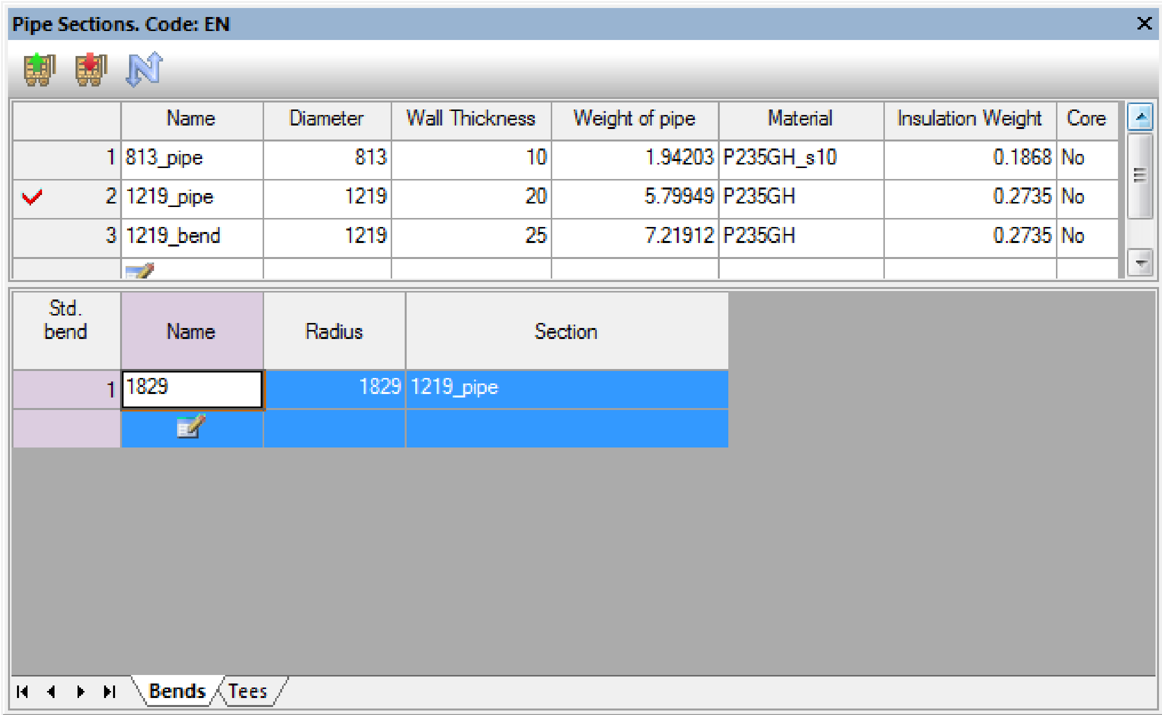
Valves – view and edit the characteristics of the valves (see V, V1, V2, VA, VO commands):


Valves

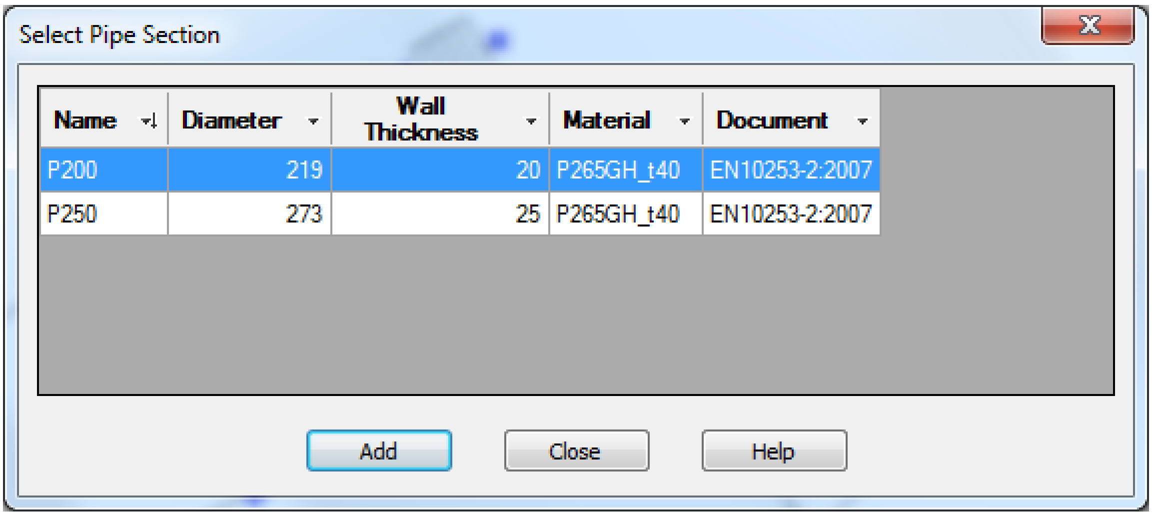
	Nodes	Designation	Type	W1	W2	W3	Wop	Data Base	Le
58	37610-37620	10LBU30BS101	General	10000					
59	37750-37760	57!!!	General					114	
60	24010-24350	10LBA40AA101	General	627.2				114	
61	25100-25110	10LBA40AA401	General	78.4				93	
62	37460-26570	10LBU40AA502	General	125.44				113	
63	26590-26600	10LBU40AA501	General	125.44				113	
64	26680-26690	LBA40 БРУ-А	Left Half	6621				103	
65	26690-26700	LBA40 БРУ-А	Right Half	6621				104	
66	37630-37640	10LBU40BS101	General	10000					

The window can also be activated by pressing the button. The button allows to set permissible loads on the valve's nozzle (see Appendix X). In order to call the individual dialog for each valve, the "Properties" item of the spreadsheet context menu is used (to be called by right click).

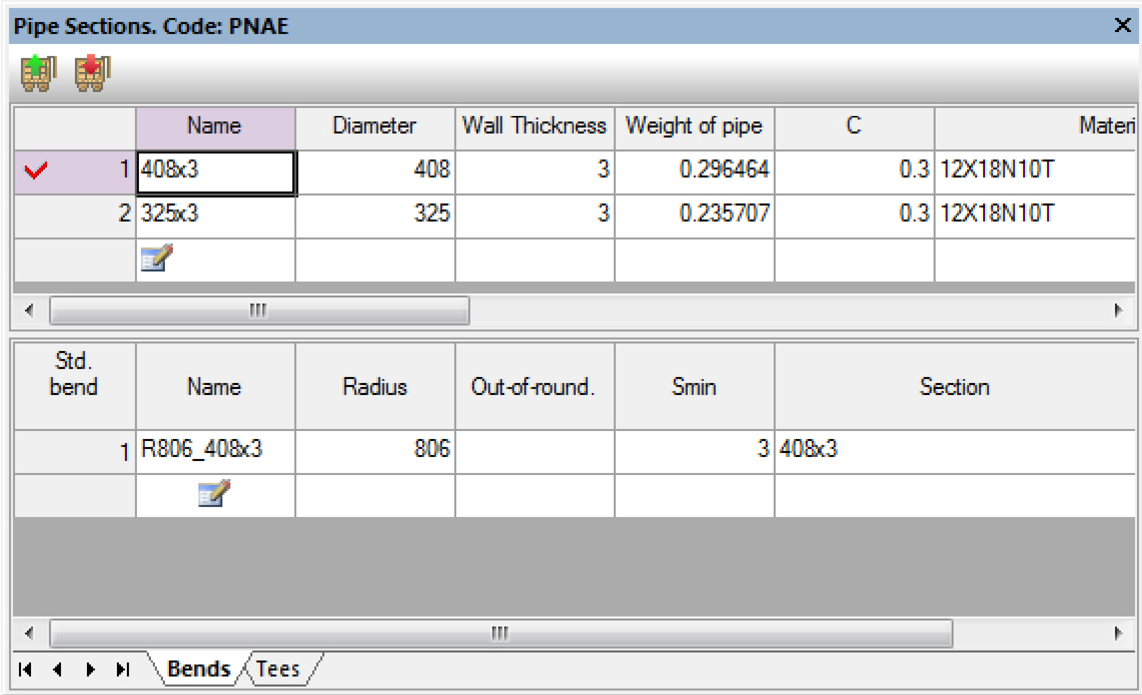
Pipe Sections – input of the characteristics of pipe cross-sections (see PIPE command):



The window can also be activated by pressing the button. In operations with this spreadsheet it is possible to add data from the database pipe.dbs (button) or export data to another database (button):



At the bottom of the window there are tabs for the piping fittings ("standard" elements), coupled with the current section of the pipe. Tab "Bends" corresponds to the parameter *BEND* :



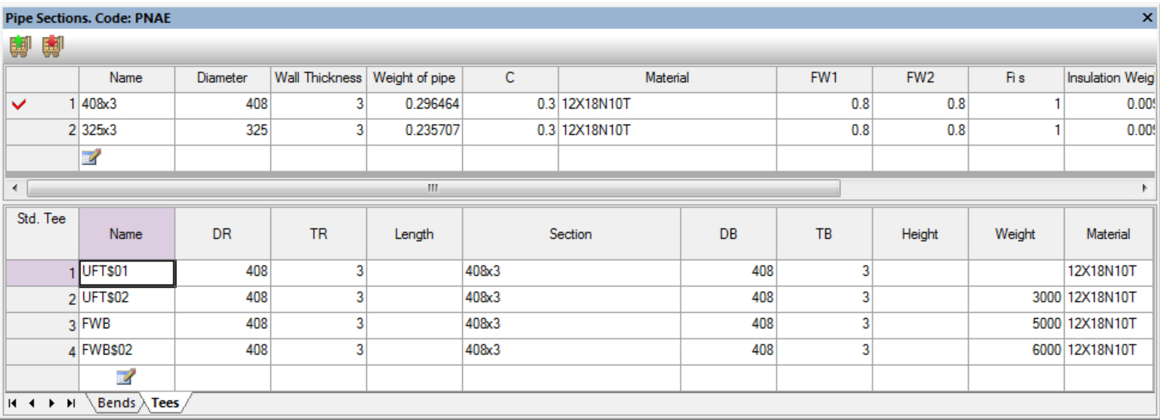
Pipe Sections. Code: PNAE

	Name	Diameter	Wall Thickness	Weight of pipe	C	Materi
1	408x3	408	3	0.296464	0.3	12X18N10T
2	325x3	325	3	0.235707	0.3	12X18N10T

Std. bend	Name	Radius	Out-of-round.	Smin	Section
1	R806_408x3	806		3	408x3

Bends | Tees

While tab "Tees" corresponds to the *TEE* parameter:



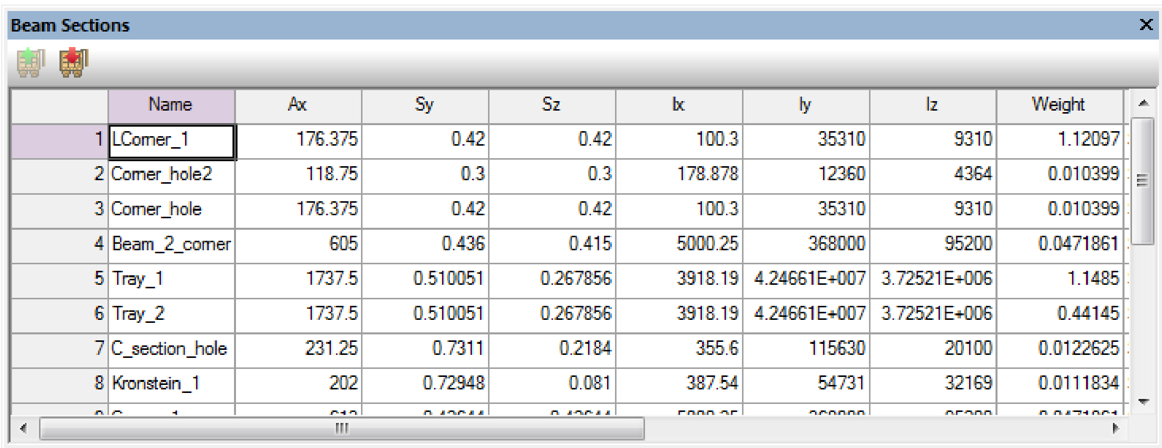
Pipe Sections. Code: PNAE

	Name	Diameter	Wall Thickness	Weight of pipe	C	Material	FW1	FW2	Fi s	Insulation Weig
1	408x3	408	3	0.296464	0.3	12X18N10T	0.8	0.8	1	0.00
2	325x3	325	3	0.235707	0.3	12X18N10T	0.8	0.8	1	0.00

Std. Tee	Name	DR	TR	Length	Section	DB	TB	Height	Weight	Material
1	UFT$01	408	3		408x3	408	3			12X18N10T
2	UFT$02	408	3		408x3	408	3		3000	12X18N10T
3	FWB	408	3		408x3	408	3		5000	12X18N10T
4	FWB$02	408	3		408x3	408	3		6000	12X18N10T

Bends | Tees

Beam Sections – input of the characteristics of cross-sections for beam elements (see BEAM command):

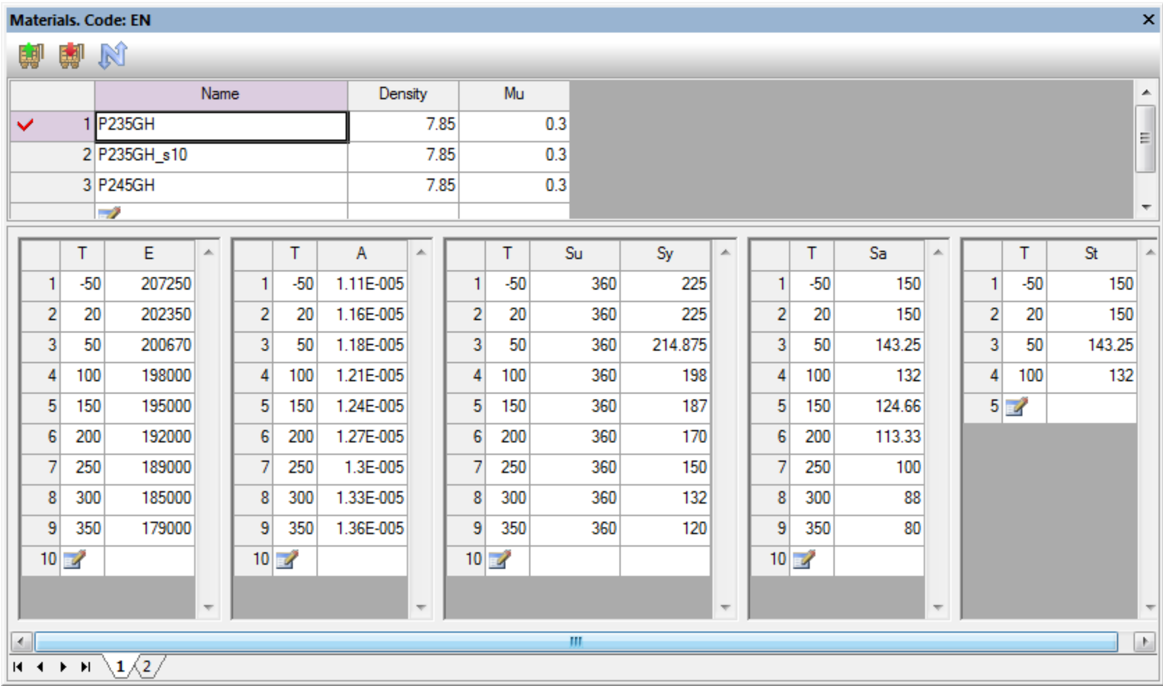


Beam Sections

	Name	Ax	Sy	Sz	Ix	Iy	Iz	Weight
1	LCorner_1	176.375	0.42	0.42	100.3	35310	9310	1.12097
2	Corner_hole2	118.75	0.3	0.3	178.878	12360	4364	0.010399
3	Corner_hole	176.375	0.42	0.42	100.3	35310	9310	0.010399
4	Beam_2_corner	605	0.436	0.415	5000.25	368000	95200	0.0471861
5	Tray_1	1737.5	0.510051	0.267856	3918.19	4.24661E+007	3.72521E+006	1.1485
6	Tray_2	1737.5	0.510051	0.267856	3918.19	4.24661E+007	3.72521E+006	0.44145
7	C_section_hole	231.25	0.7311	0.2184	355.6	115630	20100	0.0122625
8	Kronstein_1	202	0.72948	0.081	387.54	54731	32169	0.0111834

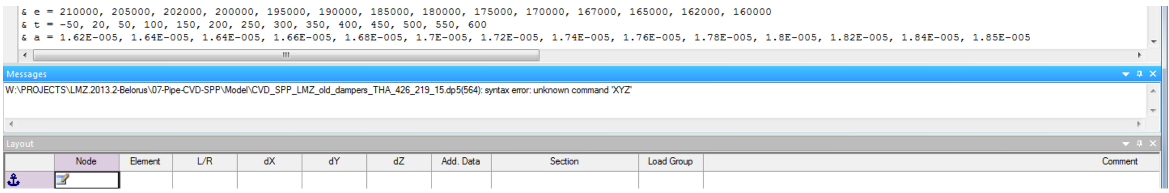
The window can also be activated by pressing the button. In operations with this spreadsheet, it is possible to add data from the database (button) or export data to another database (button).

Materials – input of the characteristics of materials to be used in piping model (see MAT command):

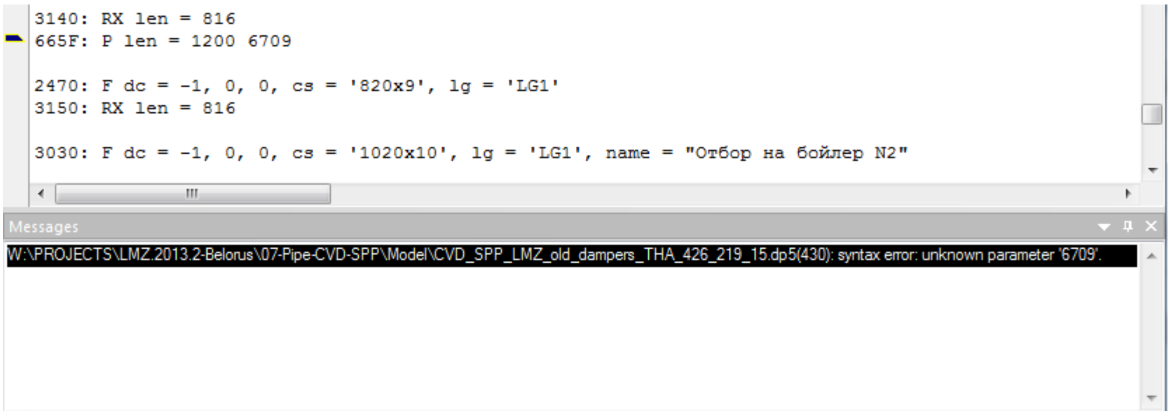


The window can also be activated by pressing the button. In operations with this spreadsheet, cross-sections can be added from the database (button) or export into another database (button).

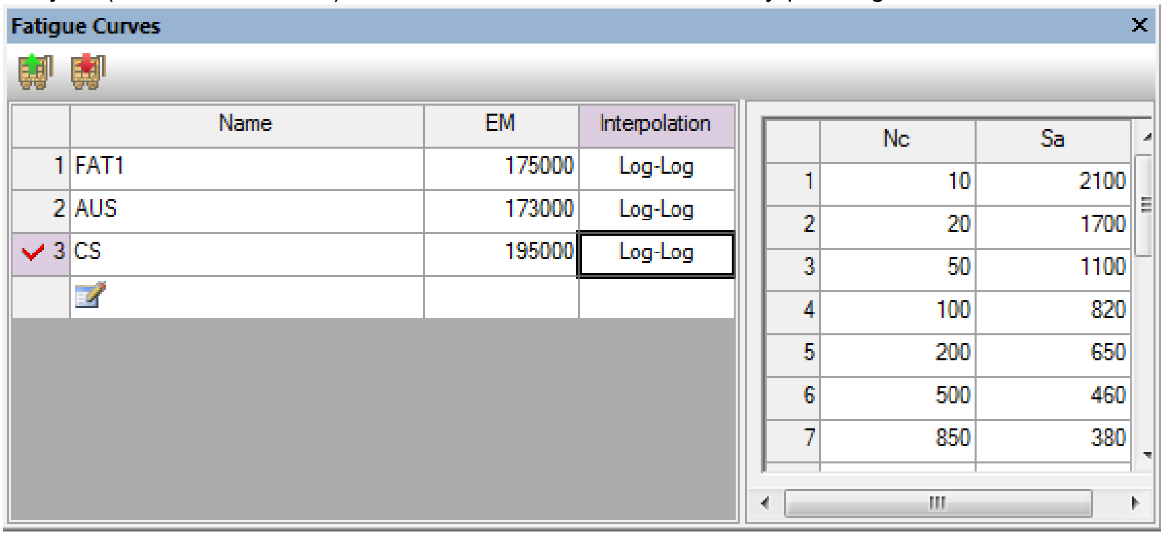
The "Messages" window contains information about errors arising in the course of reading the Input Data file. The window will be activated automatically in case when an error occurs:



Double click on the message line switches the graphics (main) window to the text mode and places the cursor on the line, in which the error is detected:

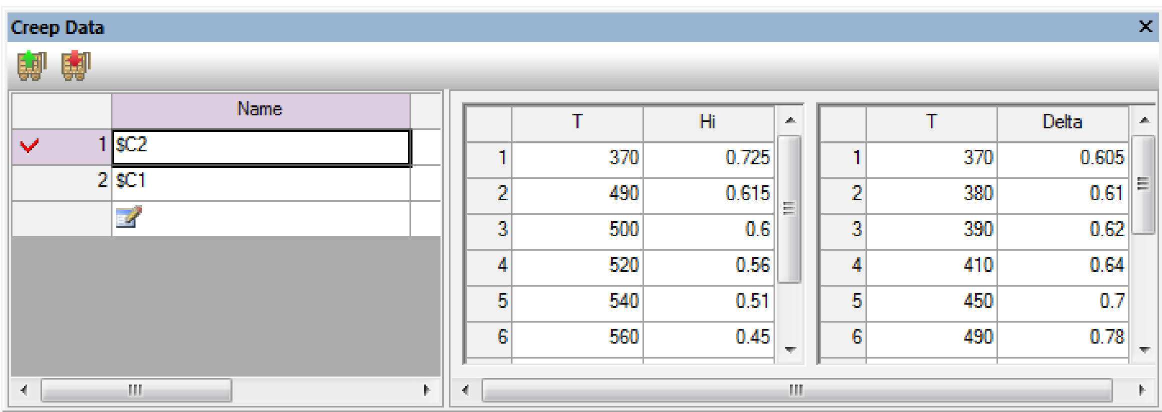


The "Fatigue Data" window contains information about the data being necessary for fatigue analysis (see FAT command). The window can also be activated by pressing the button:



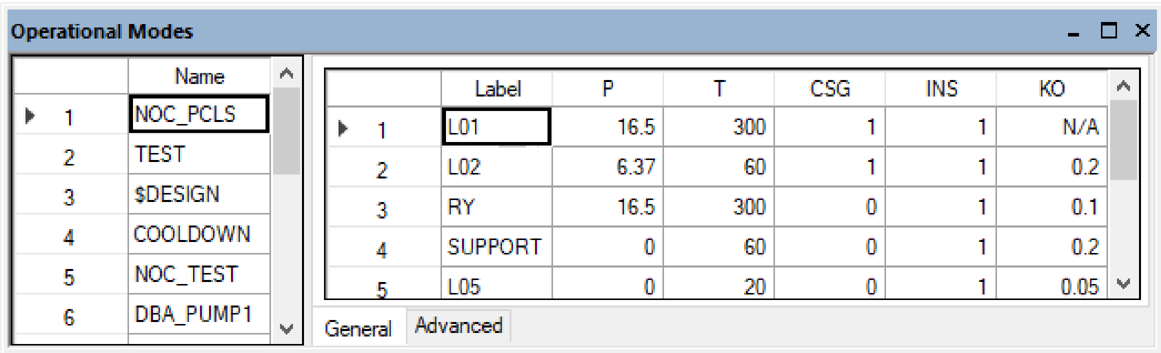
In operations with this spreadsheet, it is possible to add data from the database (button) or export data to another database (button).

- **The "Creep Data"** window contains information about the data being necessary for the analysis of high temperature piping systems (see CREEP commands). The window can also be activated by pressing the button:

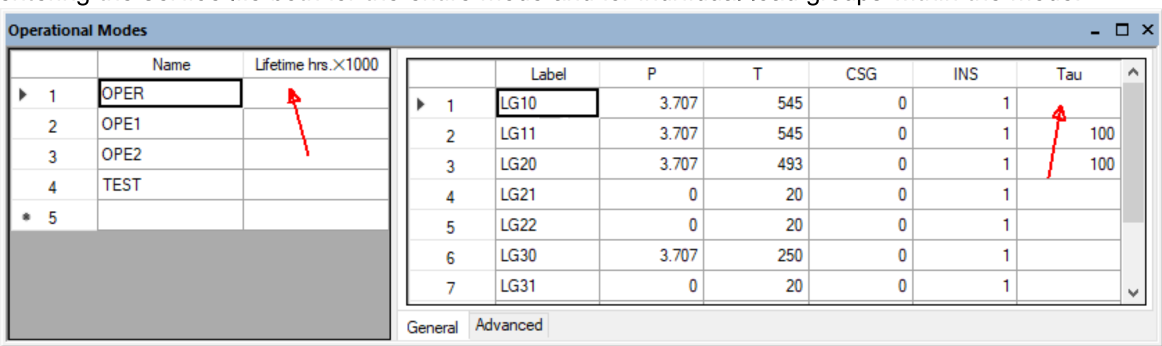


In operations with this spreadsheet, it is possible to add data from the database (button) export data to another database (button).

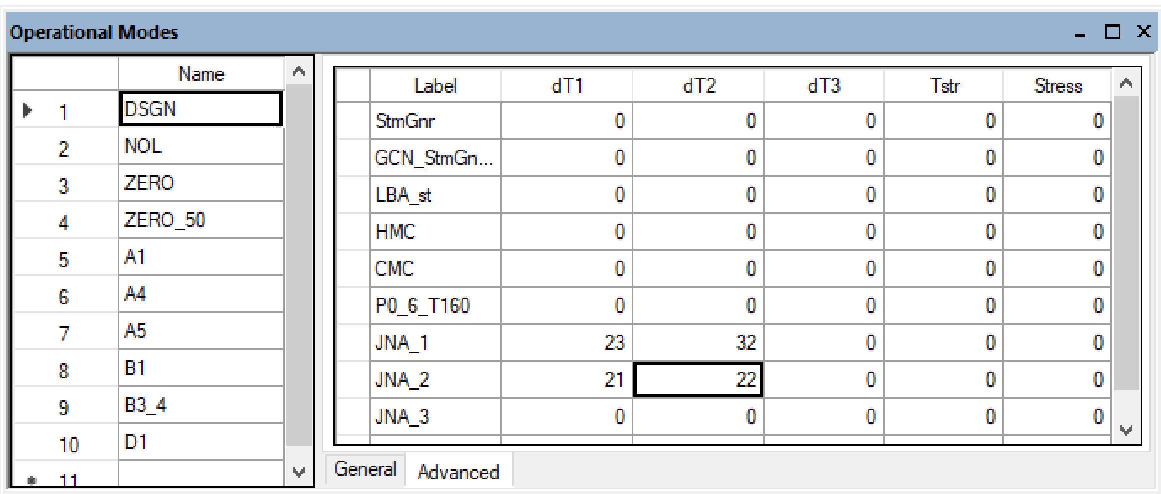
The **"Operation Modes"** window contains information about various operating modes of the piping system with specification of the load groups being assigned to the different piping segments (see OPVAL command). The window can be activated either from the menu or by pressing the button:



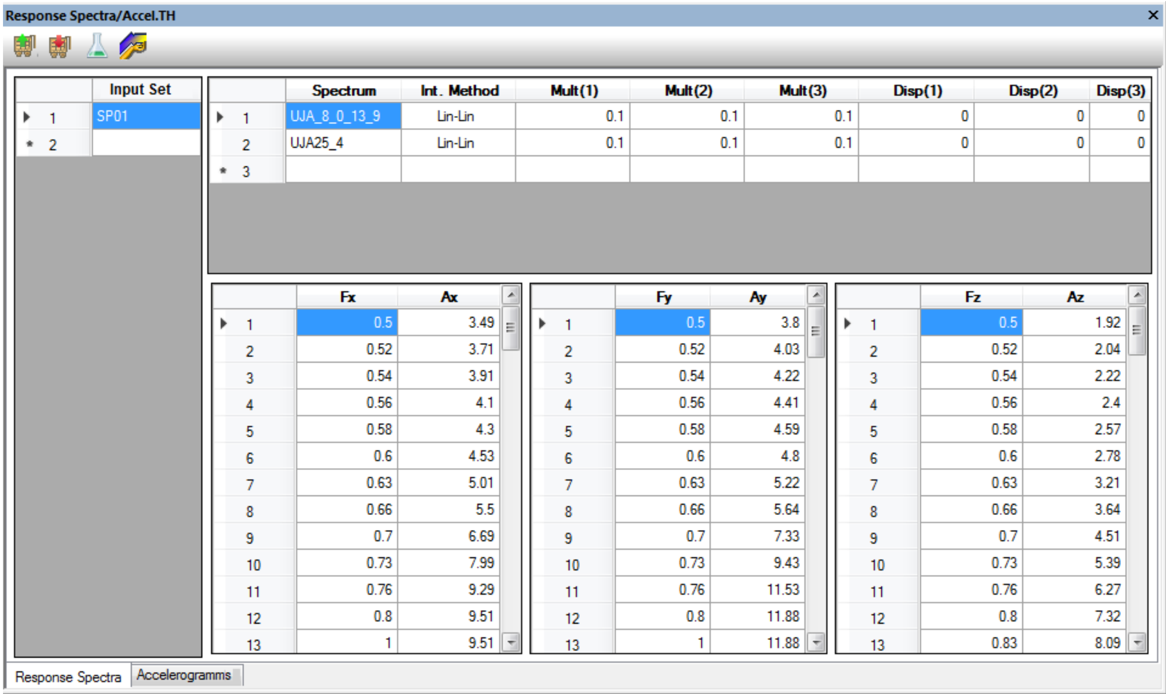
The type of dialog depends on the Code used for analysis. So, for those Codes that deal with elevated temperature piping (CODE='RD/EN/PNAE_T'), an additional fields become available for entering the service life both for the entire mode and for individual load groups within the mode:



The "Advanced" tab is used to enter the data required to calculate the stresses due to the temperature gradient across the wall thickness and the effect of stratification (GRAD command):



The "**Response Spectra/Accelerograms**" window contains data for the seismic input expressed in the form of response spectra (RSM analysis, see SPEC command), or accelerations time histories records (THA, ACCE command). The window can be activated either from the menu or by pushing the button:



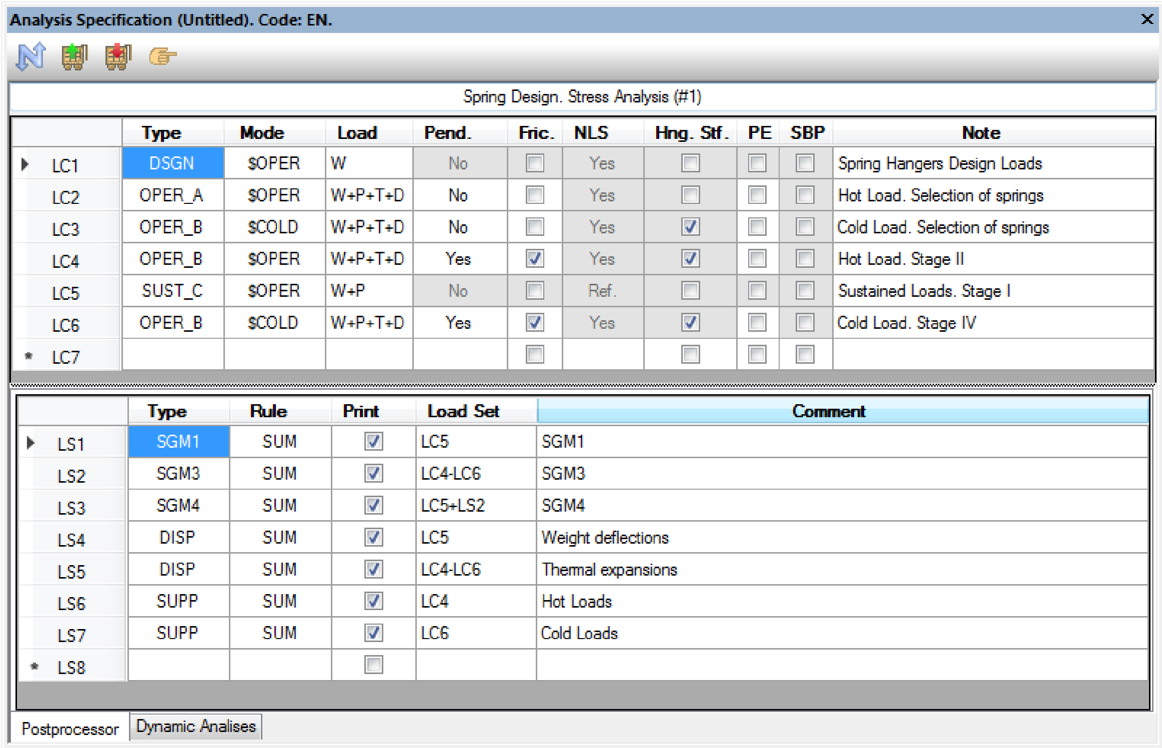
In order to add response spectrum from the existing text files, it is necessary to use the button. The files shall contain digital response spectrum in the "frequency - acceleration" format. Vice versus, for the export of the spectrum data to the external file the button should be used. In this case, instead of digital data, the relative or absolute links will be written in the *.dP5 file.

All accelerations scaled by "mult" should be expressed in g (gravity)!!!

The button is used to launch CVSpec-TH program for viewing and processing of the response spectra, and acceleration time history records. The program also can generate artificial acceleration time histories from the given response spectra.

Watch Video

The **"Analysis Specification"** window is used to specify the sequence of the required analyses (Load Cases) and post-processor directives (Load Sets). The input shall be performed in the tabular form according to instructions given for SOLV, POST and DCASE commands:



Analysis Specification (Untitled). Code: EN.

Spring Design. Stress Analysis (#1)

	Type	Mode	Load	Pend.	Fric.	NLS	Hng. Stf.	PE	SBP	Note
LC1	DSGN	$OPER	W	No	☐	Yes	☐	☐	☐	Spring Hangers Design Loads
LC2	OPER_A	$OPER	W+P+T+D	No	☐	Yes	☐	☐	☐	Hot Load. Selection of springs
LC3	OPER_B	$COLD	W+P+T+D	No	☐	Yes	☑	☐	☐	Cold Load. Selection of springs
LC4	OPER_B	$OPER	W+P+T+D	Yes	☑	Yes	☑	☐	☐	Hot Load. Stage II
LC5	SUST_C	$OPER	W+P	No	☐	Ref.	☐	☐	☐	Sustained Loads. Stage I
LC6	OPER_B	$COLD	W+P+T+D	Yes	☑	Yes	☑	☐	☐	Cold Load. Stage IV
LC7					☐		☐	☐	☐	

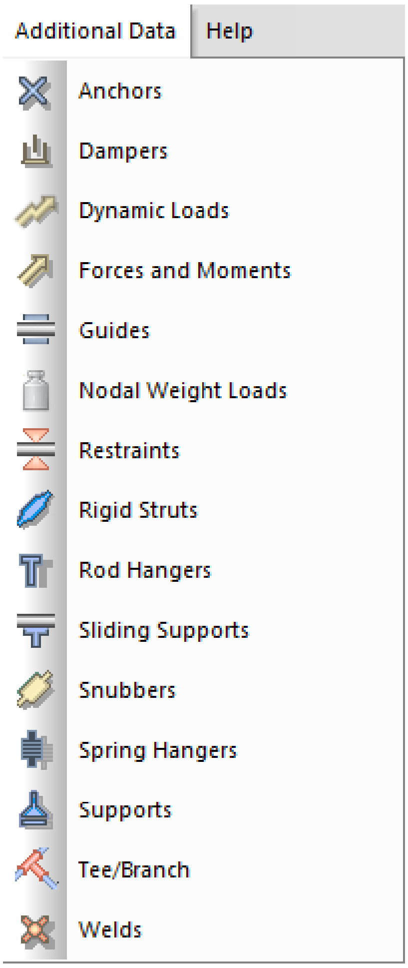
	Type	Rule	Print	Load Set	Comment
LS1	SGM1	SUM	☑	LC5	SGM1
LS2	SGM3	SUM	☑	LC4-LC6	SGM3
LS3	SGM4	SUM	☑	LC5+LS2	SGM4
LS4	DISP	SUM	☑	LC5	Weight deflections
LS5	DISP	SUM	☑	LC4-LC6	Thermal expansions
LS6	SUPP	SUM	☑	LC4	Hot Loads
LS7	SUPP	SUM	☑	LC6	Cold Loads
LS8			☐		

Postprocessor | Dynamic Analises

The buttons in the dialog header allow defining the Stress Evaluation Code (CODE), import the standard calculation's set from the solv.dbs file, export the user-defined set to the custom's database and viewing the commands in the text form:

Additional data

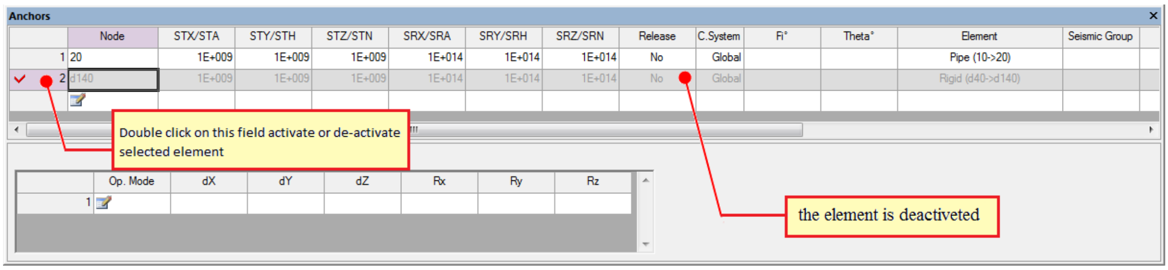
The next group of windows contains the data related to the Nodes of Piping Calculation Model presented in the tabular form. The windows can be opened from the drop-down menu (**"Additional data"** item):



The menu items and dPIPE commands are correlated as follows:

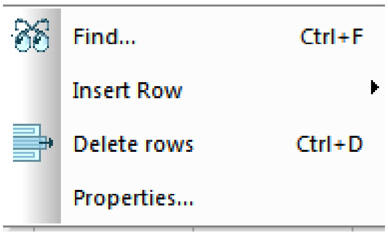
Anchors	ANC
Dampers	DMP
Dynamic Loads	DFRC
Forces and Moments	FOR
Guides	STG, STG-
Nodal Weights	CW
Restraints	STS, SRS, STS+/-
Rigid Struts	STRT
Rod hangers	ROD
Sliding Supports	STZ, STZ-
Snubbers	SNUB
Spring Hangers	SPR
Supports	SUP
Tees/Branch	TEE
Welds	WLD

In operations with "additional data" presented in the tabular form, there is the possibility to "disable" them within the analysis performed. For this purpose, it is necessary to double click over the grey field with the sequential number of element:



In doing so, the corresponding line in the Input Data file is commented by two ";;" signs. Any subsequent inclusion of elements in Piping Calculation Model occurs similarly. Such working method is convenient in variants calculations.

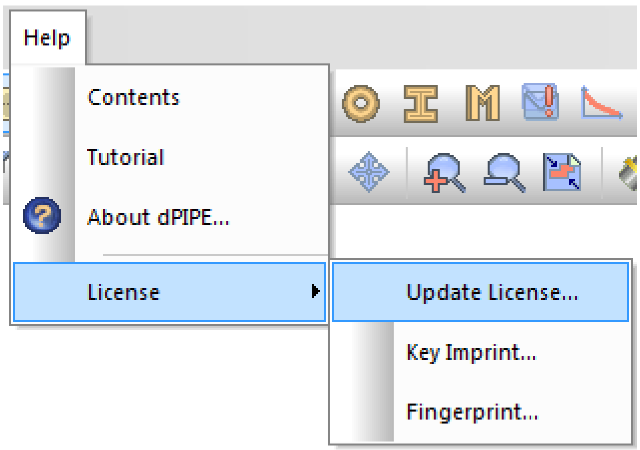
Additional data are presented in the form of summary tables that display the most important fields of their properties. The "Properties" item of the context menu (to be called by right click) in any of spreadsheets with additional data allows opening the dialog with full properties of the component considered:



Summary tables can be copied to the clipboard. The command CTRL-C copies only the contents of the table. The command CTRL-Shift-C copies the contents of the table with the heading row of the table. Clipboard contains also commented out (disabled) elements. In this case the last column of the copied table will contain symbol "!".

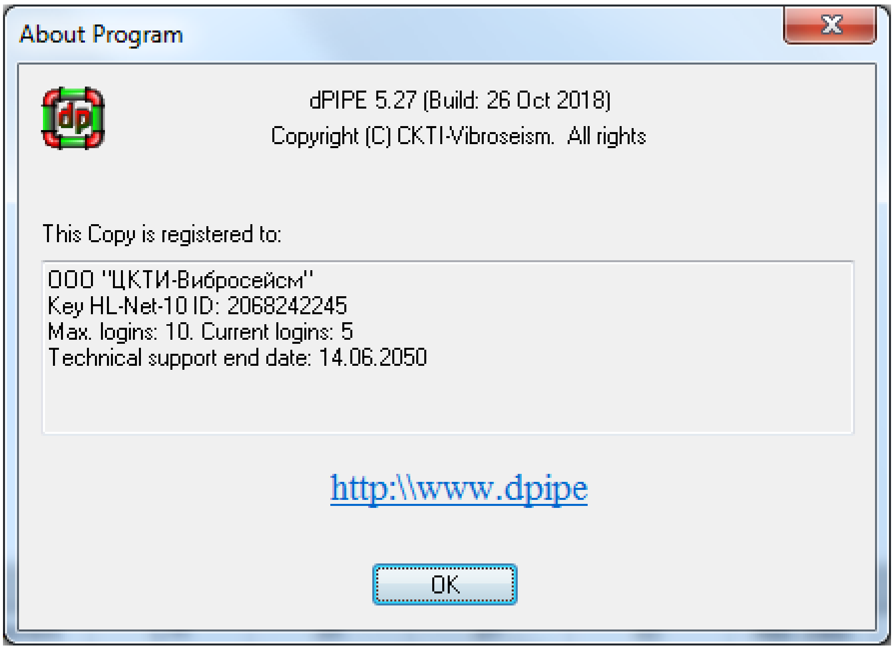
Help

The following topics are available under this menu item:



- Contents: lunch help file for dPIPE 5

- Tutorial: lunch help with dPIPE Tutorial
- About dPIPE: shows a window with information about the program's release and build, used protection key and number of used licenses:



The item "License" contains sub-items for working with security keys (see the section "Protection keys and License Management")

8 Input data language

The language for description the input data for the dPIPE 5 program consists of commands. The commands can contain internal subcommands as well as parameter subcommands and parameters, which values can be individual values or arrays.

Types of commands

The commands describe both general data relating to the whole model or a part thereof and local data referenced to a particular node or elements of the analysis model. Depending upon the availability of internal subcommands, general commands can be divided into one-line commands (without internal subcommands) and multi-line commands containing internal subcommands or parameter subcommands. Each command (subcommand) shall be located in a separate line. Before specification of a local command it is necessary to indicate the label of the node with subsequent colon.

Parameter values

The following types of parameter values can be distinguished:

- Numbers – integers (hereinafter designated as ***INTEGER***) and real numbers (hereinafter designated as ***REAL***). Real values can be written both in real form and in exponential form.

```
Example :  3, .3,-3.3, 3.2e-2
```

- Text – a set consisting only of alphanumeric characters enclosed in simple quotes (hereinafter designated as ***TEXT***). The maximum length of the text type parameter value is limited by 16 characters. In the text data, the upper case letters and the lower case letters are equal. No space or tab are permitted inside the text data. The following restrictions apply when entering text data in DDE:

a) only letters of the Latin alphabet are allowed;

b) except letters the following symbols are allowed: **#** **$** ***** **_**

Example: `'108x9', '08H18N10T'`

- String – a set of characters enclosed in double quotes (hereinafter designated as ***STRING***). The maximum length of the STRING type data is limited by 64 characters. In the string type data, the upper case letters and the lower case letters are different. The availability of spaces and special characters is permitted.

```
Example:  "10RA01 piping system"
```

Node labels

The analysis model node labels are text variables with the length up to 8 characters. In describing the node marks, the upper case letters and the lower case letters are equal.

Delimiters

The following characters are used as delimiters between the data being entered: «_» spaces, «,» commas or tabulation characters. Several spaces and/or tabulation characters entered successively shall be interpreted as one delimiter. A comma shall directly follow the magnitude entered. It is allowed to use the «=» character between the name of parameter and its value.

Special characters and commands

";" is the comment character. All information after the «;» character up to the end of the current line will not perceive by the program; the «;» character is followed by the transfer to the next line.

"\", the backslash character, is used for delimiting lines in the command. Pre-processor "glues" the previous and next lines and interprets several lines as one line. It should be kept in mind that the «;» comment character can immediately follow the line delimitation character.

```
Example :
  T =   20  50  100  150   200   250   300   350   400  450   500   550
  600

  is equivalent to

  T =   20  50  100  150   200   250   300   350   400   \
  450   500   550   600
```

& is the symbol of the beginning of a subcommand or a parameter command. It is used only in multi-line commands and shall be placed directly before the name of subcommand (parameter command).

Parameter input sequence

In case of explicit indication of the name, the parameters inside one command (subcommand) can be entered in arbitrary order. It is allowed to enter parameters without name provided that the parameters entered are passing in the order determined herein. As soon as a named parameter is encountered within one command, all subsequent parameters shall be entered with their names.

Language commands

The description of commands of the input data entry language is given below. The parameters or subcommands, for which no default value has been determined, shall be mandatory. They are highlighted in the text in red bold type, for example, **T** .

System of Units

dPIPE uses a consistent system of units. Basic Newton (N) and millimeter (mm) are used by default. All other units are derived from these two:

displacements, sizes, thickness, etc: mm;
accelerations: in parts of g (acceleration of gravity);
forces: Newton
rotation angles: radians or degrees, (as specified in this manual)
pressure, stresses: N/mm^2 or MPa;

The following designations have been used in the text hereof:

rigid – "absolutely rigid" (the stiffness value is determined in accordance with the RGD_TRN and RGD_ROT parameters)
blank – "empty line"

General commands

Local commands

General commands

Insert data from another file (INCLUDE)

Analysis title (TITLE)

Control parameters (CTRL)

Cyclic strength curves (FAT)

Curves for high temperature piping system analysis (CREEP)

Materials (MAT)

Pipe cross-section characteristics (PIPE)

Beam element cross-section characteristics (BEAM)

Piping system operating modes (OPVAL)

Stresses from the temperature drop over the wall thickness and from the stratification effect (GRAD)

Data for spring supports (SDEF)

Seismic response spectrums (SPEC)

Seismic load accelerograms (ACCE)

Specification for Analysis (SOLV)

Specification for post-processing (POST)

Specification for fatigue strength analysis (FATG)

Specification for report generation (POST_REP)

Managing the databases connected (DBF)

Suppressing results by headers ($NOHEAD)

End of input data (END_OF_DATA)

Insert data from another file (INCLUDE)

The INCLUDE command allows connecting the data contained in another file to the file with input data. The command syntax is as follows:

```
INCLUDE  "file"
```

file *is the file name and its path written in double quotes. If the file is indicated without path, then the program attempts to find it either in the current directory or in the directory, in which the program is installed.*

Model title (TITLE)

type: general one-line command

Function: title for printout

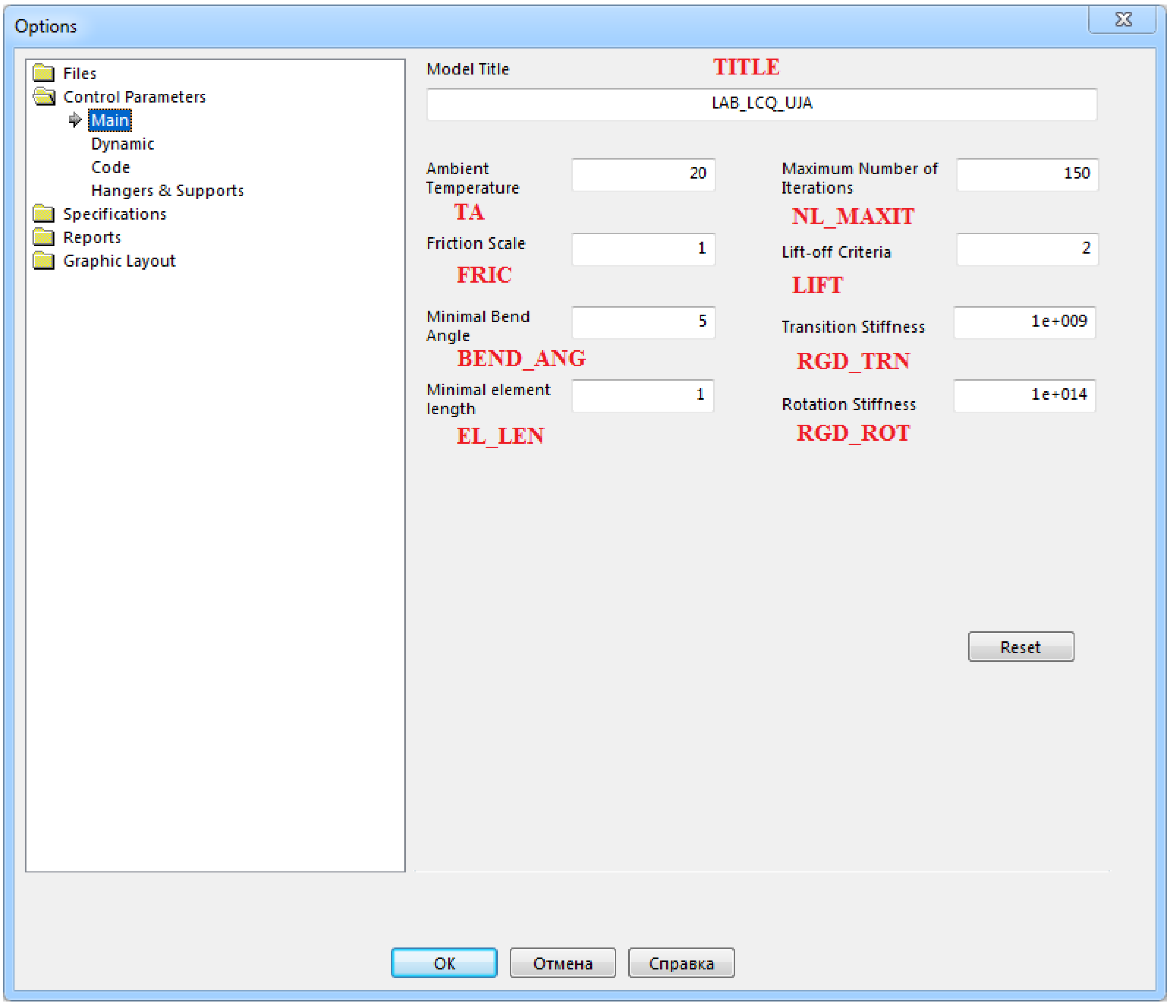
Parameters:

"text" title for printout

type:	STRING
unit:	-
default value:	blank
limitations:	-

```
Example:    TITLE "Feed water piping analysis"
```

Control parameters (CTRL)



type: general one-line command

Function: determine control parameters for analysis

Parameters:

TA is the piping system installation (ambient) temperature. It is used for determination of the characteristics of piping system in the "cold" state. If it is required to determine the installation temperature of various piping segments, the mode with the '$INST' pre-defined name (OPVAL command) should be used, in which it is possible to set the individual temperature for each load group.

type:	REAL
unit:	°C
default value:	20°C
limitations:	from -50°C to +60°C

NC is the number of cycles. It is used for fatigue analysis.

Type:	INTEGER

Unit:	-
Default value:	see Default values Table
Range of possible values:	from 1 to $1*10^7$

DYN is the dynamic analysis flag

Type:	TEXT
Unit:	-
Default value:	'NO'
Range of possible values:	'NO', 'RSM', 'THA'

'NO' - dynamic analysis is not performed;
'RSM' – seismic load analysis by the Response Spectrum Method,
'THA' – dynamic load method by the Time History Analysis method (time integration of the equations of motion)

FMAX is the cutoff frequency, up to which the natural frequencies of the piping system will be computed

Type:	REAL
Unit:	Hz
Default value:	33
Range of possible values:	>0

BEND_CODE - option for computing the bend's flexibility factor

Type:	TEXT
Unit:	-
Default values:	see Default values Table
Range of possible values:	'PNAE/RD', 'ASME', 'CASE'

'PNAE/RD' – *flexibility factor to be computed according to Russian PNAE [REF 1] and RD 10-249-98 [REF 2],*
'ASME' – analysis according to ASME codes [REF 3];
'CASE' – determination of the factor according to ASME CODE CASE N-319-3

FRIC - scaling factor to change friction coefficient in supports

Type:	REAL
Unit:	-
Default values:	1
Range of possible values:	≥ 0

By means of this factor, it is possible to change the value of MU friction coefficients in all one-component or guide supports. It is recommended to use it for evaluating the effect of friction forces upon the results.

BEND_PRES - option to take into account pressure for the bend's flexibility factor

Type:	TEXT

Unit: -
Default values: see Default values Table
Range of possible values: 'YES', 'NO'

T_REF - reference temperature, from which the Coefficient of Linear thermal expansion is set. It is also used for determination of the E_ref modulus of elasticity (see FAT command)

Type: REAL
Unit: °C
Default values: 20
Range of possible values: from -50°C to +60°C

W_DEN - density of water

Type: REAL
Unit: N/mm^3
Default values: $9.80665*10^{-6}$
Range of possible values: ≥ 0

The W_DEN value is used for conversion of the media weight in accordance with the CSG parameter value of the OPVAL command.

RGD_TRN - the parameter corresponding to the "absolutely rigid" definition (RIGID) for translational degrees of freedom

Type: REAL
Unit: N/mm
Default values: $1*10^9$
Range of possible values: ≥ 0

RGD_ROT - the parameter corresponding to the "absolutely rigid" definition (RIGID) for rotational degrees of freedom

Type: REAL
Unit: N*mm/rad
Default values: $1*10^{14}$
Range of possible values: ≥ 0

PSHEAR - option for the shear for straight pipe (0 - no shear allowance, /= 0: shear coefficient is taken equal to 2)

type: REAL
unit: -
default value: 2
limitations: ≥ 0

V_STF - a multiplier for the wall thickness in valve modeling

type: REAL
unit: -

default value:	3
limitations:	≥ 1

NL_MAXIT - the maximum number of iterations in non-linear computations

type:	INTEGER
unit:	-
default value:	99
limitations:	≥ 1

NL_FTOL - the accuracy in determination of the friction forces

type:	REAL
unit:	in fractions of 1
default value:	0.01 (corresponds to 1 %)
limitations:	≥ 0

NL_RTOL - the accuracy in determination of the non-linear support reaction force

type:	REAL
unit:	in fractions of 1
default value:	0.01 (corresponds to 1 %)
limitations:	≥ 0

NL_STOL - the threshold deformation value, after which the support sliding begins

type:	REAL
unit:	mm
default value:	0.1
limitations:	≥ 0

GRAV - dimensional value of the mass acceleration

type:	REAL
unit:	mm/s^2
default value:	9806.65
limitations:	> 0

FREQ_TOL the accuracy of eigenvector determination

type:	REAL
unit:	-
default value:	$1*10^{-5}$
limitations:	> 0

E_MOD - modulus of elasticity to be used for generation of the stiffness matrix (either as to the hot state – 'HOT', or as to the T_REF temperature – 'REF')

type:	TEXT

unit: -
default value: see [Default values] Table["по умолчанию"]
limitations: 'HOT', 'REF'

CODE - piping stress analysis codes

type: [TEXT]
unit: -
default value: 'PNAE'
limitations: 'PNAE', 'PNAE_T', 'RD', 'ASME_NC' , 'ASME_NB', 'EN', 'ASME_B311', *'NTD_ACI'*

The CODE parameter defines the selection of the Strength analysis codes:

- *'PNAE':*
  [CODE_YEAR] = 1986: analysis of low-temperature piping according to PNAE Code [REF 1];
  [CODE_YEAR] = 2022: analysis according according to GOST R 59115.9-2021 [REF 24] and GOST R 59115.15-2021, Appendix A [REF 25];
- *'PNAE_T' – analysis of high-temperature piping systems according to PNAE Codes [REF 1];*
- *'RD' – analysis of piping systems according to RD 10-249-98 codes [REF 2];*
- *'ASME_NC' - analysis of piping systems according to ASME NC-3600 (Class 2) codes [REF 3]*
- *'ASME_NB' - analysis of piping systems according to ASME NB-3600 (Class 1) codes [REF 3]*
- *'EN' - analysis of piping systems according to EN 13480-3 European codes [REF 10]*
- *'ASME_B311' - analysis of piping systems according to ASME B31.1 codes [REF 12]*
- *'NTD_ACI' - analysis of piping systems according to Czech Code NTD A.C.I. [REF 17]*

CODE_YEAR - year of publication of the codes (edition)

type: [INTEGER]
unit: -
default value: see [Default values] Table
limitations: ↓

CODE	PNAE	RD	ASME_NC	PNAE_HT	ASME_B314	ASME_NB	EN	ASME_B311	NTD_ACI
CODE_YEAR	1989 2022	2001	1992 2010	1989	2006	1992 2010	2002 2012 2020	2008	2016

OVAL(3) - a flag of the allowance for ovality (out of roundness) in bends (Item 5.2.6.8 of RD 10-249-98 [REF 2];)

type: [INTEGER]
unit: -
dimension: array of 3 elements
default value: see [Default values] Table
limitations: 1 or 0

OVAL(1) and OVAL(2) parameters are used only in evaluation of the piping system strength according to RD 10-249-98 codes [REF 2], see Item 5.2.6.8. OVAL(1) - whether the ovality should be increased by 1.8 times for low-temperature piping systems (1 - yes, 0 - no), OVAL(2) - whether the ovality should be taken into account if a <= 3% (1 - yes, 0 - no); OVAL(3) - check of stresses both with the account of actual ovality and without account of the ovality (1 - yes, 0 - no).

KS - overload factor (Item 5.2.6.2.4 of RD 10-249-98 [REF 2])

type:	REAL
unit:	-
default value:	see Default values Table
limitations:	≥ 1

WLD_CHK - flag of checking all cross-sections of the piping system analysis model with the account of the reduction coefficient of the circumferential weld strength (see FW(2) in the PIPE command), except the points corresponding to the bend center.

type:	TEXT
unit:	-
default value:	see Default values Table
limitations:	'YES' или 'NO'

HI_E - a factor for conversion of the coefficient of secondary (thermal) stress averaging, χ, to be set by the CREEP command, into the $\chi_Э$ coefficient to be used for the computation of s_{RK} stresses in bends at the analysis of high-temperature piping systems according to PNAE codes (CODE = 'PNAE_T')

type:	REAL
unit:	-
default value:	see Default values Table
limitations:	0 < HI_E < 1

SN_T - a flag for the computation of allowable stresses for occasional and emergency loads at the analysis of high-temperature piping systems according to PNAE codes (CODE = 'PNAE_T')

type:	TEXT
unit:	-
default value:	see Default values Table
limitations:	'YES' или 'NO'

*When SN_T = 'YES' – the strength rupture limit shall be taken into account in the computation of nominal allowable stresses [σ] for stresses of categories σ_2 at the computation of S2_HDR, S2_NNUE, S2_MRZ, S2_PZ1, S2_PZ2 stresses (see the POST command). When SN_T = 'NO' – the allowable stresses for the above-mentioned stress categories shall be calculated as for low-temperature piping systems. The allowable stresses for the S2_NUE category shall **always** be calculated with the account of the strength rupture limit.*

FMESH - partial frequency for automatic meshing of the piping model into finite elements

type:	REAL
unit:	Hz
default value:	0
limitations:	≥ 0

In case when the FMESH parameter is set to be different from zero, the program will perform automatic splitting of the "straight pipe" and "bend" type elements into smaller elements according to the following criterion:

$$L_{max} \le \frac{1}{2}\sqrt{\frac{\pi}{2*FMESH}} * \sqrt[4]{\frac{E*I*g}{w}}$$

where

E *Young modulus;*
I *moment of inertia of the piping system cross-section;*
g *acceleration of gravity ;*
w *piping weight per length (together with the working fluid).*

In doing so, the internal nodes beginning with the "¤" character will appear in the model.

LIFT - the "uplift" criterion for one way supports bearing the weight load (supports of "STZ-", "STG-" and "STN-" type)

type:	REAL
unit:	mm
default value:	2
limitations:	≥ 0

ZPGA - the zero period of ground acceleration - parameter to be used within the Seismic Margin Analysis (SMA)

type:	REAL
unit:	in fractions of the acceleration of gravity (g)
default value:	0
limitations:	≥ 0

BOW_PITCH - the pipe slope angle with respect to the horizontal plane, above which the temperature stratification effect will not be taken into account (see Appendix VIII)

type:	REAL
unit:	degrees
default value:	3
limitations:	≥ 0

PNAE_KE - a flag for the execution of simplified elastic-plastic analysis within PNAE codes (CODE='PNAE')

type: TEXT
unit: -
default value: see Default values Table
limitations: 'YES' or 'NO'

K_OL - amplification factor for allowable stresses used for accidental loads. It is used as a multiplier for comparison with the stresses of *SGM2* category (analysis according to the European Codes [*REF 10*], CODE = 'EN')

type: REAL
unit: -
default value: see Default values Table
limitations: ≥1

EL_LEN - minimum permissible element length. When elements with the length smaller than EL_LEN are detected in the model, the program will output a warning.

type: REAL
unit: mm
default value: 1
limitations: >0

MIS_MAX - the maximum allowable value for the mismatch of dimensions found during piping tracing. If there is a mismatch greater than MIS_MAX in the model, the program displays a warning

type: REAL
unit: мм
default value: EL_LEN
limitations: >0

RGD_SPR - the stiffness of the vertical "rigid" supports to be used at the stage of determination of the design loads on spring hanger/supports (calculations No.1 and 8)

type: REAL
unit: N/mm
default value: RGD_TRN
limitations: >0

BEND_PSTR - pressure allowance at computation of the stress intensification factor *i* for bends (effective only when CODE = 'ASME_B311'). It is recommended to include this option for thin-walled large-diameter piping systems (ASME B31.1-2007. Table D-1 Flexibility and Stress Intensification Factors, Note 5)

type: TEXT
unit: -
default value: see Default values Table
limitations: 'YES', 'NO'

WLD_SUST - allowance for the reduction coefficient of the circumferential weld strength in the computation of stresses from sustained loads (SL). It is effective only when CODE = 'ASME_B311'. It is recommended to include this option for piping systems operated at the temperatures causing creep

type:	TEXT
unit:	-
default value:	see Default values Table
limitations:	'YES', 'NO'

SL_PRES - method for the computation of stresses from pressure in the SL category stress analysis (CODE = 'ASME_B311): at SL_PRES = 1

$$S_{lp} = \frac{PD_0}{4t_n}$$

at SL_PRES = 2:

$$S_{lp} = \frac{Pd_n^2}{D_0^2 - d_n^2}$$

type:	INTEGER
unit:	-
default value:	see Default values Table
limitations:	1, 2

SA_LBRL - method for the computation of allowable stresses Sa for SE category stresses (stresses from secondary loads caused, for example, by thermal expansions, CODE = 'ASME_B311').

If SA_LBRL = 'NO', then:	Sa = f(1.25Sc + 0.25Sh)
If SA_LBRL = 'YES', then:	Sa = f(1.25Sc + 1.25Sh-SL)

where

Sc – nominal allowable stresses for the cold state; Sh - nominal allowable stresses for the hot state; f – coefficient of strength reduction from cyclic load:

$$f = 6/N^{0.2}$$

N – number of cycles (NC parameter)
SL – stresses from permanently acting non-self-balanced loads (POST command RES = 'SL' parameter). The option will operate only when Sh > SL.

type:	TEXT
unit:	-
default value:	see Default values Table
limitations:	'YES', 'NO'

BRN_RUN - the ratio of the typical sizes of the tee connection branch and run. If the corresponding value is less than BRN_RUN, the tee shall be considered as non-equal. It is used for CODE = 'ASME_B311' and for CODE = 'RD'

type:	REAL
unit:	-
default value:	see Default values Table
limitations:	0 < BRN_RUN ≤ 1

SH_140 - option for the limitation of allowable Sh and/or Sc stresses by the value of 140 MPa if the ultimate tensile strength SU (MAT command) exceeds the value of 480 MPa. In case of explicit setting of SU, the limitation will occur by default. If SU is not determined in the material properties, the program will compute the Sh/Sc stresses depending upon this option

type:	TEXT
unit:	-
default value:	see Default values Table"
limitations:	'YES', 'NO'

FREQ_OUT - option for writing the piping vibration eigen modes in a binary file with the for the subsequent view by means of the PIPE3DV program.

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO'

ARC_ANG - minimum allowable angle for the Bend (2) element

type:	REAL
unit:	degrees
default value:	*BEND_ANG*
limitations:	> 0 ; <= 10^{o}

RH_STF - stiffness to be used by default for rod hangers and for locked springs in the analysis for hydraulic tests (see also the TEST calculation type in the SOLV command)

type:	REAL
unit:	N/mm
default value:	$1*10^{5}$
limitations:	≥ 0

RH_PND - a flag for the allowance of swing (pendulum) effect for rod hangers within the framework of dynamic analysis (SOLV command, TYPE = 'MODAL'). The load to be used for the computation of lateral stiffness of the hanger shall be determined in the analysis, which number is indicated in the PEND parameter of the SOLV command.

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

BEND_ANG - minimum permissible angle for the Bend (1) element

type:	REAL
unit:	degrees
default value:	5^{o}
limitations:	> 0 ; < 90^{o}

TBRC_TOL tolerance for the inclination of the branch from 90°

type:	REAL
units	degrees
default value:	3°
limitations	≥ 0 ; < 90°

TRUN_TOL tolerance for the inclination of the run from 180°

type:	REAL
units	degrees
default value:	3°
limitations	≥ 0 ; < 90°

RD_WLD_IV - a flag of the allowance for the reduction coefficient of circumferential weld strength at the verification calculation stage IV (to be used only for CODE = 'RD')

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO'

SPR_SFPMIN - safety coefficient relating to the minimum load. It is used at the selection of springs for spring supports/hangers. With the use of this coefficient, the following relationship shall be checked: Ph*(1 - SPR_SFPMIN*(PFAC-1)) ≥ Pmin

type:	REAL
unit:	-
default value:	1.0
limitations:	0.0 ÷ 1.0

SPR_VARTOL - reference variability value for spring selection. When the condition PVAR < SPR_VARTOL is met, the spring selection will stop even if the safety margin condition is not met.

type:	REAL
unit:	-
default value:	0.05
limitations:	0.0 ÷ 0.1

SPR_TRTRAV - spring travel's computation method: SPR_TRTRAV = 'YES' -> the spring travel shall be calculated with the account of horizontal deflection according to a "triangle"; SPR_TRTRAV = 'NO' -> only deformation from vertical movement will be taken into account.

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

SWING_SH - a flag of the allowance for swing effect for spring hangers (SPR): 'NO' - no account, 'YES' - to be taken into account without geometric non-linearity, 'GNL' - to be taken into account with the effect of geometric non-linearity.

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO', 'GNL'

SWING_RH - a flag of the allowance for swing effect for rod hangers (RH): 'NO' - no account, 'YES' - to be taken into account without geometric non-linearity, 'GNL' - to be taken into account with the effect of geometric non-linearity.

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO', 'GNL'

SWING_ST - a flag of the allowance for swing effect for rigid struts (STRT): 'NO' - no account, 'YES' - to be taken into account without geometric non-linearity, 'GNL' - to be taken into account with the effect of geometric non-linearity.

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO', 'GNL'

TEE_RD Use of CKTI Engineering methodology for Stress Analysis of Tees and Branch Connections (only within 'RD' Code), See Appendix XII. Could be used only for "standard" tees.

type:	TEXT
units:	-
default value:	'CODE'
limitations:	'CODE', 'CKTI'

TEE_FLEX option for the flexibility of tee/branch connection's joints (see Appendix XIV). Could be used only for "standard" tees (see TEE). Default value is 'CODE', that means calculation of the tee's flexibility factor strictly according to the Code approach. 'NO' means ignoring of tee's flexibilities, even if they are required by CODE or defined in the input data. 'NB' means use of ASME BPVC NB-3600 approach for the branch connection's flexibility. 'PRG' prescribes use of the methodology proposed by Paulin Research Group (PRG), *[REF 15]*. Depending on used Strength Code this methodology is applied for the certain types of tee/branch connections. If CODE = 'PNAE'/'PNAE_HT' /'RD', flexibility will be calculated for BRC, UFT and RFT types.For other Codes methodology is acceptable for 'WLT', 'BRC', 'UFT', 'RFT', 'EXT', 'SOL', 'WOL', 'FWB' types.

type:	TEXT
units:	-
default value:	'CODE'
limitations:	'CODE', 'NO', 'NB', 'PRG'

RD_E0330 option to set allowable stresses σ_2 category (S2_NNUE, S2_MRZ, S2_PZ1, S2_PZ2, S2_HDR) according to RD EO 1.1.2.05.0330-2012, *[REF 16]* document. Could be used only for PNAE analysis (CODE = ‘PNAE’).

type:	TEXT
units:	-
default value:	"NO'
limitations:	'NO', 'YES'

E_MOD_EN option to account hot modulus for the allowable stress range f_a. Applicable for CODE = 'EN' and affects calculation of SGM3 and SGM4. If E_MOD_EN = 'YES', fa is reduced on (Eh/Ec), if E_MOD_EN = 'NO', fa keeps unchanged, but internal forces are scaled to cold modulus

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO'

NC_SEISM number of equivalent seismic cycles

type:	INTEGER
unit:	-
default value:	50
limitations:	> 0

SH_LOAD Flag used to define that presetting load for spring hangers/supports is defined as R0. In this case, the program does not check the presetting load values (P) against the PMAX and PMIN parameters. When performing a calculation with data, the first load case has to be LC with type 'OPER_R'

type:	TEXT
unit:	-
default value:	-
limitations:	'R0'

EN_CORR option for accounting of the corrosion in pipes for calculation of SGM1, SGM1T и SGM2 stresses (CODE = 'EN', CODE_YEAR = '2020').

type:	TEXT
unit:	-
default value:	see Default values Table
limitations:	'YES', 'NO'

Default values for control parameters depending upon strength analysis codes.

Parameter	Analysis codes							
	PNAE	PNAE_T	RD	ASME_NB	ASME_NC	EN	ASME_B311	ASME_B314
NC	3000	3000	3000	7000	7000	7000	7000	7000
BEND_CODE	PNAE/RD	PNAE/RD	PNAE/RD	ASME	ASME	ASME	ASME	ASME
BEND_PRES	YES	YES	YES	YES	NO	NO	NO	NO
E_MOD	HOT	HOT	HOT	HOT	REF	HOT	REF	REF
OVAL(3)	-	-	1,0,1	-	-	-	-	-
KS	-	-	1.4	-	-	-	-	-
WLD_CHK	-	YES	YES	-	-	-	YES	-
HI_E	-	0.6	-	-	-	-	-	-
SN_T	-	NO	-	-	-	-	-	-
PNAE_KE	NO	-	-	-	-	-	-	-
K_OL	-	-	-	-	-	1	1.15	-
BEND_PSTR	-	-	-	-	-	NO	NO	-
WLD_SUST	-	-	-	-	-	-	NO	-
SL_PRES	-	-	-	-	-	-	1	-
SA_LBRL	-	-	-	-	-	-	NO	-
BRN_RUN	-	-	0.77	-	-	-	1	-
SH_140	-	-	-	-	-	-	YES	-
RD_WLD_IV	-	-	YES	-	-	-	-	-
CODE_YEAR	1989	1989	2001	1992	1992	2012	2008	2006
TEE_RD	-	-	CODE	-	-	-	-	-
TEE_FLEX	CODE	CODE	CODE	CODE	CODE	CODE	CODE	CODE
RD_E0330	NO	-	-	-	-	-	-	-
E_MOD_EN	-	-	-	-	-	YES	-	-
EN_CORR[1)]	-	-	-	-	-	NO	-	-

Note:
[1)] This option is available only for stress analysis according to EN Code, Edition 2020

Example :

```
CTRL TA 50 NC 1000
```

Fatigue Data (FAT)

type: general multi-line command

Function: input of the fatigue curves

Parameters:

ID - identification name of the fatigue curve

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

EM[1)] - Young Modulus used to develop the Calculational Fatigue Curve from the measured strains to the conditionally elastic stresses .

type:	REAL
unit:	N/mm^2
default value:	modulus of elasticity at the T_REF temperature (see CTRL command)
limitations:	> 0

INT - identifier of the interpolation method for intermediate points. Depending upon the INT value, the interpolation will be carried out either along the linear or logarithmic scale of the fatigue curve axes.

type:	INTEGER
unit:	-
default value:	11
limitations:	0 (LIN-LIN); 1(LOG-LIN); 10(LIN-LOG); 11(LOG-LOG)

Parameter subcommands

NC - array of the numbers of cycles for the fatigue curve being entered

type:	INTEGER
unit:	-
dimension:	array from 1 to 32 elements
default value:	-
limitations:	from 1 to $1*10^{12}$. Each subsequent element of the array shall be larger than the previous one. It is allowed to enter an integer number of cycles in the exponential form

SA - array of the values of amplitudes of conditional elastic stresses corresponding to the numbers of cycles being entered.

type:	REAL
unit:	MPa
Dime nation:	array from 1 to 32 elements
default value:	-
limitations:	> 0. Each subsequent element of the array shall not be larger than the previous one.

Note:

1) In computation of $(\sigma_{aF})_K$ *stresses, the design stress value shall be multiplied by the value of (Em/E_ref) in accordance with Item 5.6.5 of the Codes [REF 1] where E_ref is the modulus of elasticity at the T_REF* temperature *(see CTRL command).*

```
Example:
FAT ID 'AUS' E 1.75E5 INT 11
&   NC     10     20     50     100    200    500    850    1000   \
     2000   5000   10000 12000 20000 50000 100000  200000  5. E 5   1. E 6

&   SA    3194    2307   1519   1123   842    593    493    468    \
     379    297   234   221   189    150    130   116    104     98
```

Creep Data (CREEP)

type: general multi-line command

Function: input the set of curves for high-temperature piping system analysis (to be used at CODE='RD' or CODE='PNAE_T').

Parameters:

ID - identification name.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

T0 - initial temperature, above which the piping system is considered as high-temperature one (see Item 5.2.1.2 of RD 10-249-98 [REF 2]) .

type:	REAL
unit:	°C
default value:	370
limitations:	> 0.

Parameter subcommands:

T - array of temperatures, for which the data are set.

type:	REAL, array from 1 to 32 elements
unit:	°C
default value:	-
limitations:	Each subsequent element of the array shall be larger than the previous one

HI - array of values of the coefficients of averaging of secondary (thermal) stresses depending upon working temperature (see Fig. 5.5 of RD 10-249-98 [REF 2]).

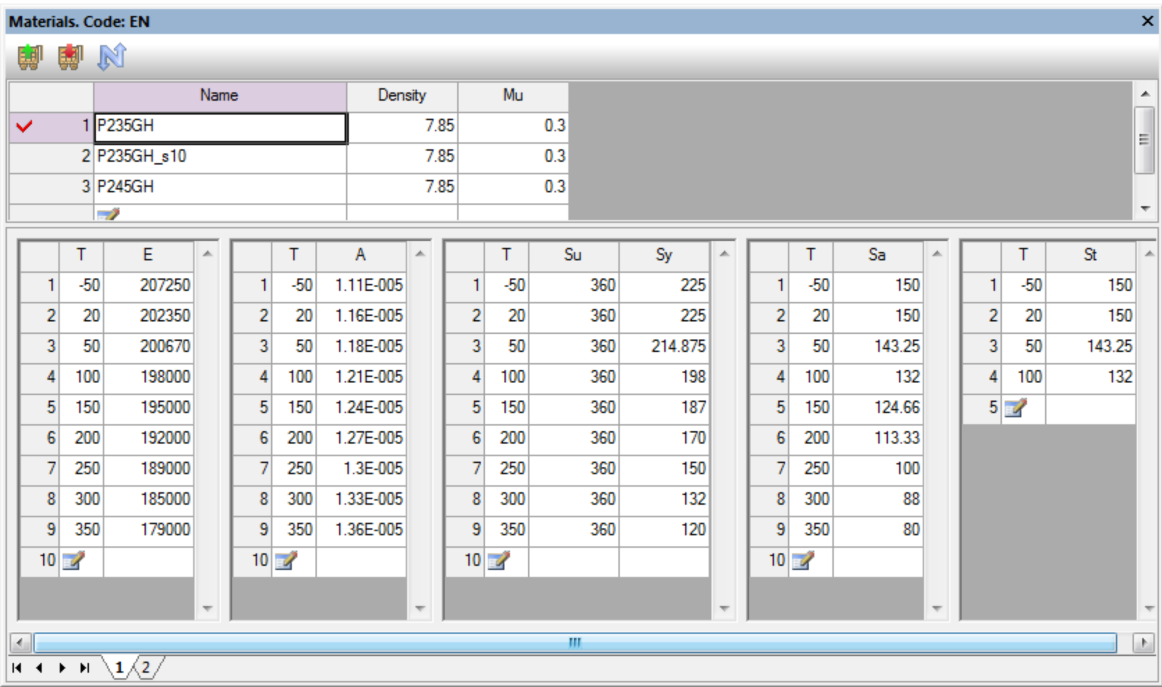
type: REAL
unit: -
dimension: array from 1 to 32 elements
default value: -
limitations: 0 < Hi ≤ 1.

DELTA - array of values of the coefficients of relaxation of secondary (thermal) stresses depending upon working temperature (see Fig. 5.6 of RD 10-249-98 [REF 2]).

type: REAL
unit: -
dimension: array from 1 to 32 elements
default value: -
limitations: 0 < DELTA ≤ 1

```
Example :

CREEP ID '1' TO 370
& T =  370    380    390    400    410    420    430    440    450
460    470
& HI  =  0.59   0.58   0.57   0.56   0.55   0.55   0.53   0.51   0.50
0.48   0.45
& T  =  370    380    390    400    410    420    430    440    450
460    470
& DELTA  = 0.76   0.77   0.78   0.79   0.81   0.83   0.85   0.87   0.89
  0.92   0.94
```

Materials (MAT)


Materials. Code: EN

	Name	Density	Mu
1	P235GH	7.85	0.3
2	P235GH_s10	7.85	0.3
3	P245GH	7.85	0.3

	T	E
1	-50	207250
2	20	202350
3	50	200670
4	100	198000
5	150	195000
6	200	192000
7	250	189000
8	300	185000
9	350	179000
10		

	T	A
1	-50	1.11E-005
2	20	1.16E-005
3	50	1.18E-005
4	100	1.21E-005
5	150	1.24E-005
6	200	1.27E-005
7	250	1.3E-005
8	300	1.33E-005
9	350	1.36E-005
10		

	T	Su	Sy
1	-50	360	225
2	20	360	225
3	50	360	214.875
4	100	360	198
5	150	360	187
6	200	360	170
7	250	360	150
8	300	360	132
9	350	360	120
10			

	T	Sa
1	-50	150
2	20	150
3	50	143.25
4	100	132
5	150	124.66
6	200	113.33
7	250	100
8	300	88
9	350	80
10		

	T	St
1	-50	150
2	20	150
3	50	143.25
4	100	132
5		

type: general multi-line command

Function: determination of piping material properties

Parameters:

ID - identification name of the material

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

FAT - reference identification name of the fatigue curve (see FAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

FAT _ B - reference identification name of the fatigue curve for bend (see FAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. It is used for checking the piping system strength according to RD 10-249-98 code [REF 2]

CREEP reference identification name of the creep curve

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters.

DEN - density of the material

type:	REAL
unit:	in fractions of the water density (for water DEN = 1)
default value:	7.85
limitations:	> 0.

MU -Poisson's ratio

type:	REAL
unit:	-
default value:	0.3
limitations:	0 ≤ MU ≤ 0.5

M, N - parameters of the material to be used for simplified elastic-plastic analysis (to be taken into account only for CODE='ASME_NB' or CODE='PNAE'). It is recommended to determine then according to the NB-3228.5(b)-1 table [REF 3, REF_11].

type: REAL
unit: -
default value: 0, 0
limitations: ≥ 0

Table XIII-3450-1
Values of m, n, and T_{max} for Various Classes of Permitted Materials

Materials	m	n	T_{max}, °F (°C)
Carbon steel	3.0	0.2	700 (370)
Low alloy steel	2.0	0.2	700 (370)
Martensitic stainless steel	2.0	0.2	700 (370)
Austenitic stainless steel	1.7	0.3	800 (425)
Nickel–chromium–iron	1.7	0.3	800 (425)
Nickel–copper	1.7	0.3	800 (425)

GENERAL NOTE: T_{max} is the maximum metal temperature.

JF - weld joint efficiency factor, ASME B31.1 [REF 12]). This factor is used for the conversion of allowable stresses entered from Appendix A. It is used when CODE = 'ASME_B311'

type: REAL
unit: -
default value: 0.3
limitations: 0 < JF ≤ 1

TYPE - type of material. Is used when CODE = 'PNAE', CODE_YEAR = 2022 are applied. Parameter defines the type of material: 'CS' – carbon steels, 'CMS' - alloyed chromium molybdenum and chromium molybdenum vanadium steels, 'AUS' – stainless austenitic steels. This parameter is used for fatigue analysis: calculation of allowable number of cycles considering environmental effects (Appendix B of Code [24])

тип: TEXT
единицы: -
значение по умолчанию: -
область возможных значений: 'AUS', 'CS', 'CMV'

Parameter subcommands

T - array of temperatures, for which the characteristics of the material are determined.

type: REAL, array from 1 to 32 elements
unit: °C
default value: -
limitations: Each subsequent element of the array shall be larger than the previous one.

E - modulus of elasticity

type:	REAL, array from 2 to 32 elements
unit:	MPa
default value:	-
limitations:	> 0.

A - average temperature linear expansion coefficient

type:	REAL, array from 2 to 32 elements
unit:	1/°C
default value:	-
limitations:	> 0.

SU - ultimate strength

type:	REAL, array from 2 to 32 elements
unit:	MPa
default value:	-
limitations:	> 0.

SY - yield strength

type:	REAL, array from 2 to 32 elements
unit:	MPa
default value:	-
limitations:	> 0.

SA3) - allowable stresses

type:	REAL, array from 1 to 32 elements
unit:	MPa
default value:	-
limitations:	> 0.

SOL2) - allowable stresses for occasional loads (for example, seismic, wind, hydraulic shock).

type:	REAL, array from 1 to 32 elements
unit:	MPa
default value:	SA
limitations:	> 0.

SR3) - creep-rupture strength

type:	REAL, array from 1 to 32 elements
unit:	MPa
default value:	-
limitations:	> 0.

ST allowable stress under the proof test conditions

	type:	REAL, array from 1 to 32 elements
	units:	MPa
	default value:	-
	limitations:	> 0.
WLD	- reduction coefficients of weld joint strength	
	type:	REAL, array from 1 to 32 elements
	unit:	-
	default value:	-
	limitations:	0.5 < WLD ≤ 1
Z	относительное сужение поперечного сечения образца после разрыва	
	type:	REAL, array from 1 to 32 elements
	unit:	%
	default value:	-
	limitations:	100 > Z > 0

Note:

1) It is recommended to use the SOL subcommand for high-temperature piping systems in order to re-determine the SA ([σ]) value taken according to the 2.1 – 2.7 Table RD 10-249-98 [REF 2] and limited for high temperautres by the creep-rupture strength. As the current RD edition does not contain any direct instructions on the computation of nominal allowable stresses for occasional loads, it is recommended to use the following approach, which is similar to the procedure recommended by the American codes - ASME B31.3 [REF 8]:

at T ≤ Tв, SOL = SA
*at T > Tв, Sol = 0.8 * (Syt/1.5),*

where

T – working temperature, °C;
Tв – temperature corresponding to the "high-temperature piping system" definition (Item 5.2.1.2 of RD), °C;
Syt – yield strength of the material at the working temperature, MPa.

The reduction coefficient 0.8 is to be entered in the last formula for taking account of the material aging effect at high temperatures.

For materials included into the database, which is supplied along with the program (MAT.DBS file), the SOL values have been determined on the basis of yield strength values taken from [REF 9]

*The allowable stresses SOL are used at the comparison of stress values S_I_PZ (POST command): the strength condition will be met if S_I_PZ ≤ 1.8 * SOL*

2) The MAT command parameters are used by the program depending upon the strength analysis codes in accordance with the following table:

CODE	ID	FAT	FAT_B	CREEP	DEN	MU	M,N	TYPE	JF	E	A	SU	SY	SA	SOL	SR	WLD	ST	Z
PNAE	x	x			x	x	x			x	x	x	x						
PNAE 2022	x	x			x	x	x	x		x	x	x	x						x
PNAE_T	x	x		x	x	x	x			x	x	x	x			x			
RD	x	x	x	x	x	x				x	x			x	x				

EN	x				x	x				x	x	x	x	x	x	x		x	
NC	x				x	x				x	x	x	x	x					
NB	x	x			x	x	x			x	x	x	x	x					
B311	x				x	x			x	x	x	x	x	x			x		
B314	x				x	x				x	x			x					

3) Parameters SA & SR may be entered as functions of lifetime (in thousands hours) depending on used CODE:

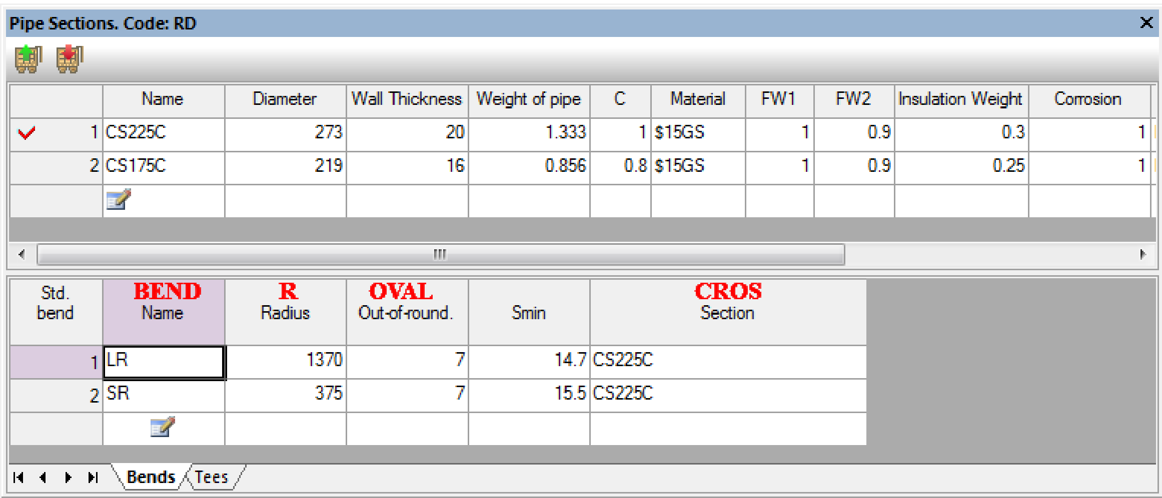
CODE	SA[TAU]	SR[TAU]
RD	+	-
EN	-	+
PNAE_T	-	+

The following syntax is used in this case: : SA[TAU]/SR[TAU], where TAU is lifetime:

```
& T          -20  20 150 250 300 350 400 420 440 450 460 480 500 510 520 530 540 550 560 570 580
& SA[10]     173 173 173 166 159 152 145 142 139 138 136 133 130 120 112 100  88  80  72  65  59
& SA[100]    173 173 173 166 159 152 145 142 139 138 136 133 113 101  90  81  73  66  59  53  47
& SA[200]    173 173 173 166 159 152 145 142 139 138 136 120  96  86  77  69  62  56  50  44  39
```

```
Example :

MAT 'ST20' FAT 'CS' DEN 7.8 MU 0.3
& T = 20 50 100 150 200 250 300 350
& SU = 402.00 392.00 392.00 392.00 373.00 373.00 363.00 353.00
& SY = 216.00 206.00 206.00 206.00 196.00 196.00 177.00 157.00
& E = 2.000E+05 1.970E+05 1.950E+05 1.920E+05 1.900E+05 1.850E+05
1.800E+05 1.750E+05
& A = 1.150E-05 1.150E-05 1.190E-05 1.220E-05 1.250E-05 1.280E-05
1.310E-05 1.340E-05
```

Pipe cross-section (PIPE)



type: general multi-line command

Function: determination of the characteristics of piping system cross-sections

Parameters:

ID - identification name of the cross-section

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

OD - outside diameter

type:	REAL
unit:	mm
default value:	-
limitations:	> 0

T - nominal wall thickness of the pipe

type:	REAL
unit:	mm
default value:	-
limitations:	0 < T < OD/2

W[1)] - weight per length

type:	REAL
unit:	N/mm
default value:	$\pi * (OD - T) * T * DEN * W\_DEN$
limitations:	≥ 0

C[2)] - mill tolerance

type:	REAL
unit:	mm or %
default value:	0
limitations:	if C > 0, then T – C > 0, if C < 0, then |C| < 100.

MAT - reference identification name of the material (see MAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	the name should coincide with the materials names determined earlier.

FW(2)[3)] - reduction coefficients of weld strength. FW(1) – reduction coefficient of longitudinal (spiral) weld strength; FW(2) – reduction coefficient of circumferential weld strength;

type:	REAL
unit:	-
default value:	1, 1

limitations: 0 < FW ≤ 1

FI_S - cyclic strength reduction coefficient for welding connections (for CODE = 'PNAE' and 'PNAE_T', it is to be set by the user in accordance with Item 5.6.12 of the codes [REF 1], see also the WLD command)

type: REAL
unit: -
default value: 1
limitations: 0 < FI_S ≤ 1

IWGT[1)] - insulation's weight per length

type: REAL
unit: N/mm
default value: 0
limitations: ≥ 0

CORR - corrosion allowance.

type: REAL
unit: mm
default value: 0
limitations: CORR ≥ 0
if C ≥ 0, then T – (C+CORR) > 0,
if C < 0, then T – (0.01*|C|*T + CORR) > 0

Parameter subcommand

BEND - identification name of the bend. It is used for "standard" bends assigned under the current cross-section.

type: TEXT
unit: -
default value: -
limitations: see limitations for the text values of parameters

Subcommand parameters:

R - radius of the bend

type: REAL
unit: mm
default value: -
limitations: > 0

OVAL - ovality (out-of-roundness) of the cross-section

type: REAL
unit: %
default value: 0
limitations: $0 \le OVAL \le 100$

SMIN - minimum wall thickness of the bend (with no account of the pipping wall thinning)

type:	REAL
unit:	mm
default value:	T - C
limitations:	$SMIN - corr > 0$

CROS - reference to the PIPE name determined earlier and being used for the bend cross-section characteristics

type:	TEXT
unit:	-
default value:	'ID' name of the main PIPE command
limitations:	the name should coincide with the cross-section names determined earlier

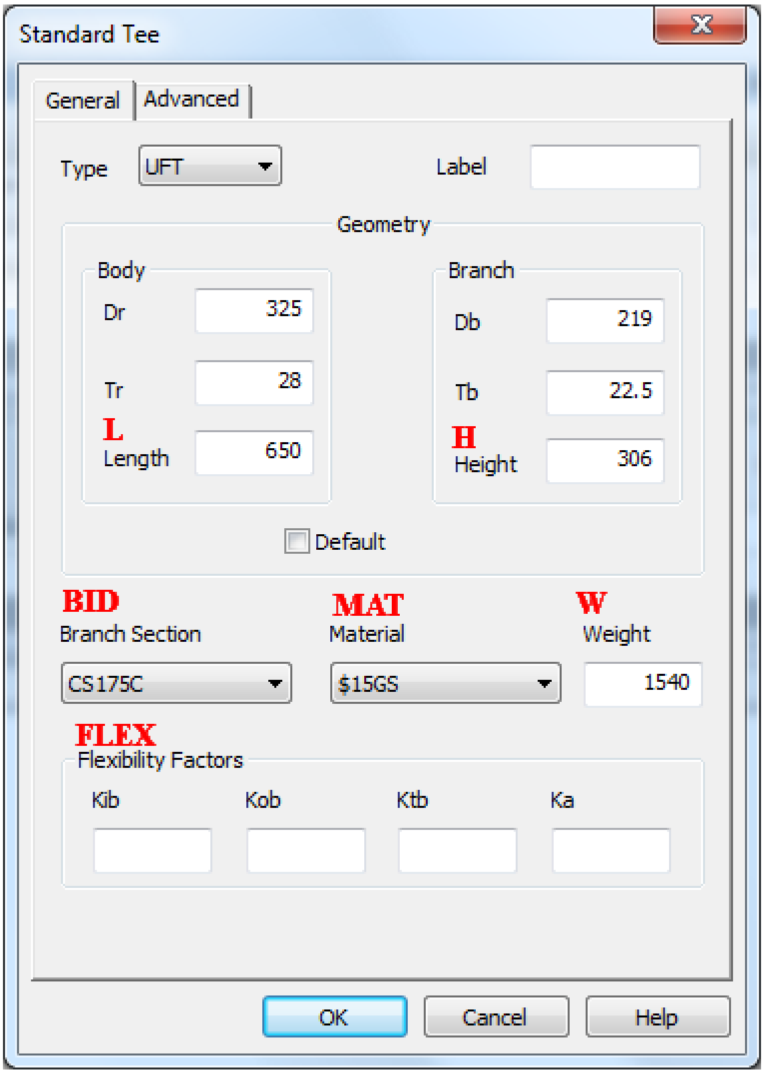
Subcommand parameters

TEE ID of the piping intersection. It's used to set input data for standard fittings adjacent to the matched pipes.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

ID for "standard" fittings is composed as follows: TEE='TYPE[$...]', where TYPE is predefined name that depends from used Code and can be set to one of the following values: BRC, WLT, RFT, UFT, EXT, SOL, WOL, 'FWB'. Second part of the ID beginning from "$" sign is optional. It's used to distinguish between several standard tees matched to the same pipe sizes but having different parameters. For example: &TEE 'BWT' ...; &TEE 'BWT$01' ...; &TEE 'BRC' .... For Russian Codes (CODE = 'PNAE', 'PNAE_HT' and 'RD') any ID can be used.

Main parameters (applicable for all Codes):



BID ID of the BRANCH pipe, required parameter, should refer to the existing command PIPE

type:	TEXT
units:	-
default value:	-
limitations:	only sections previously set by PIPE command

DR, TR outer diameter and wall thickness of the RUN pipe

type:	REAL
units:	mm
default values:	corresponding data of the matched "parent" PIPE
limitations:	> 0

DB, TB outer diameter and wall thickness of the BRANCH pipe

type:	REAL
units:	mm
default values:	corresponding data of the BRANCH pipe defined by ***BID*** name
область возможных	> 0

значений:

L length of the body. If ***DR, TR*** are set explicitly, ***L*** is required .

type:	REAL
units:	mm
default values:	-
limitations:	> 0, length of any adjacent elements should not be greater than L/2

H height of branch (measured from the run pipe axis line)). If ***DB, TB*** are set explicitly, **H** is required.

type:	REAL
units:	мм
default values:	-
limitations:	> 0, length of any adjacent elements should not be greater than H

W weight of the fitting

type:	REAL
units:	N
default values:	If H or L are set, then: W = w_r*L + w_b*(H-DR/2), where: w_r = MAX(□*(DR-TR)*TR*DEN; w_rp); w_rp – weight per length for adjacent RUN pipe; DEN – density of the material MAT; w_b = MAX(□*(DB-TB)*TB*DEN; w_bp), where: w_bp - weight per length for adjacent BRANCH pipe (BID);
limitations:	≥ 0

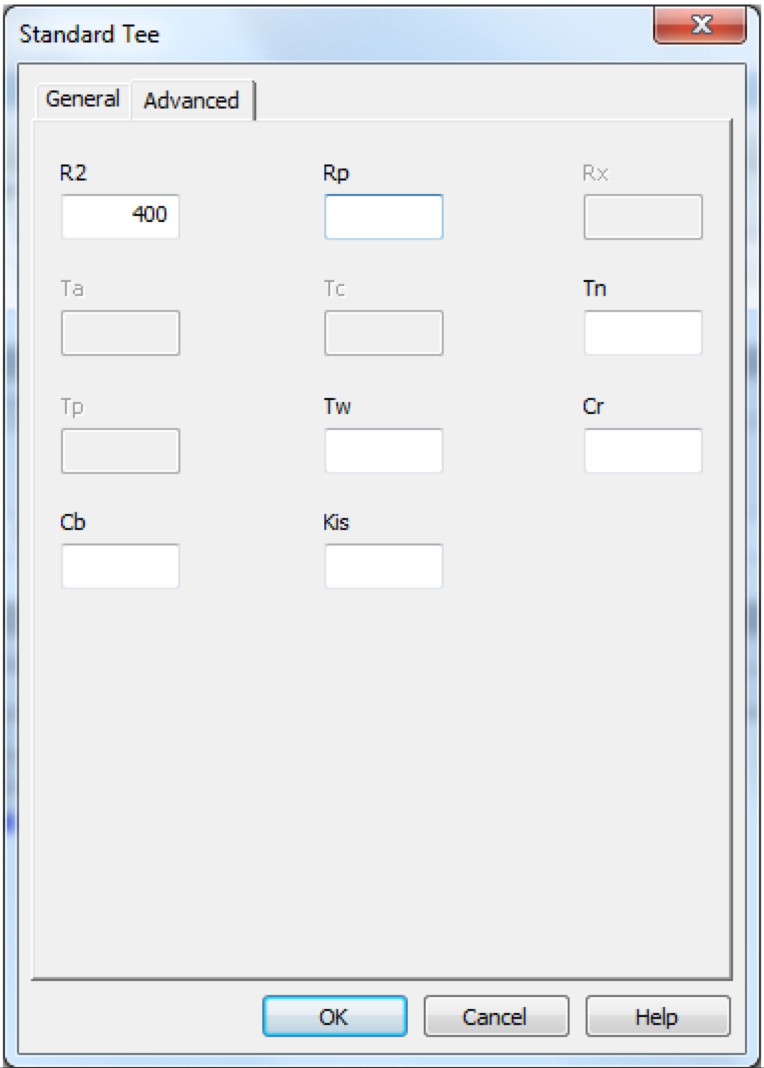
MAT reference name of the fitting's material (see command MAT)

type:	TEXT
units:	-
default values:	material of the RUN adjacent pipe
limitations:	name should be defined by command MAT

FLEX(4) flexibility factors: FLEX(1) – in plane bending (k_{ib}), FLEX(2) – out of plane bending (k_{ob}), FLEX(3) – torsion of the branch (k_{tb}), FLEX(4) – axial branch's flexibility (k_a)

type:	REAL
units:	-
dimension:	4-elements array
default values:	are set in accordance with TEE_FLEXparameter. If one of the elements = 0, then flexibility in this direction is not considered despite the TEE_FLEX parameter.
limitations:	≥ 0

Additional parameters (are compatible only with certain types of tees, depend on the used stress Codes):



Type of TEE	Description	Code	Additional parameters										
			R2	**RP**	**RX**	**TA**	**TC**	**TN**	**TP**	**TW**	**CR**	**CB**	**KIS**
WLT	Welding tee per ASME B16.9	NB, NC, B311, EN			**X**		**X**			**X**	**X**	**X**	**X**
RFT	Reinforced fabricated tee	NC, B311, EN		**X**					**X**	**X**	**X**	**X**	**X**
UFT	Unreinforced fabricated tee	NC, B311, EN	**X**	**X**				**X**		**X**	**X**	**X**	**X**
BRC	Branch connection	NB, NC, B311, EN	**X**	**X**				**X**		**X**	**X**	**X**	**X**
EXT	Extruded outlet	B311			**X**	**X**	**X**			**X**	**X**	**X**	**X**
WOL	Branch welded-on fitting/Weldolet	B311		**X**						**X**	**X**	**X**	**X**
SOL	Welded-in contour insert/Sweepolet	B311			**X**		**X**			**X**	**X**	**X**	**X**
FWB	Fillet welded and partial penetration	NC-2010		**X**				**X**		**X**	**X**	**X**	**X**

	welded branch connections												
-		PNAE, PNAE_HT								x	x		x
-		RD									x	x	

1) Type of the tee is defined by its identification name identification name;
2) ASME, EN Codes:
R2 – transition radius of branch reinforcement, mm;
RP - outside radius of branch reinforcement, mm;
RX - external crotch radius of welded-in contour inserts and welding tees, mm;
TA – mean thickness of the header pipe;
TC - crotch thickness of welded-in contour inserts and welding tees, mm;
TN - nominal wall thickness of branch pipe, mm;
TP - reinforcement pad or saddle thickness, mm

PNAE, RD Codes:
TW - reinforcement pad or saddle thickness, mm;
CR - mill tolerance of the header pipe;
CB - mill tolerance of the branch pipe;
KIS – stress index (PNAE)
3) See also Appendix XIV

Note:

1) See also Appendix IV for comments relating to the weight load setting for piping systems.
2) If the C parameter value is set < 0, then it is assumed that the reduction of the pipe wall thickness is set in percentage of the nominal piping wall thickness.
3) For piping systems to be analyzed according to the RD boiler codes and for high-temperature piping systems to be analyzed according to PNAE (CODE = 'RD', CODE = 'PNAE_T'), there is an alternative form of setting the reduction coefficients of weld strength. Instead of FW(1) and FW(2) array elements, it is possible to determine the following two parameters: FW1 and FW2. The FW1 parameter is fully similar to the first element FW(1) of the array. The FW2 parameter is a text mark corresponding to the type of circumferential weld. For CODE = 'RD' the FW2 parameter can take the following values: 'CS' - carbon steels; 'AUS' - austenite steels; 'CMV' - chrome-molybdenum-vanadium steels (see Table 4.2 of RD). For PNAE codes, it is permitted to use only FW2 = 'CMV'. In using such a form, the program will automatically assign the reduction coefficients of circumferential welds depending upon the values entered and the working temperature of the piping system.

```
Example :

PIPE '133x14' OD 133 T 14 C -5 MAT   '08H18N10T' CORR 1.  FW  0.7 1
&  BEND  ' R 175'  R  175
```

Beam cross-section (BEAM)

type: general one-line command

Function: determination of the characteristics of beam element cross-sections

Parameters:

ID - identification name of the cross-section

type: TEXT
unit: -
default value: -
limitations: see limitations for the text values of parameters

AX - cross-sectional area

type: REAL
unit: mm^2
default value: -
limitations: > 0

SY2) - shear shape factor along Y axis (at SY = 0, no shear is taken into account)

type: REAL
unit: -
default value: 0
limitations: ≥ 0

SZ2) - shear shape factor along Z axis (at SZ = 0, no shear is taken into account)

type: REAL
unit: -
default value: 0
limitations: ≥ 0

IX - torsional moment of inertia

type: REAL
unit: mm^4
default value: -
limitations: > 0

IY - cross-section moment of inertia in respect to Y axis

type: REAL
unit: mm^4
default value: -
limitations: > 0

IZ - cross-section moment of inertia in respect to Z axis

type: REAL
unit: mm^4
default value: -
limitations: > 0

W[3)] - weight per length

type:	REAL
unit:	N/mm
default value:	0
limitations:	≥ 0

MAT - reference identification name of the material (see MAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	the name should coincide with the names of materials determined earlier

B - cross-section width - the size along the local Y axis (for displaying in 3D mode)

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

H - cross-section height - the size along the local Z axis (for displaying in 3D mode)

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

TYPE - cross-section type (for displaying in Pipe3DV)

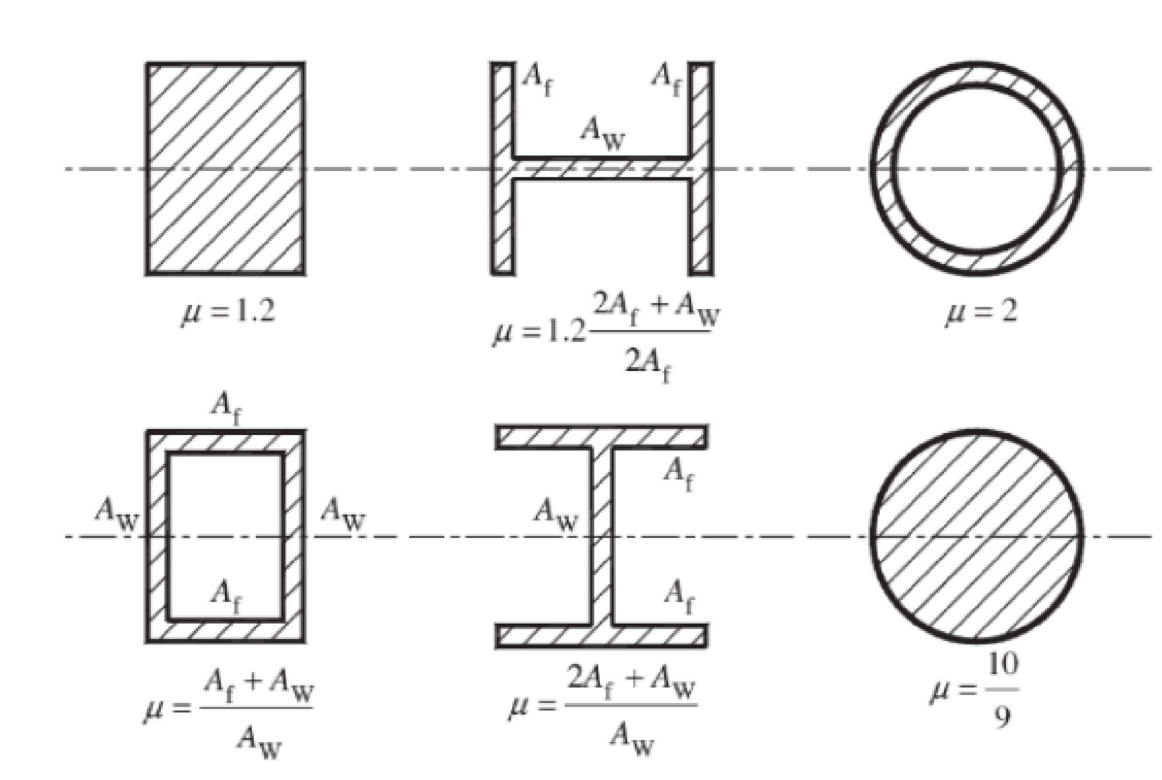
type:	TEXT
unit:	-
default value:	'I'
limitations:	'I', 'U', 'L', 'P', 'B', 'O', 'T'

'I' - I-beam; 'U' - channel; 'L' - angle; 'P' - rectangular pipe; 'B' - rectangular solid cross-section; 'O' - circular solid cross-section; "T'- T-section

Note:

1) *The geometric characteristics of beam element cross-sections are determined in the local coordinate system (see Appendix I).*

2) *Shear in beams becomes available if "shear shape factors" (Sy and Sz parameters) are set as non zero values. Typical values of shear shape factors (μ) are shown below:*



3) See also [Appendix IV](#) for comments relating to the weight load setting.

Example :

```
BEAM 'BEAM1' AX  6.40000E+03   IX  5.76000E+06 IY  3.41000E+06 \
IZ 3.41000E+06 SY  1.200 SZ  1.200 \
W 0.0  MAT '08H18N10T' B 100 H 200 type = 'L'
```

Operating modes (OPVAL)

type: general multi-line command

Function: setting the operating parameters of the piping system and its operating modes

Parameters:

ID 1) - identification name of the piping system operating mode

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

TAU4) - lifetime

type:	REAL

unit:	thous. hours
default value:	200.
limitations:	≥ 0

Parameter subcommand

LG2) - name of the piping system load group, i.e. the group of elements with the same working parameters

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

Subcommand parameters:

P - internal pressure

type:	REAL
unit:	MPa
default value:	0
limitations:	≥ 0

T - temperature

type:	REAL
unit:	°C
default value:	TA (see CTRL command)
limitations:	-

CSG3) - content specific gravity (CSG = 0 - empty pipe, CSG=1.0 pipe with water)

type:	REAL
unit:	in fractions of the water density (for water DEN = 1)
default value:	0
limitations:	≥ 0

INS3) - option for insulation

type:	REAL
unit:	in fractions of the insulation weight assigned for the pipe section (see parameter IWGHT, Command PIPE)
default value:	1
limitations:	≥ 0

TAU4) - lifetime

type:	REAL
unit:	thous. hours
default value:	*TAU* (see above)
limitations:	≥ 0.

Note:

1) *It's assumed that first operating mode described by this command corresponds to the Normal Operating Conditions (NOC). It could be mentioned as alias "**$OPER**" in the parameter MOD, command SOLV. A similar reference to the mode named **$COLD** means using the cold state in the analysis: i.e. parameters of piping before startup: no medium, no pressure and at temperature T = TA (the mode with this name can be skipped in the command: the program will automatically determine its parameters). Other standard names are: '**TEST**' – for the regime of hydraulic tests with locked springs, '**$INST**' – for the mode of piping installation and assembling (to redefine value of TA temperature); **$DESIGN** and **$PEAK** are used to define the design and peak pressure respectively within the analysis according to ASME BPVC.*

2) *Depending upon the working parameters (temperature, pressure, working fluid, insulation) the piping model is broken down into load groups. Each operating mode shall have a description of parameters for all load groups being present in the analysis model.*

3) *See also Appendix IV for comments relating to setting the weight load for piping elements.*

4) *Lifetime TAU is used for calculations according to CODE = 'RD', 'EN' and 'PNAE_T'. Depending on this number the following parameters are calculated: allowable stresses SA or creep rupture strength SR. If TAU = 0, then stresses which values are dependent on lifetime: S_II, S_IV(CODE = 'RD') & SGM5 (CODE= 'EN') will be skipped from calculations.*

Example:

```
OPVAL 'NOL'
&  'Line_1' P 12.0 T 250 CSG 1
&  'Line_2' P  8.0 T 350 CSG 0

OPVAL 'ZERO'
&  'Line_1' P 0 T 20 CSG 1
&  'Line_2' P 0 T 20 CSG 0
```

Discont. stresses due to temp. gradient and stratification (GRAD)

type: general multi-line command

Function: setting the parameters for taking into account of additional stresses from the temperature drop over the wall thickness as well as from the stratification effect (they are used in the analysis according to the PNAE codes [REF 1] and ASME BPVC NB-3600 (Class1) codes [REF 3]). The command shall be placed only after the OPVAL command. The *MODE and LG* parameters shall conform to the similar parameters defined in the OPVAL command.

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

Parameter subcommand

LG - name of the load group of the piping system, i.e. the group of elements with the same working parameters

type: TEXT
unit: -
default value: -
limitations: see limitations for the text values of parameters

Subcommand parameters:

DT1 - linear part of the temperature gradient over the wall thickness

type: REAL
unit: °C
default value: 0
limitations: –

DT2 - non-linear part of the temperature gradient over the wall thickness

type: REAL
unit: °C
default value: 0
limitations: –

STRAT - linear part of the temperature gradient over the cross-section height due to the stratification effect

type: REAL
unit: °C
default value: 0
limitations: –

DT3 - non-linear part of the temperature gradient over the cross-section height due to the stratification effect

type: REAL
unit: °C
default value: 0
limitations: –

STRESS - pre-calculated stress from the temperature drop over the wall thickness

type: REAL
unit: MPa
default value: 0
limitations: -

Example:

```
GRAD 'MODE_2S'
& 'SECT_A'        DT1 0.2   DT2 0.03 STRESS 45 STRAT 27      DT3 10.04
& 'SECT_B'        DT1 2.8   DT2 0.5  STRESS 30 STRAT 20      DT3 6.6
```

```
& 'SECT_C_D'            DT1 0.2  DT2 0.03 STRESS 58 STRAT 19.17   DT3 14.1
& 'SECT_E'        DT1 0.2  DT2 0.03 STRESS 35 STRAT 29.42   DT3 7
& 'SECT_F'        DT1 0.2  DT2 0.03 STRESS 56 STRAT 21.67   DT3 13.38
& 'FW'                  DT1 20.6 DT2 3.6  STRESS 34 STRAT 42.8    DT3 15
& 'FW_SG'         DT1 65.4 DT2 19.2 STRESS 34 STRAT 42.8    DT3 15
```

Note:

1) *Depending upon the analysis codes selected, the program will use the following parameters: CODE = 'ASME_NB': ΔT_1, ΔT_2, STRAT and ΔT_3; CODE = 'PNAE' and CODE = 'PNAE_T': STRESS, STRAT and ΔT_3, the STRAT parameter can be used in the analysis according to any Codes.*

2) *The determination and methods of calculation of ΔT_1 and ΔT_2 are given in section NB-3653.2 [REF 3].*

3) *The STRESS parameter corresponds to the swing of full maximum temperature stress from the drop over the wall thickness |*
$(\sigma)^*_T$ – see Item 2.3.3.4 of Appendix 5 of the PNAE codes [REF 1].

4) *The determination and methods of calculation of STRAT and ΔT_3 are given in Appendix VIII.*

```
Example:

OPVAL 'MODE_2S'
& 'SECT_A'        DT1 0.2   DT2 0.03  STRESS 45 STRAT 27     DT3 10.04
& 'SECT_B'        DT1 2.8   DT2 0.5   STRESS 30 STRAT 20     DT3 6.6
& 'SECT_C_D'      DT1 0.2   DT2 0.03  STRESS 58 STRAT 19.17  DT3 14.1
& 'SECT_E'        DT1 0.2   DT2 0.03  STRESS 35 STRAT 29.42  DT3 7
& 'SECT_F'        DT1 0.2   DT2 0.03  STRESS 56 STRAT 21.67  DT3 13.38
& 'FW'            DT1 20.6  DT2 3.6   STRESS 34 STRAT 42.8   DT3 15
& 'FW_SG'         DT1 65.4  DT2 19.2  STRESS 34 STRAT 42.8   DT3 15
```

Water oxidation (ENVFAT)

type: general multi-line command

Function: Defines the values of water oxidation for piping sections. The command is used in frame of the Environmental Fatigue Analysis according to GOST Code [REF 24], when the following parameters are set: CODE = 'PNAE', CODE_YEAR= 2022, FAT_ENV = 'YES'. The command must only come after the OPVAL command. The MODE and LG parameters must match those defined in the OPVAL command. The procedure for calculating fatigue curves according to GOST R 59115.9-2021, [REF 24] in the dPIPE program is described in Appendix XVII.

Parameters:

MODE identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

Parameter subcommand

LG name of the load group of the piping system, i.e. the group of elements with the same working parameters

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

Subcommand parameters:

KO water oxidation

type:	REAL
unit:	mg/kg
default value:	KO
limitations:	≥0.

Example:

```
OPVAL 'NOC'
& 'SECT_A'          KO 0.02
& 'FW'                     KO 0
```

Data for spring hangers and supports (SDEF)

type: general one-line command

Function: specification for the standard spring tables

Parameters:

STAB - identification name of the springs table

type:	TEXT
unit:	-
default value:	'OST80'
limitations:	the corresponding parameters prescribed in the SH.DBS file (see Appendix VI), for example, 'MVN63', 'OST80', 'OST93', 'LISEGA')

PVAR - load variation coefficient

type:	REAL
unit:	-
default value:	0.35
limitations:	0 ≤ PVAR ≤ 1

PFAC - load safety factor

type:	REAL
unit:	-
default value:	1.3
limitations:	≥ 1

ZMAX - maximum structure of the chain

type: INTEGER
unit: -
default value: (2)
limitations: ≥ 1

ZMIN - minimum structure of the chain

type: INTEGER
unit: -
default value: 1
limitations: 1 ≤ ZMIN ≤ ZMAX;

Note:

1) The SDEF command initiates the data to be used further by default in SPR commands by the characteristics of the hanger's/support's springs. One file with the input data can contain several SDEF commands.
2) By default, ZMAX is a maximal number of the working ranges predefined in the Manufacturer Catalogue (spring's table) being under consideration (see Appendix VI)

```
Example:

SDEF 'OST93' PVAR 0.3 PFAC 1.4
SDEF 'LISEGA' PVAR 0.25 PFAC 1.1 ZMAX 3
```

Seismic response spectra (SPEC)

type: general multi-line command

Function: setting the seismic input in terms of response spectra

Parameters:

SET The identification name of the spectra' set. For each set, separate SPEC commands are entered. SET is referenced in the command DCASE (parameter INP)

type: TEXT limited by four symbols
unit: -
default value: -
limitations: see limitations for the text values of parameters

GROUP - identification name of the seismic group of supports. A separate SPEC command is entered for each group of supports.

type: TEXT
unit: -
default value: -
limitations: see limitations for the text values of parameters

INT - identifier of the interpolation for intermediate points. Depending upon the INT value the interpolation is performed over either linear or logarithmic scale.

type:	INTEGER
unit:	-
default value:	0
limitations:	0 (LIN-LIN); 1(LOG-LIN); 10(LIN-LOG); 11(LOG-LOG)

MULT - array of 3 numbers containing scaling factors for accelerations for each of the seismic excitation directions.

type:	REAL
unit:	-
default value:	1, 1, 1
limitations:	≥ 0

DISP - array of 3 numbers containing the components of seismic movements for the support group being specified.

type:	REAL
unit:	mm
default value:	0, 0, 0
limitations:	≥ 0

Parameter subcommands

FX, FY, FZ - array of frequencies, for which the response accelerations are set along the corresponding direction: X, Y, Z

type:	REAL
unit:	Hz
dimension:	array from 2 to 1000 elements
default value:	-
limitations:	> 0. Each subsequent element of the array shall be larger than the previous one.

AX, AY, AZ - array of response accelerations of seismic excitation along X, Y, Z directions

type:	REAL
unit:	**g**
dimension:	array from 2 to 1000 elements
default value:	-
limitations:	> 0.

Otherwise, it is possible to indicate the paths to files containing digital response spectrums in the form of the following commands:

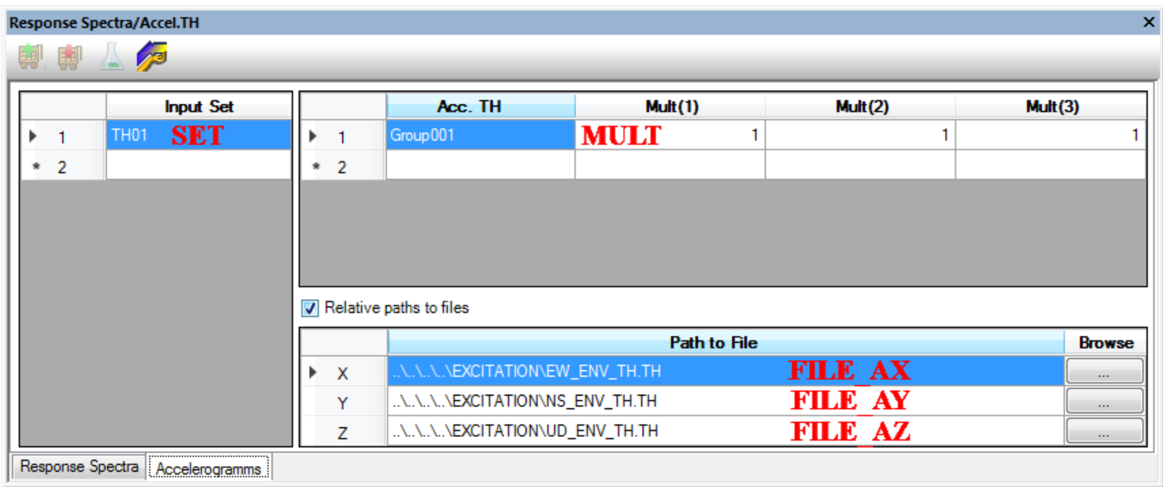
```
SPEC group = '03_60', int = 0, mult = 0.049, 0.049, 0.049, disp = 0, 0,
0
& file_sx = ".\SPECTRA\03.60_Y_SSE.DAT"
& file_sy = ".\SPECTRA\03.60_X_SSE.DAT"
```

```
& file_sz = ".\SPECTRA\03.60_Z_SSE.DAT"
```

```
Example:

SPEC GROUP 'RB'  MULT =  1.00  1.00  1.00
&FX =    0.01    0.20    1.27    2.68    3.06    3.40    3.74    5.06
5.75 \
         6.09    7.22    9.77   10.35   11.53   15.60   19.55   21.27
35.80   57.50 \
        100.00
&AX =    0.054    0.054    0.420    0.900    1.142    1.897    2.067
2.067 \
         1.745    1.725    0.982    0.982    0.777    0.669    0.669
0.578    0.483 \
         0.445    0.362    0.362
&FY =    0.01    0.20    1.27    2.04    3.06    3.57    5.06    7.47
8.40 \
         9.35   12.00   18.00   23.00   57.50  100.00
&AY =    0.054    0.054    0.344    0.684    1.406    2.215    2.215
0.974 \
         0.860    0.627    0.963    0.963    0.625    0.417    0.417
&FZ =    0.01    0.20    1.19    2.69    3.57    5.50    6.80    9.20
9.77 \
        12.30   14.45   19.55   21.27   27.60   31.32   57.50  100.00
&AZ =   0.028    0.028    0.130    0.269    0.483    0.483    0.401
0.401 \
        0.452    0.488    0.677    0.677    0.487    0.449    0.255
0.140    0.140
```

Seismic accelerograms (ACCE)

Interface for the input this data may be found in the "Response Spectra/Accelerograms" window, tab "Accelerograms":



type: general multi-line command

Function: setting the seismic input in terms of accelerograms

Parameters:

SET The identification name of the acceleration time histories set. For each set, separate ACCE commands are entered. SET is referenced in the command DCASE (parameter INP)

type:	TEXT limited by four symbols
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters

MULT **array of** 3 numbers containing scaling multipliers of accelerations for each of the seismic load direction

type:	REAL
unit:	-
default value:	1, 1, 1
limitations:	≥ 0

Parameter subcommands

FILE_AX[1)] name of file containing digital accelerogram values[2)] for X direction

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for string values of parameters. The string length shall not exceed 128 characters.

FILE_AY name of file containing digital accelerogram values for Y direction

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for string values of parameters. The string length shall not exceed 128 characters.

FILE_AZ name of file containing digital accelerogram values for Z direction

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for string values of parameters. The string length shall not exceed 128 characters.

Note:

1) *If the file with digital accelerogram values is in the working directory, then it is sufficient to specify only its name with extension. In the remaining cases, it is required to specify the full path to the file.*
2) *In describing the accelerograms, the first line of the file shall contain 2 numbers: NT - number of accelerogram digitizing points and the DT - the value of the accelerogram digitizing step. Upon settings these values, the NT accelerogram digitizing points shall be specified in an*

arbitrary format. It should be noted that the step and number of digitizing points for all three accelerograms shall be identical. ***The accelerations shall be specified in fractions of g!***

3) *If analysis specification contains two or more dynamic load cases or several sets of dynamic excitations, to run THA considering seismic anchor movement, the names of the sets of accelerograms and response spectra must match*

```
Example:

ACCE set = 'TH01', group = 'Group001', mult = 0.102, 0.102, 0.102
& file_ax = ".\ACC\OGIB_PND3_Y_k002_korr-acc.th"
& file_ay = ".\ACC\OGIB_PND3_X_k002_korr-acc.th"
& file_az = ".\ACC\OGIB_PND3_Z_k002_korr-acc.th"
```

Specification for analysis (SOLV)

type: general multi-line command

Function: assignment of the sequence and parameters for the execution of individual stages of analysis

Parameters:

NAME name of analysis (description)

type: STRING
unit: -
default value: -
limitations: see limitations for string values of parameters. The length shall not exceed 80 characters.

Subcommand

LC specification for an individual analysis (Load Case)

Subcommand parameters:

TYPE type of analysis

type: TEXT
unit: -
default value: -
limitations: the type of analysis shall either be specified (user type) or correspond to one of the following standard names: 'DSGN'; 'OPER_A'; 'OPER_B', 'OPER_R', 'SUST_A'; 'SUST_C'; 'MODAL', 'TEST', 'TEST_B', 'SAM' (see Note 1)

MOD - identification name of the piping system operating mode

type: TEXT
unit: -
default value: -

	limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command or corresponds to one of the following standard names: $COLD – "cold" state: assumes empty pipeline at ambient temperature (T=TA, P = 0, CSG = 0); $OPER – operating state corresponding to the first mode described by the OPVAL command (usually means Normal Operation of the system)

LOAD set of loads

type:	TEXT
unit:	-
default value:	-
limitations:	see (2)

PEND (see also Note**) pendulum or swing effect option. Takes into account in spring and rod hangers

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

FRIC support friction option

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

NLS[3)] status of supports nonlinearities

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'REF', 'LIN'

HNG_STF inclusion of stiffness from spring hangers in the general stiffness matrix

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO'

PE pressure elongation option (see Appendix IX)

type:	TEXT
unit:	-
default value:	'NO'

limitations: 'YES', 'NO'

NOTE Note / Comment

type: STRING
unit: -
default value: blank
limitations: see limitations for string values of parameters

Note:

1) Standard types of Load Cases assume the certain interpretation for analysis options:

Type of analysis	Load from spring hangers/supports	inclusion the spring hangers stiffness in the global stiffness matrix	Friction and swing effect	Inclusion of the vertical one way supports and rigid hangers in analysis
OPER_A	P_h	No	User	Yes
OPER_B	R_0	Yes	User	Yes
OPER_R	R_0	Yes	User	Yes
SUST_A	P_h	No	No	Yes
SUST_C	P_h	No	No	Reference (depending on their status on the previous Load Case)
DSGN	P_h	rigid/no	No	Yes
TEST	-	lock	User	Yes
TEST_B	R_0	lock/yes	User	Yes
MODAL	-	Yes	No	linearization
SAM	-	Yes	No	linearization

In the table above:

P_h - spring hangers presetting, hot load. This value could be defined by User, otherwise will be calculated by Program. In case of OPER_A type of analysis this value obtained from the Input Data, for OPER_B is taken from the previous Load Case

R_0 - Spring Hanger Installation or Theoretical Cold Load: entered as input data for OPER_R load case, in other cases is computed by dPIPE

For OPER_A/OPER_B friction and swing effects may be included in analysis by USER. SUST_C does not permit nonlinear effects. In the frame of MODAL analysis parameter RH_PND controls use of the additional lateral horizontal stiffness for rod hangers.

Nonlinear work of one way vertical restraints is considered for all types of analyses. Exceptions are: SUST_C and MODAL types. Within SUST_C analysis one way supports are considered depending *on their status on the previous, operating Load Case (NLS='REF'). MODAL analysis assumes that all restraints are linear.*

Load Case 'DSGN' is used for spring hanger design to determine design load (see Appendix VI). In doing so, the hangers, for which design load is not defined, are replaced with rigid vertical restraints. At the same time hangers with defined design loads are included in analysis by considering this load acting on the piping system.

The analysis of 'MODAL' type is used for determination of natural frequencies and mode shapes of piping system. These data is used in postprocessor for Response Spectrum or Time History Analysis.

'TEST' type of analysis is designed for simulation of hydraulic test procedure implemented just after piping installation: it's assumed that piping has no insulation and all springs are locked. Piping internal pressure, temperature and content are defined by OPVAL command. Stiffness of locked spring hangers is defined by RH_STF parameter.

The analysis of 'TEST_B' type is used for the simulation of the Hydraulic Test conducted during operating life of the piping system. It is assumed that before hydraulic test piping system is deformed. Some spring hangers may be locked being preset in this pre-testing operating state.

"OPER_R" type of analysis is designed for calculations when information for spring hanger installation load R_0 is available. In this case, the program does not check if the presetting load for the spring hanger is within the range from PMIN to PMAX

The Table below specify default and available options for standard types of analyses:

Parameter	Type of Analysis:										
	DSGN	OPER_A	OPER_B	OPER_R	SUST_A	SUST_C	MODAL	TEST	TEST_B	SAM	-
LOAD (*)	W	W+P+T+D+[...]	W+P+T+D+[...]	W+P+T+D+[...]	W+P+[F]	W+P+[F]	-	W+P+T+D	W+P+T+D	D	ANY
HNG_STF	RGD_SPR	NO	YES	YES	NO	NO	YES	RH_STF	YES/RH_STF	YES	NO/YES
NLS(**)	YES	YES	YES	YES	YES	REF	-	YES	YES	LIN	YES/REF
FRIC	NO	YES/NO	YES/NO	YES/NO	NO	NO	NO	YES/NO	YES/NO	NO	NO/YES
PEND	NO	YES/NO	YES/NO	YES/NO	NO	NO	(***)	YES/NO	YES/NO	NO	NO/YES

(*) LOAD is a text parameter consisting of pre-defined characters and delimiters ("+"):

***W** – weight load;*
***P** – pressure;*
***T** – temperature loads;*
***F** – concentrated forces;*
***D** – displacements of supports;*
***CS** - Cold Spring Load;*

__BOW__ - bending moment arising from the temperature stratification effect (see Appendix VIII)

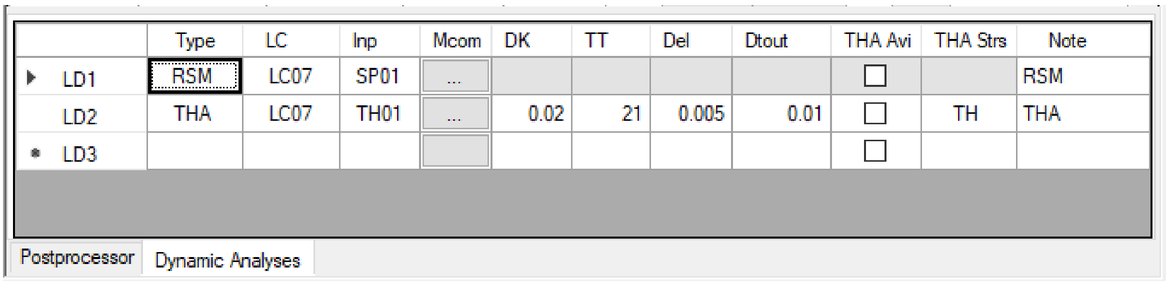
For 'OPER_A', 'OPER_B', 'OPER_R' types the following loads can optionally be included in […]: concentrated forces (F), cold spring loads (CS) and bowing loads from the stratification effect (BOW);

(**) NLS = 'REF' means that support nonlinearities are considered depending on the previous load case: for example, if a one way support has "uplifted" in the analysis under "full load" (W+P+T+D), then in the subsequent analysis, for which the option NLS = 'REF' is specified, this support will not be taken into account at all.

(***) PEND parameter for 'MODAL' analysis could be set as 'NO' or specified as 'LCXX', where XX is a number of the relevant Load Case for calculation of loads affecting on the lateral stiffness due to swing effect.

```
Example:

 SOLV "Analysis with determination of operating loads and selection of
springs (#1)"

 &LC MOD='$OPER' TYPE='DSGN' Note="Determination of working loads on
springs" ; LC1
 &LC MOD='$OPER' TYPE='OPER_A' PEND='NO' FRIC= 'NO' Note="Analysis for
full load" ; LC2
 &LC MOD='$COLD' TYPE='OPER_B' PEND='NO' FRIC ='NO' Note="Selection of
springs" ; LC3
 &LC MOD='$OPER' TYPE='OPER_B' PEND='YES' FRIC='YES' Note="Stage II
(full load)" ; LC4
 &LC MOD='$OPER' TYPE='SUST_C' Note="Stage I" ; LC5
 &LC MOD='$COLD' TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Stage IV
(cold load')"; LC6
```

Specification for Dynamic Analyses (DCASE)



DCASE command defines dynamic analysis cases and installs links between the modal load cases (see SOLV Command, MODAL type) and the set of the seismic or dynamic excitations (see SPEC, ACCE and DFRC commands). In the input data file, this command should be placed between SOLV and POST commands.

type: general multi-line command

Function: specification for the options of dynamic analyses

Parameter subcommand

LD

Subcommand parameter :

TYPE defines type of the dynamic analysis

type:	TEXT
unit:	-
default value:	DYN
limitations:	'RSM', 'THA'

LC reference on the modal analysis Load Case defined in the SOLV command

type:	TEXT
unit:	-
default value:	-
limitations:	LC should be defined in the command SOLV

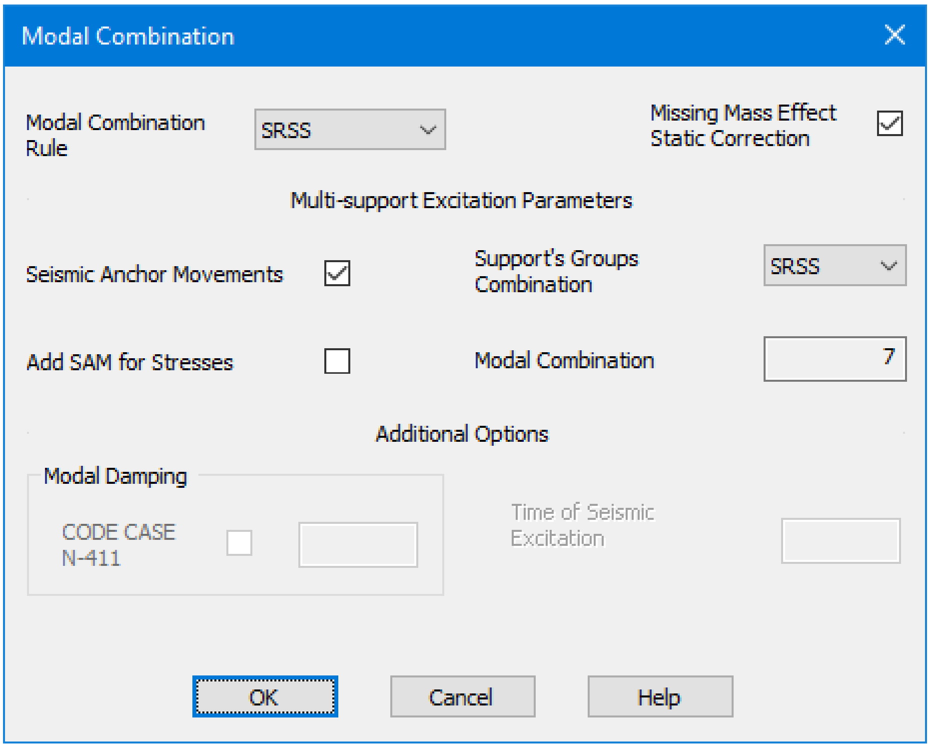
INP reference on the set of dynamic excitations (parameter SET of the SPEC/ACCE/DFRC comands)

type:	TEXT
unit:	-
default value:	-
limitations:	the set should be defined before the appearance of this reference

MCOM is the type of modal combination of seismic responses

Type:	INTEGER
Unit:	-
Default value:	2

The MCOM parameter is an integer, which bit mask contains several keys at once for setting the rules of the modal combination of seismic responses. In order to generate this parameter, it is recommended to use the following dialog from the DDE spreadsheet ("Service/Options/Control parameters/Dynamic" menu):



The fields of the given dialog can take the following values:

Determination of the rules of the modal combination of seismic responses

Dialog field	allowable values	Notes
Modal Combination Rule	SRSS	"Square root from the sum of squares" method
	NRC_GRM	Grouping method in US NRC edition
	NRC_TPM	10 % summation method in US NRC edition
	NRC_DSC	"Double sum" method in US NRC edition
	ISM_CQC	"Complete Quadratic Combination" method for multi-support excitation
	DSC	"Double sum" method
	CQC	"Complete Quadratic Combination" method
Missing Mass Effect Static Correction	Yes/No	
Multi-support Excitation Parameters		
Seismic Anchor Movements (SAM)	Yes/No	
Add SAM for stresses	Yes/No	
Support's Groups combination	ABS/SRSS	
Additional options1)		
Modal damping (DK) in fractions of the critical value	0.05	at DK = -1 the modal damping according to Code Case N-411 is assumed
Time (duration) of seismic excitation (TT), s	15.0	

1) Both parameters, DK and TT, are used for the "double sum" method (NRC_DSC and DSC); the DK parameter is used for the CQC (ISM_CQC, CQC) method.

TT - The total time of the the dynamic excitation. This parameter may be used also in the frame of RSM for the DSC combination rule (see MCOM).

type:	REAL
unit:	sec
Default values:	0
limitations:	≥ 0

DEL - integration step

Type:	REAL
Unit:	sec
Default values:	0
Range of possible values:	≥ 0

If DEL = 0, then the integration step will be computed by the program automatically. If the DEL1 integration step computed by the program is less than the specified value, then DEL = DEL1

DTOUT - the results output step in case of Time History Analysis

Type:	REAL
Unit:	sec
Default values:	DEL
Range of possible values:	≥ 0

DK - modal damping (fractions of critical). When RSM is used, DK is applicable as a parameter for CQC or DSC modal rule (see MCOM).

Type:	REAL
Unit:	-
Default values:	0
Range of possible values:	0 ≤ DK ≤ 1

THA_STRS - the method of stress computation in piping elements in case of analysis according to THA (DYN='THA')

type:	TEXT
unit:	-
default value:	'TH'
limitations:	'SRSS', 'TH' , 'FAST'

When THA_STRS = 'SRSS' the program will find maximum modal responses of the system by time integration of the equations of motion. In doing so, for computing stresses in the piping system, a procedure is used, which is similar to the analysis according to RSM. It is a "fast" but less accurate stress evaluation method than that when THA_STRS = 'TH'. In the latter case, the stresses are to be found for each integration step, and the maximum values are indicated in the printout. THA_STRS = 'FAST' corresponds to the method of "fast" computation of response parameters of the piping system based on the analysis of system kinetic energy maximums for the time of integration of the equations of motion. It is recommended to use this option in debugging calculations of models when the main analysis time falls to integration of the equations of motion. From the experience of calculations performed it follows *that the use of this options allows to correctly determine the response parameter maximums ~ in 90% of cases. For final calculations the procedure for response parameter computation at each integration step shall be used. For RD and PNAE Codes stresses dependent from the dynamic transient loads are calculated several times for selected time points, while for all other Norms stresses are calculated once based on maximal components of elements loads.*

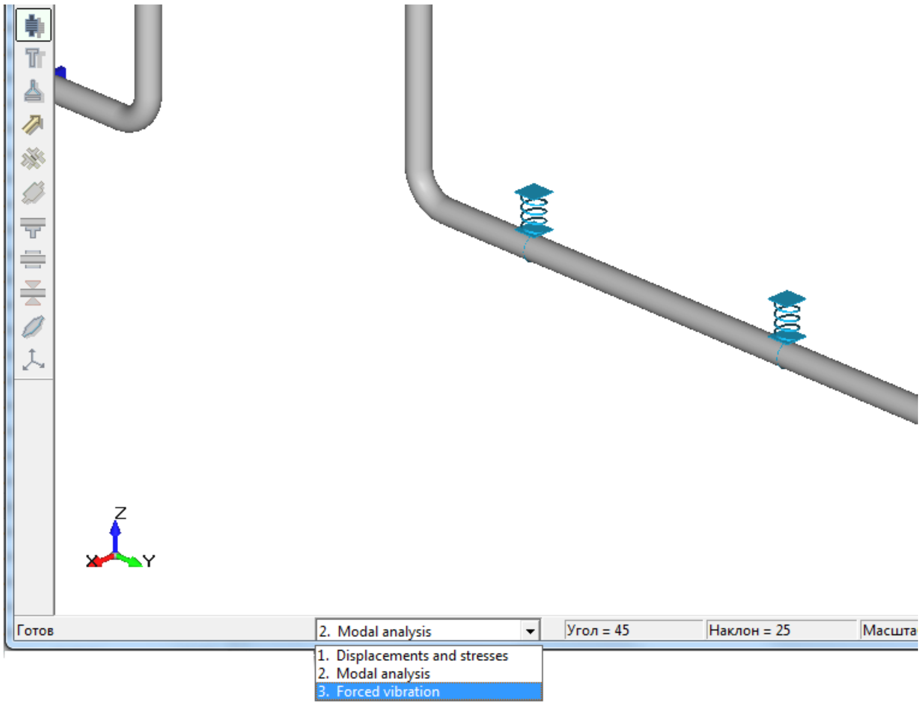
Printout of the THA results is carried out according to the table given below:

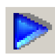
output value	*maximum*
stresses S2_MRZ, S2_PZ1, S2_PZ2, S_I_PZ (code = 'PNAE'/'PNAE_HT'/'RD')	*maximal values are printed for stresses calculated on each time step*
stresses EQ9_B, EQ9_C, EQ9_D, SGM2, SOL (code = 'ASME_NB'/'ASME_NC'/, 'B311'/'EN')	*calculated for maximal value of* $M=\sqrt{(M_x^2+M_y^2+M_z^2)}$
internal forces for PIPE, BEND and REDU elements	*all 6 components are printed for time when* $M=\sqrt{(M_x^2+M_y^2+M_z^2)}$ *is maximum*
internal forces for all elements except PIPE, BEND and REDU	*output of the maximal magnitude for each directional component independently*
displacements	
accelerations	
support's loads	
dampers (loads and displacements)	*independent output for horizontal (X and Y SRSSSquare Root of Sum Squares) and vertical directions*

THA_AVI - a flag for the output of forced vibration animation in performing the analysis according to THA (DYN='THA')

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES' or 'NO'

When THA_AVI = 'YES' the program will record the deformed state of the system for each time step. These results can be viewed thereafter in the PIPE3DV program by selecting the "Forced vibration" line in the bottom window with the list of output results:



and then it is possible to use  *and*  *buttons for viewing the animation. It should be kept in mind that when the forced vibration is recorded the size of the <model>.bin file will increase significantly!*

NOTE Note / Comment

type:	STRING
unit:	-
default value:	-
limitations:	see limitations for the string values of parameters

```
Example:

DCASE
& LD  type = 'RSM', lc = 'LC7', inp = 'SET1', mcom = 6 ; LD1
& LD  type = 'THA', lc = 'LC7', inp = 'THA', mcom = 6, dk = 0.02, tt = 20, del =
0.001, dtout = 0.001 ; LD2
```

Specification for postprocessor (POST)

The POST command defines the specification for post-processing of the results and shall be placed in the input data file only after SOLV command. The values of parameters to be used in the POST command depend upon the piping stress codes. The following codes are used in this version: Russian Nuclear Code PNAE [REF 1], Russian Power Piping Code RD [REF 2], American Nuclear Code ASME BPVC NB/NC-3600 [REF 3], European Code EN 13480-3 [REF 10], American Power Piping Code ASME B31.1 [REF 12].

type: general multi-line command

Function: specification for post-processing of the results.

Parameter subcommand

RES type of the results being processed

type:	TEXT
unit:	-
default value:	-
limitations:	see (1)

Subcommand parameter :

LS load set: specification for results output

type:	TEXT
unit:	-
default value:	-
limitations:	see (2)

RULE rule for results processing

type:	TEXT
unit:	-
default value:	'SUM'
limitations:	see (3)

OUT option for printing and output of results

type:	TEXT
unit:	-
default value:	'YES'
limitations:	'YES', 'NO'

NOTE Note / Comment

type:	STRING
unit:	-
default value:	-
limitations:	see limitations for the string values of parameters

Note:

1) RES =

Designation:	Description
DISP	displacements
SUPP	support reactions
FORC	internal forces

Stresses depending upon the piping stress codes:

CODE = 'PNAE', 'PNAE_T', 'NTD_ACI':

S1	Stresses of S1 category. Allowable values are calculated for the same service level as the previous load set.

S2_NUE	stress of S2 category (NOC - Normal operating conditions)
S2_NNUE	stress of S2 category (AOC - Abnormal operating conditions)
S2_MRZ	stress of S2 category (SSE)
S2_PZ1	stress of S2 category (OBE, category 1)
S2_PZ2	stress of S2 category (OBE, category 2)
SRK	stress of SRK category (for High Temperature Piping see Appendix V)
SAF	stress of SAF category **(Load set should consist of two arguments!!!)**
S2_HDR	stress of S2 category (hydraulic tests)

CODE = 'RD':

S_I	effective stress according to stage I
S_II	effective stress according to stage II
S_III	effective stress according to stage III
S_IV	effective stress according to stage IV
S_I_PZ	effective stress for NOC + OBE
S_H	effective stress under stage I for hydraulic test mode

CODE = 'ASME_NC':

EQ8	stress due to sustained loads (design conditions), equation (8) NC-3652
EQ9_B	stress due to sustained and occasional loads for level B Service Limits, equation (9), NC-3653.1
EQ9_C	stress due to sustained and occasional loads for level C Service Limits, equation (9), NC-3654
EQ9_D	stress due to sustained and occasional loads for level D Service Limits, equation (9), NC-3655
EQ10	stress due to thermal expansion, equation (10), NC-3653.2-a
EQ10A	stress due to single nonrepeated anchor movement, equation (10a), NC-3653.2-b
EQ11	stress due to a combination of sustained loads and thermal expansion, equation (11), NC-3653.2-c
EQ11A	stress due to reverse loads, equation (11a), NC-3653.2-d

CODE = 'ASME_NB':

EQ9_DC	stress due to sustained loads, Consideration of Design Conditions, equation (9) NB-3652
EQ9_B	stress due to sustained loads compatible with Service Limit B, equation (9) NB-3654
EQ9_C	stress due to sustained and occasional loads for level C Service Limits, equation (9), NB-3655
EQ9_D	stress due to sustained and occasional loads for level D Service Limits, equation (9), NB-3656
EQ10	Primary Plus Secondary Stress Intensity Range, equation (10), NB-3653.1
EQ11	Peak Stress Intensity Range, equation (11), NB-3653.2
EQ12	Simplified Elastic-Plastic Discontinuity Analysis, equation (12), NB-3653.6-a
EQ13	the primary plus secondary membrane plus bending stress intensity, excluding thermal bending and thermal expansion

	stresses, equation (13), NB-3653.6-b

CODE = 'EN':

SGM1	longitudinal stress due to sustained loads, equation (12.3.2-1)
SGM1T	longitudinal stress due to sustained loads under Test Conditions: see equation (12.3.2-1). Allowable stresses are defined according chapter 5.2 through the material data (see parameter ST)
SGM2	longitudinal stress due to to sustained and occasional or exceptional loads, equation (12.3.3-1)
SGM3	Stress range due to thermal expansion and alternating loads, equation (12.3.4-1)
SGM4	stress range due to sustained (weight + pressure) and alternating loads, equation (12.3.4-2)
SGM5	Additional conditions for the creep range: stress range due to sustained (weight + pressure) and alternating loads, equation (12.3.5-1)
SGM6	Stresses due to a single non-repeated support movement,equation (12.3.6-1)

CODE = 'ASME_B311':

SL	stress due to sustained loads
SOL	stress due to sustained and occasional loads
SE	stress Due to Displacement Load Ranges

Appendix V contains recommendations and rules for composing of typical specifications for analysis and post-processing.

Stress evaluation according to Russian RD 10-249-98 [REF 2] for Hydraulic Test mode is conducted in dPIPE with use of S_H stress category. It corresponds to the stress calculation for the first stage of analysis. The allowable stress for this category shall be calculated as follows: in the procedure for determination of the trial pressure (Item 2.8 of RD), an allowable stress is defined according to Table 2.8, where the value of [σ] is taken as [σ] = $\sigma_{0,2}$/1.1 for carbon, heat-resistant and austenite steel (rolled and forge). Strength criterion set for the first stage of analysis is $\sigma_{eff} \le 1.1[\sigma]$. So the allowable stress for the hydraulic test mode shall be written in the following form: $\sigma_{add.\ h.t.} = \sigma_{0.2}$.

If the input data of the dPIPE model contain the yield strength values for the materials being used (SY parameter, MAT command), then the program will directly calculate the allowable stress values in accordance with the formula given above. In case of such data being missing, it is assumed that $\sigma_{0.2}$ = 1.5*[σ] where nominal allowable stress [σ] corresponds to the SA parameter of the MAT command. An example of the specification for analysis for the hydraulic test mode is given in Appendix V.

2) *LS parameter determines how the results of analysis should be combined in order to obtain the required values. The expression for LS may contain references to the Load Cases (LC) described in the SOLV command, to the dynamic load case (LD) defined in DCASE command and to the Load Sets (LS) being created by the previously defined post-processor commands. For example, the expression 'LC2-LC1+LS1' defines that within the current LS it is necessary to generate the results as the difference between the values of LC N 2 and LC N 3 + the value from the N 1 set described earlier. The following characters are used as delimiters between the LC and LS sets: "+" (plus), "-" (minus) "," (comma) " " (space), ":" (colon). The "+" and "-"*

characters determine the sign of weight coefficients. The " " (space) and "," (comma) characters denote the list of sets, to which the rule will then be applied as determined by the RULE parameter. The ":" (colon) character is used for a short form of listing of the range of values. For example, the record LS = 'LC1:LC3' is equivalent to the record LS = 'LC1, LC2, LC3' . ***The order of references defined in the specification is important. Firstly should come references for Load Cases (LC), then references on dynamic analyses (LD) and finally Load Sets (LS).***

3) *RULE =*

- ***'SUM'*** *summation of sets;*
- ***'RANGE'*** *search with determination of the maximum difference.*
- ***'REF'*** *correction of the elements internal forces to the "cold" (reference) modulus of elasticity (to be used only in combination with RES = 'FORC'): F = (Ec/Eh)*F ;*
- ***'SEISM'*** *"seismic" rule of component summation for most "unfavourable" case: if the LS parameter is specified as LS = 'LC1+LC4', then the resulting value will be determined as the sum of absolute values from the set of |LC1i| + |LC4i|, and the sign shall be taken according to the first summand. For example:*

```
LC1:        43295    70 -3425    928 -2626   -137
LC4:        43041   938  2078  22344  3939   1379
SEISM:      86336  1008 -5503  23272 -6565  -1516
```

  the rule: RULE = "SEISM" is used only in the combination with RES = 'FORC', 'DISP' u 'SUPP';
- ***'MAX'*** *determination of maximum in absolute value of the components of displacements, supports reactions, stresses and element's forces and moments*
- ***'ABS'*** *summation of sets in absolute value*
- ***'H_REL'*** *identifier of the transfer of loads to the supports with the account of relaxation of temperature forces. It is used only for high-temperature piping systems in combination with RES = 'SUPP'*
- ***'C_REL'*** *identifier of the transfer of loads to the supports with the account of cold springing. It is used only for high-temperature piping systems in combination with RES = 'SUPP'*
- ***SRSS*** *combination by Square Root of the Sum of Squares rule. The rule is applicable for the internal forces ('FORC'), support's reactions (SUPP) and displacements ('DISP')*
- ***S_SRSS*** *"seismic" SRSS rule: magnitude of resulting value is SRSS of two combined componenets and the sign is taken by the first term. The rule is applicable for the internal forces ('FORC'), support's reactions (SUPP) and displacements ('DISP')*
- ***SAM*** *option to extract displacements, internal forces and supports reactions from the action of Seismic Anchor Movements. Applicable for RES = 'DISP'/'SUPP'/'FORC'. The LOAD parameter should refer on the dynamic load case (LD). It is also possible to use this option for calculating stresses from the action of the Seismic Anchor Motion for Service Levels B, C, and D (RES = 'EQ_9B (C, D)') in accordance with ASME NB/NC Code edition 2010 and later*
- ***RVRS*** *option to indicate that applied dynamic loads are considered as "reversing loads" for piping stress analysis. Compliant with ASME NB/NC Code Revision 2010 and later (RES = 'EQ_9B (C, D)') and EN code, Revision 2020 (RES = 'SGM2')*

4) If the multiplier of the first term in the "seismic" combination is set to zero, then internal pressure for stress calculations is set to zero as well. It allows to estimate the contribution of the "pure" seismic loads in stresses:

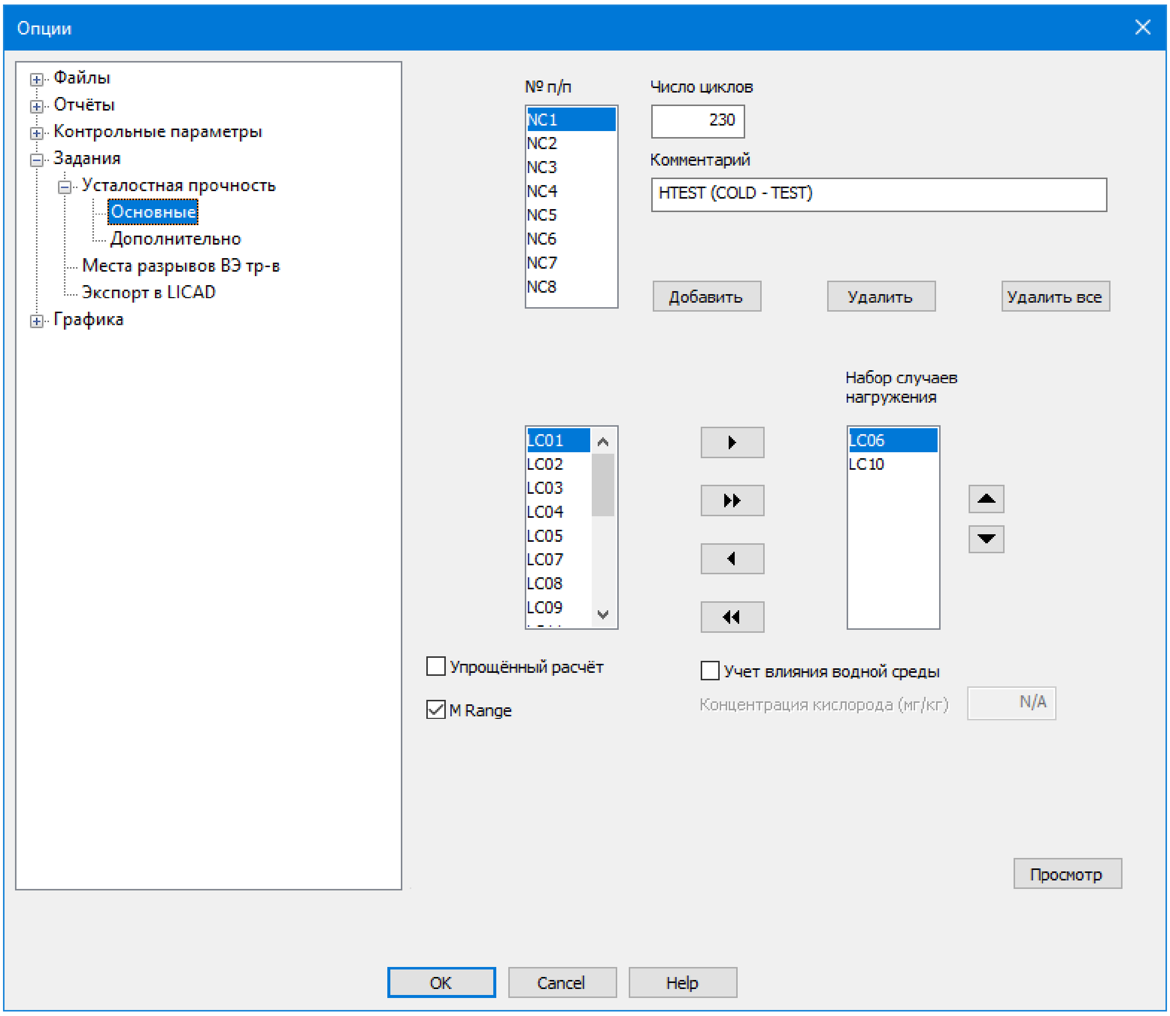
& res = 'S2_MRZ', ls = "**0***LC2+LC4", note = "Stresses S2 (SSE)" ;
LS4

```
Example:
```

```
 POST
 &RES='S2_NUE' LS="LC2" RULE='SUM' OUT='YES' NOTE="Stress of S2
category"
 &RES='SRK' LS="LC1-LC3" RULE='SUM' OUT='YES' NOTE="Stress of SRK
category"
 &RES='SAF' LS="LC1-LC3" RULE='SUM' OUT='YES' NOTE="Stress of SAF
category"
 &RES='DISP' LS="LC2" RULE='SUM' OUT='YES' NOTE="Weight movements"
 &RES='DISP' LS="LC1-LC3" RULE='SUM' OUT='YES' NOTE="Thermal expansion"
 &RES='SUPP' LS="LC1" RULE='SUM' OUT='YES' NOTE="Hot Loads"
 &RES='SUPP' LS="LC3" RULE='SUM' OUT='YES' NOTE="Cold loads"
```

Specification for fatigue analysis (FATG)

The FATG command defines the specification for fatigue analysis and shall be placed in the input data file only after description of the SOLV/*POST* command. The following dialog corresponds to this command in the DDE interface:



type: general multi-line command

Function: specification for piping Fatigue Analysis and calculation of the Cumulative Usage Factor (CUF)[1].

Parameter

FATG_SAF flag for "simplified" analysis (available for CODE = 'PNAE', 'PNAE_T', 'ASME_NB') in order to determine the individual design number of cycles in case of system transients from one operational state into another. With the option being activated, it is allowed to specify only paired load cases for each of the transients under consideration.

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

M_RANGE option to use ***range*** *of mechanical loads* for equation (13) according to ASME NB-3600 or SRK" stress for PNAE in case of analysis according to the simplified elastic-plastic analysis (see PNAE_KE parameter).

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

FAT_ENV option if the effect of water environment will be used for calculation of the fatigue curves. Effective only if CODE = 'PNAE' and CODE_YEAR= 2022. See Appendix XVII for details

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

KO global value for water oxidation. Could be redefined by ENVFAT command.

type:	REAL
unit:	mg/kg
default value:	0.
limitations:	≥0.

Subcommands:

NC number of cycles for the loading history specified

type:	INTEGER
unit:	-
default value:	-
limitations:	> 0

Subcommand parameter :

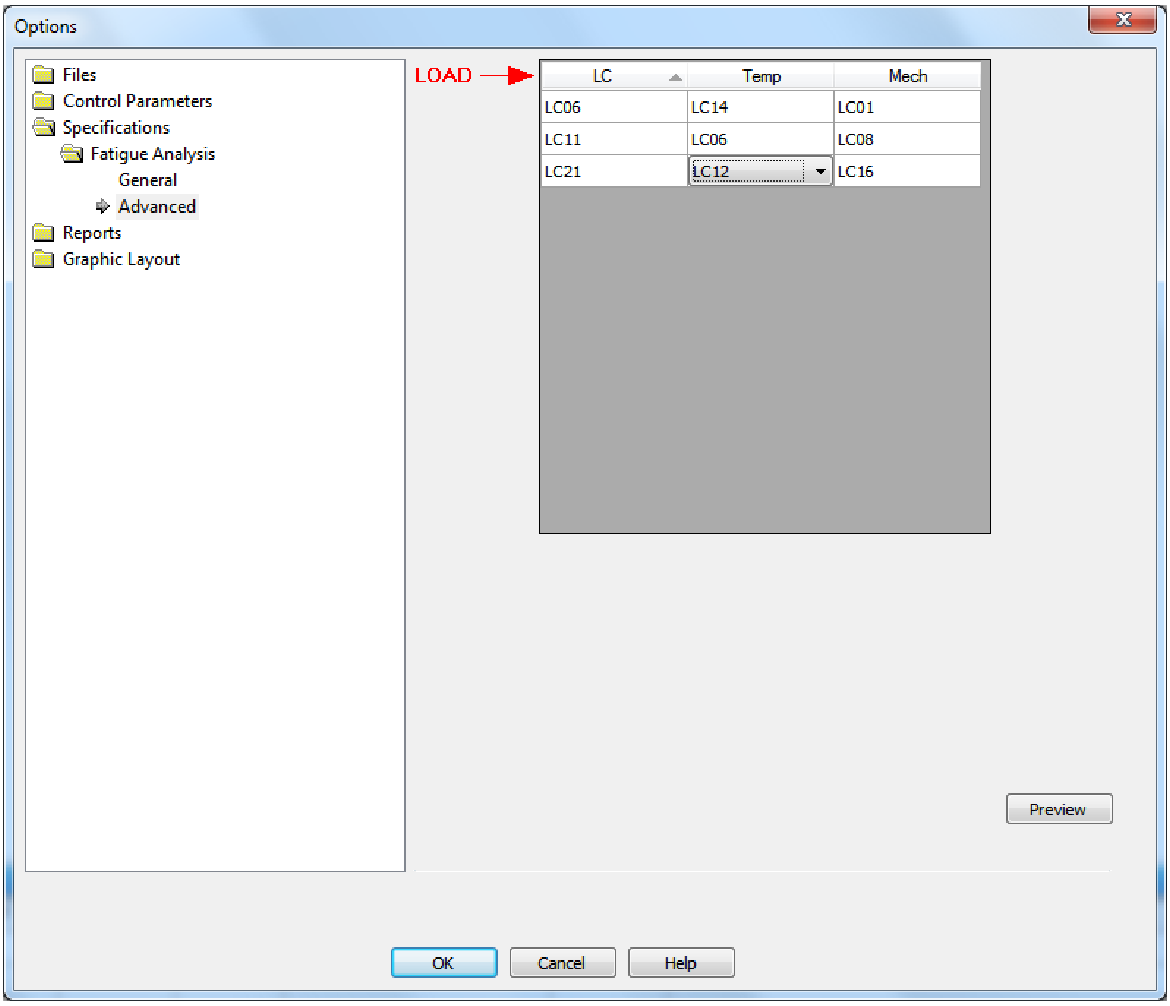
SEQ loading history, sequence of load cases or load sets

type: [TEXT](#)
unit: -
default value: -
limitations: load cases (load sets) described preliminarily in the SOLV (POST) command

NOTE Note / Comment

type: [STRING](#)
unit: -
default value: [blank](#)
limitations: see limitations for the string values of parameters

Additional data for fatigue strength analysis:



LOAD reference to LC or LS that already included in SEQ parameter

type: [TEXT](#)
unit: -
default value: -

limitations: load cases or load sets described in one of the NC subcommands (SEQ parameter)

Subcommand parameter :

TEMP reference to LC or LS, in which the temperature loads are determined for the account in equation (12) according to ASME NB-3600 or SRK' stress according to the simplified elastic-plastic analysis procedure (see PNAE_KE parameter).

type:	TEXT
unit:	-
default value:	-
limitations:	load cases or sets of results described in one of the NC subcommands (SEQ parameter)

MECH reference to LC or LS, in which the mechanical loads are determined for the account in equation (13) according to ASME NB-3600 or SRK" stress according to the simplified elastic-plastic analysis procedure (PNAE_KE parameter).

type:	TEXT
unit:	-
default value:	-
limitations:	load cases or sets of results described in one of the NC subcommands (SEQ parameter)

Example:

```
FATG
& nc = 3000, seq = "LC1, LC6"
& nc = 3800, seq = "LC1, LC4, LC6"
& nc = 10000, seq = "LC1, LC2, LC3, LC4, LC5, LC6"
```

Note:

(1) The fatigue strength analysis shall be performed only for the stress codes of PNAE (CODE = 'PNAE'), NTD A.C.I. (CODE='NTD_ACI') and ASME BPVC NB-3600 (CODE= 'ASME_NB'). For piping working in the creep range this command is available only for PNAE (CODE = 'PNAE_T') with option simplified analysis.

(2) An example of the fatigue analysis is given in [REF 11]

Specification for HELB analysis (POST_HELB)

POST_HELB provides parameters for High Energy Lines Break (HELB) analysis (location of intermediate postulated ruptures). Command is applicable only for the following Codes: PNAE [REF

1] or ASME NC/NB Code [REF 3] or EN Code [REF 10]. The command should be located after SOLV and POSTspecifications. See Appendix XVIII for HELB analysis criteria and its adaptation for different Codes.

Type: general multi-line command

Function: Specification for HELB Analysis

Parameters:

CUF

Criterion for Cumulative Usage Factor (CUF) calculated from Fatigue analysis and used to locate piping rupture

type:	REAL
unit:	-
default value:	0.4
limitations:	>0; < 1

RUPTURE

Scale factor used for allowable stresses to locate piping rupture

type:	REAL
unit:	-
default value:	0.8
limitations:	>0; <= 1

CRACK

Scale factor used for allowable stresses to locate through-wall crack

type:	REAL
unit:	-
default value:	0.5*RUPTURE
limitations:	>0; <= 1

MECH_LS

Reference on Load Set (POST command) that depends on Code used for analysis. For PNAE and ASME_NC refers to Load Set for stresses due to Sustained Loads + OBE. For ASME_NB Class 1 analysis refers to the stress range excluding thermal expansions analysis

type:	TEXT
unit:	-
default value:	-
limitations:	PNAE Code: LS for S2_PZ1, S2_PZ2, ASME_NC Code: LS for EQ9_B ASME_NB Code: LS for EQ13

RANGE_LS

Reference on Load Set (POST command) with thermal expansion stress

type:	TEXT
unit:	-

default value: 'FATG' (range stresses calculated within fatigue analysis procedure. Command FATG
limitations: PNAE Code: LS for 'SRK' or 'FATG'
ASME_NC Code: LS for EQ10
ASME_NB Code: LS for EQ12 or 'FATG'

HELB_STR prefix used in the name of piping classified as High Energy piping. If prefix does not specified all pipes in the model are considered as High Energy.

type: TEXT
unit: -
default value: -
limitations: 8 symbols length

DOC an identifier for selecting a criterion document by which the locations of the postulated piping ruptures are determined. There are two options available: DOC = 'ANSI' and DOC = 'SRP'. In the first case, the criteria are determined according to the document ANSI/ANS 58.2 [REF 19], in the second in accordance with SRP Section BTP 3-4, NUREG-0800, [REF 20]

type: TEXT
unit: -
default value: blank
limitations: blank, 'ANSI', 'SRP'

```
Example:
```

POST_HELB CUF = 0.4, RUPTURE = 0.8, MECH_LS = 'LS03', RANGE_LS = 'FATG', HELB_STR = 'HELB'

Specification for report generation (POST_REP)

The POST_REP command specify options for generation of results summary tables. The command shall be placed in the input data file after the SOLV and POST commands.

type: general one-line command

Function: generation of results summary tables

Parameters:

REP_TYPE user-defined type of report

type: TEXT
units: -
default value: -
limitations: 'SPBAEP', 'MOAEP', 'NIAEP', 'ADVANCED'

LOAD_HOT reference to the Load Case (SOLV command) that defines Hot Load for Spring Hangers.

type: TEXT
units: -
default value: -

	limitations:	existing Load Cases specified by SOLV command

LOAD_COLD

reference to the Load Case (SOLV command) that defines Cold Load for Spring Hangers.

type:	TEXT
units:	-
default value:	-
limitations:	existing Load Cases specified by SOLV command

LOAD_DES

reference to the Load Case (SOLV command) that defines Design Load for Spring Hangers

type:	TEXT
units:	-
default value:	-
limitations:	existing Load Cases specified by SOLV command

LOAD_HT

reference to the Load Case (SOLV command) that calculates Operating Load for Spring Hangers corresponding to the Hydraulic Test

type:	TEXT
units:	-
default value:	-
limitations:	existing Load Cases specified by SOLV command

LOAD_SEISM

reference to the Dynamic Load Case (DCASE command) or Load Set with type = 'SUPP' (POST command) representing seismic loads to be printed in supports' summary tables

type:	TEXT
units:	-
default value:	LD1
limitations:	existing Load Cases specified by SOLV command

SKIP_SUP

identifier for designation of the support in order to exclude it from the results. The set of characters determined by the SKIP_SUP parameter shall be placed at the beginning of the string parameter NOTE (for example, if SKIP_SUP = "$", then the following record shall be present in the command for the support being excluded:
2450: SUP note = "$support")

type:	TEXT
units:	-
default value:	'$_NULL'
limitations:	length up to 8 characters

SKIP_STR

identifier for designation of the piping segment in order to exclude it from the results. The set of characters determined by the SKIP_STR parameter shall be placed at the beginning of the string parameter

NAME (for example: *10: F dc = -1, 1, 0, cs = '630x10', lg = 'LG1', name = "$"*)

type:	TEXT
units:	-
default value:	'$_NULL'
limitations:	length up to 8 characters

ANC_CS

option for the output coordinate system for the anchor and 6-component supports reactions printed in the summary tables with results (file *.sup)

type:	TEXT
units:	-
default value:	'G' (global c.s.)
limitations:	'G' (global c.s..) , 'L' (local c.s.)

RSTR_CS

option for the output coordinate system for the restraints reactions printed in the summary tables with results (file *.sup)

type:	TEXT
units:	-
default value:	'G' (global c.s.)
limitations:	'G' (global c.s..) , 'L' (local c.s.)

SUP_CRD

option for additional output of the global coordinates in the summary tables with results containing supports reactions (file *.sup, option is valid for all types of supports)

type:	TEXT
units:	-
default value:	'NO'
limitations:	'YES', 'NO'

SUP_SKIP

признак автоматического исключения опор, расположенных на участках трубопровода с меткой SKIP_STR, из результирующих таблиц

type:	TEXT
units:	-
default value:	'YES'
limitations:	'YES', 'NO'

SKIP_OUT

flag of automatic exclusion of the supports located at the piping segments with the SKIP_STR mark from the all resulting tables

type:	TEXT
units:	-
default value:	'NO'

limitations: 'YES', 'NO'

OTT_W

reference to the Load Case (SOLV command) or the Load Set (POST command), which contains calculated weight loads. It is used for the evaluation of valve nozzles(2).

type: TEXT
units: -
default value: -
limitations: existing Load Case specified by SOLV command or existing Load Set (POST command) with type = 'FORC'

OTT_T

reference to the Load Case (SOLV command) or the Load Set (POST command), which contains calculated thermal expansion loads. It is used for the evaluation of valve nozzles(2).

type: TEXT
units: -
default value: -
limitations: existing Load Case specified by SOLV command or existing Load Set (POST command) with type = 'FORC'

OTT_PZ

reference to the Load Set (POST command), which contains combination of sustained (weight) and seismic loads corresponding to OBE. It is used for the evaluation of valve nozzles(2).

type: TEXT
units: -
default value: -
limitations: existing Load Set (POST command) with type = 'FORC'

OTT_MRZ

reference to the Load Set (POST command), which contains combination of sustained (weight) and seismic loads corresponding to SSE. It is used for the evaluation of valve nozzles(2).

type: TEXT
units: -
default value: -
limitations: existing Load Set (POST command) with type = 'FORC'

OTT_AS

reference to the Load Set (POST command), which contains combination of the sustained (weight) and accidental piping break loads . It is used for the evaluation of valve nozzles(2).

type: TEXT
units: -
default value: -
limitations: existing Load Set (POST command) with type = 'FORC'

ECOLD

option to ignore scaling of the internal forces and moments to the cold (reference) Young Modulus. If ECOLD = 'YES' internal forces/moments are kept unchanged. This parameter suppresses E_MOD option for stress calculation.

type: TEXT
units: -
default value: 'NO'
limitations: 'YES', 'NO'

SRK_RMT

option to take into account tensile stresses for calculation of allowable stresses for SRK stress category (in accordance with item 5.4.7 of PNAE), [*REF 1*].

type: TEXT
units -
default value: 'NO'
limitations: 'YES', 'NO'

IL_SBP

reference to the one of the stress categories taken into account the seismic inertia loads for small bore pipes, see Appendix XV

type: TEXT
units -
default value: -
limitations: reference to the Load Set (LS) previously defined by the POST command. Depending on the CODE it should be type of: S2_MRZ, S2_PZ1, S2_PZ2, S_I_PZ, EQ9_B, EQ9_C, EQ9_D, SOL, SGM2, SOL

IL_LBP

reference to the one of the stress categories taken into account the seismic inertia loads for "big" diameter pipes, see Appendix XV

type: TEXT
units -
default value: -
limitations: reference to the Load Set (LS) previously defined by the POST command. Depending on the CODE it should be type of: S2_MRZ, S2_PZ1, S2_PZ2, S_I_PZ, EQ9_B, EQ9_C, EQ9_D, SOL, SGM2, SOL

SL_SBP

reference to the one of the stress categories taken into account secondary seismic loads for small bore pipes (Seismic Anchor Movement), see Appendix XV

type: TEXT
units -
default value: -
limitations: reference to the Load Set (LS) previously defined by the POST command. Depending on the CODE it should be type of: SRK, S_III, EQ10, SGM3, SE

```
Example:

POST_REP load_hot = 'LC4', load_cold = 'LC6', load_des = 'LC1', skip_str = '$', skip_out = 'ye
```

Note:

(1) See. Appendix VI.;
(2) Reference to the values of permissible loads shall be determined by the *OTT_REF* parameters for the commands describing valves of various configuration. The permissible loads shall be set in the vlv_ott.dbs file.
(3) In case when REP_TYPE = 'ADVANCED' is set, the form of summary tables in the *.sup file changes, and the results of spring design are brought into conformity to the SPR_SFPMIN, SPR_VARTOL and SPR_TRTRAV parameters. See also Appendix VI

Specification for Support's Allowable Loads (SUP_LOADS)

Command SUP_LOADS is used to specify correspondence between sets of Support's allowable Loads defined in file SUP_LDS.MDB and Load Sets and Load Cases defined by SOLV and POST commands.

Type: general multi-line command

Parameters:

TABLE

reference on standard's name

type: STRING
units: -
default value: -
limitations: should coincide with one of the standards names existing in file SUP_LDS.MDB (see field Table Name, Appendix XII)

Subcommand:

MODE

The reference name of the operation mode or load combination for which the calculated and allowable loads are compared. The mode name must strictly correspond to the names predefined in the SUPPORTS (MOD1, MOD2, MOD3) or LOADS tables ("Labels" column) in the SUP_LDS.MDB file

type: TEXT
units: -
default value: -

limitations: predefined names in SUPPORTS or LOADS tables in SUP_LDS.MDB file

LOAD reference to the Load Case (SOLV command) or the Load Set (POST command), which contains supports reactions to be evaluated versus allowable values defined in the database, SUP_LDS.MDB file.

type: TEXT
units: -
default value: -
limitations: existing Load Case specified by SOLV command or existing Load Set (POST command) with type = 'SUPP'

Notes:

1) The previous syntax of this command is acceptable as well:

```
SUP_LOADS table = "LISEGA 2010RS", sup_mod1 = 'LC1', sup_mod2 =
'LS7', sup_mod3 = 'LS8'
```

but after editing the dp5 file in the DDE spreadsheet it will be converted in a new format:

```
SUP_LOADS  table = "LISEGA 2010RS"
&  mode = 'НУЭ', load = 'LC1'
&  mode = 'НУЭ+МРЗ', load = 'LS7'
&  mode = 'НУЭ+ПЗ',  load = 'LS8'
```

2) it is allowed to have several different tables with permissible support loads in the same dp5 file

Examples:

```
SUP_LOADS  table = "LISEGA2010RSENR2"
&  mode = 'NOC', load = 'LC4'
&  mode = 'NOC+OBE', load = 'LS11'
SUP_LOADS  table = "TITAN-2_R1_7"
&  mode = 'NOC', load = 'LC5'
&  mode = 'ANOC', load = 'LC4'
&  mode = 'NOC+SSE', load = 'LS3'
```

Export of piping Supports Loads to LICAD (DP2LCD)

The DP2LCD command is used to specify parameters for exporting a set of loads from dPIPE to LICAD®. See Appendix XVI.

Type: general multi-line command

Parameters:

LCD_VER LICAD version.

type: TEXT
units: -
default value: LICAD_RS_EN
limitations: 'LICAD_RS_EN' or 'LICAD_10'

ANGLE The permissible angle of deflection of the support from the direction of the axes perpendicular to the axis of the pipe. If the deviation is exceeded, the program gives a warning or error.

type: [REAL]
units: degree.
default value: 2.5°
limitations: ≥ 0°; ≤ 30°

SH_MODE Mode set in the LICAD for selection of the springs

type: TEXT
units: -
default value: 'COLD'
limitations: 'COLD' or 'HOT'

SH_MODE parameter defines the way how LICAD will treat spring hanger load. If the COLD mode is defined, then HOT loads will be recalculated through the spring stiffness and vertical travel. This value should corresponds to the LICAD settings

SKIP Control of the support's data to be exported from dPIPE to LICAD

type: TEXT
units: -
default value: 'STRICT'
limitations: 'KKS'; 'DP5'; 'STRICT'; 'OFF'

SKIP may have the following values

KKS – *Export for only supports having KKS identification*

DP5 – *Export for only supports having output according to dPIPE rules (see SKIP_SUP and SUP_SKIP)*

STRICT – *both KKS and DP5 rules are applicable + checking if supports are located in the nodes belonged to pipes*

OFF – *all checks are disabled, loads for all supports are exported*

S_CAT seismic category of piping

type: TEXT
units: -
default value: 'I'
limitations: 'I'; 'II'; 'III'

Sub-command parameters:

TYPE predefined type of loads

type: TEXT
units: -
default value: -
limitations: depends on LCD_VER, see Appendix XVI

LOAD Combination of loads for output

type: TEXT
units: -
default value: -
limitations: See Note (1)

SF amplification factors compatible with used Code

type:	REAL
units:	-
default value:	1.
limitations:	> 0, see. Notes (2)

NOTE Note/Comment

type:	STRING
units:	-
default value:	-
limitations:	-

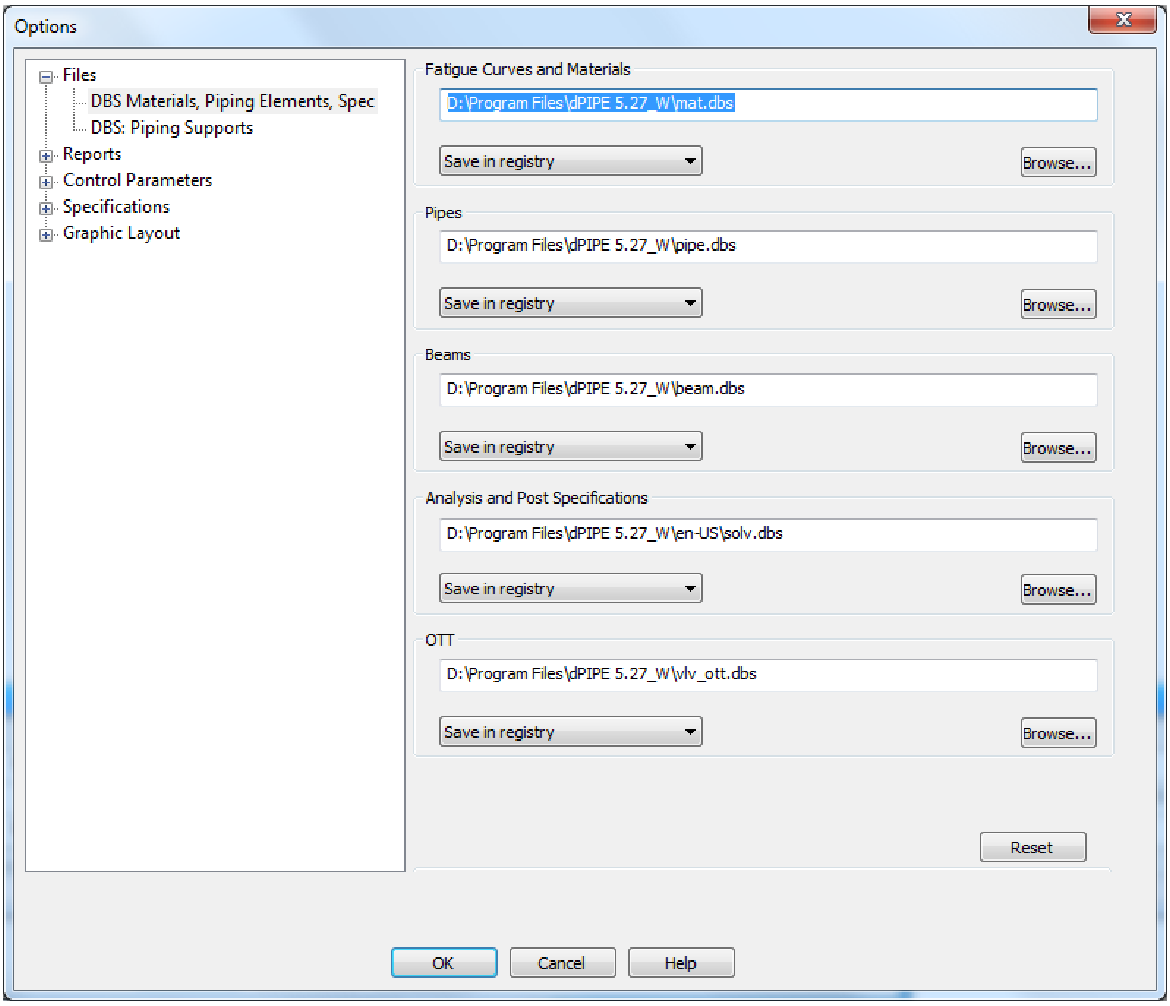
```
Пример :

DP2LCD lcd_ver = 'LICAD_10', skip = 'OFF', sh_mode = 'COLD', angle =
2.5, s_ctg = 'I'
&        'COLD' "LC6" note = "COLD"
&        'HOT' "LC4" note = "HOT"
&        'Test' "LC7" note = "TEST"
&        'Level_AB' "LC1" SF 1
&        'Level_C' "LD1 + LS11" sf = 1.6 note = "C1"
&        'Level_D' "LD1 + LS18" sf = 1.7 note = "D1"
```

Notes:

1. The LOAD parameter can refer on any already existing Load Cases (including Dynamic Cases): LC&LD or on the Load Set with type ‘SUPP’. For types that include seismic loads, it has to be combined with an appropriate static Load Case (Load Set)..
2. SF parameter is applicable only for export to LICAD-10 (LCD_VER = 'LCD_VER_10'), see also Appendix XVI

Managing the databases (DBF)

The DBF command determines the location of databases to be used by the program:



User can manage the paths to database files depending upon the options selected:

- in case of the ***"Save in the register"*** option being selected, the paths to databases will be recorded in the system register and connected automatically at the following session;
- in case of the ***"Save in a file"*** option being selected, the paths to databases will be stored in the <model name>.dp5 file and become active upon opening the existing model;
- in case of the ***"Save relative paths in a file"*** option being selected, the paths to databases will be recoded as relative links, otherwise the full paths will be written, which can cause problems when the model is transferred to another computer.

type: general one-line command

Parameter subcommands:

SPR	path to file containing the spring database;
MAT	path to file containing the materials database;
PIPE	path to file containing the pipe and bend database;
BEAM	path to file containing the beam database;
DAMP	path to file containing the damper database;
SOLV	path to file containing the database of the specifications for analysis and post-processing of the results;
VALVES	path to file containing the database of the permissible loads on valve nozzles;
SUPLOAD	path to file containing the database of the permissible loads on piping system supports;

PRE_FMT path to file with template used for input data listing file (file *.OUT);
POST_FMT path to file with template used for files with analyses results (files *.RES and *.SUP).

type:	STRING
unit:	-
default value:	blank
limitations:	The line length shall not exceed 259 characters.

Parameters PRE_FMT and PST_FMT are set in the window Options/Report/Templates

Example:

```
DBF
& spr = "D:\Program Files\dPIPE 5\sh.dbs"
& mat = "D:\Program Files\dPIPE 5\mat.dbs"
& pipe = "D:\Program Files\dPIPE 5\pipe.dbs"
& beam = "D:\Program Files\dPIPE 5\beam.dbs"
& damp = "D:\Program Files\dPIPE 5\dmp.dbs"
& solv = "D:\Program Files\dPIPE 5\solv.dbs"
& valves = "vlv_ott.dbs"
& supload = "D:\DBS\sup_lds.mdb"
```

Suppressing results by headers ($NOHEAD)

The $NOHEAD command suppresses the printing of summary tables, in accordance with headers initiated by NAME commands. Simultaneously, it maintains the segmentation of the model by headers when browsing in PIPE3DV. Consequently, summary tables are printed for the entire model, excluding sections designated as 'boundary conditions,' which are addressed by the SKIP_STR command.

End of input data (END_OF_DATA)

The **END_OF_DATA** command limits the input data in the *.dp5 file. All information following this command will be ignored by the program.

Local Commands

There are two types of dPIPE Local commands: commands that define piping spatial layout, so called "geometrical" commands, and commands that define data related to the node.

The piping system geometry is determined by means of successive chain of the geometrical commands. One such continuous chain generates a **branch** of the piping. Each branch begins by the FROM command. A straight piping segment between two nodes, in which the direction is set in the explicit form, is called a **span.** Branch can be looped. All elements located within the same span shall have the nonzero length, except cases specially stated.

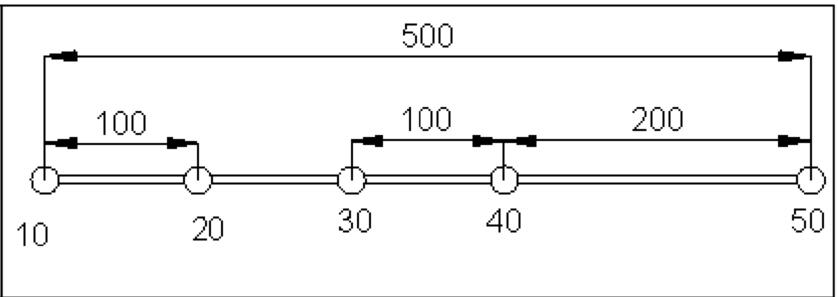
In order to determine geometry of the piping system, it is possible to use either Cartesian or spherical coordinate system. Each element of the analysis model has the direction and the length. The DC or DS parameter are used for setting the direction. The DC parameter is an array of three numbers setting the direction of the element axis in the Cartesian coordinate system. As the values of this array, the projections of the element (or span of the piping system onto the global XYZ coordinates or its directional cosines are specified (Fig. 2). In addition, by means of the DC parameters the length of span of the piping system can be determined:

$$LEN = \sqrt{DC(1)^2 + DC(2)^2 + DC(3)^2}$$

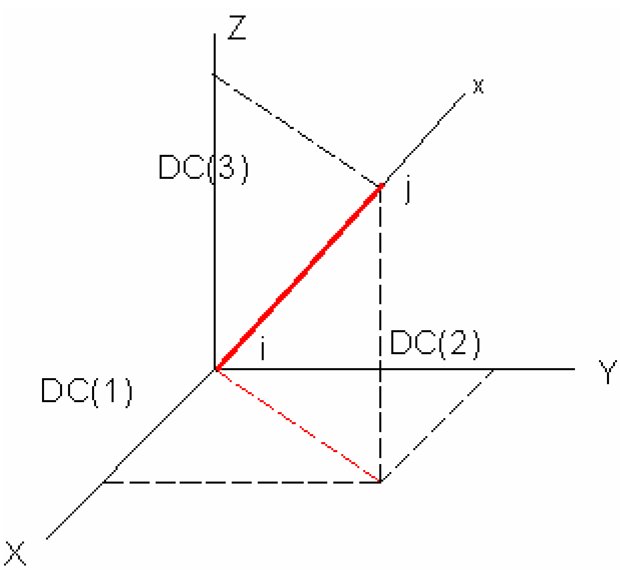
In doing so, it is permitted to avoid determining the length of one of the elements in the span; at that it will be computed as the difference between the LEN value and the sum of length of the remaining elements (the exception are elements that can have zero length: cold spring "CS" element and flexible joint element, FJ).

Example:

```
10: f dc 500 0 0
20: p 100
30: p
40: p 100
50: p 200
```

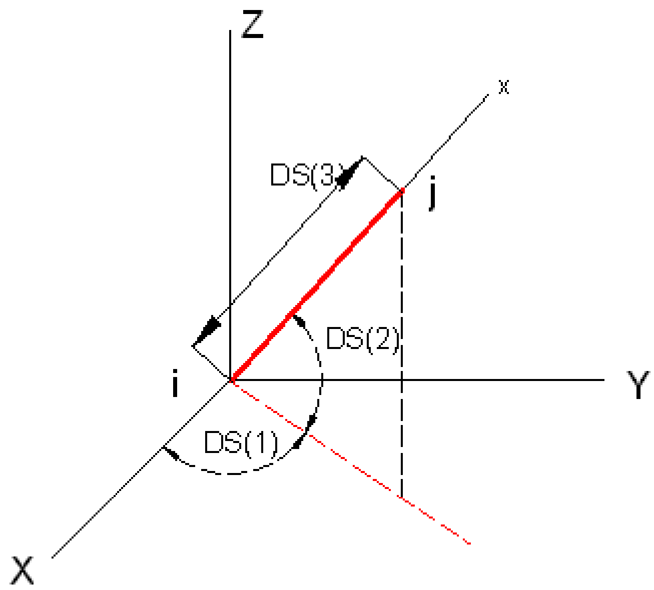


In this example the length of element 20 – 30 is automatically computed as 500 – (100 + 100 + 200) = 100



Determination of the element direction in the Cartesian coordinate system

DS being a similar parameter defines the direction of the element (piping segment) in the spherical coordinate system. In doing so, the first element of the array, DS(1), corresponds to the azimuth, i.e. to the angle in degrees between the global OX axis and the element projection onto the horizontal plane XOY. The positive direction of the angle is counted from the global OX axis to the horizontal projection of the element in the counterclockwise direction. The second element of the array, DS(2), defines the slope, i.e. the angle between the axis of the element and its projection onto the horizontal plane. The positive direction is counted from the horizontal projection of the element upwards to the element axis. The third value of the DS parameter can determine the span length.



Determination of the element direction in the spherical coordinate system

The syntax of DC and DS parameters is common for all local commands and is given below:

DC(3) - setting the direction of the current span in Cartesian coordinates

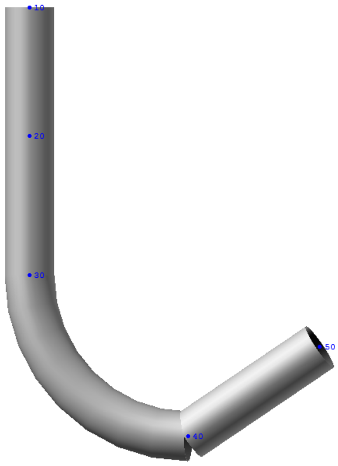
type:	REAL
unit:	either non-dimensional parameter or dimensional (in mm), if the span length is determined (see general notes)
dimension:	array of 3 elements
default value:	-
limitations:	all three elements of the array cannot simultaneously be equal to zero

DS(3) - setting the direction of the current span in spherical coordinates

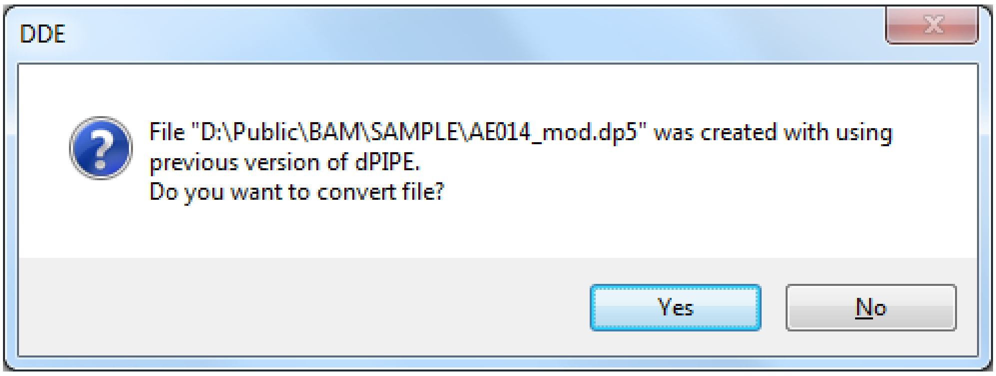
type:	REAL
unit:	the first and the second elements of the array are degrees, the last element is the length of span in mm (see general notes)
dimension:	array of 3 elements
default value:	0 0 0
limitations:	DS(3) ≥ 0

Note:

The DC and DS parameters can appear in almost any geometrical command. The exceptions are the Bend (2) element as well as any elements following the Bend (1) element. If it is required to set a new direction immediately after the bend, the User should begin a new branch:

	WRONG: 10: F dc = 1, 1000, 0, cs = 'Pipe1', lg = 'LG1' 20: P 30: B r = 300, dc = -1, 0, 0 40: P dc = -1000, 500, 0.5, cs = 'Pipe1', lg = 'LG1'
	CORRECT: 10: F dc = 1, 1000, 0, cs = 'Pipe1', lg = 'LG1' 20: P 30: B r = 300, dc = -1, 0, 0 30: F dc = -1000, 500, 0.5, cs = 'Pipe1', lg = 'LG1' 40: P

The last limitation has been introduced into the program since Version 5.20 (July 2010). In case when old models are opened, which were created in the previous versions of the program, the following warning will appear:



Similar commands are also used for one-directional restraints:

DC/DS/DIRL(3) - direction of the action of restraint in Cartesian/spherical or local coordinates.

DC is set in the form of an array of three numbers with projections of the axis of action of the restraint onto the global axes (for example, DC = 0, 1, 2), or if the line of action of the restraint coincides with one of the global axes, then the following short form is possible: DC = 'X' ('Y', 'Z')

DS defines the direction of the line of action of the restraint in spherical coordinates. It is set in the form of an array of two numbers, i.e. angles in degrees in the horizontal and vertical planes (see the similar command DS, which is used for entering the piping system geometry)

DIRL is set in the form of an array of three numbers with projections of the axis of action of the restraint onto the local axes of the piping segment (for example, DIRL = 0, 1, 1), or if the line of action of the restraint coincides with one of the local axes, then the following short form is possible: DIRL = 'A' ('H', 'N')

From (F)

Straight pipe (P)

Bend-1 (B)

Bend-2 (B)

Miter bend (MTR)

Reducer (R)

Valve (V)

Angle valve (V1, V2 command)

Expansion joint (EJ)

Axial expansion joint (EA)

Tied expansion joint (ET)

Hinge expansion joint (EH)

Gimbal expansion joint (EG)

Rigid link (RX/RP)

Flexible element (FJ)

Cold spring (CS)

Structural element (beam) (S)

Locate position (POS)

Tee (TEE)

Welding (WLD)

Weight (CW)

Force (FOR)

Anchor (ANC)

Support (SUP)

Transverse restraint (STS)

Skewed restraint (SRS)

One-way restraint (STS+, STS- commands)

Guide support (STG, STG-)

Spring hanger/spring support (SPR)

Rod hanger (ROD)

Damper (DMP)

Snubber (SNUB)

Dynamic one-way restraint (DGAP)

Dynamic transient force (DFRC)

Time history output/travel indicator (TH_OUT)

Discontinuity stress (STR_DISC)

Initialization parameters

Initialization parameters are combined with local geometrical commands describing the piping layout

Command	Description	Allowable values	Default values	Notes
CS	Initialization of the piping Section	Reference on the names predefined by PIPE or BEAM commands	-	(1), (2)
LG	Initialization of the Load Group	Reference on the names predefined by LG parameters of OPVAL command	-	(1), (2)
NAME	Identification name of the piping segment	String with length up to 32 symbols	-	(3)
CLS	Safety Class of the piping segment (applicable for ASME NB/NC Code)	cls = 1 or cls = 2	2	(2)
SBP	Small Bore Pipes marker	sbp = 'yes', sbp = 'no'	'no'	(3)
SCTG	Seismic Category (applicable for PNAE and RD Codes)	sctg = 'I' or 'II' or 'III'	'N/A'	(2)

Notes:

(1) The command has to be located at the beginning of the branch (FROM command)

(2) Initializes the current piping segment up to the end of the branch, or until the next command

(3) Initializes the current piping segment up to the next command. The command allows to assign a logical names to piping sections for subsequent partial display of the model and printing of the main calculation results for the selected sections at the end of the file with a printout of the

results (*.res). The effect of this command on the entire model can be suppressed using the command $NOHEAD = 'YES'

From (F)

type: local geometrical command

Function: determines the first point of the Branch

Parameters:

DC or DS: see "Local commands" section for the description of parameters

CS initialize the cross-section type

type:	TEXT
unit:	-
default value:	mandatory parameter if the current cross-section is not initialized. Otherwise it takes the value of the current cross-section.
limitations:	the cross-section shall be preliminarily described by the PIPE command or BEAM command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature defines segment of the piping as "small bore pipe" (initialization parameter)

type:	TEXT
unit:	-
default value	'NO'
limitations:	'YES', 'NO'

```
Example:

   1000: F DC 0 0 1 CS '108x10' LG 'Line1' name = "Line 1"
```

or:

```
1000: F 0 0 1 '108x10' 'Line1' "Line 1"
```

Straight pipe (P)

type: local geometrical command

Function: determines the "straight pipe" element

Parameters:

LEN length of element

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or DS: see "Local commands" section for the description of parameters

CS initialize the cross-section type

type:	TEXT
unit:	-
default value:	mandatory parameter if the current cross-section is not initialized. Otherwise it takes the value of the current cross-section.
limitations:	the cross-section shall be preliminarily described by the PIPE command

or:

XS change of the cross-section type only for the current element without changing the current value for the whole remaining segment

type:	TEXT
unit:	-
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

NAME - identification name of the piping segment

type: STRING
unit: -
default value: blank line or current identification name
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

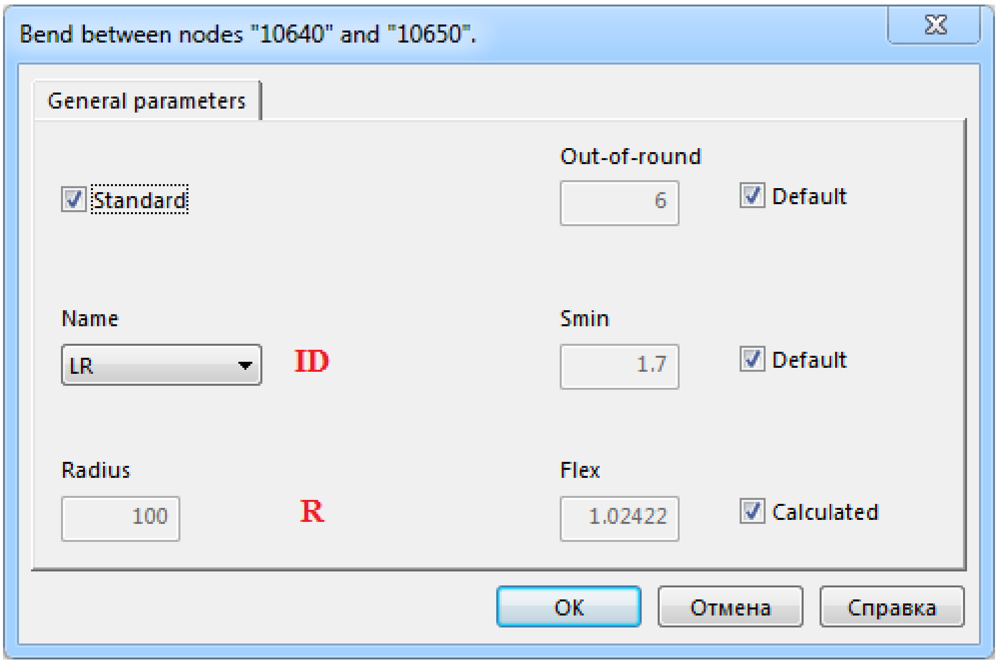
Example:

```
2000: P 2300 DC 1 CS '108x10' LG 'Line1'
```

or:

```
2000: P 2300 DC 1 0 0 CS '108x10' LG 'Line1'
```

Bend-1 (B)



type: local geometrical command

Function: determines the element for modeling the curved pipe (bend or elbow)

Parameters:

ID[1)] - identification name of the bend determined by the BEND subcommand of the PIPE general command

type: TEXT
unit: -

default value: -
limitations: the name shall be preliminarily described by the BEND subcommand of the PIPE command

or

R[2)] radius of bend

type: REAL
unit: mm
default value: -
limitations: R > 0; the bend shall adjust to the geometry of the segment (see Example of setting the geometry in the "Local commands" section)

DC or DS: see "Local commands" section for the description of parameters. The parameters determine the direction of span being next to the bend. The direction of the BEND element shall differ from the current direction (see Example of setting the geometry)

XS[1)] set the type of the cross-section used for BEND if it differs from the cross-section of the adjacent pipe

type: TEXT
unit: -
default value: current cross-section type
limitations: the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type: TEXT
unit: -
default value: current load group
limitations: the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

OVAL out-of-roundness (ovality) of the cross-section (this parameter redefines the similar value that is set in the BEND subcommand of the PIPE general command)

type: REAL
unit: %
default value: 0 or value defined by the OVAL parameter in the BEND subcommand of the PIPE general command
limitations: 0 ≤ OVAL ≤ 100

SMIN minimum thickness of the bend wall (this parameter redefines the similar value that is set in the BEND subcommand of the PIPE general command)

type: REAL
unit: mm

default value:	value defined by the SMIN parameter in the BEND subcommand of the PIPE general command
limitations:	0 < SMIN ≤ T

FLEX flexibility factor of curvilinear pipe, which is used in generating the stiffness matrix of the element

type:	REAL
unit:	-
default value:	it is computed by the program automatically depending upon the Analysis codes being used.
limitations:	FLEX ≥ 1.

FLNG number of flanges at the ends of the bend (available only for CODE = 'ASME_B311')

type:	INTEGER
unit:	-
default value:	0
limitations:	0, 1, 2

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

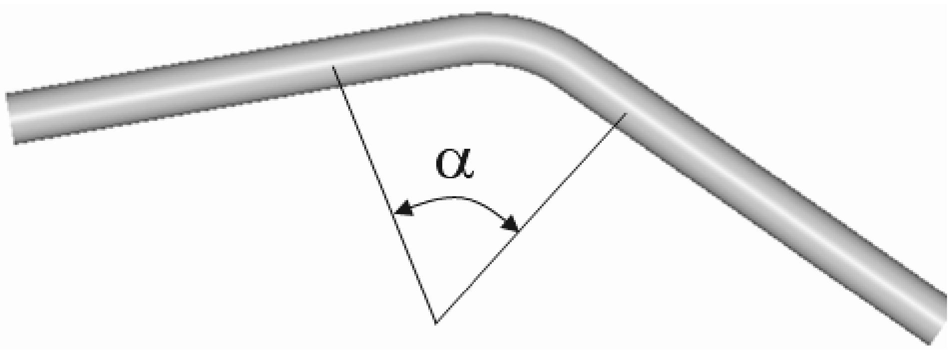
SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

When using ASME/EN CODES, an additional tab with stress indexes appears in the dialog

Note:

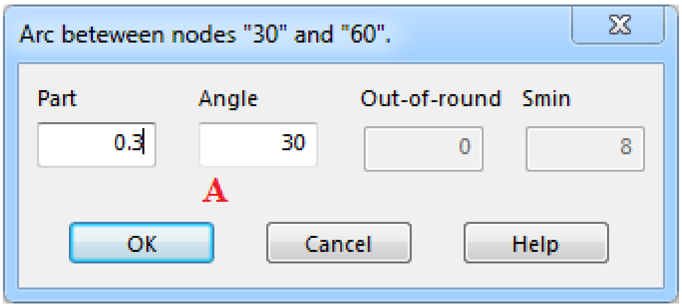
1) *In setting the bend by means of the ID parameter, it is not allowed to use the XS parameter*
2) *designation "R" is mandatory*
3) *The full angle of bend shall be within the following range: BEND_ANG < a < 180° - BEND_ANG:*

.



```
Example:

 3000: b r 1000 dc 1 0 0 xs '108x12' OVAL 3 SMIN 10 FLEX 1.

 or:

 1000: b ID 'LONG'
```

Bend-2 (B)



type: local geometrical command

Function: allows splitting one bend into several elements. It is used only in combination with the BEND-1 command. The BEND-2 command shall precede the BEND-1 command.

Parameters:

A[1)] parameter of the division of a curved pipe into parts. At A > 1 it defines the angle (in degrees) from the beginning of the bend to the current node. At 0 < A ≤ 1 it defines a part of the full bending angle.

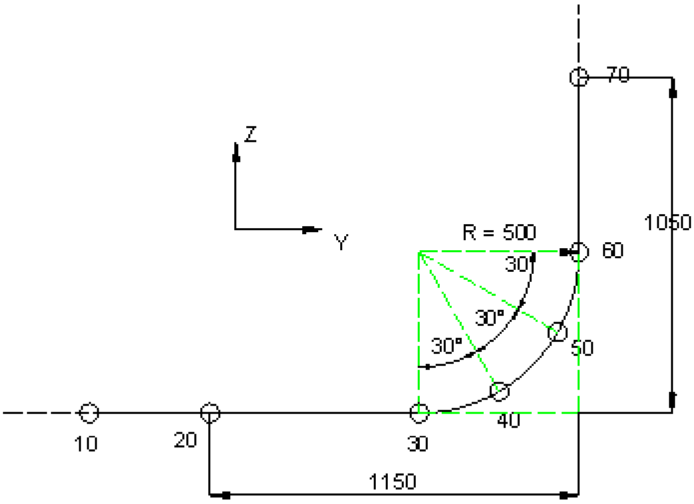
type:	REAL
unit:	degrees or non-dimensional parameter
default value:	-
limitations:	the angle cannot be more than the full angle of bend

Note:

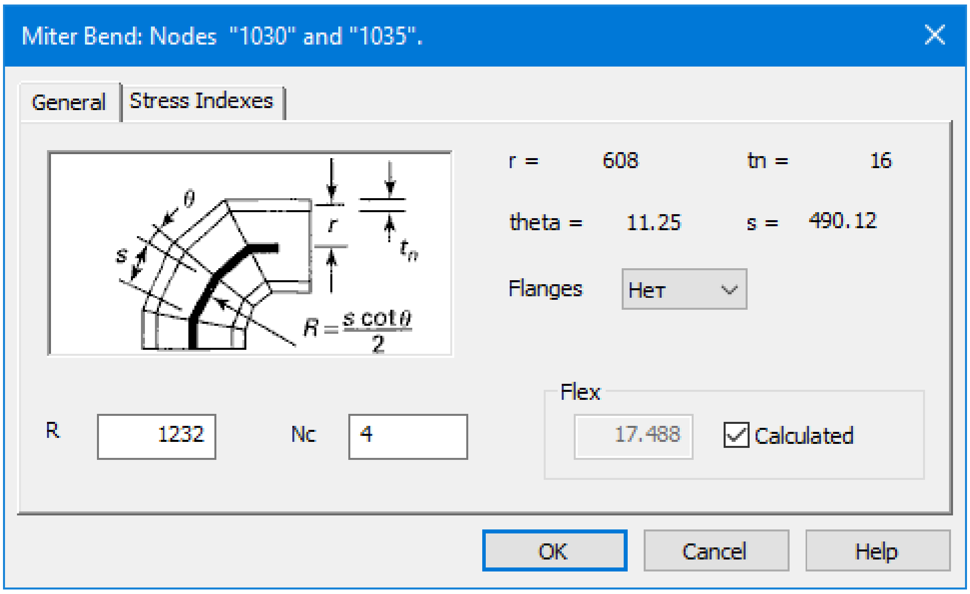
1) *designation "A" is mandatory*

```
Example:
```

```
10: f 0 1 0 '108x5' 'Line1'
20: p 400
30: p 1150
40: b a 30
50: b a 60
60: b r 500 dc 0 0 1
70: p 1050

or:

10: f 0 1 0 '108x5' 'Line1'
20: p 400
30: p 1150
40: b a .33333
50: b a .66666
60: b r 500 dc 0 0 1
70: p 1050
```



Miter bend (MTR)



type: local geometrical command

Function: determines the "miter bend" element 1)

Parameters:

R equivalent radius of the miter bend

type: REAL
unit: mm
default value: -
limitations: R > 0; the element shall adjust to the geometry of the segment (see Example of setting the geometry in the "Local commands" section)

Nc number of miter cuts (if NC> 1 - closely spaced miter bend, NC = 1 for widely spaced miter bend)

type:	INTEGER
unit:	-
default value:	-
limitations:	>0

DC or DS: see "Local commands" section for the description of parameters. The parameters determine the direction of span being next to the bend. The direction of the BEND element shall differ from the current direction (see Example of setting the geometry)

XS setting the cross-section type for the bend, if it differs from the cross-section of the adjacent pipe

type:	TEXT
unit:	-
default value:	current cross-section type
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	current load group
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

FLNG number of flanges at the ends of the bend (available only for CODE = 'ASME_B311')

type:	INTEGER
unit:	-
default value:	0
limitations:	0, 1, 2

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

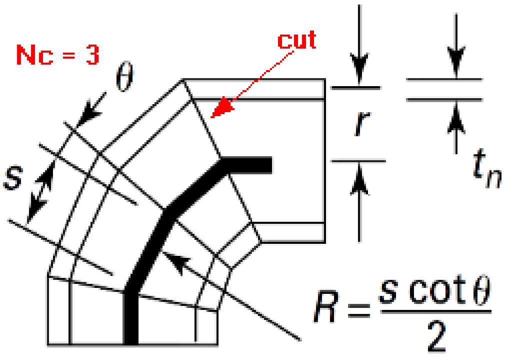
SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

When using ASME/EN CODES, an additional tab with stress indexes appears in the dialog

Note:

1) *The "Miter bend" element is a variation of the "Bend" element and is mainly used within the framework of analysis according to international codes (ASME, EN) for specification of the flexibility factor and stress indexes. When the element is set within the analysis according to Russian Codes (PNAE or RD) the program will interpret it as a conventional bend with radius R and specified characteristics of the cross-section.*

   To define the element one should set an equivalent radius R and number of cuts Nc:



   *If Nc = 1 the radius (R) should not be set: the program will determine it based on the relation: Re = 0.5 * r * (1 + ctg (θ)), where: r is the mean radius of the pipe; θ is half the total angle of the bend. If* Nc > 1, then $\theta = \alpha/(2 N_c)$, and $s = 2 R_e \cdot tg(\theta)$

   Check for Errors:

   1. If Nc > 1, then s < r*(1+tg(θ))
   2. $B = s(1 - r_0/R_e) \geq 5 t$ (r0 - outside pipe radius, t - wall thickness)
   3. For ASME B31.1: B > 6t

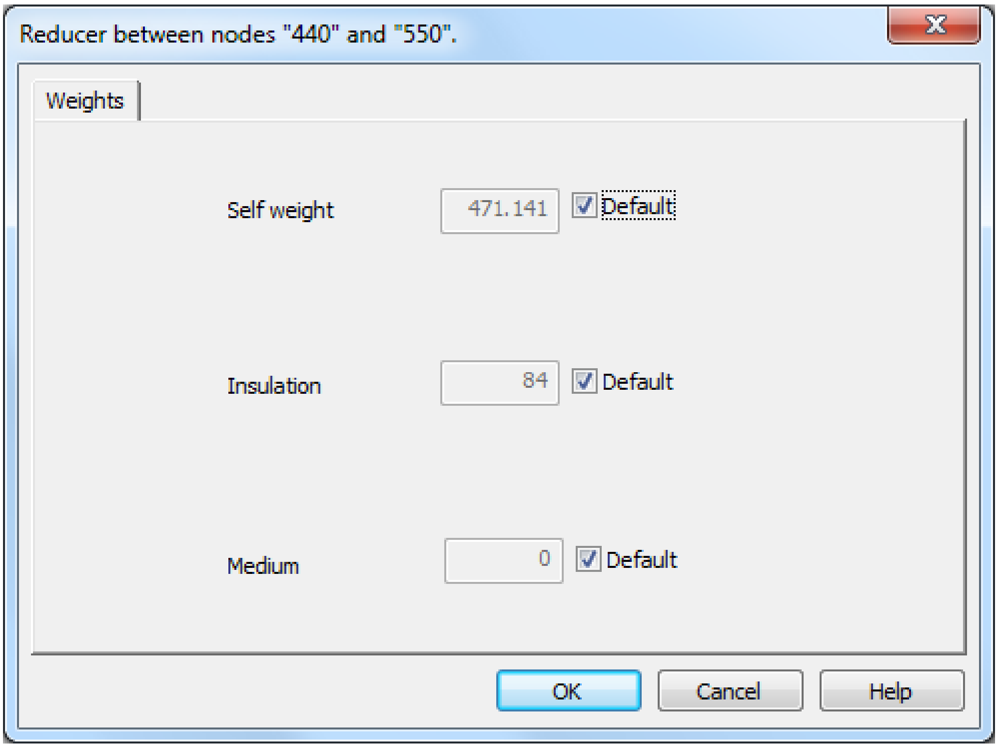
   Program will issue the warnings, if:

   1) the total miter bend's angle α is less than 15°;
   2) angle θ < 15° (for Russian PNAE and RD Codes);
   3) angle θ < 22.5° (for the rest of Codes (except PNAE and RD)

Example:

```
30: MTR r = 862.089, nc = 2, ds = 45, 0, 5000, xs = 'Pipe2', name = "LAB10"
```

Reducer (R)



type: local geometrical command

Function: determines the Reducer element between pipes of various diameter. The stiffness matrix of the element is formed as for the "straight pipe" element with averaged characteristics of the cross-section from the adjacent pipes.

Parameters:

LEN length of element

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

ANGLE cone angle (it is used for computation of the stress indexes in analysis according to international codes)

type:	REAL
unit:	degrees
default value:	see Note 1
limitations:	≥ 0

MAT - reference identification name of the material (see MAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	the name should coincide with the materials names determined earlier.

DC or **DS:** see "Local commands" section for the description of parameters

CS initialize the type of cross-section being next to the Reducer element

type:	TEXT
unit:	-
default value:	mandatory parameter. Must define different cross-section in respect to adjacent pipe
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the current load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

W(3) weight characteristics of the element: see Appendix IV.

type:	REAL
unit:	Newton
default value:	all three components of the array are determined as average values from the characteristics of the adjacent pipes
dimension	array of 3 elements
limitations:	see Appendix IV.

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

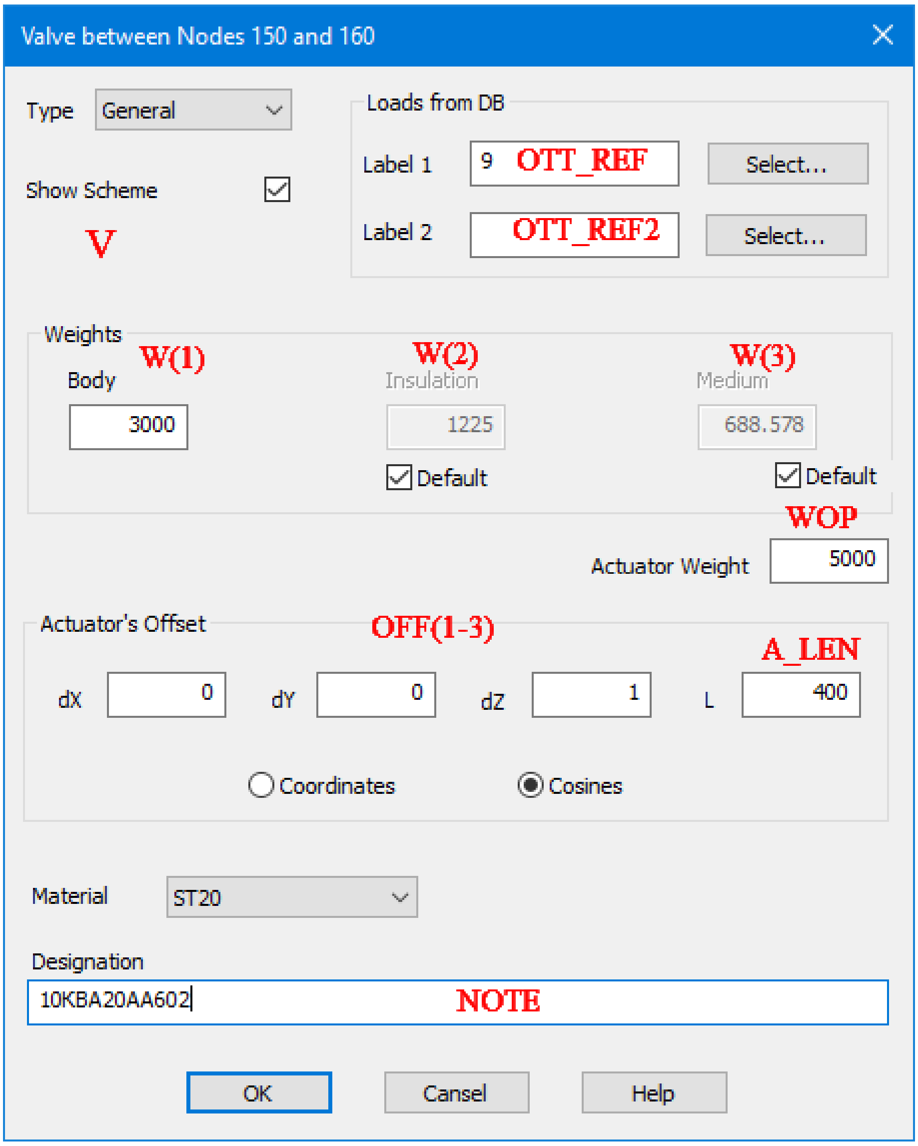
When using ASME/EN CODES, an additional tab with stress indexes appears in the dialog

Note:

1) *By default **ANGLE** parameter is not defined. In this case a maximal values are used for stress indexes/stress intensification factors:* $B_1 = 1, i = 2$

```
Example:

2000: r 400 CS '108x10' w 10
```

Valve (V)



The input window provides separate setting of the valve weights: e.g. weight of the valve body and the weight of the actuator. With the flags marked by default, the weight of insulation and piping content will be added to the element depending upon the characteristics of the adjacent pipe and the current load group. In doing so, the weight of insulation is calculated with the coefficient 1.75 of the pipe's weight, and the weight of the piping content is added to be equal to the weight of the working fluid in the straight pipe. The user can redefine these parameters having removed the flag marks taken by default. If a positive number is entered, then it will be taken by the program as weight (of insulation or piping content) in Newtons. In case of negative values being entered the program will take them as multipliers to the weight of insulation or working fluid. If the dead weight is set equal to zero,then the weight of insulation and working fluid will not be added to the element by default.

When setting the weight of actuator, it is mandatory to enter information about the offset of the center of mass of the actuator with respect to the central point of the element. The offset can be set either in relative coordinates ("Coordinates" flag) or by means of the direction cosines and the length of driver.

The "Loads from DB" field serves for selecting permissible loads in accordance with NP-068-05 "Valves for nuclear plants. General technical requirements".

The "Designation" field is used for setting the identification name of valves. This information is output for printing in listings of the input data and the results of analysis.

type: local geometrical command

Function: determines the element for the modeling of valves and similar pipe accessories. The stiffness matrix for an element is formed the same as for the "straight pipe" element of the current cross-section with the wall thickness being multiplied by the V_STF factor.

Parameters:

LEN length of element

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS set the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	mandatory parameter if the current cross-section is not initialized. Otherwise it takes the value of the current cross-section.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

W(3) weight characteristics of the element[1)]

type:	REAL
unit:	Newton
dimension:	array of three numbers
default value:	see Appendix IV.
limitations:	-

MAT - reference identification name of the material (see MAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	the name should coincide with the materials names determined earlier.

WOP weight of drive

type:	REAL
unit:	Newton
default value:	0
limitations:	WOP ≥ 0

OFF(3) relative coordinates of the center of mass of the drive or direction cosines (X, Y, Z)[2)]

type:	REAL
unit:	mm
dimension:	array of three numbers
default value:	0, 0, 0
limitations:	-

A_LEN length of drive[3)]

type:	REAL
unit:	mm
default value:	-
limitations:	A_LEN > 0

NOTE Note/Comment - identification name of the valve

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

OTT_REF label identifying a record in the database with permissible loads on the valve nozzle from the side of first element's node (see Appendix X)

type:	TEXT
unit:	-
default value:	-
limitations:	to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

OTT_REF2 label identifying a record in the database with permissible loads on the valve nozzle from the side of second element's node (see Appendix X)

type:	TEXT
unit:	-
default value:	-
limitations:	to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

Note:

1) *See Appendix IV .*
2) *The coordinates are set in the global system combined with the center of mass of the valve housing. When the A_LEN parameter is present, the values are interpreted as direction cosines.*
3) *The parameter is used only for setting the coordinates of the center of mass of the actuator by means of direction cosines.*

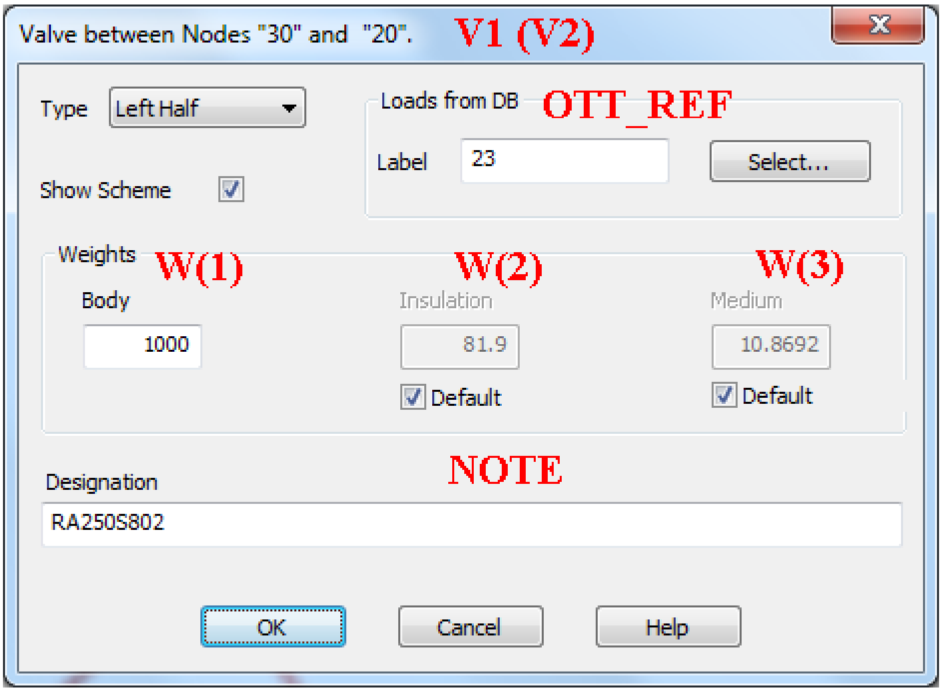
Example:

```
30: V len = 800, w = 2000,,, wop = 1000, off = 0, 0, 1, a_len = 400, note = "RA250S
```

or:

```
30: V len = 800, w = 2000,,, wop = 1000, off = 0, 0, 400, note = "RA250S802", ott_r
```

"Half-valve" (V1, V2 commands)



"Half-valve" is used to model different types of piping fittings. It is possible to set the left-side and right-side parts of the valve.

The input window provides setting of the valve weights: with the flags marked by default, the weight of insulation and piping content will be added to the element depending upon the characteristics of the adjacent pipe and the current load group. In doing so, the weight of insulation is calculated with the coefficient 1.75 of the pipe's weight, and the weight of the piping content is added to be equal to the weight of the working fluid in the straight pipe. The user can redefine these parameters having removed the flag marks taken by default. If a positive number is entered, then it will be taken by the program as weight (of insulation or piping content) in Newtons. In case of negative values being entered the program will take them as multipliers to the weight of insulation or working fluid. If the dead weight is set equal to zero,then the weight of insulation and working fluid will not be added to the element by default.

The "Loads from DB" field serves for selecting permissible loads in accordance with NP-068-05 "Valves for nuclear plants. General technical requirements".

The "Designation" field is used for setting the identification name of valves. This information is output for printing in listings of the input data and the results of analysis.

type: local geometrical command

Function: commands for the modeling different types of piping fittings, for example, angle valve. When valves are set by means of V1, V2 commands, the weight of valve shall be divided equally between these two elements.

Parameters:

LEN length of element

type:	REAL
unit:	mm

default value: 0
limitations: LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	mandatory parameter if the current cross-section is not initialized. Otherwise it takes the value of the current cross-section.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group .
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

W(3) weight characteristics of the element[1)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

MAT - reference identification name of the material (see MAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	the name should coincide with the materials names determined earlier.

NOTE Note/Comment/- identification name of the valve

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

NAME - identification name of the piping segment

type:	STRING
unit:	-

default value:	blank line or current identification name
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

OTT_REF label identifying a record in the database with permissible loads on the valve nozzles (see Appendix X)

type:	TEXT
unit:	-
default value:	-
limitations:	to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

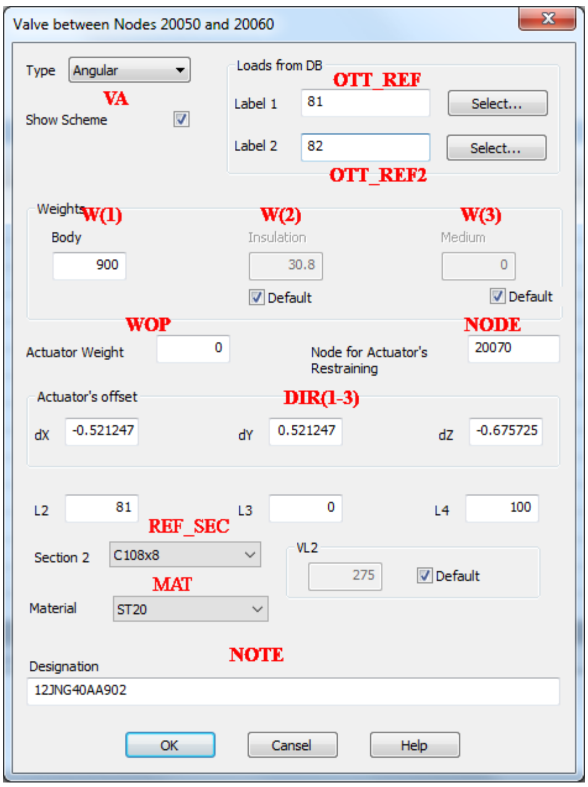
Note:

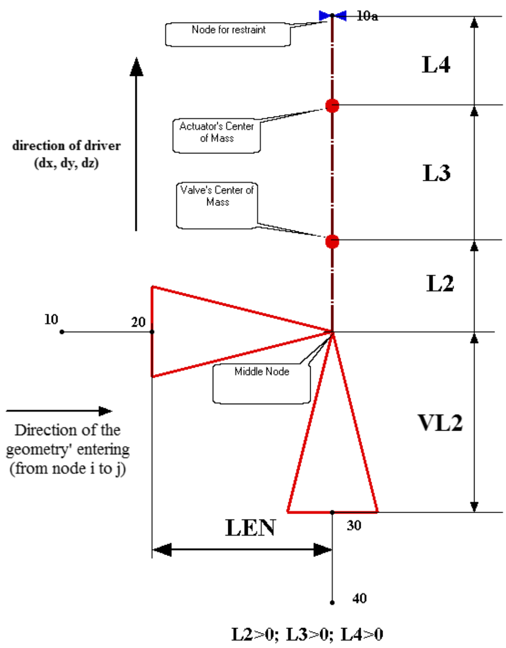
1) See Appendix IV .

Example:

2000: V1 1000 W 230 NOTE "Valve 30RAS201" DC 1 0 0
2000: V2 1000 W 230 NOTE "Valve 30RAS201" DC 0 0 -1

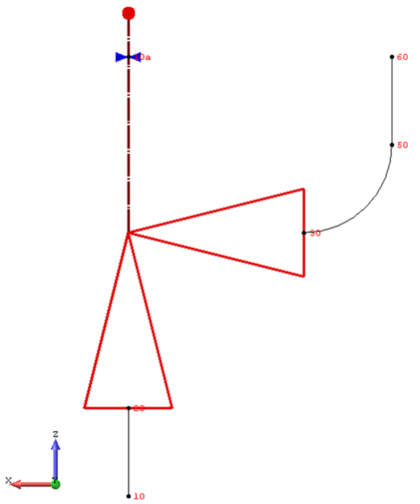
Angle valve (VA)





Calculation model of the angle valve

The "angle valve" changes the direction of the current branch. That is why the string where this element is present should have commands for change of direction. Caution: If the angle valve is adjacent to the Bend element his length should not be corrected on the projection of bend radius like it’s doing for a straight pipe:



Layout

	Node	Element	L/R	dX	dY	dZ	Add. Data
1	10	From		0	0	600	
2	20		0				
3	30	Valve	400	-600	0	0	Restraint
4	50	Bend	200	0	0	1	
5	60		400				

Element’s dialog allows to set points for the Valve’s center of mass, center of mass of the actuator and the node, at which the drive restraining is possible ("Node for Actuator’s restraining" field). It is supposed that these points lie on the same line, which direction is determined by means of direction cosines in the "Actuator's offset" field. The distance to the points specified is measured from the central point of the element according to the dimension chaining principle. Below the examples are given for determination of L2 - L4 parameters depending upon location of the points being determined.

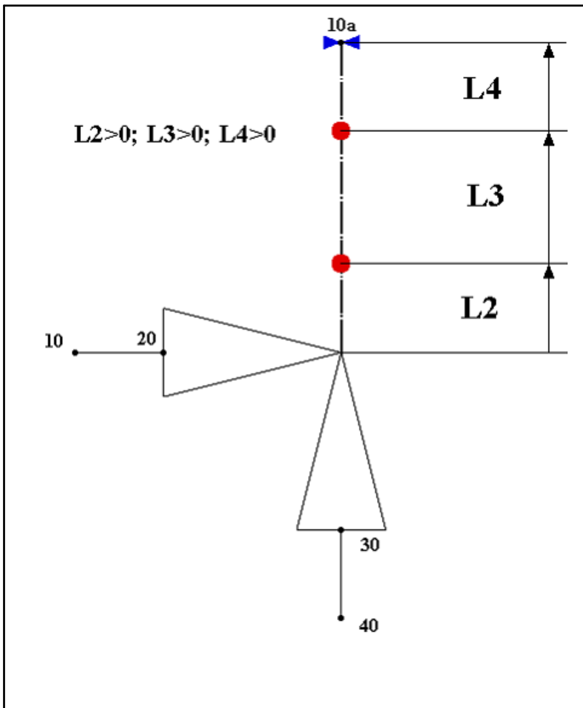


Diagram 1

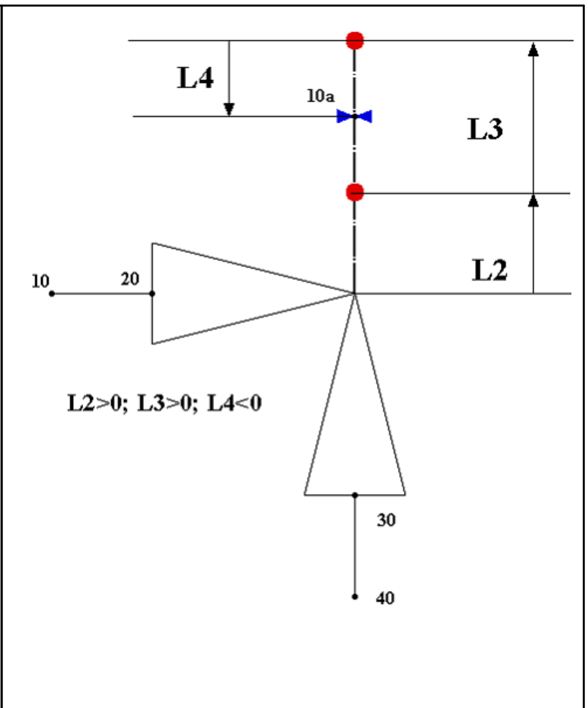


Diagram 2

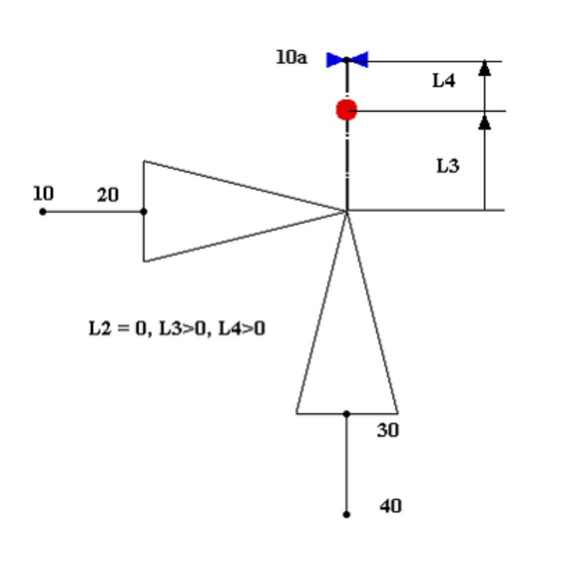
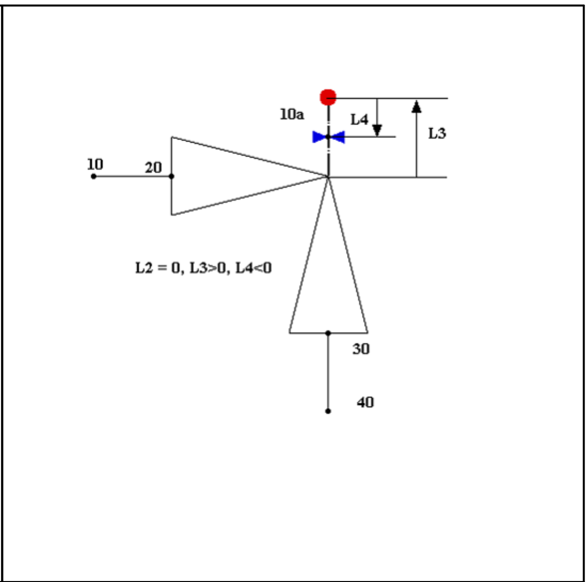
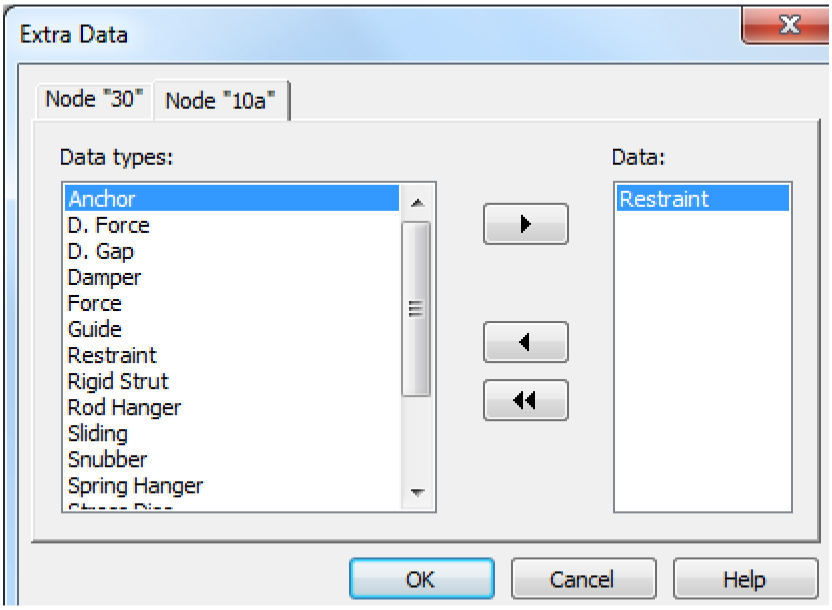
	

Diagram 3	**Diagram 4**

To set the supports in node 10a, it is necessary to call the "additional data" dialog (in the same line where the element is described) and transfer to the tab with the name of this node:



The input window provides separate setting of the valve weights: e.g. weight of the valve body and the weight of the actuator. With the flags marked by default, the weight of insulation and piping content will be added to the element depending upon the characteristics of the adjacent pipe and the current load group. In doing so, the weight of insulation is calculated with the coefficient 1.75 of the pipe's weight, and the weight of the piping content is added to be equal to the weight of the working fluid in the straight pipe. The user can redefine these parameters having removed the flag marks taken by default. If a positive number is entered, then it will be taken by the program as weight (of insulation or piping content) in Newtons. In case of negative values being entered the program will take them as multipliers to the weight of insulation or working fluid. If the dead weight is set equal to zero,then the weight of insulation and working fluid will not be added to the element by default.

The "Loads from DB" field serves for selecting permissible loads in accordance with NP-068-05 "Pipeline valves for nuclear plants. General technical requirements".

The "Designation" field is used for setting the identification name of valves. This information is output for printing in listings of the input data and the results of analysis.

type: local geometrical command

Function: determines the element for the simulation of valves and fittings (slides, gates, valves, etc.). The stiffness matrix for an element is formed the same as for the "straight pipe" element of the current cross-section with the wall thickness being multiplied by the V_STF factor.

Parameters:

LEN length of element (see the analytical model of the valve)

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS**: see "Local commands" section for the description of parameters. **For the angle valve the entry of these parameters is mandatory!**

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	mandatory parameter if the current cross-section is not initialized. Otherwise it takes the value of the current cross-section.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command.

W(3) weight characteristics of the element[1)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

MAT - reference identification name of the material (see MAT command)

type:	TEXT
unit:	-
default value:	-

limitations: the name should coincide with the materials names determined earlier.

WOP weight of drive

type: REAL
unit: Newton
default value: 0
limitations: WOP ≥ 0

DIR(3) direction cosines for determination of the drive direction

type: REAL
unit: mm
dimension: array of three numbers
default value: 0, 0, 0
limitations: -

LEN2, LEN3, LEN4 distances determining the center of mass of the housing, center of mass of the actuator and the drive restraining point (see Fig. with the analytical model).

type: REAL
unit: mm
default value: -
limitations: -

VL2 length of the second part of the valve (see Fig. with the analytical model)

type: REAL
units: mm
default value: LEN
limitations: -

NODE[2)] name of the node for detachment of the drive

type: TEXT
unit: -
default value: -
limitations: see limitations for node labels

NOTE Note/Comment/- identification name of the valve

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

NAME - identification name of the piping segment

type: STRING
unit: -

default value: blank line or current identification name
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

OTT_REF label identifying a record in the database with permissible loads on the valve nozzle from the side of first element's node (see Appendix X)

type: TEXT
unit: -
default value: -
limitations: to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

OTT_REF2 label identifying a record in the database with permissible loads on the valve nozzle from the side of second element's node (see Appendix X)

type: TEXT
unit: -
default value: -
limitations: to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

REF_SEC Reference on the pipe cross-section for the second half of the valve

type: TEXT
unit: -
default value: current cross-section of the element
limitations: the cross-section shall be preliminarily described by the PIPE command

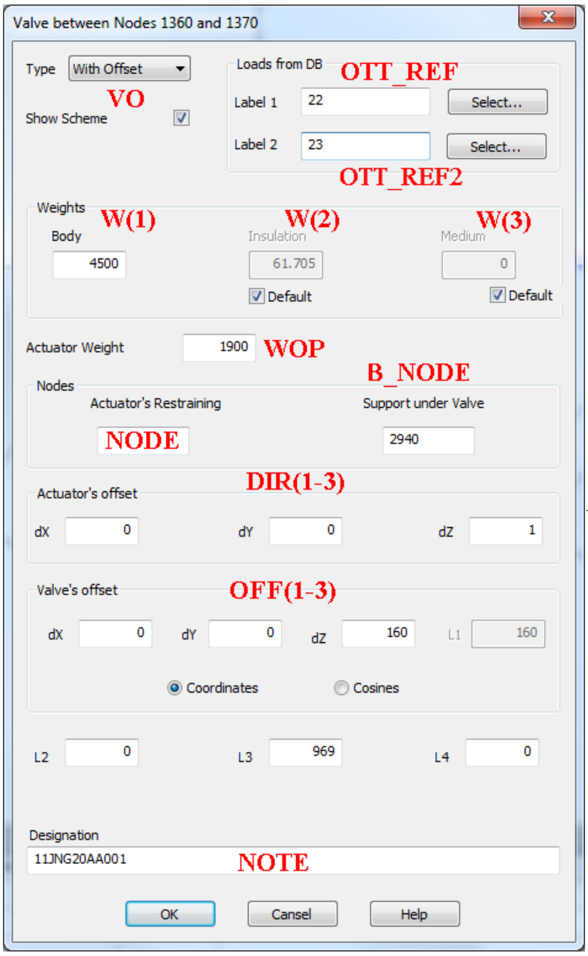
Note:

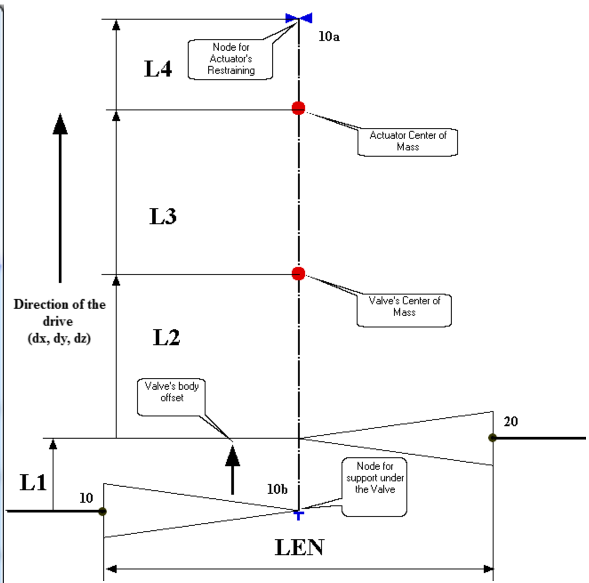
1) See Appendix IV .
2) The internal nodes of the valve can be used only for referencing a support thereto. It cannot be used as a reference value (for example, in the POS command or in operations with copying the model segments)

Example:

```
30: VA len = 400, dc = 0, 0, -600, xs = 'Pipe2', w = 2000,,, wop = 1000, dir = 0, (
```

Valve with offset (VO)





Valve with Offset model

Element's dialog allows to set points for the Valve's center of mass, center of mass of the actuator and the node, at which the drive restraining is possible ("Node for Actuator's restraining" field). It is supposed that these points lie on the same line, which direction is determined by means of direction cosines in the "Actuator's offset" field. The distance to these points is measured from the central point of the element according to the dimension chaining principle. Below the examples are given for determination of L1 - L4 parameters depending upon location of the points being determined.

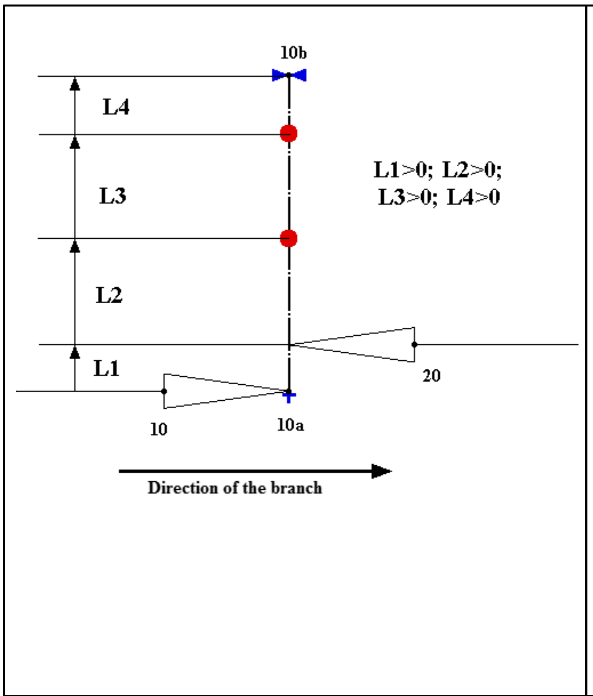


Diagram 1

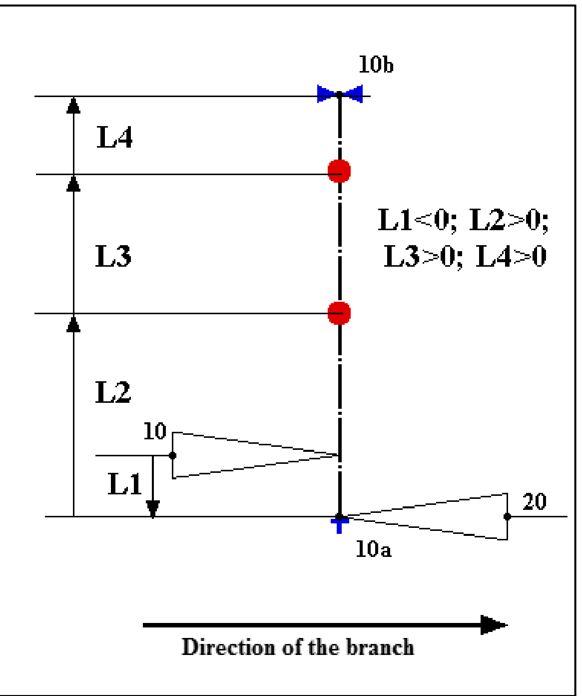


Diagram 2

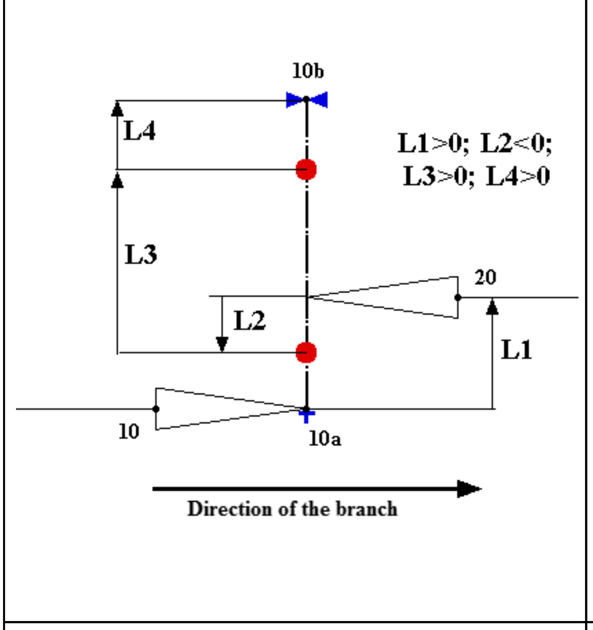


Diagram 3

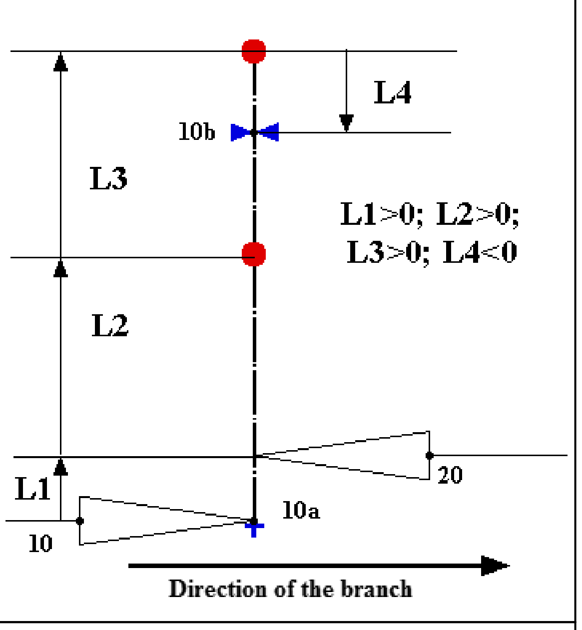
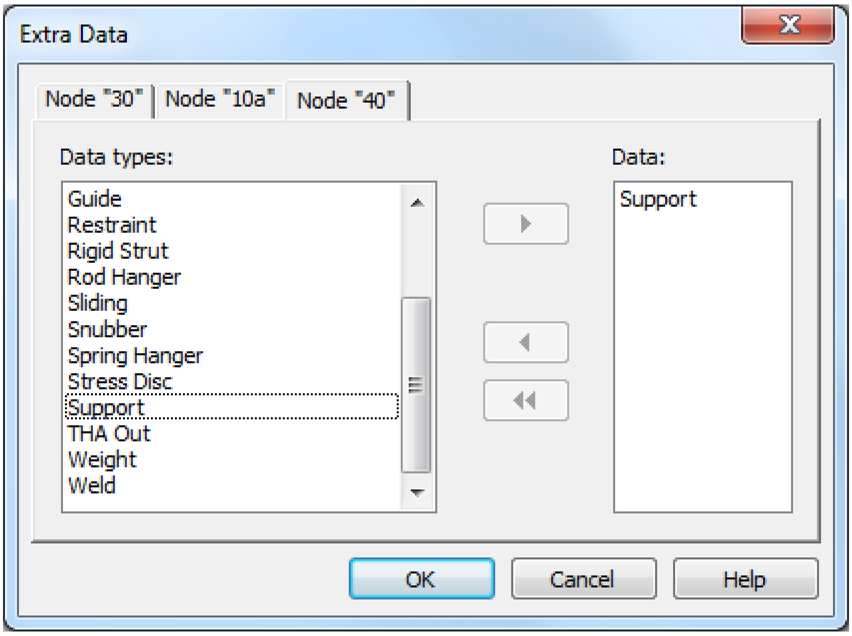


Diagram 4

To set the supports in nodes 10a and 10b, it is necessary to call the "additional data" dialog (in the same line, in which the element is described) and и transfer to the tab with the name of this node:



The input window provides separate setting of the valve weights: e.g. weight of the valve body and the weight of the actuator. With the flags marked by default, the weight of insulation and piping content will be added to the element depending upon the characteristics of the adjacent pipe and the current load group. In doing so, the weight of insulation is calculated with the coefficient 1.75 of the pipe's weight, and the weight of the piping content is added to be equal to the weight of the working fluid in the straight pipe. The user can redefine these parameters having removed the flag marks taken by default. If a positive number is entered, then it will be taken by the program as weight (of insulation or piping content) in Newtons. In case of negative values being entered the program will take them as multipliers to the weight of insulation or working fluid. If the dead weight is set equal to zero,then the weight of insulation and working fluid will not be added to the element by default.

The "Loads from DB" field serves for selecting permissible loads in accordance with NP-068-05 "Pipeline valves for nuclear plants. General technical requirements".

The "Designation" field is used for setting the identification name of valves. This information is output for printing in listings of the input data and the results of analysis.

type: local geometrical command

Function: determines the element for the simulation of valves and fittings (slides, gates, valves, etc.). The stiffness matrix for an element is formed the same as for the "straight pipe" element of the current cross-section with the wall thickness being multiplied by the V_STF factor.

Parameters:

LEN length of element (see analytical model of the valve)

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS**: see "Local commands" section for the description of parameters.

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	mandatory parameter if the current cross-section is not initialized. Otherwise it takes the value of the current cross-section.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command.

W(3) weight characteristics of the element[1)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

MAT - reference identification name of the material (see MAT command)

type:	TEXT
unit:	-
default value:	-
limitations:	the name should coincide with the materials names determined earlier.

WOP weight of drive

type:	REAL
unit:	Newton
default value:	0
limitations:	WOP ≥ 0

DIR(3) direction cosines for determination of the drive direction

type:	REAL
unit:	mm
dimension	array of three numbers
default value:	0, 0, 0
limitations:	-

OFF(3) direction cosines or relative coordinates for determination of the housing displacement direction 2)

type:	REAL
unit:	mm
dimension	array of three numbers
default value:	0, 0, 0
limitations:	-

LEN1, LEN2, LEN3, LEN4 distances determining the center of mass of the housing, center of mass of the drive and the drive detachment point (see Fig. with the analytical model).

type:	REAL
unit:	mm
default value:	-
limitations:	-

NODE3) name of the node for detachment of the drive

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for node marks

B_NODE3) name of the node for the support under the valve housing

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for node marks

NOTE Note/Comment/- identification name of the valve

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

OTT_REF label identifying a record in the database with permissible loads on the valve nozzle from the side of first element's node (see Appendix X)

type: TEXT
unit: -
default value: -
limitations: to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

OTT_REF1 label identifying a record in the database with permissible loads on the valve nozzle from the side of second element's node (see Appendix X)

type: TEXT
unit: -
default value: -
limitations: to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

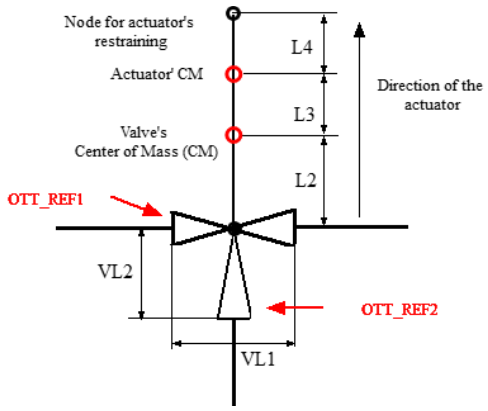
Note:

1) See Appendix IV .
2) If theLEN1 parameter is set in the command, then the values of OFF(1-3) will be interpreted by the program as a set of direction cosines. Otherwise OFF(1-3) will be considered as relative coordinates.
3) The internal nodes of the valve can be used only for referencing a support thereto. They cannot be used as a reference value (for example, in the POS command or in operations with copying the model segments)

Example:

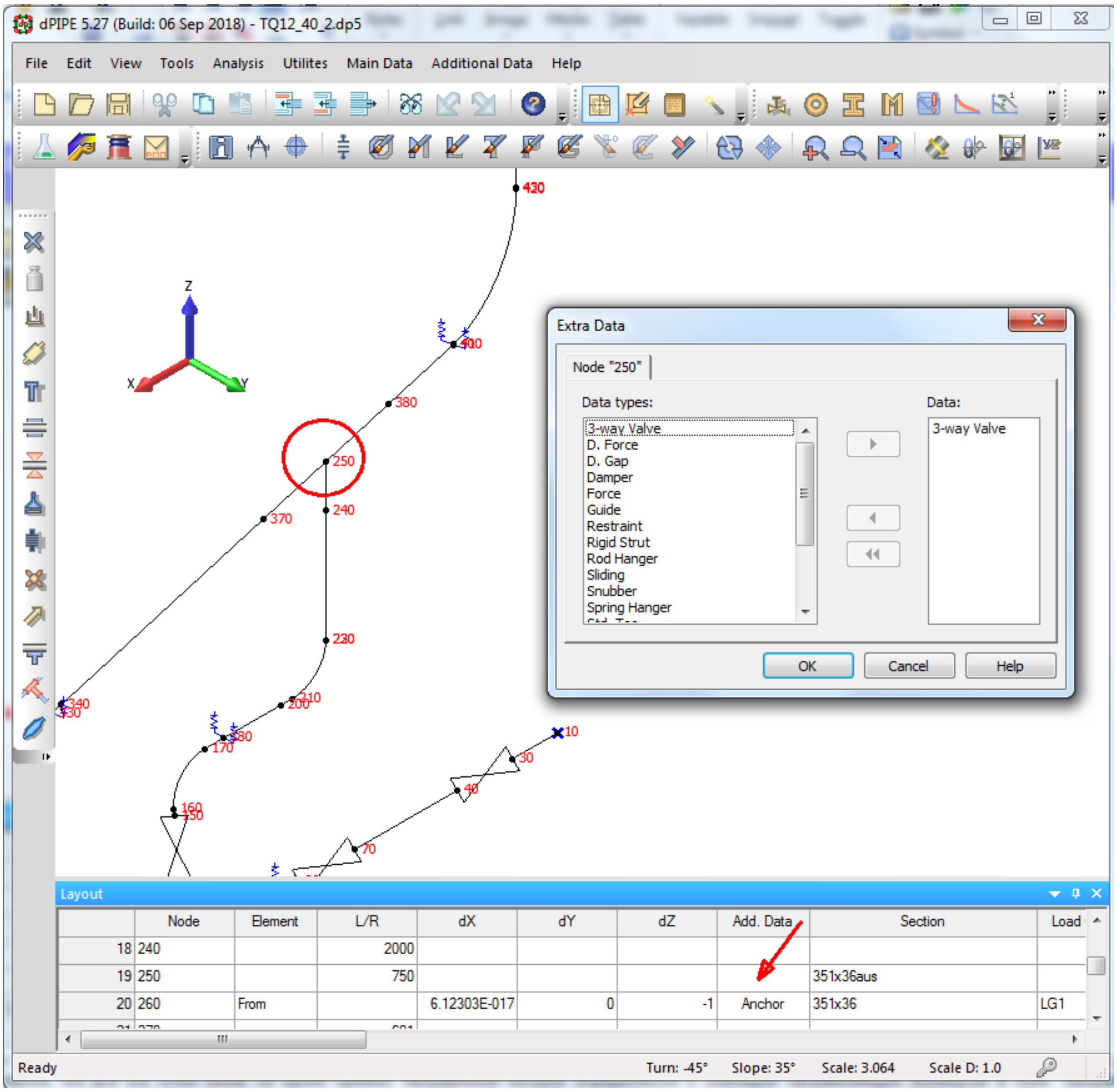
```
20: VO len = 130, w = 323,,, off = 0, 0, 1, len1 = 24, wop = 100, dir = 0, 0, 1, le
```

3-way valve (V3W)



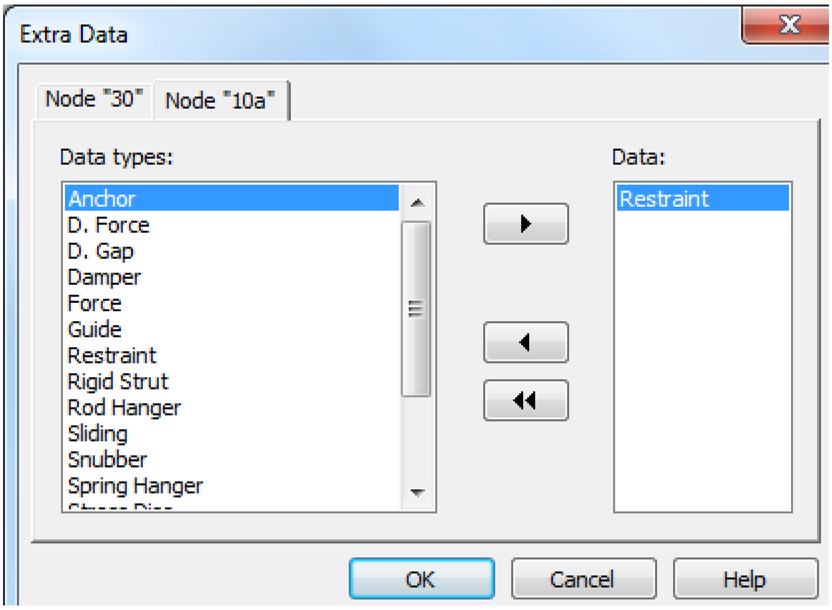
Three-way Valve model

The element "three-way valve" should be defined in the piping model in the same way as a TEE element through the entering in the “Add. Data” field:



In the dialog with the characteristics of the element, one can specify points for the center of mass of the valve body, the center of mass of the actuator and the node in which the actuator can be restrained (field "Node for actuator's restraining"). It is assumed that all these points lie on one straight line, the direction of which is determined by the direction cosines in the field "actuator's offset". The distance to the indicated points is measured from the central point of the element according to the principle a chain of dimensions. The following are examples of determining the parameters L2-L4 depending on the location of the points to be determined.

The support, for the actuator's restraining (if any), is entered in the same dialog "additional data" as the element itself on the tab with the name of this node:



The input window provides separate setting of the valve weights: e.g. weight of the valve body and the weight of the actuator. With the flags marked by default, the weight of insulation and piping content will be added to the element depending upon the characteristics of the adjacent pipe and the current load group. In doing so, the weight of insulation is calculated with the coefficient 1.75 of the pipe's weight, and the weight of the piping content is added to be equal to the weight of the working fluid in the straight pipe. The user can redefine these parameters having removed the flag marks taken by default. If a positive number is entered, then it will be taken by the program as weight (of insulation or piping content) in Newtons. In case of negative values being entered the program will take them as multipliers to the weight of insulation or working fluid. If the dead weight is set equal to zero,then the weight of insulation and working fluid will not be added to the element by default.

The "Loads from DB" field serves for selecting permissible loads in accordance with NP-068-05 "Pipeline valves for nuclear plants. General technical requirements".

The "Designation" field is used for setting the identification name of valves. This information is output for printing in listings of the input data and the results of analysis.

type: local geometrical command

Function: determines the element for the simulation of valves and fittings (slides, gates, valves, etc.). The stiffness matrix for an element is formed the same as for the "straight pipe" element of the current cross-section with the wall thickness being multiplied by the V_STF factor.

Parameters:

VL1 length of element (see model of the valve)

type:	REAL
unit:	mm
default value:	0
limitations:	VL1 > 0

VL2 length of the three-way valve's nozzle (see model of the valve)

type: REAL
unit: mm
default value: VL1/2
limitations: VL2 > 0

W(3) weight characteristics of the element[1)]

type: REAL
unit: Newton
dimension array of three numbers
default value: see Appendix IV.
limitations: -

MAT - reference identification name of the material (see MAT command)

type: TEXT
unit: -
default value: -
limitations: the name should coincide with the materials names determined earlier.

WOP weight of drive

type: REAL
unit: Newton
default value: 0
limitations: WOP ≥ 0

DIR(3) direction cosines for determination of the drive direction

type: REAL
unit: mm
dimension array of three numbers
default value: 0, 0, 0
limitations: -

LEN2, LEN3, LEN4 distances determining the center of mass of the housing, center of mass of the drive and the drive detachment point (see Fig. with the model).

type: REAL
unit: mm
default value: -
limitations: -

NODE[3)] name of the node for actuator's restraining

type: TEXT
unit: -
default value: -
limitations: see limitations for node marks

NOTE Note/Comment/- identification name of the valve

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

OTT_REF1 label identifying a record in the database with permissible loads on the outlet valve's nozzles (see Appendix X)

type: TEXT
unit: -
default value: -
limitations: to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

OTT_REF2 label identifying a record in the database with permissible loads on the inlet valve's nozzle (see Appendix X)

type: TEXT
unit: -
default value: -
limitations: to be determined by the values being present in the vlv_ott.dbs file. The length shall not exceed 8 characters

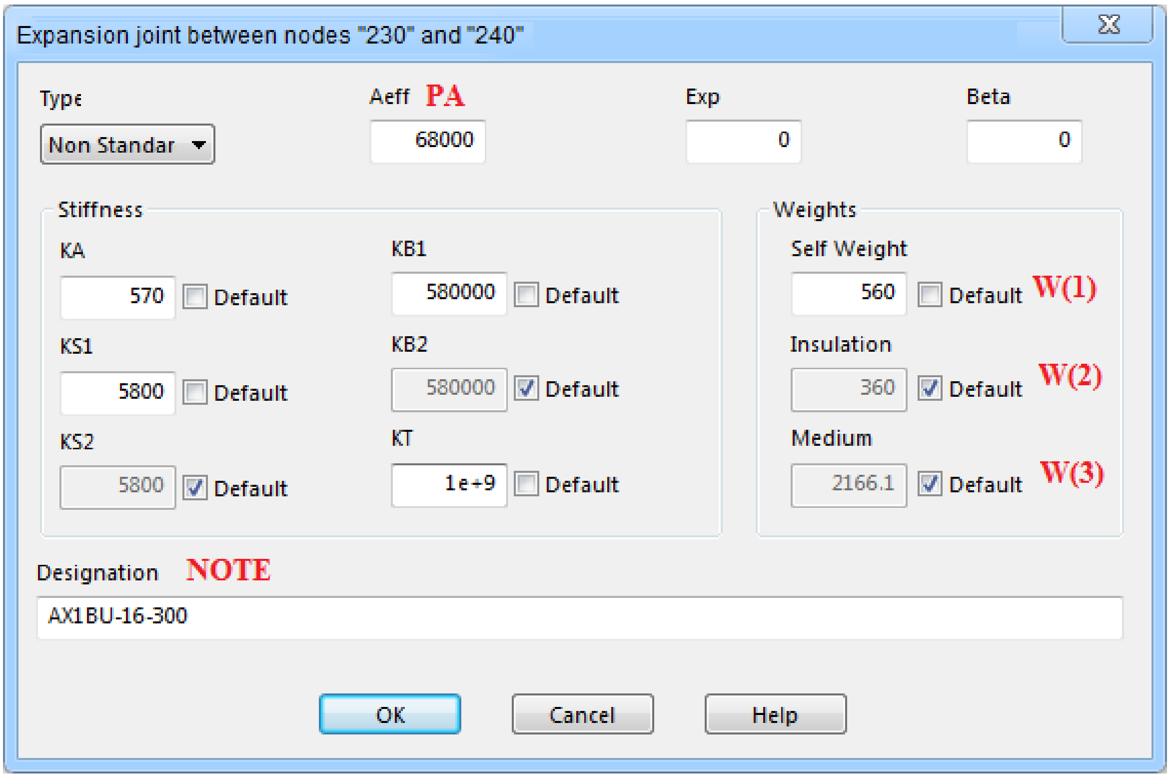
Note:

1) See Appendix IV .
2) The internal nodes of the valve can be used only for referencing a support thereto. They cannot be used as a reference value (for example, in the POS command or in operations with copying the model segments)

Example:

```
NODE: V3W vl1 = 800, vl2 = 300, w = 22000,,, note = "V1", ott_ref1 = '12', ott_ref2
= '15'
```

Expansion Joint (EJ)



type: local geometrical command

Function: command for modeling an expansion joint of the Custom (nonstandard) type

Parameters:

LEN length of element

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	the cross-section shall be preliminarily described by the PIPE command.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT

unit: -
default value: mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations: the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

KA axial spring rate (stiffness)

type: REAL
unit: N/mm
default value: axial stiffness of the matching pipe (E*A/L)
limitations: ≥ 0

KS(2) lateral (shear) spring rate (1 - along the local Y axis of the element; 2 - along the local Z axis)

type: REAL
unit: N/mm
dimension: array of 2 elements
default value: KS(1) = $\frac{12 * E * I}{L^3}$ (shear stiffness of the matching pipe);
KS(2) = KS(1)
limitations: ≥ 0

KB(2) bending spring rate(1 - with respect to the local Y axis of the element; 2 - with respect to the local Z axis)

type: REAL
unit: N*mm/rad
dimension: array of 2 elements
default value: KB(1) = $\frac{E * I}{L}$ (bending stiffness of the matching pipe);
KB(2) = KB(1)
limitations: ≥ 0

KT torsional spring rate

type: REAL
unit: N*mm/rad
default value: torsional stiffness of the matching pipe:
$$\frac{E * I}{L * (1 + \mu)}$$
limitations: ≥ 0

PA effective area of the expansion joint

type: REAL
unit: mm^2
default value: 0
limitations: ≥ 0

W(3) weight of the expansion joint[1)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

EXP thermal expansion multiplier [3)]

type:	REAL
unit:	-
default value:	0
limitations:	≥ 0

BETA angle to set orientation of element with respect to its axis (see Example of setting)

type:	REAL
unit:	degrees
default value:	0
limitations:	|BETA| ≤ 360

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name .
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

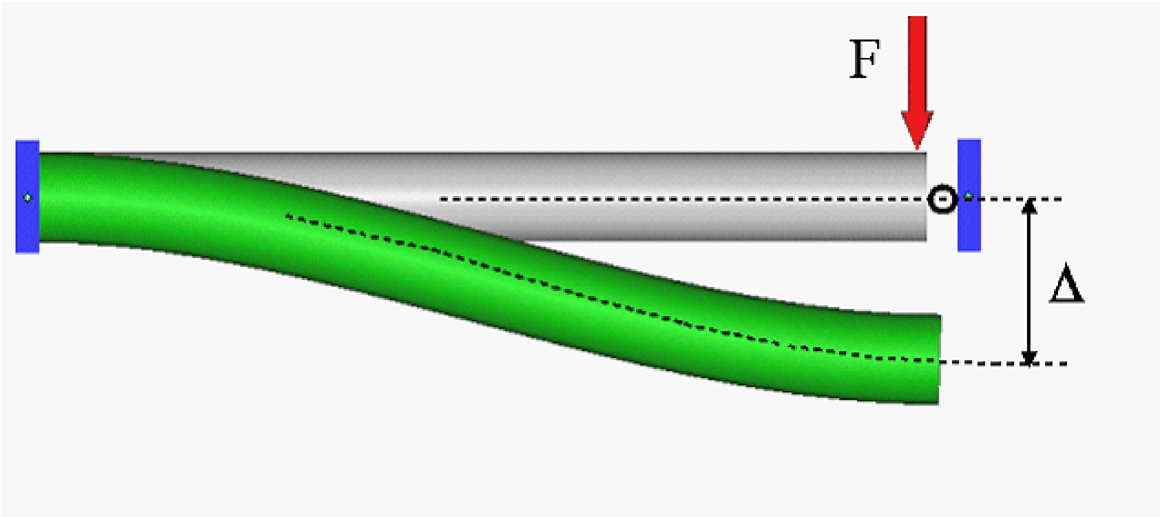
NOTE Element ID

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

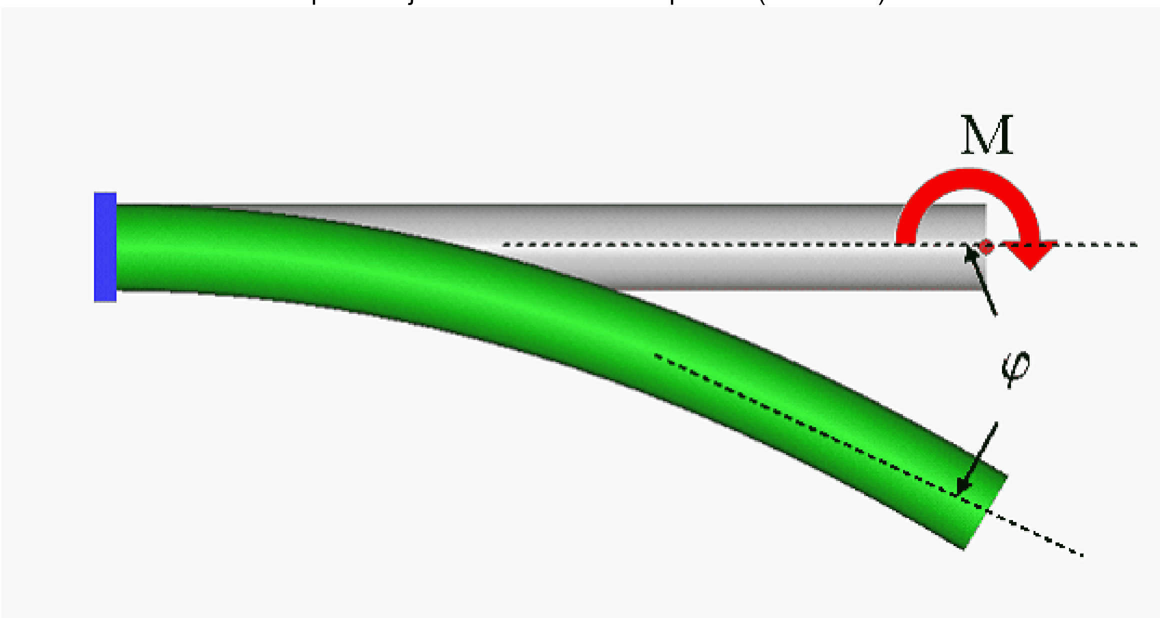
Note:

1) *When PA > 0 the longitudinal load (pressure thrust) R = PA*P(op) will be applied at the ends of the element (P(op) is the internal pressure concurrent with operation mode)*
2) *Thermal expansion multiplier EXP is used to scale current thermal expansion factor of the pipe adjacent to the bellow to the expansion joint material property: TEXP (EXPJ) = EXP * TEXP (pipe)*
3) *The lateral stiffness KS of the expansion joint corresponds to the parallel-plane deformation pattern for the ends of the element. The bending stiffness KB of the expansion joint*

corresponds to the deformation of a cantilevered beam under the action of concentrated moment:



Expansion joint lateral movement pattern (F = KS*Δ)



Expansion joint bending deflection pattern (M = KB*φ)

4) To get results of analysis for expansion joints (transverse and angular deformations) it's necessary to set the output of internal forces in the [POST](#) command for the load combination being of interest (RES = 'FORC' parameter). In doing so, a separate table containing deformations (linear and angular) will be printed out in the <model name>.sup file for each expansion joint for all sets of results with internal forces.

Example:

```
2000: EJ 1000 KA 100. KS 150 200. KB 320 340 KT 1000 PA 203 W 80, -1.2, -1
```

All expansion joint subtypes: A[xial (EA)](#) , [Tied (ET)](#) , [Hinge (EH)](#) , [Gimbal (EG)](#) are modeled with use the following values of its parameters:

Modeling of Expansion Joints

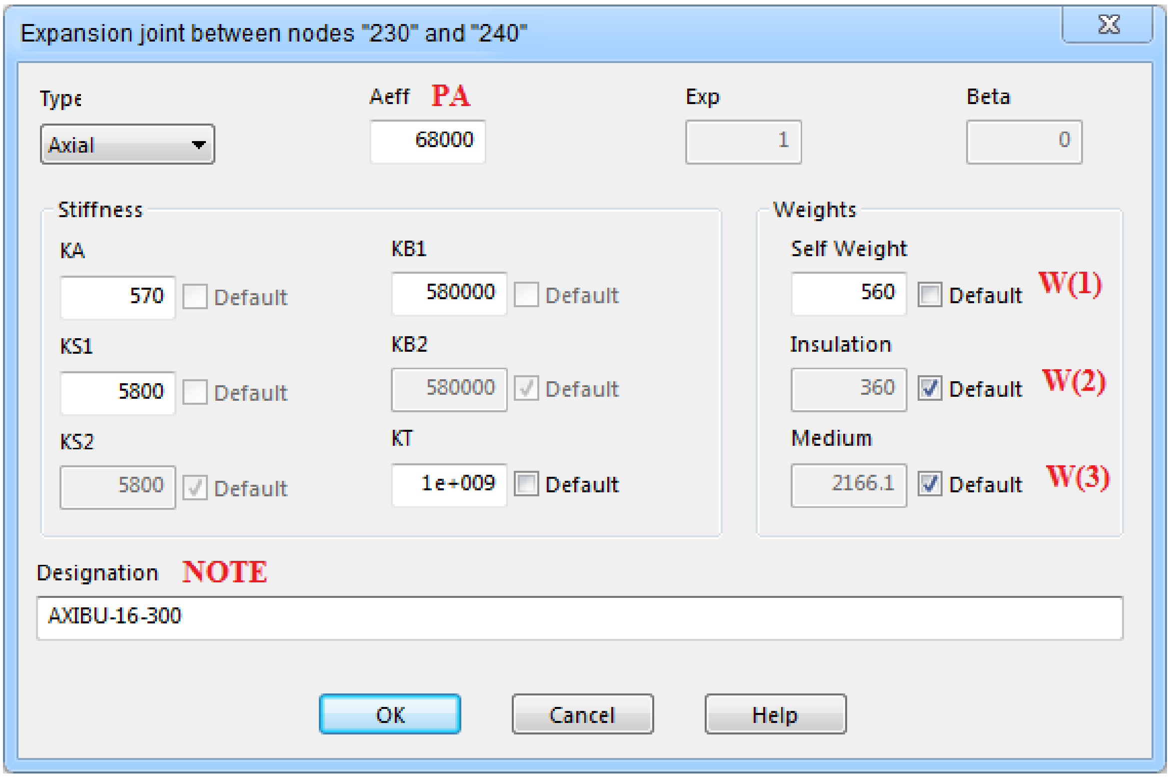
TYPE	KA	KS(1)	KS(2)	KB(1)	KB(2)	KT	PA	EXP

EA	ka	ks	ks	kb	kb	R	pa	1
ET (>2 tie rods)	R	ks	ks	R	R	R	0	0
ET (2 tie rods)	R	ks	ks	kb	R	R	0	0
EH	R	R	R	kb	R	R	0	0
EG	R	R	R	kb	kb	R	0	0

Note: R - "rigid" value, corresponds to relevant stiffnesses of the matched pipe:

Stiffness	Формула
Axial (KA)	$\frac{E * A}{L}$
Shear (KS)	$\frac{E * I}{L^3}$
Bending (KB)	$\frac{E * I}{L}$
Torsion (KT)	$\frac{E * I}{L * (1 + \mu)}$

Axial Expansion Joint (EA)



type: local geometrical command

Function: command for modeling an axial expansion joint

Parameters:

LEN length of element

type: REAL
unit: mm
default value: 0
limitations: LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type: TEXT
unit: -
default value: the cross-section shall be preliminarily described by the PIPE command.
limitations: the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type: TEXT
unit: -
default value: mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations: the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

KA axial spring rate (stiffness)

type: REAL
unit: N/mm
default value: -
limitations: ≥ 0

KS lateral (shear) spring rate

type: REAL
unit: N/mm
default value: -
limitations: ≥ 0

KB bending spring rate

type: REAL
unit: N*mm/rad
default value: -
limitations: ≥ 0

KT torsional spring rate

type: REAL
unit: N*mm/rad

default value: torsional stiffness of the matching pipe: $\frac{E * I}{L * (1 + \mu)}$

limitations: ≥ 0

PA effective area of the expansion joint

type: [REAL]
unit: mm^2
default value: -
limitations: ≥ 0

W(3) weight of the expansion joint[1)]

type: [REAL]
unit: Newton
dimension array of three numbers
default value: see [Appendix IV].
limitations: -

NAME - identification name of the piping segment

type: [STRING]
unit: -
default value: blank line or current identification name .
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

NOTE Element ID

type: [STRING]
unit: -
default value: [blank]
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

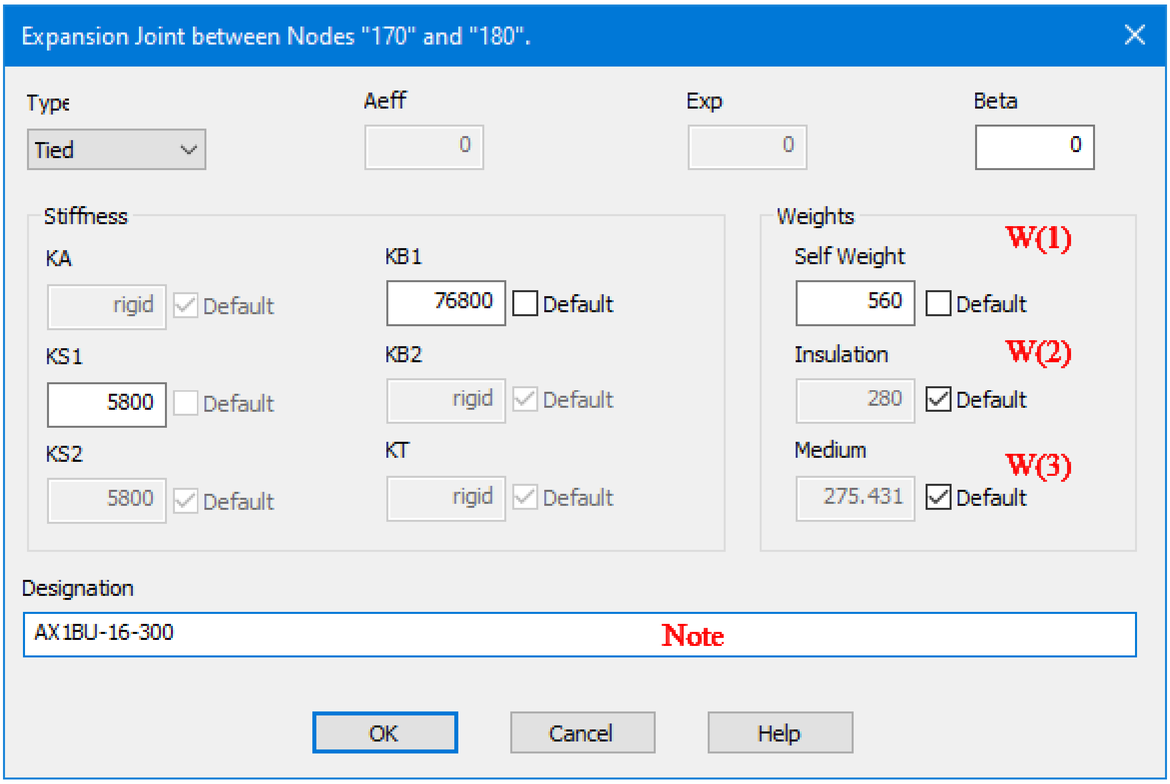
Note:
1) *Mandatory parameter to be used for computation of the pressure thrust consistent with operating modes.*
2) *See Appendix IV.*

Example :

```
A10: EA 340, ka = 61.294, ks = 252.358, kb = 1.84E+006, pa = 1.88E5, w = 747.3
```

See also: *Modeling of Expansion Joints*

Tied Expansion Joint (ET)



type: local geometrical command

Function: command for modeling a tied Expansion Joint

Parameters:

LEN length of element

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	the cross-section shall be preliminarily described by the PIPE command.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

KS lateral (shear) spring rate

type:	REAL
unit:	N/mm
default value:	-
limitations:	≥ 0

KB[1)] bending spring rate

type:	REAL
unit:	N*mm/rad
default value:	-
limitations:	≥ 0

BETA[1)] angle to set orientation of element with respect to its axis (see Example of setting)

type:	REAL
unit:	degrees
default value:	0
limitations:	|BETA| ≤ 360

W(3) weight of the expansion joint[2)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name .
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

NOTE Element ID

type:	STRING

unit: -
default value: [blank](#)
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

Note:

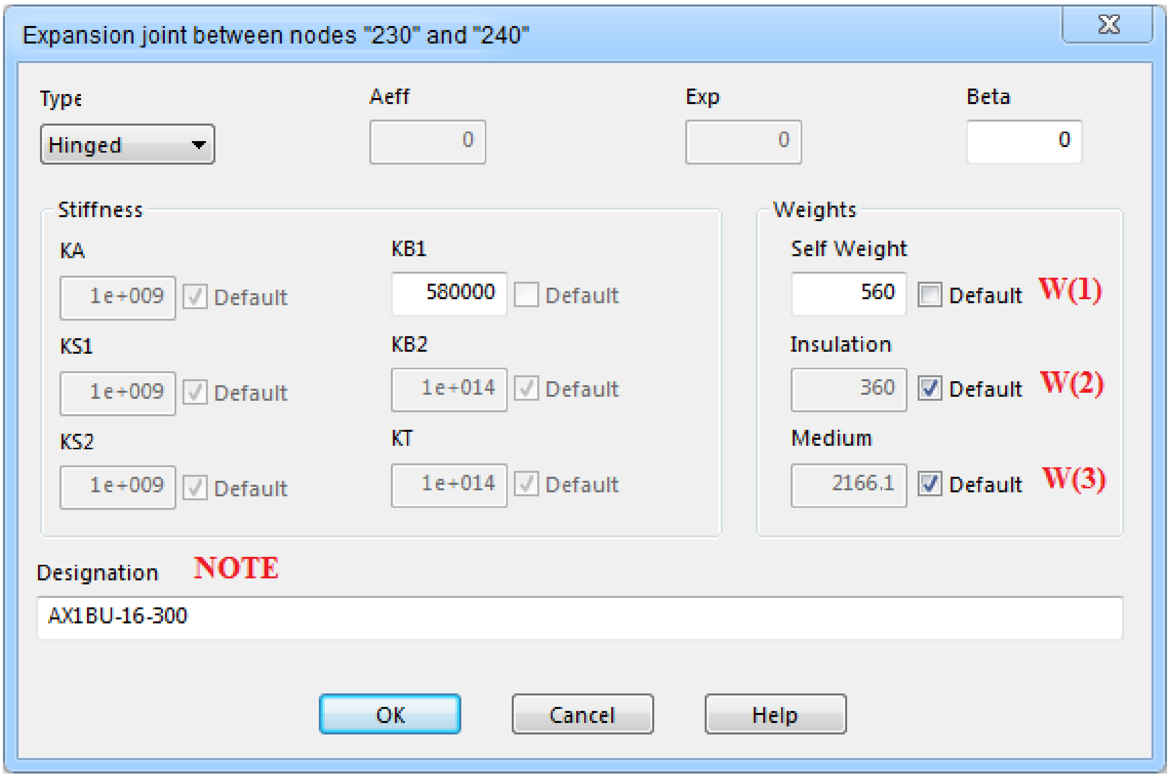
1. ***For modeling the expansion joint having 2 tie rods the setting of KB and BETA parameters is mandatory****. In case of the angle BETA = 0, it is assumed that the tie rods are located in the local X-Y plane of the element.*
2. *See [Appendix IV](#) .*

Example:

```
2000: ET 1000 KS 100. KB 1000. BETA 30
```

See also*: [Modeling of Expansion Joints](#)*

Hinge Expansion Joint (EH)



type: local geometrical command

Function: command for modeling a hinge expansion joint[1)]

Parameters:

LEN length of element

type: [REAL](#)

unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	the cross-section shall be preliminarily described by the PIPE command.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

KB bending spring rate

type:	REAL
unit:	N*mm/rad
default value:	-
limitations:	≥ 0

BETA[1)] angle to set orientation of element with respect to its axis (see Example of setting)

type:	REAL
unit:	degrees
default value:	0
limitations:	|BETA| ≤ 360

W(3) weight of the expansion joint[2)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

NAME identification name of the piping segment

type:	STRING
unit:	-

default value: blank line or current identification name .
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

NOTE Element ID

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

Note:

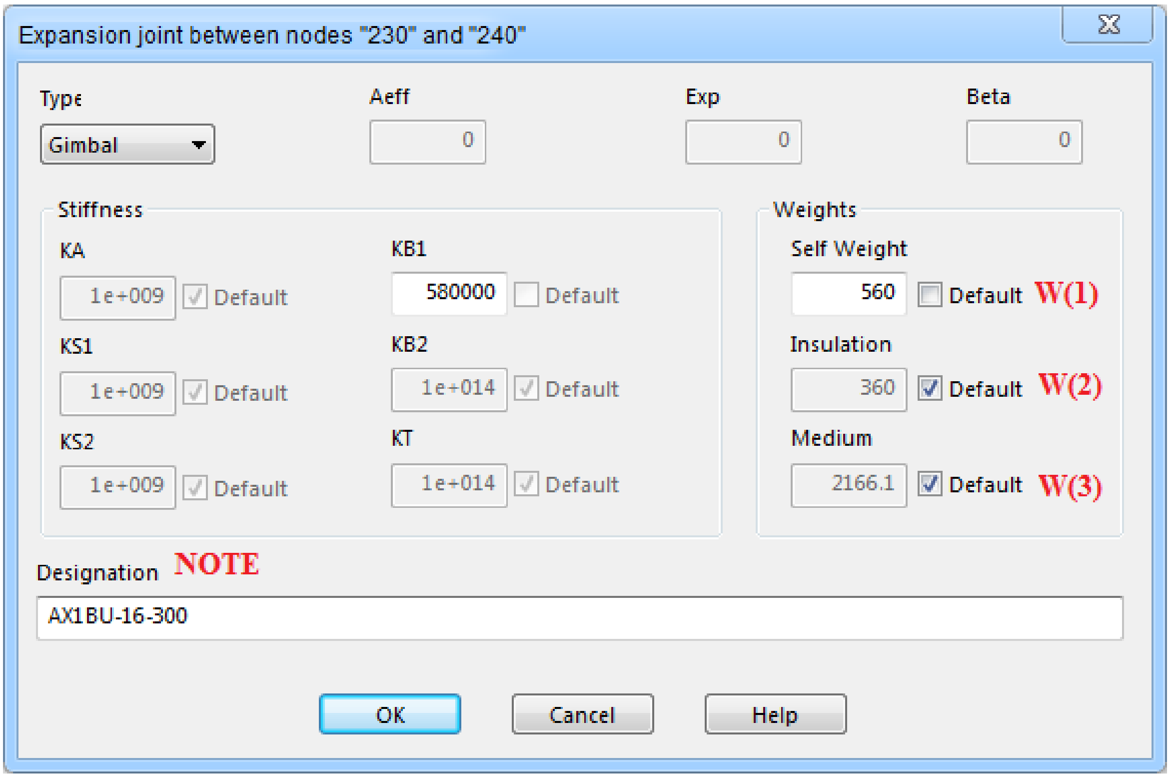
1) *In case if the angle BETA = 0, it is assumed that the hinge expansion joint allows rotating about the local Y axis of the element;*
2) *See Appendix IV .*

Example:

```
2000: EH 1000 KB 1000. BETA 30
```

See also*: Modeling of Expansion Joints*

Gimbal Expansion Joint (EG)



type: local geometrical command

Function: command for modeling a gimbal expansion joint

Parameters:

LEN length of element

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	the cross-section shall be preliminarily described by the PIPE command.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

KB bending spring rate

type:	REAL
unit:	N*mm/rad
default value:	-
limitations:	≥ 0

W(3) weight of the expansion joint[2)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

NAME identification name of the piping segment

type:	STRING
unit:	-

default value: blank line or current identification name .
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

NOTE Element ID

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

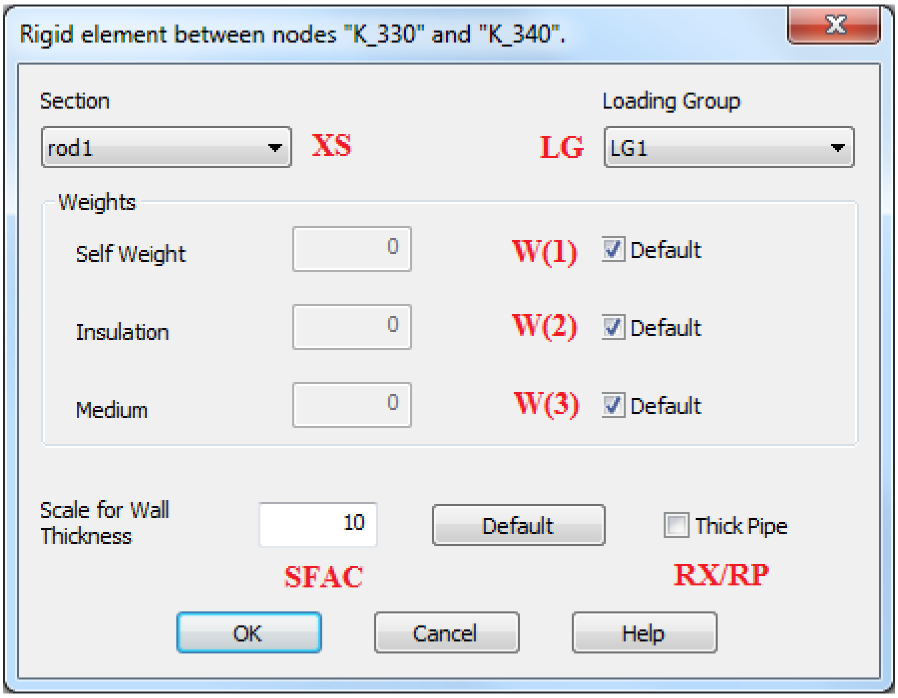
Note:
1) See Appendix IV .

Example:

```
2000: EG 1000 KB 1000.
```

See also*: Modeling of Expansion Joints*

Rigid Link (RX/RP)



type: local geometrical command

Function: command for modeling a "rigid" element. Depending upon the type of displaying, the following is used: RX - rigid link and RP - "thick" pipe.

Parameters:

LEN length of element

type:	REAL
unit:	mm
default value:	0
limitations:	LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
default value:	the cross-section shall be preliminarily described by the PIPE command.
limitations:	the cross-section shall be preliminarily described by the PIPE command

LG initialize the load group

type:	TEXT
unit:	-
default value:	mandatory parameter if the load group is not initialized. Otherwise it takes the value of the current load group.
limitations:	the load group shall be preliminarily described by one of the "LG" subcommands of the OPVAL command

W(3) weight of the element[2)]

type:	REAL
unit:	Newton
dimension	array of three numbers
default value:	see Appendix IV.
limitations:	-

SFAC scaling factor at the pipe wall thickness[1)]

type:	REAL
unit:	-
default value:	10
limitations:	> 0

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name .
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

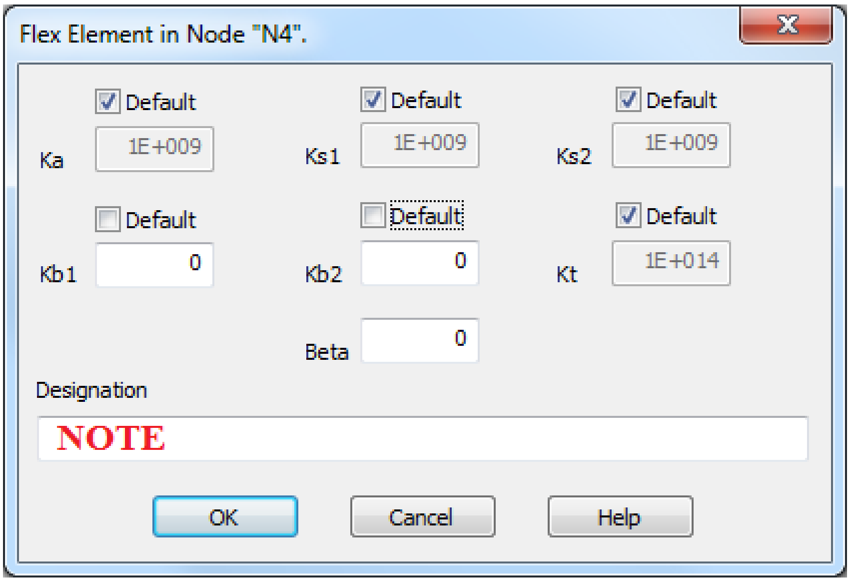
Note:

1) *The element is simulated using a pipe with the current internal diameter and the wall thickness increased by SFAC times: T = T*SFAC*
2) *See Appendix IV .*

Example:

```
2000: RX 1000 SFAC .5
```

Flexible Joint (FJ)



type: local geometrical command

Function: command for modeling a flexible joint

Parameters:

DC or **DS:** see "Local commands" section for the description of parameters

KA axial spring rate (stiffness)

type:	REAL
unit:	N/mm
default value:	rigid
limitations:	≥ 0

KS(2) lateral (shear) spring rate

type:	REAL
unit:	N/mm
dimension	array of two numbers

	default value:	KS(1) = rigid, KS(2) = KS(1)
	limitations:	≥ 0
KB(2)	bending spring rate	
	type:	REAL
	unit:	N*mm/rad
	dimension	array of two numbers
	default value:	KB(1) = rigid, KB(2) = KB(1)
	limitations:	≥ 0
KT	torsional spring rate of the joint	
	type:	REAL
	unit:	N*mm/rad
	default value:	rigid
	limitations:	≥ 0
BETA	angle to set orientation of element with respect to its axis (see Example of setting)	
	type:	REAL
	unit:	degrees
	default value:	0
	limitations:	|BETA| ≤ 360
NAME	identification name of the piping segment	
	type:	STRING
	unit:	-
	default value:	blank line or current identification name .
	limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters
NOTE	Element ID	
	type:	STRING
	unit:	-
	default value:	blank
	limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

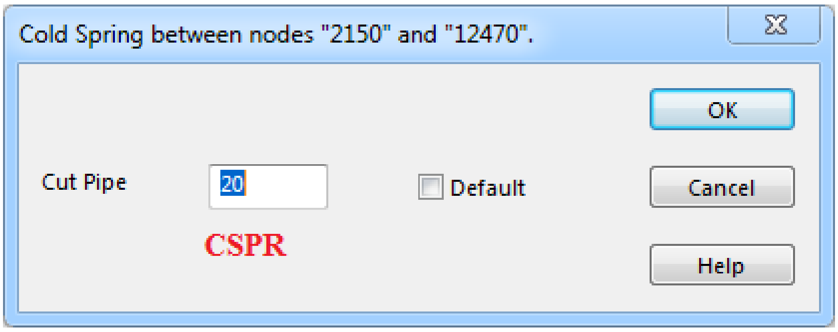
Note:

The flexible joint has the zero length. It is used for setting the local flexibility (stiffness) in the model. The values KA and KT shall be set with respect to the local "A" axis, KS(1) and KB(1) - with respect to the local "H" axis, KS(2) and KB(2) - with respect to the local "N" axis (see Example of setting);

Example :

```
2000: FJ KA 1.e2 KB 3.e4 KS 1.e4 2.1e4 KT 1.e4 beta 45 dc 0 1 1
```

Cold Spring (CS)



type: local geometrical command

Function: element for modeling a cold spring

Parameters:

LEN[1)] length of element

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

DC or **DS**: see "Local commands" section for the description of parameters

XS change the cross-section type only for the current element without changing the current value for the rest of the segment

type:	TEXT
unit:	-
limitations:	the cross-section shall be preliminarily described by the PIPE command

CSPR cold spring value

type:	REAL
unit:	mm
default value:	LEN
limitations:	-

NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name .

limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

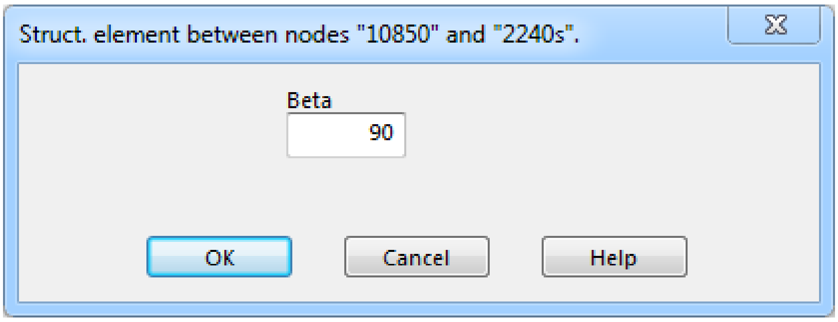
Note:

1) *The element length can be equal to zero.*
2) *To set Cold Spring in force a corresponding option 'CS' should be used within SOLV command. See Appendix VII for comments to the specification for analysis of the piping system with the cold spring*

Example :

```
10: CS 145 dc 1,0,0 xs 'CS250A' CSPR = 100.
```

Beam (S)



type: local geometrical command

Function: command for modeling a beam elements (structural elements)

Parameters:

LEN length of element

type: REAL
unit: mm
default value: 0
limitations: LEN ≥ 0

DC or **DS:** see "Local commands" section for the description of parameters

BEAM - identification name of the cross-section

type: TEXT
unit: -
default value: -
limitations: the cross-section shall be preliminarily described by the BEAM command

BETA[1)] angle to set orientation of element with respect to its axis

type:	REAL
unit:	degrees
default value:	0
limitations:	|BETA| ≤ 360

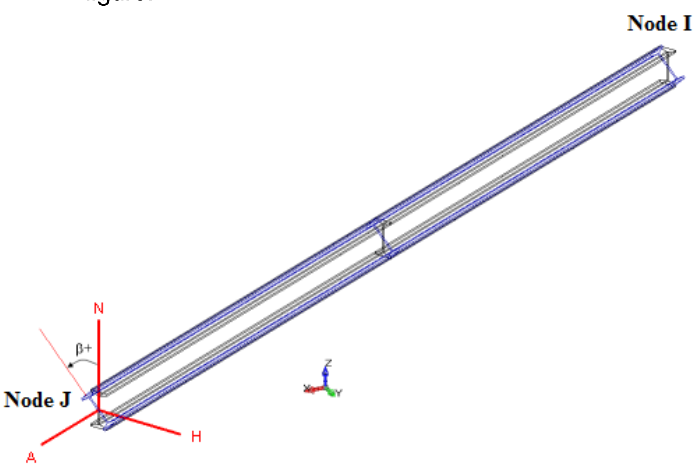
NAME - identification name of the piping segment

type:	STRING
unit:	-
default value:	blank line or current identification name .
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

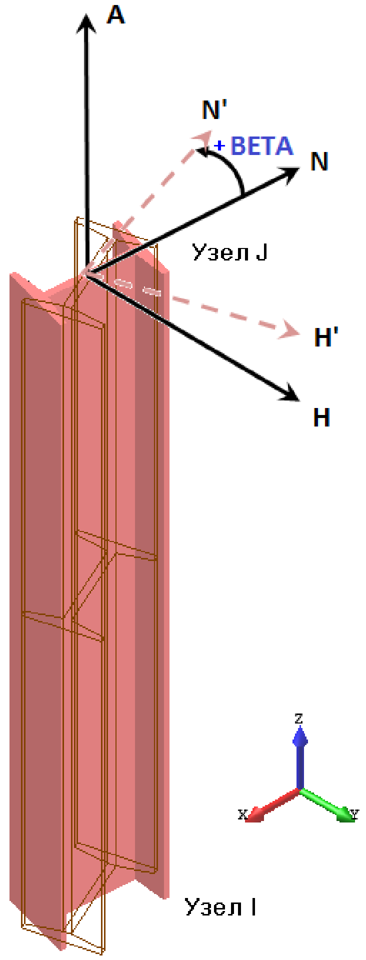
SBP feature that defines the element and following portion of the piping as "small bore pipe" (initialization parameter), see SBP

Notes:

1) The angle BETA defines the rotation of cross-section around the local A axis. An example of setting the BETA angle and the positive direction of its counting are shown in the following figure:



1) the local A axis is not parallel to the global Z axis.



2) the local A axis is parallel to the global Z axis (H is directed along the global Y axis).

Example:

```
2000: S LEN 1000 BEAM 'L10'
```

Set Position (POS)

type: local geometrical command

Function: command for setting the global or relative coordinates of the node. If model does not contain the POS command, then the global coordinates of the first node of the model are set to zero. When POS is assigned to some node, the global coordinates of the model are recalculated. Existing of two or more POS commands could lead to the conflict between model's global coordinates. In such a case a warning message will appear.

Parameters:

X, Y, Z global coordinate of the node

type:	REAL
unit:	mm
default value:	-
limitations:	-

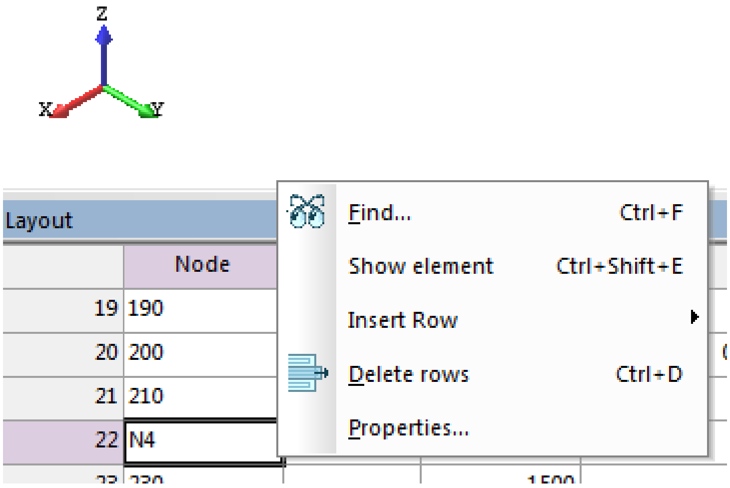
NODE label of the node to be linked

type:	TEXT
unit:	-
default value:	-
limitations:	the parameter shall refer to the label of one of the existing nodes of the model

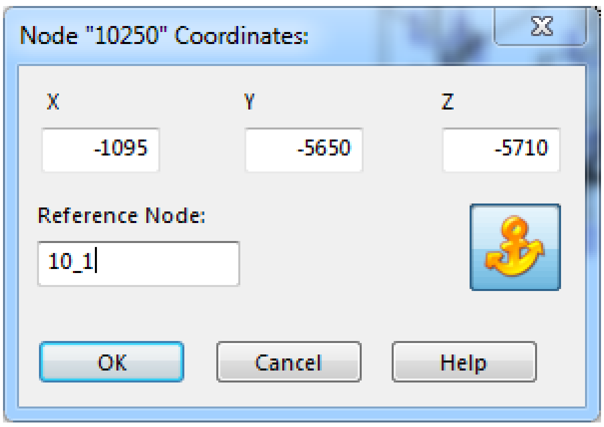
Note:

To set POS command the following actions should be executed:

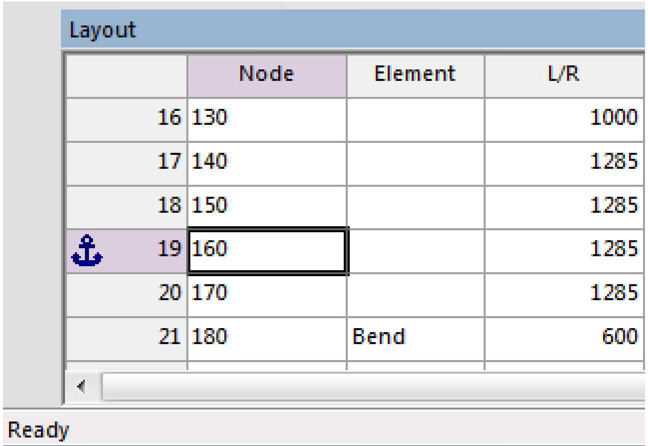
1. *Place the mouse cursor to the required node and select the "Properties" item:*



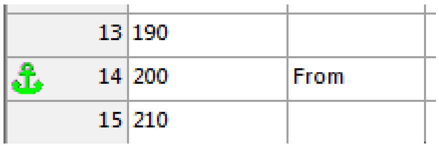
2. *Click the "Properties" item in the dialog appeared and enter either the absolute global coordinates of the node or define them with respect to the coordinates of the existing node:*



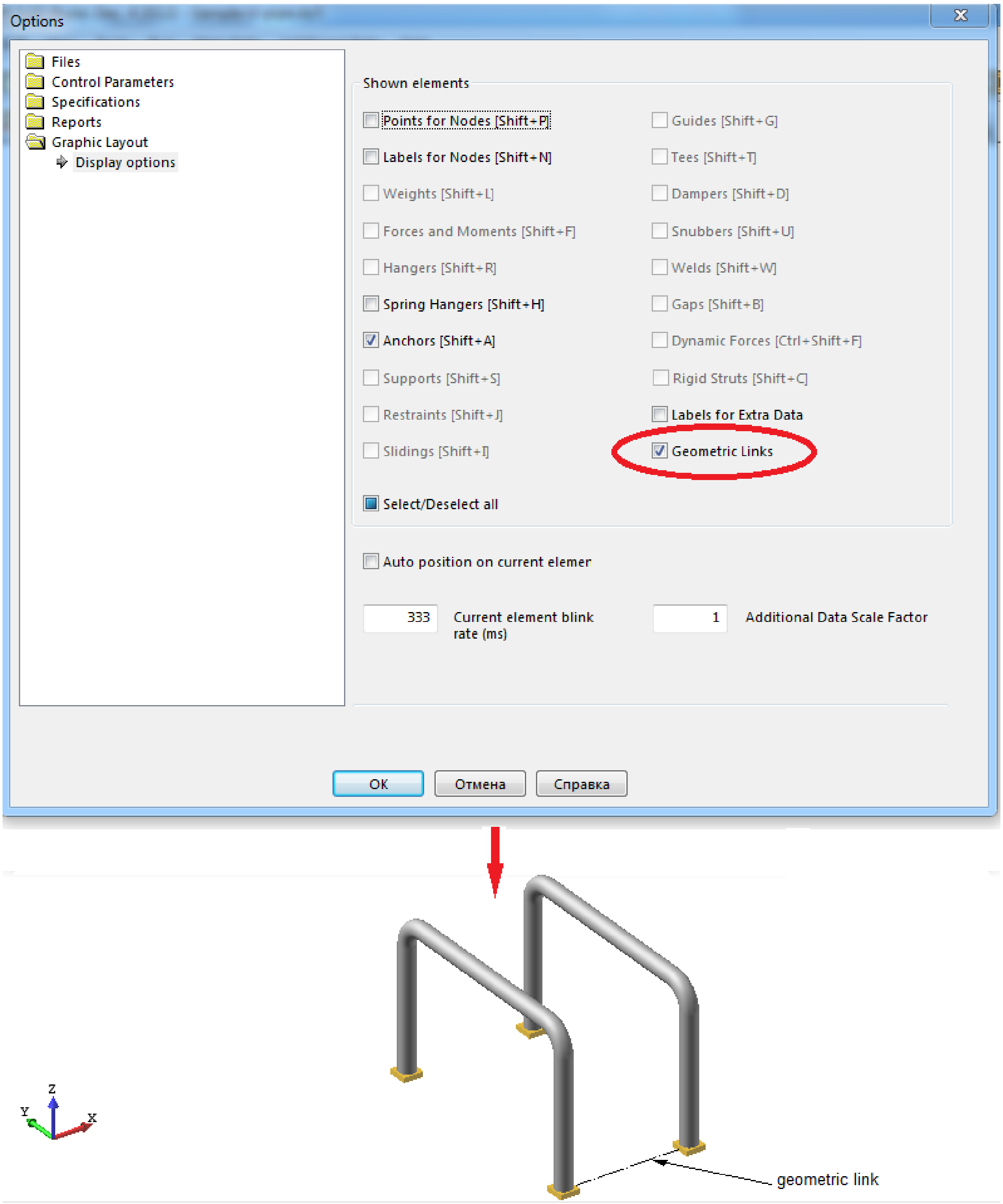
3. *Press "OK". In doing so the node, for which the absolute global coordinates are set, will be marked with the blue-colored "anchor" symbol:*



if the coordinates are set with respect to the existing node, then the anchor will be displayed in green:



If there are geometric links between various nodes in the model, they can be displayed by means of the "Display parameters" menu ("Geometric links" option):



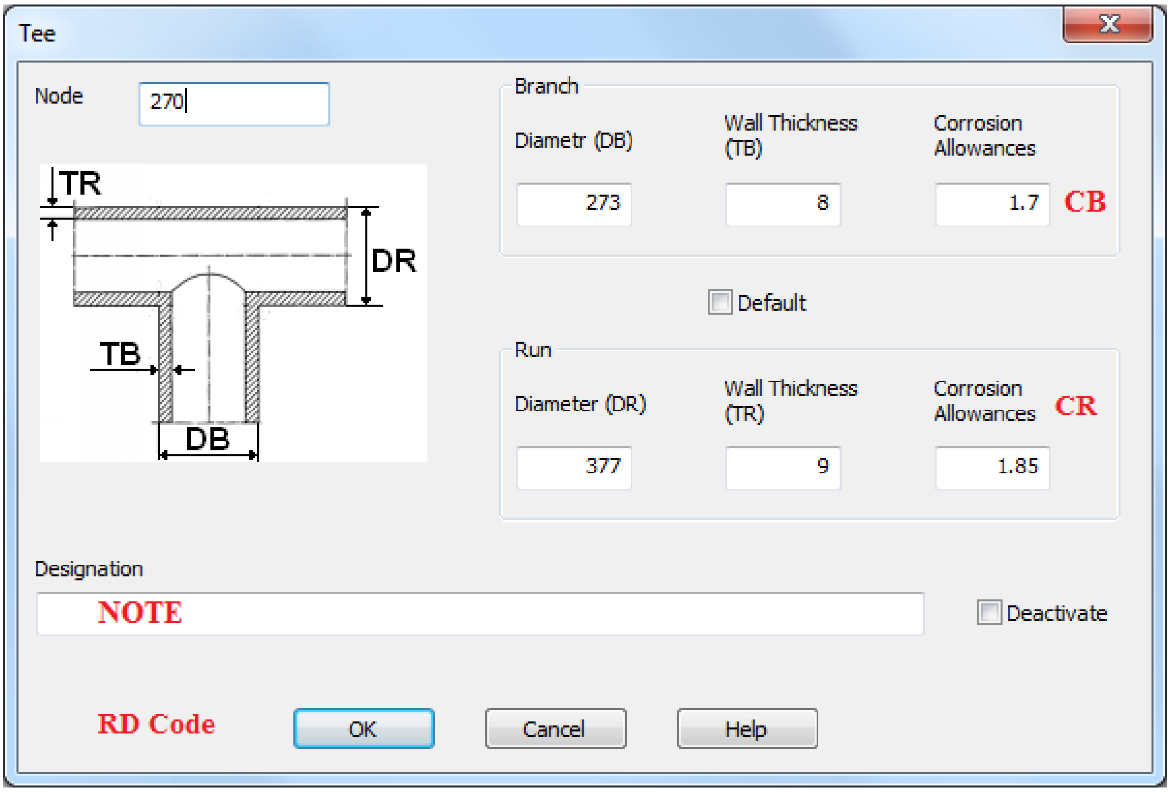
Example:

```
20: POS 0, -2000, 0 NODE '10A'
```

Tee/Branch Connection (TEE)

[1] Starting from dPIPE 5.27, this command is not recommended for the input of tees. Instead, use the command for the "standard" tees"

[2)] The command is not compatible with the calculations according to the EN Norms of the 2020 edition



type: local geometrical command

Function: Command for setting the characteristics of tees/branch connections located in the current node. Command is available only if node belongs to several interconnected elements that form branching. Dialog and list of parameters are dependent on the Stress Code used.

Parameters:

TYPE type of the tee

type:	TEXT
unit:	mm
default value:	'WLD'
limitations:	in accordance with the codes, see the following table:

CODE	CODE_YEAR	TYPE	Description in accordance with the Codes
ASME_B311	2008	WLD	Welding tee per ASME B16.9
		RF_TEE	Reinforced fabricated tee
		URF_TEE	Unreinforced fabricated tee

		RF_BRC	Branch welded-on fitting (integrally reinforced) per MSS SP-97
		EX_OUT	Extruded outlet meeting the requirements of para. 104.3.1(G)
		WLD_INS	Welded-in contour insert
		BRC	Branch connection
ASME_NC	1992	WLD	Welding Tee per ANSI B19.6 (1)
		BRC	Branch connection (2)
		RF_TEE	Reinforced fabricated tee
		URF_TEE	Unreinforced fabricated tee
	2010	WLD	Welding Tee per ANSI B19.6
		RF_TEE	Reinforced fabricated tee
		BRC	Branch connection or unreinforced fabricated tee
		WLD_BRC	Fillet welded and partial penetration welded branch connections
ASME_NB	1992	WLD	Butt welding tees
		BRC	Branch connections per NB-3643
	2010	WLD	Butt welding tees
		BRC	Branch connections per NB-3643
EN	2002	URF_TEE	tee with welded-on, welded-in or extruded nozzle
		RF_TEE	tee with welded-on, welded-in or extruded nozzle with additional reinforcing ring
		WLD	forged welded-in tee
		BRC	particular connections
ASME_B314	2006	WLD	Welding tee per ASME B16.9
		RF_TEE	Reinforced tee with pad or saddle
		URF_TEE	Unreinforced fabricated tee
		EX_OUT	Extruded welding tee

DB branch pipe outside diameter

type: REAL
unit: mm
default value: outside diameter set in the cross-section of adjacent pipe from the BRANCH side
limitations: > 0 + see Appendix 2

TB branch pipe wall thickness

	type:	REAL
	unit:	mm
	default value:	wall thickness set in the cross-section of adjacent pipe from the BRANCH side
	limitations:	TB > 0; DB-2*TB> 0 + see Appendix 2
DR	run pipe outside diameter	
	type:	REAL
	unit:	mm
	default value:	outside diameter of the RUN pipe
	limitations:	> 0 + see Appendix 2
TR	run pipe wall thickness	
	type:	REAL
	unit:	mm
	default value:	wall thickness of the RUN pipe
	limitations:	TR > 0; [DR-2*(TR-CR)] > 0 + see Note 2
CR1)	combined mill tolerance and corrosion allowance for the RUN pipe	
	type:	REAL
	unit:	mm
	default value:	inherits value from the RUN pipe cross-section data
	limitations:	CR ≥ 0, TR – CR > 0
CB1)	combined mill tolerance and corrosion allowance for the BRANCH pipe	
	type:	REAL
	unit:	mm
	default value:	inherits value from the BRANCH pipe cross-section data
	limitations:	CB ≥ 0, TB – CB > 0
TW	reinforcement of the branch in the zone of welding (see Figure)	
	type:	REAL
	unit:	mm
	default value:	TB
	limitations:	≥ TB
TN	wall thickness of nozzle or branch connection reinforcement ; it is used for 'BRC' or WLD_'BRC' type	
	type:	REAL
	unit:	mm
	default value:	TB
	limitations:	≥ TB
RP	outside radius of reinforced nozzle or branch connection; it is used for a tee of 'BRC' and WLD_'BRC' type (see Appendix XI, Figure NC-3673.2(b)-2)	

type:	REAL
unit:	mm
default value:	TB
limitations:	≥ TB

R2 radius of rounding in the region of branch connection; it is used for a tee of 'BRC' type (see Appendix XI, Figure NB-3643.3(a)-1)

type:	REAL
unit:	mm
default value:	TB
limitations:	≥ TB

TE thickness of the Pad or reinforcement; it is used for a tee of 'RF_TEE' type (see Appendix XI, Figure NC-3673.2(b)-2)

type:	REAL
unit:	mm
default value:	TB
limitations:	≥ TB

SI (10) array of stress indexes, which are interpreted by the program depending upon the Stress Code being used, see the following table:

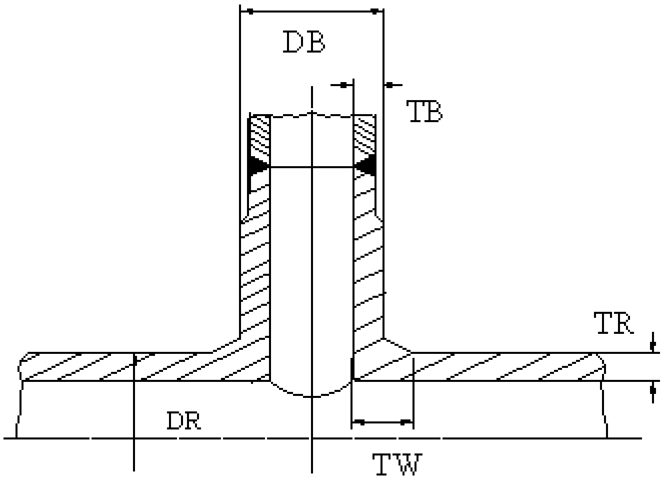
CODE	(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
PNAE, PNAE_T	$K_{и(s)}$	a_s								
ASME_NC	B_1	B_{2R}	B_{2B}	S_{IR}	S_{IB}	B_{2R}'	B_{2B}'			
ASME_NB	C_1	K_1	B_{2R}	B_{2B}	C_{2R}	C_{2B}	K_{2R}	K_{2B}	B_{2R}'	B_{2B}'
EN	S_{IR}	S_{IR}*0.75	S_{IB}	S_{IB}*0.75						
ASME_B311	S_I	S_I*0.75								

type:	REAL
unit:	-
default value:	to be determined by the program automatically
limitations:	SI(1) ≥ 2 ; SI(2) ≥ 1 (for PNAE Codes)

NOTE Element ID

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

Note:



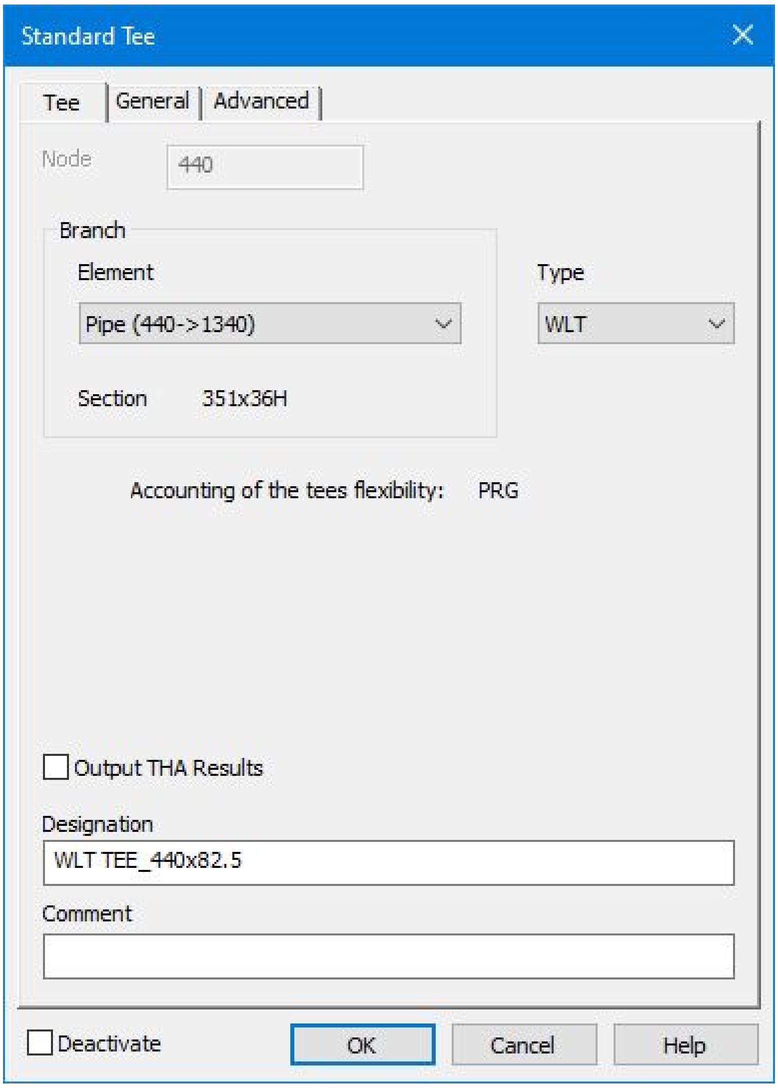
Cross-section of the tee joint (Russian PNAE Code)

1) CR and TW parameters are used only for PNAE and RD Codes; CB parameter is used only for RD.
2) The diameter and thickness of the RUN (DR, TR) and BRANCH (DB, TB) cannot be less than similar parameters for the pipes being connected

Example:

```
2000: TEE DB 108 TB 9 DR 325 TR 16
```

"Standard" (TEE)



Type: local geometrical command

Function: Command for setting the characteristics of the standard tee/branch connection joints. Data for such fitting should be previously defined in the pipe sections (see &TEE subcommand). Elements that form header pipe should be located inline. Branch pipe should be orthogonal to the header pipe. Allowable tolerances from above angles are defined by TBRC_TOL and TRUN_TOL parameters.

Parameters:

ID identification name of the standard fitting. It could be set from the available set appeared in the drop down list (required parameter)

type:	TEXT
units:	-
default value:	-
limitations:	see &TEE subcommand

BP Reference to the existing node located on the branch pipe. BP is optional parameter that could be used to set collectors, pipe crosses, etc.:

Watch Video

type: TEXT
units: -
default value: -
limitations: existing node

CODE Reference on strength analysis code (applicable only for CODE = 'NTD_ASI')

type: TEXT
units: -
default value: CODE
limitations: same as CODE

CODE_YEAR year of publication of the codes (edition)

type: INTEGER
units: -
default value: CODE_YEAR
limitations: same as CODE_YEAR

NOTE Element ID

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

Example:

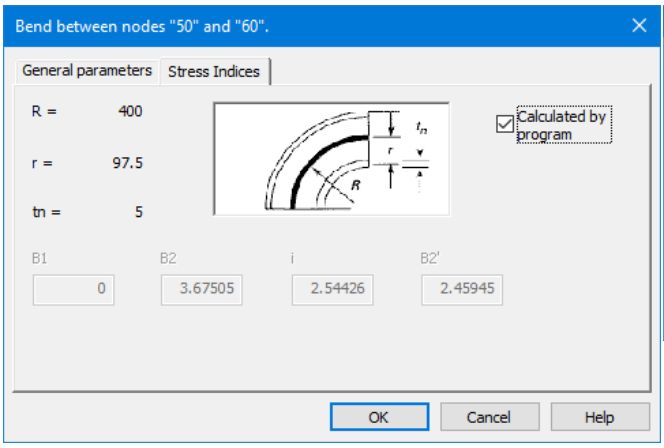
```
20: TEE id = 'BRC'
```

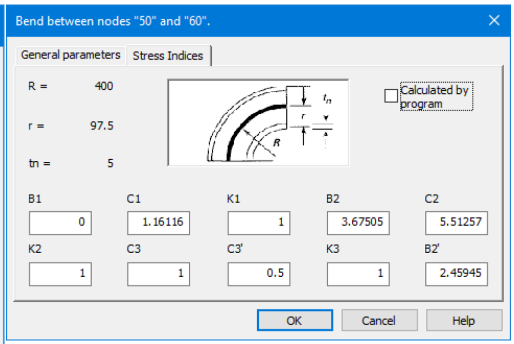
Stress Indexes

These tabs show stress intensification factors/stress indexes calculated by the program depending on the CODE used for stress analysis. By default, the coefficients are calculated in accordance with the requirements of the Norms, but if one uncheck the “calculated by program” checkbox, these coefficients may be redefined by User.

BEND/ELBOW:

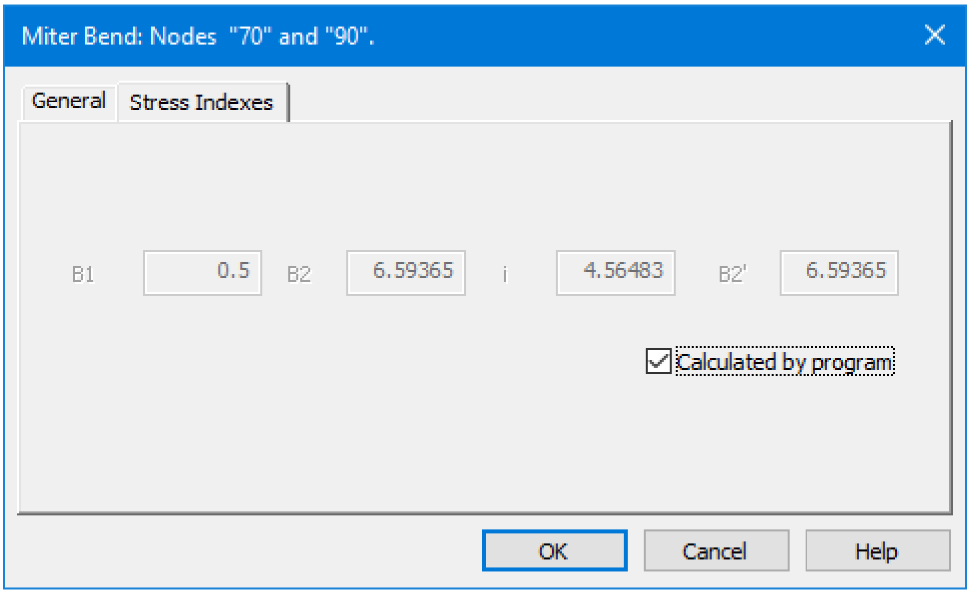
CODE = 'ASME_NC', 'EN' CODE = 'ASME_NB', CLS = 1





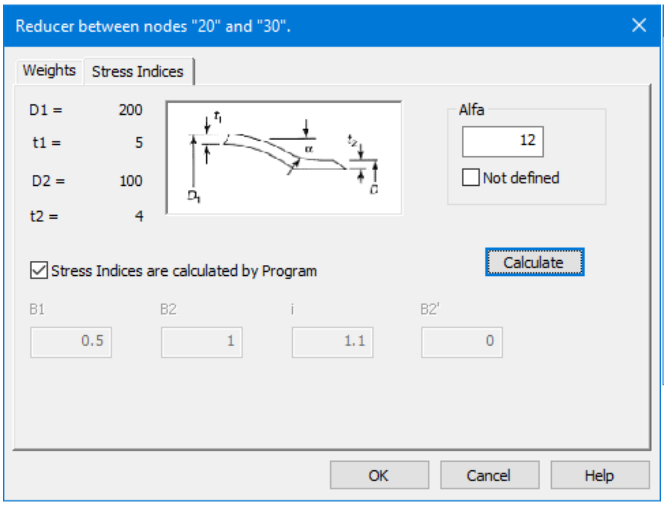
MITER BEND:

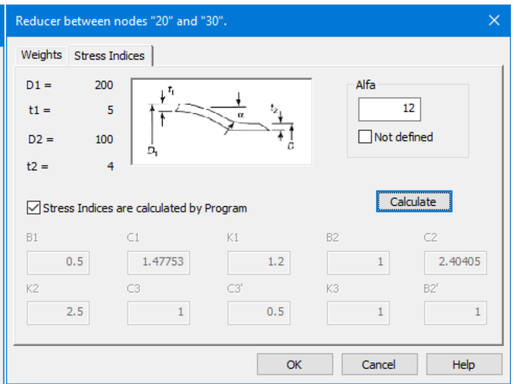
CODE = 'ASME_NC'



REDUCER

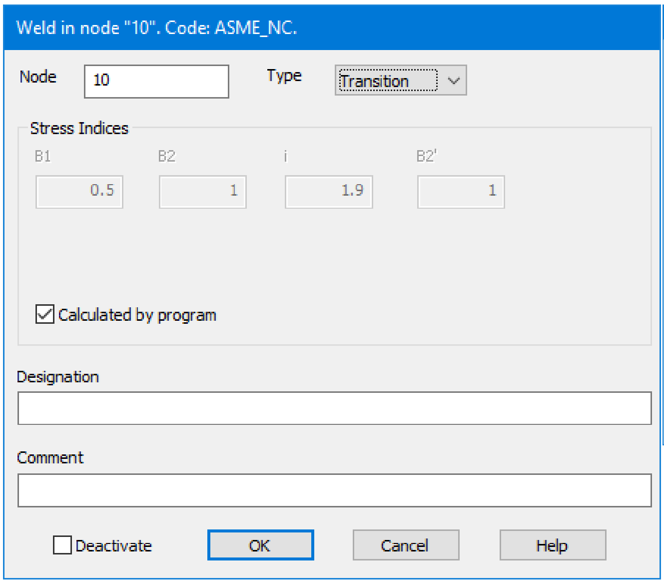
CODE = 'ASME_NC', 'EN' CODE = 'ASME_NB', CLS = 1

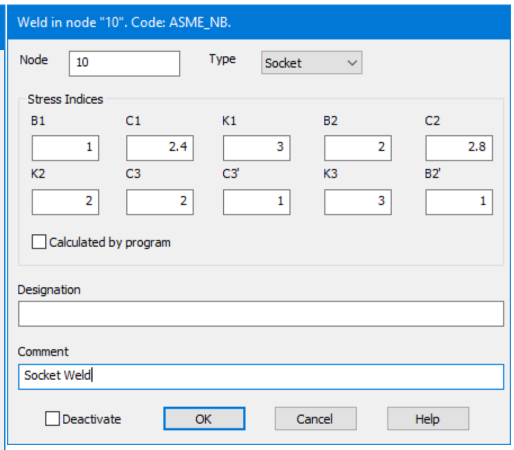




WELD

CODE = 'ASME_NC', 'EN' CODE = 'ASME_NB', CLS = 1

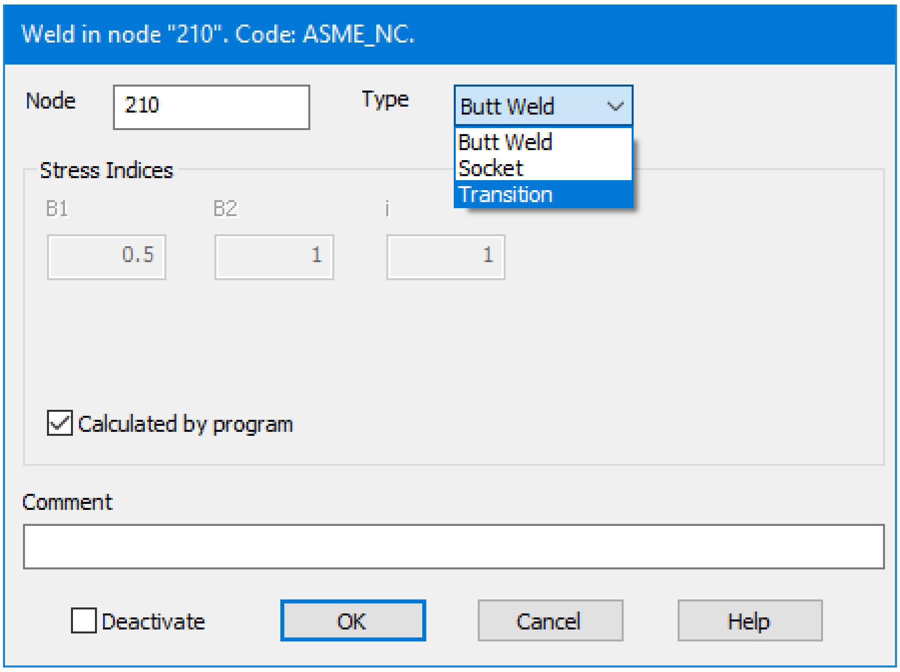




If these data are redefined by USER, they are recorded in dp5 file in the SI array:

CODE	1	2	3	4	5	6	7	8	9	10
ASME_NB	**B1**	**C1**	**K1**	**B2**	**C2**	**K2**	**C3**	**C3'**	**K3**	**B2'**
ASME_NC	**B1**	**B2**	**i**	**B2'**						
EN	**i**	**0.75*i**								
ASME B31.1	**i**	**0.75*i**								

Welds (WLD)



type: local geometrical command

Function: setting stress indexes for circumferential welding, Parameters are set depending on CODE.

Parameters:

FIB strength reduction factor

type:	REAL
unit:	-
default value:	0.9
limitations:	0 < FIB ≤ 1

FWS fatigue strength reduction factor (to be determined in accordance with Item 5.6.12 of Codes [REF 1], it is applicable in case when CODE = 'PNAE' and 'PNAE_T')

type:	REAL
unit:	-
default value:	0.9
limitations:	0 < FWS ≤ 1

TYPE type of material of the piping (to be used for RD and PNAE codes - high temperature piping systems)

type:	TEXT
unit:	-
default value:	'CS'
limitations:	'AUS', 'CMV', 'CS', 'BWELD', 'SOCK', 'TRANS'

When using ASME/EN CODES, an additional fields with stress indexes appears in the dialog

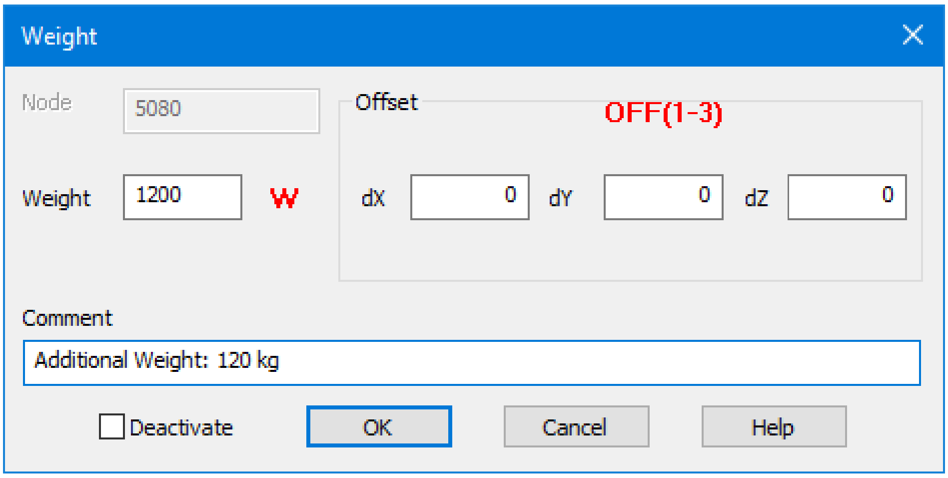
Note:

In evaluating the piping system strength according to RD 10-249-98 codes [REF 2], the values of strength reduction factors for welded joints shall be taken according to Table 4.2 depending upon the TYPE parameter. When TYPE='AUS' the austenite chromium-nickel and high-chromium steels are considered; TYPE='CMV' corresponds to chromium-molybdenum steels; TYPE='CS' – to carbon, manganese and chromium-molybdenum steels. Types 'BWELD' (Girth Butt Weld), 'SOCK' (Socket Weld) and 'TRANS' (Transition) are used in frame of ASME and EN analyses.

Example:

```
2000: WLD FIB 0.9 FWS 1

or

2000: WLD TYPE 'CS'
```

Concentrated weight (CW)



type: local geometrical command

Function: setting the weight in the node

Parameters:

W concentrated weight

type:	REAL
unit:	Newton
default value:	0
limitations:	≥ 0

OFF(3) offset of the weight load (relative coordinates, X, Y, Z, shall be set along the direction of global axes)

type:	REAL
unit:	mm
dimension	array of three numbers
default value:	0, 0, 0
limitations:	-

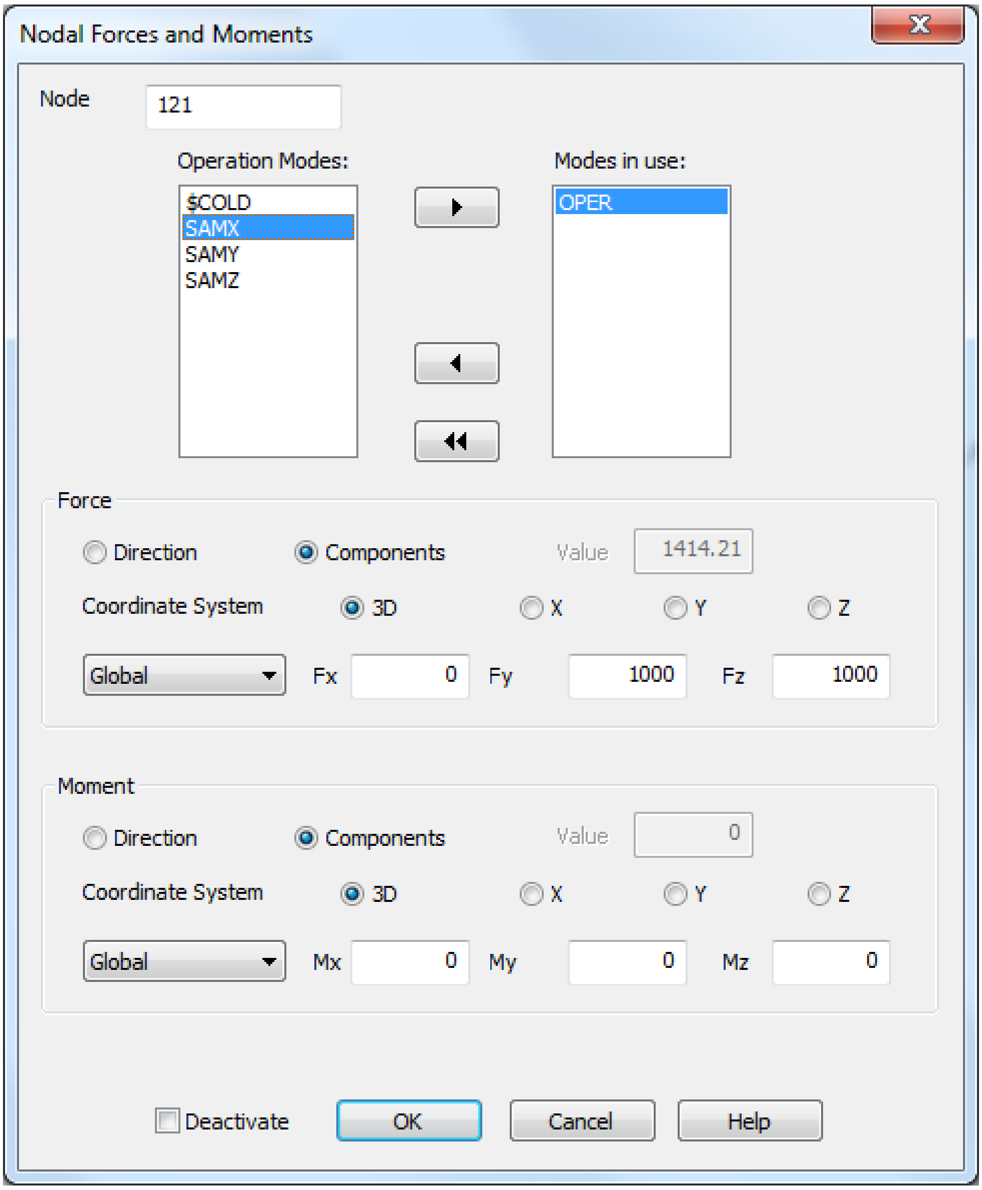
Note:

1) *In dynamic analysis, the concentrated weight is interpreted as a concentrated mass over all translational degrees of freedom. See also Appendix IV.*

Example:

```
2000: CW 1000.
```

Concentrated Loads (FOR)



type: local geometrical multi-line command

Function: command for applying the forces/moments concentrated in the node[1)]

Parameters:

MODE - identification name of the corresponding operating mode

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

FX, FY, FZ Components of forces acting along global axes

type: REAL
unit: N
default value: -
limitations: -

MX, MY, MZ Components of moments acting along global axes

type: REAL
unit: N*mm
default value: -
limitations: -

or:

F (M) force's (moments') magnitude

type: REAL
unit: N (N*mm)
default value: -
limitations: -

+

DC_F (DC_M)$^{2)}$ projections of the line of action of the force (moment) on the global coordinates XYZ, or the corresponding direction cosines.

type: REAL
unit: -
dimension: array of three elements
default value: -
limitations: all three elements of the array cannot simultaneously be equal to zero.

DIRL_F (DIRL_M)$^{2)}$ alternative way for specification of load's directions (in local coordinates)

type: REAL или TEXT
unit: -
dimension: array of three elements
default value: -
limitations: Either 3 numbers are directional cosines, or 'A', 'H', 'N' - directions of local axes

DS_F (DS_M)$^{2)}$ alternative way for specification of load's directions (in spherical coordinates). See command DS

type: REAL
units: -
dimension: array of three elements
default value: -
limitations: both elements of the array cannot simultaneously be equal to zero.

`Example:`

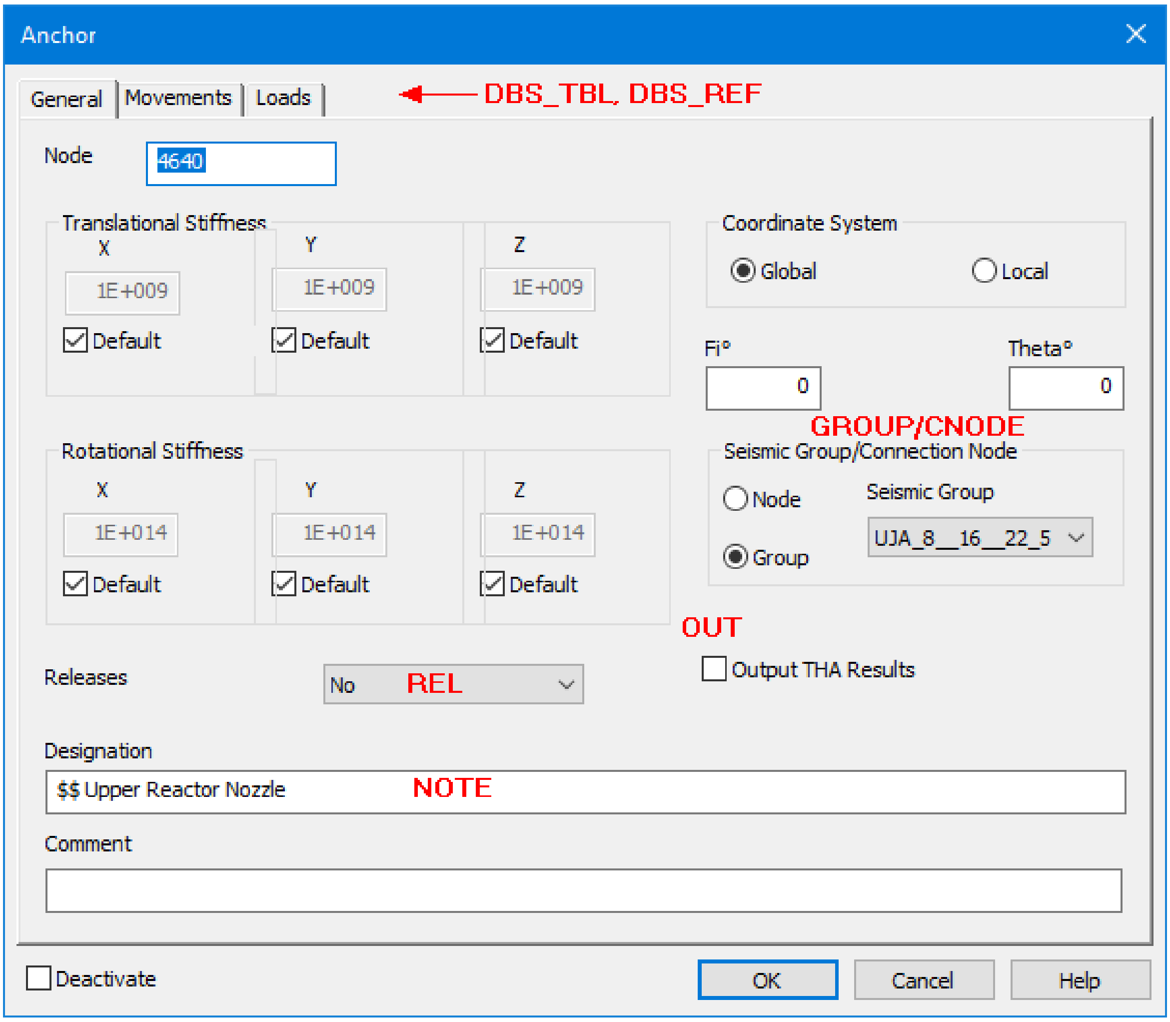
```
DJ10N: FOR
& mode = 'HOT',    fz = 2000
& mode = 'REG_1', fz = 1000
& mode = 'REG_2', fz = 20
```

Note:

*1) To activate concentrated loads within current analysis the corresponding reference shall be specified in the [SOLV](#) command: ... LC MOD='$OPER' ... **[LOAD](#)** = 'W+P+**F**'*

2) Parameters specifying the load's directions: DC_, DIRL_ and DS_, are mutually exclusive. To define the local coordinate system, see [Appendix II](#)

Anchor (ANC)



type: local geometrical multi-line command

Function: model an anchor (fixed support)

Parameters:

STX(STA) anchor stiffness in the direction of global X axis (STX) or local A axis (STA)

type: REAL
unit: N/mm
default value: rigid
limitations: ≥ 0

STY(STH) anchor stiffness in the direction of global Y axis (STY) or local H axis (STH)

type: REAL
unit: N/mm
default value: rigid
limitations: ≥ 0

STZ(STN) anchor stiffness in the direction of global Z axis (STZ) or local N axis (STN)

type: REAL
unit: N/mm
default value: rigid
limitations: ≥ 0

SRX(SRA) angular spring rate of the anchor about the global X axis (SRX) or local A axis (SRA)

type: REAL
unit: N*mm/rad
default value: rigid
limitations: ≥ 0

SRY(SRH) angular spring rate of the anchor about the global Y axis (SRY) or local H axis (SRH)

type: REAL
unit: N*mm/rad
default value: rigid
limitations: ≥ 0

SRZ(SRN) angular spring rate of the anchor about the global Z axis (SRZ) or local N axis (SRN)

type: REAL
unit: N*mm/rad
default value: rigid
limitations: ≥ 0

REL[3)] release degrees of freedom at the determination of loads on hangers

type: TEXT
unit: -
default value: -
limitations: see Note 3

NOTE support ID

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP[4)] name of the seismic group of supports

type: TEXT
unit: -
default value: name of the first group of spectra described by the SPEC command
limitations: name of the group to be selected from the 'GROUP' parameter of the SPEC command

CNODE[4,5)] connecting node

type: TEXT
unit: -
default value: -
limitations: the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT[6)] write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

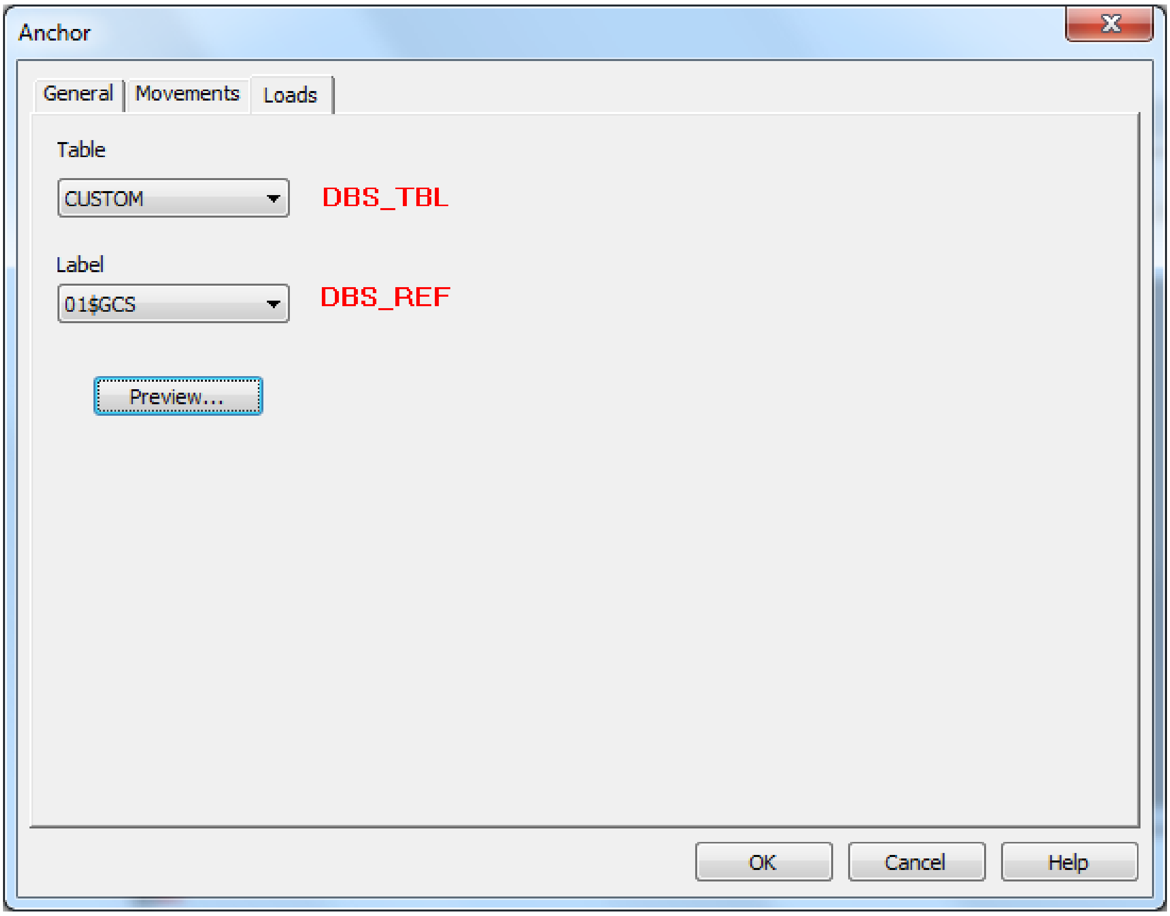
type: TEXT
unit: -
default value: 'NO'
limitations: 'YES' or 'NO'

FI angle of rotation of the anchor global axes about Z axis (similar to the DS(1) angle of the spherical coordinate system)

type: REAL
unit: degrees
default value: 0
limitations: ≥ 0

THETA angle of rotation of the turned X axis of the anchor with respect to the horizontal plane (similar to the DS(2) angle of the spherical coordinate system)

type: REAL
unit: degrees
default value: 0
limitations: ≥ 0



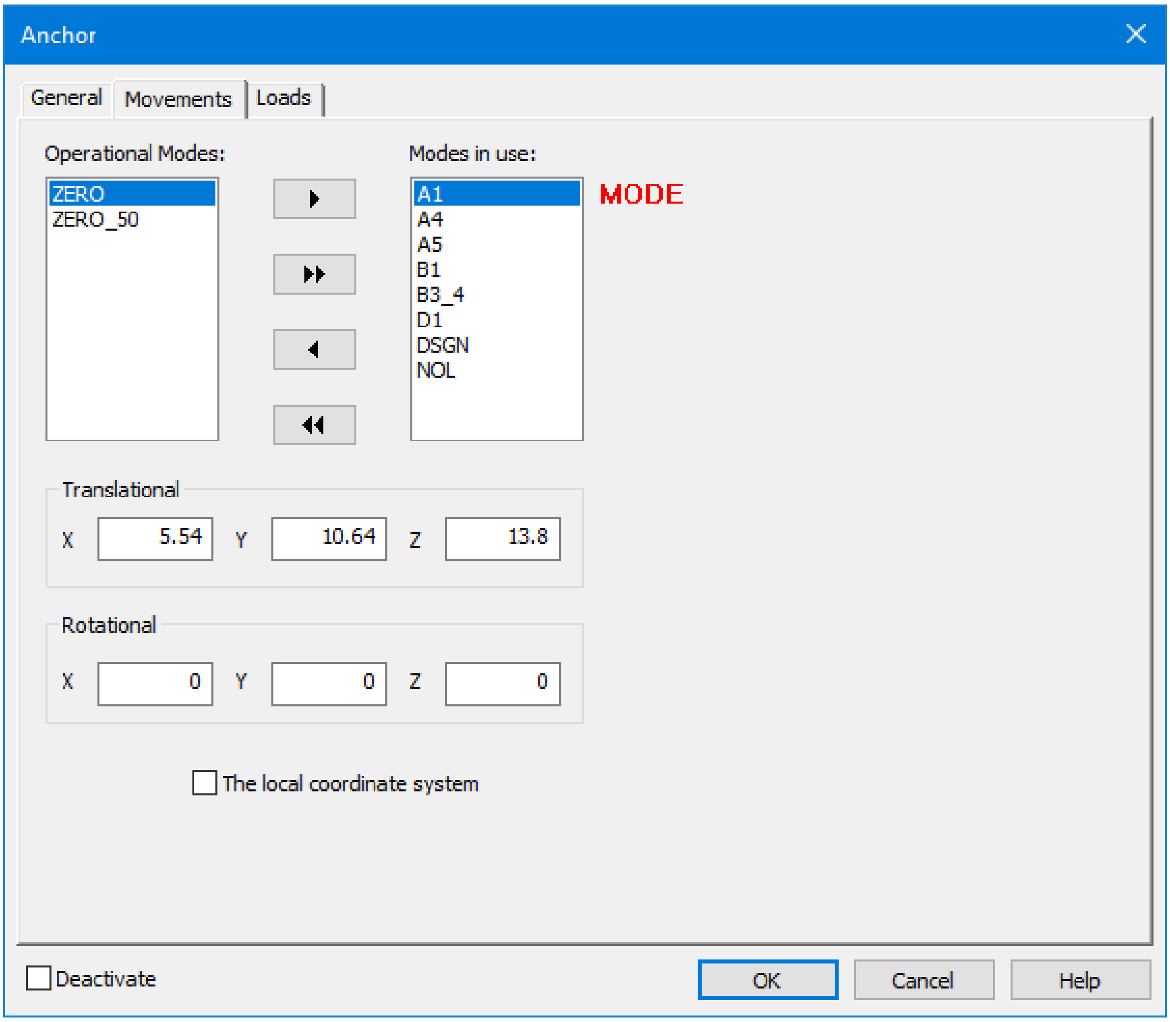
DBS_TBL reference to the table in DB for supports

type:	STRING
unit:	-
default value:	-
limitations:	existing table in DB

DBS_REF reference to the record from the DBS_TBL table

type:	STRING
unit:	-
default value:	-
limitations:	existing mark

Parameter subcommand



MODE - identification name of the piping system operating mode. It is used for setting the pre-defined displacement of anchors

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

Subcommand parameter :

DX anchor displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY anchor displacement along the global Y axis

type:	REAL

unit: mm
default value: 0
limitations: -

DZ anchor displacement along the global Z axis

type: REAL
unit: mm
default value: 0
limitations: -

RX angular displacement of the anchor about the global X axis

type: REAL
unit: rad
default value: 0
limitations: -

RY angular displacement of the anchor about the global Y axis

type: REAL
unit: rad
default value: 0
limitations: -

RZ angular displacement of the anchor about the global Z axis

type: REAL
unit: rad
default value: 0
limitations: -

Note:

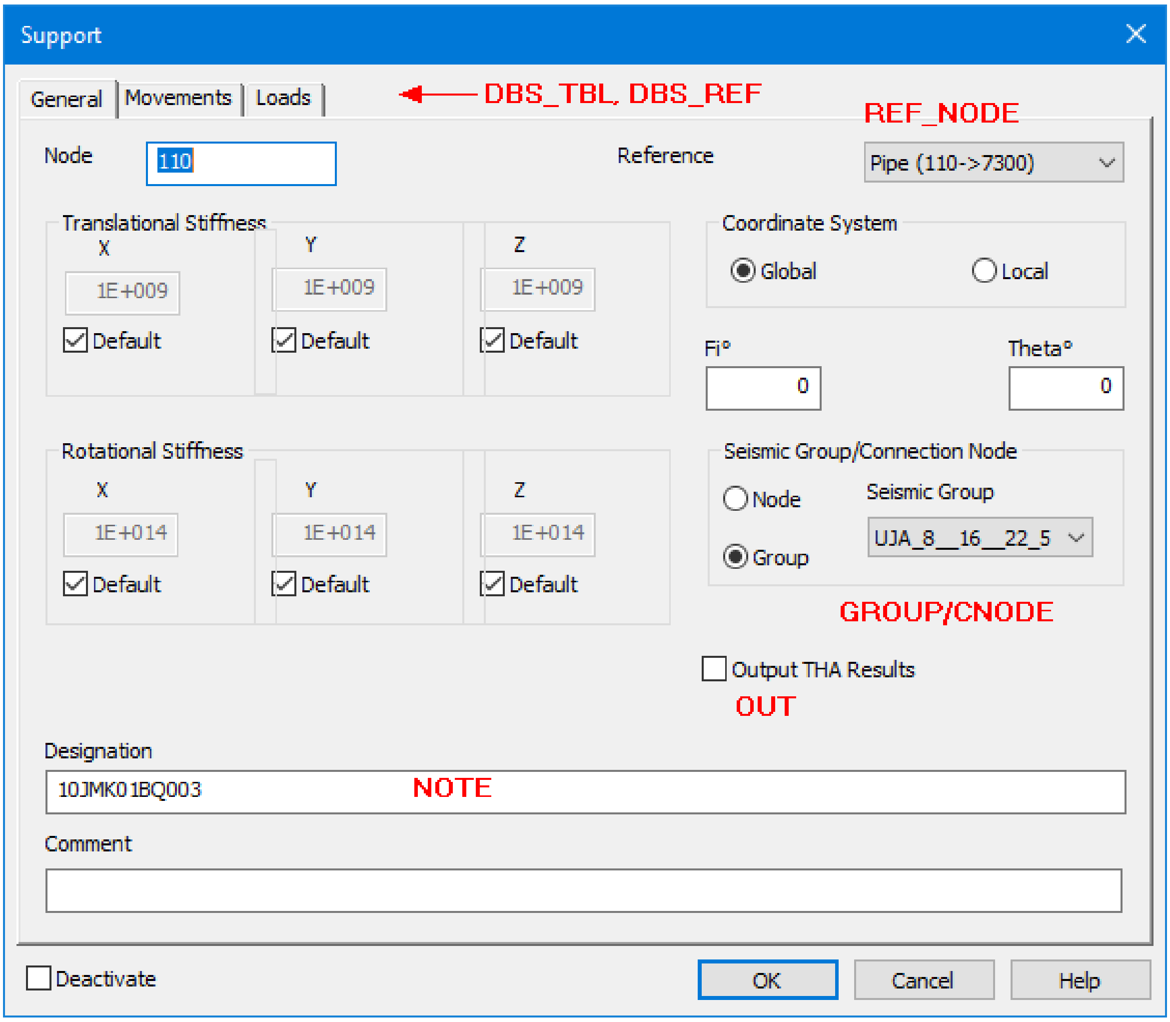
1) *Local coordinate system used for supports is defined according to the rules set forth in Appendix II;*
2) *The ANC command cannot be set in the middle of the span;*
3) *The **REL** parameter is taken into account only in selecting the spring characteristics, namely, at the stage of analysis, for which the T**YPE = 'DSGN'** parameter is set in the SOLV command. The possible variants of values of the REL parameter are as follows: 'V' - release the vertical; 'T' - release all translational degrees of freedom (X, Y, Z/ A, H, N); 'A' - release all;*
4) *The GROUP and CNODE parameters are mutually exclusive;*
5) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no anchor movement could be defined*
6) *With the parameter value of OUT = 'YES', a file with the name of "SUPP_001_100.dat" type will appear in the directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the anchor or FIX support located in the "100" node. The file contains 13 columns of numbers: first column - time, 2nd - 7th columns - components of forces in the anchor/support (Fx, Fy, Fz, Mx, My, Mz); 8th - 13th columns - dynamic deformations of the anchor/support (Dx, Dy, Dz, Rx, Ry, Rz).*

Example:

```
2000: ANC REL 'V'
& mode 'NOL' DX -10. DY -23. DZ 30. RX 3.
```

```
& mode 'ZERO'
& mode 'HYDR' DX -5. DY -13. DZ 20.
```

Support (SUP)



type: local geometrical command

Function: model fixed support, unlike anchor could be placed in the piping mid-span

Parameters:

STX(STA) support stiffness in the direction of the global X axis (STX) or local A axis (STA)

type: REAL
unit: N/mm
default value: rigid
limitations: ≥ 0

STY(STH) support stiffness in the direction of the global Y axis (STY) or local H axis (STH)

type: REAL
unit: N/mm

default value: rigid
limitations: ≥ 0

STZ(STN) support stiffness in the direction of the global Z axis (STZ) or local N axis (STN)

type: REAL
unit: N/mm
default value: rigid
limitations: ≥ 0

SRX(SRA) angular spring rate of the support about the global X axis (SRX) or local A axis (SRA)

type: REAL
unit: N*mm/rad
default value: rigid
limitations: ≥ 0

SRY(SRH) angular spring rate of the support about the global Y axis (SRY) or local H axis (SRH)

type: REAL
unit: N*mm/rad
default value: rigid
limitations: ≥ 0

SRZ(SRN) angular spring rate of the support about the global Z axis (SRZ) or local N axis (SRN)

type: REAL
unit: N*mm/rad
default value: rigid
limitations: ≥ 0

NOTE support ID

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP[2)] name of the seismic group of supports

type: TEXT
unit: -
default value: name of the first group of spectrums described by the SPEC command
limitations: the group name is selected from the 'GROUP' parameter of the SPEC command

CNODE[2,3)] connecting node

type: TEXT
unit: -
default value: -
limitations: the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT[4)] write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type: TEXT
unit: -
default value: 'NO'
limitations: 'YES' or 'NO'

FI angle of rotation of the anchor global axes about Z axis (similar to the DS(1) angle of the spherical coordinate system)

type: REAL
unit: degrees
default value: 0
limitations: ≥ 0

THETA angle of rotation of the turned X axis of the anchor with respect to the horizontal plane (similar to the DS(2) angle of the spherical coordinate system)

type: REAL
unit: degrees
default value: 0
limitations: ≥ 0

DBS_TBL reference to the table in DB for supports

type: STRING
unit: -
default value: -
limitations: existing table in DB

DBS_REF reference to the record from the DBS_TBL table

type: STRING
unit: -
default value: -
limitations: existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DZ support displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

RX angular displacement of the support about the global X axis

type:	REAL
unit:	rad
default value:	0
limitations:	-

RY angular displacement of the support about the global Y axis

type:	REAL
unit:	rad
default value:	0
limitations:	-

RZ angular displacement of the support about the global Z axis

type:	REAL
unit:	rad
default value:	0
limitations:	-

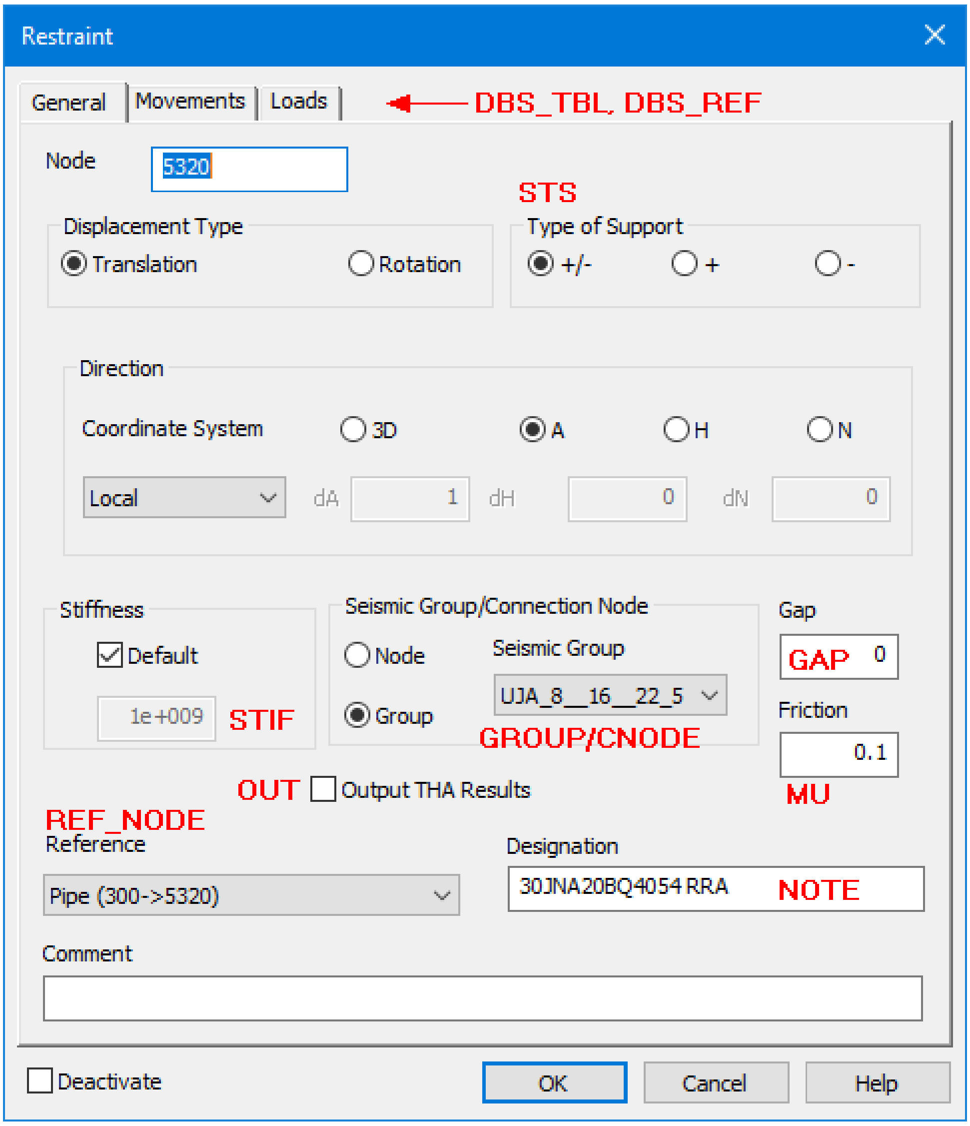
Note:

1) *Local coordinate system used for supports is defined according to the rules set forth in [Appendix II](#);*
2) *The GROUP and CNODE parameters are mutually exclusive;*
3) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no support's movement could be defined*
4) *With the parameter value of OUT = 'YES', a file with the name of "SUPP_001_100.dat" type will appear in the directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the anchor or FIX support located in the "100" node. The file contains 13 columns of numbers: first column - time, 2nd - 7th columns - components of forces in the anchor/support (Fx, Fy, Fz, Mx, My, Mz); 8th - 13th columns - dynamic deformations of the anchor/support (Dx, Dy, Dz, Rx, Ry, Rz).*

Example:

```
2000: SUP SRA 0
```

Translational Restraint (STS)



type: local geometrical multi-line command

Function: restraint acting in both directions along the line of action (STS);[1)]

Parameters:

STIF restraint's stiffness

type:	REAL
unit:	N/mm
default value:	rigid
limitations:	≥ 0

DC or DS: direction of the action of restraint; see "Local commands" section for the description of parameters

GAP[2)] gap

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

MU friction coefficient

type:	REAL
unit:	-
default value:	0
limitations:	0 ≤ MU ≤ 1

NOTE Note/Restraint ID

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP[3)] name of the seismic group of supports

type:	TEXT
unit:	-
default value:	name of the first group of spectrums described by the SPEC command
limitations:	name of the group to be selected from the 'GROUP' parameter of the SPEC command

CNODE[3,4)] connecting node

type:	TEXT
unit:	-
default value:	-
limitations:	the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT[5)] write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type: TEXT
unit: -
default value: 'NO'
limitations: 'YES' or 'NO'

DBS_TBL reference to the table in DB for supports

type: STRING
unit: -
default value: -
limitations: existing table in DB

DBS_REF reference to the record from the DBS_TBL table

type: STRING
unit: -
default value: -
limitations: existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the restraint

Parameters:

MODE identification name of the piping system operating mode

type: TEXT
unit: -
default value: -
limitations: see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX restraint's displacement along the global X axis

type: REAL
unit: mm
default value: 0
limitations: -

DY restraint's displacement along the global Y axis

type: REAL
unit: mm
default value: 0
limitations: -

DZ restraint's displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

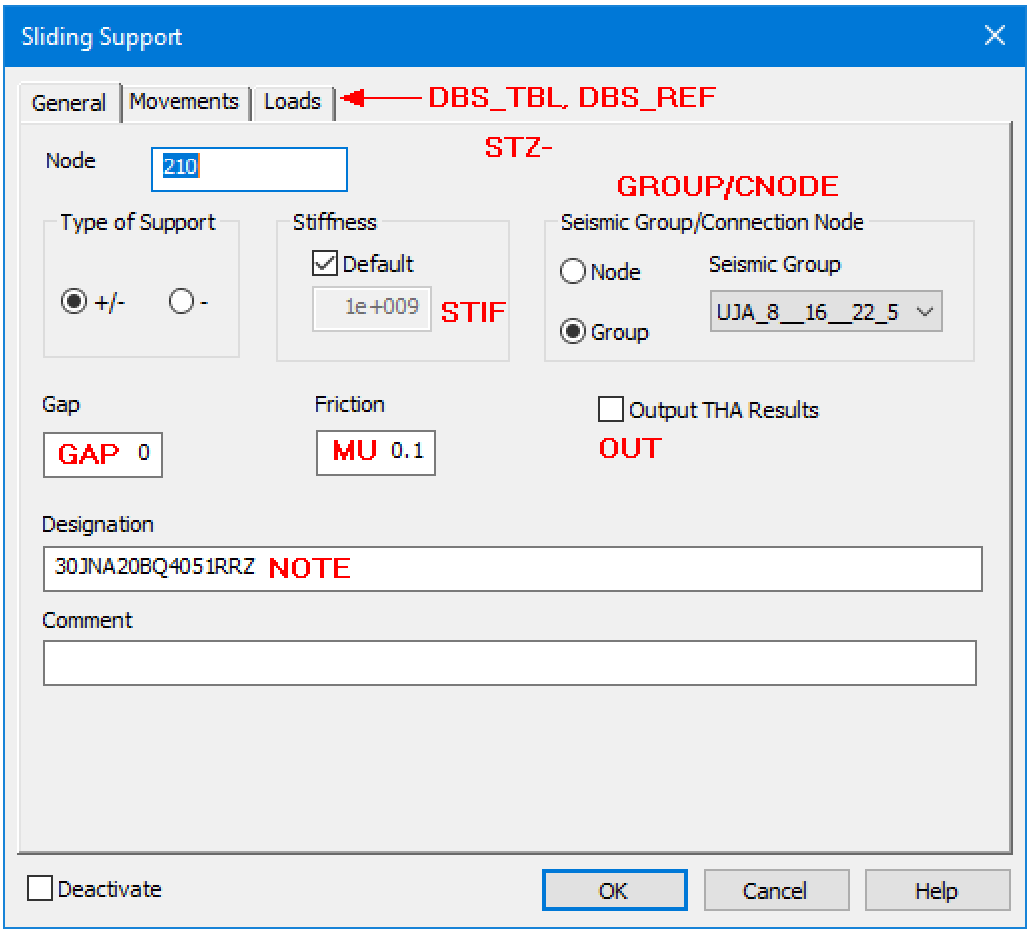
Note:

1) *Short variants of the designation of this command are possible when the direction of action of the restraint coincides with the global coordinates (STX, STY, STZ* commands*) or with the local coordinates of the support (STA, STH, STN). In case when a short designation of commands is used, the parameters indicating the direction (DC or DS) are nor used;*
2) *The gap is symmetric. The gap value is set from the piping centerline "zero" (unloaded) state;*
3) *The GROUP and CNODE parameters are mutually exclusive;*
4) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined*
5) *With the parameter value of OUT = 'YES', a file with the name of "RSTR_001_100.dat" type will appear in the directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the support (Fx, Fy, Fz); 5th - 7th columns - dynamic deformations of the support (Dx, Dy, Dz).*

Example :

```
2000: STS 1.e4 DC 1 1 0 GAP 10 MU 0.3 NOTE "12RABQ001"
```

One-way restraint (STS+, STS- commands)



type: local geometrical multi-line command

Function: restraint acting in one directions along the line of action;

Parameters:

STIF restraint's stiffness

type: REAL
unit: N/mm
default value: rigid
limitations: ≥ 0

DC or DS: direction of the action of support; see "Local commands" section for the description of parameters

GAP2) gap

type: REAL
unit: mm
default value: 0
limitations: ≥ 0

MU friction coefficient

	type:	REAL
	unit:	-
	default value:	0
	limitations:	0 ≤ MU ≤ 1
NOTE	note (Support ID)	
	type:	STRING
	unit:	-
	default value:	blank
	limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters
GROUP[4)]	name of the seismic group of supports	
	type:	TEXT
	unit:	-
	default value:	name of the first group of spectrums described by the SPEC command
	limitations:	name of the group to be selected from the 'GROUP' parameter of the SPEC command
CNODE[4,5)]	connecting node	
	type:	TEXT
	unit:	-
	default value:	-
	limitations:	the parameter shall refer to the label of one of the existing nodes of the analytical model
OUT[6)]	write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')	
	type:	TEXT
	unit:	-
	default value:	'NO'
	limitations:	'YES' or 'NO'
DBS_TBL	reference to the table in DB for supports	
	type:	STRING
	unit:	-
	default value:	-
	limitations:	existing table in DB
DBS_REF	reference to the record from the DBS_TBL table	
	type:	STRING
	unit:	-
	default value:	-
	limitations:	existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the restraint

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DZ support displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

Note:

1) *"+" or "-" sign indicates the direction, in which the restraint is acting. Short designations of this command are possible (without indication of the direction by means of the DIR command): in the global coordinate system: STX+/STX-, STY+/STY-, STZ+/STZ- . In case when the local coordinate system is used, the following commands are applied: STA+/STA-; STH+/STH-; STN+/STN-.*
2) *For one-way restraints the gap is considered in the direction of action of the support; in the opposite direction the gap is considered unlimited. The gap value is set from the "zero" (unloaded) state.*
3) *The supports of "STZ-" and "STN-" type (the latter - in case of location on the horizontal segment of the piping system with a slope of not more than10°) are considered by the program as those carrying the weight load. In case of their uplift for a value exceeding the critical one (see the LIFT parameter) at the Load Cases of "OPER_A" or "OPER_B" type (see the SOLV command, TYPE parameter); at the subsequent stage of analysis of "SUST_C" type, these supports are not taken into account.*
4) *The GROUP and CNODE parameters are mutually exclusive.*
5) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined*

6) *With the parameter value of OUT = 'YES', a file with the name of "RSTR_001_100.dat" type will appear in the directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the support (Fx, Fy, Fz); 5th - 7th columns - dynamic deformations of the support (Dx, Dy, Dz).*

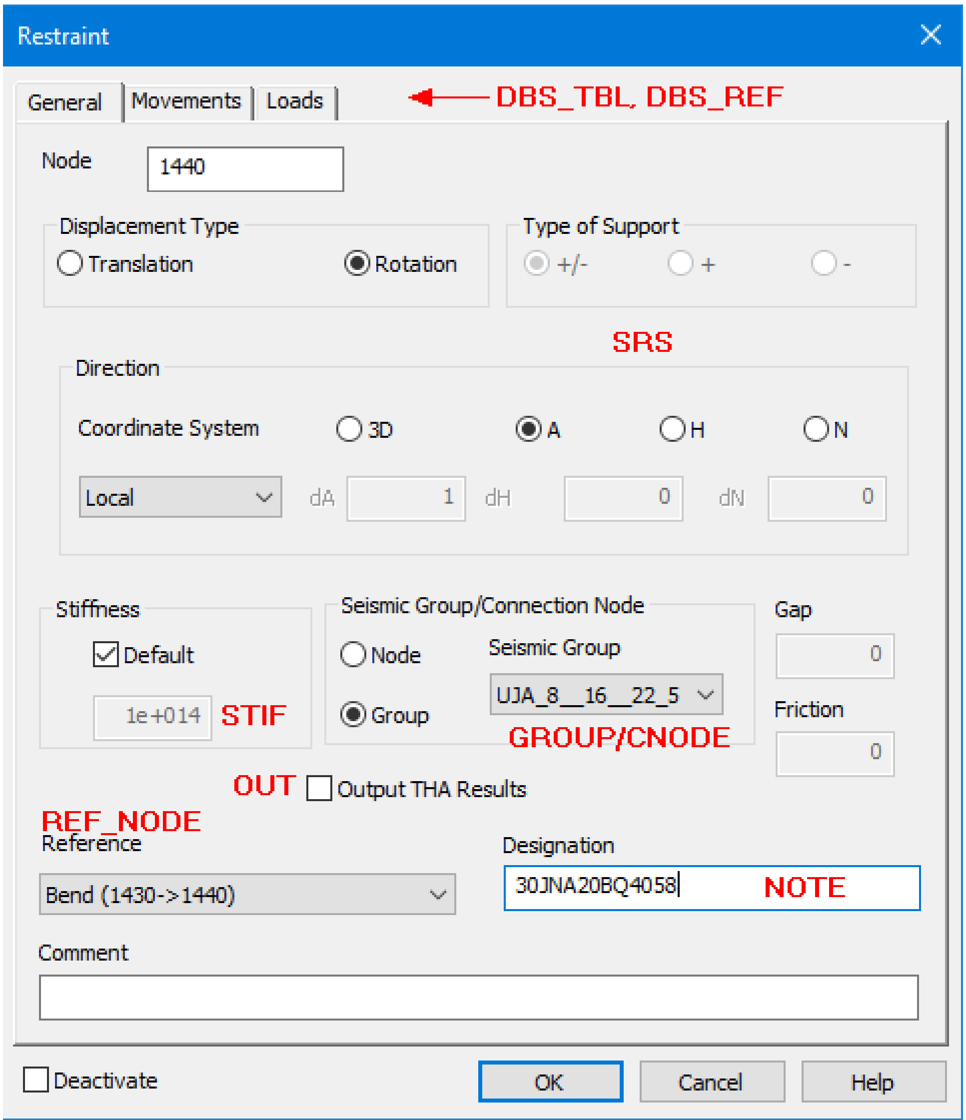
Example :

```
2000: STS+ 1.e4 DC 1 1 0 GAP 10 MU 0.3 NOTE "12RABQ001"
```

or :

```
2000: STX+ 1.e4 GAP 10 MU 0.3 NOTE "12RABQ001"
```

Skewed Restraint (SRS)



type: local geometrical multi-line command

Function: restraint limiting the angular movements;

Parameters:

STIF

angular spring rate of the restraint about its line of action

type: REAL
unit: N*mm/rad
default value: rigid
limitations: ≥ 0

DC or DS:

direction of the action of support; see "Local commands" section for the description of parameters

NOTE

support ID

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP2)

name of the seismic group of supports

type: TEXT
unit: -
default value: name of the first group of spectrums described by the SPEC command
limitations: name of the group to be selected from the 'GROUP' parameter of the SPEC command

CNODE2,3)

connecting node

type: TEXT
unit: -
default value: -
limitations: the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT4)

write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type: TEXT
unit: -
default value: 'NO'
limitations: 'YES' or 'NO'

DBS_TBL

reference to the table in DB for supports

type: STRING
unit: -
default value: -

limitations: existing table in DB

DBS_REF reference to the record from the DBS_TBL table

type:	STRING
unit:	-
default value:	-
limitations:	existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

RX angular displacement of the support about the global X axis

type:	REAL
unit:	rad
default value:	0
limitations:	-

RY angular displacement of the support about the global Y axis

type:	REAL
unit:	rad
default value:	0
limitations:	-

RZ angular displacement of the support about the global Z axis

type:	REAL
unit:	rad
default value:	0
limitations:	-

Note:

1) *Short variants of the designation of this command are possible when the direction of action of the support coincides with the global coordinates (STX, STY, STZ* commands*) or with the local coordinates of the support (STA, STH, STN). In case when a short designation of commands is used, the parameters indicating the direction (DC or DS) are nor used;*
2) *The GROUP and CNODE parameters are mutually exclusive;*
3) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined;*

4) *With the parameter value of OUT = 'YES', a file with the name of "RSTR_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the support (Mx, My, Mz); 5th - 7th columns - dynamic deformations of the support (Rx, Ry, Rz).*

Example :

```
2000: SRS 1.e4 DC 1 1 0 NOTE "12RABQ001"
```

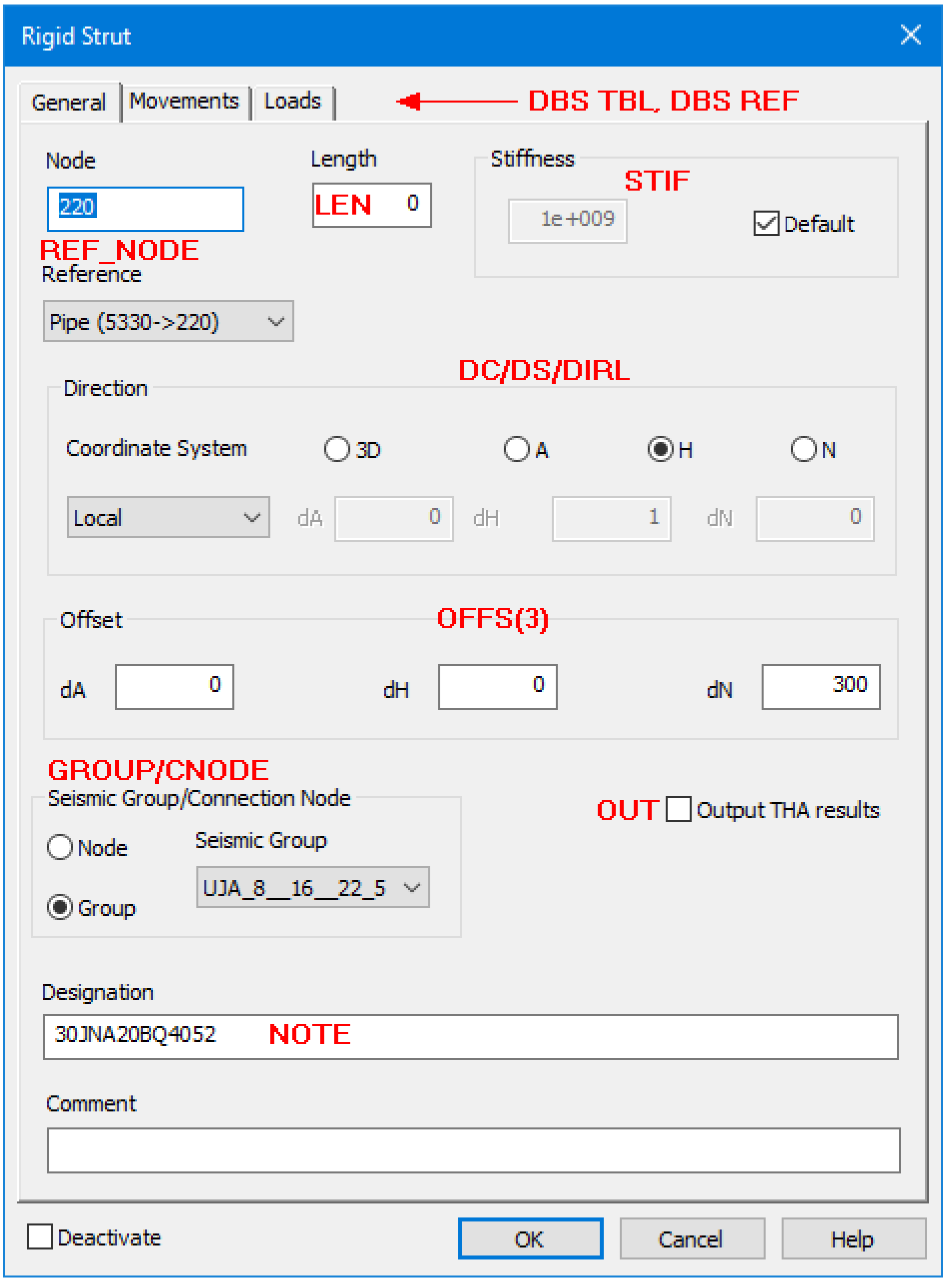
or

```
2000: SRN
```

or

```
2000: SRZ
```

Rigid Strut (STRT)



type: local geometrical multi-line command

Function: modeling a restraint forming a rigid connection in axial direction between piping and structure. Ability to model the geometric nonlinearity due to short length of restraint.

Parameters:

LEN length of the rod

type:	REAL
unit:	mm
default value:	0

	limitations:	≥ 0
STIF	stiffness of the restraint	
	type:	REAL
	unit:	N/mm
	default value:	rigid
	limitations:	≥ 0

DC/DS/DIRL(3) direction of the action of support in Cartesian/spherical or local coordinates; see "Local commands" section for the description of parameters

OFFS(3) offsets of the point of restraint attachment to the pipe. Set in the local coordinates of pipe

type:	REAL(3)
unit:	mm
dimension	array of three numbers
default value:	0, 0, 0
limitations:	-

NOTE support ID

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP[1)] name of the seismic group of supports

type:	TEXT
unit:	-
default value:	name of the first group of spectrums described by the SPEC command
limitations:	name of the group to be selected from the 'GROUP' parameter of the SPEC command

CNODE[1,2)] connecting node

type:	TEXT
unit:	-
default value:	-
limitations:	the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT[3)] write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type:	TEXT
unit:	-
default value:	'NO'

	limitations:	'YES' or 'NO'
DBS_TBL	reference to the table in DB for supports	
	type:	STRING
	unit:	-
	default value:	-
	limitations:	existing table in DB
DBS_REF	reference to the record from the DBS_TBL table	
	type:	STRING
	unit:	-
	default value:	-
	limitations:	existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DZ support displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

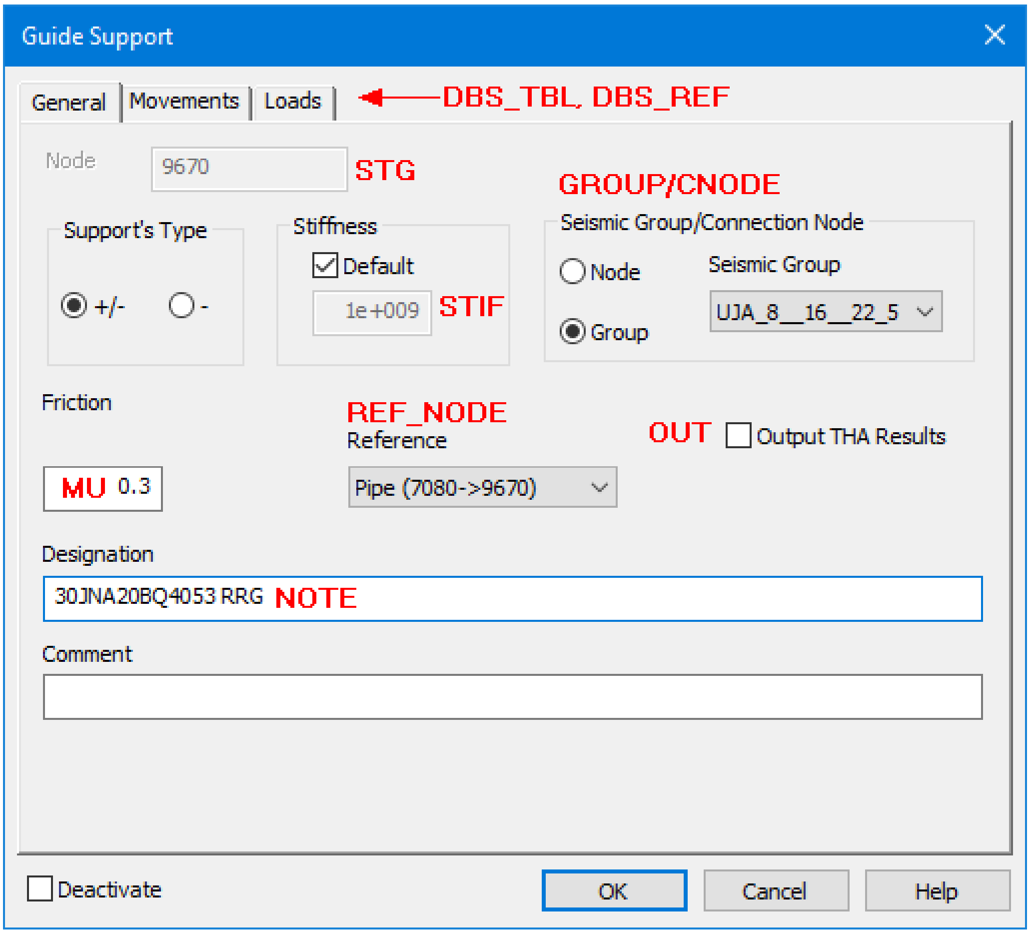
Note:

1) The GROUP and CNODE parameters are mutually exclusive;
2) CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined;
3) With the parameter value of OUT = 'YES', a file with the name of "STRT_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the support (Mx, My, Mz); 5th - 7th columns - dynamic deformations of the support (Rx, Ry, Rz).

Example:

```
  50: STRT LEN = 1000, DC = 0, 0, 1 OFFS = 0, 123, 0 DBS_TBL = "LISEGA 2010RS", DBS
& MODE = 'NOL', DZ = 300
```

Guide Support (STG, STG-)



type: local geometrical multi-line command

Function: support restraining lateral movements of pipe.

Parameters:

STIF support's stiffness

	type:	REAL
	unit:	N/mm
	default value:	rigid
	limitations:	≥ 0

MU friction coefficient

type:	REAL
unit:	-
default value:	0
limitations:	0 ≤ MU ≤ 1

NOTE Note / Comment

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP3) name of the seismic group of supports

type:	TEXT
unit:	-
default value:	name of the first group of spectrums described by the SPEC command
limitations:	name of the group to be selected from the 'GROUP' parameter of the SPEC command

CNODE3,4) connecting node

type:	TEXT
unit:	-
default value:	-
limitations:	the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT5) write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES' or 'NO'

DBS_TBL reference to the table in DB for supports

type:	STRING
unit:	-
default value:	-
limitations:	existing table in DB

DBS_REF reference to the record from the DBS_TBL table

type:	STRING

unit: -
default value: -
limitations: existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type: TEXT
unit: -
default value: -
limitations: see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type: REAL
unit: mm
default value: 0
limitations: -

DY support displacement along the global Y axis

type: REAL
unit: mm
default value: 0
limitations: -

DZ support displacement along the global Z axis

type: REAL
unit: mm
default value: 0
limitations: -

Note:

1) *The STG- command is not used at the vertical segment. At the piping segment with the direction different from the vertical, it is possible to use both forms of the STG command, e.g. "STG- mu" command is equivalent to the "STH mu" + "STN- mu" commands;*
2) *The support of "STG-" type located at the horizontal segment of the piping with a slope of not more than 10° is considered by the program as a support carrying the weight load. In case of its uplift for a value exceeding the critical one (see the LIFT parameter) at the analysis stages of "OPER_A" or "OPER_B" type (see the SOLV command, TYPE parameter), at the subsequent stage of analysis of "SUST_C" type, this support will not be taken into account;*
3) *The GROUP and CNODE parameters are mutually exclusive;*

4) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined;*
5) *With the parameter value of OUT = 'YES', a file with the name of "RSTR_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the support (Fx, Fy, Fz); 5th - 7th columns - dynamic deformations of the support (Dx, Dy, Dz).*

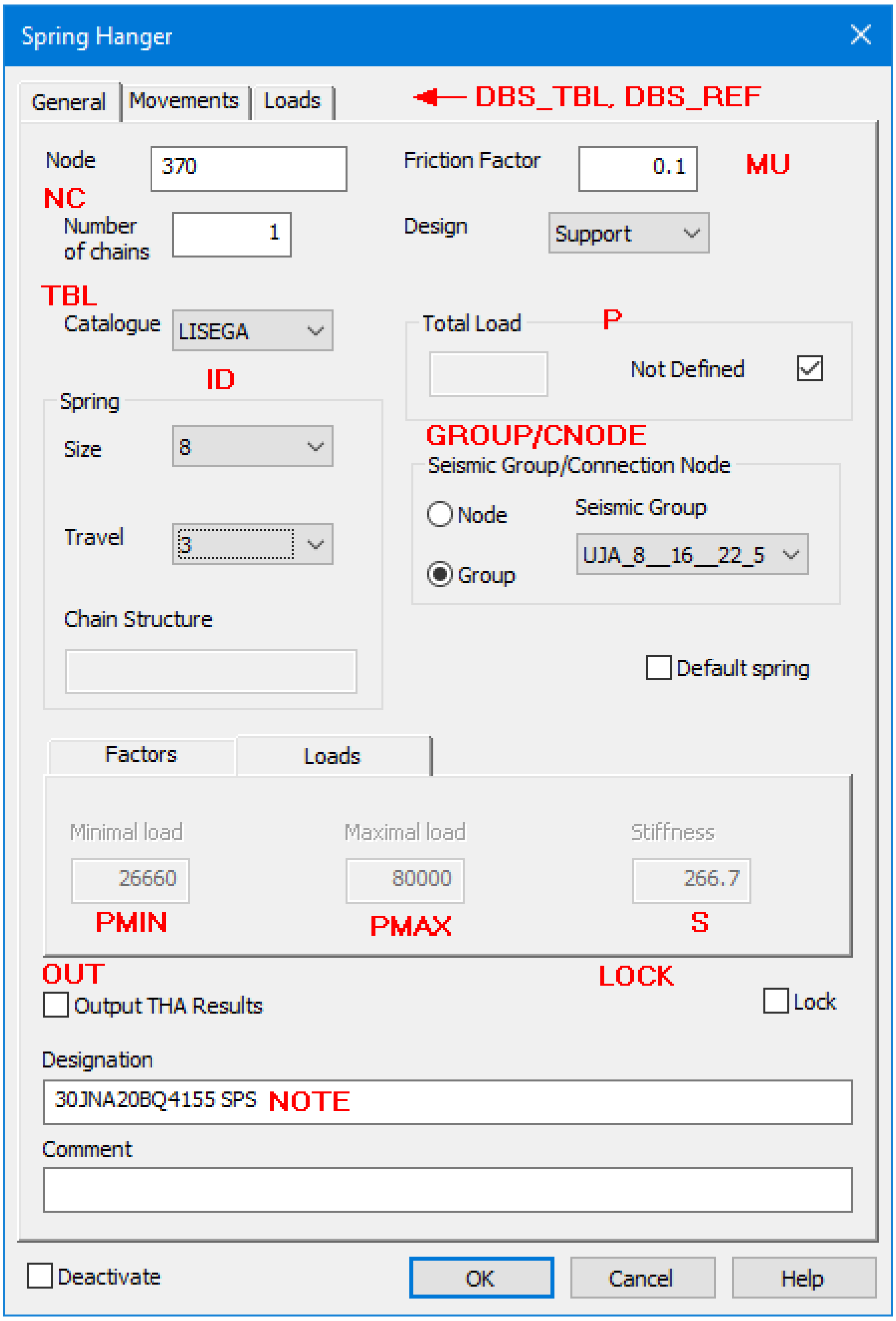
Example:

```
2000: STG MU 0.3
```

or:

```
2000: STG-
```

Spring Hanger/Support (SPR)



type: local geometrical command

Function: simulating the spring hanger or spring support;

Parameters:

NC1) number of hangers/supports at the location

type:	INTEGER
unit:	-
default value:	1
limitations:	$\|NC\| \geq 1$

P total hanger/support load

type: REAL
unit: N
default value: -
limitations: ≥ 0

S spring rate (per one hanger/support)

type: REAL
unit: N/mm
default value: -
limitations: ≥ 0 (except for LC TYPE = 'OPER_R')

TBL - name of the spring Table

type: TEXT
unit: -
default value: to be determined in accordance with the parameters given in the database or indicated in the SDEF command
limitations: to be determined by the spring Table used (SH.DBS file)

ID2) spring identifier

type: TEXT
unit: -
default value: -
limitations: to be determined by the spring Table used (SH.DBS file)

PMAX maximum load per hanger/support

type: REAL
unit: N
default value: 0
limitations: ≥ 0

PMIN minimum load per hanger/support

type: REAL
unit: N
default value: 0
limitations: ≥ 0

LEN length of the hanger

type: REAL
unit: mm
default value: 0
limitations: ≥ 0

PVAR load variation factor

type: REAL

unit: -
default value: 0.35
limitations: 0 ≤ PVAR ≤ 1

PFAC load capacity factor

type: REAL
unit: -
default value: 1.3
limitations: PFAC ≥ 1

ZMAX maximum structure of the chain

type: INTEGER
unit: -
default value: to be determined in accordance with the parameters given in the database or indicated in the SDEF command
limitations: ZMAX ≥ 1

ZMIN minimum structure of the chain

type: INTEGER
unit: -
default value: to be determined in accordance with the parameters given in the database or indicated in the SDEF command
limitations: ZMIN ≥ 1

NOTE Note / Support ID

type: STRING
unit: -
default value: blank
limitations: see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP[4)] name of the seismic group of supports

type: TEXT
unit: -
default value: name of the first group of spectrums described by the SPEC command
limitations: name of the group to be selected from the 'GROUP' parameter of the SPEC command

CNODE[4,5)] connecting node

type: TEXT
unit: -
default value: -
limitations: the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT[6)] write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES' or 'NO'

LOCK lock spring during HT (Hydraulic Test during piping operational life, Load Case of TEST_B type)

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES' or 'NO'

DBS_TBL reference to the table in DB for supports

type:	STRING
unit:	-
default value:	-
limitations:	existing table in DB

DBS_REF reference to the record from the DBS_TBL table

type:	STRING
unit:	-
default value:	-
limitations:	existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DZ support displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

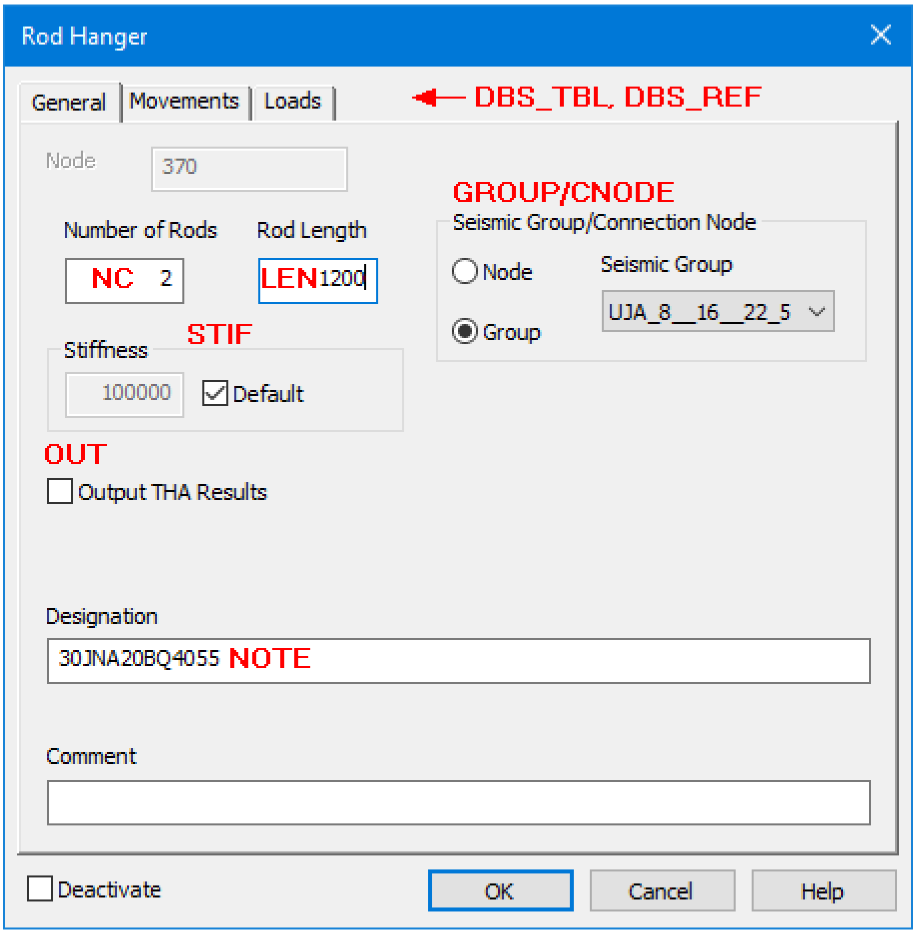
Note:

1) *At NC ≥ 1 the support will be displayed as a spring hanger; at NC ≤ -1 - as a spring support;*
2) *The spring identifier consists of 3 fields: 'size/travel/type' where the spring size corresponds to the maximum load on the spring, "travel" corresponds to the maximum possible working travel of the spring, hanger type is used only in those catalogues, in which same springs could be used in various design versions of variable spring supports (for example, LISEGA catalogue);*
3) *See also Appendix VI with comments on selecting springs for spring hangers/supports;*
4) *The GROUP and CNODE parameters are mutually exclusive;*
5) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined;*
6) *With the parameter value of OUT = 'YES', a file with the name of "SPRH_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the support (Fx, Fy, Fz); 5th - 7th columns - dynamic deformations of the support (Dx, Dy, Dz).*

Example :

```
2000: SPR NC 2 LEN 1000. NOTE "12RABQ003"
```

Rod Hanger (ROD)



type: local geometrical command

Function: modeling of the rigid rod hanger;

Parameters:

NC number of rods

type:	≥NT
unit:	-
default value:	1
limitations:	≥ 1

LEN length of hanger

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

STIF rod's stiffness

type:	REAL

	unit:	N/mm
	default value:	RH_STF
	limitations:	≥ 0

NOTE support ID

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

GROUP2) name of the seismic group of supports

type:	TEXT
unit:	-
default value:	name of the first group of spectrums described by the SPEC command
limitations:	name of the group to be selected from the 'GROUP' parameter of the SPEC command

CNODE2,3) connecting node

type:	TEXT
unit:	-
default value:	-
limitations:	the parameter shall refer to the label of one of the existing nodes of the analytical model

OUT4) write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES' or 'NO'

DBS_TBL reference to the table in DB for supports

type:	STRING
unit:	-
default value:	-
limitations:	existing table in DB

DBS_REF reference to the record from the DBS_TBL table

type:	STRING
unit:	-
default value:	-
limitations:	existing mark

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DZ support displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

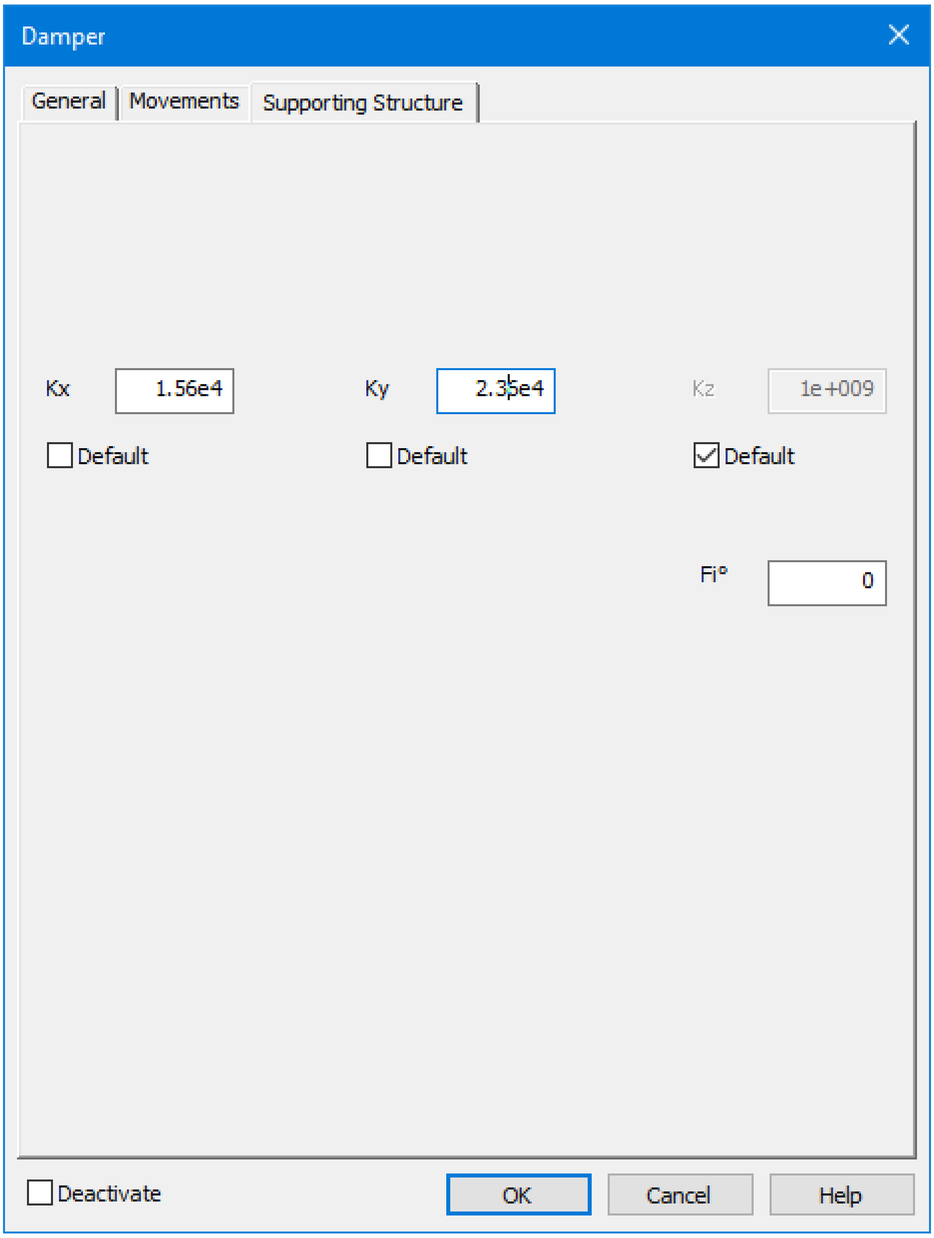
Note:

1) *The "rod hanger" element is a one-way link limiting movements of the piping system vertically downwards.*
2) *The GROUP and CNODE parameters are mutually exclusive;*
3) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined;*
4) *With the parameter value of OUT = 'YES', a file with the name of "SPRH_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the support (Fx, Fy, Fz); 5th - 7th columns - dynamic deformations of the support (Dx, Dy, Dz).*

Example :

```
2000: ROD NC 2 LEN 1000. NOTE "12RABQ005"
```

Viscous Damper (DMP)



type: local geometrical command

Function: modeling of the viscous damper (viscoelastic damper or High Viscous Damper);

Parameters:

NAME damper name

type:	TEXT
unit:	-
default value:	blank
limitations:	(1)

TYPE damper model

type:	TEXT
unit:	-
default value:	'VED'
limitations:	'VED', 'VIS', 'MXW'

TD2) temperature of the damper's working fluid

type:	REAL
unit:	°C
default value:	20
limitations:	from -10°C to +100°C , (2)

LH nominal load in the horizontal direction

type:	REAL
unit:	Newton
default value:	0
limitations:	≥ 0

LV nominal load in the vertical direction

type:	REAL
unit:	N
default value:	0
limitations:	≥ 0

DH permissible movements in the horizontal direction

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

DV permissible movements in the vertical direction

type:	REAL
unit:	mm
default value:	0
limitations:	≥ 0

FH characteristic frequency in the horizontal direction

type:	REAL
unit:	Hz
default value:	0
limitations:	≥ 0

FV characteristic frequency in the horizontal direction

type: REAL
unit: Hz
default value: 0
limitations: ≥ 0

CH stiffness in the horizontal direction

type: REAL
unit: N/mm
default value: 0
limitations: ≥ 0

CV stiffness in the vertical direction

type: REAL
unit: N/mm
default value: 0
limitations: ≥ 0

VH viscosity in the horizontal direction

type: REAL
unit: N*s/mm
default value: 0
limitations: ≥ 0

VV viscosity in the vertical direction

type: REAL
unit: N*s/mm
default value: 0
limitations: ≥ 0

K1_H, C1_H, K2_H, C2_H coefficients of the 4-parameter Maxwell model of the damper for the horizontal direction

type: REAL
unit: (K1_H, K2_H: N/mm, C1_H, C2_H: rad/s)
default value: 0
limitations: ≥ 0

K1_V, C1_V, K2_V, C2_V coefficients of the 4-parameter Maxwell model of the damper for the vertical direction.

type: REAL
unit: (K1_V, K2_V: N/mm, C1_V, C2_V: rad/s)

	default value:	0
	limitations:	≥ 0

GROUP name of the seismic group of supports

type:	TEXT
unit:	-
default value:	name of the first group of spectrums described by the SPEC command
limitations:	name of the group to be selected from the 'GROUP' parameter of the SPEC command

OUT3) write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES' or 'NO'

NOTE4) support ID

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

CNODE4,5) connecting node

type:	TEXT
unit:	-
default value:	-
limitations:	the parameter shall refer to the label of one of the existing nodes of the analytical model

KX, KY, KZ stiffnesses of the civil supporting structures

type:	REAL
unit:	N/mm
default value:	rigid
limitations:	≥ 0

FI angle of rotation of the global axes of the supporting structure about Z axis (similar to DS(1) angle for spherical coordinate system)

type:	REAL
unit:	degrees
default value:	0
limitations:	≥ 0

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DZ support displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

Note:

1) *Dampers are active only within dynamic analysis. The program provides 2 types of damper's characteristics: in explicit and implicit forms (see the table below). In case when the damper is set in the implicit form, the program will automatically determine the damper support characteristics using the data containing in the 'DMP.DBS' file. In doing so, the damper name (NAME parameter) shall exactly correspond to one of the standard names given in this file. The nomenclature and parameters of the dampers of VD type are given in Appendix III .*

Different variants for modeling the viscous damper

Model	Parameters used		Type of Analysis
	explicit mode	implicit mode	
elastic (TYPE = 'VED')	TYPE, LH, LV, DH, DV, CH, CV, GROUP, NOTE	NAME, TYPE, TD, FH, FV, GROUP, NOTE	RSM, THA
ideal viscous (TYPE = 'VIS')	TYPE, LH, LV, DH, DV,	NAME, TYPE, TD, FH, FV, NOTE	THA

	VH, VV, GROUP, NOTE		
Maxwell (TYPE = 'MXW')	TYPE, LH, LV, DH, DV, K1_H, C1_H, K2_H, C2_H, K1_V, C1_V, K2_V, C2_V	NAME, TYPE, TD, NOTE	THA

2) *The TD parameter shall be used only for dampers filled with silicone gel fluid (for example, dampers of VD series). In case when the temperature of the damper fluid is within the range from -10 ° C to +100 ° C, the dynamic characteristics of the damper shall be corrected by means of the following empirical expression:*

$$Si_t = 1.47 \bullet Si_{20^\circ} \bullet e^{-0.0193 \bullet t}\text{, where}$$

Si_t - *stiffness parameter at the working temperature t;*

Si_{20° - *stiffness parameter at the temperature of +20 °C (the damper model characteristics of VD type are set in the database namely for this temperature);*
t - working temperature of the fluid in the damper (in degrees).

In order to determine the working temperature in the damper fluid, it is recommended to use the following empirical relation:

$$t = k \bullet (t_m - t_s) + t_s\text{, where:}$$

t - working temperature of the fluid in the damper;
t_m - *operating temperature of the piping content;*
t_s - *ambient temperature in the compartment;*
k - heat transfer coefficient to be taken depending upon the connection between the damper piston and the piping. In case of direct connection, k=0.136; in case of connection via a heat-insulating gasket, k=0.1; in case of connection via spacer made from hollow profile, k=0.071.

3) *With the parameter value of OUT = 'YES', a file with the name of "DAMP_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the one-component support located in the "100" node. The file contains 7 columns of numbers: first column - time, 2nd - 4th columns - components of forces in the damper (Fx, Fy, Fz); 5th - 7th columns - dynamic deformations of the support (Dx, Dy, Dz). The OUT parameter is used only for the viscous or Maxwell model of the damper (TYPE = 'VIS' or 'MXW').*

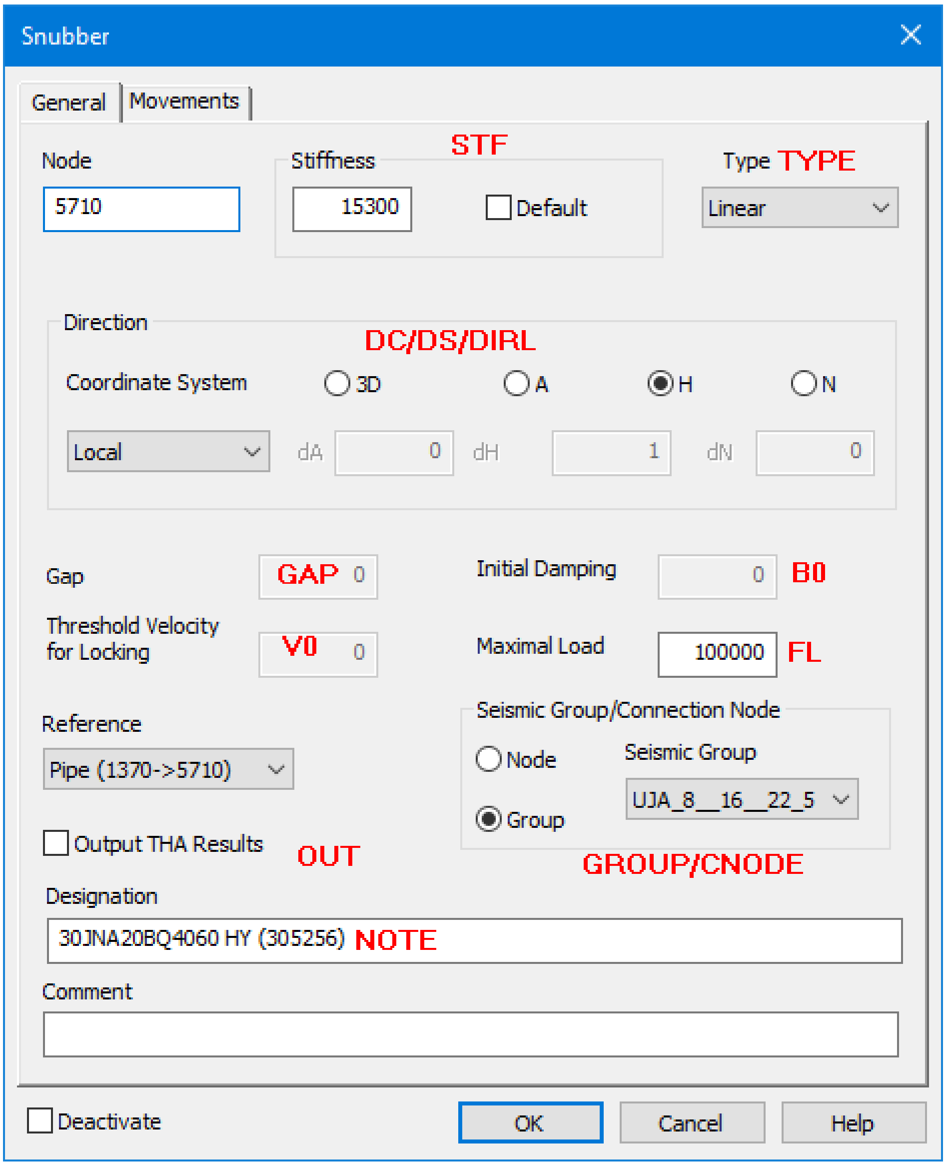
4) The GROUP and CNODE parameters are mutually exclusive;

5) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined.*

Examples:

```
2000: DMP TYPE = 'MXW' 'VD-426/219-3' TD 50
2000: DMP TYPE = 'VED' 'VD-426/219-3' TD 50 FH 5 FV 7
```

Snubber (SNUB)



type: local geometrical command

Function: mechanical or hydraulic snubber

Parameters:

TYPE type of snubber's model

type: TEXT
unit: -
default value: 'LIN'
limitations: 'LIN', 'MCH', 'HDR'

STF snubber spring rate

type: REAL
unit: N/mm
default value: rigid

limitations: ≥ 0

DC or DS: direction of the action of snubber; see "Local commands" section for the description of parameters.

DIRL[2]): alternative form of setting the direction of the snubber action line

type: TEXT
unit: -
default value: -
limitations: 'H', 'N', 'A'

GAP gap (available only for mechanical snubber model within THA)

type: REAL
unit: mm
default value: 0
limitations: ≥ 0

V0 threshold rate of locking of the hydraulic snubber (only for THA)

type: REAL
unit: mm/s
default value: 0
limitations: ≥ 0

B0 initial damping in the hydraulic snubber

type: REAL
unit: N*s/mm
default value: 0
limitations: ≥ 0

FL maximum load capacity

type: REAL
unit: N
default value: 0
limitations: ≥ 0

GROUP name of the seismic group of supports

type: TEXT
unit: -
default value: name of the first group of spectrums described by the SPEC command
limitations: name of the group to be selected from the 'GROUP' parameter of the SPEC command

OUT[3]) write the "time - force - deformation" records in a text file (available only for THA: DYN='THA')

type: TEXT

unit:	-
default value:	'NO'
limitations:	'YES' or 'NO'

NOTE4) support ID

type:	STRING
unit:	-
default value:	blank
limitations:	see limitations for the string values of parameters, the length shall not be more than 32 characters

CNODE4,5) connecting node

type:	TEXT
unit:	-
default value:	-
limitations:	the parameter shall refer to the label of one of the existing nodes of the analytical model

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

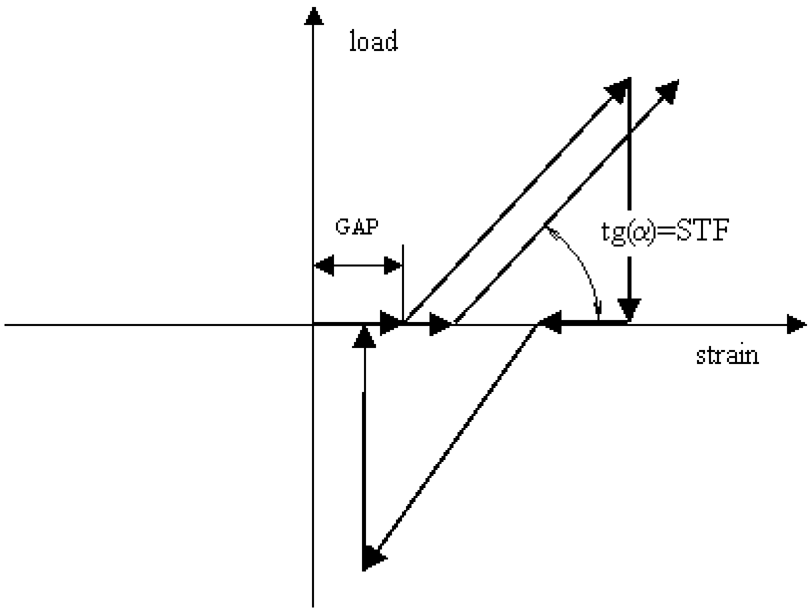
DZ support displacement along the global Z axis

type:	REAL

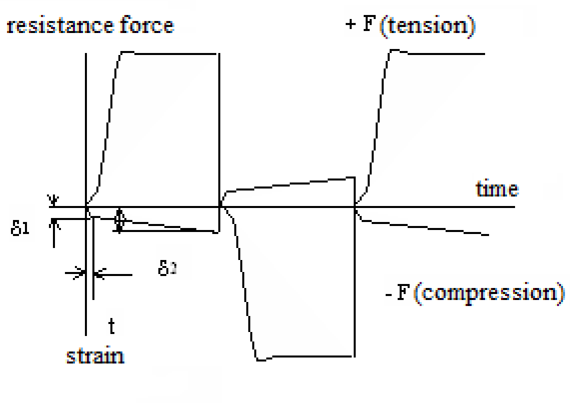
unit: mm
default value: 0
limitations: -

Note:

1) Snubbers are active only within dynamic analysis. Within RSM analysis only linear model of snubber is valid (TYPE='LIN').The figures given below illustrate different snubber models available within THA . Depending upon the snubber type, the command can contain the parameters defined in [Table](#)



Mechanical snubber model



Hydraulic snubber model

Combination of parameters with various forms of setting the SNUB command

Snubber model	Parameters	Type of analysis
Linear (TYPE='LIN')	TYPE, STF, DC (DS), FL, GROUP, NOTE	RSM, THA
Mechanical (TYPE='MCH')	TYPE, STF, DC (DS), GAP, FL, GROUP, NOTE	THA

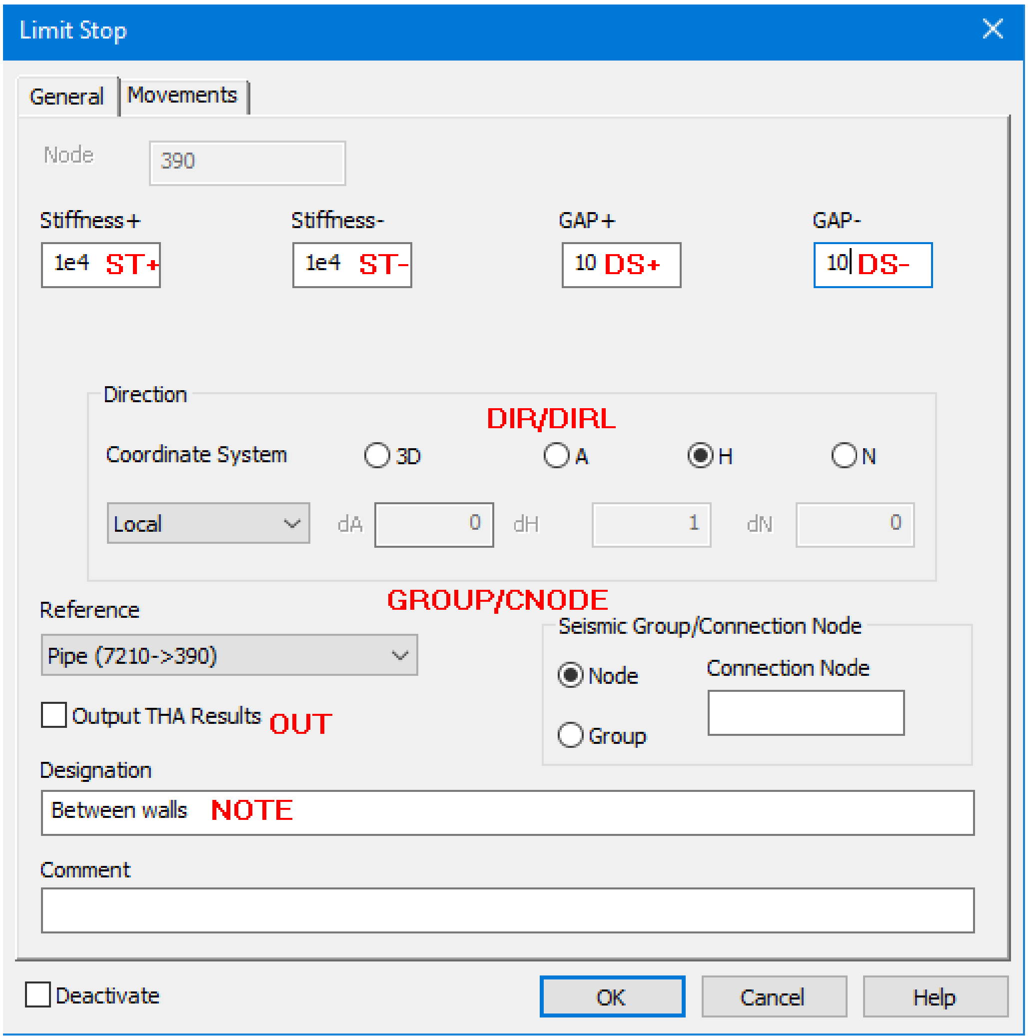
Hydraulic (TYPE='HDR')	TYPE, STF, DC (DS), V0, B0, FL, GROUP, NOTE	THA

2) *The direction of action of the snubber axis is set either by means of direction cosines (DC or DS) in the global coordinate system of the piping system or in the local axis of the support with respect to the piping system axis (DIRL parameter). In the latter case, DIRL can take the following values: 'H' and 'N' - perpendicularly to the piping system axis, 'A' - along the piping system axis. The rule for local coordinates of the support is given in Appendix II.*
3) *With the parameter value of OUT = 'YES', a file with the name of "SNUB_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the snubber located in the "100" node. The file contains 3 columns of numbers: the first column - time, 2nd column - snubber's reaction, 3rd column - deformation of the snubber.*
4) *The GROUP and CNODE parameters are mutually exclusive;*
5) *CNODE can be used to tie one node in the piping system to any other node in the system. If CNODE is set no restraint's movement could be defined.*

Examples :

```
2000: SNUB TYPE 'MCH' DC 1, 0, 0 STF 1.e3 GAP 2. FL 1.4e5
2000: SNUB TYPE 'HDR' DC 1, 0, 0 STF 1.e3 V0 2. B0 600 FL 1.4e5
2000: SNUB STF 1.e3 DIRL 'H' FL 1.4e5
```

Dynamic Limit Stop (DGAP)



The specification of the integration step is required when using Dynamic Limit Stops. It must be determined by the user based on the maximum frequency (minimum period) of the partial subsystem in which this element is installed.

Type: local geometrical command

Function: modeling of limit stop (restraint with gaps) active only within dynamic analysis[1)]

Parameters:

ST+ stiffness of the restraint acting after closure of the positive gap

type:	REAL
unit:	N/mm
default value:	0
limitations:	≥0

ST- stiffness of the restraint acting after closure of the negative gap

type:	REAL
unit:	N/mm

	default value:	0
	limitations:	≥0

DS+ positive gap value

type:	REAL
unit:	mm
default value:	0
limitations:	≥0

DS- negative gap value

type:	REAL
unit:	m
default value:	0
limitations:	≥0

DIR2) projections of the line of action of the element on the global coordinates XYZ, or the corresponding direction cosines.

type:	REAL
unit:	-
dimension:	array of three element
default value:	-
limitations:	all three elements of the array cannot be simultaneously equal to zero

DIRL2) alternative form of setting the direction of action of the element (in local coordinates)

type:	TEXT
unit:	-
default value:	-
limitations:	'A', 'H', 'N'

OUT3) flag of output of the time history of forces/deformations for printing

type:	TEXT
unit:	-
default value:	'NO'
limitations:	'YES', 'NO'

NOTE Note / Comment

type:	STRING
unit:	-
default value:	blank
limitations:	see limitation for string parameter values

Subcommand

type: subcommand

Function: to be used for setting the pre-defined displacements of the supports

Parameters:

MODE - identification name of the piping system operating mode.

type:	TEXT
unit:	-
default value:	-
limitations:	see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

DX support displacement along the global X axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DY support displacement along the global Y axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

DZ support displacement along the global Z axis

type:	REAL
unit:	mm
default value:	0
limitations:	-

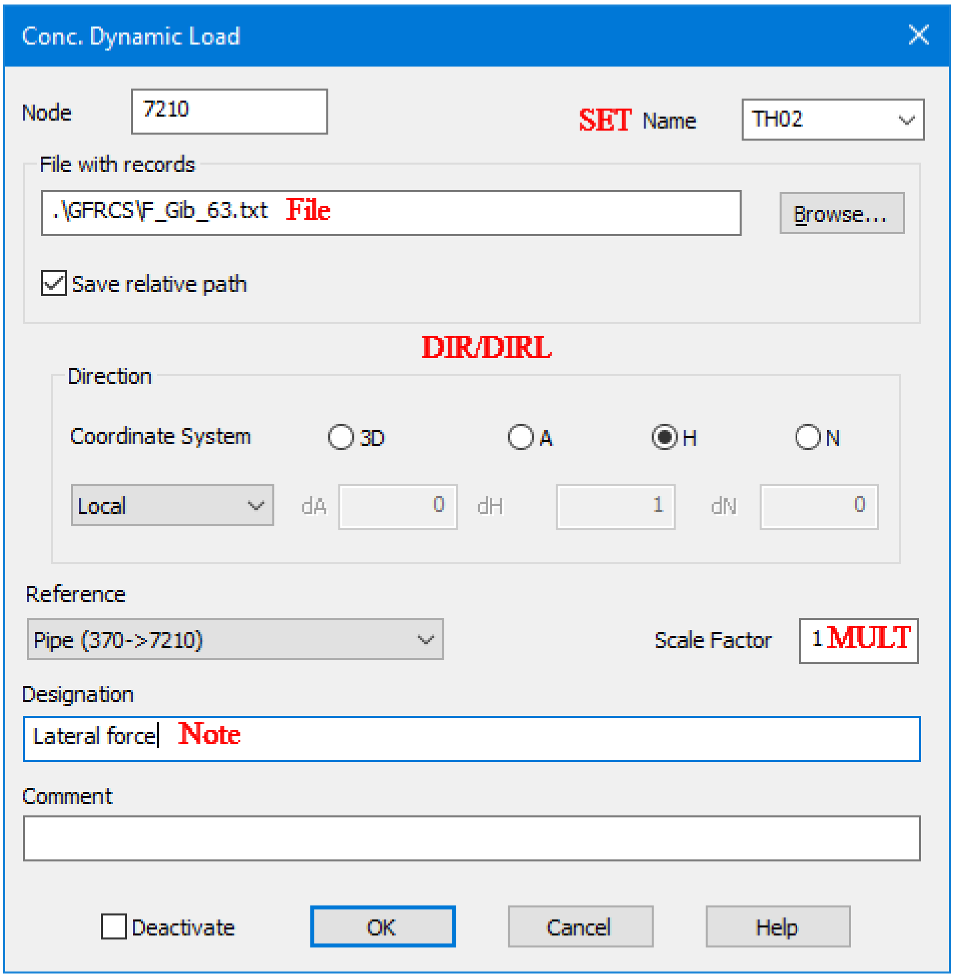
Note:

1) *The command is valid only within THA (DYN='THA')*
2) *The parameters determining the direction of action of the DIR and DIRL elements are mutually exclusive. See Appendix II for determination of the local coordinate system.*
3) *With the parameter value of OUT = 'YES', a file with the name of "DGAP_001_100.dat" type will appear in the working directory of the model after execution of the analysis. In the file name: "100" is the node name, "001" -is the sequence number of the snubber located in the "100" node. The file contains 3 columns of numbers: the first column - time, 2nd column - forces in the link, 3rd column - movements in the element.*

Example:

```
123: DGAP ST+ = 1.e4 ST- = 1.e4 DS+ = 2 DS- = 2 DIRL='H' OUT='YES'
```

Concentrated Transient Dynamic Load (DFRC)



Type: local geometrical command

Function: command for applying transient force for THA[1]

Parameters:

SET The identification name of the set for dynamics loads' family. SET is referenced in the command DCASE (parameter INP)

FILE[2,3] name of the file containing digital records of the applied force

type: STRING
unit: -
default value: -
limitations: see limitations for string parameter values. The length of string shall not exceed 128 characters.

MULT scaling factor

type: REAL
unit: -
default value: 1

limitations: -

DIR[4)] projections of the line of action of the force on the global coordinates XYZ, or the corresponding direction cosines.

type: REAL
unit: -
dimension: array of three elements
default value: -
limitations: all three elements of the array cannot simultaneously be equal to zero.

DIRL[4)] alternative form of setting the direction of action of the concentrated dynamic load (in local coordinates)

type: TEXT
unit: -
default value: -
limitations: 'A', 'H', 'N'

NOTE Note / Comment

type: STRING
unit: -
default value: blank
limitations: see limitations for string parameter values

Note:

1) *The command is valid only within THA (DYN='THA')*
2) *If the file with digital dynamic force values is located in the current directory of the model, it is sufficient to indicate only its name with the extension. In other cases, it is necessary to indicate the full path to the file.*
3) *The file containing the digital values of the force is an ASCII file containing 2 columns of numbers: "time-force" typed in free format. The force shall be set in Newtons. The file must contain a point for the "zero" time moment. If the total time of dynamic action, TT, is greater than the force action time specified in the file, it is assumed that the after that an action of force has stopped (=0).*
4) *The parameters determining the directions of action of the force, DIR and DIRL, are mutually exclusive. See Appendix II for determination of the local coordinate system.*

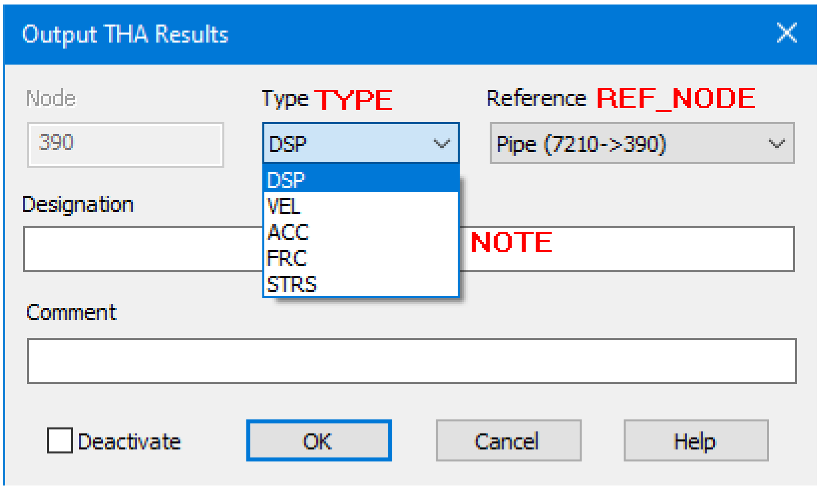
Example:

```
123: DFRC file = 'f1.dat', dir=1,0,0 note = 'Force from operation of the safety val
```

or:

```
123: DFRC file = 'f1.dat', dirl='H' note = 'Force from operation of the safety valv
```

Time History output /Travel indicator (TH_OUT)



Type: local geometrical command

Function: output of Time History response parameters

Parameters:

TYPE type of output parameter

type: TEXT
unit: -
default value: 'DSP'
limitations: 'DSP', 'VEL', 'ACC', 'FRC'

NOTE Note / Comment

type: STRING
unit: -
default value: blank
limitations: see limitations for string parameters values

Note:

1) Within THA (DYN='THA'), depending upon the value indicated in the TYPE parameter, the following relations will be output in the course of post-processing of the results:

TYPE	Parameter	Name of file	Note
DSP	displacements	DSP_NODE.dat	
VEL	velocities	VEL_NODE.dat	(a)
ACC	accelerations	ACC_NODE.dat	
FRC	internal forces/deformations in the element	TYPE_NODE1_NODE2 .dat	(b)
STRS	dynamic stresses S2 category (only for CODE = 'PNAE')	TYPE_STRS_NODE1-NODE2.dat	(c)

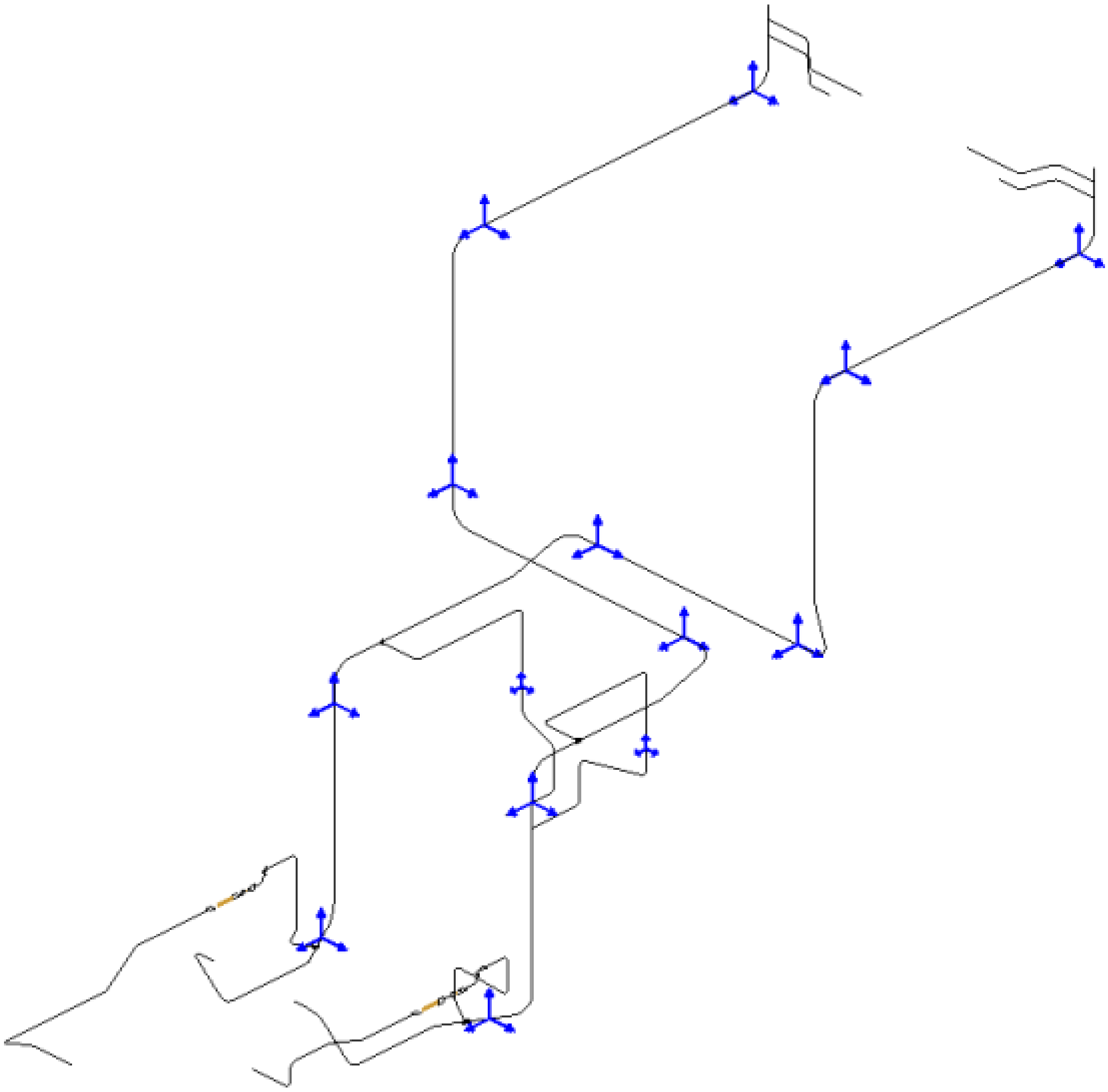
(a) four columns are output into the file: time + 3 components along the global axes X, Y, Z

(b) depending upon the element type, the "time - components" columns are output into the file (in the local coordinate system)":

Type of element	Beginning of the file name	Components being output
Bend	BEND…	19 columns: time + 3*6 components of the internal forces at the beginning, at the end and in the middle of the element
Straight Pipe	PIPE…	
Reducer	REDU…	13 columns: time + 2*6 components of the internal forces at the beginning and at the end of the element
Expansion Joint	EJ…	19 columns: time + 2*6 forces at the beginning and at the end of the element + 6 deformations in the center of the element
Rigid Link	RIGD…	13 columns: time + 2*6 components of the internal forces at the beginning and at the end of the element
Beam	BEAM…	
Flexible Joint	FLEX…	
Valve	VALV…	

(c) four number columns are writing in the file: time – stress at the beginning of the element, in the middle and at the end

2) The command TH_OUT with DISP parameter (node: TH_OUT = 'DSP') is considered also as a "travel indicator" and results to a summary table of displacements associated with indicated node for all static Load Sets required displacement output. In viewing the results in the PIPE3DV program, the travel indicators are displayed in the form of three orthogonal arrows:



and the following generalized table will be printed in the <model name>.sup file:

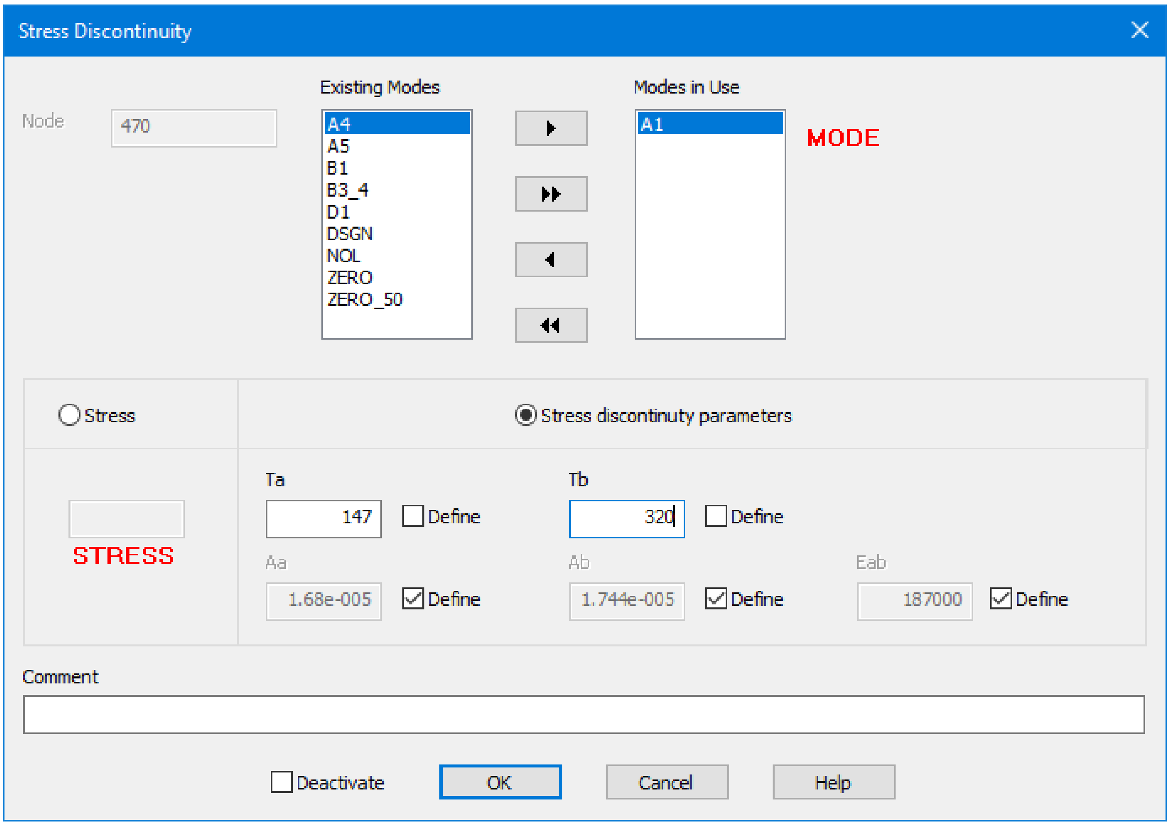
```
>>>  Piping movements (displacement monitoring).
========================================================================
===============================================
Point N           DX        DY        DZ        RX            RY
RZ        Mode
   (node)                ( mm )                           ( rad )
========================================================================
===============================================
№ 1                 0        -1        -1    -2.109E-04   -3.839E-04
7.167E-05   Weight Deflections
                  -15         3        10     1.791E-03   -8.742E-04
9.215E-04   Thermal Expansions
========================================================================
===============================================
```

3) The command TH_OUT with ACC parameter (node: TH_OUT = 'ACC') leads to the summary table with maximal accelerations in the specified node:

```
>>>  Accelerartions (g)
==================================
   Node      AX       AY       AZ
==================================
3            2.63     2.38     1.48   AV204-1A
10          -9.76     2.33     4.24   VZ04-1A
13           6.66     2.32    -2.97   V204-2A
==================================
```

```
Example:

123: TH_OUT TYPE = 'ACC'

or

30: TH_OUT type = 'DSP', note = "N 01"
```

Discontinuity stresses (STR_DISC)



type: local geometrical multi-line command

Function: Input data for assessment of Stress Intensity Range due to structural or material discontinuity.The stresses are used in the analysis according to the "nuclear" codes: PNAE: stress $(s)_{T0}$ (Appendix 5, Item 2.3.2.4, [REF 1]) and ASME BPVC NB-3653.2, [REF 3].

Subcommand

Parameters:

MODE identification name of the operating mode consistent with input parameters.

type: TEXT
unit: -
default value: -
limitations: see limitations for the text values of parameters. The operating mode shall preliminarily be described by the OPVAL command

TA, TB range of average temperature on side a(b of gross structural discontinuity or material discontinuity

type: REAL
unit: °C
default value: -
limitations: -

EAB average modulus of elasticity of the two sides of a gross structural discontinuity or material discontinuity at room temperature (Eab = 0.5*(Ea+Eb))

type: REAL
unit: MPa
default value: to be determined by the program automatically from the input data
limitations: ≥ 0

AA, AB coefficient of thermal expansion on side a(b) of a gross structural discontinuity or material discontinuity, at room temperature

type: REAL
unit: 1/°C
default value: to be determined by the program automatically from the input data
limitations: ≥ 0

STRESS pre-computed discontinuity stress value

type: REAL
unit: MPa
default value: 0
limitations: -

Note: *in one command, it is permitted to set either TA, TB, EAB, AA, AB parameters or STRESS. If no specific reference for operational mode is defined, it's assumed that temperature stresses due to gross structural discontinuity occur for any piping transient from one operational state to another. In this case stresses are calculated automatically. Average temperatures Ta and Tb are taken according to the properties of adjacent parts for given mode.*

Example:

```
 2000: STR_DISC
& MODE 'NOL' TA 23 TB 45
```

```
& MODE 'MODE1' STRESS 345.5
```

9 References

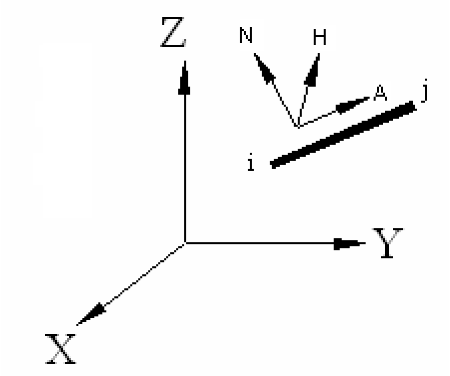
REF 1. PNAE G-7-002-86 Equipment and pipelines strength analysis codes for nuclear power plants
REF 2. RD 10-249-98 Strength Design Code for Stationary Boilers and Steam and Hot Water Pipelines
REF 3. ASME BPVC, Section III, Subsections NB-3600, NC-3600
REF 4. Report No. Rep 02-05/04-99, Software package for strength analysis of piping systems under the action of operating and seismic loads. dPIPE 5. Theoretical basics
REF 5. Piping system strength analysis for nuclear power plants. RTM 108.020.01-75
REF 6. LISEGA Standard supports 2020 - Catalogue, June 2015
REF 7. V.A. Nakhalov, R.K. Balashova. Adjustment of the piping system anchorage for thermal power plants, M., Energia, 1975
REF 8. ASME B31.3-2004, Process Piping, ASME Code for Pressure Piping, B31, an American National Standard
REF 9. Service properties of boiler materials - Guidelines, Issue No.43 - NPO CKTI, 1981
REF 10. EN 13480-3 Metallic industrial piping - Part 3: Design and calculation
REF 11. Document No.SM01-08 Example of fatigue strength analysis and fatigue accumulation coefficient for a test piping system, CKTI-Vibroseism Ltd., fatigue_sample.pdf
REF 12. ASME B31.1-2007, Power Piping, ASME Code for Pressure Piping, B31, an American National Standard
REF 13. S.P. Timoshenko. Mechanics of materials, 1976
REF 14. NP-068-05, Piping valves for nuclear power plants. General technical requirements
REF 15. ASME ST-LLC 07-02 SIF and K-factor Alignment Project (R1), Paulin Research Group (PRG), March 27, 2011
REF 16. RD EO 1.1.2.05.0330-2012. Lifetime Extension Guideline for the strength calculations of piping and equipment of RBMK, VVER and EGP reactor facilities
REF 17. Normativně Technická Dokumentace Asociace Strojních Inženýrů pro použití na jaderných elektrárnách typu VVER, NTD A.S.I. Sekce III, Příloha A, Hodnocení pevnosti zařízení a potrubí jaderných elektráren typu VVER, 2016
REF 18. WRC-300 (Welding Research Council). Technical position n industry practice, The Design Process for Small Bore Piping, December 1984
REF 19. ANSI/ANS-58.2-1988, Design basis for potential of light water NPP against the effects of postulated pipe rupture
REF 20. NUREG-0800, US NRC Standard Review Plan, Section 3.6.2 “Determination of Rupture Locations and Dynamic Effects associated with the Postulated Rupture of Piping”
REF 21. OTT 1.5.2.01.999.0157-2013 «Supporting structures for elements of nuclear power plants with water-cooled power reactors. General technical requirements»
REF 22. Addendum to LISEGA catalogue „Standard Supports 2010" in reference to Russian standard pipe dimensions for nuclear application, Document No.: 902205, Rev. 1
REF 23. GOST R 59115.3-2021, Justification of Strength of Equipment and Pipelines of Nuclear Power Plants. ***Short-term mechanical properties of structural materials***
REF 24. GOST R 59115.9-2021, Justification of Strength of Equipment and Pipelines of Nuclear Power Plants. ***Calculation of strength.***
REF 25. GOST R 59115.15-2021, Justification of Strength of Equipment and Pipelines of Nuclear Power Plants. ***Calculation of strength of typical pipeline units.***
REF 26. Piping Flexibility and Stress Analysis Software dPIPE 5. Verification Manual, VR01-07
REF 27. TU 4192-001-20503039-01. Viscoelastic dampers VD series. Specifications

10 Appendix I

Sign convention and directions of the local coordinates for internal forces in elements

I. "Straight pipe", "Beam" elements

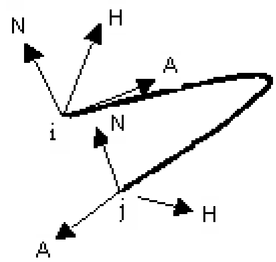
1. The local A axis is directed along the element axis from node i to node j.
2. If the local A axis is not parallel to the global Z axis, then the local N axis lies in the "local A – global Z" plane and is directed so that its projection onto the global Z axis is positive. The local H axis is constructed according to the right-hand rule (H = N x A)



3. If the local A axis is parallel to the global Z axis, then the local H axis coincided with the global Y axis, and the local N axis is constructed according to the right-hand rule (N = A x H)

II. Bend element

1. The local A axis is directed along the tangent to the bend element axis from node i to node j.
2. The local H axis lies in the bend plane perpendicular to the local A axis and is directed from the center of curvature.
3. The local N axis is constructed according to the right-hand rule (N = A × H)

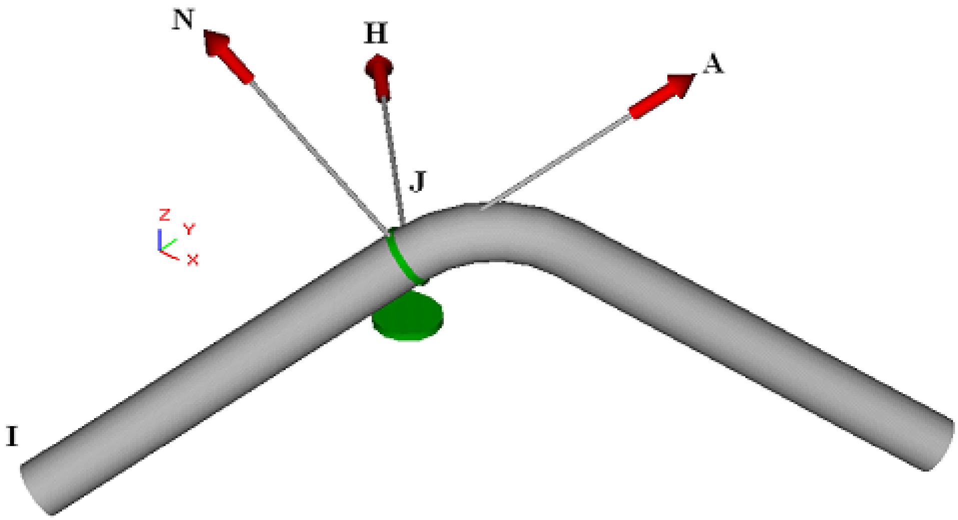


11 Appendix II

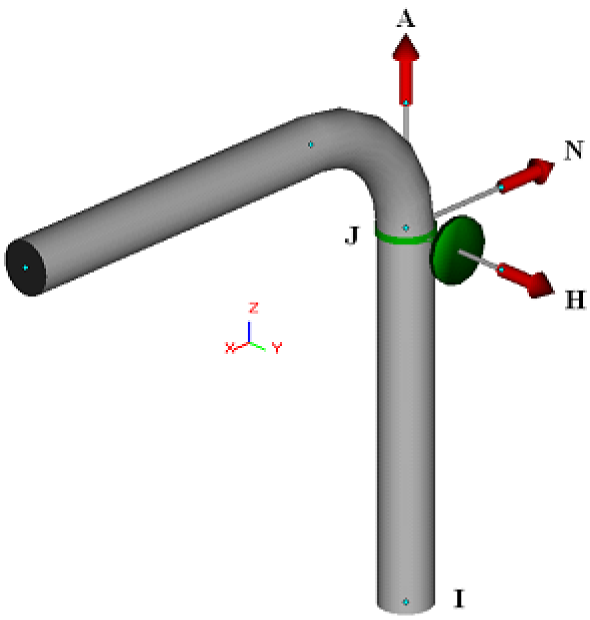
Direction of local coordinates for supports

The local coordinate system for supports is determined as follows:

1. The A axis is constructed along the tangent to the axial line of the element from node I to node J.
2. If the element axis is not parallel to the global Z axis, then the H axis is constructed as a perpendicular to the4 vertical plane being formed by the A and Z axes: H = Z x A. If the A axis is parallel to the Z axis, then the H axis coincides with the Y axis.
3. The N axis is constructed perpendicular to the A and H axes: N = A x H (see Fig.)

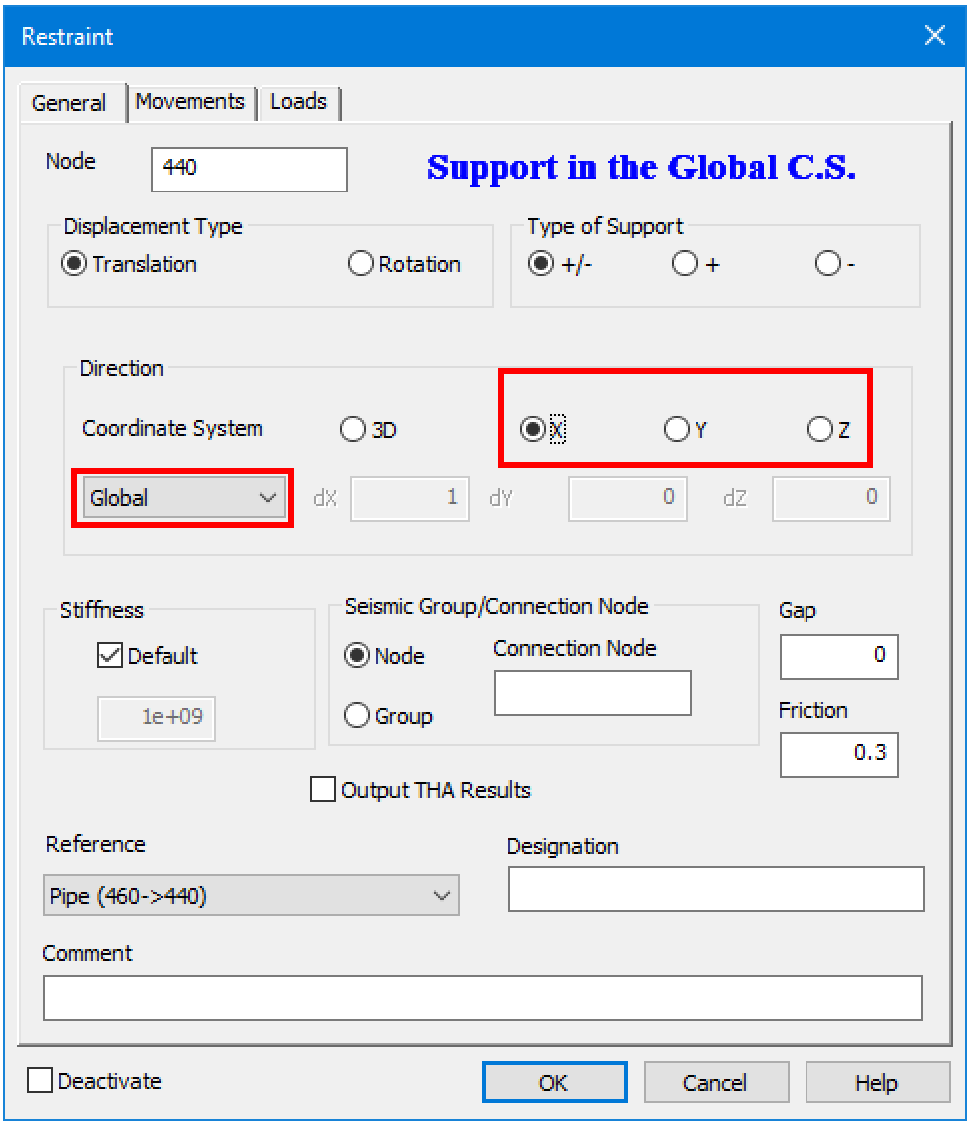


Direction of local coordinates of the support for a non-vertical piping segment

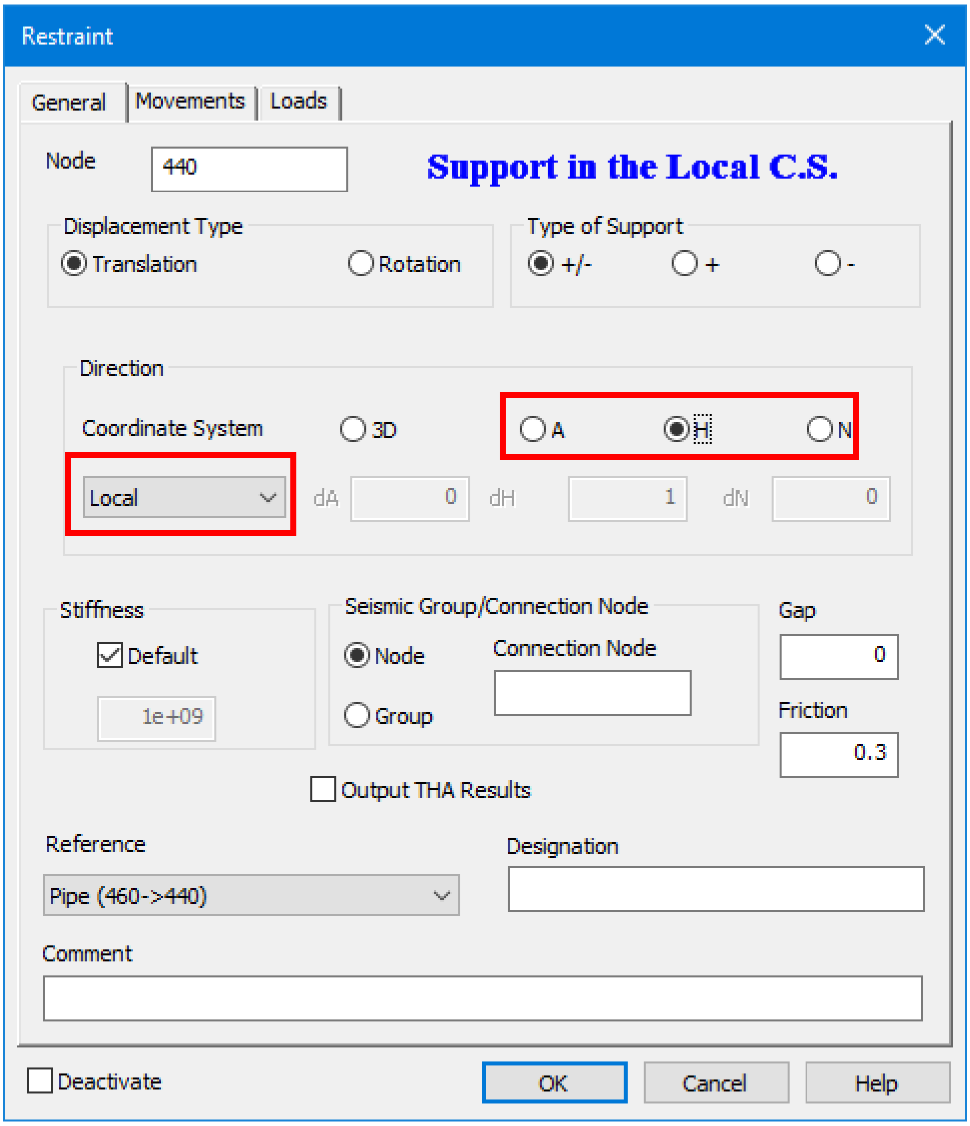


Direction of local coordinates of the support for a vertical piping segment

The characteristics of the ***one-component*** supports (restraints) can be defined in both the global (XYZ) and the local (AHN) coordinate systems. The last one is defined relative to the adjacent element. In this case, the direction of the support's action should be aligned along the global or local axis. In the *.dp5 file, these directions correspond to the commands STX, STY, STZ for the global axis directions, and the commands STA, STH, STN for the local axis directions, respectively.



Restraint
General
Movements
Loads
Node
440
Support in the Global C.S.
Displacement Type
Translation
Rotation
Type of Support
+/-
+
-
Direction
Coordinate System
3D
X
Y
Z
Global
dX
1
dY
0
dZ
0
Stiffness
Default
1e+09
Seismic Group/Connection Node
Node
Group
Connection Node
Gap
0
Friction
0.3
Output THA Results
Reference
Pipe (460->440)
Designation
Comment
Deactivate
OK
Cancel
Help



In all other cases, it is assumed that the support is defined in an arbitrary direction (command STS).

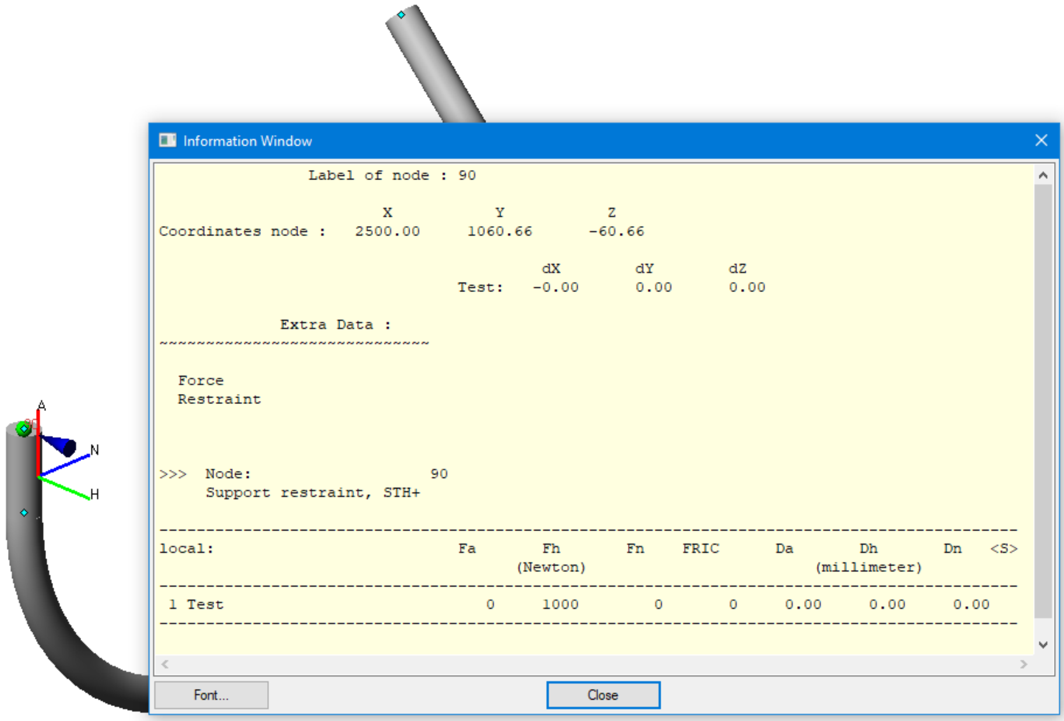
In the calculation results, the representation of reaction's components for the one-directional supports (restraints) depends on how they are defined:

Definition along global coordinates: reactions are displayed in the global coordinates system (STX, STY, STZ).

Definition along local axis of the element (STA, STH, STN): reactions are displayed in the local coordinates of the element.

Definition in an arbitrary direction (STS): reactions are displayed in the local coordinates of the support. In this case, the "A" axis is considered as the line of the support's action, and the other axes (H and N) are built according to the rules of the local coordinates for the element (see Appendix 1).

The label for the support's direction is provided in the results printouts (*.sup file). This information can also be seen when viewing the results in the PIPE3DV program.



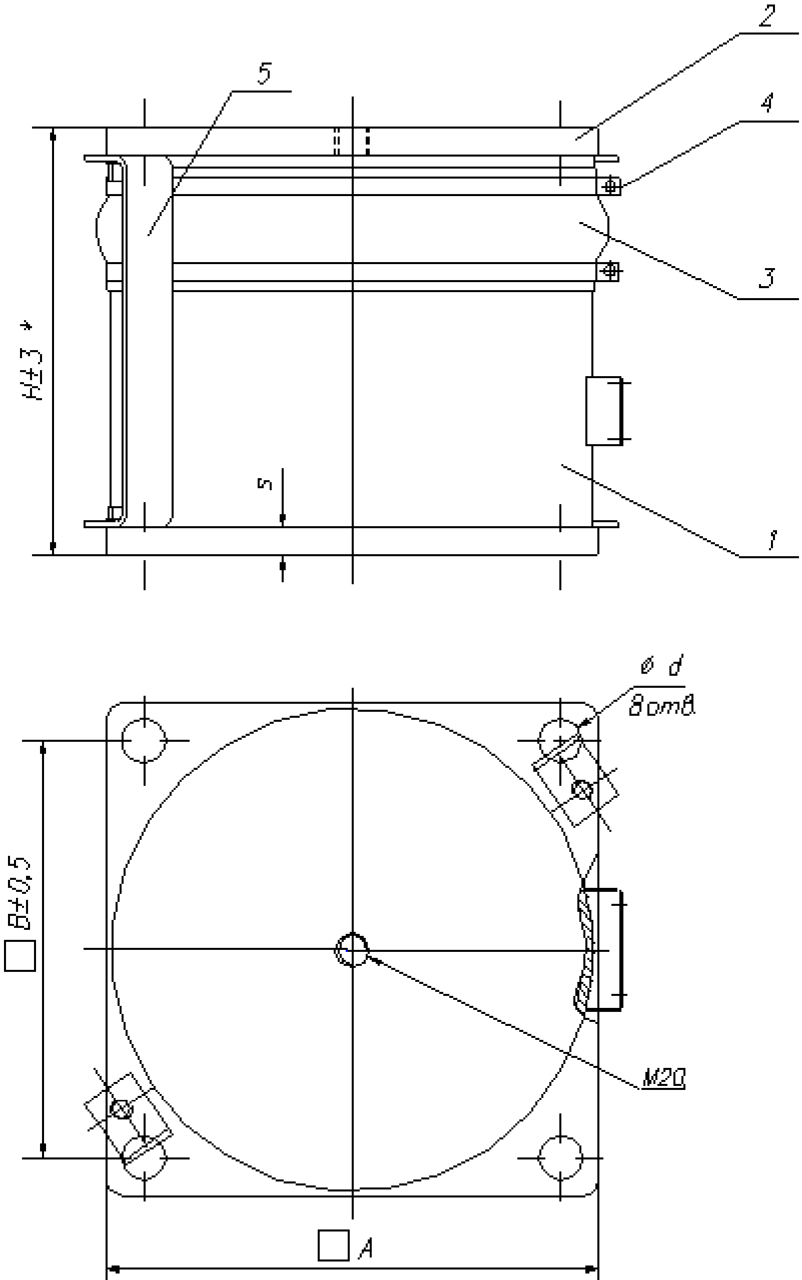
12 Appendix III

Nomenclature and parameters of VD dampers (ТУ 4192-001-20503039-01)

Damper Type	Nominal Load		Allowable Displacement (±) From Normal Position		
	Horizontal	Vertical	Horizontal	Vertical	Angular
	H	H	mm	mm	degree
VD-108/57-3	1750	1200	13	13	9
VD-159/76-3	3800	2650	27	25	11
VD-159/76-7	8100	4500	25	25	11
VD-219/108-3	7200	5050	41	24	11
VD-219/108-7	15500	8500	39	24	11
VD-219/159-3	10000	7000	15	24	6
VD-325/159-3	16000	11000	67	40	14
VD-325/159-7	34000	18500	64	40	14
VD-325/159-15	68000	27000	58	40	14
VD-325/219-3	21000	15000	37	40	9
VD-325/219-7	46000	25000	34	40	9
VD-426/219-3	27000	19000	87	45	18
VD-426/219-7	58000	32000	84	45	18
VD-426/219-15	120000	47000	78	45	18
VD-426/325-3	36000	25000	34	45	7

VD-426/325-7	80000	44000	31	45	7
VD-630/325-3	60000	42000	134	74	11
VD-630/325-7	130000	70000	130	74	11
VD-630/325-15	260000	100000	122	74	11
VD-630/426-3	80000	56000	84	74	11
VD-630/426-7	175000	95000	80	74	11
VD-630/426-15	350000	140000	72	74	11

Damper Type	Weight, kg ±5%	Dimensions, mm				
		H*	A	B	d	s
VD 108/57-3	6	152	130	106	14	8
VD 159/76-3	16	197	180	150	18	10
VD 159/76-7	17	197	180	150	18	10
VD 219/108-3	35	236	238	200	22	15
VD 219/108-7	36	236	238	200	22	15
VD 219/159-3	35	236	238	200	22	15
VD 325/159-3	90	333	342	286	33	20
VD 325/159-7	93	333	342	286	33	20
VD 325/159-15	99	333	342	286	33	20
VD 325/219-3	110	333	342	286	33	20
VD 325/219-7	113	333	342	286	33	20
VD 426/219-3	176	378	434	368	39	25
VD 426/219-7	181	378	434	368	39	25
VD 426/219-15	190	378	434	368	39	25
VD 426/325-3	223	378	434	368	39	25
VD 426/325-7	229	378	434	368	39	25
VD 630/325-3	535	556	646	542	60	35
VD 630/325-7	551	556	646	542	60	35
VD 630/325-15	579	556	646	542	60	35
VD 630/426-3	635	556	646	542	60	35
VD 630/426-7	652	556	646	542	60	35
VD 630/426-15	685	556	646	542	60	35



1 – housing; 2 – piston; 3 – protecting sleeve; 4 – hoop; 5 – spacer.

13 Appendix IV

Assignment of the weight characteristics of elements

The weight of any piping component could be considered as sum of three parts:

W_1 – weight of material
W_2 – weight of insulation;
W_3 – weight of piping content.

For elements of "Straight pipe" (PIPE) and "Bend" (BEND) the weight characteristics are computed in accordance with the data defined for the corresponding cross-sections (PIPE command) and information contained in the command for setting the operating modes of the piping system (OPVAL). For beam elements (S command) the weight shall be determined in accordance with the W parameter (weight per length), see BEAM command.

For piping cross-sections the following parameters shall be set: W - piping weight per length and IWGT - insulation weight per length. In addition, for each operating mode the CSG parameter (Content Specific Gravity), shall be assigned, which is set in fractions of the water weight (for water this parameter is equal to unity: CSG = 1). In turn, the density of water is set by the W_DEN parameter (CTRL command), and the relative density of the piping material - by the DEN parameter (MAT command).

Hence the corresponding weight components for each of the "Straight pipe" and "Bend" type element shall be calculated according to the following expressions:

$$W_1 = W*L$$
$$W_2 = IWGT*L$$
$$W_3 = CSG*W\_DEN*A*L,$$

where A is the area of the internal cross-section of the pipe, L is the element's length.

By default, the W parameter of piping cross-sections (PIPE command) is determined according to the formula:

$$W = \pi * (OD - T) * T * DEN * W\_DEN$$

where OD is the outside diameter of the pipe, T is the wall thickness.

The total weight load for the element is determined as follows:

$$W_e = W_1 + W_2 + W_3$$

In addition, the concentrated weight characteristics, which can be set by CW command, shall be added to the above-listed components to the total weight of the piping.

For elements simulating the expansion joints (EJ, EA, ET, EH, EG), piping valves (VALV, V1, V2), reducers (REDU) as well as "rigid" links (RX, RP), the weight characteristics components are to be set by their parameters. (W_1, W_2, W_3).
In doing so, the following rules are used:

1. If $W_i > 0$, then the program will interpret the corresponding value as the component weight (1 – material, 2 – insulation, 3 – content). **For W_3 it is necessary to set the weight of water** (in the course of analysis the program will recalculate this weight depending upon the CSG value determined for the corresponding operating conditions).

2. If W1 = 0 and the rest of components are not set, then the total element's weight will be taken equal to zero.

3. If $W_i < 0$, then the program will perceive the data as coefficients and use the following formula for determination of the weight load components:
$$W_i = |W_i| * W_{pipe} * L$$

where W_{pipe} is the weight per length (material/insulation/water) determined according to the data for the "current" cross-section of the piping system. For reducers (REDU) the average values of adjacent cross-sections are used.

4. The total weight of the element is determined as the following sum:

$$W_e = W_1 + W_2 + CSG * W_3$$

5. Depending upon the element type by default the following values of the W_1, W_2, W_3 parameters are used:

	REDU	VALV, V1, V2	EJ, EA, ET, EH, EG	RX, RP
W_1	-1	≠0	0	0
W_2	-1	-1.75	0	0
W_3	-1	-1	0	0

If the value of parameter W_1 differs from the value specified by default in the table above, then the W_2 and W_3 parameters will take the following values by default:

	REDU	VALV, V1, V2	EJ, EA, ET, EH, EG	RX, RP
W_1	**0**/≠0	**0**	≠0	≠0
W_2	0/-1	0	-1	0
W_3	0/-1	0	-1	0

14 Appendix V

Standard sets of Analysis and Postprocessor Specifications

1. **Analysis according to PNAE Codes [REF 1]**

2. **Analysis according to the RD Boiler Codes [REF 2]**

Analysis according to PNAE Code

In case of analysis according to the PNAE Codes, the CODE parameter can take the values CODE = 'PNAE' for low-temperature piping systems and CODE = 'PNAE_T' for high-temperature piping calculations.

The full verification analysis according to the PNAE Code assumes performing of the following stages of analysis [REF 5]:

Stage	Content of analysis	Load factors taken into account	Purpose	Stress category
I	Analysis for Sustained Loads	Internal pressure (P); weight load (W), spring hangers/supports loads acting in the operating condition (operating load)	Calculation of Primary Stress Intensity	$(\sigma)_2$
II	Analysis for the concurrent action of all loads specified for the operating state	Same as above + thermal expansion (T) + "displacement" loads due to support's movements (D) + cold spring (CS)	Calculation of Support's Reactions and Equipment's Nozzle Loads	(1)
IIIa	Analysis for the effect of the secondary loads (transient of system from one to other load set)	Pressure difference (ΔP); thermal expansion (T); support's displacements (D)	Fatigue strength assessment according to the adaptability criterion	$(\sigma)_{RK}$
III	Analysis of the effect of variable loads with the allowance for stress concentration and additional stress from irregularity of the cross-sectional shape	The same	Satisfaction of Peak Stress Intensity Range, Fatigue strength assessment. Calculation of displacements from thermal expansion.	$(\sigma_{aF})_K$
IV	Analysis for the cold state	Weight load (W); cold spring (CS)	Calculation of Support's Reactions and Equipment's Nozzle Loads	-

Note:
(1) For high-temperature piping systems Stage II serves for check of additional conditions for the creep range (assessment of stress of category (s)$_{RK}$). In doing so, the components of the stressed state from the compensation of temperature expansions shall be determined with the account of relaxation due to creep (see the CREEP command). The main recommended specifications for the execution of analysis according to the PNAE Codes are given in the 'SOLV.DBS' file. At that the following classification of the calculations being performed has been taken:

LOW-TEMPERATURE PIPING SYSTEMS

Calculation No.1 – Spring design + Stress Analysis;

In order to select springs for hanger supports , a set of "starting" computations shall be performed (Load Case):

```
&LC MOD='$OPER' TYPE='DSGN' Note="Spring Hangers Design Load" ; LC1
&LC MOD='$OPER' TYPE='OPER_A' PEND='NO' FRIC= 'NO' Note="Hot Load.
Selection of springs" ; LC2
&LC MOD='$COLD' TYPE='OPER_B' PEND='NO' FRIC ='NO' Note="Cold Load.
Selection of springs" ; LC3
```

In the example above, the first line (LC1) determines "restrained weight" analysis of the system on the rigid supports at the places of installation of spring hangers being selected. In doing so, the "required" values of operating loads in hangers, P_h, which shall balance the weight load of the piping system. The calculation shall be performed for the operating state; at that the piping system is considered without account of the friction forces in supports and swing effect in hangers.

In case of successful execution of this calculation, it will be followed by the full load analysis (second line, LC2): $W+P+T+D+P_h$. It is an analog of Stage II but once again without account of the friction forces and swing effect.

The results of this analysis is determination of the full movement of the system from the "zero" state into the operating state.

The third line of the specification for analysis (LC3) corresponds to the selection of springs with the account of fulfillment of the PVAR variation condition between the operating (hot) and reference (cold) states. The calculation for spring selection is iterative. The closest springs as to the load are taken as the first approximation for the spring stiffness with the account of the load safety factor PFAC. In addition, the calculation is performed without account of friction and swing effect. On the basis of the results of this calculation the spring type is selected for each spring hanger/support and the value of installation load R_0 is computed.

The successful execution of the "starting" set is followed by the set of main computations:

```
&LC MOD='$OPER' TYPE='OPER_B' PEND='YES' FRIC='YES' Note="Hot Load.
Stage II"; LC4
&LC MOD='$OPER' TYPE='SUST_C' Note="Sustained Loads. Stage I"
                   ; LC5
&LC MOD='$COLD' TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Cold Load.
Stage IV"; LC6
```

LC4 – full load calculation (stage II) for the operating state with the account of all support's nonlinearities;
LC5 - analysis for the effect of weight and pressure (stage I). The loads on a hanger and the state of one-way supports perceiving the weight load are determined according to the results of LC4. In doing so, the supports, which "uplifted" at the previous stage of analysis, are excluded. Friction and swing effect are not taken into account.
LC6 – analysis for the reference (cold) state (stage IV) with the account of all support's nonlinearities;

The specification for post-processing of the results of analysis for the example under consideration consists of the following set of commands:

```
&RES='S2_NUE' LS="LC5" Note="S2 stress (NOC)" ; LS1
&RES='SRK' LS="LC4-LC6" Note="Srk Stress"; LS2
&RES='SAF' LS="LC4-LC6" Note="Saf stress" ; LS3
&RES='DISP' LS="LC5" Note ="Weight deflections" ; LS4
&RES='DISP' LS="LC4-LC6" Note ="Thermal expansions" ; LS5
&RES='SUPP' LS="LC4" Note="Hot Loads" ; LS6
&RES='SUPP' LS="LC6" Note="Cold Loads" ; LS7
```

LS1 – determines computations of the stress of category $(\sigma)_2$ for the combination of loads corresponding to NOC (weight + pressure). For stress computations the internal forces in elements are used, which have been calculated in the LC5 calculation.
LS2 – computation of the stress of category $(\sigma)_{RK}$ (shakedown condition) as the transient from the cold state into the operating state. In order to determine the stress, the internal forces in elements

are used, which have been calculated as the difference between the loads in the operating and cold states: LC4-LC6.

LS3 – computation of the stress of category $(\sigma_{aF})_K$ (fatigue strength assessment). The load combination is the same as for stress $(\sigma)_{RK}$: LC4-LC6.

LS4, LS5 – printout of the movements from weight loads corresponding to Stage I (LC5) and "visible" movements (differences between the system deformation in the operating state and in the cold state): LC4-LC6.

LS6, LS7 – printout of the loads on supports for the operating (hot) state (LC4) and cold state (LC6).

In case of successful completion of this calculation, a file of <model name>.dp5_ type will be created in the working directory, which contains data for hangers with the spring types and working loads specified (thereafter this file can be renamed into <model name1>.dp5 and used as the main file for subsequent computations).

Calculation No.2 – Verification calculation with specified characteristics of spring hangers/supports

In this calculation it is assumed that the spring characteristics and working loads for spring hangers/supports have been determined; therefore, the starting calculations are missing in the specification for analysis. As for the rest, the calculation repeats the previous example:

```
&LC MOD='$OPER' TYPE='OPER_A' PEND='YES' FRIC='YES' Note="Stage II (hot
load)"; LC1
&LC MOD='$OPER' TYPE='SUST_C' Note="Stage I"
; LC2
&LC MOD='$COLD' TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Stage IV
(cold load)"; LC3
```

Calculation No.8 – Verification calculation with determination of the design load on spring hangers/supports

```
&LC MOD='$OPER' TYPE='DSGN' Note="Determination of design load" ; LC1
&LC MOD='$OPER' TYPE='OPER_A' PEND='YES' FRIC='YES' Note="Stage II (hot
load)"; LC2
&LC MOD='$OPER' TYPE='SUST_C' Note="Stage I"
; LC3
&LC MOD='$COLD' TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Stage IV
(cold load)"; LC4
```

In this calculation it is assumed that the spring characteristics are known and it is required to determine the working load on hangers. Therefore, the calculation on rigid supports will be performed first (LC1, *TYPE = 'DSGN'*). Then a set of command follows, which is similar to the previous examples.

Any of the calculations listed above can be supplemented by the specification for *analysis under dynamic or seismic load*. For this purpose, the following command shall be added to the specification for analysis:

```
&LC MOD='$OPER' TYPE='MODAL' Note="Modal analysis" ; LC4
```

In accordance with this command, the modal analysis (determination of natural frequencies and mode shapes of the piping system) will be performed. The conditions for the execution of dynamic analysis are set by the DYN, FMAX, FMESH and MCOM parameters of the DCASE and CTRL commands. The conduction of modal analysis implies a linear system; and, therefore, all piping system supports are interpreted as two-way ones, and the gaps in supports are not taken into account.

In case when the response spectrum method is used (DYN='RSM), the seismic load is specified by a set of floor response spectrums (SPEC command).
According to the results of analysis, the stress of category $(\sigma)_2$ is determined for the following combinations of loads NOC + SSE (S2_MRZ), NOC + OBE (S2_PZ1) - for piping systems of the first seismic category and NOC + OBE (S2_PZ2) – for piping systems of the second seismic category :

```
&RES='S2_MRZ' LS="LC2 + LC4" Note="S2 stress (SSE)"
&RES='S2_PZ1' LS="LC2 + 0.5*LC4" Note="S2 stress (OBE)"
```

It should be noted that the sequence of location of the reference loads in the parameters for stress computation, *S2_MRZ, S2_PZ1, S2_PZ2.* is significant, namely: the load calculated for Stage I shall be indicated first (LC2 in this example), and the next load shall be a reference to the modal analysis (LC4). In the example given above, it is assumed that the OBE intensity is equal to a half of SSE; therefore, the factor of 0.5 shall be indicated with the reference to seismic load. In case when the seismic load for OBE and SSE has a different form, it is necessary to perform 2 separate calculations with assignment of their own spectrums for each of the variants.

Calculation of the "hot" hydraulic test mode

Below the specification for analysis and post-processing of the results is given for the "hot" hydraulic test , which is similar to Stages 5 and 6 in the RAMPA93 software program.

1. By means of the OPVAL command, set the values of pressure, temperature and weight of working fluid corresponding to the hydraulic test mode:

```
OPVAL 'OPER' ; normal operating conditions
& 'LG1' P= 4.0 T= 350 CSG= 0
OPVAL 'TEST' ; hydraulic test
& 'LG1' P= 5.0, T= 70, CSG= 1 ; csg= 1 -> weight of water is added!
```

2. if necessary, set the values of temperature displacements of fixed supports:

```
10:ANC
& 'OPER' DX= 0 DY= 100 DZ= 0
& 'TEST' DX= 0 DY= 20 DZ= 0
```

3. Add the following commands to the specification for analysis(calculation No.2 should be selected as the basic variant):

```
Existing commands:

SOLV "Verification calculation No.2 + hydraulic test"
&LC MOD='$OPER' TYPE='OPER_A' PEND='YES' FRIC='YES' Note="Stage II"
;LC1
&LC MOD='$OPER' TYPE='SUST_C' Note="Stage I" ;LC2
&LC MOD='$COLD' TYPE='OPER_B' PEND='YES' FRIC='YES' Note="Stage IV"
;LC3


Additional commands:

&LC MOD='$TEST' TYPE='OPER_B' PEND='YES' FRIC='YES' ; LC4 /P+W+T+D
&LC MOD='$TEST' TYPE='SUST_C' ; LC5 /P+W
```

```
Existing commands:

POST

&RES='S2_NUE' LS="LC2" Note="S2 stress (OPER)"
&RES='SRK' LS="LC1-LC3" Note="SRK stress (COLD->OPER)"
&RES='SAF' LS="LC1-LC3" Note="SAF stress (COLD->OPER)"

&RES='SUPP' LS="LC1" Note="Loads in the operating state"
&RES='SUPP' LS="LC3" Note="Loads in the cold state"

&RES='DISP' LS="LC1" Note="Full movements (OPER)"
&RES='DISP' LS="LC2" Note="Movements from weight (OPER)"
&RES='DISP' LS="LC1-LC3" Note="Visible movements (COLD->OPER)"

Additional commands:

&RES='S2_HDR' LS="LC5" Note="S2 stress (TEST)"
&RES='SRK' LS="LC4-LC3" Note="SRK stress (COLD->TEST)"
&RES='SAF' LS="LC4-LC3" Note="SAF stress (COLD->TEST)"

&RES='SUPP' LS="LC4" Note="Loads at hydraulic test"

; if movements are required

&RES='DISP' LS="LC4" Note="Full movements (TEST)"
&RES='DISP' LS="LC5" Note="Movements from weight (TEST)"
&RES='DISP' LS="LC4-LC3" Note="Visible movements (COLD->TEST)"
```

HIGH-TEMPERATURE PIPING SYSTEMS

The specification for analysis (SOLV command) for high-temperature piping systems is the same as described above. Below only post-processor commands for stress computation are considered.

Since the creep-rupture strength is used for computing the nominal allowable stress [σ] for high-temperature piping systems (SR parameter of the MAT command), it is necessary to check if the corresponding data are present in the properties of material.

The postprocessor command for computation of the $(\sigma)_2$ stress category is completely similar to low-temperature piping (see above). In computation of allowable stress for operating modes implying short-term loads such as hydraulic tests (S2_HDR), AOC (S2_NNUE) as well as seismic loads (S2_MRZ , S2_PZ1, S2_PZ2), it is possible, due to the User's decision, to avoid the account of the creep-rupture strength (see description of the SN_T parameter, CTRL command).

As the formulae for computation the stress $(\sigma)_2$ include the strength reduction coefficient for the transverse welded joint φ_w, then the results of analysis influence upon the WLD_CHK parameter and the WLD command.

In order to compute the stress of category $(\sigma)_{RK}$, it is necessary to specify the following two post-processor commands:

```
&RES='FORC' LS="LC1-LC2" OUT='NO' Note="Internal forces (Stage 2)" ; LS2
&RES='SRK' LS="LC2+LS2" Note="Srk stress" ; LS3
```

LS2 determines the internal forces from the thermal expansions, which, in accordance with the Codes, can be determined with the account of gradual decrease (relaxation) in time due to creep.

The combination of loads to be specified in LS3 includes the forces as per Stage I (LC2) and the forces computed at the previous stage (LS2). The averaging factor for thermal stresses shall be taken equal to 0.5*χ where χ is set in input data by means of the CREEP command. The coefficient $\chi_Э$ being contained in the formulae for computation of stress from the ovality of bends shall be determined according to the following formula:

$$\chi_Э = HI\_E*\chi,$$

where the conversion coefficient HI_E is determined in the CTRL command.

The sequence of location of the reference loads in the parameters for computation the SRK stress for high-temperature piping systems shall be the same as given above.

The stress of category $(\sigma_{aF})_K$ for high-temperature piping systems is computed similar to low-temperature piping systems:

```
&RES='SAF' LS="LC1-LC3" Note="Saf stress" ; LS3
```

The combination of loads specified above determines the transient from the operating state (LC1, Stage II) into the cold state (LC3, Stage IV).

Printout of the loads on supports with the account of relaxation and cold springing

1. The loads on supports with the account of relaxation and cold springing shall be output for anchors and supports.
2. The loads with the account of relaxation of temperature forces shall be computed according to the following formula:

$$LS_{hot} = LC2 + (LC1 - LC2)*(1-\delta)$$

where
LC2 - loads from weight in the operating state;
LC1 - loads from weight and temperature in the operating state
δ - thermal stress relaxation coefficient (DELTA parameter in the CREEP command)
3. The loads on supports in the cold state with the account of cold springing shall be computed according to the following formula:

$$LS_{cld} = LC3 - (LC1 - LC2)*\delta*(E_{LC3}/E_{LC1})$$

where
LC3 - loads from weight in the cold state;
(E_{LC3}/E_{LC1}) - reduction of loads to the cold modulus of elasticity.

In order to implement the above-mentioned formulae, the following commands shall be added to the specification for post-processing with the use of H_REL and C_REL identifiers:

```
&RES='SUPP' LS="LC1-LC2"  OUT  = 'NO'    Note="Loads from temperature"
           ; LS10
&RES='SUPP' LS="LC2+LS10" RULE = 'H_REL' Note="loads in the operating
state, relaxation"          ; LS11
&RES='SUPP' LS="LC3-LS10" RULE = 'C_REL' Note="loads in the cold state,
cold springing"    ; LS12
```

Analysis according to the Russian Boiler Code (RD)

Stress Analysis according to the Russian Boiler Code RD 10-249-98 "Strength Design Code for Stationary Boilers and Steam and Hot Water Pipelines" assumes execution of the similar calculation stages as described above for PNAE Code.

The specifications for different kinds of analyses (SOLV command) performed according to Boiler Code are the same as for PNAE Code. Differences exist only in for results post-processing in stress calculations :

Computation of the effective stress at Stage I of the analysis:

```
&RES='S_I' LS="LC2" Note="S_I stress (Stage 1)" ; LS1
```

As the reference load LC2, the load for Stage I shall be specified.

Computation of stress at Stage II of the analysis (high-temperature piping systems):

```
&RES='FORC' LS="LC1-LC2" OUT='NO' Note="Internal forces (Stage 2)" ; LS2
&RES='S_II' LS="LC2+LS2" Note="S_II stress (Stage 2)" ; LS3
```

The specification for *S_II* stress analysis shall be formed similar to SRK for high-temperature piping systems according to PNAE.

In the example given, the LS2 calculation determines the internal forces from the thermal expansions.

The combination of loads to be specified in LS3 includes the forces as per Stage I (LC2) and forces from the thermal expansions (LS2) computed by the previous command. The averaging coefficient for thermal stresses shall be taken equal to $0.5*\chi$ where χ is set in input data by means of the CREEP command.

The sequence of location of the reference loads in the parameters for computation the S_II stress shall be the same as given above.

Computation of the stress at Stage III of the analysis (low-temperature piping systems, fatigue strength assessment):

```
&RES='S_III' LS="LC1-LC3" Note="S_III stress (Stage 3)" ; LS2
```

The specification for *S_III* stress analyses shall be formed similar to SAF for PNAE: the transient from the cold state into the operating state is evaluated: LC1 (Stage II) – LC3 (Stage IV).

It should be noted that in accordance with RD (Item 5.2.7, Fig. 5.15) the permissible stress $[\sigma_a]$ for Stage III depends both upon the type of material (carbon or austenite steels) and from the type of element being evaluated (straight pipe or bend/tee). The data for these stresses shall be given in the FAT command referenced in the *FAT* and *FAT_B* parameters of the MAT command.

Determination of the stress at Stage IV of the full analysis (high-temperature piping systems):

```
&RES='FORC' LS="LC1-LC2" OUT='NO' RULE = 'REF' Note="Internal forces
(Stage 4)" ; LS4
&RES='S_IV' LS="LC3-LS4" Note="S_IV stress (Stage 4)"           ; LS5
```

At Stage IV for high-temperature piping systems the check is carried out for the absence of plastic overloads in the cold state. For computation of the S_IV stress, one auxiliary combination of loads

(Line LS4) is required, which determines the difference between the full load (LC1, Stage II) and the load LC2 corresponding to Stage I: LC1 – LC2. The *RULE* = *'REF'* parameter is specified for reduction of the combination of internal forces computed to the "cold" (reference) modulus of elasticity. The next line (LS5) determines the calculation of the S_IV stress on the basis of forces and moments computed as the difference between the forces in the cold state (Stage IV) and the loads computed in LS4 multiplied by the relaxation coefficient δ (DELTA parameter, CREEP command).

The sequence of location of the reference loads in the parameters for computation of the S_IV stress shall be the same as given above.

The recommended sequence of commands for the specification for analysis including the ***hydraulic test*** mode:

Low-temperature piping systems:

```
SOLV "Verification calculation with specified characteristics of spring
hangers/supports  + hydraulic tests"
&LC MOD='$OPER' TYPE='OPER_A' PEND='YES' FRIC='YES' Note="Stage II (full
load)"      ; LC1
&LC MOD='$OPER' TYPE='SUST_C' Note="Stage I"
                   ; LC2
&LC MOD='$COLD' TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Stage IV
('cold load')"; LC3
&LC MOD='TEST'  TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Hydraulic
test mode"; LC3    ; LC4
&LC MOD='$OPER' TYPE='SUST_C' Note="Stage I for HT"
                   ; LC5
POST
&RES='S_I'    LS="LC2"        Note="S_I stress (Stage 1)"          ; LS1
&RES='S_III'  LS="LC1-LC3"    Note="S_III stress (Stage 3)"        ; LS2
&RES='S_H'    LS="LC5"        Note="S_H stress (Stage 1)"          ; LS3
&RES='DISP'   LS="LC2"        Note ="Weight movements"             ; LS4
&RES='DISP'   LS="LC1-LC3"    Note ="Visible movements"             ;
LS5
&RES='SUPP'   LS="LC1"        Note="Loads in the operating state"
 ; LS6
&RES='SUPP'   LS="LC3"        Note="Loads in the cold state"        ;
LS7
&RES='SUPP'   LS="LC4"        Note="Loads at HT"                  ; LS8
```

High-temperature piping systems :

```
SOLV "Verification calculation with specified characteristics of spring
hangers/supports (#2) + hydraulic tests"
&LC MOD='$OPER' TYPE='OPER_A' PEND='YES' FRIC='YES' Note="Stage II (full
load)"      ; LC1
&LC MOD='$OPER' TYPE='SUST_C' Note="Stage I"
                   ; LC2
&LC MOD='$COLD' TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Stage IV
('cold load')"; LC3
&LC MOD='TEST'  TYPE='OPER_B' PEND='YES' FRIC = 'YES' Note="Hydraulic
test mode"; LC3    ; LC4
&LC MOD='$OPER' TYPE='SUST_C' Note="Stage I for HT"
                  ; LC5
POST
&RES='S_I'  LS="LC2"              Note="S_I stress (Stage 1)"     ; LS1
```

```
&RES='FORC' LS="LC1-LC2" OUT='NO' Note="Internal forces (Stage 2)"
; LS2
&RES='S_II' LS="LC2+LS2"          Note="S_II stress (Stage 2)"    ; LS3
&RES='FORC' LS="LC1-LC2" OUT='NO' RULE = 'REF' Note="Internal forces
(Stage 4)"      ; LS4
&RES='S_IV' LS="LC3-LS4"          Note="S_IV stress (Stage 4)"    ; LS5
&RES='S_H'  LS="LC5"        Note="S_H stress (Stage 1)"          ; LS6
&RES='DISP' LS="LC2"        Note ="Weight movements"             ; LS7
&RES='DISP' LS="LC1-LC3"    Note ="Visible movements"             ; LS8
&RES='SUPP' LS="LC1"        Note="Loads in the operating state"       ;
LS9
&RES='SUPP' LS="LC3"        Note="Loads in the cold state"        ; LS10
&RES='SUPP' LS="LC4"        Note="Loads at HT"                   ; LS11
```

15 Appendix VI

Appendix VI. Spring hangers design

[Basic definitions](#)

[Execution of analysis, errors and warnings](#)

[Results of analysis](#)

[SH.DBS file structure](#)

[Selection of springs from LISEGA Catalog](#)

Basic definitions

design load (P_d) is a target hanger load that should balance weight of the piping. Design load is calculated as reaction of the vertical rigid restraint installed in the location of the designed spring hanger (the stiffness of these restrains is set by RGD_SPR parameter). For this case it is assumed that piping is subjected to sustained loads only (weight of insulated pipe with medium content). Sometimes Design Load is also referred to as Hot Load due to the fact that for most cases a weight balance should be achieved in the operational (hot) state.

operating or Hot load (P_h) – spring hanger/support reaction in Hot state of piping. By value Hot load shall be close to the design load. The difference in the values between these reactions can be caused by the account of nonlinearities(friction forces, one-way supports, swing effect).

installation (cold) load (P_c) – reaction of the spring hanger/support in the cold state (at the installation temperature when piping is empty).

Hanger Travel (u_0) characterizes vertical piping movement that could be measured in the location of spring hanger support between installed (cold) and operating (hot) positions

theoretical installation load (R_0): physically it's a load that corresponds to the spring preset out of the piping (such procedure is used in Russian practice for "one-stage" spring presetting, see REF 7). Numerically it can be described as: $R_0 = P_H + k_S * u_0$, where: ks is a spring rate, u_0 - hanger travel.

Spring identification comes from the corresponding spring table (catalog). The spring identification consists of 3 fields: 'size/travel/type' where "size" corresponds to the maximum load capacity of the spring, "travel" corresponds to the maximum allowable travel, the hanger type ("type") is used only in those catalogs, in which various springs are used in different design versions of spring hangers/supports (for example, LISEGA catalog).

maximum travel of the spring – spring travel between the minimum and maximum loads.

chain – set of springs connected sequentially

chain structure – number characterizing the maximum operating travel of the chain. In Russian standards, the springs with working travel of 70 mm (Z1) and 140 mm (Z2) mm are used. Then the coupling of several springs into a single assembly (chain) can be described as follows:

Z3 = Z1 + Z2
Z4 = Z2 + Z2
Z5 = Z1 + Z2 + Z2
Z6 = Z2 + Z2 + Z2 … и т.д.

For international standards, each consequent number of the chain structure corresponds to a spring with the larger operating travel. In selecting the springs, the concepts of maximum and minimum chain structures (ZMAX и ZMIN) are introduced, which corresponds to limitation of the maximum and minimum values of the working travel.

load safety factor (PFAC) – ratio of the maximum permissible load on spring, P_{MAX}, to the load, equal to max(P_h, P_c). In selecting the springs, the program provides the ratio of these values being equal or greater than the PFAC value. In the course of designing. the safety factor allows compensating the uncertainties related to the difference between the nominal weight of pipe taken in the analysis and the actual data, which can become known only during installation. Hence the introduction of load safety factor allows to correct the spring pre-load during installation, if necessary. On the other hand, the load safety factor enables the spring to perceive additional loads (for example, seismic loads).

Load variation factor (PVAR) – value to be determined by the following relation:

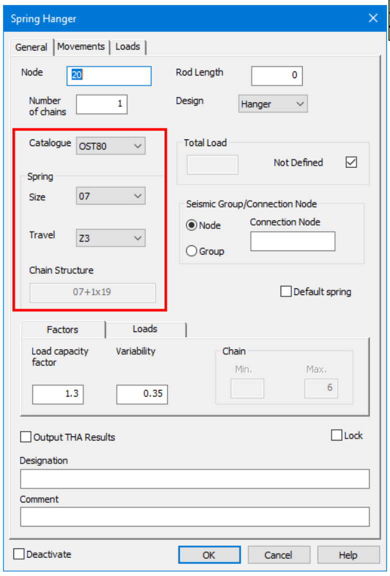
$$PVAR = \frac{|P_d - P_c|}{P_d}$$

The SPR command is used to define the spring support/hanger in dPIPE. The spring is identified by reference to the table (catalogue) and the spring size. For example, for a spring manufactured in accordance with OST 108.764.01-80 with a load capacity of 2005 kgf and a travel of 210 mm, the following command will be used in the dPIPE input data:

```
20: SPR nc = 1, id
   = '07/Z3' tbl
     'OST80'
```

OST 108.764.01-80

λ (мм)	Pmax (кгс) 128	278	534	816	1190	1666	2005	2686	3325	4080	4955	5960	
70	01	02	03	04	05	06	07	08	09	10	11	12	Z1
140	13	14	15	16	17	18	19	20	21	22	23	24	Z2
210	01+13	02+14	03+15	04+16	05+17	06+18	07+19	08+20	09+21	10+22	11+23	12+24	Z3
280	13+13	14+14	15+15	16+16	17+17	18+18	19+19	20+20	21+21	22+22	23+23	24+24	Z4
350	01+13+13	02+14+14	03+15+15	04+16+16	05+17+17	05+18+18	06+19+19	08+20+20	09+21+21	10+22+22	11+23+23	12+24+24	Z5
420	13+13+13	14+14+14	15+15+15	16+16+16	17+17+17	18+18+18	19+19+19	20+20+20	21+21+21	22+22+22	23+23+23	24+24+24	Z6
	01	02	03	04	05	06	07	08	09	10	11	12	dPipe



MVN 049-63

λ (mm)	Pmax (kgf) 97	197	292	514	815	1155	1562	2050	2420	3420	4620	5720	6400	
70	11	12	13	14	15	16	17	18	19	20	24	25		Z1
140	01	02	03	04	05	06	07	08	09	10	21	22	23	Z2
210	01+11	02+12	03+13	04+14	05+15	06+16	07+17	08+18	09+19	10+20	21+24	22+25		Z3
280	01+01	02+02	03+03	04+04	05+05	06+06	07+07	08+08	09+09	10+10	21+21	22+22	23+23	Z4
350	01+01+11	02+02+12	03+03+13	04+04+14	05+05+15	06+06+16	07+07+17	08+08+18	09+09+19	10+10+20	21+21+24	22+22+25		Z5
420	01+01+01	02+02+02	03+03+03	04+04+04	05+05+05	06+06+06	07+07+07	08+08+08	09+09+09	10+10+10	21+21+21	22+22+22	23+23+23	Z6
	01	02	03	04	05	06	07	08	09	10	11	12	13	dPipe

Run of analysis, errors and warnings

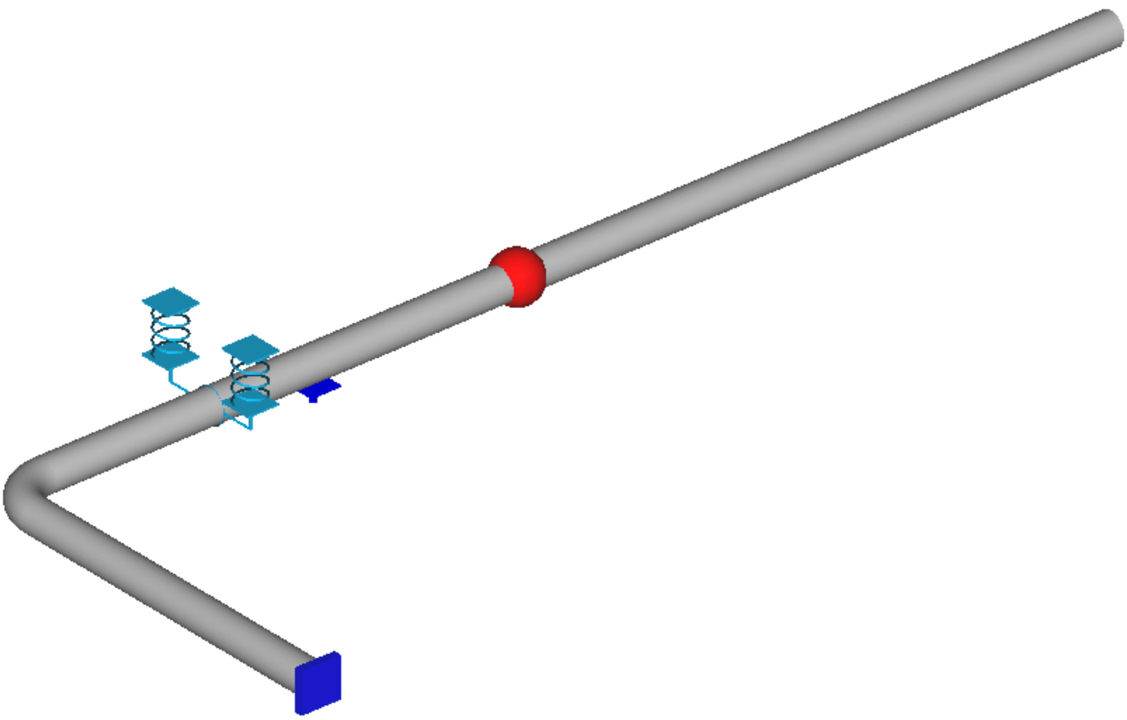
In order to select the characteristics of springs for hangers (supports), it is necessary to specify the "starting" set of computations in the specification for analysis, which is described in Appendix V. The program performs the selection of spring characteristics depending upon the spring tables specified (SDEF command), load safety factor and load variation factors (*PFAC and PVAR* parameters of the SPR command) as well as the values of minimum and maximum chain structures (ZMAX and ZMIN). In case of successful completion of the "starting" set of computations, the program performs the execution of main analysis stages using the spring characteristics computed. In addition, a file of the type: <model name>.dp5_ will be created in the working directory, which contains the data for hangers with computed characteristics of springs (further this file can be renamed into <model name1>.dp5 and used as the basic file for subsequent computations).

In case of unsuccessful completion of the starting set of computations, the program will stop its execution and will inform about the cause, according to which the selection of spring characteristics has failed. In order to analyze the causes for unsuccessful selection of springs, then intermediate results can be viewed in the listing of results or in the PIPE3DV program. After opening the PIPE3DV program and "activation" of the hangers, the red exclamation mark will be displayed in the upper right corner, and the "problem" supports will be flashing (the flashing can be activated - deactivated by the following combination of keys: ***ALT-E***). Having placed the cursor to the hanger, clicking the right button of the mouse and selected the "Node parameters" item from the context menu, one can view information about the intermediate results.

Below most typical situations related to the failures during spring design are given and commented:

Example 1

Diagram:



Program messages:

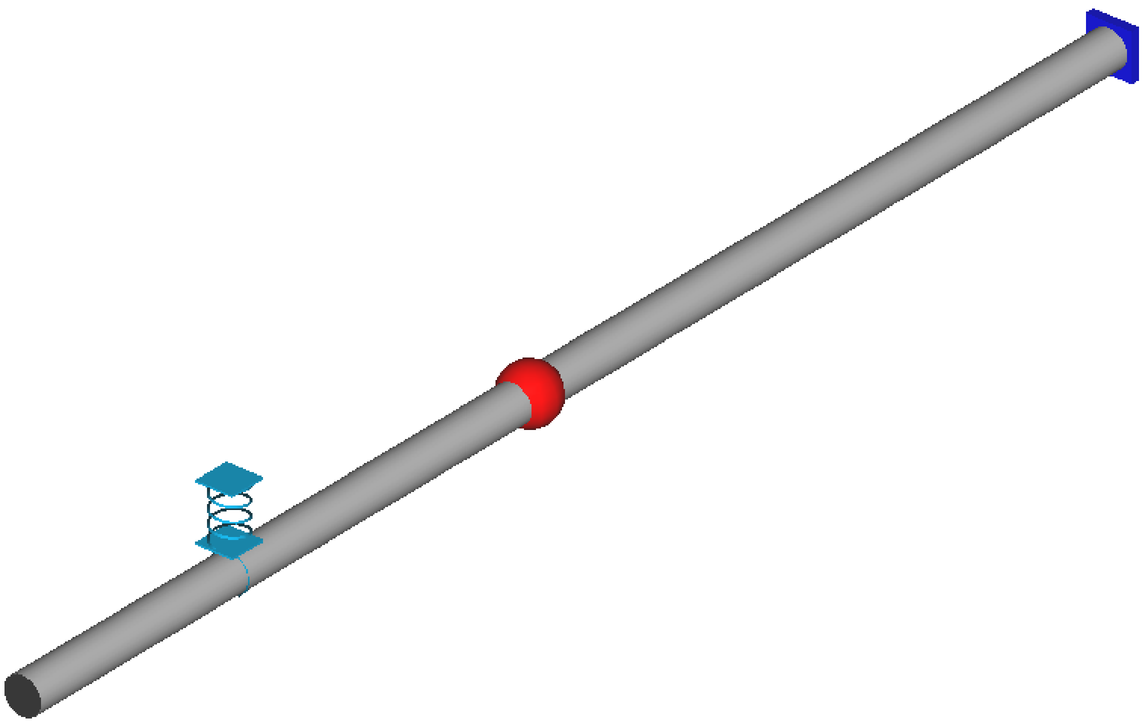
```
File Model_spring.bin was created successfully.
Solve > Op 1(R): W                       -> done
Error: spring load out of working range


dPIPE 5 Post  Build 231213
 POST: Design Load for spring hanger could not be defined
 POST: See results
```

Comment:

Design Load could not be calculated ("restrained weight" Load Case, TYPE = 'DSGN'). It could happen if the reaction of the hanger turned out to be negative, i.e. a wrong place for spring hanger/support has been selected.

Example 2

Diagram:



Program messages:

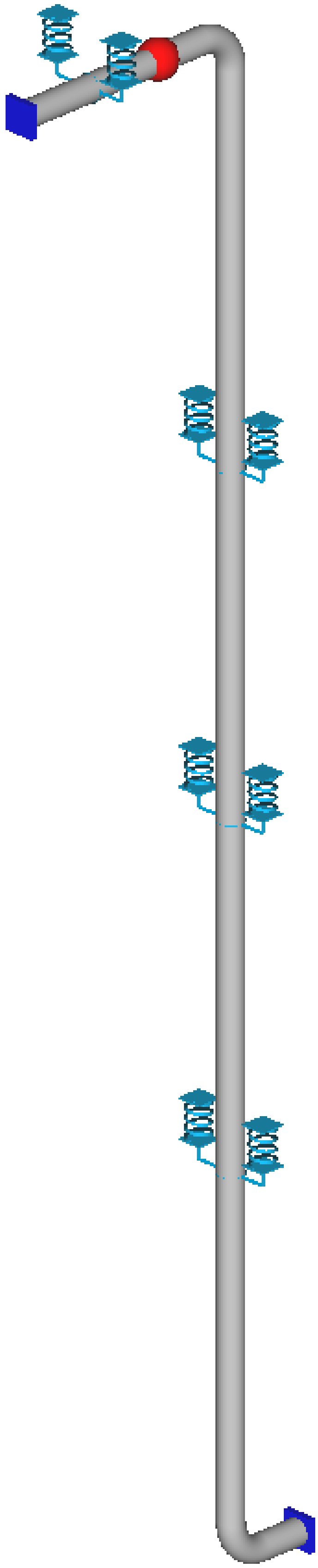
```
File Model_spring_straight.bin was created successfully.
Solve > Op 1(R): W                       -> done
Solve > Op 1(A): W+P+T+D                 -> done
Error: spring load out of working range


dPIPE 5 Post  Build 231213
 POST: Design Load for spring hanger is out of range
 POST: See results
```

Comment:

In this example, a very large weight of the valve has been specified intentionally, and the program cannot choose a spring with the required load capacity. Normally, in order to solve such kind of problems it is usually sufficient to increase the number of springs or install an additional supports.

Example 3

Diagram:



Program messages:

```
Solve > Op 1(R): W                    ->  1/1
Solve > Op 1(R): W                    -> done
Solve > Op 1(A): W+P+T+D              ->  1/1
Solve > Op 1(A): W+P+T+D              -> done
Solve > Cold(B): W+P+T+D              ->  1/1
Solve > Cold(B): W+P+T+D              -> done
Error: spring load out of working range



dPIPE 5 Post  Build 231213
 POST: Work travel for spring hanger is out of range (Pref > Pmax)
 POST: See results
Stop - Program terminated.
```

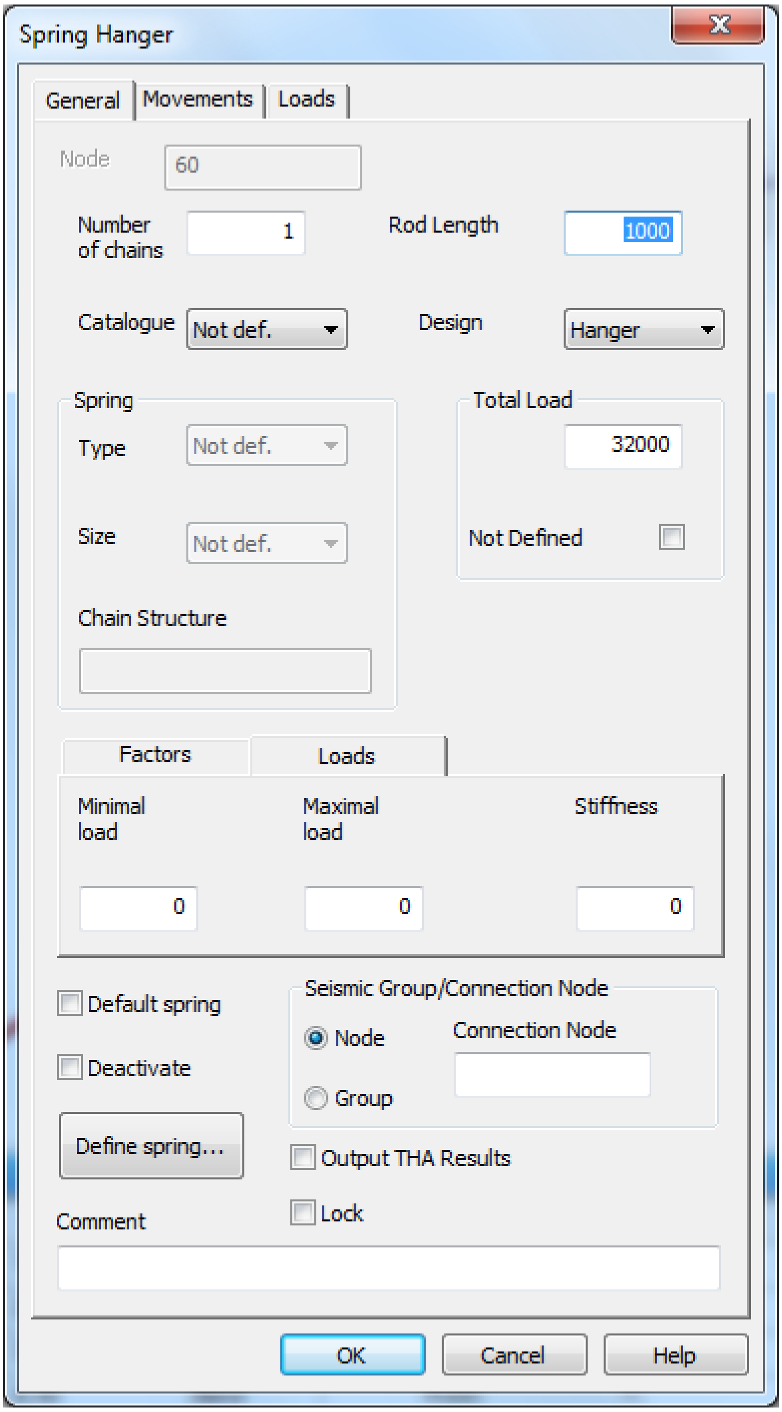
Comment:

In this example, it has appeared impossible to select a spring for the reference load (Cold Load in the given case) . At the third LC of starting computations the program checks the following condition:

PMIN < Pref < PMAX/PFAC

In case when this inequality is not valid, the program will try to select a spring with PFAC = 1. If it fails, then the program will terminate the execution of calculations with an error message being issued.

Such situations. as a rule, arise at large vertical temperature movements of the piping system. This problem can be solved only after analysis of the intermediate results: if the working load P_h on the hanger is close to the maximum and the piping being cooled moves down, then it is necessary to unload this hanger either by increasing the number of springs (NC) or by installing an additional supports carrying the weight load. Another method is to enable the program to select more flexible springs with a large working travel (for this purpose it is necessary to increase the ZMAX parameter). If no one of these solutions is satisfactory, then it is possible to use constant loads hangers (for example, from LISEGA catalog). For simulating such hangers, it is necessary to select a "user" spring in the program with the "zero" stiffness and required working load:



If in selecting the springs it turns out impossible to provide one of the required parameters (load safety factor *PFAC* or required variability *PVAR*), then the program will issue a warning and continue running:

```
Solve  > Op 1(R): W                    -> done
Solve  > Op 1(A): W+P+T+D+F            -> done
Warning: lift-off from one-way supports
Solve  > Cold(B): W+P+T+D+F            -> done
Warning: spring load variation > desired
Solve  > Op 1(B): W+P+T+D+F+FR+SW      -> done
Warning: lift-off from one-way supports
Solve  > Op 1(C): W+P                  -> done
Solve  > Cold(B): W+P+T+D+F+FR+SW      -> done


dPIPE 5 Post   Build 231213
 POST: LS1     S2_NUE           LC5  SUM
 POST: LS2     SRK          LC4-LC6  SUM
 POST: LS3     SAF          LC4-LC6  SUM
 POST: LS4     DISP             LC5  SUM
 POST: LS5     DISP         LC4-LC6  SUM
 POST: LS6     SUPP             LC4  SUM
 POST: LS7     SUPP             LC6  SUM
 POST: LS8     FORC         LC4-LC5  SUM
```

In the example given, the message ***"Warning: spring load variation > desired"*** means that it failed to achieve fulfillment of the variability conditions (PVAR) for one or several hangers.

The message ***"Warning: spring load safe factor < desired"*** means that it failed to achieve fulfillment of the conditions for load safety factor (PFAC) for one or several hangers.

The message ***"Warning: lift-off from one-way support"*** means that at the corresponding stage of analysis one of the one-way supports has uplifted and cannot bear the load.

Further, in the course of viewing the results in PIPE3DV, the "problem" supports will be flashing, and the following message will be placed in the listing of results with summarized tables opposite these supports.

Results of analysis

In case of successful completion of the analysis, the information about spring hangers/supports will be printed out in the following tables: "Spring Hanger Design Data." (file <>*.sup*). In this table, only the data for spring hangers/supports referenced to the catalogs included in the file sh.dbs will be printed. A separate table is printed for each catalog. For hangers from Russian standards (OCT, MBH), the table is printed in the following form:

```
>>>  Spring Hanger Design Data. Catalogue  OCT 108.764.01-80
==========================================================================================
Sup N      NC   Chain        Springs Heights              Support's Loads        Movements        ALPHA
(node)          struct. H_free  H_hot  H_cold  H_inst  P_hot   P_cold  P_seis    DX     DY     DZ
==========================================================================================
N1          2   1*18     369     277    258     257     21.41   25.88              4     -1     19  0.1
------------------------------------------------------------------------------------------
N2          1   1*06+    201     165    157     154      8.31   10.37             15      5     26  0.9
                1*18     369     298    280     275
==========================================================================================
Notes:
NC          - number of springs
H_free      - unloaded spring height, mm
H_hot       - spring height in hot state, mm
H_cold      - spring height in cold state, mm
H_inst      - spring height before installation, mm
P_hot       - hot load, kN
P_cold      - cold load, kN
P_seis      - seismic load, kN
DX, DY, DZ  - transient movements from cold to hot state,mm
ALPHA       - angularity (swing from the vertical), deg
```

The values of H_hot., H_cold., P_hot and P_cold. are printed in accordance with the LOAD_HOT and LOAD_COLD parameters. Similarly the computation of "movements" is performed. The value of H_inst is recalculated according to the load R_0 determined for LC with the type of 'OPER_A' encountered first in the analysis:

$$R_0 = P_h + \lambda_{ii} * k_s,$$

where k_s is the spring stiffness, λ_{ii} is the spring travel to be computed in the analysis of type 'OPER_A'

For international standards this table is printed out in the following form:

```
>>>  Spring Hanger Design Data. Catalogue  Lisega cat.2010 Type 21/25/29/20
==========================================================================================
Sup N      NC  Spring     Type          Spring Travel     Support's Loads         Movements     ALPHA
(node)                                   Dh     Dc     P_hot   P_cold  P_seis   DX     DY    DZ
==========================================================================================
N1          2   5/2        21 25         30     50     10.70   13.27             4     -2    19   0.1
------------------------------------------------------------------------------------------
N2          1   5/3        21 25         25     53      8.32   10.17            15      4    28   0.9
------------------------------------------------------------------------------------------
N3          2   5/2        21 25         22     36      9.62   11.52            26      7    14   0.8
------------------------------------------------------------------------------------------
N4          1   5/1        21 25         16     13     10.87   10.00            40      5    -3   2.8
------------------------------------------------------------------------------------------
N5          1   5/1        21 25         23     22     12.68   12.48            67     24    -1   5.9
------------------------------------------------------------------------------------------
N6          1   5/1        21 25         26     31     13.56   14.94            91     44     5   8.6
------------------------------------------------------------------------------------------
N7          1   5/2        21 25         51     65     13.40   15.28            78     51    14   8.1
==========================================================================================
Notes:
NC          - number of springs
Dc, Dh      - relative spring travel, mm (as marked on the scale)
P_hot       - hot load per one spring, kN
P_cold      - cold load per one spring, kN
P_seis      - seismic load per one spring, kN
DX, DY, DZ  - transient movements from cold to hot state,mm
ALPHA       - angularity (swing from the vertical), deg
```

Unlike Russian standards, the values are printed in this table, which are normally indicated on the spring's name plate, e.g. load on the spring in the working (hot) and cold states as well as the position of movement indicator (deformation of spring). The same as in the previous table, the printing of values for the cold and hot states is performed in accordance with the LOAD_HOT and LOAD_COLD parameters. If these parameters are not defined, then the corresponding table columns are not completed.

```
>>>  Summary table for spring hangers loads
==============================================================================
SUP N     spring      P_des  P_oper   FS     var     DX     DY     DZ   ALPHA  load set
(node)
==============================================================================
N1        5/2         21.45   21.41   1.6     0       2     -7     20    0.2   LS006
                              26.53   1.5    24      -1     -5      1    0.1   LS007
------------------------------------------------------------------------------
N2        5/3          8.35    8.32   1.2     0      12     -5     34    0.7   LS006
                              10.17   1.5    22      -3     -9      6    0.5   LS007
------------------------------------------------------------------------------
------------------------------------------------------------------------------
N7        5/2         13.46   13.40   1.5     0      76     52     16    8.0   LS006
                              15.28   1.3    14      -2      0      2    0.2   LS007
==============================================================================
Notes:
P_des               - design load, kN
P_oper              - operational load,kN
FS                  - load safety factor
var                 - variability, %
DX, DY, DZ          - movements, mm
ALPHA               - angularity (swing from the vertical), deg

LS006               - Нагрузки в раб. состоянии (Этап 2)
LS007               - Нагрузки в хол. состоянии (Этап 4)
```

The number of lines for each support is determined by the number of corresponding directives in the POST (res='TYPE') command. The value of P_пр. is taken from the calculation having the type of 'dsgn' or from the calculation specified by the LOAD_DES parameter. The value of variability is determined for each state with respect to the design load. For combinations of loads including seismic ones, the variability value is not printed.

Individual tables for each srong hanger/support.

In addition to the tables mentioned above, for each flexible support in the <*model name*>*.sup* file, the table of the following form will be printed out:

```
>>>  Node:                     N1
     Support spring hanger, number of springs: NC = 2, Rod Length L = 2050 mm

     Spring :                  5/2 (LISEGA). (Maximal travel -  100 mm)

     Design load:                        21451 N
     Theoretical installation load:      26696 N
 -----------------------------------------------------------------------------------
                                 P       Pmax      Pmin            DISP    LOAD    Var
                                       (Newton)                     mm      kN      %
 -----------------------------------------------------------------------------------
  1 Нагрузки в раб. состояни    21405    40000     13320  1.61      30    10.7      0
  2 Нагрузки в хол. состояни    26531    40000     13320  1.51      50    13.3     24
 -----------------------------------------------------------------------------------
 types: 21 25

 Displacements:
 ---------------------------------------------------------------
                                    Dx        Dy        Dz      ALPHA
                                         (millimeters)           deg
 ---------------------------------------------------------------
  1 Нагрузки в раб. состоянии      2.37     -7.15     19.83     0.2
  2 Нагрузки в хол. состоянии     -1.29     -5.05      0.62     0.1
 ---------------------------------------------------------------
```

The same table is displayed in the PIPE3DV program in the course of viewing the "information about the node". The numbers printed in this table are determined according to the rules given above for similar values.

In case of successful selection of spring characteristics in the <*model name*>*.res* file the summarized table with all hangers will be printed out:

```
>>> Table   1 . Results of spring's selection (load for one spring)
 ------------------------------------------------------------------------------------------
 Node      NC    Spring      K      Pmin     Pmax     Pd     Ph     Pc     R0    TRAVEL  var    FS
 ------------------------------------------------------------------------------------------
 N1         2   06/Z2     116.7        0    16338   9309   9291  11525  11603     19     24   1.42
 N2         1   4/2        66.7     3330    10000   7246   7218   8990   9501     27     24   1.11
 ...
 N7         1   5/2       133.4     6660    20000  11691  11641  13516  13827     14     16   1.48
 ------------------------------------------------------------------------------------------

 Notes:
 NC     - number of springs
 K      - spring's stiffness, N/mm
 Pmax   - maximal load, N
 Pmin   - minimal load, N
 Pd     - design load, N
 Ph     - hot state load, N
 Pc     - cold state load, N
 R0     - theoretical presetting load, N
 TRAVEL - operation travel, mm
 var    - variability, %
 FS     - load safety factor: FS  = MIN ( Pmax/MAX(Ph,Pc); MIN(Ph,Pc)/Pmin )
```

And in case when errors arise, this table has the following form:

```
ERROR MESSAGES:
Fail to select spring matched to operational load

>>> Table   1 . Results of spring's selection (load for one spring)
-----------------------------------------------------------------------------------------------
Node       NC    Spring     K       Pmin    Pmax     Pd      Ph      Pc    R0      TRAVEL  var    FS
-----------------------------------------------------------------------------------------------
210         2    7/1               20000   60000   45304   45304                                1.32
70          2                         40  100000  219186  219186                                0.46  (FS !)
80          2    6/1               13330   40000   37000   37000                                1.08
90          2    7/1               20000   60000   60000   60000                                1.00
-----------------------------------------------------------------------------------------------
```

warning and error messages

for hangers, which characteristics were failed to determine the minimum and maximum spring loads are printed according to the corresponding table of the springs

SH.DBS file structure

The file contains the description of spring characteristics (spring tables) in accordance with the manufacturers' industrial standards. The table rows correspond to the maximum values of the "working travel" of spring, columns – to the maximum load, which the spring can perceive.

The following commands and parameters are used for description of the spring tables.

$TITLE *"Name" "Description"*:

"Name" – name of standard being used in the SDEF command (STAB parameter). *"Description"* – information about the standard.

$UNIT_LOAD *'load' coef* - command that specifies the conversion factor for the dimension of loads given in the catalog with respect to newtons;

$UNIT_DISP *'disp' coef* – command that specifies the conversion factor for the dimension of movements given in the catalog with respect to millimeters;

$LOAD_MIN $load_1$, $load_2$ ... $load_n$ – value of the minimum load for each of the spring type-sizes;

$LOAD_MAX $load_1$, $load_2$ ... $load_n$ – value of the maximum load for each of the spring type-sizes;

$WRK_RANGE $disp_1$, $disp_2$, ... $disp_k$ – values of maximum possible deformations of springs (working travel);

$LAB_SIZE $'size_1'$ $'size_2'$ ... $'size_n'$ – marks corresponding to the loads

$LAB_TYPE $'type_1'$ $'type_2'$ ... $'tyep_k'$ – marks corresponding to the spring deformations

$Z1_HEIGHT $hght_1$, $hght_2$ ... $hght_n$ – heights of springs with the working travel of 70 mm in a free state (to be used only for domestic standards: OST 108.764.01-80, MVN 049-63, OST 24.125.109-93)

$Z2_HEIGHT $hght_1$, $hght_2$ ... $hght_n$ – heights of springs with the working travel of 140 mm in a free state (to be used only for domestic standards: OST 108.764.01-80, MVN 049-63, OST 24.125.109-93)

$Z1_DESIGN $'lab_1'$ $'lab_2'$ ... $'lab_n'$ – marks for identification of springs with the working travel of 70 mm from domestic catalogs

$Z2 DESIGN $'lab_1'$ $'lab_2'$ ... $'lab_n'$ – marks for identification of springs with the working travel of 140 mm from domestic catalogs

$SPR_PTTRN $XXXXX_1$ $XXXXX_2$ …. $XXXXX_n$ – pattern to determine whether this spring type-size with the corresponding working travel exists: 0 – missing, 1 – present; the number of figures in the pattern shall correspond to the dimension of the $WRK_RANGE array

$TYPES $'type1_1'$ $'type1_2'$ ... $'type1_j'$ – marks for possible design versions of springs;

$TYP_PTRN X_1 X_2 ... X_j - pattern for the design versions of springs: 0 – this design version is available only for spring hangers; 1 - this design version is available only for spring supports;

$TYP_MASK – integer array with the size repeating the spring table. Each integer is a "bit mask" with available types of design versions for each type-size of springs

$PFAC – PFAC parameter recommended for this table

$PVAR – PVAR parameter recommended for this table

$ZMIN – ZMIN parameter recommended for this table

$ZMAX – ZMAX parameter recommended for this table

For each spring table the presence of the following commands: $TITLE, $UNIT_LOAD, $UNIT_DISP, $LOAD_MIN, $LOAD_MAX, $WRK_RANGE, $LAB_SIZE and $LAB_TYPE is mandatory. The rest of commands can be specified or not depending upon the corresponding catalog.

In this program version, the file SH.DBS contains the spring tables for the following standards:

Title	Description of standard
OST80	OST 108.764.01-80
MVN63	MVN 049-63
OST93	OST 24.125.109-93
LISEGA	LISEGA Catalog "Standard supports 2010", Product group 2, spring hangers, spring supports, Types: 21, 25, 29, 20
LISEGA_E	LISEGA Catalog "Standard supports 2010", Product group 2, spring hangers, spring supports, Types: 22, 26, 28
GRADIOR	GRADIOR "PIPE SUPPORTS CATALOGUE", Spring Supports: types 1-4 (hangers), 7 - supports
WTZNMAN	"Pipe Hangers and Supports" Catalogue , Series FHD, FHG, FHS and FDT for spring hangers, FSS, FSP - for spring supports
UNISONS	"Variable spring hanger" Unison eTech Catalogue, Springs VSS (30 mm travel)
UNISONML	"Variable spring hanger" Unison eTech Catalogue, Springs VSM (65 mm travel) and VSL (130 mm travel)
T2	Springs with travel 50, 100 and 200 mm and load capacity from 0 to 100 kN (table 12.2) from “TITAN2, Catalogue EN. Standard elements for piping supports for ISO 4200:1991 pipes”. Types 51, 52, 53
T2_XL	Springs with travel 50, 100 and 200 mm and load capacity from 53.3 to 400 kN (table 12.4) from “TITAN2, Catalogue EN. Standard elements for piping supports for ISO 4200:1991 pipes”. Types 54, 55, 56
T2_RU	Springs with travel 70 and 140 mm and load capacity from 0 to 58.5 kN (table 12.3) from “TITAN2, Catalogue EN. Standard elements for piping supports for ISO 4200:1991 pipes”. Types 51, 52

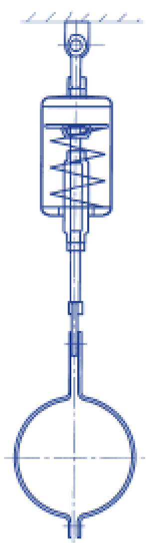
Selection of springs from LISEGA Catalog

Springs from LISEGA Catalog (product group 2, [REF 6]) differ by types (design versions), load capacity and by the range of spring's travels. The catalog contains 5 variants of the working travel: 50, 100, 200, 300 and 400 mm. In doing so, the last two variants *"The use of extra long springs is only to be recommended in limited cases because of the relatively large spring hysteresis"* [REF 6].

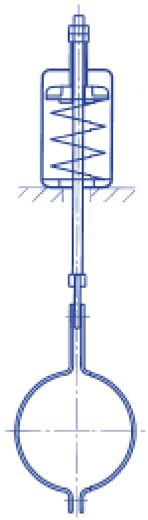
LISEGA offers the following types of springs:

For hangers (supports) with load capacity **up to 100 kN**

Type 21 and Type 25 are the most frequently used springs for hangers and differ by the design version:



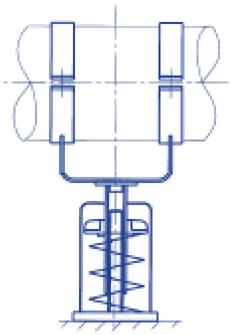
Type 21



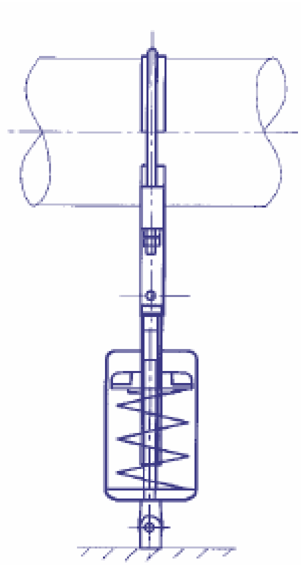
Type 25

Type 29 - is used for spring supports.

Type 20 - Angulating spring support. Unlike spring supports of Type 29, this design can perceive horizontal movements without lateral forces arising due to friction forces:



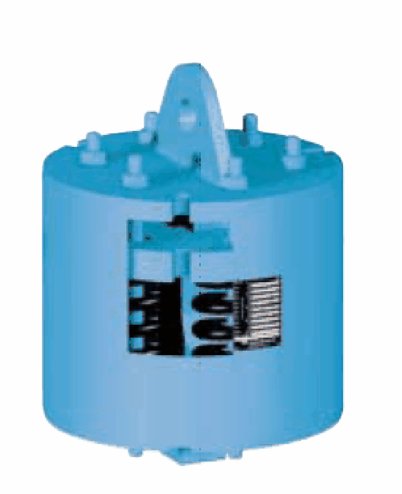
Type 29



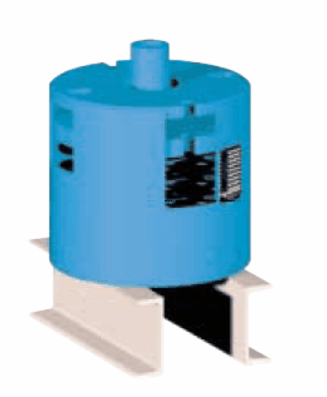
Type 20

For hangers (supports) with load capacity **from 160 kN to 400 kN** the spring assembly are used:

Type 22 – spring hangers (similar to Type 21):
Type 26 – spring hangers (seated) (similar to Type 25):
Type 28 – spring supports (similar to Type 29)



Type 22

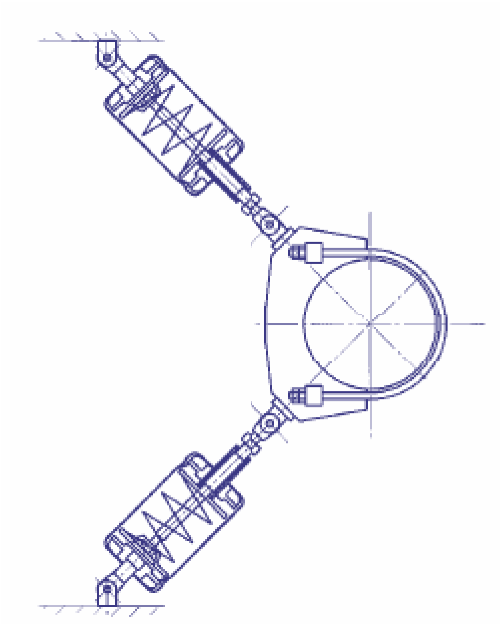


Type 26

Type 28

For these types of springs only the first 3 ranges of maximum working travels are possible (from 50 to 200 mm).

In addition to the spring types listed above, the catalog offers "sway braces" (Type 27) and spring hanger trapezes (Type 79). These types are not considered herein below.



Type 27

Type 29

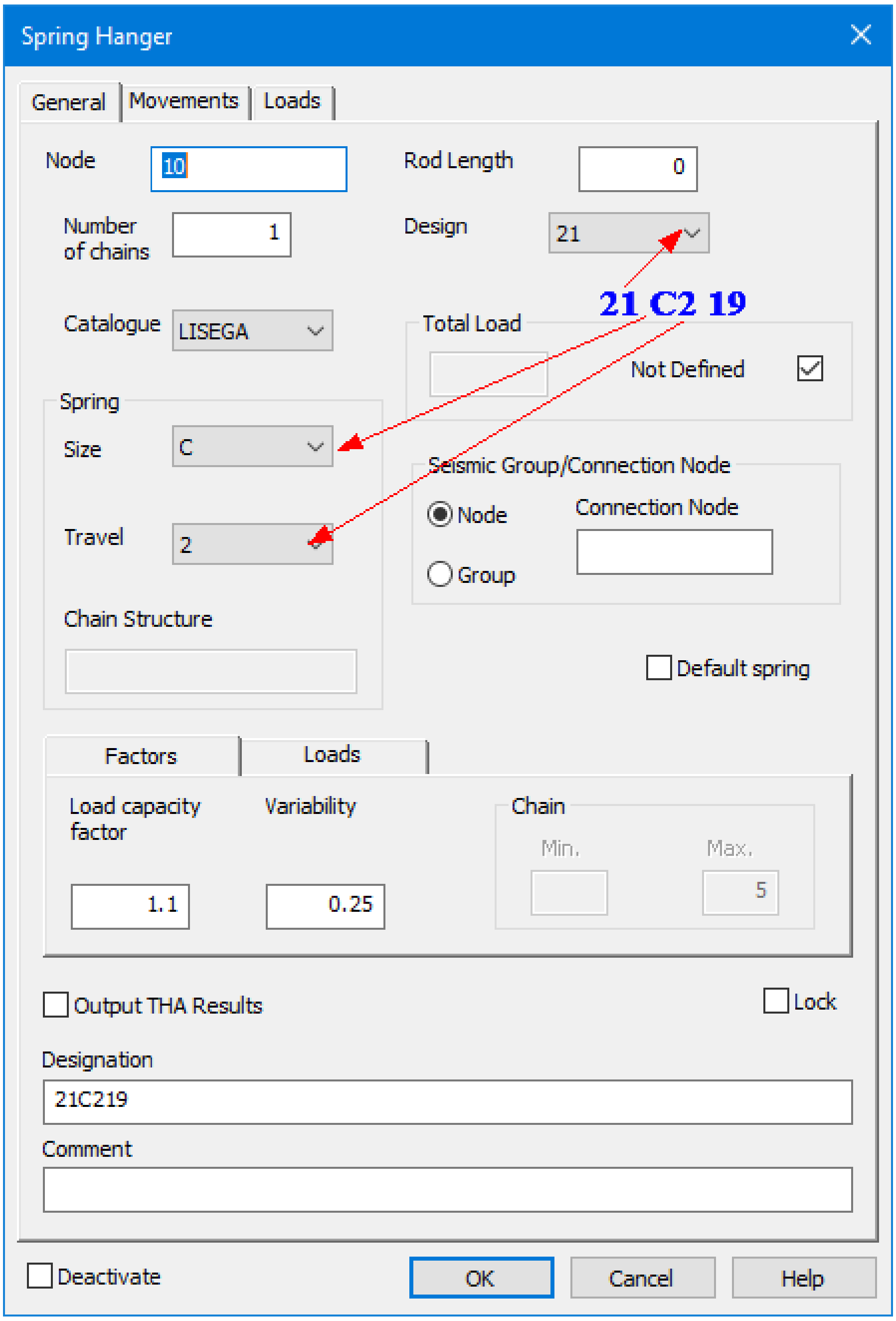
The springs of types 21, 25, 29 and 20 are included in the spring hanger database file (SH.DBS) under the "LISEGA" Table . Springs with increased load capacity (types 22, 26 and 28) are contained in "LISEGA_E" table.

The recommended values of parameters affecting the spring design when the LISEGA Catalog is used (SDEF and SPR commands) are as follows:

ZMIN = 1; ZMAX = 3; PFAC = 1.1, PVAR = 0.25

To set the identification of a spring from the LISEGA catalog in dPIPE, the following fields from the spring designation are used: the first 2 digits are the spring design, the third digit corresponds to the load group, the fourth digit is the travel range, the last 2 digits determine the production series and in the spring code are not used. For example, the spring "21C219" is specified by the following command:

10: SPR nc = 1, tbl = 'LISEGA', id = 'C/2/21', note = "21C219"



16 Appendix VII

Consideration of cold springing in piping system design and analysis.

Cold-springing is a fabrication of piping to an actual length shorter than its nominal length so that it is stressed in the installed condition, thus compensating partially for the effects produced by expansion due to an increase in temperature.

In using the cold springing, it is necessary to take into account the following circumstances:

- cold spring reduces the loads on equipment in the hot state and increases them in the cold state;
- during installation of the piping system it is difficult to provide the required tolerance of the cold spring and further, in the course of operation of the piping system, it is impossible to control it. Taking into account this factor, a number of international stress codes for piping systems (ASME BPVC NB 3600, ASME B31.3), require to perform allowance for only 2/3 of the cold spring design value in determination of loads on supports and equipment in the working (hot) state and the full value of the cold spring in determination of loads in the cold state;
- the cold springing does not affect the stress range between the hot and cold states, and, therefore, it shall not be taken into account for computation of the corresponding stress categories;
- the relaxation of stresses due to thermal expansion in high-temperature piping systems leads to the cold springing of the piping system in the cold state irrespective of the use of the cold spring. The effect of use of the cold spring shows itself only at the initial stage of the piping system operation.

Within the framework of dPIPE program, the cold spring is specified, on the one hand, in the form of an analysis model element (see *CS* command) and, on the other hand, it is necessary to indicate the cold spring in the specification for analysis as a load component (*load* parameter, *SOLV* command). The standard specifications for analysis for the PNAE and RD Codes contain the recommended set of commands for taking into account the cold spring.

17 Appendix VIII

Effect of non-uniform temperature distribution over the height of piping system cross-section (thermal stratification).

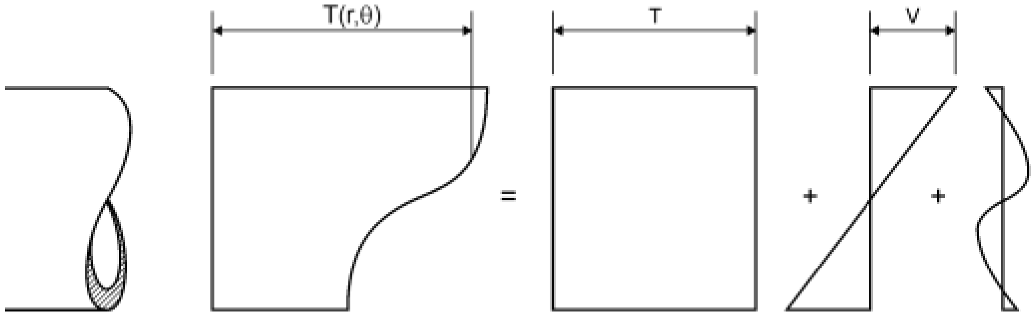
The thermal stratification arises in the horizontal segments of the piping system as a result of mixing the two flows of the working fluid with different temperatures and low velocities. Under certain operating conditions, due to this effect, the non-linear temperature distribution occurs over the height of piping system cross-section, which leads to additional temperature stress that can be divided into the following two categories:

- global bending stress caused by the thermal stratification top-to-bottom gradient: the upper part of the pipe tends to expand at higher temperatures, and the colder (lower) part of the pipe restrains these expansions and vice versa;
- a local stratification stress is produced by the nonlinearities in the top to bottom temperature distribution

It should be noted that the existing strength analysis codes for piping systems do not consider stratification as the design load, however in the course of operation of the piping system the stratification effect can influence on supports and equipment as well as on the fatigue strength.

The following procedure has been implemented within the framework of the dPIPE program for taking account of the stratification:

similar to the approach given in ASME NB-3653.2 for the temperatures gradient over the wall thickness of the piping system, the temperature distribution over the cross-section height of the piping can be presented by superposition of the following three parts: T – constant part, V – linear portion with zero average value and ΔT_3 – a non-linear portion with zero average value and a zero first moment across the pipe diameter.



Decomposition of Stratification Temperature Distribution Range.

The following formulae can be used for determination of the above-mentioned parameters:

$$T = \frac{2}{\pi\left(r_o^2 - r_i^2\right)} \int_{-\pi/2}^{\pi/2} \int_{r_i}^{r_o} r\,T(r,\theta)\,dr\,d\theta$$

$$V = \frac{6}{\pi(r_o^3 - r_i^3)} \int_{-\pi/2}^{\pi/2} \int_{r_i}^{r_o} r^2\,T(r,\theta)\sin\theta\,dr\,d\theta$$

$$\Delta T_3 = \max\left|\left(T(r,\theta) - T - Vr\sin\theta / r_o\right)\right|$$

where

T(r,θ) – function of distribution of temperature over the cross-section depending upon the radius and the angle, °C,

r_o – outside radius of pipe, mm

r_i – internal radius of pipe, mm

θ – angle to be counted from the horizontal central line of the cross-section

r – coordinate along the radius

After determination of the values of V and ΔT_3 (these parameters are described by the STRAT and DT3 identifiers respectively, see GRAD command), the program will compute the distributed bending moment M_{eq}, which complements the load from thermal expansions:

$$M_{eq} = \frac{E * I * \alpha * V}{D_0}$$

where

E – Young modulus (of elasticity) of the pipe material (MPa);

I – moment of inertia of the piping cross-section (mm^4);

α – temperature expansion coefficient (mm/mm/°C);

V – linear component of the temperature distribution,°C;

D_0 – outside diameter of the pipe, mm.

In order to take into account the load from stratification for the specified case of analysis, it is necessary to indicate the "BOW" identifier in the LOAD parameter (SOLV command) (for example, *LOAD='P+T+W+BOW'*). It should be noted that only those segments of the piping system that lie in the horizontal plane will be loaded by the distributed bending moment M_{eq} (BOW_PITCH parameter of the CTRL command establishes the acceptable deviation of pipe from the horizontal plane).

- in conducting the estimation of the fatigue strength according to the ASME NB-3600 Codes, equation 11 (NB-3653.2), the calculation of local peak stress shall be supplemented by the following part: $E\alpha|\Delta T_3|$ where $|\Delta T_3|$ is the maximum value of the non-linear component of the stratification temperature distribution, °C.
- within the framework of analysis according to the PNAE Codes, the additive $E\alpha|\Delta T_3|$ shall be taken into account in computation of the stress of category $(\sigma_{aF})_K$. To do this, one should explicitly define these stresses by STRESS parameter.

18 Appendix IX

Pressure Elongation in piping system

When piping is pressurized, its entire inner surface is subjected to a uniform pressure loading. This pressure loading creates stresses in the directions normal to the wall, parallel to the pipe axis, and tangential to the cross sectional circle:

- hoop stresses:

$$\sigma_h = \frac{P \cdot D}{2t}$$

- longitudinal stresses:

$$\sigma_z = \frac{P \cdot D}{4t}$$

where

P – internal pressure; D – mean diameter of the piping system; t – wall thickness.

At the same time it elongates the pipe again in the axial and circumferential directions. The stretch in the axial direction is generally referred to as ***pressure elongation***.

As a rule, these components of stresses are taken into account directly (without computation of internal forces) in the corresponding formulae set forth in the Strength analysis codes for piping systems in combination with permanent loads (weight + pressure). However, apart from the above-mentioned stresses, the internal pressure cause radial and longitudinal deformations in the piping system:

$$\varepsilon_h = \frac{\sigma_h}{E} - \frac{\mu \cdot \sigma_z}{E} = \frac{P \cdot D}{2 \cdot t \cdot E}(1 - 0.5\mu) \quad \text{and} \quad \varepsilon_z = \frac{\sigma_z}{E} - \frac{\mu \cdot \sigma_h}{E} = \frac{P \cdot D}{4 \cdot t \cdot E}(1 - 2\mu)$$

where

ε_h – hoop strain, ε_z – longitudinal strain, E – modulus of elasticity, μ - Poisson's coefficient.

And if the hoop strain, due to the absence of their restraint, can be neglected, then the allowance for internal pressure, from the point of view of longitudinal deformations,can, in certain cases, make a contribution comparable with temperature expansions. In order to evaluate this effect, it is possible to use the formula for conversion of the equivalent temperature corresponding to the piping system deformation from internal pressure:

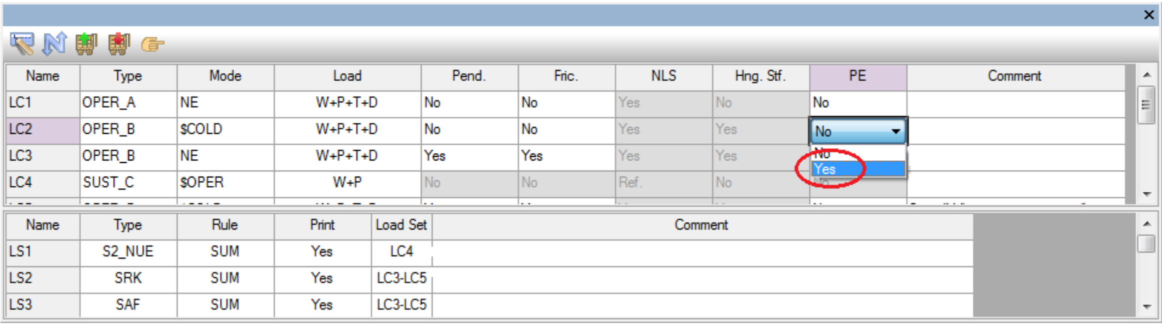
$$\Delta T_{eq.} = \frac{P \cdot D}{4 \cdot t \cdot E \cdot \alpha}(1 - 2\mu)$$

where α is the temperature coefficient of linear expansion.

For most piping systems being operated at high temperatures on plants, this effect is insignificant and ΔT is equal to a value of about 5-10 °C but for extent thin-walled piping systems being operated at moderate temperatures, the value of equivalent temperature can reach 40 - 50 °C. In doing so, additional loads on supports and connected equipment arise in the piping system apart from stresses caused by its tension due to pressure.

As the load caused by "tension" of the piping system under the action of internal pressure, is, by its nature, related to self-balanced (secondary) load, then the effect of piping system tension due to internal pressure shall be taken into account in combination with deformation loads (temperature expansions and displacements of supports) and the additional stresses shall be evaluated by the corresponding stress categories.

In dPIPE, the allowance for axial deformations from internal pressure has been implemented by introduction of the PE option (SOLV command). If necessary, for the corresponding case of analysis this flag should be set as PE = 'YES':



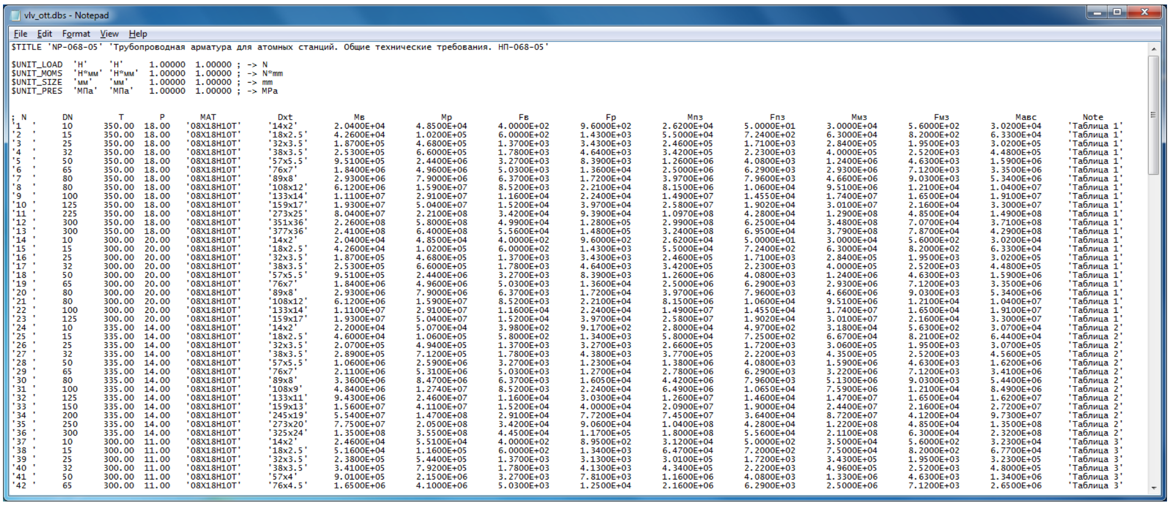
Name	Type	Mode	Load	Pend.	Fric.	NLS	Hng. Stf.	PE	Comment
LC1	OPER_A	NE	W+P+T+D	No	No	Yes	No	No	
LC2	OPER_B	$COLD	W+P+T+D	No	No	Yes	Yes	No	
LC3	OPER_B	NE	W+P+T+D	Yes	Yes	Yes	Yes	No Yes	
LC4	SUST_C	$OPER	W+P	No	No	Ref.	No		

Name	Type	Rule	Print	Load Set	Comment
LS1	S2_NUE	SUM	Yes	LC4	
LS2	SRK	SUM	Yes	LC3-LC5	
LS3	SAF	SUM	Yes	LC3-LC5	

19 Appendix X

Assessment of loads on the valve nozzles

The assessment is performed in accordance with the document: "Piping valves for nuclear plants. General technical requirements NP-068-05", *[REF 14]*. The data from Appendix 8 to this document have been recorded in the database contained in the "*vlv_ott.dbs*" file. This file has a text format and the following structure:



```
vlv_ott.dbs - Notepad
File  Edit  Format  View  Help
$TITLE 'NP-068-05' 'Трубопроводная арматура для атомных станций. Общие технические требования. НП-068-05'

$UNIT_LOAD  'Н'    'Н'     1.00000  1.00000 ; -> N
$UNIT_MOMS  'Н*мм' 'Н*мм'  1.00000  1.00000 ; -> N*mm
$UNIT_SIZE  'мм'   'мм'    1.00000  1.00000 ; -> mm
$UNIT_PRES  'МПа'  'МПа'   1.00000  1.00000 ; -> MPa

; N    DN   T       P      MAT          Dxt        Mв          Mp          Fв          Fp          Mпз         Fпз         Mмз         Fмз         Mавс        Note
'1  '  10   350.00  18.00  '08X18H10T'  '14x2'     2.0400E+04  4.8500E+04  4.0000E+02  9.6000E+02  2.6200E+04  5.0000E+01  3.0000E+04  5.6000E+02  3.0200E+04  'Таблица 1'
'2  '  15   350.00  18.00  '08X18H10T'  '18x2.5'   4.2600E+04  1.0200E+05  6.0000E+02  1.4300E+03  5.5000E+04  7.2400E+02  6.3000E+04  8.2000E+02  6.3300E+04  'Таблица 1'
'3  '  25   350.00  18.00  '08X18H10T'  '32x3.5'   1.8700E+05  4.6800E+05  1.3700E+03  3.4300E+03  2.4600E+05  1.7100E+03  2.8400E+05  1.9500E+03  3.0200E+05  'Таблица 1'
'4  '  32   350.00  18.00  '08X18H10T'  '38x3.5'   2.5300E+05  6.6000E+05  1.7800E+03  4.6400E+03  3.4200E+05  2.2300E+03  4.0000E+05  2.5200E+03  4.4800E+05  'Таблица 1'
'5  '  50   350.00  18.00  '08X18H10T'  '57x5.5'   9.5100E+05  2.4400E+06  3.2700E+03  8.3900E+03  1.2600E+06  4.0800E+03  1.2400E+06  4.6300E+03  1.5900E+06  'Таблица 1'
'6  '  65   350.00  18.00  '08X18H10T'  '76x7'     1.8400E+06  4.9600E+06  5.0300E+03  1.3600E+04  2.5000E+06  6.2900E+03  2.9300E+06  7.1200E+03  3.3500E+06  'Таблица 1'
'7  '  80   350.00  18.00  '08X18H10T'  '89x8'     2.9300E+06  7.9000E+06  6.3700E+03  1.7200E+04  3.9700E+06  7.9600E+03  4.6600E+06  9.0300E+03  5.3400E+06  'Таблица 1'
'8  '  80   350.00  18.00  '08X18H10T'  '108x12'   6.1200E+06  1.5900E+07  8.5200E+03  2.2100E+04  8.1500E+06  1.0600E+04  9.5100E+06  1.2100E+04  1.0400E+07  'Таблица 1'
'9  '  100  350.00  18.00  '08X18H10T'  '133x14'   1.1100E+07  2.9100E+07  1.1600E+04  2.2400E+04  1.4900E+07  1.4550E+04  1.7400E+07  1.6500E+04  1.9100E+07  'Таблица 1'
'10 '  125  350.00  18.00  '08X18H10T'  '159x17'   1.9300E+07  5.0400E+07  1.5200E+04  3.9700E+04  2.5800E+07  1.9020E+04  3.0100E+07  2.1600E+04  3.3000E+07  'Таблица 1'
'11 '  225  350.00  18.00  '08X18H10T'  '273x25'   8.0400E+07  2.2100E+08  3.4200E+04  9.3900E+04  1.0970E+08  4.2800E+04  1.2900E+08  4.8500E+04  1.4900E+08  'Таблица 1'
'12 '  300  350.00  18.00  '08X18H10T'  '351x36'   2.2600E+08  5.8000E+08  4.9900E+04  1.2800E+05  2.9900E+08  6.2500E+04  3.4800E+08  7.0700E+04  3.7100E+08  'Таблица 1'
'13 '  300  350.00  18.00  '08X18H10T'  '377x36'   2.4100E+08  6.4000E+08  5.5600E+04  1.4800E+05  3.2400E+08  6.9500E+04  3.7900E+08  7.8700E+04  4.2900E+08  'Таблица 1'
'14 '  10   300.00  20.00  '08X18H10T'  '14x2'     2.0400E+04  4.8500E+04  4.0000E+02  9.6000E+02  2.6200E+04  5.0000E+01  3.0000E+04  5.6000E+02  3.0200E+04  'Таблица 1'
'15 '  15   300.00  20.00  '08X18H10T'  '18x2.5'   4.2600E+04  1.0200E+05  6.0000E+02  1.4300E+03  5.5000E+04  7.2400E+02  6.3000E+04  8.2000E+02  6.3300E+04  'Таблица 1'
'16 '  25   300.00  20.00  '08X18H10T'  '32x3.5'   1.8700E+05  4.6800E+05  1.3700E+03  3.4300E+03  2.4600E+05  1.7100E+03  2.8400E+05  1.9500E+03  3.0200E+05  'Таблица 1'
'17 '  32   300.00  20.00  '08X18H10T'  '38x3.5'   2.5300E+05  6.6000E+05  1.7800E+03  4.6400E+03  3.4200E+05  2.2300E+03  4.0000E+05  2.5200E+03  4.4800E+05  'Таблица 1'
'18 '  50   300.00  20.00  '08X18H10T'  '57x5.5'   9.5100E+05  2.4400E+06  3.2700E+03  8.3900E+03  1.2600E+06  4.0800E+03  1.2400E+06  4.6300E+03  1.5900E+06  'Таблица 1'
'19 '  65   300.00  20.00  '08X18H10T'  '76x7'     1.8400E+06  4.9600E+06  5.0300E+03  1.3600E+04  2.5000E+06  6.2900E+03  2.9300E+06  7.1200E+03  3.3500E+06  'Таблица 1'
'20 '  80   300.00  20.00  '08X18H10T'  '89x8'     2.9300E+06  7.9000E+06  6.3700E+03  1.7200E+04  3.9700E+06  7.9600E+03  4.6600E+06  9.0300E+03  5.3400E+06  'Таблица 1'
'21 '  80   300.00  20.00  '08X18H10T'  '108x12'   6.1200E+06  1.5900E+07  8.5200E+03  2.2100E+04  8.1500E+06  1.0600E+04  9.5100E+06  1.2100E+04  1.0400E+07  'Таблица 1'
'22 '  100  300.00  20.00  '08X18H10T'  '133x14'   1.1100E+07  2.9100E+07  1.1600E+04  2.2400E+04  1.4900E+07  1.4550E+04  1.7400E+07  1.6500E+04  1.9100E+07  'Таблица 1'
'23 '  125  300.00  20.00  '08X18H10T'  '159x17'   1.9300E+07  5.0400E+07  1.5200E+04  3.9700E+04  2.5800E+07  1.9020E+04  3.0100E+07  2.1600E+04  3.3000E+07  'Таблица 1'
'24 '  10   335.00  14.00  '08X18H10T'  '14x2'     2.2000E+04  5.0700E+04  3.9800E+02  9.1700E+02  2.8000E+04  4.9700E+02  3.1800E+04  5.6300E+02  3.0700E+04  'Таблица 2'
'25 '  15   335.00  14.00  '08X18H10T'  '18x2.5'   4.6000E+04  1.0600E+05  5.8000E+02  1.3400E+03  5.8000E+04  7.2500E+02  6.6700E+04  8.2100E+02  6.4400E+04  'Таблица 2'
'26 '  25   335.00  14.00  '08X18H10T'  '32x3.5'   2.0700E+05  4.9400E+05  1.3700E+03  3.2700E+03  2.6600E+05  1.7200E+03  3.0600E+05  1.9500E+03  3.0700E+05  'Таблица 2'
'27 '  32   335.00  14.00  '08X18H10T'  '38x3.5'   2.8900E+05  7.1200E+05  1.7800E+03  4.3800E+03  3.7700E+05  2.2200E+03  4.3500E+05  2.5200E+03  4.5600E+05  'Таблица 2'
'28 '  50   335.00  14.00  '08X18H10T'  '57x5.5'   1.0600E+06  2.5900E+06  3.2700E+03  1.2300E+04  1.3800E+06  4.0800E+03  1.5900E+06  4.6300E+03  1.6200E+06  'Таблица 2'
'29 '  65   335.00  14.00  '08X18H10T'  '76x7'     2.1100E+06  5.3100E+06  5.0300E+03  1.2700E+04  2.7800E+06  6.2900E+03  3.2200E+06  7.1200E+03  3.4100E+06  'Таблица 2'
'30 '  80   335.00  14.00  '08X18H10T'  '89x8'     3.3600E+06  8.4700E+06  6.3700E+03  1.6050E+04  4.4200E+06  7.9600E+03  5.1300E+06  9.0300E+03  5.4400E+06  'Таблица 2'
'31 '  100  335.00  14.00  '08X18H10T'  '108x9'    4.8400E+06  1.2740E+07  8.5200E+03  2.2400E+04  6.4900E+06  1.0650E+04  7.5900E+06  1.2100E+04  8.4900E+06  'Таблица 2'
'32 '  125  335.00  14.00  '08X18H10T'  '133x11'   9.4300E+06  2.4600E+07  1.1600E+04  3.0300E+04  1.2600E+07  1.4600E+04  1.4700E+07  1.6500E+04  1.6200E+07  'Таблица 2'
'33 '  150  335.00  14.00  '08X18H10T'  '159x13'   1.5600E+07  4.1100E+07  1.5200E+04  4.0000E+04  2.0900E+07  1.9000E+04  2.4400E+07  2.1600E+04  2.7200E+07  'Таблица 2'
'34 '  200  335.00  14.00  '08X18H10T'  '245x19'   5.5400E+07  1.4700E+08  2.9100E+04  7.7200E+04  7.4500E+07  3.6400E+04  8.7200E+07  4.1200E+04  9.7300E+07  'Таблица 2'
'35 '  250  335.00  14.00  '08X18H10T'  '273x20'   7.7500E+07  2.0500E+08  3.4200E+04  9.0600E+04  1.0400E+08  4.2800E+04  1.2200E+08  4.8500E+04  1.3500E+08  'Таблица 2'
'36 '  300  335.00  14.00  '08X18H10T'  '325x24'   1.3500E+08  3.5500E+08  4.4500E+04  1.1700E+05  1.8000E+08  5.5600E+04  2.1100E+08  6.3000E+04  2.3200E+08  'Таблица 2'
'37 '  10   300.00  11.00  '08X18H10T'  '14x2'     2.4600E+04  5.5100E+04  4.0000E+02  8.9500E+02  3.1200E+04  5.0000E+02  3.5000E+04  5.6000E+02  3.2300E+04  'Таблица 3'
'38 '  15   300.00  11.00  '08X18H10T'  '18x2.5'   5.1600E+04  1.1600E+05  6.0000E+02  1.3400E+03  6.4700E+04  7.2000E+02  7.5000E+04  8.2000E+02  6.7700E+04  'Таблица 3'
'39 '  25   300.00  11.00  '08X18H10T'  '32x3.5'   2.3800E+05  5.4400E+05  1.3700E+03  3.1300E+03  3.0100E+05  1.7200E+03  3.4300E+05  1.9500E+03  3.2300E+05  'Таблица 3'
'40 '  32   300.00  11.00  '08X18H10T'  '38x3.5'   3.4100E+05  7.9200E+05  1.7800E+03  4.1300E+03  4.3400E+05  2.2200E+03  4.9600E+05  2.5200E+03  4.8000E+05  'Таблица 3'
'41 '  50   300.00  11.00  '08X18H10T'  '57x4'     9.0100E+05  2.1500E+06  3.2700E+03  7.8100E+03  1.1600E+06  4.0800E+03  1.3300E+06  4.6300E+03  1.3400E+06  'Таблица 3'
'42 '  65   300.00  11.00  '08X18H10T'  '76x4.5'   1.6500E+06  4.1000E+06  5.0300E+03  1.2500E+04  2.1600E+06  6.2900E+03  2.5000E+06  7.1200E+03  2.6500E+06  'Таблица 3'
```

The first line, command : $TITLE contains a short and full description of the document:

$TITLE 'NP-068-05' 'Piping valves for nuclear plants. General technical requirements NP-068-05'

Commands: ***$UNIT_XXX*** determine the measurement unit system and conversion factors (from one measurement unit system into another) for the data contained in the database:

```
$UNIT_LOAD  'Н'    'Н'    1.00000  1.00000 ; -> N
$UNIT_MOMS  'Н*мм' 'Н*мм' 1.00000  1.00000 ; -> N*mm
$UNIT_SIZE  'мм'   'мм'   1.00000  1.00000 ; -> mm
$UNIT_PRES  'МПа'  'МПа'  1.00000  1.00000 ; -> MPa
```

where

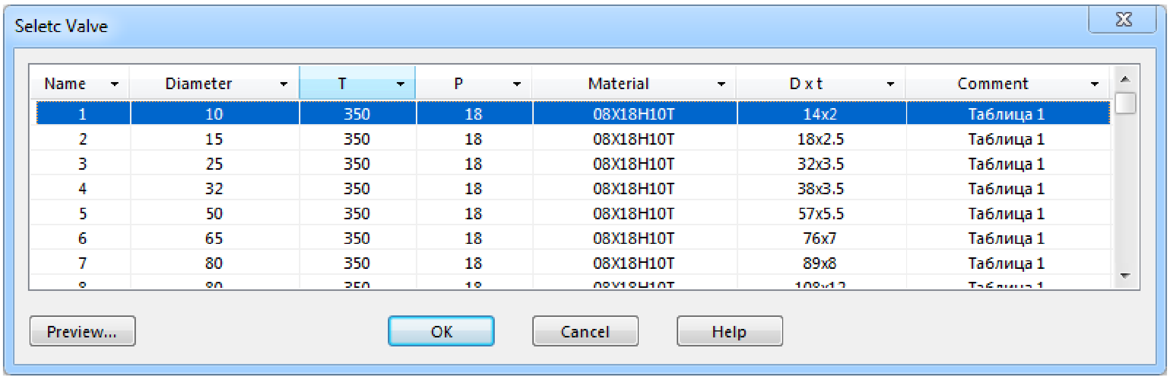
$UNIT_LOAD - measurement unit system for forces; ***$UNIT_MOMS*** - measurement unit system for moments; ***$UNIT_SIZE*** - measurement unit system for linear dimensions; ***$UNIT_PRES*** - measurement unit system for pressure

Blank lines and lines beginning from the semicolon sing ";" will be interpreted by the program as comment.

The data themselves consist of a set of records (lines) consisting of 16 values (columns):

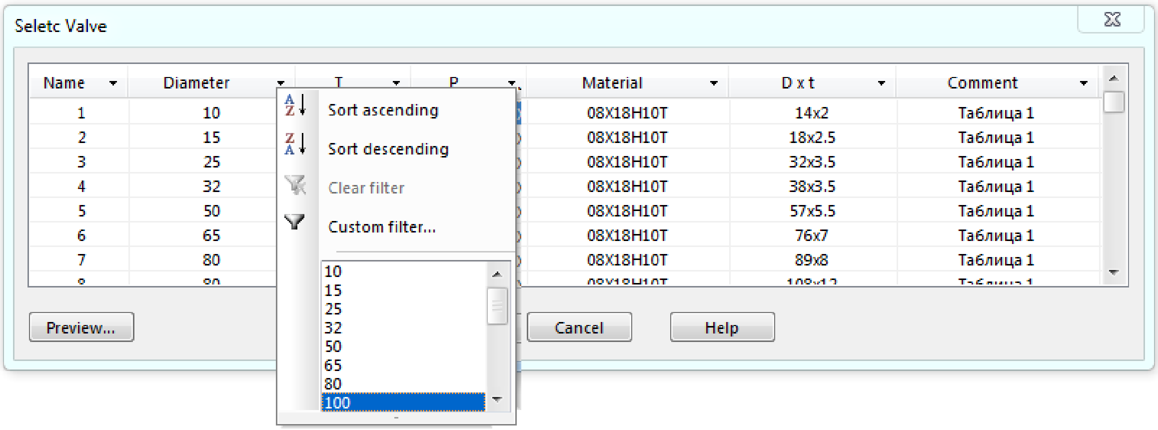
Designation	Column number	Value	Format
N	1	record identifier	TEXT
DN	2	nominal diameter	INTEGER
T	3	design temperature	REAL
P	4	design pressure	REAL
MAT	5	material	TEXT
Dxt	6	dimensions of connecting pipes	TEXT
Мв	7	moment from the piping weight	REAL
Мр	8	range of moment from thermal expansion of the piping system	REAL
Fв	9	force from piping weight	REAL
Fр	10	range of force from thermal expansion	REAL
Мпз	11	moment and force from the joint action of the piping weight and OBE	REAL
Fпз	12		REAL
Ммз	13	moment and force from the joint action of the piping weight and SSE	REAL
Fмз	14		REAL
Мавс	15	moment from the joint action of the piping weight and reactive force in case of piping system rupture	REAL
Note	16	comment (reference to table from Appendix 8)	TEXT

Referencing a certain valve (V, V1, V2, VA, VO commands) to the database is performed via the corresponding dialogs with the use of the *OTT_REF parameter*. In case when the "Select" button is pressed, the following dialog appears:

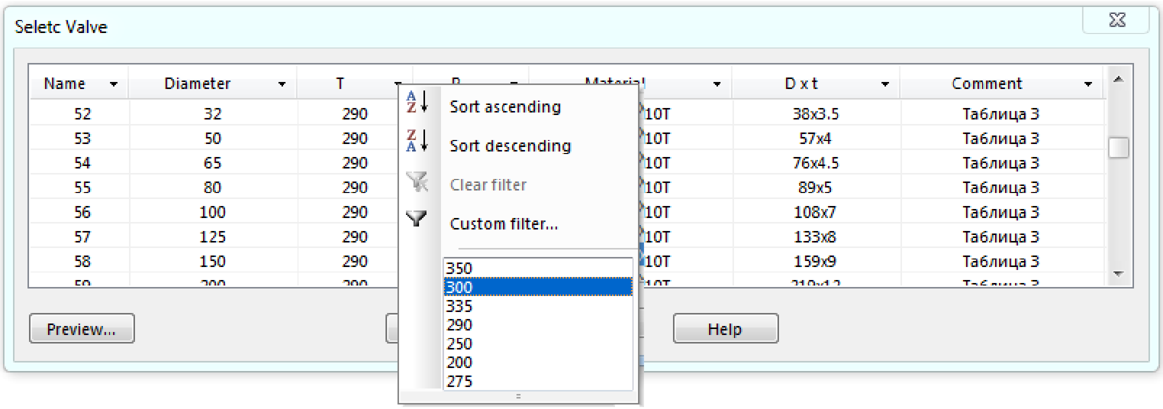


In order to select the required record in this dialog, one can use the integrated filters. For example, in order to select the allowable loads for the electrically driven shutoff bellows DN 100, PN=20 MPa, T=300 °C from stainless steel, it is necessary to execute the following operations:

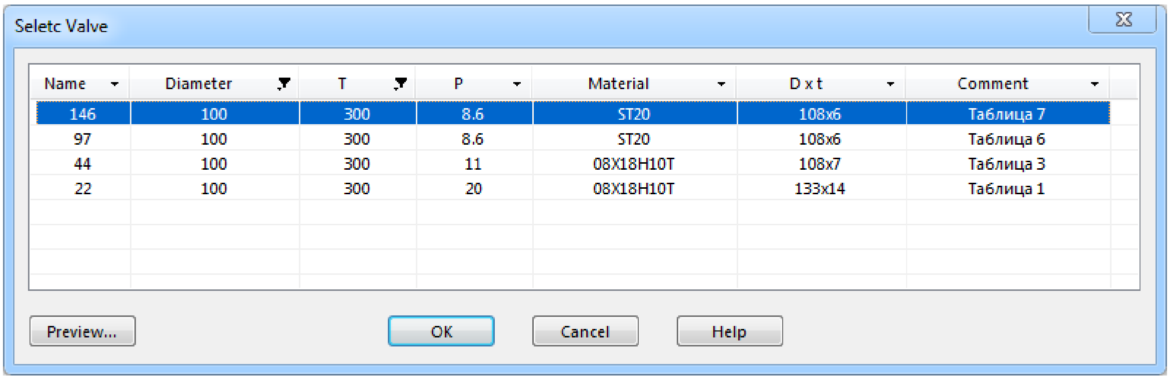
1. Click on the "Diameter" title and select the nominal diameter 100 by a double click:



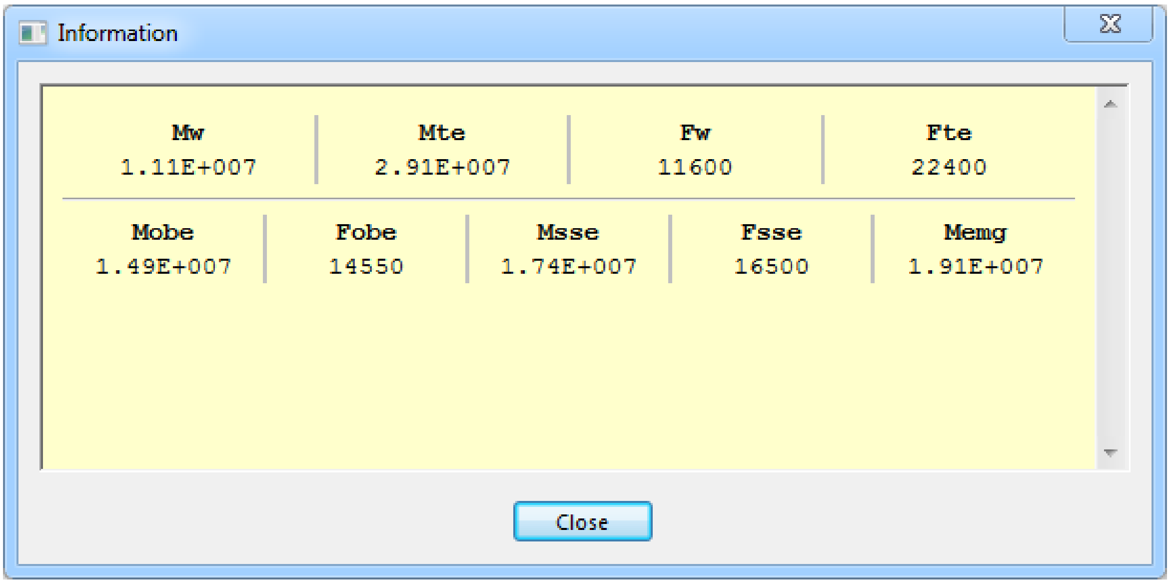
2. Click on the "Temperature" title and select the temperature 300 by a double click:



3. Form the remaining data in the table, select the required record and press OK:



Using the "View" button you can preview the values of allowable loads:



In addition, in the "Options/Reports/Valves" dialog, it is necessary to specify the corresponding references for LC & LS so that the assessment of loads on the valve nozzles should be performed.

Upon execution of analysis, two tables will appear in the file with the input data listing, *<model name>.out*, containing information about the links between the valve data and the reference to the database record as well as a table with the values of allowable loads:

```
>>>  Table   11. Assessment of valve's nozle loads according to NP-068-05
--------------------------------------------------------------------------------------------------------
designation        Node 1   Node 2   body mass   actuator   label   DN    Dxs        Pd   Td    material
--------------------------------------------------------------------------------------------------------
10LCQ25AA201       2460     6230     ¤0000028               80       80   89x5       2.5 250.0 08X18H10T  Таблица 5
10LCQ35AA201       3760     3770     ¤0000029               80       80   89x5       2.5 250.0 08X18H10T  Таблица 5
10LCQ15AA201       5090     5100     ¤0000030               80       80   89x5       2.5 250.0 08X18H10T  Таблица 5
10LCQ45AA201       6410     6420     ¤0000031               80       80   89x5       2.5 250.0 08X18H10T  Таблица 5
вход охлаждающ 1   1780     6110                            58      150   159x9      9.2 290.0 08X18H10T  Таблица 3

>>>  Table   11.a Allowable loads for valve's nozzles according to NP-068-05
----------------------------------------------------------------------------------------------------------
label          Fw         Mw         Ft         Mt         Fobe       Mobe       Fsse       Msse       Memerg
----------------------------------------------------------------------------------------------------------
80           4.000E+03 2.330E+06 5.940E+03 3.450E+06 4.900E+03 2.870E+06 5.610E+03 3.230E+06 3.460E+06
58           1.520E+04 1.320E+07 3.880E+04 3.370E+07 1.900E+04 1.750E+07 2.160E+04 2.030E+07 2.190E+07
----------------------------------------------------------------------------------------------------------
Notes:
Fw,   Mw   - resulting force and moment from the weight loads: H, H*мм
Ft,   Mt   - range of force and moment due to thermal expansion: H, H*мм
Fobe, Mobe - force and moment due to weight and seismic (OBE) loads: H, H*мм
Fмрз, Mмрз - force and moment due to weight and seismic (SSE) loads: H, H*мм
Mавс       - moment due to weight and emergency  loads:H*мм
```

The results of analysis will be recorded in a file with summary tables, *<model name>.sup,* in the form of a table containing ratios of the design values to the allowable ones:

```
>>>  Assessment of valve's nozzle loads according to NP-068-05 (demand to capacity ratio)
----------------------------------------------------------------------------------------------------------------------
designation     Fw       Mw       Fr       Mr       Fobe     Mobe     Fsse     Msse     Memr     Ab_h    Ab_v    Aa_h
----------------------------------------------------------------------------------------------------------------------
10LCQ25AA201    0.007    0.081    0.067    0.287    0.263    0.483    0.454    0.803    -        1.78    0.36    -
10LCQ35AA201    0.003    0.087    0.322    0.206    0.091    0.307    0.158    0.494    -        1.75    0.30    -
10LCQ15AA201    0.000    0.025    0.100    0.251    0.236    0.332    0.411    0.573    -        1.63    0.27    -
10LCQ45AA201    0.006    0.087    0.233    0.239    0.076    0.344    0.128    0.550    -        1.70    0.26    -
вход охлажда    0.004    0.024    0.102    0.241    0.105    0.253    0.182    0.421    -        0.33    0.22    -
----------------------------------------------------------------------------------------------------------------------
Notes:
Fw, Mw            - force and moment from the weight loads;
Fr, Mr            - range of force and moment from the thermal expansion loads
Fobe, Mobe        - force and moment from the OBE loads;
Fsse, Msse        - force and moment from the SSE loads;
Memr              - moment from combination of weight load and reactive load due to pipe break
Ab_h, Ab_v, Aa_h  - horizontal (h) and vertical (v) accelerations of the valve's body and actuator (in g)
```

In case of the seismic load being present in the analysis, the values of horizontal and vertical accelerations of the valve housing center of mass will additionally be printed out in comparison with the values of 3g in the horizontal direction and 2g in the vertical direction for the valve housing and 8g in the horizontal direction for the valve drive center of mass.

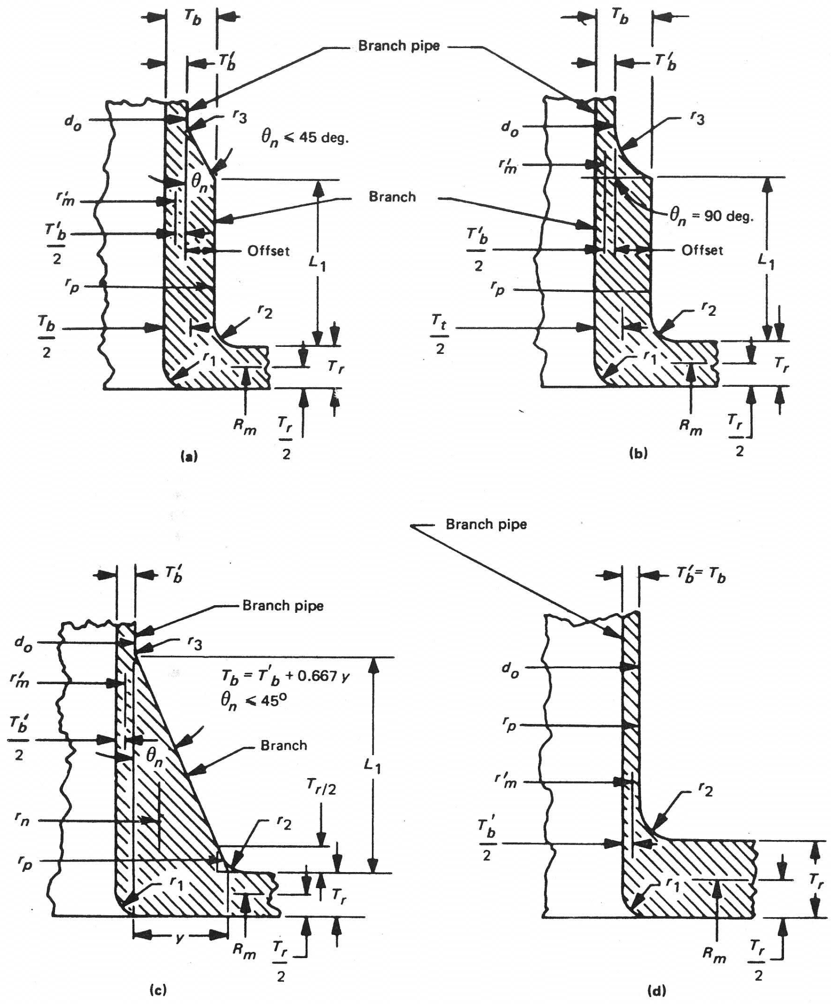
20 Appendix XI

ASME BPVC Codes. NB/NC-3600 sections. Stress Indices

NC_3600

NC-3600, 1992 (CODE = 'ASME_NC', CODE_YEAR = '1992', TRN command, TYPE = 'BRC')

NC-3000 — DESIGN Fig. NC-3673.2(b)-2



d_o = outside diameter of branch pipe, in.
r_m = mean radius of branch pipe in.
T'_b = nominal thickness of branch pipes, in.
R_m = mean radius of run pipe, in.
T_r = nominal thickness of run pipe, in.

GENERAL NOTES:

(1) T_b, θ, r_1, r_2, r_3, r_p, and y are defined in this figure.

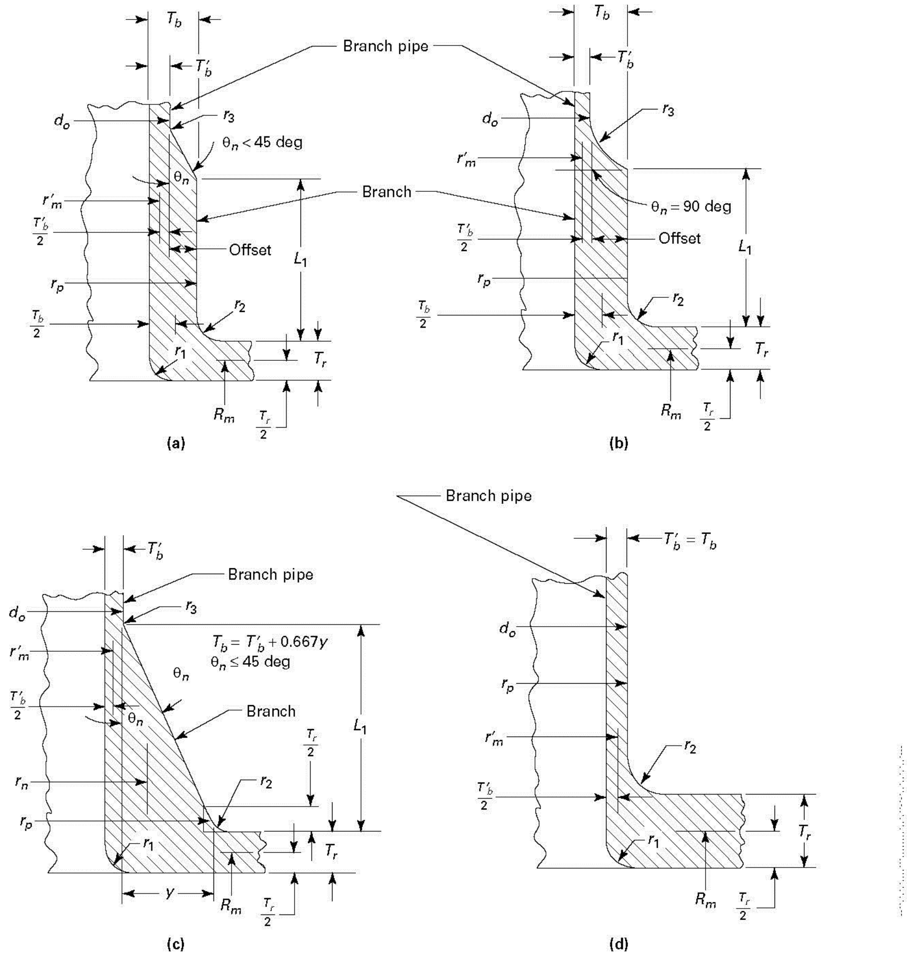
(2) If L_1 equals or exceeds $0.5\sqrt{r_i T_b}$ then r'_m can be taken as the radius to the center of T_b.

FIG. NC-3673.2(b)-2 BRANCH DIMENSIONS

183

NC-3600, 2010 (CODE = 'ASME_NC', CODE_YEAR = '2010', TRN command, TYPE = 'BRC')

FIG. NC-3673.2(b)-2 BRANCH CONNECTION NOMENCLATURE

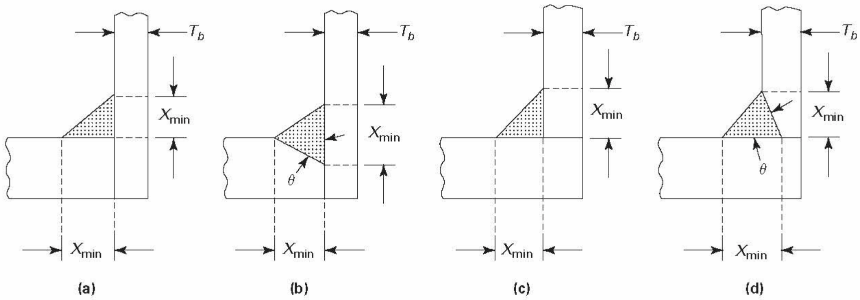


d_o = outside diameter of branch pipe, in. (mm)
r'_m = mean radius of branch pipe, in. (mm)
T'_b = nominal thickness of branch pipes, in. (mm)
R_m = mean radius of run pipe, in. (mm)
T_r = nominal thickness of run pipe, in. (mm)

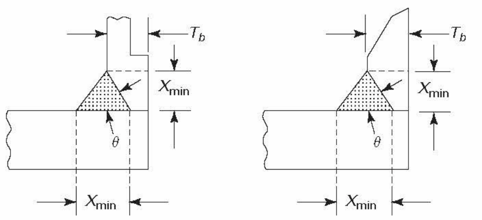
GENERAL NOTES:
(a) T_b, θ, r_1, r_2, r_3, r_p, and y are defined in this figure.

NC-3600, 2010 (CODE = 'ASME_NC', CODE_YEAR = '2010', TRN command, TYPE = 'WLD_BRC')

FIG. NC-3643.2(b)-2 TYPICAL RIGHT ANGLE BRANCH CONNECTIONS MADE USING A FILLET WELD OR A PARTIAL PENETRATION WELD



T_b = nominal branch pipe wall thickness
X_{min} = $1^1/_4\ T_b$
θ = partial penetration weld groove angle ≥ 45 deg



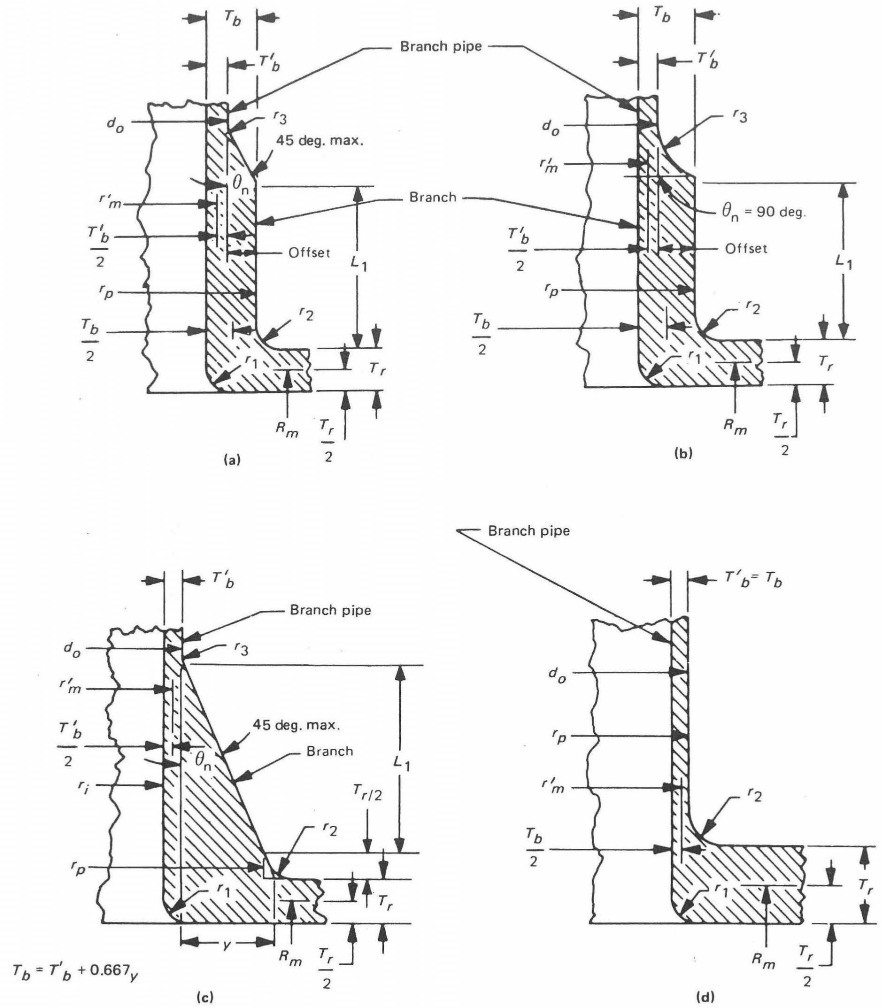
(e) ASME B16.11 Coupling (f) Welded Outlet Fitting

T_b = fitting wall thickness in the reinforcement zone (when the fitting is tapered in the reinforcement zone, use average wall thickness)
X_{min} = $1^1/_4\ T_b$
θ = partial penetration weld groove angle ≥ 45 deg

NB_3600

NB-3600, 1992 (CODE = 'ASME_NB', CODE_YEAR = '1992', TRN command, TYPE = 'BRC')

NB-3000 — DESIGN **Fig. NB-3643.3(a)-1**



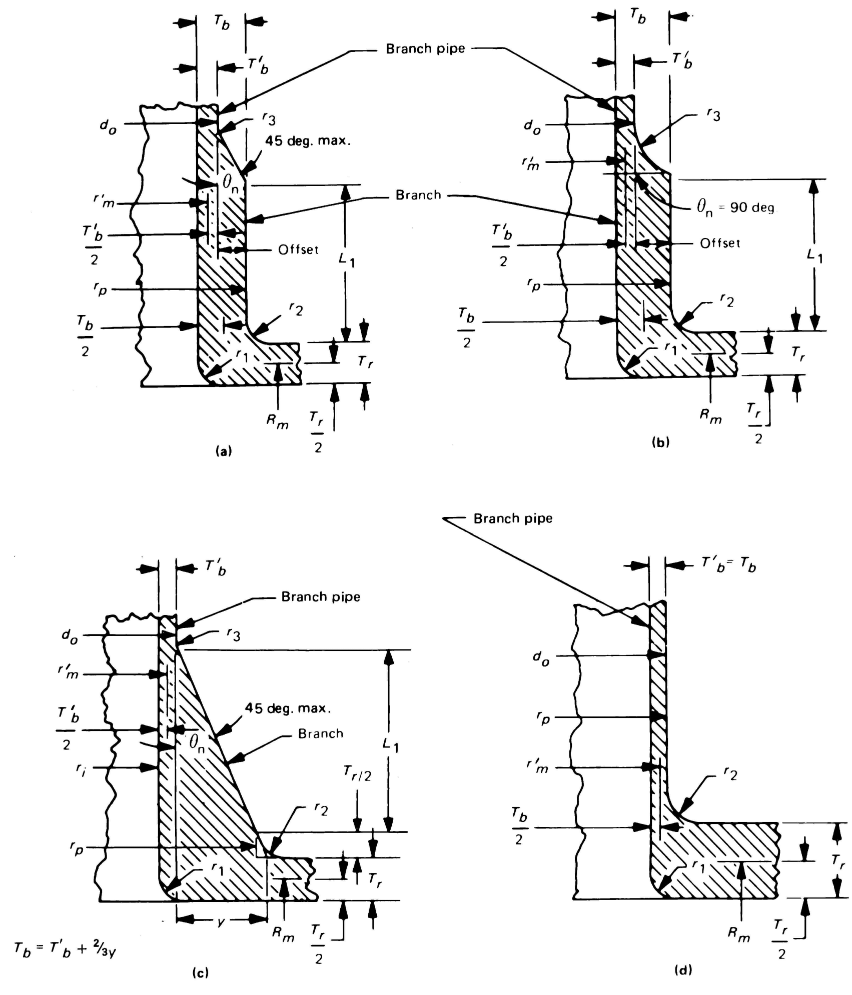
GENERAL NOTE:
If L_1 equals or exceeds $0.5\sqrt{r_i T_b}$, then r'_m can be taken as the radius to the center of T_b.

FIG. NB-3643.3(a)-1 BRANCH CONNECTION NOMENCLATURE

139

NB-3600, 2010 (CODE = 'ASME_NB', CODE_YEAR = '2010', TRN command, TYPE = 'BRC')

FIG. NB-3643.3(a)-1 BRANCH CONNECTION NOMENCLATURE

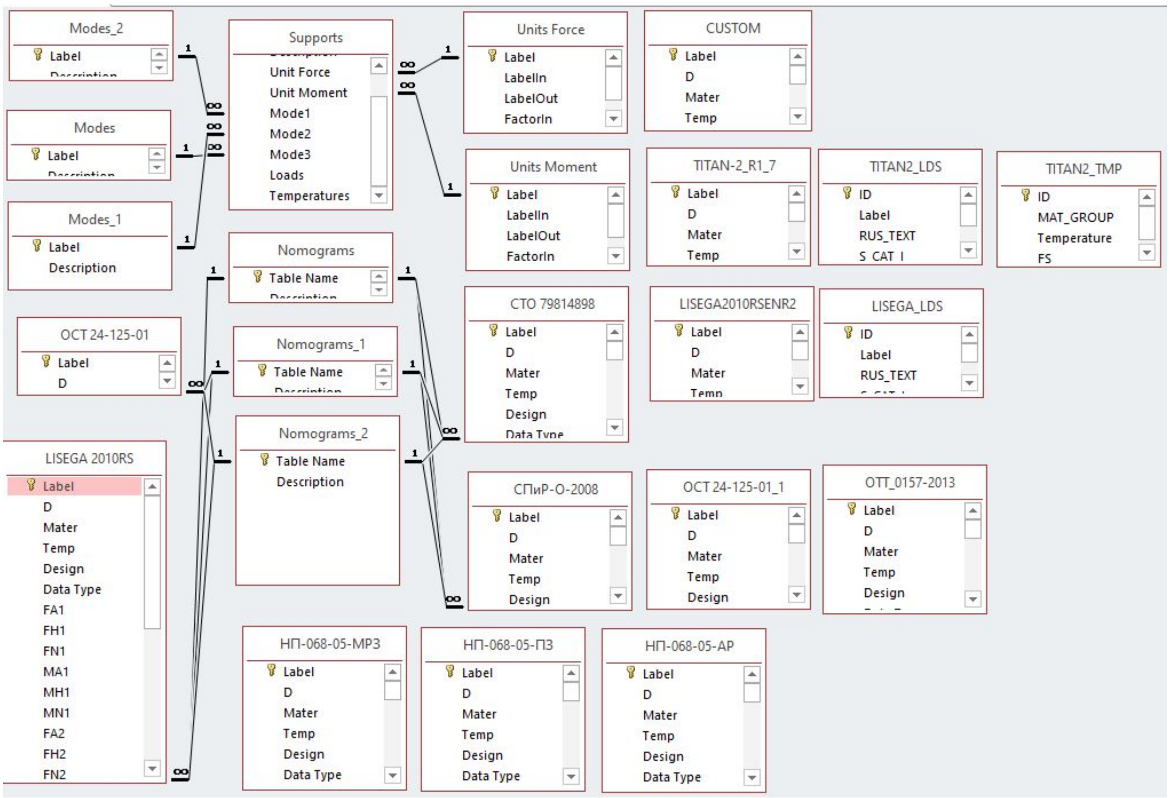


21 Appendix XII

Structure of the database with Pipe supports Allowable loads

After installation of the program the file with database in MS Access format containing pipe supports allowable loads is located in the following folder: ...\dPIPE 5.XX\DB\sup_lds.mdb. The MS Access program itself is not required for PC operation but it can become necessary if the User independently determines to edit the database.

General structure of the database:



Description of the main DB tables:

➢ ***«Supports»*** table with a list of standards being present in DB:

Table Name	Description	Unit Force	Unit Moment	Mode 1	Mode2	Mode 3	Loads	Temperatures
CUSTOM	Table for the Custom input	Newton	Newton *m	NOC	HT	NOC+ OBE		
LISEGA 2010RS	LISEGA. Standard Supports 2010 RS. Document No. 900081-4	Kilonewton	Newton *m	НУЭ	НУЭ+МР 3	НУЭ+ П3		
LISEGA2010RSENR2	Standard Supports 2010 RS EN, Document No.: 902205, Rev. 2	Kilonewton	Newton *m	N/A	N/A	N/A	LISEGA _LDS	
OTT_0157-2013	Support structures of WWER NPP elements,	Kilonewton	Knewton*m	OTT_ 1.0	OTT_1.4	OTT_ 1.5		

	OTT 1.5.2.01.999.0157-2013							
TITAN-2_R1_7	Catalog of standard supports for ISO 4200 1991 pipes	Kilonewton	Newton*m	N/A	N/A	N/A	TITAN2_LDS	TITAN2_TMP
НП-068-05-АР	Piping valves for nuclear power plants. OTT	Newton	Newton*mm	Вес	Температура	Вес+АР		
НП-068-05-МРЗ	Piping valves for nuclear power plants. OTT	Newton	Newton*mm	Вес	Температура	Вес+МРЗ		
НП-068-05-ПЗ	Piping valves for nuclear power plants. OTT	Newton	Newton*mm	Вес	Температура	Вес+ПЗ		
ОСТ 24-125-01	Permissible loads for supports of high-pressure pipes. OST 24.125-01	Newton	Newton*mm	НУЭ	НУЭ+МРЗ	НУЭ+ПЗ		
СПиР-О-2008	Allowable Loads for Standard Support Structures (Appendix O-2)	Kilonewton	KNewton*m	НУЭ	НУЭ+МРЗ	НУЭ+ПЗ		
СТО 79814898	Supports for NPP piping working under pressure up to 4.0 MPa	Newton	Newton*mm	НУЭ	НУЭ+МРЗ	НУЭ+ПЗ		

Fields:

Field Name	Data Type	Field Size	Descriptor
Table Name	Short Text	16	Name of the table
Description	Short Text	128	Table Description
Unit Force	Short Text	16	The label used for the units of force, reference to the "Units Force"
Unit Moment	Short Text	16	The label used for the units of moments, reference to the "Units Force"
Model	Short Text	16	Description of the operational mode "MOD1"
Mode2	Short Text	16	Description of the operational mode "MOD2"
Mode3	Short Text	16	Description of the operational mode "MOD3"
Loads	Short Text	255	The name of the table with amplification factors used for scaling nominal loads depending on load combination and pipe’s seismic category
Temperatures	Short Text	255	The name of the table with correction factors used for scaling nominal loads depending on the piping operational temperature

- The ***"Modes"*** table with a description of the operational modes, which are referenced by the Mode1, Mode2 and Mode3 fields of the SUPPORTS table:

Label	Description
HT	Hydraulic Test
N/A	not applicable
NOC	Normal Operation Conditions
NOC+OBE	NOC+OBE
NOC+SSE	NOC+SSE
OTT_1.0	AF = 1.0
OTT_1.4	AF = 1.4
OTT_1.5	AF = 1.5
Вес	Weight loads
Вес+АР	Combination of weight and piping break loads
Вес+МРЗ	Combination of weight and SSE loads
Вес+ПЗ	Combination of weight and OBE loads
ГИ	Hydraulic Test
НУЭ	NOC, normal operation conditions
НУЭ+МРЗ	Combination of NOC and SSE loads (applicable only for seismic category I pipes)
НУЭ+ПЗ	Combination of NOC and OBE loads (applicable only for seismic category II pipes)
Температура	Thermal expansion loads

Fields:

Field Name	Data Type	Field Size	Descriptor
Label	TEXT	16	Short name of the Mode (will be shown in the printouts)
Description	TEXT	64	Mode Description (will be shown in the printouts)

- ***«Nomograms» Table*** with references to the nomograms, which are used for determination of permissible loads:

Supports | **Nomograms**

Table Name	Description
Table 001	D = 273, НУЭ, Рис. 5.24
Table 002	D = 325, НУЭ, Рис. 5.25
Table 003	D = 377, НУЭ, Рис. 5.26
Table 004	D = 426, НУЭ, Рис. 5.27
Table 005	D = 465, НУЭ, Рис. 5.28
Table 006	D = 530, НУЭ, Рис. 5.29

Fields:

Field Name	Data Type	Field Size	Descriptor
Table Name	TEXT	16	Table Name
Description	TEXT	64	Description/Comment

➢ Spreadsheet containing digital nomograms (***Table 001, Table 002 … Table 069***):

Table 001

_ID	FH	FV
1	0	978.7
2	7142.8	5016.2
3	10212	6716.2

Fields:

Table 001 | **Table 006**

Field Name	Data Type	Description
_ID	AutoNumber	AutoNumber
FH	Number	Horizontal Reaction
FV	Number	Vertical Reaction

«Units Force» Table with measurement units for forces:

Units Force

Label	LabelIn	LabelOut	FactorIn	FactorOut
Kilonewton	kN	N	1000	1
Newton	N	N	1	1

➢ ***«Units Moments»*** Table with measurement units for moments, :

Units Moment

Label	LabelIn	LabelOut	FactorIn	FactorOut
Newton*m	N*m	N*mm	1000	1
Newton*mm	N*mm	N*mm	1	1

Fields:

Field Name	Data Type	Field Size	Descriptor
Label	Text	16	Name of the Unit
LabelIn	Text	8	Abbreviation of input units
LabelOut	Text	8	Abbreviation of output units
FactorIn	Number	-	Conversion factor between input units and Newton
FactorOut	Number	-	Conversion factor between Newton and output units

➢ Spreadsheets with values of permissible loads on supports

СТО 79814898

Temp	Design	Data Type	FA1	FH1	FN1	MA1	MH1	MN1	FA2	FH2
200	1	0	0	0	9800	0	0	0	0	0
200	19	0	0	0	150100	0	0	0	0	0
200	92	0	0	12500	44500	0	0	0	0	17500
200	94	0	0	13000	46500	0	0	0	0	18000
200	96	0	0	20500	54500	0	0	0	0	28000
200	98	0	0	24000	61500	0	0	0	0	33000
200	100	0	0	27000	68000	0	0	0	0	37500
200	102	0	0	29000	71500	0	0	0	0	40500
200	104	0	0	46500	123000	0	0	0	0	64000
200	106	0	0	66000	126500	0	0	0	0	90000
200	108	0	0	72500	126500	0	0	0	0	100000
200	1	0	0	0	3000	0	0	0	0	0
200	21	0	0	0	157200	0	0	0	0	0
200	3	0	0	0	5500	0	0	0	0	0
200	5	0	0	0	7000	0	0	0	0	0
200	7	0	0	0	11500	0	0	0	0	0

Fields:

Field name	Data type	Field Size	Description
Label	Text	8	Label of the current record (for reference in the DBS_REF parameter)
D	Text	8	Nominal diameter of the piping system; it is used as auxiliary information
Mater	Text	16	Type of the piping system material (CS - carbon steel, AUS - austenite steel, SS - stainless steel); it is used as auxiliary information
Temp	Text	8	Temperature, C; it is used as auxiliary information
Design	Text	16	Support designation in the corresponding catalog; it is used as auxiliary information and appears in printouts
Data Type	Number	-	Data type: integer from 0 to 4; see Note (1)
FA1	Number	-	Set of permissible loads for the first mode indicated in parameters MODE и LOAD of the SUP_LOADS command, see Note (2)
FH1	Number		
FN1	Number		
MA1	Number		
MH1	Number		
MN1	Number		
FA2	Number	-	Set of permissible loads for the second mode indicated in parameters MODE и LOAD of the SUP_LOADS command, see Note (2)
FH2	Number		
FN2	Number		
MA2	Number		
MH2	Number		

MN2	Number		
FA3	Number	-	Set of permissible loads for the third mode indicated in parameters MODE и LOAD of the SUP_LOADS command, see Note (2)
FH3	Number		
FN3	Number		
MA3	Number		
MH3	Number		
MN3	Number		
Nomogram1	Text	16	It is used if DataType = 2: name of spreadsheet with digital nomogram for the first analysis mode
Nomogram2	Text	16	It is used if DataType = 2: name of spreadsheet with digital nomogram for the second analysis mode
Nomogram3	Text	16	It is used if DataType = 2: name of spreadsheet with digital nomogram for the third analysis mode
Sup Type	Text	8	Type of support (ANC, NZL, SUP, STG, etc.); it is used as auxiliary information
Note	Text	255	Comment; it is used as auxiliary information

Note:

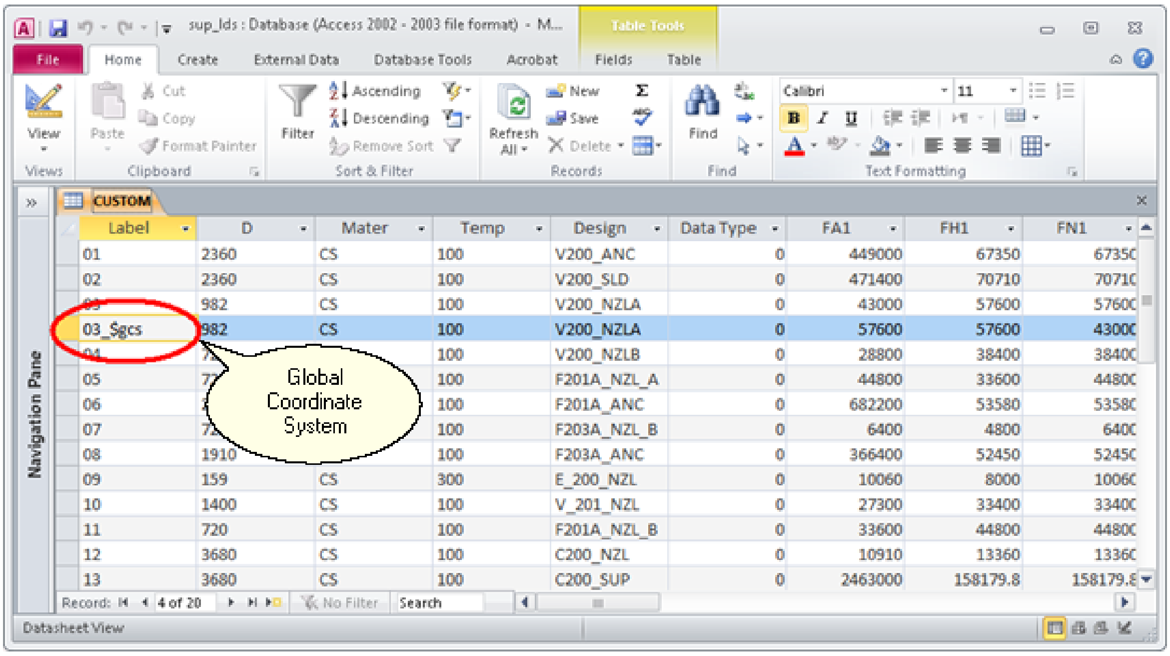
1) Data type - integer, which specifies the processing of permissible loads:
   - 0 - component-wise comparison of design and permissible values;
   - 1 - vertical component in case of reaction with the sign "+" accounts for 50% of the permissible value;
   - 2 - permissible lateral load is a function of the vertical component : FH = f(FV). The relations are specified in the form of nomograms. The nomogram number is entered into the field for forces with the negative sign. The nomograms themselves are described in the «Nomograms» spreadsheet;
   - 3 - vertical component in case of reaction with the sign "+" is determined by the value in the MN column;
   - 4 - load-bearing capacity of the support is determined according to the following iteration formula: fn/FN + fh/FH <= 1;
   - 5 - assessment of support’s load capacity according to NP-068-05, *[REF 14]*: calculated axial forces and SRSSSquare Root of Sum Squares of resulting moment are compared with FA and MA values. Applicable only for SUP and ANC.
   - 6 - Applicable only for SUP and ANC. Comparison is made for axial and shear forces and torsional/bending moments. Shear Force and Bending Moment are calculated as SRSS from corresponding components. Allowable values should be set in the following order: F_A F_S 0. M_T M_B 0., where: F_A and F_S – allowable axial and shear forces; M_T and M_B – allowable torsional and bending moments.
   - 7 - assessment is made by interaction method for all load components: SUM(|(f)/(F)|) < 1. For supports with different upward and downward loads, the positive vertical component is defined in the field for FA2
   - 8 - Assessment of fix supports on the component by component basis. For supports with different upward and downward loads, the positive vertical component is defined in the field for FN2

9 - Allowable load is set in the field FN1. Comparison with vertical and lateral components of calculated loads is made by SRSS rule: $F_N \geq \sqrt{F_X^2 + F_Z^2}$; Axial load is limited by the value of $0.3*F_N$

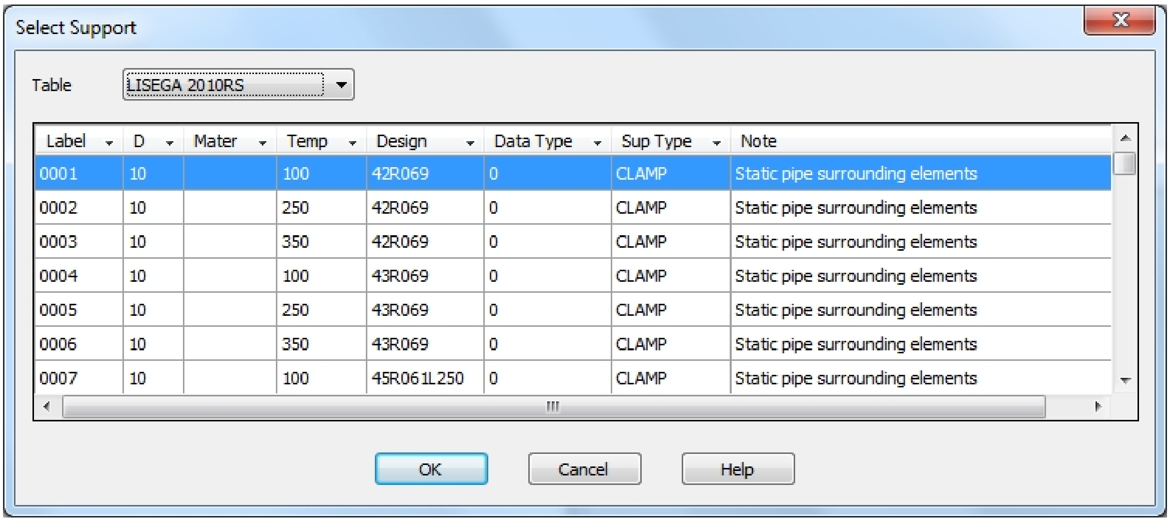
10 - Allowable load is set in the field FN1. Comparison with all three components of calculated loads is made by SRSS rule: $F_N \geq \sqrt{F_X^2 + F_Y^2 + F_Z^2}$

Name of the Table	Data Type										
	0	1	2	3	4	5	6	7	8	9	10
LISEGA 2010RS	X	X		X	X						
LISEGA2010RSENR2	X			X	X					X	X
OTT 0157-2013								X			
TITAN-2_R1_7	X			X					X		
НП-068-05-АР						X					
НП-068-05-МРЗ						X					
НП-068-05-ПЗ						X					
ОСТ 24-125-01	X		X								
СПиР-О-2008	X										
СТО 79814898	X	X									
CUSTOM	User Defined										

2) The coordinate system for the values of permissible loads is determined depending upon the mark of the current record: if the '$GCS' chain of characters is present in the mark name (case is of no importance), then it is assumed that the loads are specified in the global coordinate system. Otherwise, they are specified in the local coordinate system:



3) When the "View" button of the load selection dialog is pressed, the following dialog will be displayed:



which fields correspond to the values described above. Using integrated filters, one can quickly find the required reference.

➢ Tables "LOADS" with amplification factors used for scaling nominal loads depending on load combination and pipe's seismic category (field LOADS of the SUPPORTS table)

ID	Label	RUS_TEXT	S_CAT_I	S_CAT_II	S_CAT_III
1	NOC	НУЭ	1	1	1
2	AOO	ННУЭ	1.2	1.2	1.2
4	NOC+OBE	НУЭ+ПЗ	1.2	1.5	0
5	AOO+OBE	ННУЭ+ПЗ	1.2	1.5	0

6	DBA	УПА	1.4	1.4	1.4
7	NOC+SSE	НУЭ+МРЗ	1.4	0	0
8	AOO+SSE	ННУЭ+МРЗ	1.4	0	0
9	DBA+OBE	УПА+ПЗ	1.5	0	0

Note: zero value indicates that the specified load combination is not applicable for corresponding seismic category

Field name	Data type	Field Size	Description
Label	Short Text	255	Label
RUS_TEXT	Short Text	255	Name to be shown in GUI
S_CAT_I	Number	-	Seismic Category I
S_CAT_II	Number		Seismic Category II
S_CAT_III	Number		Seismic Category III

- Tables ***Temperature*** with correction factors used for scaling nominal loads depending on the piping operational temperature (field TEMPERATURES of the SUPPORTS table)

ID	MAT_GROUP	Temperature, °C	FS
1	1	50	1
2	1	150	1
3	1	250	0.8
4	2	350	0.7

22 Appendix XIII

CKTI approach for stress analysis of TEEs and Branch Connections in compliance with chapter 5.2 of RD 10-249-98.

Computation of the Stage I effective stresses (item 5.2.6.2.5):

$$\sigma_{эф} = 0.5\sigma_{пр} + \max(\Omega; 1.0)\frac{k_п\sqrt{M_x^2 + M_y^2 + M_z^2}}{W}$$

Computation of the Stage II effective stresses (item 5.2.6.3.4):

$$\sigma_{эф} = \sigma_{пр} + \max(0.6\gamma_m; 1.0)\frac{k_п\sqrt{M_x^2 + M_y^2 + M_z^2}}{W}$$

Computation of the Stage III effective stresses (item 5.2.6.4.4):

$$\sigma_{эф} = 2\sigma_{пр} + \max(\gamma_m; 3.0)\frac{k_п\sqrt{M_x^2 + M_y^2 + M_z^2}}{W}$$

Computation of the Stage IV effective stresses (item 5.2.6.5.4):

$$\sigma_{эф} = \max(0.6\gamma_m; 1.0)\frac{k_п\sqrt{M_x^2 + M_y^2 + M_z^2}}{W}$$

Calculations according to the above equations are performed for three elements joined in the intersection node (sections 1-1, 2-2 and 3-3 from Figure 5.12, Chapter 5.2, RD 10-249-98)

Equivalent stresses due to pressure ($\sigma_{пр}$) are calculated according to equation (3), item 5.2.6.2.2 RD 10-249-98 with use of the actual element's sizes.

Bending moments M_X, M_Y, M_Z acting in the considered sections are determined according to Figure 5.14, Chapter 5.2, RD 10-249-98.

Overload factor *kn* is taken according to item 5.2.6.2.4, RD 10-249-98.

Section modulus W used for sections 1-1 and 2-2 is defined according to the actual dimensions of the TEE body. Section modulus for section 3-3 is defined on the basis of branch dimensions (for extruded and forged tees matched pipe should be used)

Coefficients Ω и γ_m are defined depending on location and type of tee joint with use of the following formulas:

Type of Tee joint	Branch (sec. 3-3)	RUN (sec. 1-1 and 2-2)
Branch Connections, unreinforced and reinforced tees.	$\Omega = 0.8\left(\frac{R}{T}\right)^{0.67}\left(\min(\frac{r}{R};0.5)\right)^{0.5}\left(\frac{t}{T}\right)$	$\Omega = 0.9\left(\frac{r}{t}\right)^{0.3}$
	$\gamma_m = A\left(\frac{R}{T}\right)^{0.67}\left(\frac{r}{R}\right)^{0.5}\left(\frac{t}{T}\right)^{1}$ $A = \begin{cases}3.0\ if\ (r/R) \le 0.9\\ 1.8\ if\ (r/R) = 1.0\end{cases}$	$\gamma_m = A\left(\frac{R}{T}\right)^{0.67}\left(\frac{r}{R}\right)$

Welded, extruded and forged ($r_c > 0.1r$) tees	$\Omega = 0.8\left(\frac{R}{T}\right)^{0.67}\left(\min(\frac{r}{R};0.5)\right)^{0.5}\left(\frac{t}{T}\right)$	$\Omega = 0.9\left(\frac{r}{t}\right)^{0.3}$
	$\gamma_m = A\left(\frac{R}{\left(1+\frac{r_c}{R}\right)T}\right)^{0.67}\left(\frac{r}{R}\right)^{0.5}$ $A = \begin{cases}3.0\ if\ (r/R) \le 0.9\\ 1.8\ if\ (r/R) = 1.0\end{cases}$	$\gamma_m = 1.6\left(\frac{R}{\left(1+\frac{r_c}{R}\right)T}\right)^{0.67}$

Notes:

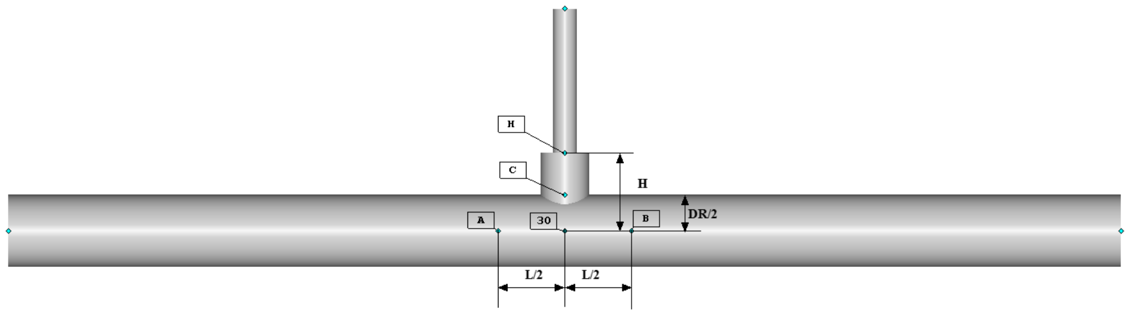
1) Designations:
   R – RUN mean radius;
   T – RUN wall thickness;
   r – BRANCH mean radius (for extruded and forged tees dimensions of the matched pipe should be used)
   t – BRANCH wall thickness (for extruded and forged tees dimensions of the matched pipe should be used)
   r_c – radius of curvature of external contoured portion of outlet
2) Value of A in the range of 0,9 < (r/R) < 1.0 is defined by linear interpolation.
3) $\tau = \begin{cases}0.4\ if\ (t/T) < 1\\ 1.0\ if\ (t/T) \ge 1\end{cases}$
4) For reinforced tees run thickness T could be substituted by effective value: $T_C=T+0.5t_r$, where tr is pad's thickness ($t_r \le 1.5T$).

.

23 Appendix XIV

Modeling of the tee/branch connection's joints.

Depending from the TEE_FLEX option dPIPE may analyze tee/branch connection's joints taking into account their local flexibility. To do that dPIPE automatically split existing mesh according to the data for "standard tee" (see command TEE):



a. dPIPE creates weightless element «30 – C» with length equal to the half diameter of the header pipe;

b. Flexibility of the joint is located in the point “C” that is intersection of the branch pipe and header’s outer surface;

c. If the length of the header is set, dPIPE creates two elements: «30-A» and «30-B» with outer diameter = DR and wall thickness = TR. If the weight W of the fitting is not explicitly defined, these elements inherit weigh per length that was set for header pipe. Otherwise, their weight per length is calculated as W/L. Insulation weight is defined as $IWGT = IWGT_{RP}*(DR/OD\_rp)$, where $IWGT_{RP}$ and OD_rp are insulation’s weigh per length and outer diameter of the header pipe.

d. If the height H of the branch is set, dPIPE creates element “C-H”: OD = DB, t = TB, mat = MAT. If the weight W of the fitting is not explicitly defined, this element inherits weigh per length from the branch pipe. If the length of the header L is set, element “C-H” is weightless (w = 0), else: w = W/(H-DR/2). Insulation’s weigh per length in this case is taken from the branch pipe: $IWGT = IWGT_{BP}$

24 Appendix XV

Appendix XV. Engineering methodology for seismic assessment of small bore pipes (SBP).

Dividing the seismic load into two parts is an accepted approach for piping seismic analysis:

- primary loads due to inertia. When corresponding stresses are calculated these loads are combined with other sustained loads, such as weight loads and internal pressure;
- secondary loads due to seismic anchor movements (SAM).

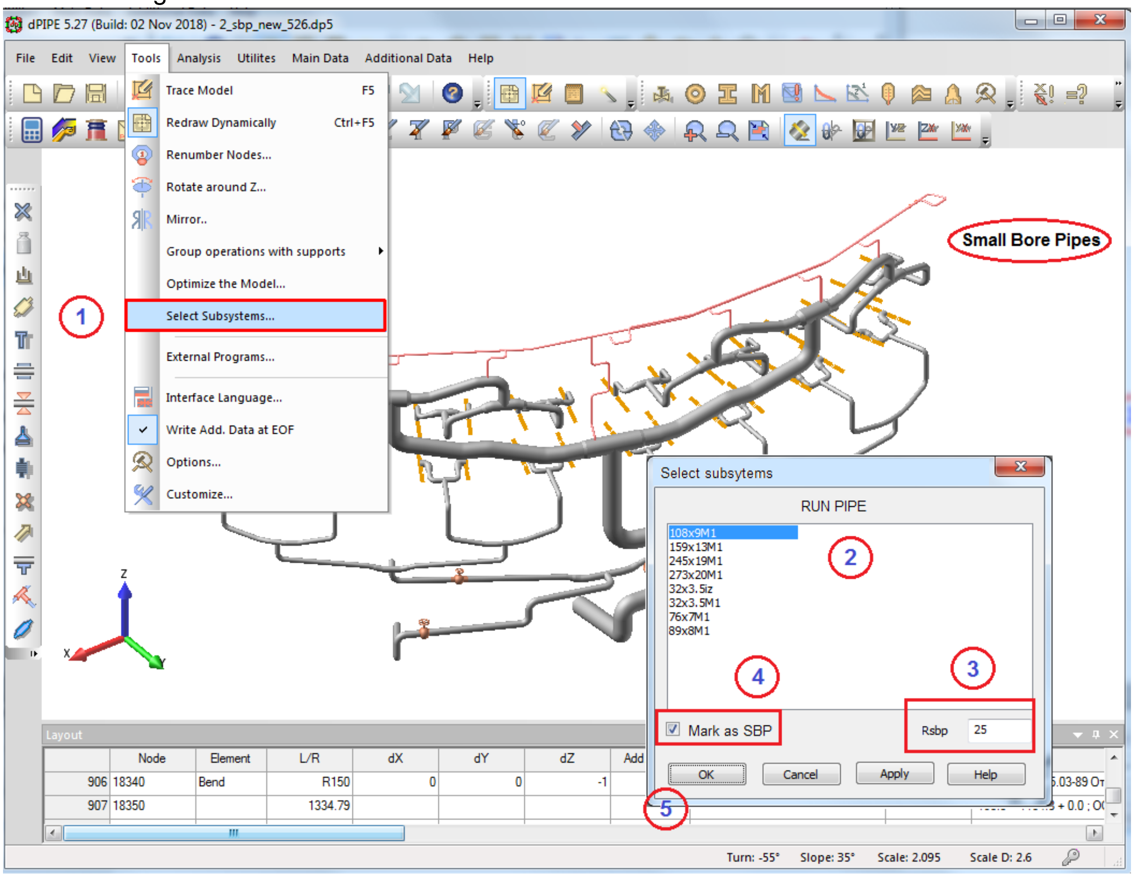
The presented methodology is aimed to reduce the conservatism of the seismic analyses for small bore pipes when they are considering in one complex model together with big size pipes. The seismic load acting on the small bore pipes is transmitted both: via their supporting system and through the big pipes to which small bore pipes are connected. Seismic stresses in this case are calculated on the base of internal loads derived from the combination of inertial and SAM loads. The last ones in this case may be considered as loads from the motion of big size pipes.

To avoid this effect, the following sequence of steps is proposed as a part of the dPIPE calculations:

1. Select piping segments that may be classified as “small bore pipes” with help of the decoupling criteria. One of such criteria proposed in the literature is the ratio of run to branch pipe moment of inertia [REF 18]:

$$R_{SBP} = \frac{I_{RUN}}{I_{BRANCH}} > 25$$

⊟ How to manage that in dPIPE:



2. Run conventional seismic analysis of the whole system. Resulting internal forces in the small bore pipes would consist from the combination of the primary (inertial) seismic loads, F_{SBP}^{INRT}, and secondary (displacement based) components, F_{SBP}^{SAM}:

$$F_{SBP}^{(1)} = F_{SBP}^{INRT} + F_{SBP}^{SAM}$$

Under this analysis corresponding *seismic equations are checked only for big pipes*

3. Set to zero weight of small bore pipes segments and run the seismic analysis again. Now inertial seismic loads are excluded for small pipes and only secondary loads coming from the seismic motion of the big pipes are applied:

$$F_{SBP}^{(2)} = F_{SBP}^{SAM}$$

Results of this analysis are used for the check of stress equations that consider *secondary loads for small bore pipes* (like *EQ. 10 of ASME NC-3600*)

4. Create load combination:

$$F_{SBP}^{(3)} = F_{SBP}^{(1)} - F_{SBP}^{(2)} = F_{SBP}^{INRT}$$

These loads are used for the check of *seismic stresses for the small bore pipes*

25 Appendix XVI

Appendix XVI. Export of results of calculations from dPIPE to LICAD

1. The export of pipe supports loads from dPIPE to LICAD® is carried out depending on the LCD_VER parameter both for the “classic” version “LICAD-10” and for the version adapted for Russian Standards - “LICAD-RS-EN”
2. For the “LICAD-RS-EN” version, the following types of loads and their combinations are available:

Loads Combination	Designation (TYPE)	piping seismic category		
		I	II	III
Cold	COLD	1	1	1
Hot	HOT			
NOC	NE	1	1	1
Abnorm.	NNE	1.2	1.2	1.2
LOCA	UPA	1.4	1.4	1.4
HTEST	HTEST	1	1	1
NOC+OBE	NE_PZ	1.2	1.5	1.5
Abnorm+OBE	NNE_PZ	1.2	1.5	-
УПА+OBE	UPA_PZ	1.5	-	-
NOC+SSE	NE_MRZ	1.4	-	-
Abnorm+SSE	NNE_MRZ	1.4	-	-

3. For the “LICAD-10” the following loads combination and designations are applicable:

Loads Combination	Designation (TYPE)
Cold	COLD
Hot	HOT
Service Level A/B loads	Level_AB
Service Level C loads	Level_C

Service Level D loads	Level_D
Testing	TEST

Responsibility for assigning SF coefficients is after the USER: these coefficients are selected according to Code used for analysis

4. The types of loads “COLD” and “HOT” should correspond to those Load Cases from dPIPE that were used for the spring hangers design. These references are mandatory for the models with spring hangers/supports

5. Upon successful completion of the calculations, two files are created in the working folder:
   - file.lrs.csv is an Excel file (csv). To enter into LICAD, it must be saved in xls format.
   - file.lrs_log.csv is an Excel file (csv) with error messages and notes.

6. The loads on the supports/hangers are exported in kN, displacements and linear dimensions are given in mm

7. The table below presents correspondence between dPIPE types of the Supports/Hangers and LICAD designations

Supports in dPIPE	**LICAD Configuration**
Spring hanger on the horizontal run of the pipe	S13 (one chain) S02 (two chains)
Spring hanger on the riser	S32 (two chains)
Spring support on the horizontal run of the pipe	S29 (one chain)
Rigid hanger on the horizontal run of the pipe	S45 (one chain) S40 (two chains)
Rod hanger on the riser	S52 (two chains)
Fix Support	S30/FP
One-way restraint	S30
Linear Restraint	S30/G2P
Guide support	S30/40GS (OD < 32 mm) S30/49GS (OD ≥ 32 mm)
Rigid Strut	Y01
Snubber	Y02

8. The coordinate system used for fixed and guide supports come from LICAD:

- the x axis lies in a horizontal plane and is perpendicular to the axis of the pipe;
- the y axis is directed along the pipe axis;

- the z axis is directed upwards and perpendicular to the x and y axes

Accordingly, the following loads are exported to LICAD:

- Fx – lateral load;
- Fy – axial load;
- Fz(+) – vertical load (tension);
- Fz(–) – vertical load (compression);

LICAD does not provide instructions on how to interpret the loads if support is located on the vertical run of the pipe. In this case converter exporting loads makes them symmetrical:

Fx = Fz(+) = Fz(–) = MAX{ |Fx|; |Fz| }.

If the support sustains the maximal load, it can be oriented around the axis of the pipe in any direction. If it fails, then (knowing the orientation), one can try to set the load manually. More strong supports are not selecting since for each diameter there is only one 49FP/GS support (LICAD-RS-EN) for each diameter.

26 Appendix XVII

Appendix XVII. The procedure for calculating of fatigue curves according to GOST R 59115.9-2021[REF 24] in the dPIPE program

The allowable amplitude of the elastically calculated stress or the allowable number of cycles at $[N_0] \leq 10^{12}$ is calculated by the formulas, considering the maximum influence of the cycle asymmetry (clause 10.12):

$$[\sigma_{aF}] = \frac{E^T e_c^T}{(4n_N[N_0])^m} + \frac{R_c^T - R_{p0,2}^{T_{min}}}{(4n_N[N_0])^{m_c} - 1}$$

$$[\sigma_{aF}] = \frac{E^T e_c^T}{n_\sigma (4[N_0])^m} + \frac{R_c^T - R_{p0,2}^{T_{min}}}{n_\sigma \left((4[N_0])^{m_c} - 1\right)}$$

The smallest value from $[\sigma_{aF}]$ or $[N_0]$ is taken for the consequent analyses. The parameters included in the above formulas are determined in accordance with clause 10.8 only for the case:

$R^T_m \leq 700$ MPa; $n_\sigma = 2$; $n_N = 10$; $R^T_c = R^T_m (1+0.014Z^T)$;

$$e_c^T = 1.15\lg\left(\frac{100}{100 - Z_c^T}\right) - 0.025$$

$$Z_c^T = \min\left\{Z^T; 50\%\right\} \quad (Z^T \text{ according to GOST R 59115.3})$$

$$m = 0.5 \qquad npu\ R_m^T \le 700\ MPa$$
$$m = 0.36 + 0.0002R_m^T \qquad npu\ 700 < R_m^T \le 1200\ MPa$$

$$m_e = 0.132\lg\left[\frac{R_m^T}{R_{-1}^T}\left(1 + 0.014Z^T\right)\right]$$

$$R_{-1}^T = 0.4R_m^T \qquad if\ R_m^T \le 700\ MPa$$
$$R_{-1}^T = \left(0.54 - 0.0002R_m^T\right)R_m^T \qquad if\ 700 < R_m^T \le 1200\ MPa$$

Data for E^T, Z^T, $R^T{}_m$ is taken equal to the smallest values found in the range of operating temperatures for the considered cycle. The value of $R^{T\,min}{}_{p0,2}$ is determined at the minimum temperature of the cycle.

E^T - Elastic Modulus, MPa;
$R^T{}_m$ - minimal value of the tensile strength, MPa;
$R^T{}_{p0,2}$ - minimal value of the yield strength, MPa;
Z^T - reduction of area, %.

Fatigue Evaluation including Environmental Effects

1) Influence of the water medium on the fatigue has to be considered if the temperature of water T > 150°C. There is a gap between the curve in air and in water, since $F_{pn} \approx 2$ at T = 150°C.

Limitations of the methodology:

- water medium: T ≤ 350°C;
- number of cycles: $[N_0] \le 10^{12}$.

2) Allowable stress amplitude $[\sigma_{aF}]$ that corresponds to number of cycles N_0 or allowable number of cycles $[N_0]$ corresponded to (σ_{aF}) is taken as minimum between two values calculated in assumption of the maximal influence of the cycle's asymmetry (clauses B.4 and B.9):

$$[\sigma_{aF}] = \frac{E^T e_c^{20}}{\left(4n_N F_{pn}[N_0]\right)^{0,5}} + \frac{R_{cF}^T - R_{p0,2}^{T_{\min}}}{\left(4n_N[N_0]\right)^{m_{eF}} - 1}$$

$$[\sigma_{aF}] = \frac{E^T e_c^{20}}{n_\sigma\left(4F_{pn}[N_0]\right)^{0,5}} + \frac{R_{cF}^T - R_{p0,2}^{T_{\min}}}{n_\sigma\left(\left(4[N_0]\right)^{m_{eF}} - 1\right)}$$

where:

$$n_\sigma = 2\,;\; n_N = 10$$

$$e_c^T = 1.15\lg\left(\frac{100}{100 - Z_c^{20}}\right) - 0.025$$

$$Z_c^{20} = \min\left\{Z^{20}; 50\%\right\}$$

$$R_{cF}^T = R_m^{20}\left(1 + 0.014 Z_F\right)$$

$$m_{eF} = 0.132\lg\left[2.5\left(1 + 0.014 Z_F\right)\right]$$

$$Z_F = 100 - \frac{100}{\exp\left(2e_c^{20} / \sqrt{F_{pn}}\right)}$$

The coefficient F_{pn} is conservatively calculated according to clause B.8 depending on the temperature of the metal T, the sulfur content in the metal S, the oxygen concentration in the water KO and the strain rate $\dot{\varepsilon}$

- for carbon steels:

$$F_{pn} = \exp\left(0.632 - 0.101 \cdot S^* \cdot T^* \cdot O^* \cdot \dot{\varepsilon}^*\right)$$

- for alloyed chromium-molybdenum and chromium-molybdenum-vanadium steels:

$$F_{pn} = \exp\left(0.702 - 0.101 \cdot S^* \cdot T^* \cdot O^* \cdot \dot{\varepsilon}^*\right)$$

$S^* = 0.015$ (conservatively taken as $S > 0.015\%$)

$T^* = T - 150$ (T - maximal temperature of the cycle, °C)

$O^* = 0$ $\quad$ *if* $KO \le 0.04\ mg/kg$

$O^* = \ln\left(\dfrac{KO}{0.04}\right)$ $\quad$ *if* $0.04 < KO \le 0.5\ mg/kg$

$O^* = \ln(12.5)$ $\quad$ *if* $KO > 0.5\ mg/kg$

$\dot{\varepsilon}^* = \ln(0.001)$ (conservatively taken as $\dot{\varepsilon} \le 0.001\% sec^{-1}$)

- for austenitic steels:

$$F_{pn} = \exp\left(0.734 - T^* \cdot O^* \cdot \dot{\varepsilon}^*\right)$$

$T^* = \min\left\{\dfrac{T-150}{175}; 1\right\}$ (T - maximal temperature of the cycle, °C)

$O^* = 0.281$ (for all values of KO)

$\dot{\varepsilon}^* = \ln(0.001)$ (conservatively taken as $\dot{\varepsilon} \le 0.0004\% sec^{-1}$)

In accordance with clause B.13, the allowable number of cycles must be taken as the smallest of those determined in air according to 10.12 and in water according to B.4-B.8 (***the curves in air and in water are not consistent with each other: the number of allowable cycles in water can be more than in air***).

3) The allowable stress amplitude for the number of cycles N_0 or the allowable number of cycles for a given stress amplitude (σ_{aF}) for a welded joint is defined as (clause B.12):

$$[\sigma_{aF}]_s = \min\left\{\varphi_s [\sigma_{aF}]^*; [\sigma_{aF}]\right\}$$

$$[N_0]_s = \min\left\{[N_0]^*; [N_0]\right\}$$

where:

$[\sigma_{aF}]^*$ - allowable stress amplitude on air;

$[N_0]^*$ - allowable number of cycles on air, calculated for stress amplitude $(\sigma_{aF})_s = (\sigma_{aF})/\varphi_s$.

27 Appendix XVIII

Appendix XVIII. Criteria for postulated rupture locations of HE piping adopted for different Codes

1. *ASME BPVC NB-3600 (Class 1 Nuclear Piping)*

Locations of ruptures of Class 1 piping are determined by checking the following inequalities (exceeding the value of 2.4 Sm corresponds to a rupture, 1.2 Sm - to a crack):

Equation (10), NB-3653.1:

$$S_n = C_1 \frac{P_0 D_0}{2t} + C_2 \frac{D_0}{2I} M_i^{(10)} + C_3 E_{ab} \times \left| \alpha_a T_a - \alpha_b T_b \right| > \begin{Bmatrix} 2.4 S_m \\ 1.2 S_m \end{Bmatrix}$$

Equation (12) and (13), NB-3653.6:

$$S_e = C_2 \frac{D_0}{2I} M_i^{(12)} > \begin{Bmatrix} 2.4 S_m \\ 1.2 S_m \end{Bmatrix}$$

$$S = C_1 \frac{P_0 D_0}{2t} + C_2 \frac{D_0}{2I} M_i^{(13)} + C_3' E_{ab} \times \left| \alpha_a T_a - \alpha_b T_b \right| > \begin{Bmatrix} 2.4 S_m \\ 1.2 S_m \end{Bmatrix}$$

where:

Sn, Se, S are stress ranges calculated according to equations (10) – (13);

C_1, C_2, C_3, C_3' are secondary stress indices for the specific component under investigation (NB-3680);

P_0 is range of service pressure;

D_0 is outside diameter of pipe;

t is nominal wall thickness of product;

I is moment of inertia;

$M_i^{(10)}$ is resultant range of moment that occurs when the system goes from one service load set to another. Service loads and combinations are provided in the Design Specification. In the combination of moments from load sets, all directional moment components in the same direction shall be combined before determining the resultant moment. If the method of analysis is such that only magnitudes without relative algebraic signs are obtained, the most conservative combination shall be assumed;

$M_i^{(12)}$ is same as $M_i^{(10)}$ for the pair of load sets under review except it includes only moments due to thermal expansion and thermal anchor movements;

$M_i^{(13)}$ is same as $M_i^{(10)}$ for the pair of load sets under review except it excludes the moments due to thermal expansion and thermal anchor movements;

T_a, T_b, α_a, α_b are range of average temperature and coefficients of thermal expansion on side a(b) of gross structural discontinuity or material discontinuity;

E_{ab} is average modulus of elasticity of the two sides of a gross structural discontinuity or material discontinuity at room temperature;

S_m – is average of the allowable stress intensity value for the highest and the lowest temperatures of the metal during the transient.

Additional criteria for Class 1 piping ruptures is the value of the Cumulated Usage Factor U. The break is postulated if this value exceeds 0.1. However, this value can be taken as 0.4 when the effects of environmental assisted fatigue (EAF) are considered in the piping design.

2. *ASME BPVC NCD-3600 (Class 2 and 3 Nuclear Piping)*

Locations of ruptures are determined by checking the following inequalities (0.8 - break; 0.4 - crack):

$$S_{OL} + S_E > \begin{Bmatrix} 0.8(1.8S_h + S_a) \\ 0.4(1.8S_h + S_a) \end{Bmatrix}$$

$$S_a = f(1.25S_c + 0.25S_h)$$

where:

S_{OL} are stresses due to pressure, weight, and occasional loads for OBE level (NCD-3653, Equation 9)

S_E are thermal expansion stresses (NCD-3653.2, Equation 10)

S_a is allowable stress range for expansion stresses;

S_c is basic material allowable stress at minimum (cold) temperature;

S_h is basic material allowable stress at maximum (hot) temperature;

f is stress range reduction factor for cyclic conditions, is defined according to Table NC-3611.2(e)-1.

3. *Adaptation of the Methodology for Postulated Rupture Locations in High Energy Lines to the PNAE Code*

For calculation of the High Energy Lines according to the PNAE Code, [REF1], it is suggested to use methodology similar to the one used for rupture locations of ASME Class 1 piping according to ASME NB, [REF3]

For this purpose, the additional stresses are defined:

- σ'_{RK} - stress range calculated by formulas of item 2.3.2, Appendix 5 of PNAE Code, from the moments due to thermal expansion and thermal anchor movements. These stresses are analogue of Equation (12) of ASME NB-3600;
- σ''_{RK} - is stress range calculated by formulas of item 2.3.2, Appendix 5 of PNAE Code, except it excludes from the loading moments due to thermal expansion and thermal anchor movements. Additionally, these stresses include membrane temperature stresses due to gross structural or material discontinuity. These stresses are analogue of Equation (13) of ASME NB-3600

Intermediate locations of piping ruptures are postulated based on the following piping stress and fatigue analysis criteria:

- the circumferential or longitudinal break is postulated if:

  $$\sigma'_{RK} > 0.8 \times 2R^T_{p0,2} = 1.6R^T_{p0,2}$$

  or

  $$\sigma''_{RK} > 0.8 \times 2R^T_{p0,2} = 1.6R^T_{p0,2}$$

  In addition to these equations, to locate the postulated break, the value of the cumulated fatigue usage factor U is checked:

  U > 0.1

  However, when the effects of environmental assisted fatigue are considered in the piping design then instead of U>0.16 the following condition may be applied:

  U > 0.4

- The throw-wall crack is postulated if:

  $$\sigma'_{RK} > 0.4 \times 2R^T_{p0,2} = 0.8R^T_{p0,2}$$

  or:

  $$\sigma''_{RK} > 0.4 \times 2R^T_{p0,2} = 0.8R^T_{p0,2}$$

4. *Adaptation of the Methodology for Postulated Rupture Locations in High Energy Lines to the EN 13480 Code*

For calculation of the High Energy Lines according to the EN, [REF10], it is suggested to use methodology similar to the one used for rupture locations of ASME Class 2 piping according to ASME NC

Intermediate locations of piping ruptures are postulated based on the following piping stress and fatigue analysis criteria:

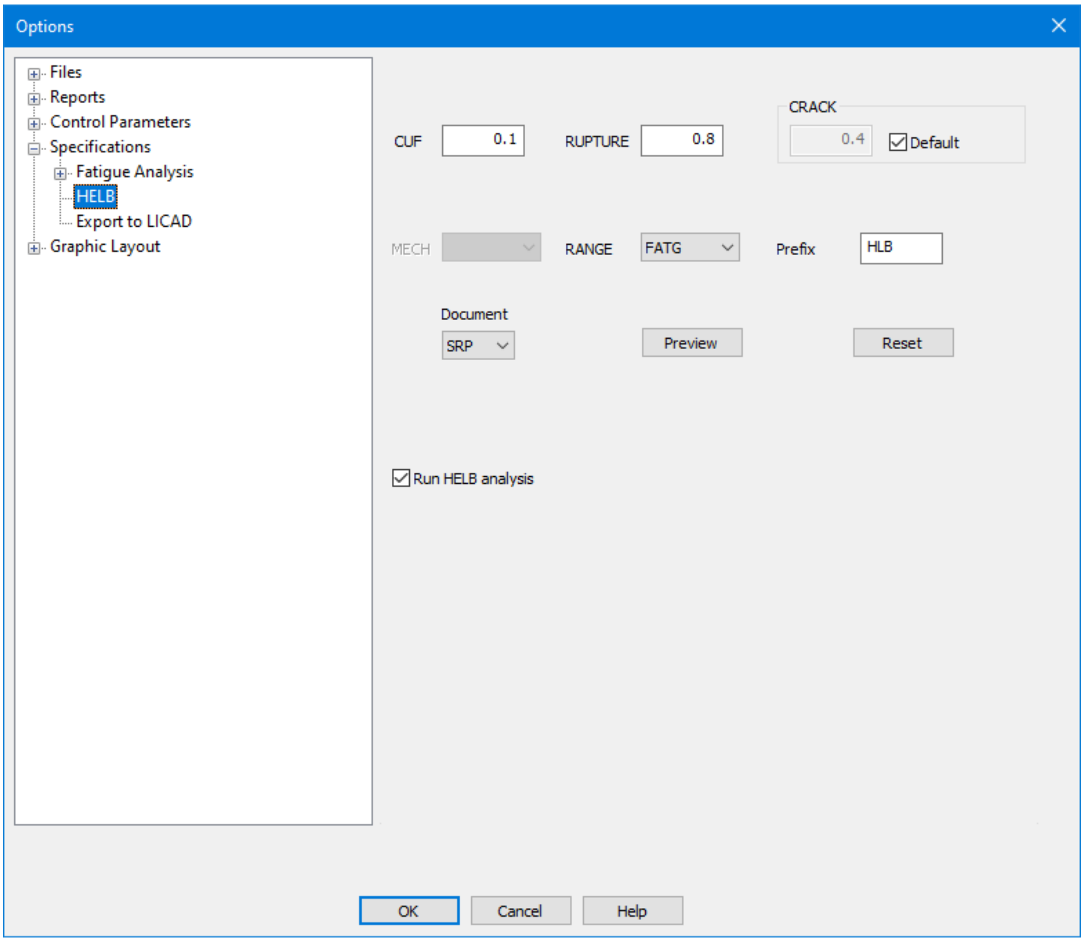
- the circumferential or longitudinal break is postulated if:

  $$\sigma_2 + \sigma_3 > 0.8 \times (1.2 f_f + f_a)$$

- the throw-wall crack is postulated if:

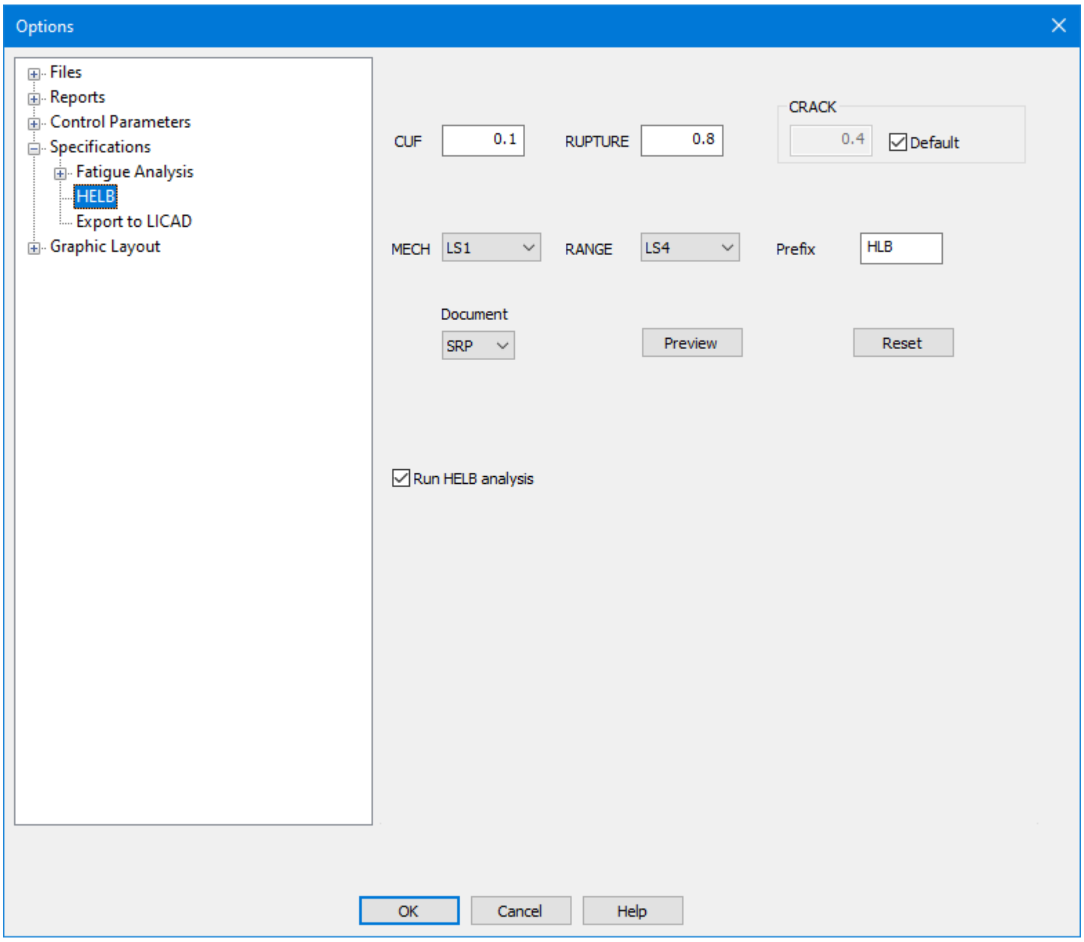
  $$\sigma_2 + \sigma_3 > 0.4 \times (1.2 f_f + f_a)$$

Sample of analysis specification for Codes PNAE and ASME NB:




```
POST HELB cuf = 0.1, rupture = 0.8, range_ls = 'fatg',
```

Sample of analysis specification for Codes EN and ASME NC:




```
POST HELB cuf = 0.1, rupture = 0.8, mech_ls = 'LS1', r
```

28 Appendix XIX

Appendix XIX. Import PCF (Piping Component File) to *.dp5

The PCF2dP preprocessor module is part of the dPIPE 5 and is designed to convert PCF files (PCF – Piping Component Files) obtained from 3D design systems into a file containing a dPIPE calculation model (dP5 files).

Program interface

Модуль The PCF2dP module is launched from the main dPIPE program window by clicking on the icon located on the Utilities toolbar, or by executing the PCF2dP command from the Utilities menu, Figure 1:

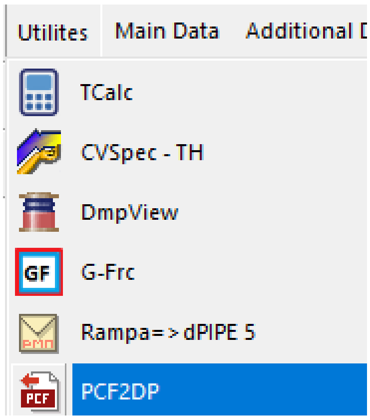


Figure 1. Launching PCF2dP

The PCF2dP dialog box contains two tabs: "General", Figure 2 and "MAP files", Figure 3:

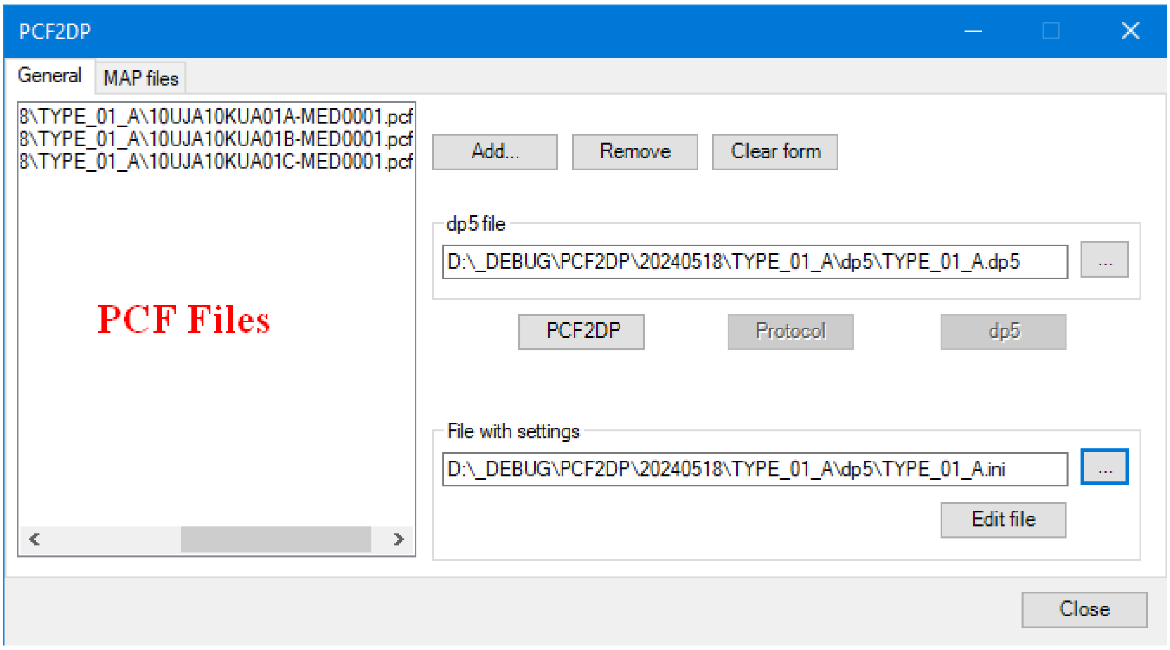


Рисунок 2. Figure 2. PCF2dP dialog box, "General" tab

On the "General" tab, using the "Add" and "Delete" buttons, a set of PCF files to be processed is formed (the files are displayed on the left side of the tab). The "dp5 file" field is used to select the dp5 file that will be generated as a result of PCF2dP operation.

The "File with settings" field allows you to specify the location of the INI configuration file. The "Edit file" button allows you to make changes to it.

The "Protocol" button is used to view the LOG file with messages about the results of data conversion.

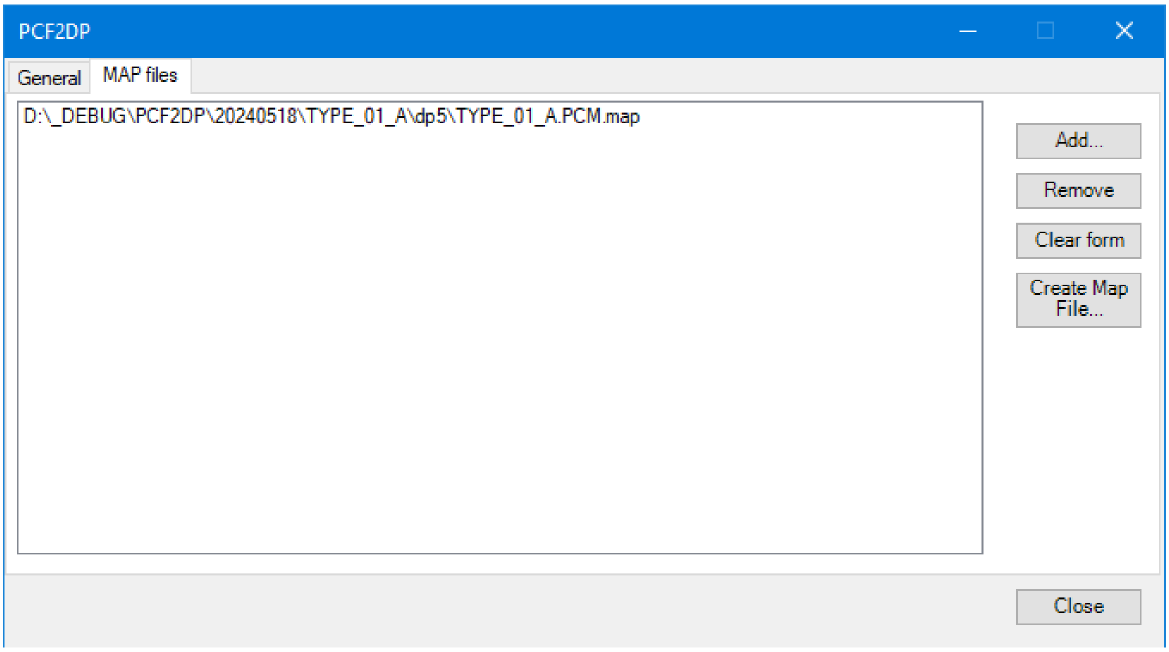


Figure 3. PCF2dP dialog box, "MAP files" tab

The "MAP files" tab is used to create and connect files of correspondence between descriptors from PCF files and dPIPE commands. MAP files are indexed databases created on the basis of PCM files. PCM files (Piping Component Map) are text ASCI files that are specifications for establishing a correspondence between records in PCF files and a description of the corresponding commands in dPIPE. The "Create MAP file" button is used to create MAP files.

The program is launched by clicking the "PCF2dP" button, and by clicking the "dP5" button, the generated file with the calculation model opens in the main dPIPE window.

Figure 4 shows the block diagram of PCF2dP operation: it is assumed that specifications (PCM files) are prepared in advance as part of work on a specific project, establishing the correspondence between the records in the PCF files and the description of the corresponding directives in dPIPE. The SPEC2MAP module transforms a set of these specifications into indexed databases ("MAP" files), which are used directly by PCF2dP. Such a structure allows accumulating and finalizing "databases" from different projects and supplementing specifications on the fly.

When the PCF2dP translator is running, both PCF files and a set of MAP files are inputted. As a result, a dP5 file is generated, which is written to the folder specified in the "dP5 file" field. The dP5 file contains a link to the "materials.inc" file, which should indicate the materials from which the piping spools are made. This file is automatically copied to the same folder where the dP5 file is saved, from the program installation folder: ... \dPIPE 5.28\Utils\PCF2DP\materials.inc. In turn, the "materials.inc" file contains all the materials used for PNAE calculations and are located in the materials database (mat.dbs file). If the names of the materials in the "MATERIALS.INC" file and in the PCF files do not match, the calculation model opens with an error message. In this case, one should edit the material name in the "MATERIALS.INC" file.

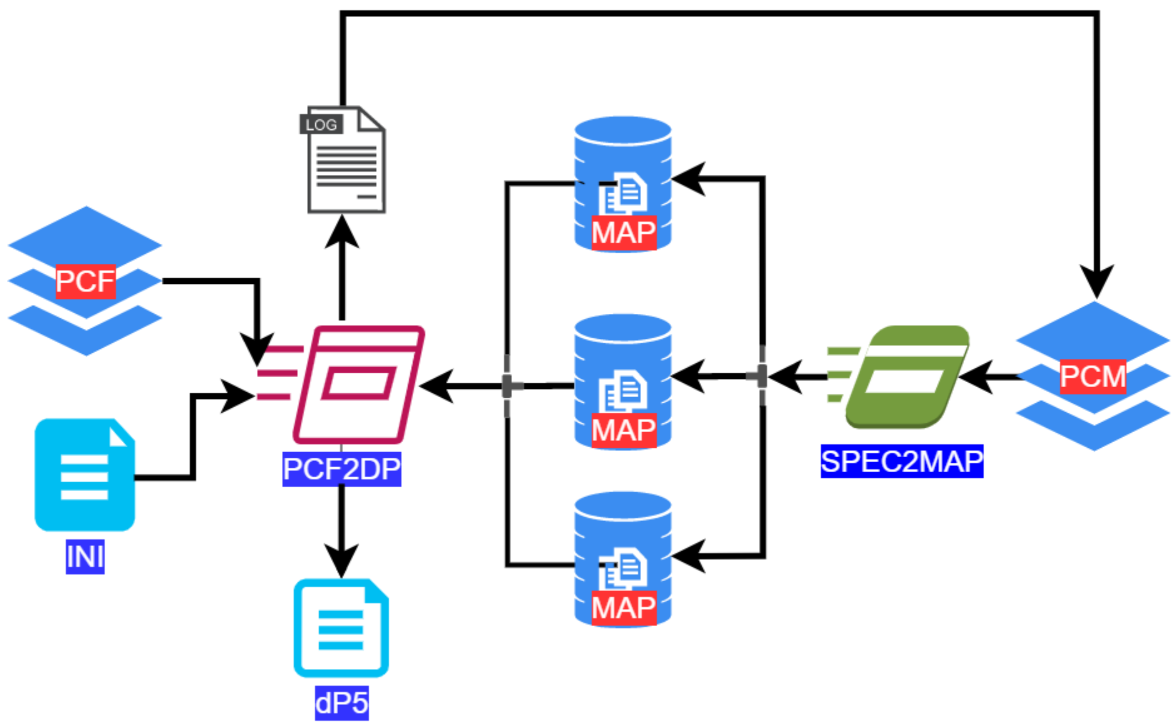


Figure 4. Block diagram of PCF2dP operation

Program settings file (INI file)

The INI file consists of the following sections:

- ✍ [COMPONENT] – description of pipeline component identifiers. The following parameters may be present in this section::
  - «DESC» – identifier of the line containing information that allows you to determine the data on the element, or a reference to the PCF file section "MATERIALS"
  - «MATL» - identifier of the line containing information that allows you to determine the name of the material of the pipeline component
  - «SDES» - identifier of the line containing information that allows you to determine the data on the supports
  - «BTAN» - used to set an attribute that is used in some cases for bent bends with straight sections
- ✍ [MATERIALS] – description of information needed to build a calculation model of the pipeline, from the section of the same name "MATERIALS", which is located at the end of the PCF file. Depending on the method of describing the data in the PCF file, this section may also contain the parameters "DESC" and "MATL", or remain empty.
- ✍ [TOLERANCE] – section containing parameters defining the accuracy of constructing a spatial pipeline model: the R_OFF parameter defines the number of digits after the decimal point in coordinate values for rounding when performing arithmetic operations.
- ✍ [LOG] – section containing the LEVEL parameter defining the level of detail of the log file – the program execution protocol: LEVEL = 0 – the most compact record. Large values are used for debugging.

The semicolon sign (“;”) is used as a comment symbol: all information after this symbol is ignored by the program.

The current version of PCF2dP assumes 3 options for identifying pipeline parts.

Variant 1 assumes that the information required to identify pipeline elements is contained in PCF sections with the components: PIPE, ELBOW, REDUCER, TEE. For example::

```
PIPE
    COMPONENT-IDENTIFIER    1
    END-POINT     46200.000    8599.999    10570.738     300
    END-POINT     46200.000    8801.628    10571.140     300
    FABRICATION-ITEM
    INSULATION-SPEC     UNDEFINED
    PAINTING-SPEC     UNDEFINED
    TRACING-SPEC     UNDEFINED
    PIPING-SPEC     ST_014U025S_SG
    WEIGHT     62.540
    UCI     {0001388C-0000-0000-F332-8811A7611F05}
    ITEM-CODE     {0001388C-0000-0000-F332-8811A7611F05}
    ITEM-DESCRIPTION     CTO/STO 95 113-2013 Tpy6a/Pipe   325x8
    COMPONENT-ATTRIBUTE1     62.54
    COMPONENT-ATTRIBUTE6     20 TY 14-3-190-2004
    COMPONENT-ATTRIBUTE7     8
    COMPONENT-ATTRIBUTE8     8
    COMPONENT-ATTRIBUTE9     0
    COMPONENT-ATTRIBUTE41     70SGA05BR010
    SPOOL-IDENTIFIER     NO SPOOL FOUND
    CUT-PIECE-LENGTH     201.63
```

Then the contents of the INI file can be written as:

```
[COMPONENT]
DESC = ITEM-DESCRIPTION
MATL = COMPONENT-ATTRIBUTE6
SDES = SUPPORT-TYPE
;[MATERIALS]
[TOLERANCE]
R_OFF = 1
[LOG]
LEVEL = 0
```

Variant 2 assumes that the "ITEM-CODE" descriptor contains a reference to the "MATERIALS" section, which provides more detailed information. For example:

```
PIPE
    COMPONENT-IDENTIFIER    3
    END-POINT     989304.000    863135.000    12512.603     50
    END-POINT     989304.000    863051.403    12657.397     50
    ERECTION-ITEM
    INSULATION-SPEC    UNDEFINED
    PAINTING-SPEC    UNDEFINED
    TRACING-SPEC    UNDEFINED
    MISC-SPEC1    10MAV16BR001
    PIPING-SPEC    ST_006N300S_TMM
    WEIGHT    4.020
    UCI    {0001388C-0000-0000-B207-8E883A63C4B9}
    ITEM-CODE    4.020+  TMM-1.00309-12  Труба   57x3+10535konan13
    ITEM-DESCRIPTION    TMM-1.00309-12  Труба   57x3
    COMPONENT-ATTRIBUTE10    .00 0.000
    COMPONENT-ATTRIBUTE4    Operator: Undefined
    COMPONENT-ATTRIBUTE5    0x0-0
    CUT-PIECE-LENGTH    167.19

MATERIALS
ITEM-CODE    0.600+  TMM-3.11073  Колено 90°-57x3-PN 25+10535konan13
    DESCRIPTION    TMM-3.11073  Колено 90°-57x3-PN 25
    MATERIAL-USER1    0.600
    MATERIAL-USER0    08X18H10T ГОСТ 9941-81
ITEM-CODE    4.020+  TMM-1.00309-12  Труба   57x3+10535konan13
    DESCRIPTION    TMM-1.00309-12  Труба   57x3
    MATERIAL-USER1    4.020
    MATERIAL-USER0    08X18H10T ГОСТ 9941-81
```

In this case, the contents of the ini file can be written as:

```
[COMPONENT]
;desc =
matl = ITEM-CODE
[MATERIALS]
desc = DESCRIPTION
matl = MATERIAL-USER0
[TOLERANCE]
R_OFF = 1
[LOG]
LEVEL = 0
```

Variant 3 is used when the reference to the "MATERIALS" section is contained in the "MATERIAL-IDENTIFIER" descriptor:

```
PIPE
    COMPONENT-IDENTIFIER    1
    END-POINT     1148.745    2846.978    -8378.010     50
    END-POINT     1148.745    2846.978    -8443.000     50
    ERECTION-ITEM
    INSULATION-SPEC    UNDEFINED
    PAINTING-SPEC    UNDEFINED
    TRACING-SPEC    UNDEFINED
    PIPING-SPEC    AH_140N335S_Z
    UCI    {0001388C-0000-0000-B418-779744658E04}
    MATERIAL-IDENTIFIER 1
    COMPONENT-ATTRIBUTE1    7.02
    COMPONENT-ATTRIBUTE6    08Х18Н10Т ТУ 14-3Р-197-2001
    COMPONENT-ATTRIBUTE7    5.5
    COMPONENT-ATTRIBUTE8    5.5
    COMPONENT-ATTRIBUTE9    0
    COMPONENT-ATTRIBUTE41    30JNB98BR001
    CUT-PIECE-LENGTH    64.99
..............................

MATERIALS
MATERIAL-IDENTIFIER    1
    ITEM-CODE {0001388C-0000-0000-B418-779744658E04}
    DESCRIPTION                                            ОСТ 24.125.01-89 Труба   57x5.5
    MATERIAL-USER31    7.02
    MATERIAL-USER32    08Х18Н10Т ТУ 14-3Р-197-2001
    MATERIAL-USER33    0.46
```

In this case, the contents of the ini file can be written as:

```
[COMPONENT]
matl = MATERIAL-IDENTIFIER
sdes = SUPPORT-TYPE
[MATERIALS]
desc = DESCRIPTION
matl = MATERIAL-USER32
[TOLERANCE]
R_OFF = 1
[LOG]
LEVEL = 0
```

The template for the INI file is located in the \dPIPE folder 5.28\Utils\PCF2DP\pcf.ini. This template is written for Variant 1, and variants 2 and 3 are commented out in it.

If the User starts editing the INI file, the program saves a copy of this file in the folder with the calculation model and assigns it the same name as the dP5 file. During subsequent work sessions, the link to the last INI file is saved.

BTAN parameter

In some cases, PCF files use a special attribute with the value True or False, which indicates whether the bend has straight sections.

```
ELBOW
    COMPONENT-IDENTIFIER    3
    END-POINT    14635.993    -84014.007    15139.003     50    BW
    END-POINT    14557.176    -83724.546    15439.003     50    BW
    CENTRE-POINT    14635.993    -84014.007    15439.003
    SKEY    ELBW
    ITEM-CODE    4.608+25 OCT 24.125.32-89 Отвод 90°-57x4-150x240-861-R
300+10165konan13
    ITEM-DESCRIPTION    25 OCT/OST 24.125.32-89 Отвод/Bend 90°-57x4-150x240-861-R
300
    COMPONENT-ATTRIBUTE1    4.61
    COMPONENT-ATTRIBUTE10    SH1: 4.0mm SH2: 4.0mm SH3:
```

```
COMPONENT-ATTRIBUTE2    CP1: 0.057 CP2: 0.057 CP3:
COMPONENT-ATTRIBUTE3    300
COMPONENT-ATTRIBUTE4    90°
COMPONENT-ATTRIBUTE5    150 240
COMPONENT-ATTRIBUTE6    20 ТУ 14-3Р-55-2001
COMPONENT-ATTRIBUTE7    4
COMPONENT-ATTRIBUTE8    4
COMPONENT-ATTRIBUTE9    4
COMPONENT-ATTRIBUTE40    ОСТ 24.125.32-89
COMPONENT-ATTRIBUTE41    10LBJ57BR001
COMPONENT-ATTRIBUTE42    150
COMPONENT-ATTRIBUTE43    240
COMPONENT-ATTRIBUTE44    True
UCI    {00013885-0000-0000-D82D-07073F5C3D50}
INSULATION ON
INSULATION-SPEC    UNDEFINED
PAINTING-SPEC    UNDEFINED
TRACING-SPEC    UNDEFINED
MISC-SPEC1    10LBJ57BR001
PIPING-SPEC    AH_086U300S_Z
ERECTION-ITEM
WEIGHT    4.608
```

In this case, the BTAN parameter should be set in the [COMPONENT] section of the ini file with the name of this attribute. Otherwise, the sections of the straight sections adjacent to the branch will remain undefined.

```
[COMPONENT]

desc = ITEM-DESCRIPTION
matl = COMPONENT-ATTRIBUTE6

btan = COMPONENT-ATTRIBUTE44

sdes = SUPPORT-TYPE
```

Program operation log (LOG file)

When the protocol is written compactly (LEVEL = 0), the following types of messages are included in the LOG file:

1) Information about sections in PCF that are ignored by the program:

```
Parsing "D:\_DEBUG\PCF2DP\20240512\TYPE_03\30JNB98BR001-1.pcf" (001)

00004: UNITS-BOLT-LENGTH -> ignored
00005: UNITS-BOLT-DIA -> ignored
00007: PIPELINE-REFERENCE -> ignored
00159: END-CONNECTION-EQUIPMENT -> ignored
00162: ELBOW -> CONTINUATION component ignored
00190: END-CONNECTION-PIPELINE -> ignored
00212: MATERIAL-IDENTIFIER -> ignored
00218: MATERIAL-IDENTIFIER -> ignored
```

2) Information about supports whose coordinates do not fall on the centerline of the pipe:

```
*** Linking Supports ***

00110_002:SUPP 01HG  DN:    50  CP:   -103  Unlinked: dmin = 1.15E+02 mm
00204_002:SUPP 01HG  DN:    50  CP:   -104  Unlinked: dmin = 1.15E+02 mm
```

Such supports are linked to the nearest node of the calculation model and a short RIGID element with the section "XSUP" is built from this node: thus, these places are easy to identify and make the necessary changes.

3) Undefined element property descriptors:

```
*** Undefined Descriptors ***

ОСТ 24.125.01-89 Труба 57x5.5:08X18H10T ТУ 14-3Р-197-2001
СТО 79814898 109-2012 Труба 57x3:08X18H10T ТУ 14-3Р-197-2001

01 СТО 79814898 111-2009 Колено В 90°-57x3-PN 25:08X18H10T ТУ 14-3Р-197-2001
05 ОСТ 24.125.05-89 Отвод 90°-57x5.5-?x?-?-R 200:08X18H10T ТУ 14-3Р-197-2001

02 СТО 79814898 122-2009 Штуцер В 10 PN25:08X18H10T ГОСТ 5949-2018
```

These lines are generated by the program in accordance with the pointers specified in the configuration INI file. For example, the MATL = ITEM-CODE pointer in the [COMPONENT] section refers to the DESC = DESCRIPTION and MATL = MATERIAL-USER0 pointers in the [MATERIALS] section. Accordingly, the element property descriptors are generated from the "DESCRIPTION" and "MATERIAL-USER0" lines:

```
MATERIALS
MATERIAL-IDENTIFIER    1
    ITEM-CODE {0001388C-0000-0000-A207-8C522E658604}
    DESCRIPTION         ОСТ 24.125.01-89 Труба   57x5.5
    MATERIAL-USER31     7.02
    MATERIAL-USER32     08X18H10T ТУ 14-3Р-197-2001
    MATERIAL-USER33     11.37
```

```
ОСТ 24.125.01-89 Труба 57x5.5:08X18H10T ТУ 14-3Р-197-2001
```

These lines are further used to prepare specifications (PCM files), from which the program generates indexed binary MAP files.

Rules for forming PCM files

1) The exclamation mark "!" is interpreted by the program as a comment: all information after it is ignored.
2) ***Descriptor lines from the LOG file are written in the first position***. If a descriptor line contains a colon ":", then the description of the part properties is followed by a description of the material used. For example:

```
СТО 79814898 109-2012 Труба 18x2.5:08X18H10T ГОСТ 9941-81
ТММ-3.42134 Труба 18x2.5:08X18H10T ГОСТ 9941-81
```

3) The descriptor lines must be followed by the corresponding command for the dPIPE calculation model. Several descriptor lines written one after another correspond to one dPIPE command, which is written immediately after the descriptors, starting from the second position:

✍ Section characteristics: PIPE command::

```
01.PA1.0.0.TM.TT.NSN083-01 Труба S-NUC-B-13.5x2:08X18H10T ГОСТ 5632-2014
01.PA1.0.0.TM.TT.NSN083-01 Труба S-NUC-C-13.5x2:08X18H10T ГОСТ 5632-2014
01.PA1.0.0.TM.TT.NSN083-01 Труба/Pipe S-NUC-B-13.5x2:08X18H10T ГОСТ 5632-2014
01.PA1.0.0.TM.TT.NSN083-01 Труба/Pipe S-NUC-C-13.5x2:08X18H10T ГОСТ 5632-2014
01.PA1.0.0.TM.TT.NSN083-01 Труба/Pipe S-T-13.5x2:08X18H10T ГОСТ 5632-2014
'13.5x2'
  PIPE '13.5x2' OD=13.5 T=2 W=0.0057 C=-12.5 MAT='08X18H10T'
```

Among the descriptor lines for the PIPE command, there must be a label for the section being described. In this example, it is ’13.5x2’

✍ Bend characteristics (&BEND command):

```
03-01 01.PA1.0.0.TM.TT.NSN083-01 Отвод гнутый/Curved bend NUC-B-45°-13.5x2-?x?-?-R100:08X18H10T ГОСТ 5632-2014
03-01 01.PA1.0.0.TM.TT.NSN083-01 Отвод гнутый/Curved bend NUC-B-90°-13.5x2-?x?-?-R100:08X18H10T ГОСТ 5632-2014
03-01 01.PA1.0.0.TM.TT.NSN083-01 Отвод гнутый/Curved bend NUC-C-30°-13.5x2-?x?-?-R100:08X18H10T ГОСТ 5632-2014
03-01 01.PA1.0.0.TM.TT.NSN083-01 Отвод гнутый/Curved bend NUC-C-45°-13.5x2-?x?-?-R100:08X18H10T ГОСТ 5632-2014
03-01 01.PA1.0.0.TM.TT.NSN083-01 Отвод гнутый/Curved bend NUC-C-60°-13.5x2-?x?-?-R100:08X18H10T ГОСТ 5632-2014
03-01 01.PA1.0.0.TM.TT.NSN083-01 Отвод гнутый/Curved bend NUC-C-90°-13.5x2-?x?-?-R100:08X18H10T ГОСТ 5632-2014
| '13.5x2' &BEND 'R100' R=100 OVAL=2.6 SMIN=1.4 CROS='13.5x2'
```

The standard dPIPE command “&BEND” must be preceded by the label of the connected pipe, and the CROS parameter (reference to the cross-section of the bend) is also mandatory, even if the cross-section of the bend does not differ from the cross-section of the connected pipe.

✍ Transition characteristics (&REDU command):

```
01-02 01.PA1.0.0.TM.TT.NSN083-03 Переход/Reducer S-NUC-B-17.2x3.2-13.5x2:08X18H10T ГОСТ 5632-2014
  '17.2x3.2' &REDU '13.5x2'
```

The &REDU command is located between the labels of the pipes being connected, in this example ’17.2x3.2’ and ’13.5x2’

✍ Tee and Branch Connection characteristics (&TEE command):

```
01-01 01.PA1.0.0.TM.TT.NSN083-02 Тройник равнопроходный/Equal tee S-NUC-C-13.5x2:08X18H10T ГОСТ 5632-2014
  '13.5x2' & TEE 'FRG' bid='13.5x2' Dr= 15.2 Tr= 2.6 Db= 15.2 Tb= 2.6 L= 60 H= 30 W= 0.6 Tw= 2.6 Cr= 0.0 mat='08X18H10T'
```

The &TEE command must be preceded by the label of the pipe adjacent to the tee body, in this example ’17.2x3.2’ and ’13.5x2’

Working with dP5 files obtained as a result of conversion from PCF.

Следует It should be noted that the dp5 files obtained from PCF are "semi-finished products", and the degree of necessary revision of the corresponding calculation models significantly depends on the content of the MAP files. In the absence of MAP files, the program generates the pipeline geometry, places supports and marks the sections of the pipeline parts with nominal diameters DNXX, see Figure 5. Elements that the program was unable to "recognize" are modeled by rigid links (RIGID element). If after conversion there are unconnected branches in the calculation model, the program marks them with the names B001, B002, etc., see Figure 6.

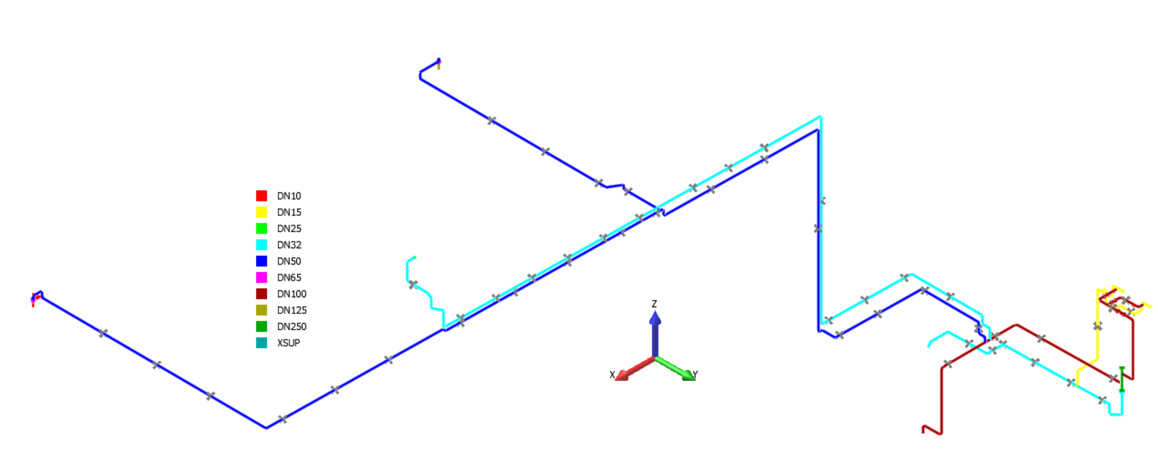


Figure 5. Example of generating a dP5 file without using MAP files. Identification of sections

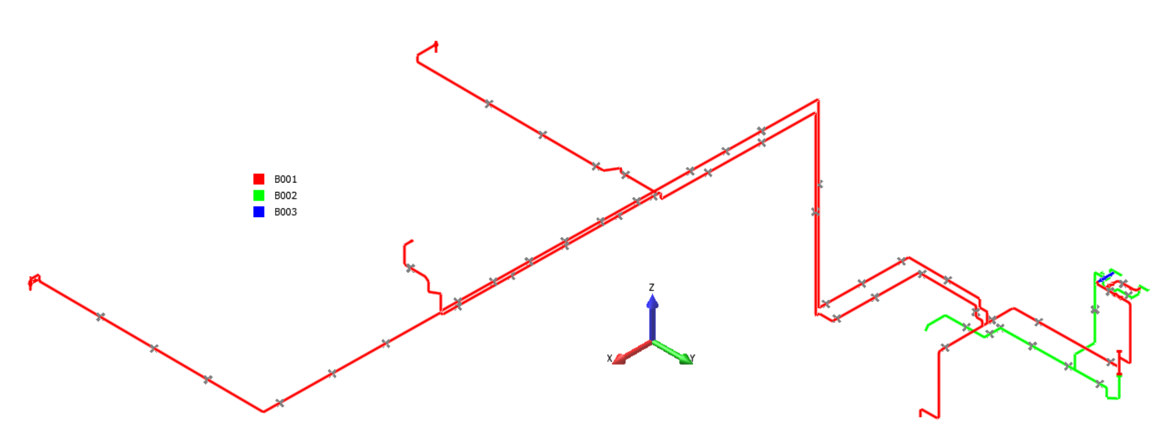


Figure 6. Example of generating a dP5 file. Identification of unconnected branches

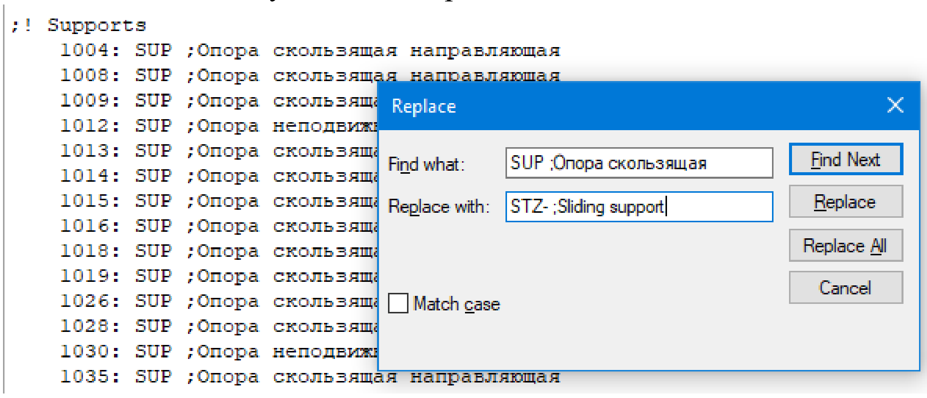
Supports and tees are written to the end of the dP5 file. If the MAP file did not contain a specification for any tee connections, these tees are added to the source data commented out:

```
;    172:TEE
;    189:TEE
;    194:TEE

     42:TEE 'UFT'
     90:TEE 'UFT'
    106:TEE 'FRG'
    124:TEE 'FRG'
    135:TEE 'FRG'
    156:TEE 'UFT'
    158:TEE 'UFT'
;    163:TEE
    168:TEE 'UFT'
;    180:TEE
```

All supports are transferred to dP5 as 6-component supports, but the support type from the specification is displayed in the comment to them (in accordance with the SDES parameter from the INI file):

```
1003:SUP ;Подвеска жесткая
1005:SUP ;Опора направляющая
1007:SUP ;Подвеска жесткая
1008:SUP ;Опора направляющая
1012:SUP ;Опора неподвижная
1014:SUP ;Опора направляющая
1015:SUP ;Опора скользящая
1022:SUP ;Опора скользящая
1023:SUP ;Опора неподвижная
1024:SUP ;Опора скользящая
1028:SUP ;Опора скользящая
1029:SUP ;Опора скользящая
1030:SUP ;Опора направляющая
1031:SUP ;Опора направляющая
1032:SUP ;Опора скользящая
1033:SUP ;Опора скользящая
1036:SUP ;Опора неподвижная
1037:SUP ;Опора скользящая
```

When working with dP5 files obtained from PCF, it is recommended to set the option "Additional data at the end of the file" in the "Service" menu of the dPIPE window. Then, when switching from graphic to text mode, it will be convenient to change the support type in accordance with the comments by contextual replacement:



29 Sample of analysis

As an example, the main steam line of the WWER-440 unit has been selected extending from the steam generator header to the containment penetration within the containment area.

Piping Data

Figure 1 shows the isometric diagram of this piping system and its main dimensions. The piping system is manufactured from steel of grade St 20, working temperature of the content: 271° C, ambient temperature 20° C, internal pressure: 4.5 MPa.

In the hot state, the steam generator headers have the following thermal displacements:

Table 1. Temperature displacements of the steam generator header in the working state.

Nozzle No.	Temperature displacements		
	X	Y	Z
P1	68	54	6
P2	63	57	6
P3	56	60	6
P4	49	64	6
P5	44	67	6

Locations of spring hangers are shown in the figure. The length of hangers rods are given in the table below:

Table 2. Rod Length and number of springs for spring hangers

Support No.	Number of springs	Rod Length, mm
N1	2	2050
N2	1	1010
N3	2	2050
N4	1	830
N5	1	700
N6	1	670
N7	1	660

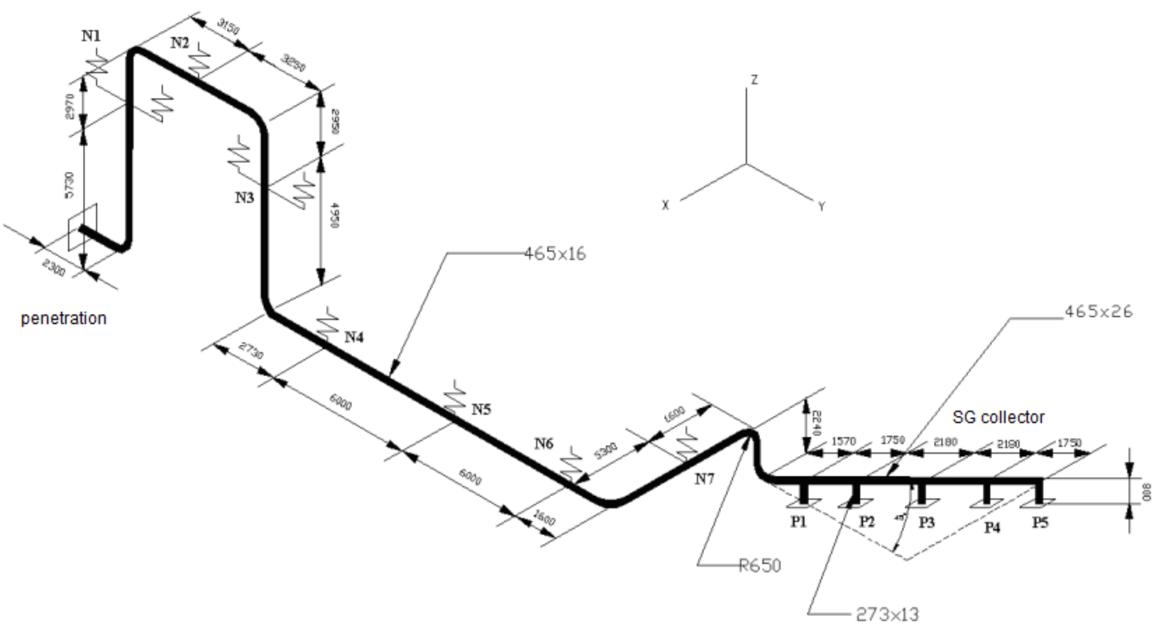


Figure 1. Isometric diagram of the sample piping system

Table 3. Cross-section characteristics (according to OST 24/125.30-89)

DN	Dn, mm	s, mm	c, mm	w, N/mm	R, mm	Smin, mm	a, %
450	465	16	0.8	1.88 (value from OST)	650	12.5	7
450	465	26	1.3	to be determined by the program	-	-	-
250	273	13	0.65	to be determined by the program	-	-	-

Seismic Load

Seismic load is specified in the form of two groups of floor response spectrums: group SG - for attachment of the piping to the steam generator and group RB - for all the rest of steam line supports (penetration from the reactor compartment and hangers). The response spectrum values are given in Table 4. For supports attributed to group SG, the seismic displacements are specified: X - 33mm, Y - 34mm, Z - 2mm. For group RB, the seismic displacements are taken to be equal to zero. The OBE earthquake is taken equal to a half of SSE.

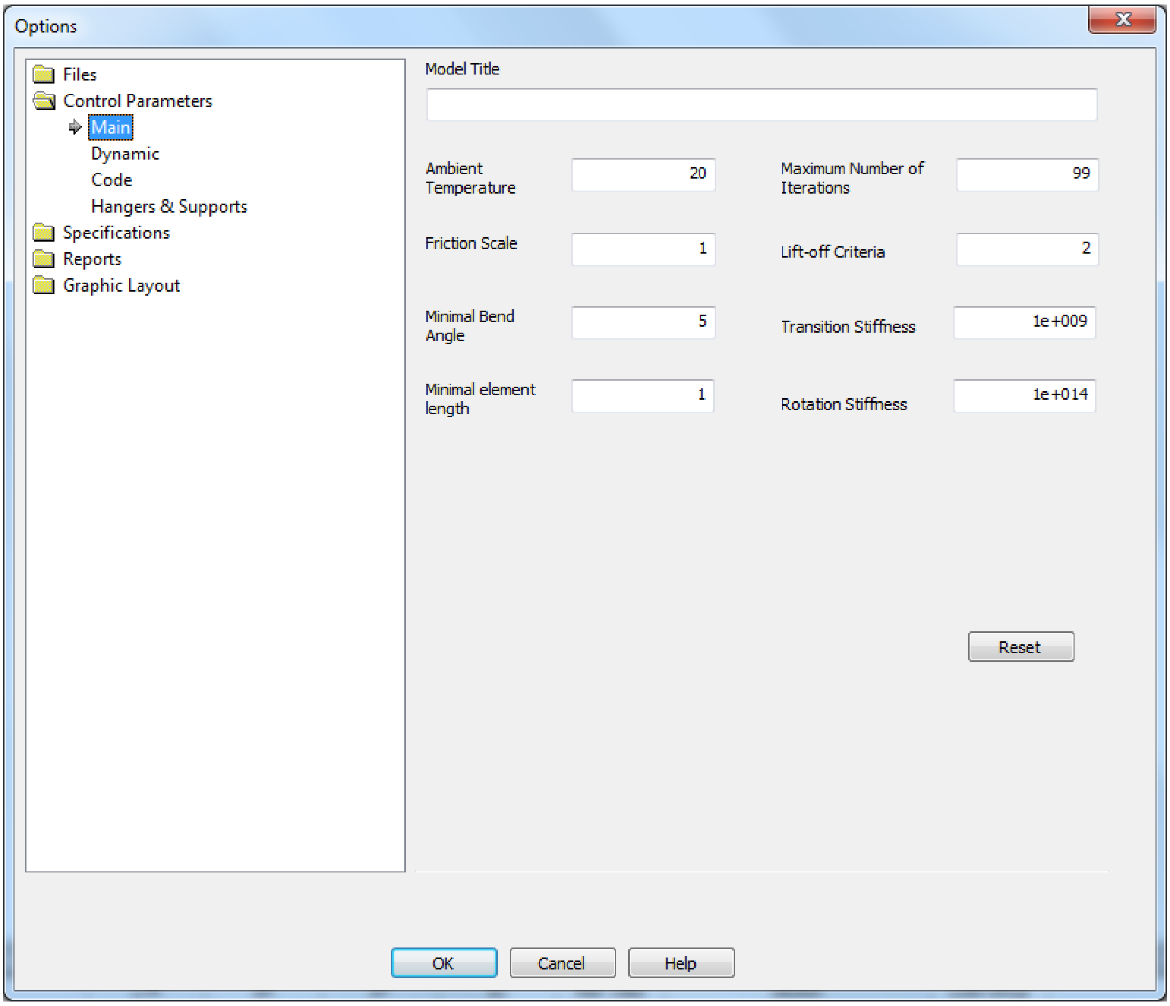
Table 4. Floor response spectrums of seismic load for the SSE level

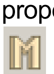
	X-Direction		Y-Direction		Z-Direction	
	Frequency, Hz	Acceleration, g	Frequency, Hz	Acceleration, g	Frequency, Hz	Acceleration, g
group SG	0.20	0.06	0.20	0.06	0.20	0.06
	2.00	1.00	2.56	1.37	2.50	0.88
	3.08	1.30	4.17	1.38	4.00	1.11
	4.60	2.50	6.22	2.57	8.00	1.10
	6.54	2.50	8.27	2.58	10.00	1.10
	11.67	1.00	11.67	1.21	12.00	1.13
	17.00	1.00	14.36	1.21	13.00	1.21
	27.00	0.68	28.72	0.61	28.00	0.46
	40.00	0.69	39.68	0.61	40.00	0.47
group RB	0.20	0.06	0.20	0.06	0.20	0.06
	2.00	1.40	2.00	1.40	2.50	0.92
	4.00	2.74	3.00	3.70	7.20	1.65
	4.20	4.00	5.40	3.70	10.50	2.25
	7.00	4.00	6.90	3.20	14.50	2.10
	12.00	1.60	9.00	3.20	17.80	1.80
	16.00	1.60	16.00	1.50	20.00	1.50
	23.00	1.00	23.00	1.00	28.00	0.78
	40.00	1.00	40.00	1.00	40.00	0.78

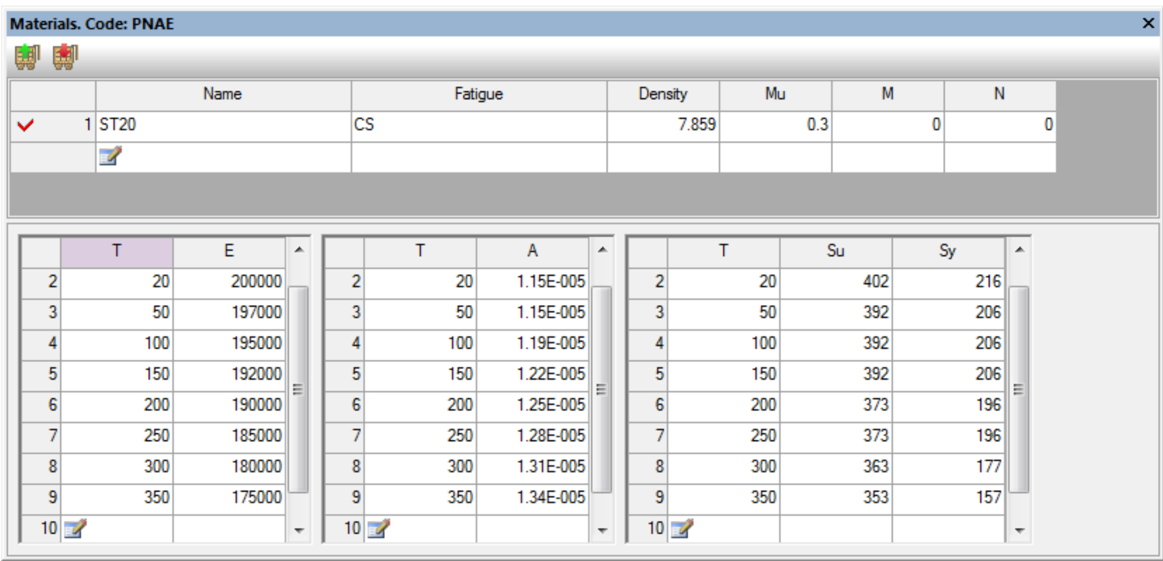
Input Data and Run of analysis

1. Create a working directory on the hard disk (for example, E:\PROJECTS\DPIPE_WORK\SAMPLE). Run the DDE program and save a "blank" model in the working directory with a suitable name (for example, "RA_SAMPLE.DP5").

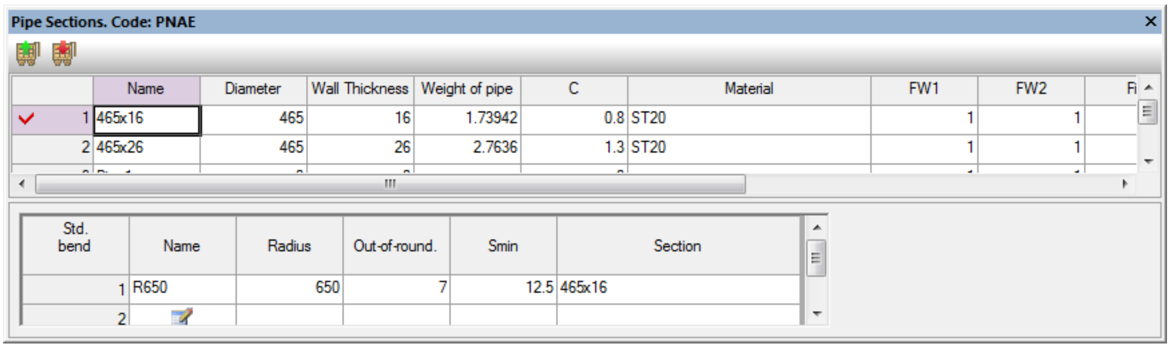
2. Give a name to the analysis model, which will be displayed in printpouts in the form of title ("Service\Options\Reference parameters" menu):



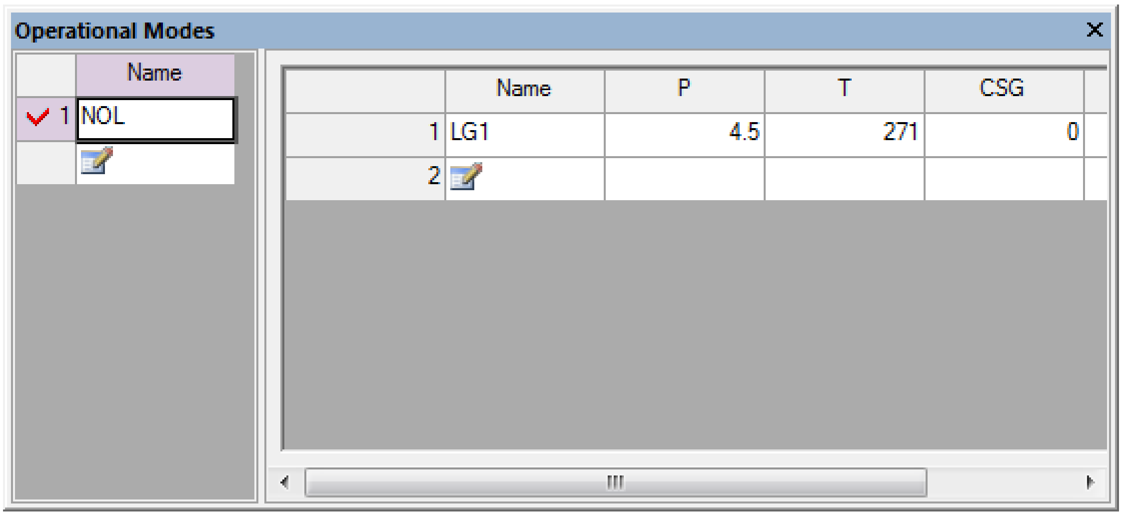
3. Determine the type of material for the piping system and specify its physical and mechanical properties (select from the database(), or type manually). "Main data\Materials" menu or  button:



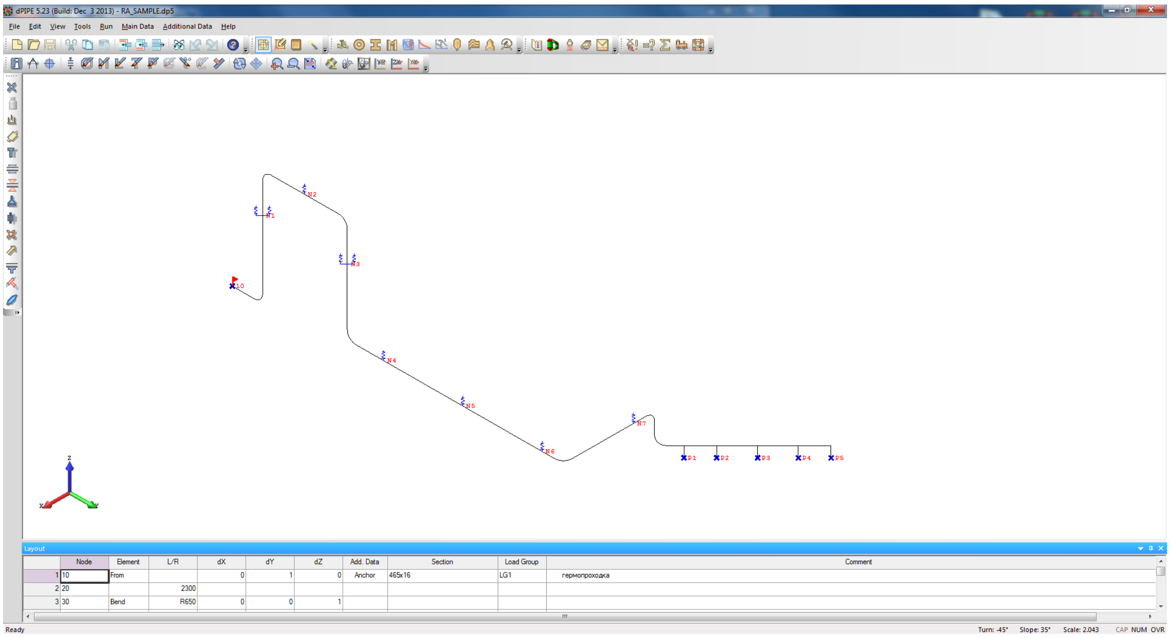
4. Determine the piping system cross-sections and the nomenclature of bends being used. "Main data\Pipe cross-sections" menu or button:



5. Specify the operating modes of the piping system ("Main data\Operating mode" menu or button):

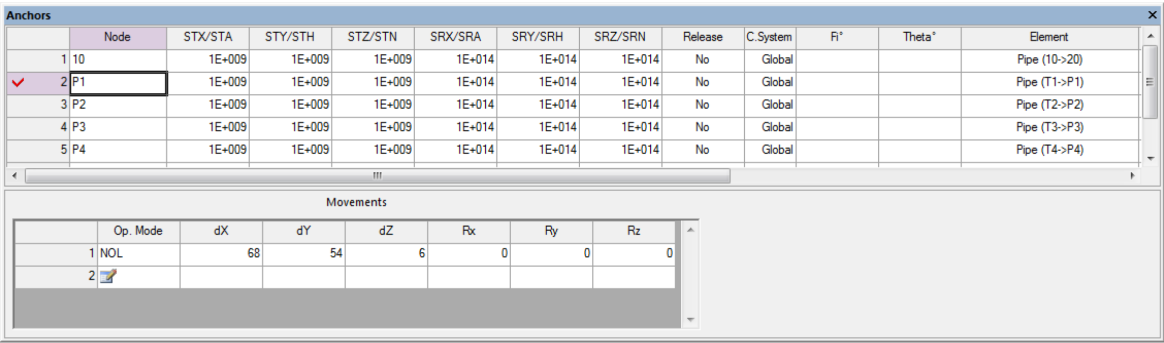


6. Specify the geometry of the piping system, location, types and characteristics of supports:



In the course of input of the geometry, it is recommended to periodically save the input data file on the hard disk!!!.

7. Specify the temperature displacements of connecting equipment (anchors simulating the connection to SG). "Main data\Additional data\Anchors" menu):



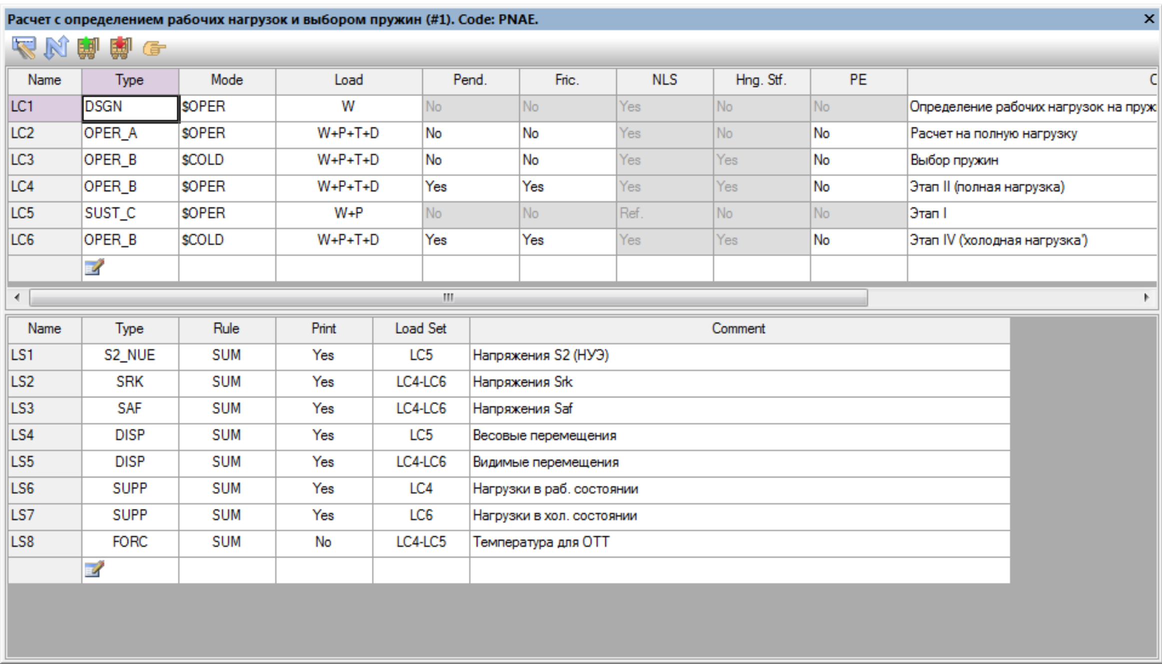
Anchors

	Node	STX/STA	STY/STH	STZ/STN	SRX/SRA	SRY/SRH	SRZ/SRN	Release	C.System	Fi°	Theta°	Element
1	10	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (10->20)
2	P1	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T1->P1)
3	P2	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T2->P2)
4	P3	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T3->P3)
5	P4	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T4->P4)

Movements

	Op. Mode	dX	dY	dZ	Rx	Ry	Rz
1	NOL	68	54	6	0	0	0
2							

8. Determined the specification for analysis and post-processing of the results for selecting springs of spring hangers/supports ("Service\Options\Tasks" menu or button):

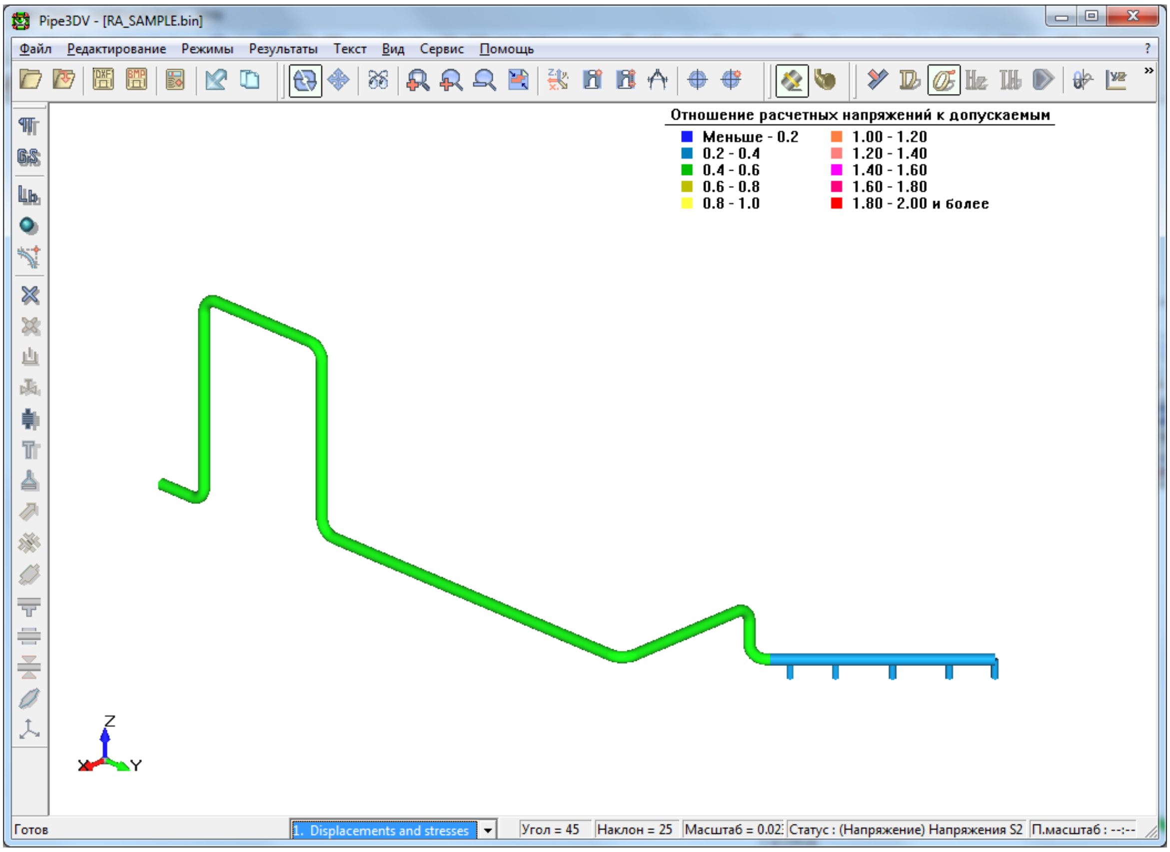


Расчет с определением рабочих нагрузок и выбором пружин (#1). Code: PNAE.

Name	Type	Mode	Load	Pend.	Fric.	NLS	Hng. Stf.	PE	C
LC1	DSGN	$OPER	W	No	No	Yes	No	No	Определение рабочих нагрузок на пруж
LC2	OPER_A	$OPER	W+P+T+D	No	No	Yes	No	No	Расчет на полную нагрузку
LC3	OPER_B	$COLD	W+P+T+D	No	No	Yes	Yes	No	Выбор пружин
LC4	OPER_B	$OPER	W+P+T+D	Yes	Yes	Yes	Yes	No	Этап II (полная нагрузка)
LC5	SUST_C	$OPER	W+P	No	No	Ref.	No	No	Этап I
LC6	OPER_B	$COLD	W+P+T+D	Yes	Yes	Yes	Yes	No	Этап IV (холодная нагрузка')

Name	Type	Rule	Print	Load Set	Comment
LS1	S2_NUE	SUM	Yes	LC5	Напряжения S2 (НУЭ)
LS2	SRK	SUM	Yes	LC4-LC6	Напряжения Srk
LS3	SAF	SUM	Yes	LC4-LC6	Напряжения Saf
LS4	DISP	SUM	Yes	LC5	Весовые перемещения
LS5	DISP	SUM	Yes	LC4-LC6	Видимые перемещения
LS6	SUPP	SUM	Yes	LC4	Нагрузки в раб. состоянии
LS7	SUPP	SUM	Yes	LC6	Нагрузки в хол. состоянии
LS8	FORC	SUM	No	LC4-LC5	Температура для ОТТ

9. Execute the analysis ("Batch mode", button) and view the results of analysis (PIPE3DV, button):



View the results on the selection of springs of spring hangers/supports ("Loads on supports",  button) :

```
>>>  Characteristics of spring supports and hangers(for
installation0). Standard OST 108.764.01-80
================================================================
=====================================
Support    NC  Chain        Spring height         Total load on
support      visible displacements ALPHA
(node)No.    struct.    H_fr. H_work. H_cold. H_1s     P_work.
P_cold     P_seism.     DX      DY     DZ
================================================================
=====================================
N1          2   1*19     414   338    319    318     21.40
26.77               4     -1     19    0.1
----------------------------------------------------------------
-------------------------------------
N2          1   1*06+    201   165    157    154      8.31
10.37              15      6     26    0.9
                1*18     369   298    280    275
----------------------------------------------------------------
-------------------------------------
N3          2   1*06     201   160    148    133     19.21
24.61              26      8     12    0.8
----------------------------------------------------------------
-------------------------------------
N4          1   1*06     201   154    160    145     10.88
9.56              40      5     -6   2.8
```

```
-----------------------------------------------------------------
-------------------------------------
N5          1   1*07     226   181    183    169     12.69
12.10              67     24    -2   5.9
-----------------------------------------------------------------
-------------------------------------
N6          1   1*07     226   178    173    162     13.55
14.86              91     44     5   8.6
-----------------------------------------------------------------
-------------------------------------
N7          1   1*20     399   328    314    311     13.38
16.01              78     51    14   8.1
=================================================================
=====================================
Note:
NC           - number of chains
H_fr.        - spring height in the free state, mm
H_work.      - spring height in the working state, mm
H_cold       - spring height in the cold state, mm
H_1s         - spring height with single-stage preload, mm
P_work.      - load on support in the working state, kN
P_cold       - load on support in the cold state (without
working fluid), kN
P_seism.     - seismic load on the support, kN
DX, DY, DZ   - visible movements between the cold and working
states, mm
ALPHA        - angle of deviation of the hanger tie rod from the
vertical, degrees


>>>  Summarized table of loads on spring supports and hangers
(all stages of analysis)
=================================================================
====================
Support  Spring     P_des.  P_work.   FS     var     DX     DY
    DZ  ALPHA  Analysis
(node)No.
               calculation
=================================================================
====================
N1       07/Z2       21.45   21.40   1.8     0        2     -7
    20   0.2  LS06
                             26.77   1.5    25       -1     -6
     1   0.2  LS07
-----------------------------------------------------------------
--------------------
N2       06/Z3        8.35    8.31   2.0     0       12     -5
    34   0.7  LS06
                             10.37   1.6    24       -3    -10
     8   0.6  LS07
-----------------------------------------------------------------
--------------------
N3       06/Z1       19.33   19.21   1.7     1       24      3
    27   0.7  LS06
                             24.61   1.3    27       -2     -6
    15   0.2  LS07
```

```
--------------------------------------------------------------
--------------------
N4        06/Z1       10.85   10.88   1.5     0       40      5
    10   2.8  LS06
                               9.56   1.7    12        1      0
    15   0.1  LS07
--------------------------------------------------------------
--------------------
N5        07/Z1       12.65   12.69   1.5     0       67     24
    12   5.8  LS06
                              12.10   1.6     4       -1      0
    14   0.1  LS07
--------------------------------------------------------------
--------------------
N6        07/Z1       13.74   13.55   1.5     1       89     44
    16   8.5  LS06
                              14.86   1.3     8       -2      0
    11   0.2  LS07
--------------------------------------------------------------
--------------------
N7        08/Z2       13.46   13.38   2.0     1       76     52
    16   8.0  LS06
                              16.01   1.6    19       -2      0
     2   0.2  LS07
==============================================================
====================
Note:
P_des.         - design load, kN
P_work.        - load for the design mode,kN
FS             - load safety margin
var            - variability, %
DX, DY, DZ     - movements for the design state
ALPHA          - angle of deviation of the hanger tie rod from
the vertical, degrees

LS06              - Loads in the working state
LS07              - Loads in the cold state



>>>  Summarized table of loads on anchors and supports
==============================================================
====================
 Support  Cross-section  FX       FY       FZ         MX
MY       MZ   Analysis
 (node)No.)             -------(forces, kN)--------- -----
(moments, kN*m)--  calculation
==============================================================
====================
10         465x16        1.2     -6.1     -1.9      34.2
2.7    -18.4   LS06
                        -0.1      0.7      7.1      29.5     -
3.2      1.4   LS07
--------------------------------------------------------------
--------------------
P1         273x13      -29.9     28.3      6.5     -10.1    -
15.0     12.1   LS06
```

```
                         2.2     -2.4      3.6        2.3     -
1.9     -0.2   LS07
---------------------------------------------------------------
--------------------
P2         273x13      -57.6     14.5    -11.8       -4.1    -
25.4     16.0   LS06
                        -0.8      0.7    -14.3        1.1     -
1.5     -0.2   LS07
---------------------------------------------------------------
--------------------
P3         273x13      -22.5     54.9    -19.9      -23.1     -
9.4     17.3   LS06
                        -0.3      0.4     -7.1        0.6     -
0.9     -0.1   LS07
---------------------------------------------------------------
--------------------
P4         273x13       56.8    -49.2    -30.5       19.9
24.2     13.9   LS06
                        -0.4      0.4     -7.0        0.3     -
0.6      0.0   LS07
---------------------------------------------------------------
--------------------
P5         273x13       46.6    -44.6     21.8       18.5
20.5     12.0   LS06
                        -0.4      0.5     -3.1        0.2     -
0.5      0.0   LS07
===============================================================
====================
Note:

LS06              - Loads in the working state
LS07              - Loads in the cold state



>>>  Node:                   10
     Type of support:        anchor

Forces:
---------------------------------------------------------------
---------------------
global:                     Fx       Fy       Fz          Mx
   My      Mz
                                   (newtons)
(newton*meter)
---------------------------------------------------------------
---------------------
 1 Loads in the working state     1223    -6142    -1883
34228     2718   -18362
 2 Loads in the cold state        -109      742     7069
29454    -3166     1430
---------------------------------------------------------------
---------------------

Movements:
---------------------------------------------------------------
---------------------
```

```
global:                          Dx       Dy       Dz
Rx       Ry      Rz
                                     (millimeters)
      (degrees)
---------------------------------------------------------------
 ---------------------
 1 Loads in the working state      0        0        0
0.00      0.00     0.00
 2 Loads in the cold state         0        0        0
0.00      0.00     0.00
---------------------------------------------------------------
---------------------



>>>  Node:                   P1
     Type of support:        anchor

Forces:
---------------------------------------------------------------
---------------------
global:                          Fx       Fy       Fz
Mx       My      Mz
                                   (newtons)
 (newton*meter)
---------------------------------------------------------------
---------------------
 1 Loads in the working state   -29938    28299     6541     -
10140   -15000    12149
 2 Loads in the cold state        2243    -2406     3621
2332    -1887     -248
---------------------------------------------------------------
---------------------

Movements:
---------------------------------------------------------------
---------------------
global:                          Dx       Dy       Dz
Rx       Ry      Rz
                                    (millimeters)
     (degrees)
---------------------------------------------------------------
---------------------
 1 Loads in the working state      0        0        0
0.00      0.00     0.00
 2 Loads in the cold state         0        0        0
0.00      0.00     0.00
---------------------------------------------------------------
---------------------



>>>  Node:                   P2
     Type of support:        anchor

Forces:
```

```
---------------------------------------------------------------
---------------------
global:                          Fx        Fy        Fz
Mx        My       Mz
                                               (newtons)
    (newton*meter)
---------------------------------------------------------------
---------------------
 1 Loads in the working state   -57570    14466   -11823      -
4060   -25412    15968
 2 Loads in the cold state        -818      697   -14303
1059    -1525     -155
---------------------------------------------------------------
---------------------

Movements:
---------------------------------------------------------------
---------------------
global:                         Dx        Dy        Dz          Rx
      Ry       Rz
                                            (millimeters)
    (degrees)
---------------------------------------------------------------
-------------------
 1 Loads in the working state   0        0        0
0.00      0.00     0.00
 2 Loads in the cold state      0        0        0
0.00      0.00     0.00
---------------------------------------------------------------
---------------------



>>>  Node:                    P3
     Type of support:         anchor

Forces:
---------------------------------------------------------------
---------------------
global:                          Fx        Fy        Fz
Mx        My       Mz
                                               (newtons)
   (newton*meter)
---------------------------------------------------------------
---------------------
 1 Loads in the working state   -22503    54858   -19866     -
23122    -9359    17336
 2 Loads in the cold state        -344      370    -7121
 563     -863      -77
---------------------------------------------------------------
---------------------

Movements:
---------------------------------------------------------------
---------------------
global:                        Dx        Dy        Dz           Rx
      Ry       Rz
```

```
                                  (millimeters)
   (degrees)
---------------------------------------------------------------
---------------------
 1 Loads in the working state   0        0        0
0.00      0.00     0.00
 2 Loads in the cold state      0        0        0
0.00      0.00     0.00
---------------------------------------------------------------
---------------------



>>>  Node:                   P4
     Type of support:        anchor

Forces:
---------------------------------------------------------------
---------------------
global:                          Fx       Fy       Fz
 Mx       My      Mz
                                       (newtons)
     (newton*meter)
---------------------------------------------------------------
---------------------
 1 Loads in the working state    56786   -49216   -30531
19880    24216    13851
 2 Loads in the cold state        -384      354    -6994
 302     -577      -37
---------------------------------------------------------------
---------------------

Movements:
---------------------------------------------------------------
---------------------
global:                       Dx       Dy       Dz           Rx
      Ry      Rz
                                  (millimeters)
   (degrees)
---------------------------------------------------------------
---------------------
 1 Loads in the working state   0        0        0
0.00      0.00     0.00
 2 Loads in the cold state      0        0        0
0.00      0.00     0.00
---------------------------------------------------------------
---------------------



>>>  Node:                   P5
     Type of support:        anchor

Forces:
---------------------------------------------------------------
---------------------
```

```
global:                           Fx        Fy        Fz
 Mx        My       Mz
                                         (newtons)
      (newton*meter)
---------------------------------------------------------------
---------------------
 1 Loads in the working state    46572   -44596    21832
18513    20494    11974
 2 Loads in the cold state        -423      497    -3143
 171     -462      -24
---------------------------------------------------------------
---------------------

Movements:
---------------------------------------------------------------
---------------------
global:                        Dx        Dy        Dz           Rx
      Ry       Rz
                                   (millimeters)
   (degrees)
---------------------------------------------------------------
---------------------
 1 Loads in the working state   0        0        0
0.00      0.00     0.00
 2 Loads in the cold state      0        0        0
0.00      0.00     0.00
---------------------------------------------------------------
---------------------



>>>  Node:                    N1
     Type of support:        spring hanger, number of chains:
NC = 2

     Spring:                 07/Z2 (OST80). Chain structure:
1*19 (Maximum working travel -  140 mm)

     Design load:                      21451    H_fr. = 226/414
mm
     Theoretical installation load:    26973    H_1s  = 178/318
mm
---------------------------------------------------------------
-------------------------
                                P      Pmax      Pmin      FS
H_70  H_140   Var
                                      (newtons)
(millimeters)    %
---------------------------------------------------------------
-------------------------
 1 Loads in the working state   21404    39325           1.84
 188    338     0
 2 Loads in the cold state      26774    39325           1.47
 178    319    25
---------------------------------------------------------------
-------------------------
```

```
Movements:
-----------------------------------------------------------------
----
                                  Dx       Dy       Dz
ALPHA
                                       (millimeters)
degrees
-----------------------------------------------------------------
----
 1 Loads in the working state    -1.34    -5.92     0.71    0.2
-----------------------------------------------------------------
----



>>>  Node:                   N2
     Type of support:        spring hanger, number of chains:
NC = 1

     Spring:               06/Z3 (OST80). Chain structure:
1*06+1*18 (Maximum working travel -  210 mm)

     Design load:                      8349    H_fr. = 201/369
mm
     Theoretical installation load:    10980    H_1s  = 154/275
mm
-----------------------------------------------------------------
-------------------------
                                  P      Pmax      Pmin      FS
 H_70  H_140   Var
                                     (newtons)
(millimeters)    %
-----------------------------------------------------------------
-------------------------
 1 Loads in the working state    8312    16338           1.97
 165    298     0
 2 Loads in the cold state      10371    16338           1.58
 157    280    24
-----------------------------------------------------------------
-------------------------

Movements:
-----------------------------------------------------------------
----
                                  Dx       Dy       Dz    ALPHA
                                       (millimeters)
degrees
-----------------------------------------------------------------
----
 1 Loads in the working state   11.54    -4.87    34.28    0.7
 2 Loads in the cold state      -3.18   -10.43     7.82    0.6
-----------------------------------------------------------------
----



>>>  Node:                   N3
```

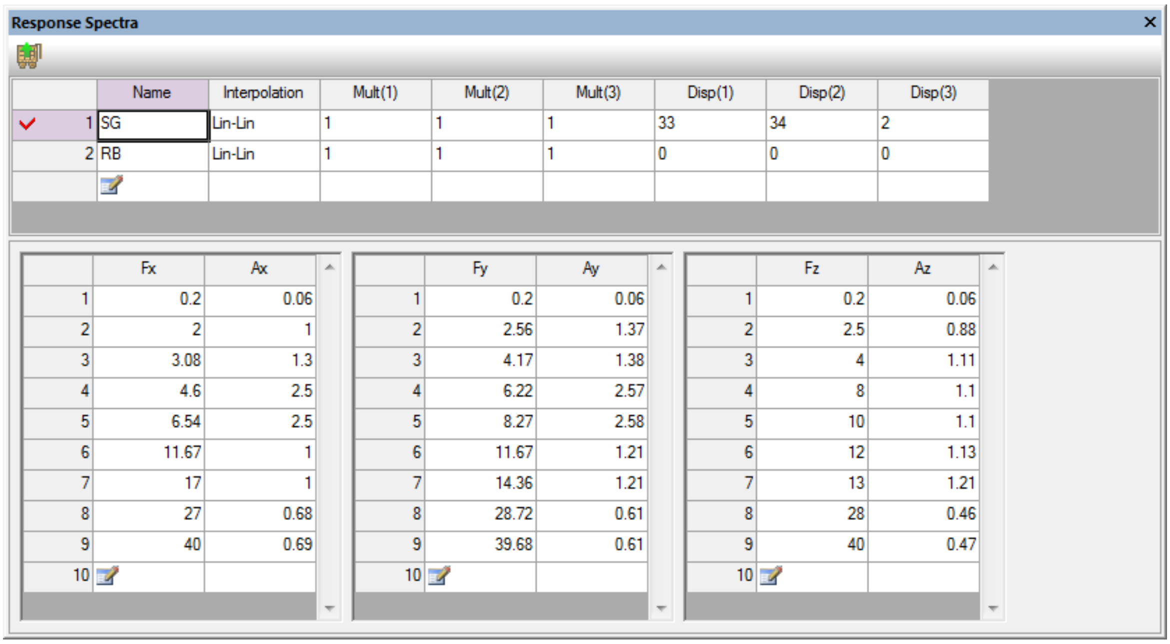
```
     Type of support:        spring hanger, number of chains:
NC = 2

     Spring:                06/Z1 (OST80). Chain structure:
1*06 (Maximum working travel -   70 mm)

     Design load:                      19326    H_fr. = 201 mm
     Theoretical installation load:    31667    H_1э  = 133 mm
----------------------------------------------------------------
-------------------------
                                P      Pmax     Pmin     FS
H_70  H_140   Var
                                     (newtons)
(millimeters)    %
----------------------------------------------------------------
-------------------------
 1 Loads in the working state   19206    32676           1.70
 160           1
 2 Loads in the cold state      24611    32676           1.33
 148          27
----------------------------------------------------------------
-------------------------

Movements:
----------------------------------------------------------------
----
                                  Dx       Dy       Dz
ALPHA
                                       (millimeters)
degrees
----------------------------------------------------------------
----
 1 Loads in the working state    24.07     2.61    26.69    0.7
 2 Loads in the cold state       -2.25    -5.59    15.12    0.2
----------------------------------------------------------------
----



>>>  Node:                   N4
     Type of support:        spring hanger, number of chains:
NC = 1

     Spring:                06/Z1 (OST80). Chain structure:
1*06 (Maximum working travel -   70 mm)

     Design load:                      10845    H_fr. = 201 mm
     Theoretical installation load:    13155    H_1э  = 145 mm
----------------------------------------------------------------
-------------------------
                                P      Pmax     Pmin     FS
H_70  H_140   Var
                                     (newtons)
(millimeters)    %
----------------------------------------------------------------
-------------------------
```

```
 1 Loads in the working state   10879    16338           1.50
 154            0
 2 Loads in the cold state       9560    16338           1.71
 160           12
----------------------------------------------------------------
-------------------------

Movements:
----------------------------------------------------------------
----
                                   Dx       Dy       Dz
ALPHA
                                        (millimeters)
degrees
----------------------------------------------------------------
----
 1 Loads in the working state    40.47     4.68     9.75    2.8
 2 Loads in the cold state        0.72    -0.15    15.40    0.1
----------------------------------------------------------------
----



>>>  Node:                   N5
     Type of support:        spring hanger, number of chains:
NC = 1

     Spring:                07/Z1 (OST80). Chain structure:
1*07 (Maximum working travel -   70 mm)

     Design load:                      12651    H_fr. = 226 mm
     Theoretical installation load:    15974    H_1э  = 169 mm
----------------------------------------------------------------
-------------------------
                                 P      Pmax     Pmin     FS
H_70  H_140   Var
                                      (newtons)
(millimeters)    %
----------------------------------------------------------------
-------------------------
 1 Loads in the working state   12691    19662           1.55
 181            0
 2 Loads in the cold state      12095    19662           1.63
 183            4
----------------------------------------------------------------
-------------------------

Movements:
----------------------------------------------------------------
----
                                   Dx       Dy       Dz
ALPHA
                                        (millimeters)
degrees
----------------------------------------------------------------
----
```

```
 1 Loads in the working state     66.76    24.14    11.69
5.8
 2 Loads in the cold state        -0.73    -0.15    13.81
0.1
-----------------------------------------------------------------
----



>>>  Node:                    N6
     Type of support:         spring hanger, number of chains:
NC = 1

     Spring:                 07/Z1 (OST80). Chain structure:
1*07 (Maximum working travel -   70 mm)

     Design load:                      13738    H_fr. = 226 mm
     Theoretical installation load:    17992    H_1s  = 162 mm
-----------------------------------------------------------------
-------------------------
                                P      Pmax     Pmin     FS
H_70  H_140   Var
                                     (newtons)
(millimeters)    %
-----------------------------------------------------------------
-------------------------
 1 Loads in the working state  13553    19662           1.45
178            1
 2 Loads in the cold state     14862    19662           1.32
173            8
-----------------------------------------------------------------
-------------------------

Movements:
-----------------------------------------------------------------
----
                                  Dx        Dy        Dz
ALPHA
                                       (millimeters)
degrees
-----------------------------------------------------------------
----
 1 Loads in the working state    89.31    43.59    15.80    8.5
 2 Loads in the cold state       -1.76    -0.15    11.14    0.2
-----------------------------------------------------------------
----



>>>  Node:                    N7
     Type of support:               spring hanger, number of
chains: NC = 1

     Spring:                 08/Z2 (OST80). Chain structure:
1*20 (Maximum working travel -  140 mm)
```

```
      Design load:                        13456    H_fr. = 221/399
mm
      Theoretical installation load:      16468    H_1s  = 177/311
mm
----------------------------------------------------------------
-------------------------
                                P       Pmax     Pmin     FS
H_70  H_140   Var
                                     (newtons)
(millimeters)    %
----------------------------------------------------------------
-------------------------
 1 Loads in the working state  13376    26341           1.97
185    328     1
 2 Loads in the cold state     16007    26341           1.65
178    314    19
----------------------------------------------------------------
-------------------------

Movements:
----------------------------------------------------------------
----
                                  Dx       Dy       Dz
ALPHA
                                       (millimeters)
degrees
----------------------------------------------------------------
----
 1 Loads in the working state    76.33    51.62    16.43    8.0
 2 Loads in the cold state       -1.89     0.21     2.45    0.2
----------------------------------------------------------------
----
```

10. In order to execute the seismic stability analysis it is necessary to enter the floor response spectrums for various support groups ("Main data\Response spectrums" menu, button):

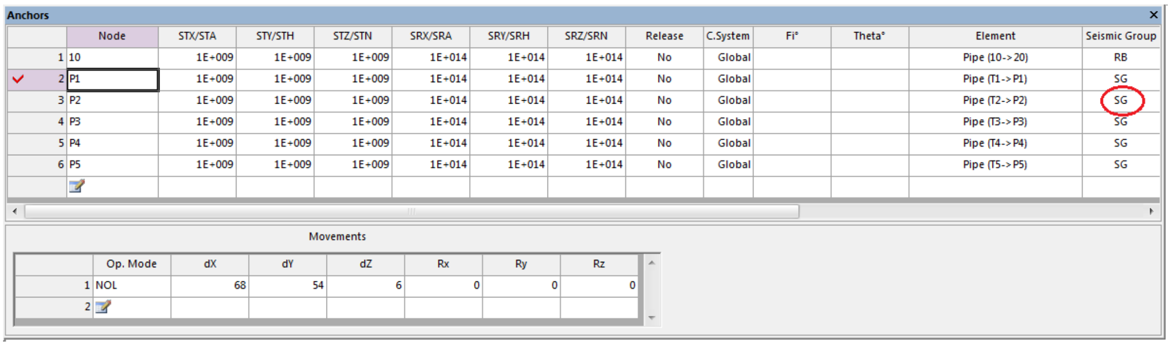


	Name	Interpolation	Mult(1)	Mult(2)	Mult(3)	Disp(1)	Disp(2)	Disp(3)
1	SG	Lin-Lin	1	1	1	33	34	2
2	RB	Lin-Lin	1	1	1	0	0	0

	Fx	Ax	Fy	Ay	Fz	Az
1	0.2	0.06	0.2	0.06	0.2	0.06
2	2	1	2.56	1.37	2.5	0.88
3	3.08	1.3	4.17	1.38	4	1.11
4	4.6	2.5	6.22	2.57	8	1.1
5	6.54	2.5	8.27	2.58	10	1.1
6	11.67	1	11.67	1.21	12	1.13
7	17	1	14.36	1.21	13	1.21
8	27	0.68	28.72	0.61	28	0.46
9	40	0.69	39.68	0.61	40	0.47
10						

If the spectrums are located in separate files, then it is possible to load them by pressing the () button.

11. For all supports of the model, it is necessary to determine the corresponding seismic group ("Additional data\Anchors" and "Additional data\Spring hangers" menus):



Anchors

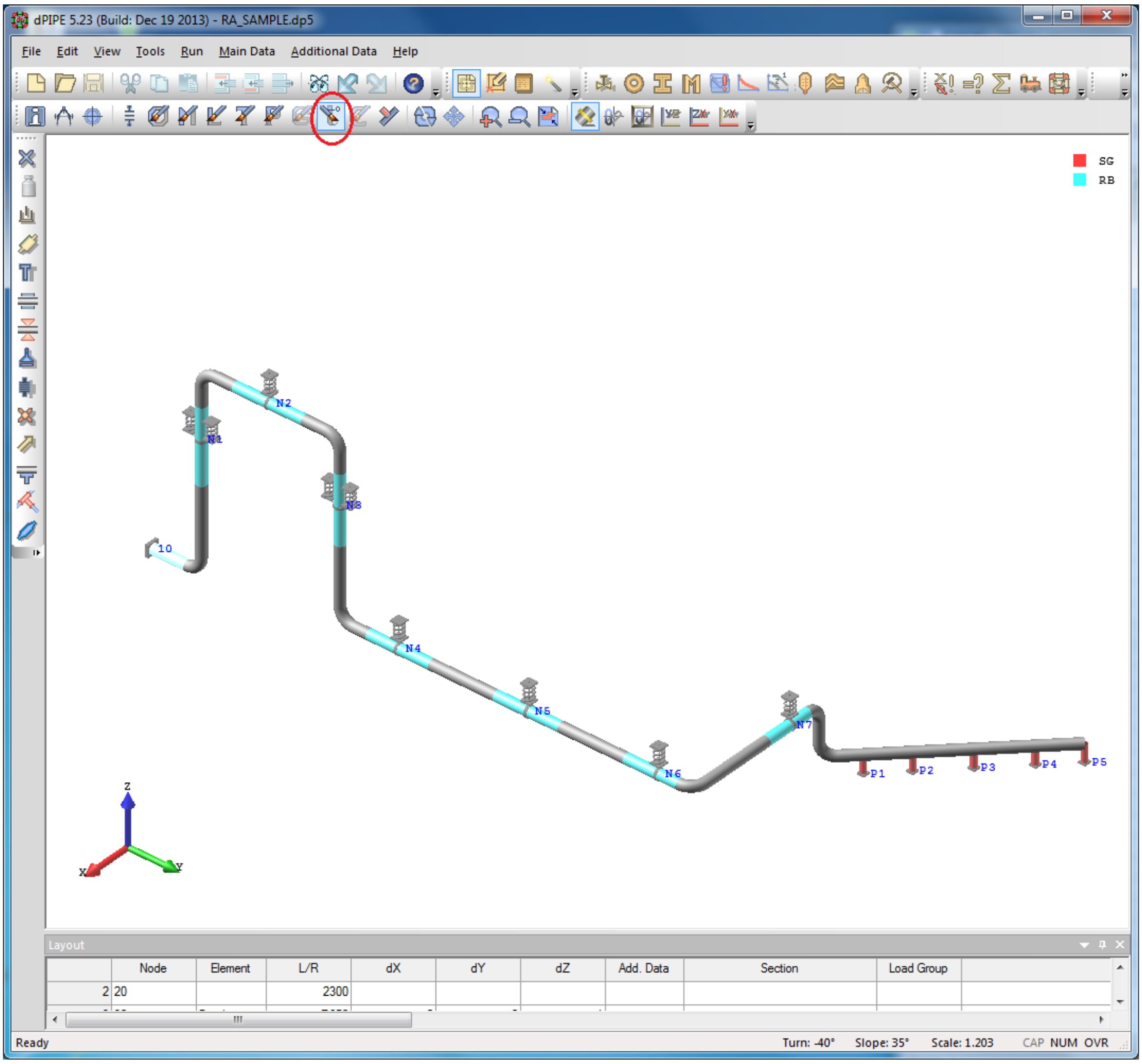
	Node	STX/STA	STY/STH	STZ/STN	SRX/SRA	SRY/SRH	SRZ/SRN	Release	C.System	Fi°	Theta°	Element	Seismic Group
1	10	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (10->20)	RB
2	P1	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T1->P1)	SG
3	P2	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T2->P2)	SG
4	P3	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T3->P3)	SG
5	P4	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T4->P4)	SG
6	P5	1E+009	1E+009	1E+009	1E+014	1E+014	1E+014	No	Global			Pipe (T5->P5)	SG

Movements

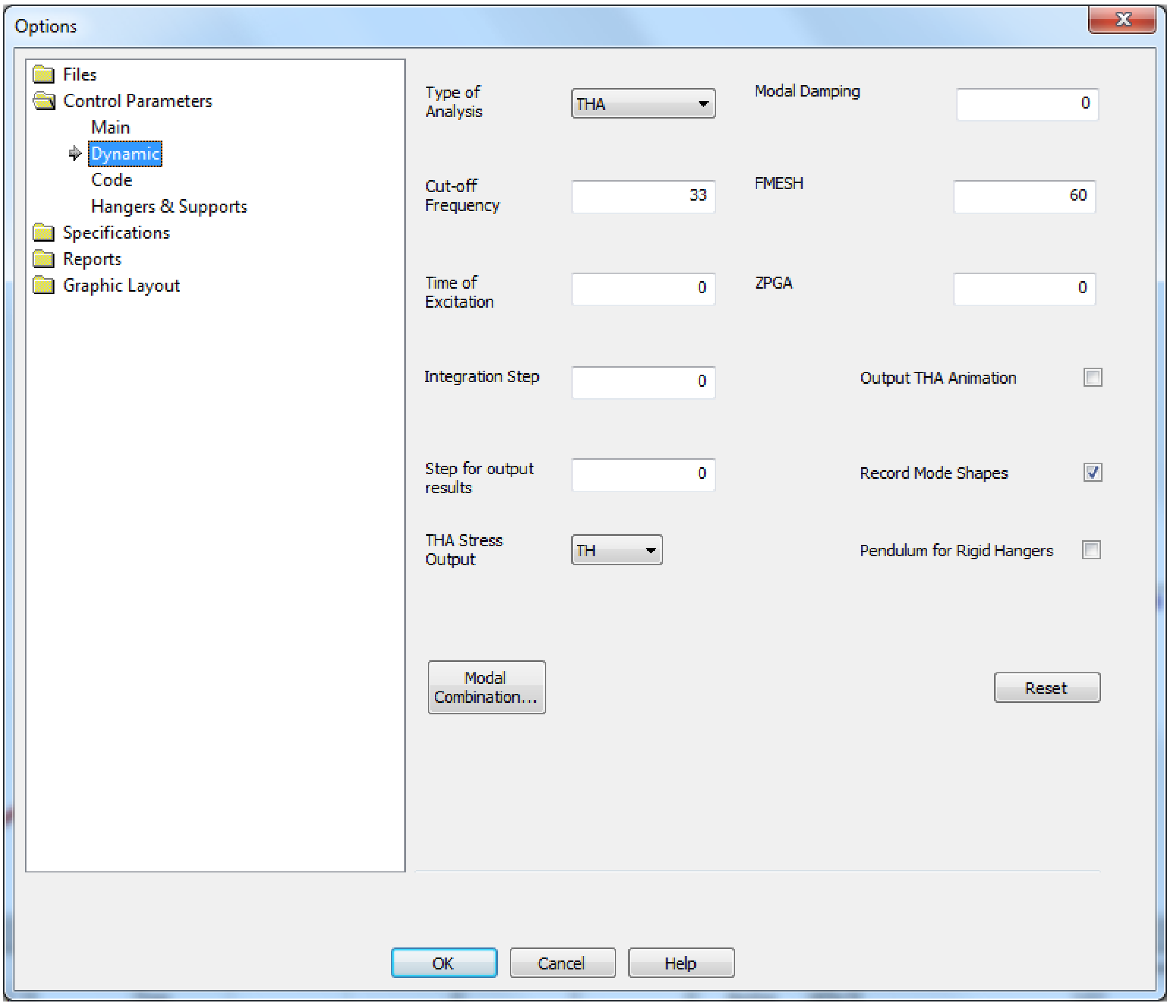
	Op. Mode	dX	dY	dZ	Rx	Ry	Rz
1	NOL	68	54	6	0	0	0
2							

The same operation can be executed via the spreadsheet interface by using the "Service\ Determine seismic support groups" menu item ()

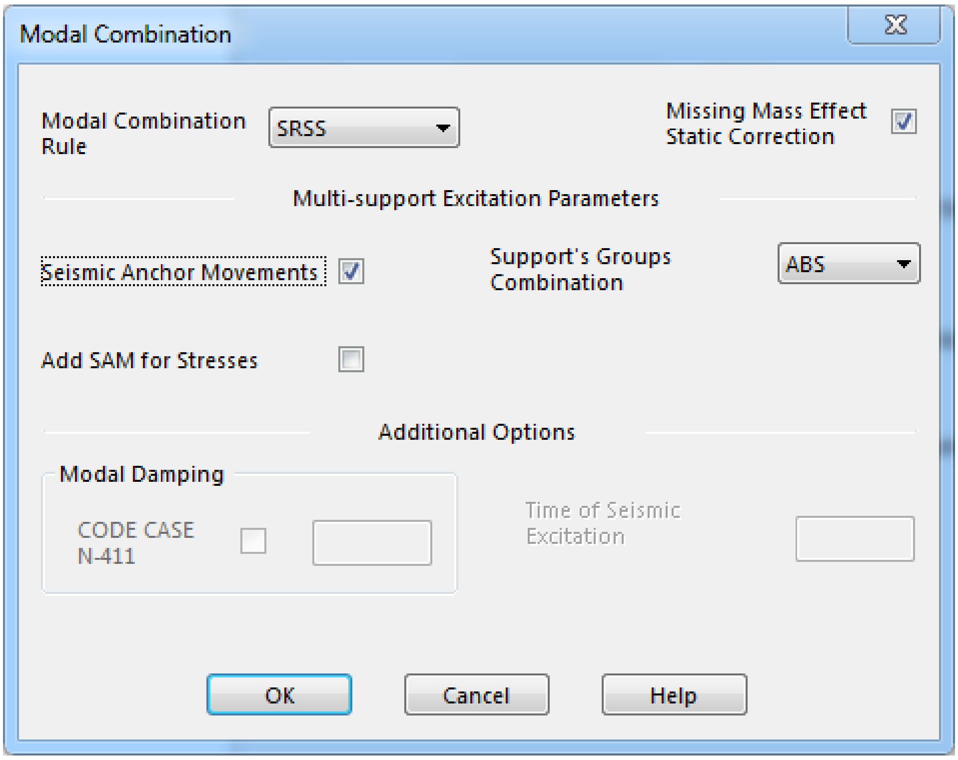
12. It is possible to check the correctness of the assignment of seismic groups for a certain support by pressing the  button on the toolbar.



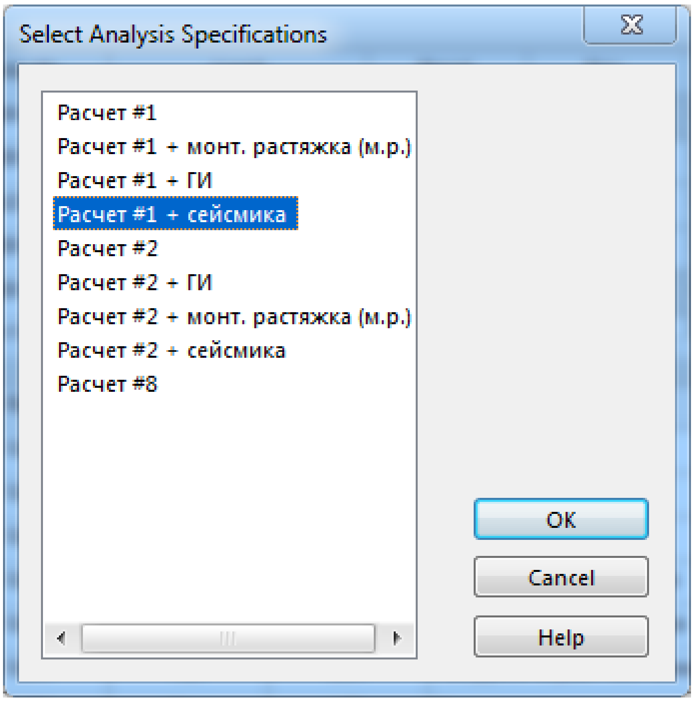
13. Specify the method of analysis for dynamic analysis (DYN) as the response spectrum method (RSM), limit frequency FMAX as 33 Hz and the parameter for automatic breakdown of the model FMESH as 60 Hz ("Service\Options\Reference parameters\Dynamic" menu):



14. In the "Modal combination..." dialog, place check-marks in the "Allowance for higher forms of vibration" and "Allowance for seismic displacement of supports" items:



15. Load the specification for analysis including seismic factors ("Service\Options\Tasks" menu or  button):



16. In order to execute the analysis and view the results, it is necessary to repeat the operations set forth in Item 9