UM – Drillstring Analysis Software

Getting started

Edition 1

2021

Powered By

Getting Started Using UM Drillstring Analysis Software: simulation of drill string behavior in well

This manual leads you through the process of UM Drillstring Analysis installation and basics of describing the models, running and analyzing results of three types of analysis:

- **Static Analysis:** Evaluation of the equilibrium position of the Bottom Hole Assembly for the defined bit location in the borehole. The analysis provides an accurate representation of the deformed shape of the assembly in the hole, forces acting on the assembly, as well as internal force factors and stresses in the assembly units.

- **Torque & Drag Analysis:** Estimation of resistance torque and axial drag forces acting on drillstring in wellbore for standard set of technical operations: rotary and slide drilling, rotation off bottom, pick up and slack off, back reaming, reaming and fishing. Distributions of axial forces and torques, internal force factors and stresses, as well deformations an assembly are estimated.
  + **Soft-string** analysis is the fast solution based on assumption of positioning of assembly along the well trajectory.
  + **Stiff-string** analysis is the detailed analysis considering local deformations and lateral displacements of assembly units in the borehole; application of Static Analysis approach for the whole drillstring analysis.

- **Time Domain Analysis:** Simulation of drillstring motion in borehole with account of side contact with wellbore walls, bit-rock interaction, and any set of harmonic excitations; determination of dangerous dynamic effects having place in the assembly, estimation of stress load of drillstring components, optimization of drilling loots characteristics and operational settings.

We will consider Static and Time Domain Analysis of bottom hole assembly on the single test model - Rotary BHA. Support of assemblies equipped by mud motors and steering tools will be added in the next release of the software. Torque&Drag Analysis will be demonstrated for the single assembly.

It assumes that you go through the manual step by step sequentially. Information that is given in one section might be further given shortly or even omit.

Compatibility

	32-bit	64-bit
Windows XP	✓	✓
Windows Vista	✓	✓
Windows 7	✓	✓
Windows 8	✓	✓
Windows 10	✓	✓

Copyright and trademarks

This manual is prepared for informational use only, may be revised from time to time. No responsibility or liability for any errors that may appear in this document is supposed.

Copyright and trademarks

This manual is prepared for informational use only, may be revised from time to time. No responsibility or liability for any errors that may appear in this document is supposed.

Contact information

Address:

Laboratory of Computational Mechanics
Bryansk State Technical University,
Kharkovskaya 10B, Bryansk, 241035,
Russia

Contact person:

Nikolay Lysikov, Gennady Mikheev

Phones: +7 4832 588329, +7 4832 568637
E-mails:

General enquiries: um@universalmechanism.com
Questions, suggestions, bug reports: support@universalmechanism.com
Sales, promotion, advertising, collaboration: sales@universalmechanism.com

CONTENT

GETTING STARTED USING UM DRILLSTRING ANALYSIS SOFTWARE: SIMULATION OF DRILL STRING BEHAVIOR IN WELL 2

INSTALLING UM DRILLSTRING ANALYSIS SOFTWARE 7

1. GRAPHICAL USER INTERFACE 12

1.1. Concept 12

1.2. Workflow 16

1.3. General GUI settings 17
1.3.1. System of Units 18
1.3.2. User Mode 19
1.3.4. Source Directory 20
1.3.5. Interface Settings 21

1.4. GUI Databases & Tools 22
1.4.1. Material, formation, fluid databases 23
1.4.1.1. Material Database 26
1.4.1.2. Formation Database 27
1.4.1.3. Fluid Database 28
1.4.2. String Part Database 29
1.4.2.1. Drillstring Part Editor 32
1.4.3. Well Trajectory Editor 39
1.4.4. Wellbore Editor 43
1.4.5. Drillstring / BHA Editor 45

1.5. GUI Drillstring Analysis Project 51
1.5.1. Start New Project 52
1.5.2. Save Project 53
1.5.3. Open Project 54
1.5.4. Project window 55
1.5.4.1. Project navigation tools 55
1.5.4.2. General project info page 56
1.5.4.3. Drilling Information page 57
1.5.4.4. Analysis GUI 60

2. TRAINING PROJECT 63

2.1. General project data input 64

2.2. Drilling information description 66
2.2.1. Well trajectory 66
2.2.2. Wellbore construction 67
2.2.3. BHA Cases 68
2.2.3.1. Rotary BHA description 69
2.2.4. Drillstring Cases 72

2.3. Static Analysis 73
2.3.1. Scenario description 74
2.3.2. Analysis run 77
2.3.3. Results output 79
2.3.3.1. Brief results 80
2.3.3.2. 3D View 81
2.3.3.1. 2D View 83

2.4. Torque & Drag Analysis 84
2.4.1. Scenario description 85
2.4.2. Analysis run 91
2.4.3. Results output 92
2.4.3.1. 2D plots 93
2.4.3.2. Table output 96

2.5. Time Domain Analysis 97
2.5.1. Scenario description 99
2.5.2. Run scenario by Simulation Desk 103
2.5.3. Wizard of variables 108
2.5.4. Replay calculated scenario by Simulation Desk 112
2.5.5. Multi-scenario run 114
2.5.6. Results output 117
2.5.6.1. Initial State 118
2.5.6.2. Variables 119
2.5.6.3. Plots 120
2.5.6.4. Functionals 121
2.5.7. Excitations 122
2.5.8. Comparison mode 124

Installing UM Drillstring Analysis Software

This section contains the description of processes necessary to install UM Drillstring Analysis Software.

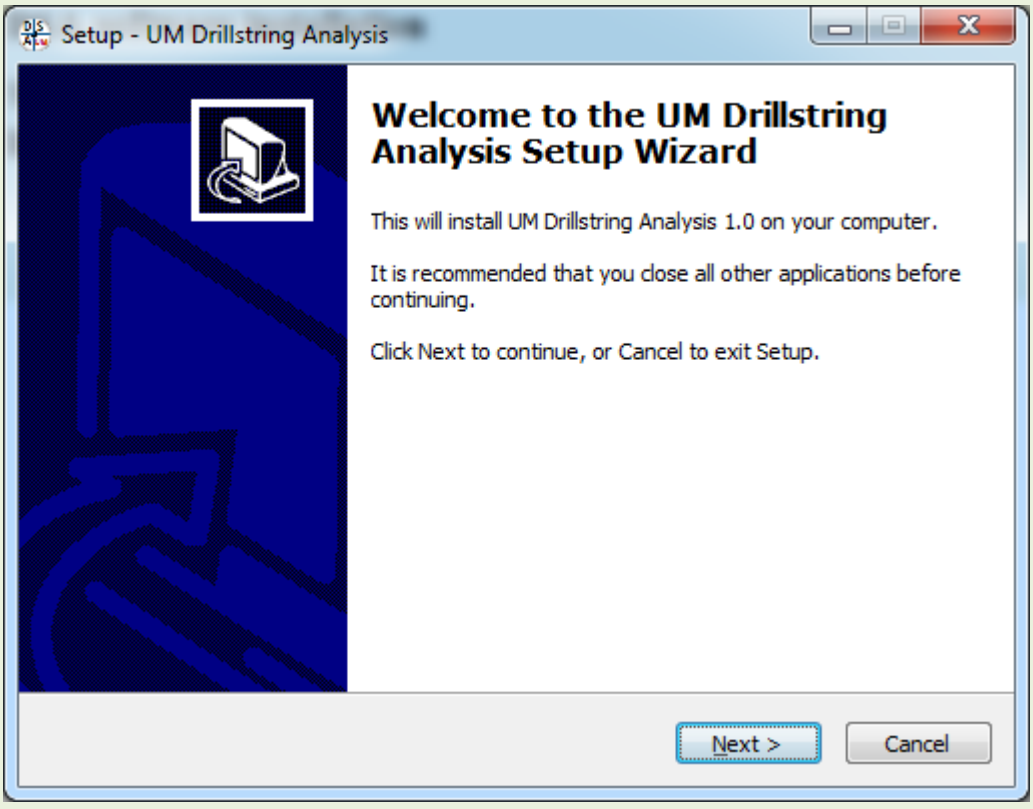
Drillstring Analysis Software Structure

UM Drillstring Analysis Software (here and below - DSA) consists of general GUI and number of executable files and DLL's using for analysis and results output.

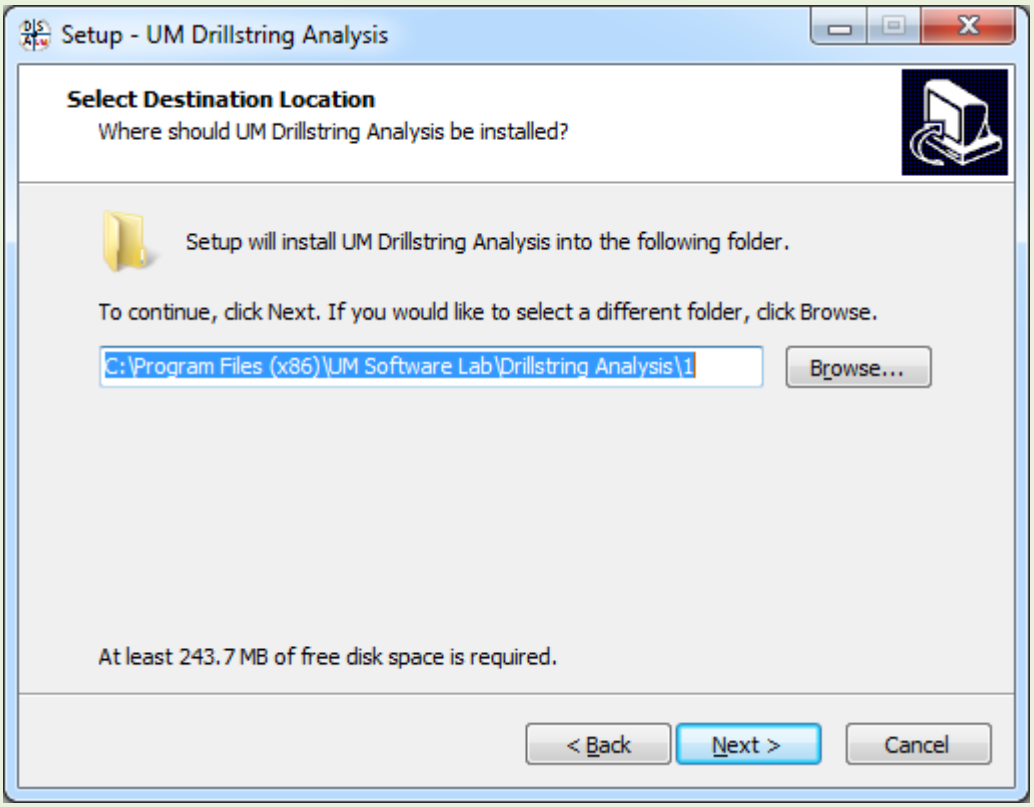
DSA software installation

Start the installation file and follow the steps.

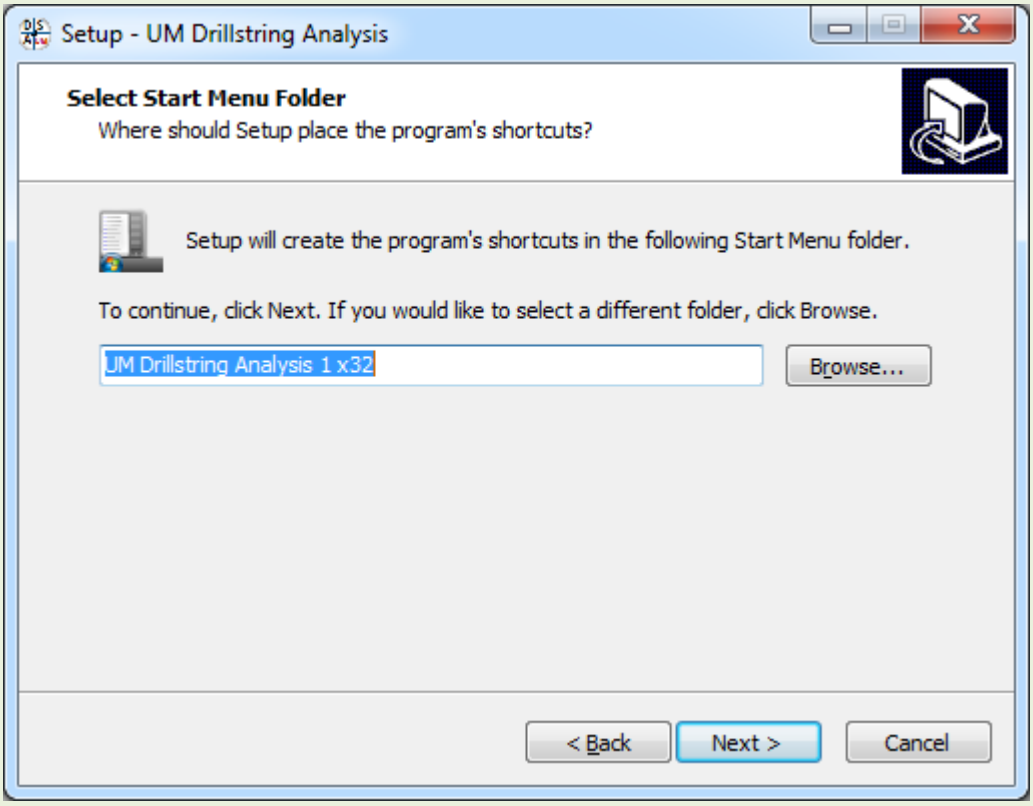
Important! Installation can be started by *Administrator* only.

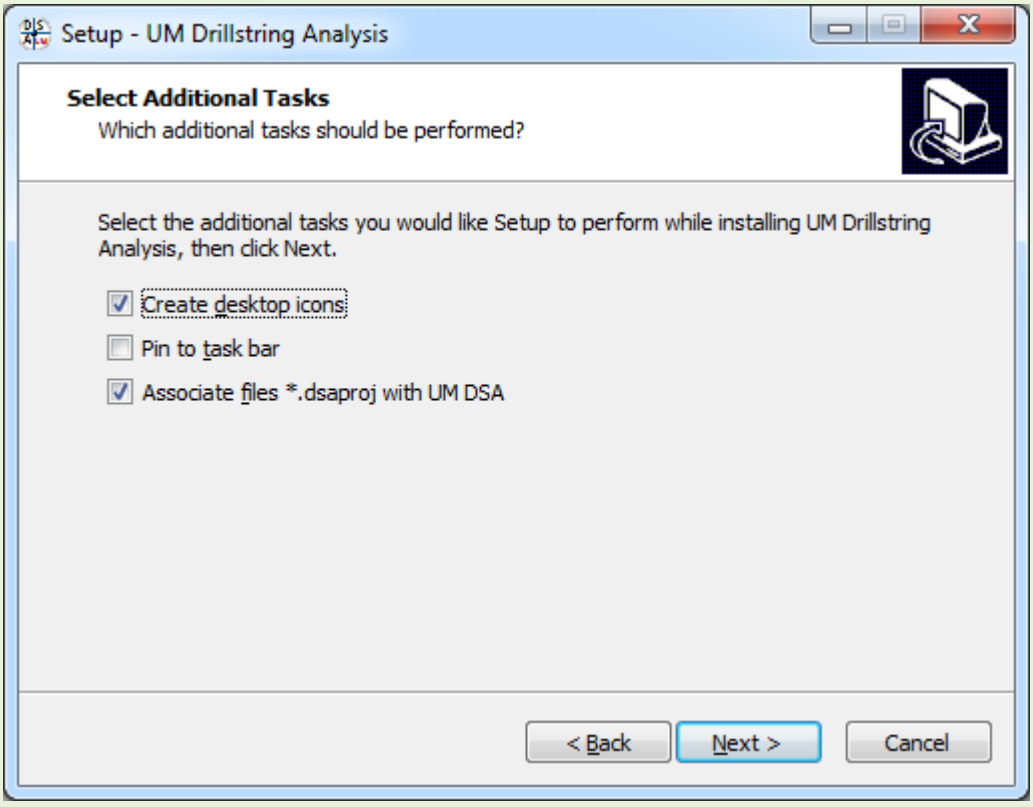


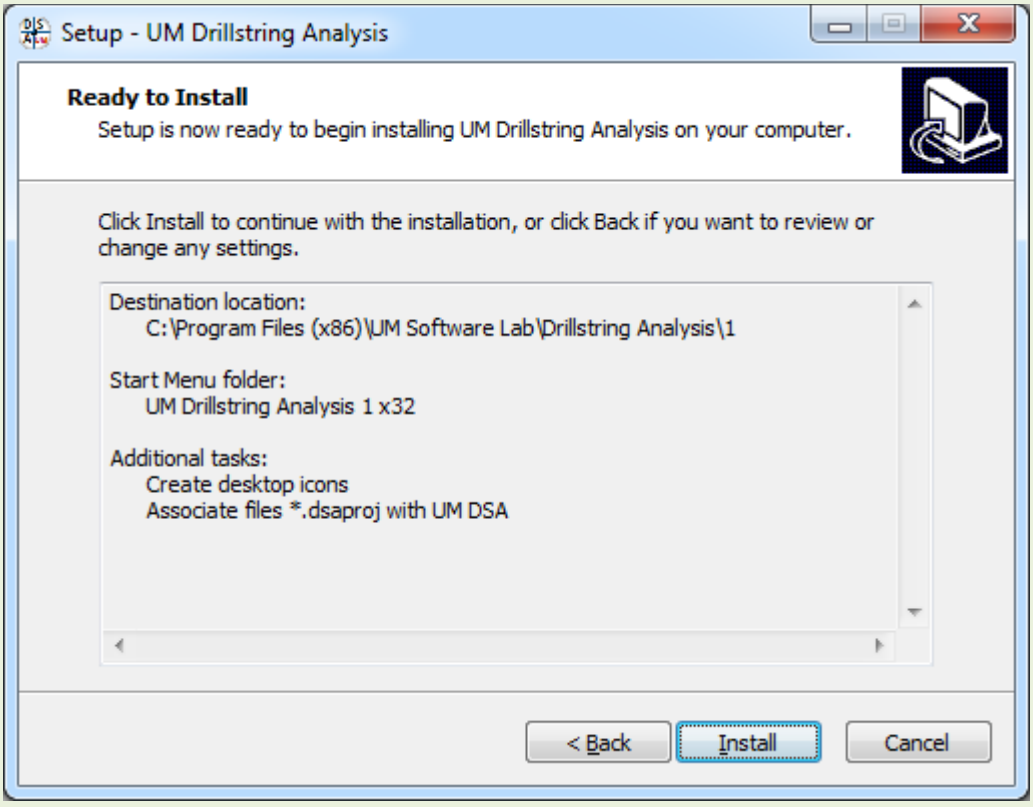
Installation proposes to install application in *C:\Program Files\ Drillstring Analysis\1* folder for Windows XP and *C:\Program Files (x86)\UM Software Lab\Drillstring Analysis\1* folder for Windows 7/8/10 by default. Executable files are copied to the destination.



Program databases and sample projects are installed in *C:\Documents and Settings\All Users\Documents\ UM Software Lab\Drillstring Analysis\1* for Windows XP and *c:\Users\Public\Documents\UM Software Lab\Drillstring Analysis\1* for Windows 7/8/10 by default.

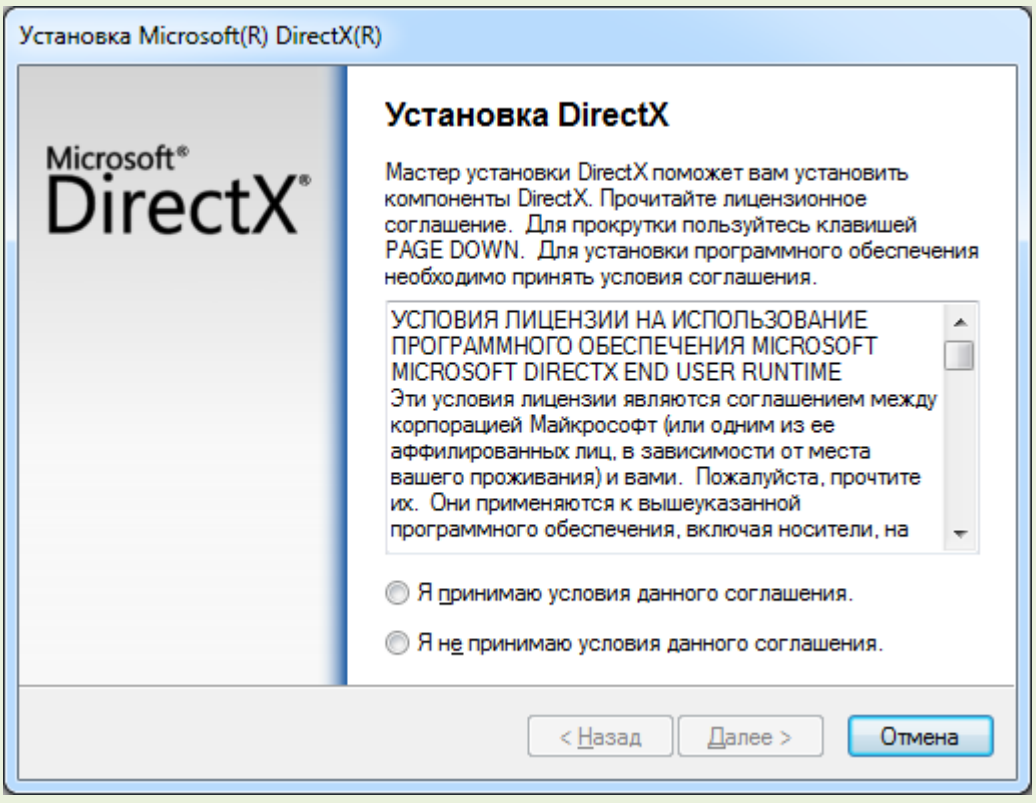

Setup - UM Drillstring Analysis
Select Start Menu Folder
Where should Setup place the program's shortcuts?
Setup will create the program's shortcuts in the following Start Menu folder.
To continue, click Next. If you would like to select a different folder, click Browse.
UM Drillstring Analysis 1 x32
Browse...
< Back
Next >
Cancel

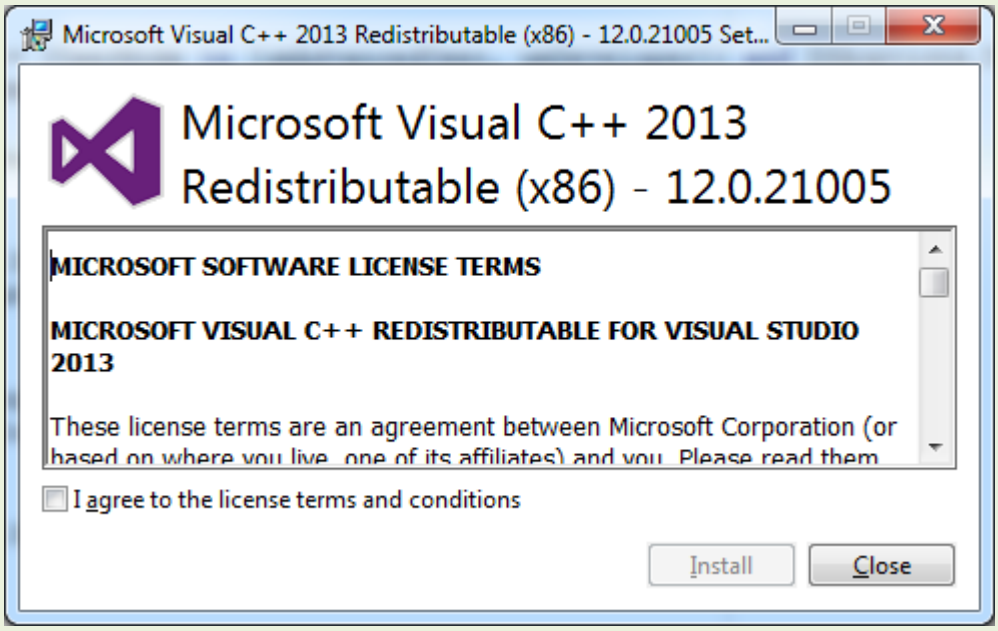

Setup - UM Drillstring Analysis
Select Additional Tasks
Which additional tasks should be performed?
Select the additional tasks you would like Setup to perform while installing UM Drillstring Analysis, then click Next.
Create desktop icons
Pin to task bar
Associate files *.dsaproj with UM DSA
< Back
Next >
Cancel



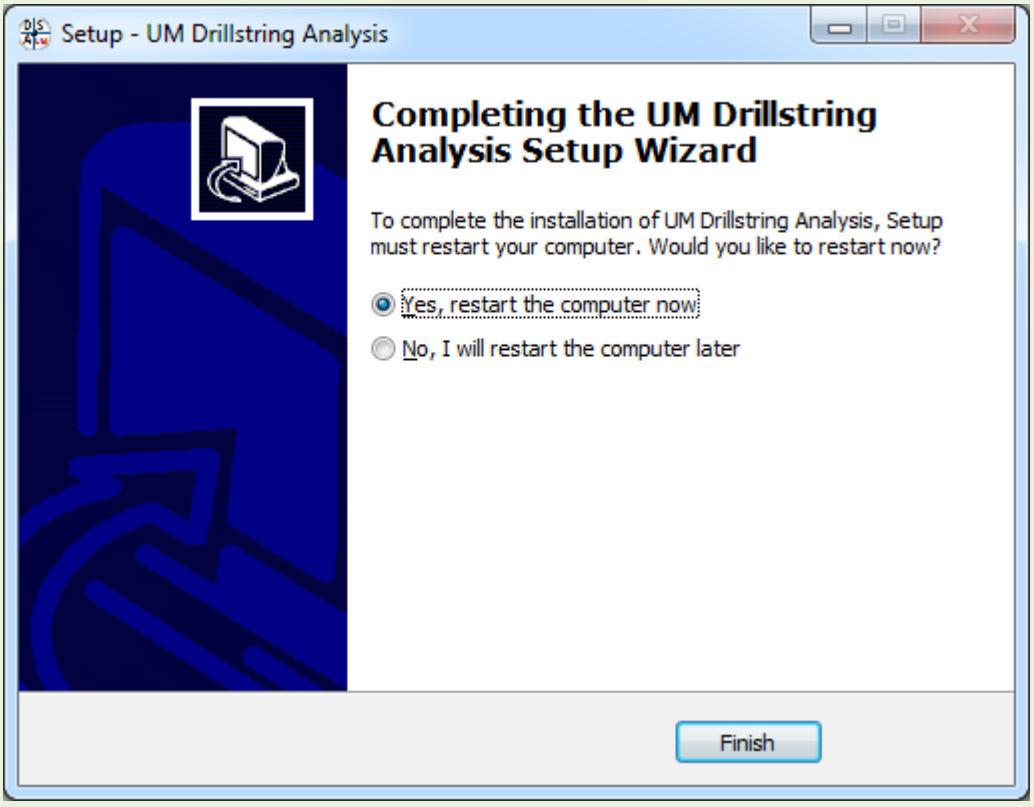
Microsoft DirectX installation

Microsoft DirectX 9 or higher, and Microsoft Visual C++ 2013 Redistributable should be installed on the workstation to enable 3D visualization of drillstring in wellbore. Installations of the components are included in DSA Installation.





Completion of installation will require the restart of workstation. After rebooting UM Drillstring Analysis Software is ready to run.



1. Graphical User Interface

This section contains general information related to UM Drillstring Analysis Software concept, main tools used for input of source data for Static, Torque & Drag and Time Domain analyses and single project structure.

1.1. Concept

Graphical User Interface of Drillstring Analysis is assigned for the input of source data, formulation of the tasks for the solvers, control of calculation process and results post-processing for Static Analysis and Torque&Drag Analysis; Time Domain analysis solver, integrated to the DSA GUI, enables results output during simulation process as well post-processing of the results.

The interface enables storage of source data related to various types of analyses in format of specialized databases, visualization of the calculated results and reporting data in text, MS Excel and PDF formats.

Static analysis

Graphical User Interface of DSA software is oriented on the choice of the bottom hole assembly optimal for work in one or several locations in the definite well on the base of comparison of the results of Static analyses: contact forces, torques, bending moments, stresses, etc.

Torque&Drag analysis

DSA GUI enables description, evaluation and results comparison for a set of combinations of operational conditions for single or several technical operations; estimation of critical levels of operational loads; analysis of friction factor influence on the axial forces and torque distribution, etc.

Time Domain Analysis

The program enables time domain simulation of BHA/drillstring motion in the wellbore with high CPU efficiency; kinematic characteristics as well as forces acting on the assembly can be output during simulation, and stored for post-processing.

Time Domain Analysis functionality is oriented on determination of dangerous dynamic effects having place in the assembly, estimation of stress load of drillstring components, optimization of drilling tools characteristics and operational settings.

Data structure

Static, Torque & Drag and Time Domain analyses are carried out within the drillstring analysis project (*Project*). GUI enables run of single or several projects of the following content:

- *Drilling Information*

  Source data on well trajectory, wellbore intervals, BHA and drillstring design cases used in the project.

- *Analysis*

  Each type of analysis within the *Project* can contain any number of parameter sets – *scenarios* – that can be run in parallel threads or one by another. *Scenario* corresponds to the definite position of drillstring/BHA in the well, set of operational settings and options specific for the analysis. GUI enables analysis of calculated results for single scenario as well as comparison of the outputs for a set of scenarios.

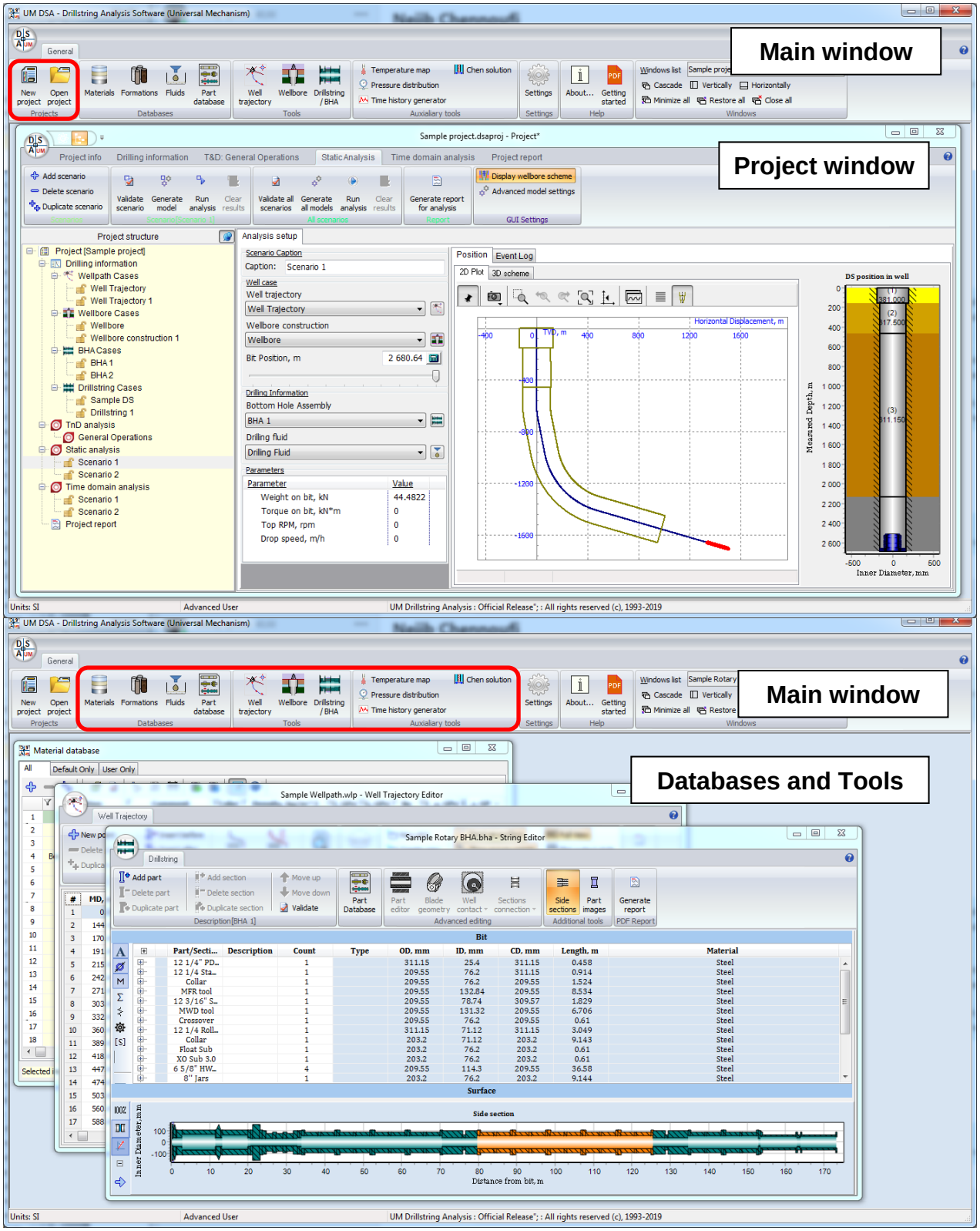
Report Generator

The interface enables automatic generation of the reports on the analysis in PDF format. The user has an ability to customize the content of the report. Reports can include all the project data or just single results or source data view.

Main window

Graphical User Interface is based on **Main Window** which enables access to the general GUI settings, databases and projects, as well as separate tools for input data items description.

Input of source data, analyses customizing and processing, results post-processing and report generation is carried out in single **Project Window**.



System of Units

The interface supports Imperial (API) and International System of Units (SI) and automatic conversion of source data and results from one system to another.

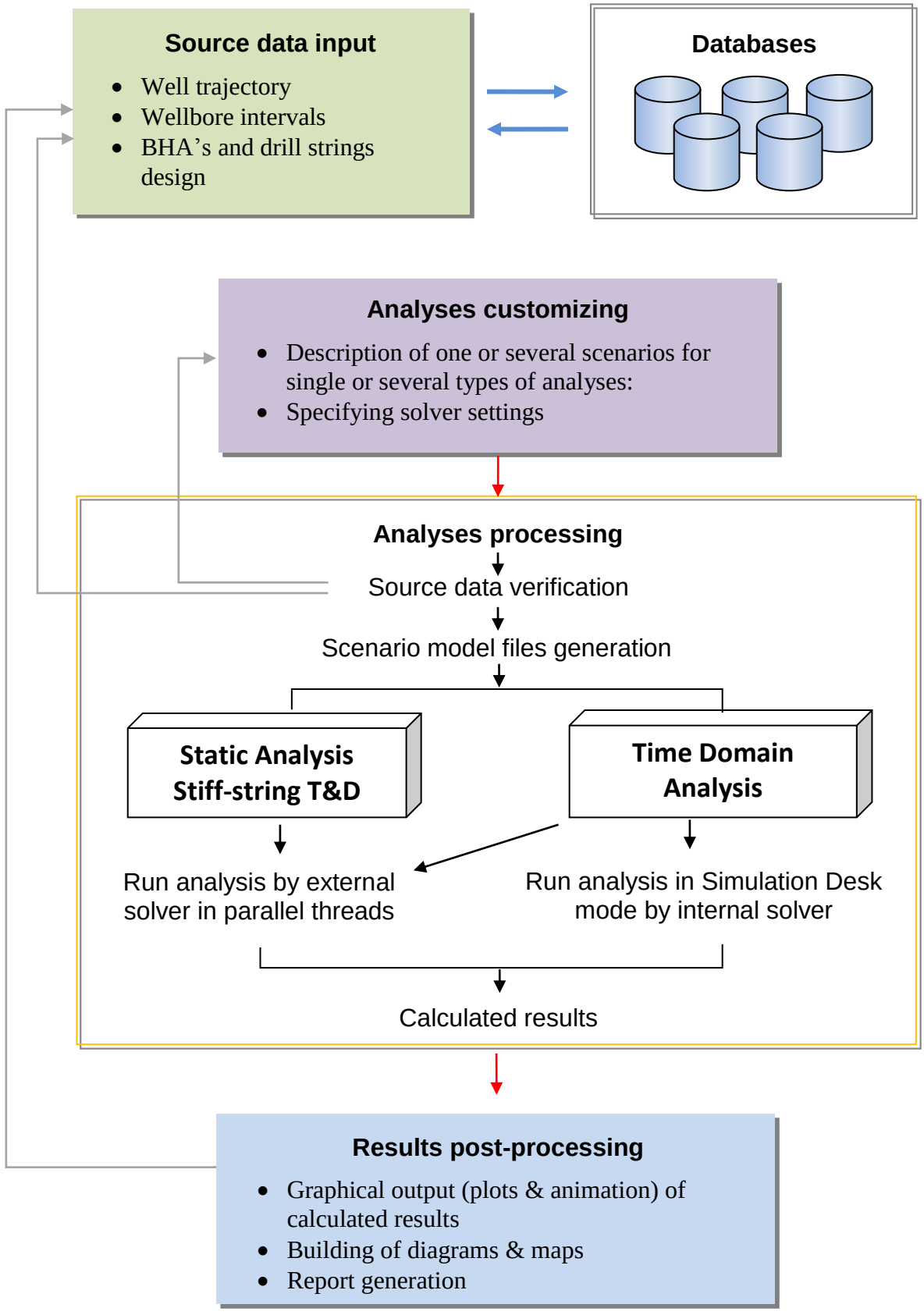
The user can customize unit system settings to select SI or Imperial system for output of different types of parameters.

User Modes

The user can customize the functionality of the interface by selecting ***User*** or ***Advanced User*** mode. ***User*** mode hides the settings which are not necessary for analyses of standard problems.

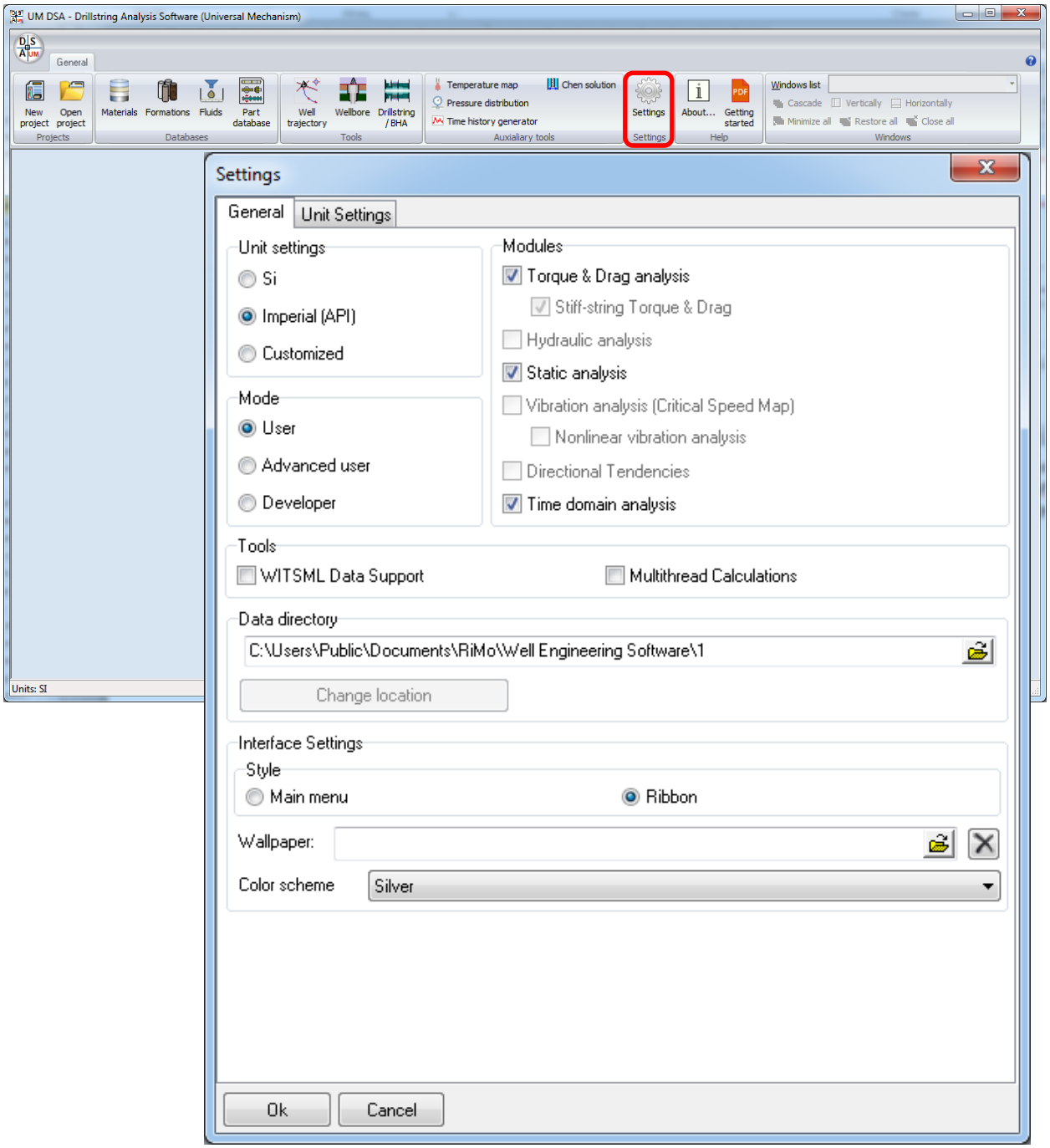
1.2. Workflow

Procedure of drill string analysis within single *Project* can be illustrated with the following scheme.



1.3. General GUI settings

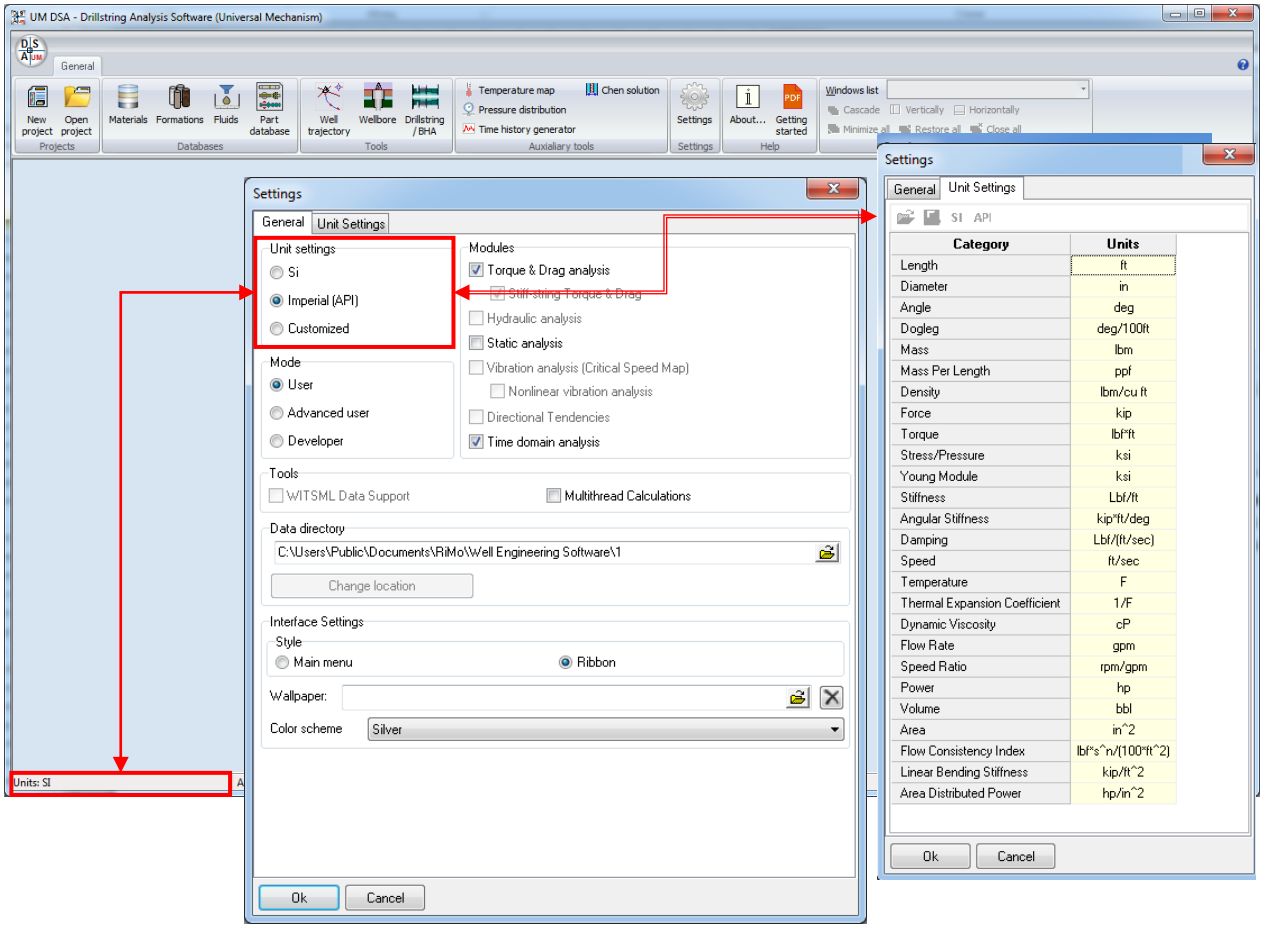
Select **Settings** item of **Main window** head menu to specify general GUI settings: system of units (Si, or Imperial, or Custom) and interface mode (***User***, or ***Advanced user***, or ***Developer***), list of analysis and tools, main menu style, desktop wallpaper, color scheme, etc.



1.3.1. System of Units

The interface supports Imperial and International System of Units (SI) and automatic conversion of source data and results from one system to another.

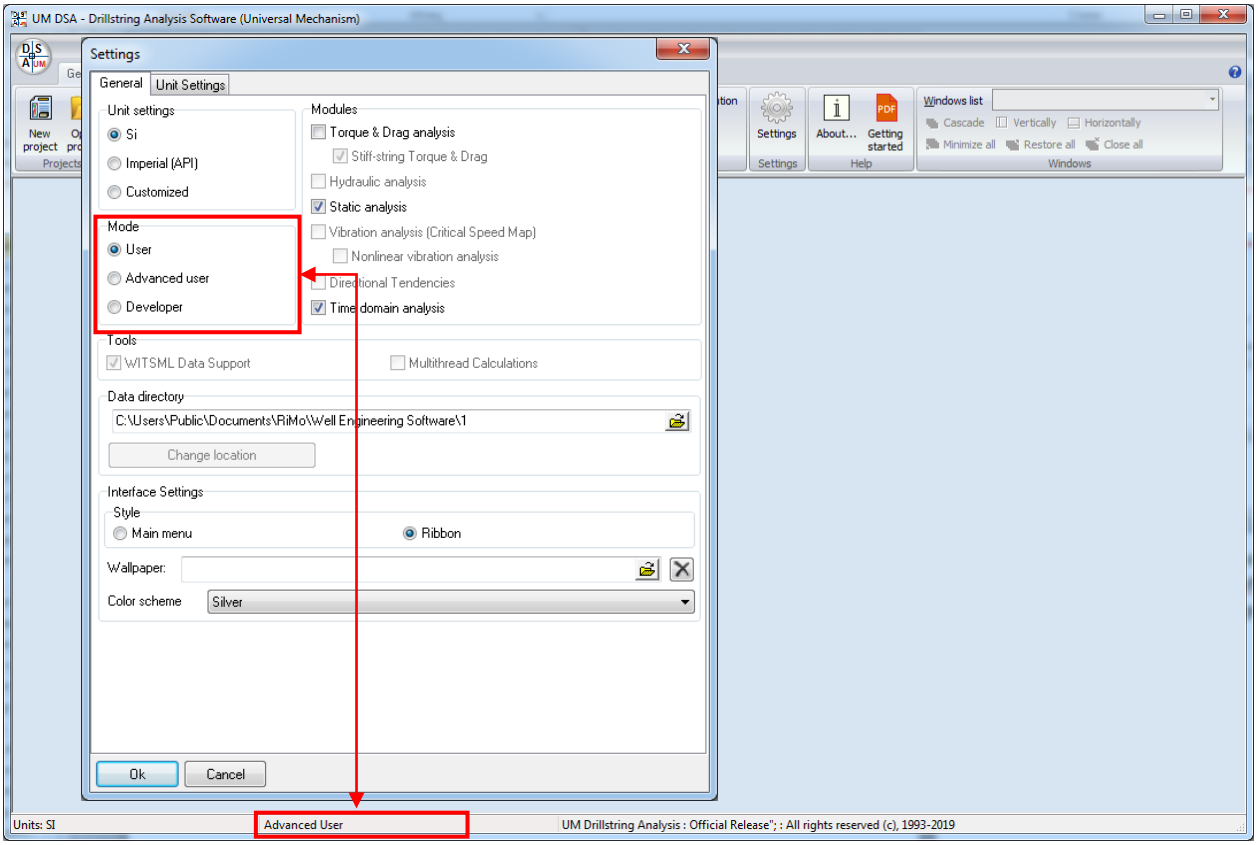
The user can customize unit system settings to select SI or Imperial system for output of different types of parameters on **Unit Settings** tab.



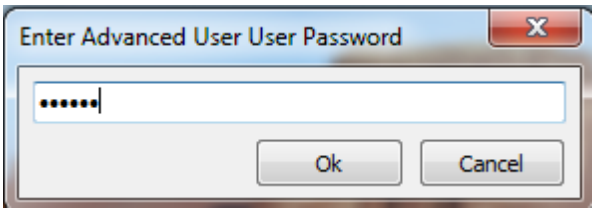
The label of current system of units (**SI**, **Imperial**, or) is displayed in the left bottom corner of the **MainWindow**. Double click the panel or use **Main Window | Options** interface to change the system of units. All data items will be converted automatically.

1.3.2. User Mode

Drillstring Analysis GUI can operate in three modes: ***User*** mode, ***Advanced User***[1] and ***Developer*** mode. ***User*** mode hides or enables settings which are not used in the most cases.



The label of current mode is displayed in the left bottom corner of the **Main Window**. Double click the panel or use **Main Window | Options** interface to change the setting.



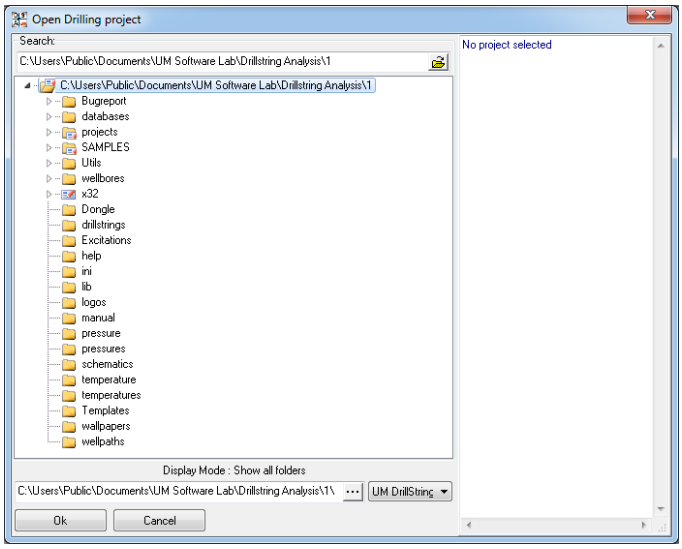
Starting of GUI in ***Advanced User*** or ***Developer*** mode or switching from ***User User*** mode is available after password entering only. If entered password is incorrect the GUI will work in ***User*** mode.

[1] Default *Advanced user* password is «DSA-DSA»

1.3.4. Source Directory

DSA program uses *Source directory* path for search of databases and projects saving. The *Source directory* is placed in *C:\Documents and Settings\All Users\Documents\UM Software Lab\Drillstring Analysis\1* for Windows XP and *C:\Users\Public\Documents\UM Software Lab\Drillstring Analysis\1* for Windows 7, 8, 10 by default.

The list of files and catalogs placed in the directory:



Options.ini – DSA GUI options;
Databases – Material, Fluid, Formation, Drillstring part databases;
Projects – Default storage of Drillstring Analysis projects;
Samples – Sample project files;

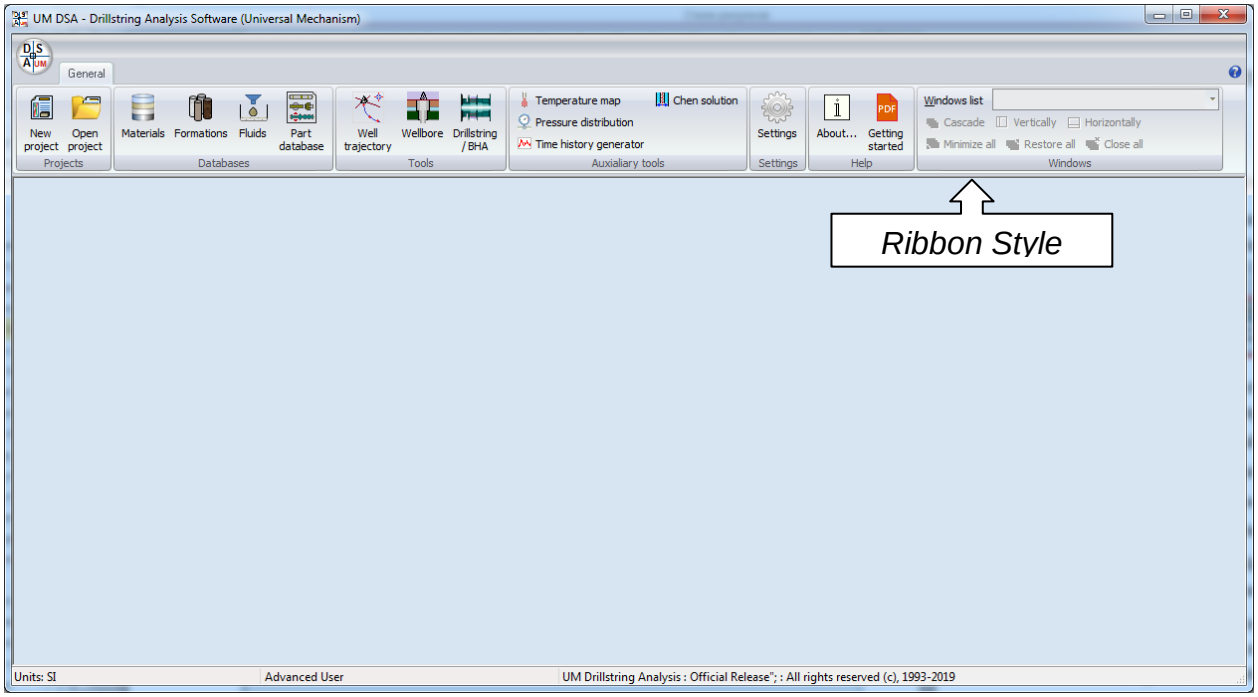
*Wellpaths * – Default storage of files with well trajectory description;
Wellbores – Default storage of files with wellbore intervals description;
Drillstrings – Default storage[1] of drillstring/BHA design files;
Excitations – Default storage of drillstring/BHA excitation description files;
Pressures – Default storage of pressure map files;
Temperatures – Default storage of temperature distribution files;

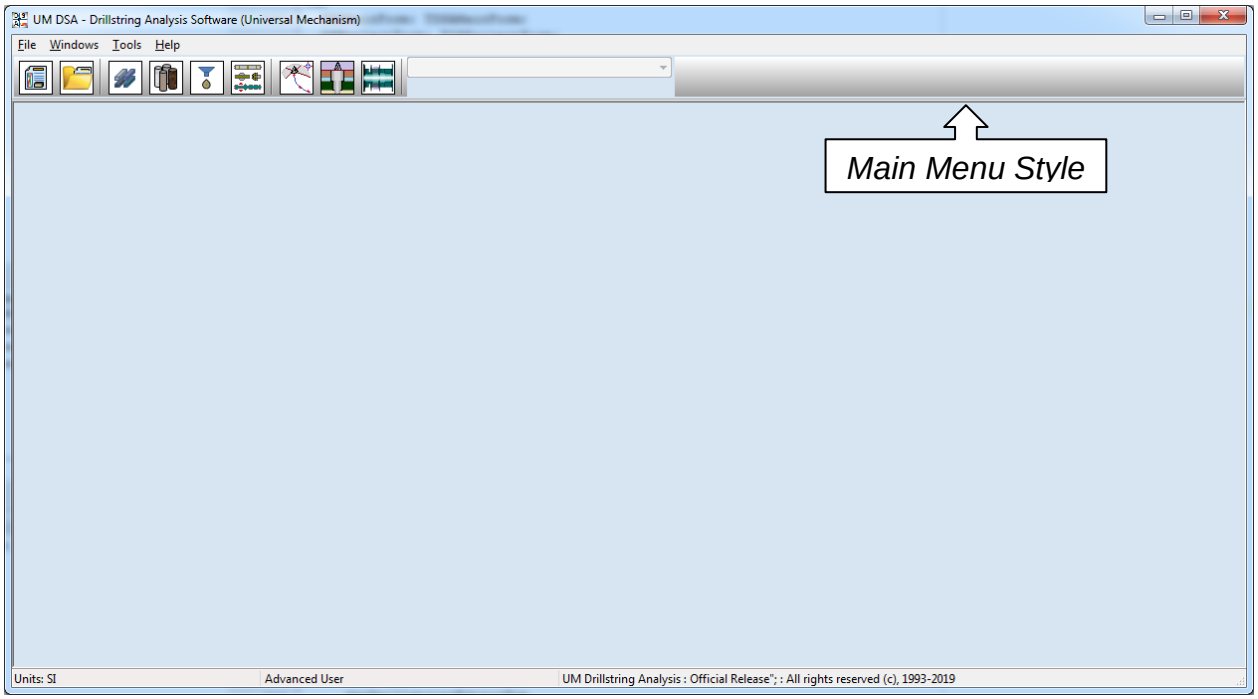
Schematics – Storage of string parts image files;
Wallpapers – Storage of main window wallpaper files;
Manual – PDF documentation files;
Utils,*Templates*,*ini*,*help*,*logos*,*umentities*,*x32*, etc. – Auxiliary file storage.

[1] Default storage means that the user can store these files anywhere he or she wants.

1.3.5. Interface Settings

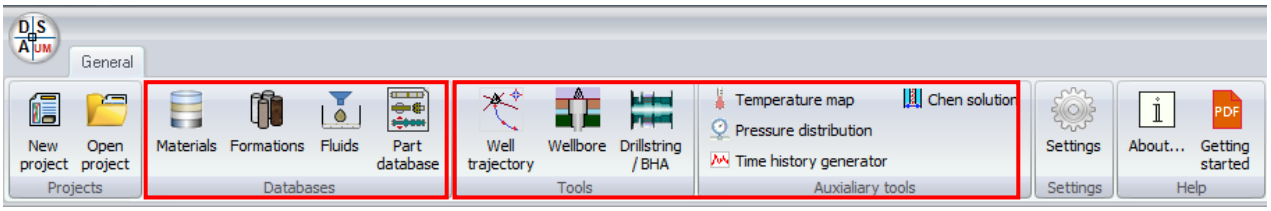
DSA Main window can be customized by selection of GUI style, and desktop wallpaper.





1.4. GUI Databases & Tools

DSA GUI includes a number of specialized databases and tools for work with source data without starting Drillstring Analysis project.



- Databases: Set of data used in drilling information description within Project or Drilling Information editors.
  - Material Database
  - Formation Database
  - Fluid Database
  - String Part Database

  The interface of databases is generalized within DSA GUI, and varies by the list of parameters only.

- Tools: Specific tools for drilling information items description without starting *Project*; one can use preliminary prepared data items for the fast *Project* initialization.
  - Well Trajectory Editor
  - Wellbore Editor
  - Drillstring/BHA Editor

  The interface of source data editors is similar to *Project* interface of data input.

- Auxiliary tools: Specific tools for scenario(s) input data items description without starting *Project*, as well as auxiliary analysis tools.
  - Temperature map – Temperature distribution along the well
  - Pressure distribution – Internal/external pressure for drillstring
  - Time history generator – Value vs. time dependencies generation for force excitations input.
  - Chen solution – Estimation of external mud damping for a pipe oscillating in a wellbore (in accordance to Chen solution).

  The interface of source data editors is similar to *Project* interface of data input.

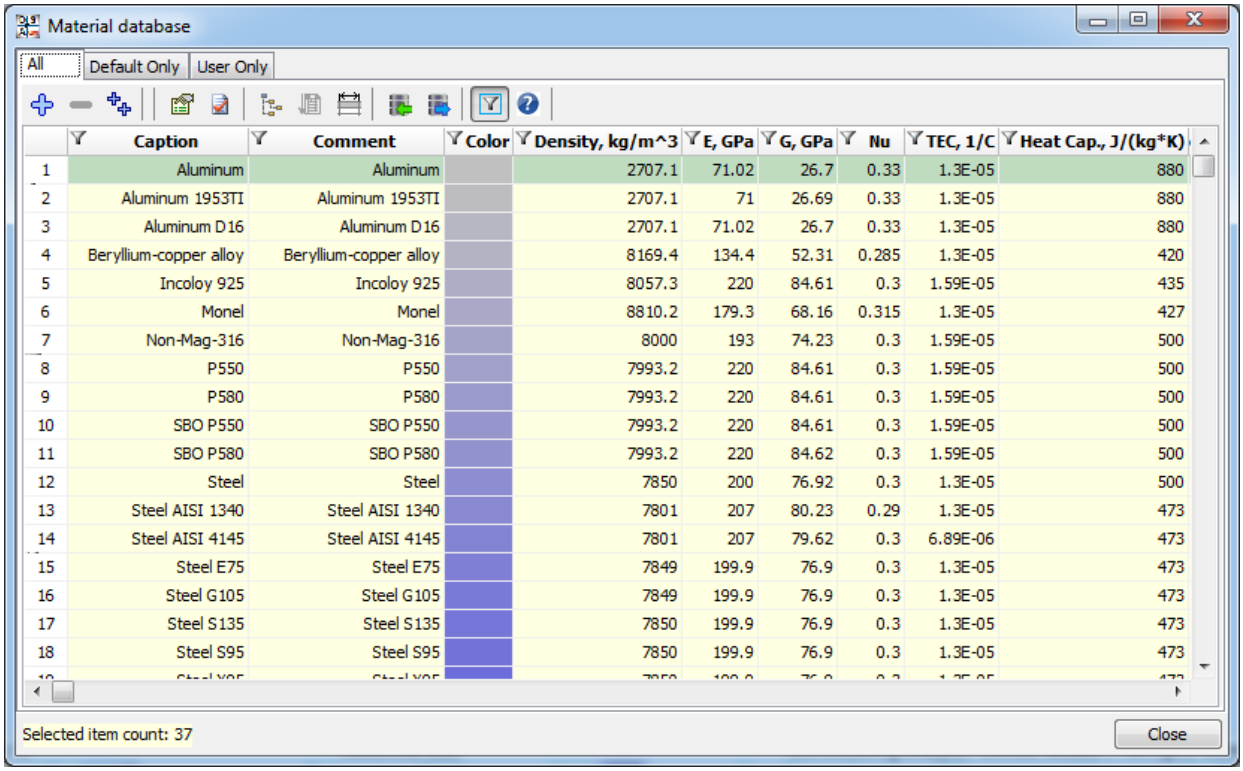
1.4.1. Material, formation, fluid databases

Database wizard GUI is common for material, formation and fluid data.

General view GUI

Each database contains the preset list of items, which parameters cannot be modified by the user[1]. One can add new item ( ) or duplicate ( ) one from the preset list to describe new item. User's items can be modified or deleted ( ).

GUI is organized as a table of database items vs. parameters. Use **All**, **Default Only** or **User Only** tabs above the table to display all items or preset or user-defined items correspondently.



	Caption	Comment	Color	Density, kg/m^3	E, GPa	G, GPa	Nu	TEC, 1/C	Heat Cap., J/(kg*K)
1	Aluminum	Aluminum		2707.1	71.02	26.7	0.33	1.3E-05	880
2	Aluminum 1953TI	Aluminum 1953TI		2707.1	71	26.69	0.33	1.3E-05	880
3	Aluminum D16	Aluminum D16		2707.1	71.02	26.7	0.33	1.3E-05	880
4	Beryllium-copper alloy	Beryllium-copper alloy		8169.4	134.4	52.31	0.285	1.3E-05	420
5	Incoloy 925	Incoloy 925		8057.3	220	84.61	0.3	1.59E-05	435
6	Monel	Monel		8810.2	179.3	68.16	0.315	1.3E-05	427
7	Non-Mag-316	Non-Mag-316		8000	193	74.23	0.3	1.59E-05	500
8	P550	P550		7993.2	220	84.61	0.3	1.59E-05	500
9	P580	P580		7993.2	220	84.61	0.3	1.59E-05	500
10	SBO P550	SBO P550		7993.2	220	84.61	0.3	1.59E-05	500
11	SBO P580	SBO P580		7993.2	220	84.62	0.3	1.59E-05	500
12	Steel	Steel		7850	200	76.92	0.3	1.3E-05	500
13	Steel AISI 1340	Steel AISI 1340		7801	207	80.23	0.29	1.3E-05	473
14	Steel AISI 4145	Steel AISI 4145		7801	207	79.62	0.3	6.89E-06	473
15	Steel E75	Steel E75		7849	199.9	76.9	0.3	1.3E-05	473
16	Steel G105	Steel G105		7849	199.9	76.9	0.3	1.3E-05	473
17	Steel S135	Steel S135		7850	199.9	76.9	0.3	1.3E-05	473
18	Steel S95	Steel S95		7850	199.9	76.9	0.3	1.3E-05	473

Select column with the parameter and click to sort items in the parameter value increasing/decreasing order. Use button to auto-size column headers; click to enable hint mode. Click to validate input data on admissible parameter values.

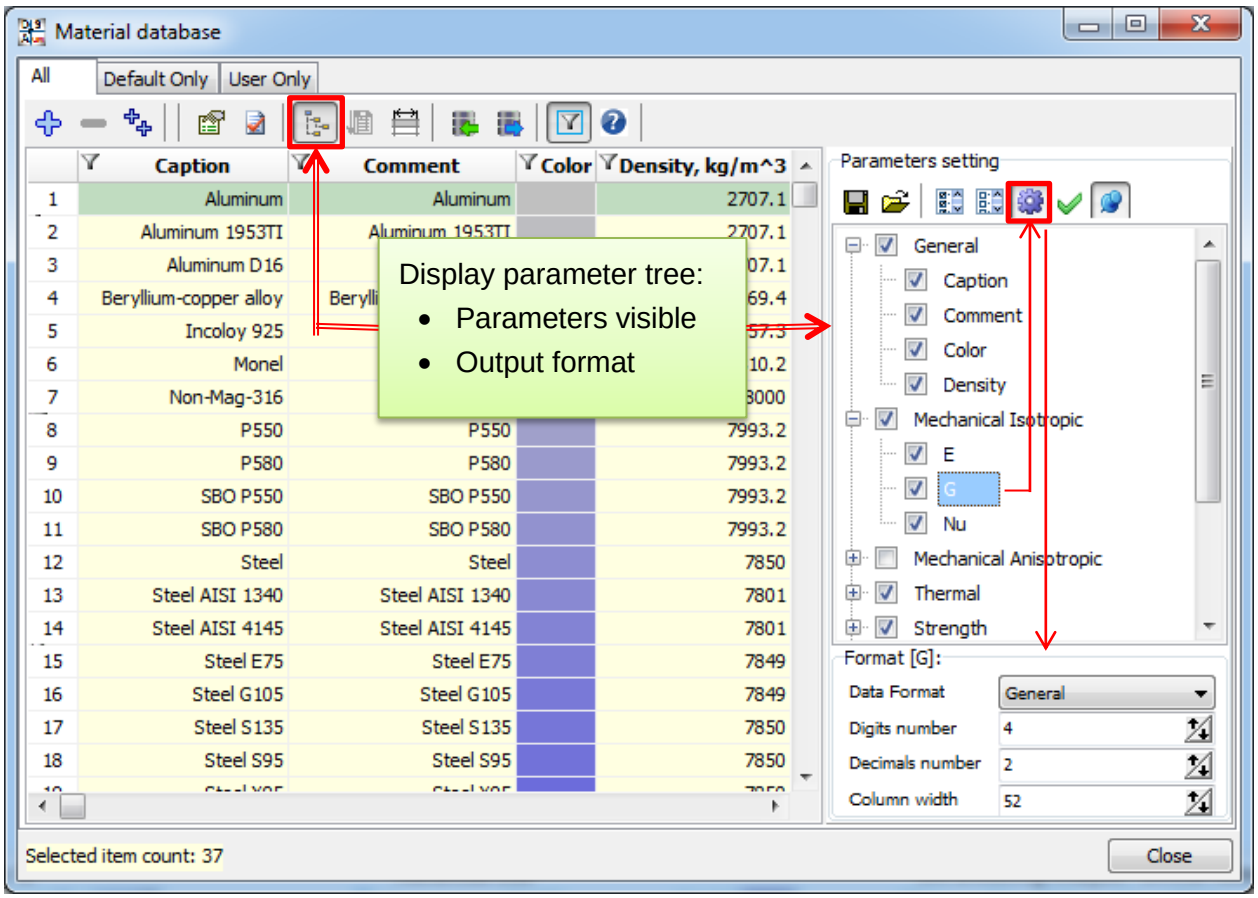
Use toolbar button to activate items filtering mode; click on a column head to turn on filtering by the column parameter.

Database info import ( ) and export ( ) of is available in **Advanced** and **Developer** mode.

[1] Modification is available in Developer mode only

GUI enables selection of parameters/columns to display in the table; this option is useful if number of item parameters is significant.

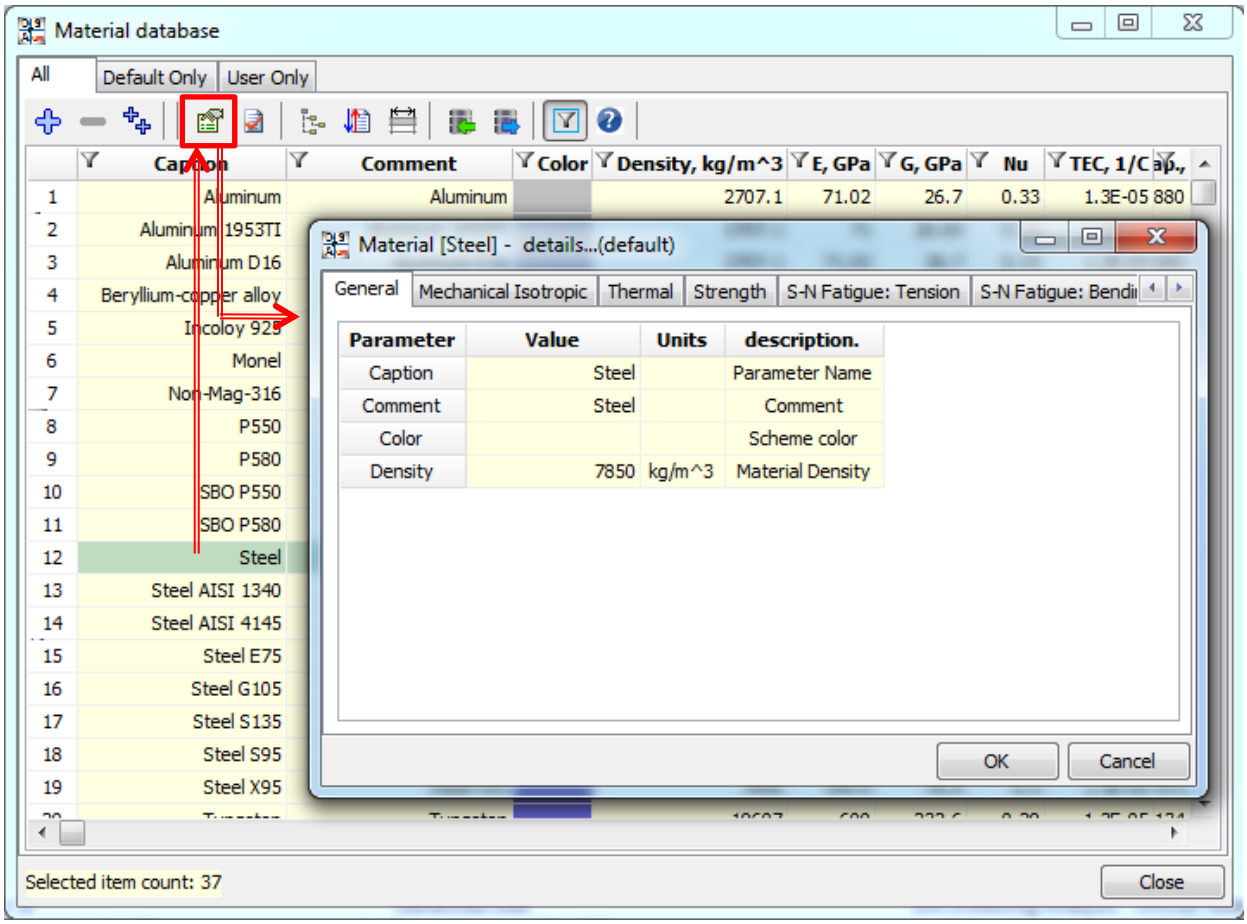
Click on button to display the **Category tree** panel, which provide an ability to hide/display parameters , and turn on output format of the parameters .



Parameters are grouped in categories; the user can hide/show a category at all by one click.

Item details view

Double click on the item in general view table, or select the item and click on button to see the item data in separate window.



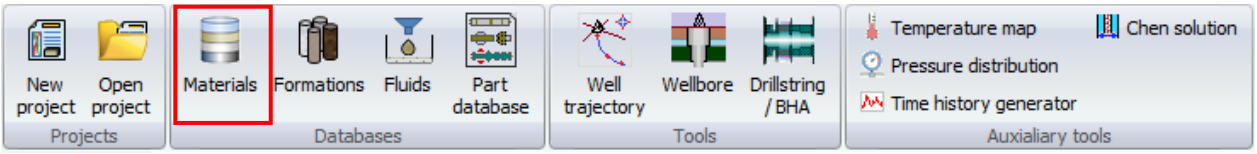
Click **Ok** to save changing to database; click **Cancel** to exit without saving.

Note: The *Item data view* mode enables modification of any item (preset or user-defined). If the user does not have access to preset data modification, the GUI will propose to save the item with another name as a user-defined item.

1.4.1.1. Material Database

Material database is stored in *MaterialDB.mtl* file placed in *Source directory\Databases* catalogue. The database is loading from the file when GUI starts.

Use **Materials** button of the head menu to open material database editor.



Material parameters

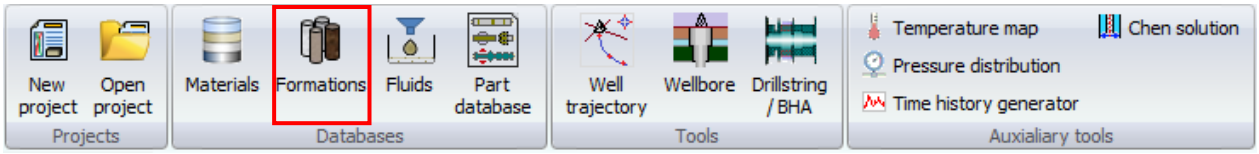
Database contains wide list of material parameters; list of parameters required for Static, T&D and Time Domain Analysis in DSA application is stated below.

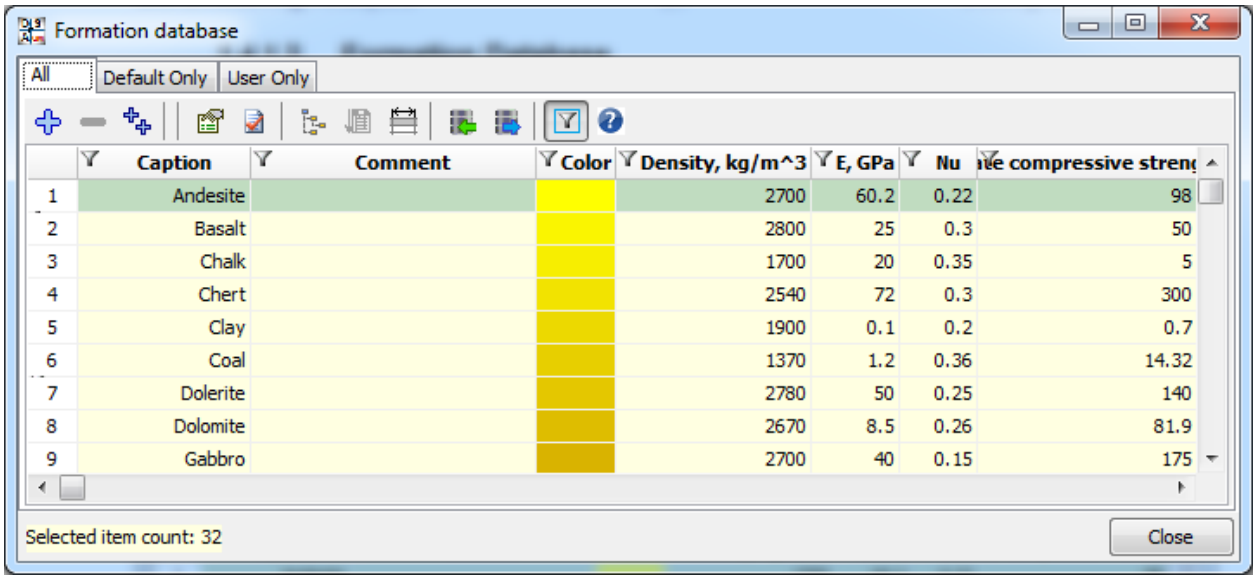
Parameter	Units		Description
	SI	Imperial	
General category			
Caption			Unique caption of the material within the database.
Color			Color the material is displayed on the drillstring side section view.
Density	Kg/m^3	Lbm/ft^3	Material density.
Mechanical Isotropic category			
Modulus of elasticity	GPa	ksi	Modulus of elasticity of the material.
Poisson's Ratio			The signed ratio of transverse strain to axial strain.
Thermal category			
TEC	1/C	1/F	Thermal expansion coefficient.

1.4.1.2. Formation Database

Formation database is stored in *FormationDB.fdb* file placed in *Source directory\Databases* catalogue. The database is loading from the file when GUI starts.

Use **Formations** button of the head menu to open formation database editor.





Formation parameters

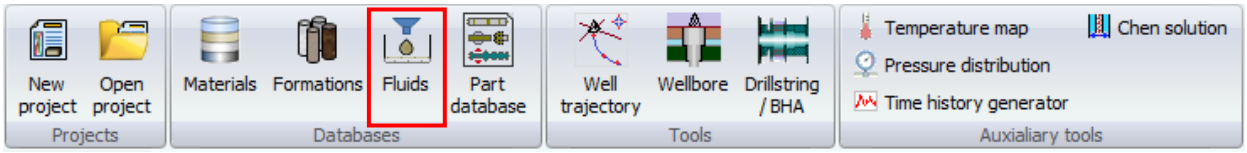
Database contains wide list of formation parameters; list of parameters required for Static, T&D and Time Domain Analysis in DSA application is stated below.

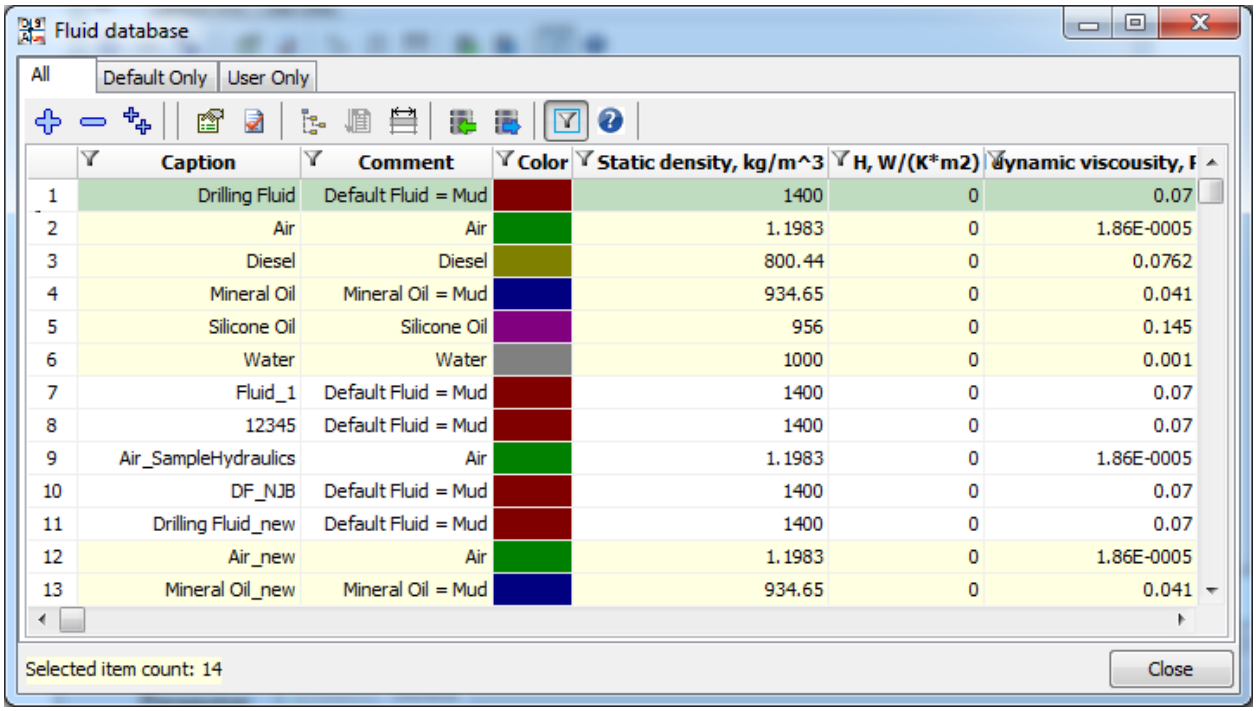
Parameter	Units		Description
	SI	Imperial	
General category			
Caption			Unique caption of the formation within the database.
Color			Color the formation is displayed on the wellbore section view.
Modulus of elasticity	MPa	psi	Modulus of elasticity of the formation.
Poisson's Ratio			The signed ratio of transverse strain to axial strain.
Density	Kg/m^3	Lbm/ft^3	Formation density.
RGD model parameters category			
Rock specific strength (epsilon)	MPa	psi	Rock specific strength parameter for RGD model of PDC bit-rock interaction
Bit-rock contact stress (sigma)	MPa	psi	Bit-rock contact stress parameter for RGD model of PDC bit-rock interaction

1.4.1.3. Fluid Database

Fluid database is stored in *FluidDB.ldb* file placed in *Source directory\Databases* catalogue. The database is loading from the file when GUI starts.

Use **Fluids** button of the head menu to open fluid database editor.





	Caption	Comment	Color	Static density, kg/m^3	H, W/(K*m2)	dynamic viscousity, F
1	Drilling Fluid	Default Fluid = Mud		1400	0	0.07
2	Air	Air		1.1983	0	1.86E-0005
3	Diesel	Diesel		800.44	0	0.0762
4	Mineral Oil	Mineral Oil = Mud		934.65	0	0.041
5	Silicone Oil	Silicone Oil		956	0	0.145
6	Water	Water		1000	0	0.001
7	Fluid_1	Default Fluid = Mud		1400	0	0.07
8	12345	Default Fluid = Mud		1400	0	0.07
9	Air_SampleHydraulics	Air		1.1983	0	1.86E-0005
10	DF_NJB	Default Fluid = Mud		1400	0	0.07
11	Drilling Fluid_new	Default Fluid = Mud		1400	0	0.07
12	Air_new	Air		1.1983	0	1.86E-0005
13	Mineral Oil_new	Mineral Oil = Mud		934.65	0	0.041

Fluid parameters

Database contains wide list of fluid parameters; list of parameters required for Static, T&D and Time Domain Analysis in DSA application is stated below.

Parameter	Units		Description
	SI	Imperial	
General category			
Caption			Unique caption of the fluid within the database.
Color			Color the fluid is displayed on the schemes.
Density	Kg/m^3	Lbm/ft^3	Fluid density.
Mechanical category			
Mud dynamic viscosity	Pa · sec	cP	Dynamic viscosity (or absolute viscosity) of the fluid: cP = 0.01P = 1 mPa·s = 0.001 Pa·s.

1.4.2. String Part Database

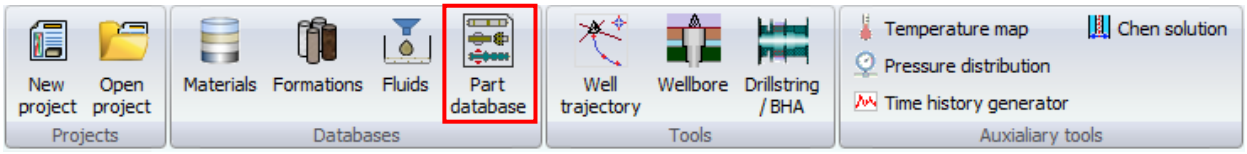
DSA GUI enables description of the bottom hole assembly and drillstring design with the set of components, that can be taken from *String Part Database* or described internally by *Drillstring Part Editor* tools.

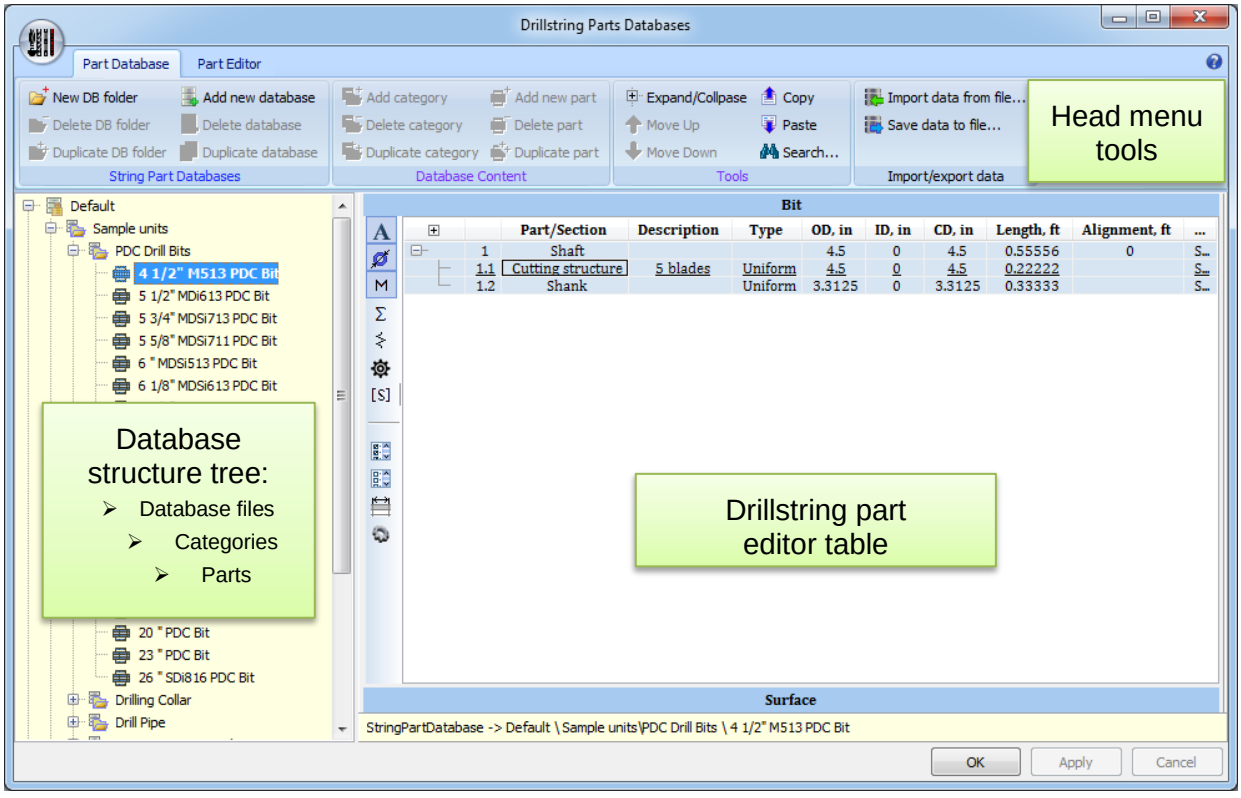
String Part Database GUI

The database is stored in **.pdb* file placed in *Source directory\Databases\StringPartDatabase* catalogue and its sub-catalogues. The database is loading from the files when GUI starts.

StringPartDatabase catalogue contains by default *Training database.pdb*, which items are read only; and *User database.pdb* available for editing.

Use **Part Database** button of the head menu to open database editor.





Database navigation is organized by database tree structure on the left panel. Database file is displayed in the database structure tree ( ), and contain a set of drillstring parts ( ) grouped into categories ( ).

Head menu tools and *Drillstring Part Editor* GUI, placed on the right panel, enables access to the database items description.

Database structure editing

Tools for the database data structure modification are available on the **Part Database** tab of the head menu:

 - add new database catalogue folder;
 - delete selected database catalogue folder with all entities;
 - duplicate selected database catalogue folder at the current level[1];

 - add new database file in the selected database catalogue folder;
 - delete selected database file with all entities;
 - duplicate selected database file at the current level[1];

 - add sub category for the selected category node;
 - delete selected category node with all entities;
 - duplicate selected category at the current level;

 - add new part to the current category;
 - delete selected part;
 - duplicate selected part;

 - collapse/expand tree branches;
 - move node up;
 - move node down;

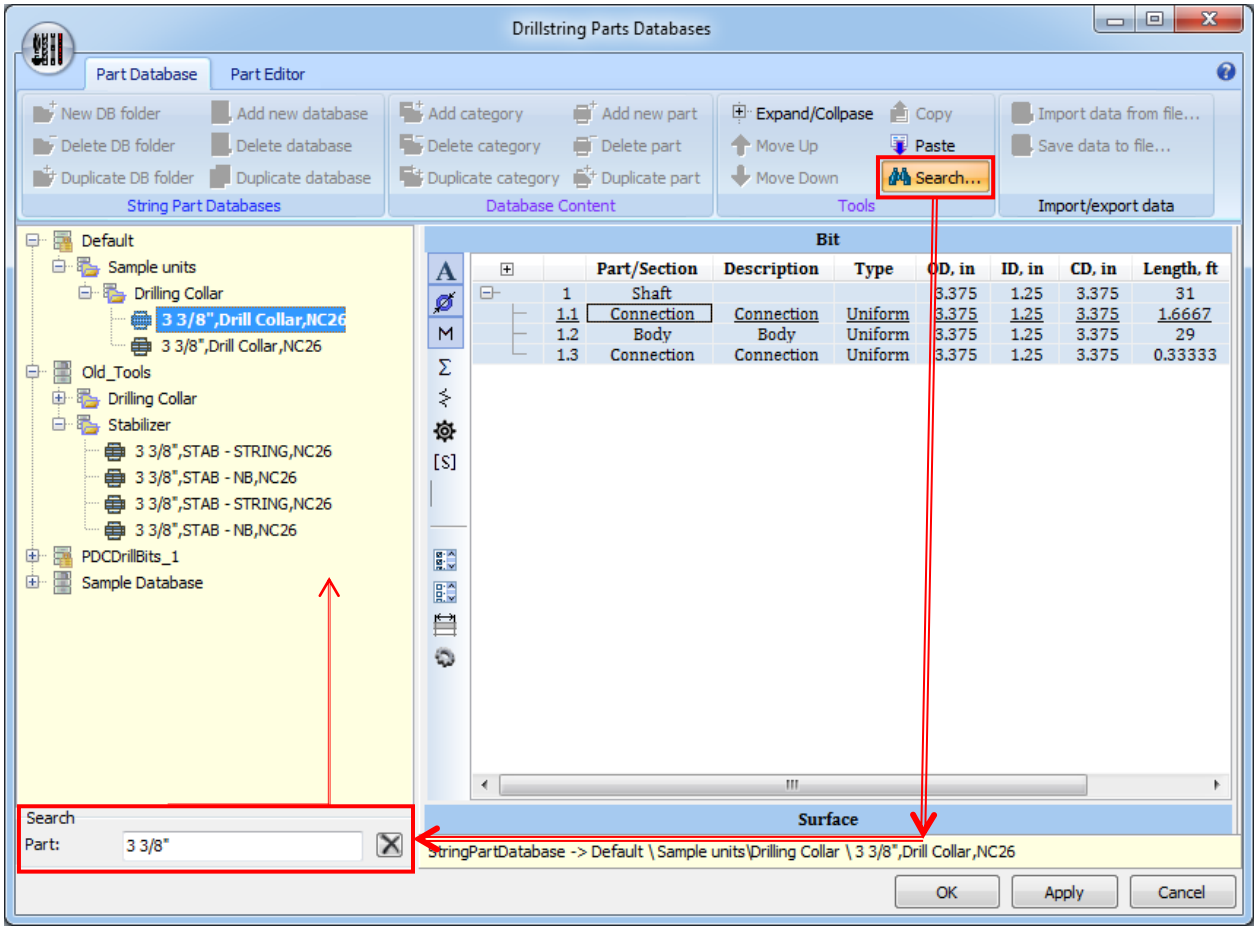
 - copy node;
 - paste copied node;
 - enable/disable *Search mode*:

 - import data from *.pdb file to the current category;
 - export data from the current category to *.pdb file.

[1] Available in **Developer** mode only

Search mode

The **Search** panel is adding to the bottom of the database tree panel if the *Search mode* is enabled. The only parts containing search mask, defined in the **Part Caption** field, are displayed in the database tree if the *Search mode* is enabled.

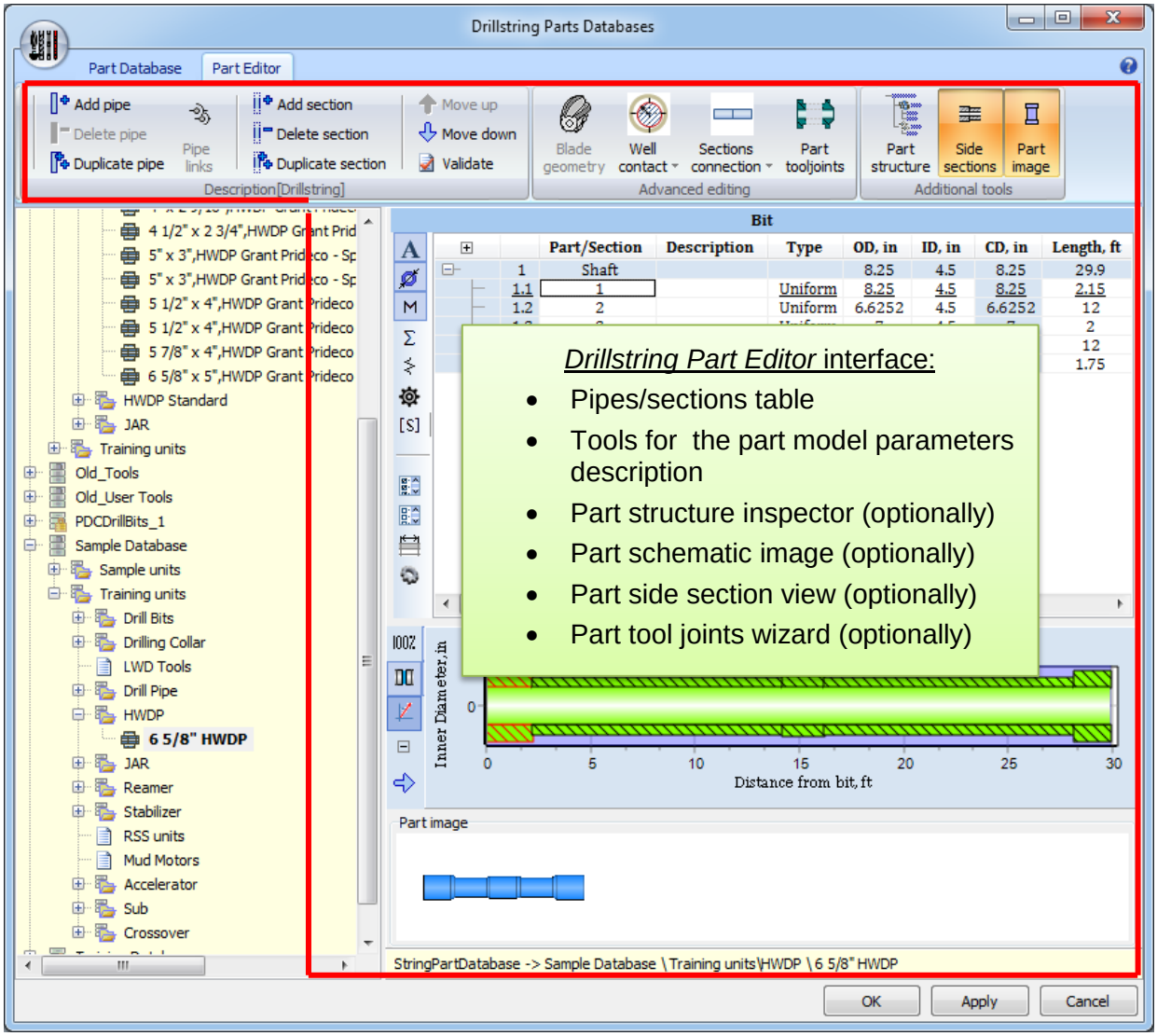


Part parameters output/editing

Parameters of the part selected in the *Database tree* are controlled by the *Part Editor GUI* - panel right to the tree panel; additional tools are placed on the **Part Editor** tab of the head menu.

1.4.2.1. Drillstring Part Editor

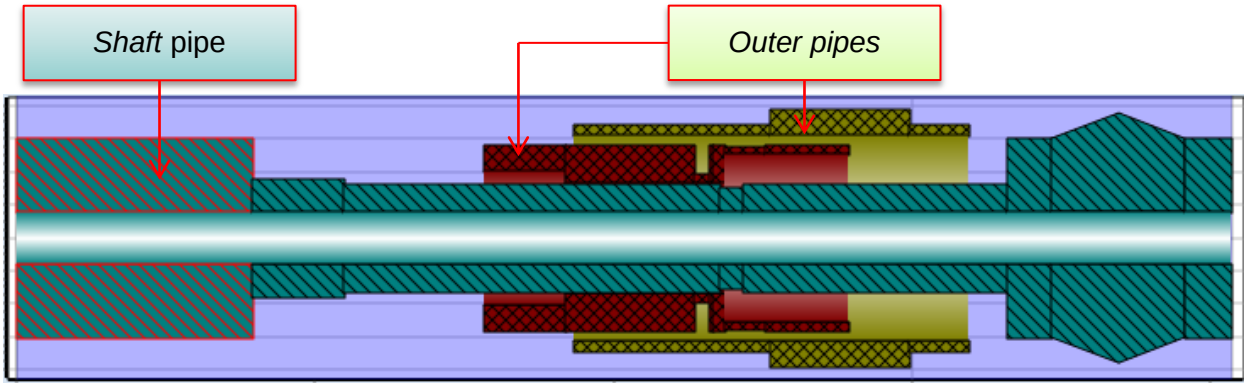
The *Drillstring Part Editor* GUI provide tools for description of the wide range of drilling tools within the generalized format: bits, drilling pipes, collars, stabilizers, mud motors, “place-the-bit” and “push-the-bit” steering systems, etc.



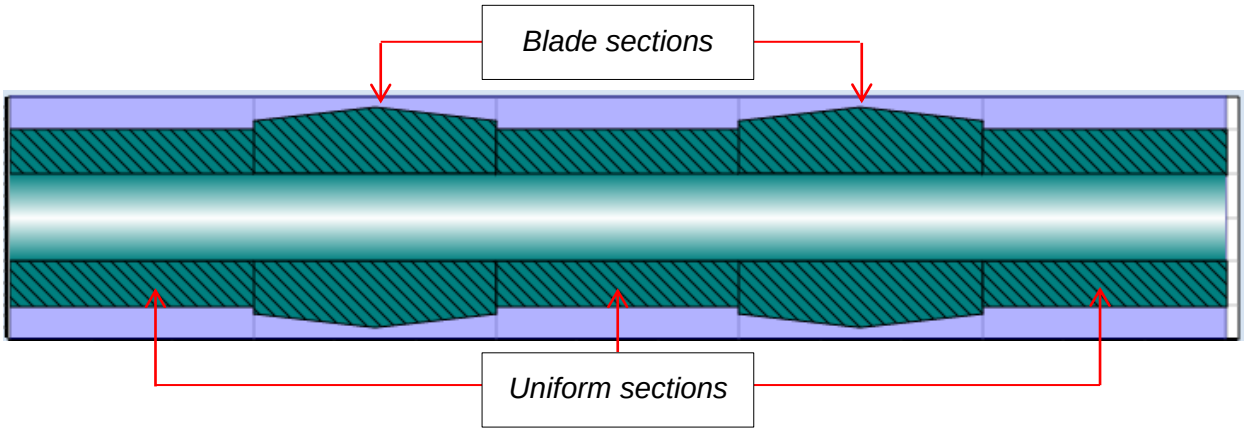
Note! Modification of part description is available for the not-read only database files only.

Part description concept

In accordance to the generalized description any part is presented by set of pipes; the first one (or the single for simple parts) is the *shaft pipe*, the rest ones are the *outer pipes*.



Each pipe consists of arbitrary set of *Uniform* and *Blade* sections. Each section can be presented by flexible beam element, or absolutely rigid body; 3D finite element model of non-uniform component can be incorporated in the description[1].



Part-wellbore interaction is defined by *circle-cylinder* and *points-cylinder* contact forces; contact circles and points are placed on the part, hole is modeled by cylinder.

Several kinds of sibling section connections are supported: tool joint, hinge, CV joint, bend sub, arbitrary bushing connection, mud motor pump. Several kinds of section ends - wellbore interaction are supported: none, contact, offset, pad, fixed node.

Any outer pipe should be linked directly or by other pipes to the shaft pipe; different types of bearings, bushings, hydraulic/mechanical positioning systems, mud motor interaction and bushings are available.

Detailed description of the drillstring part model in DSA environment is presented in Setc.x of the Manual.

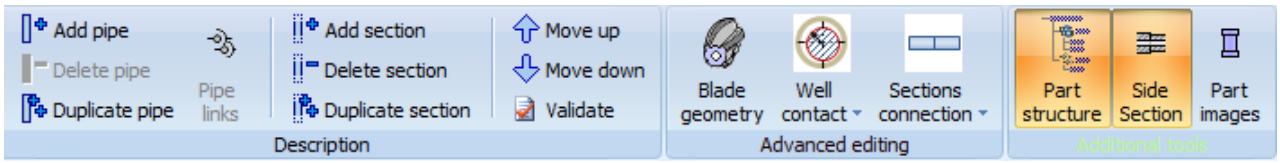
[1] The 3D FEM model usage functionality is not presented in the current documentation.

Part Editor GUI

Part Editor GUI consists of several elements:

➢ Head menu

Head menu contains basic and advanced tools for part structure description, as well as auxiliary visualization tool controls:

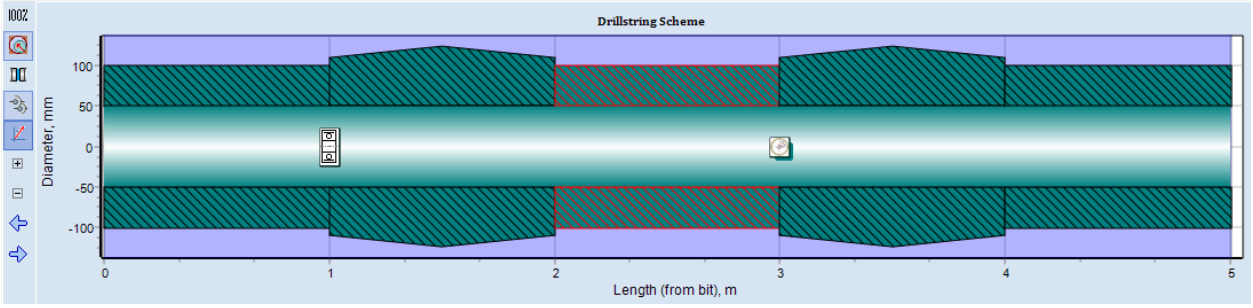


- add pipe;
- delete selected pipe (Note: *Shaft* pipe cannot be removed);
- duplicate selected pipe;
- display/hide pipe links editor[1];
- add new section for the pipe;
- delete selected section;
- duplicate selected section;
- move section close to the downhole end of part;
- move section far from the downhole end of part;
- validate part description.

Advanced editing tools enables detailed description of blade section geometry and model settings (**Blade geometry**), well-interaction model for the upper end point of section (**Well contact**) and parameters of connection with upper section (**Sections connection**).

One can use **Part structure** button to display/hide *Part structure tree* that can be useful for navigation in the description of the complex components.

Use **Side section** button to show the side section image of the part. The image is updating interactively during modification of the part description, and can be used for visualization of links, connections and nonstandard contacts marking.



[1] Description will be added in the next version of the Manual

➢ Parameter table

Table control is used for output and editing of pipes and section parameters; one can edit white-colored fields to modify the description.

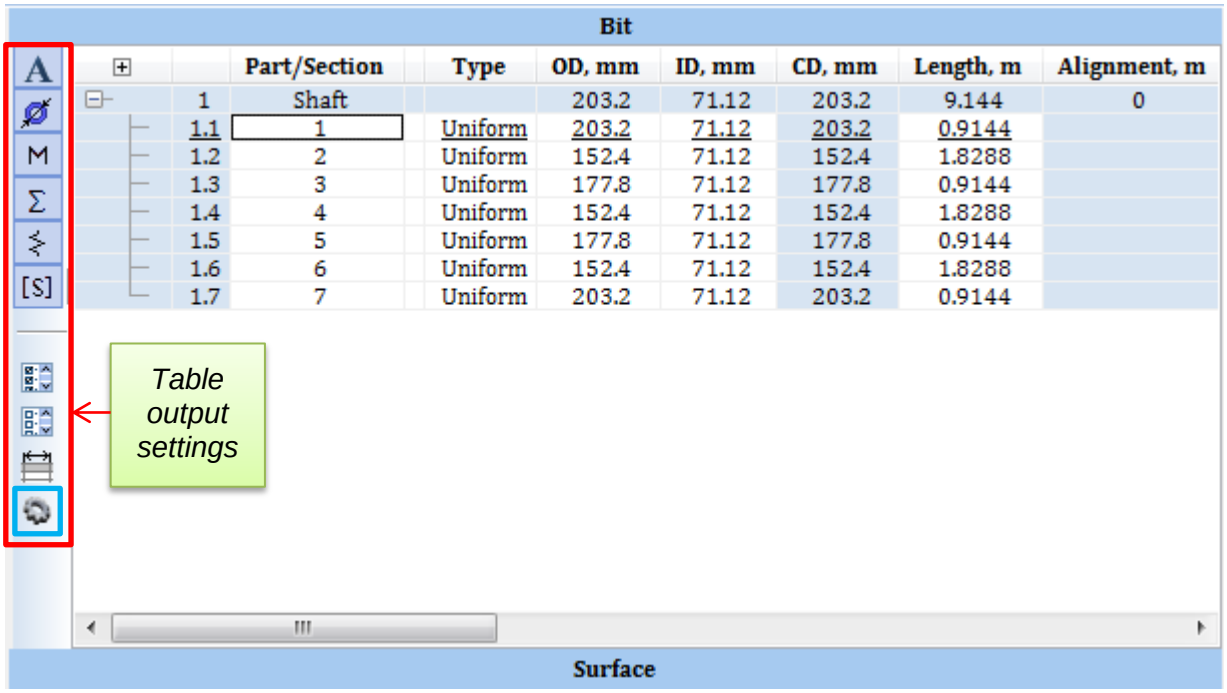
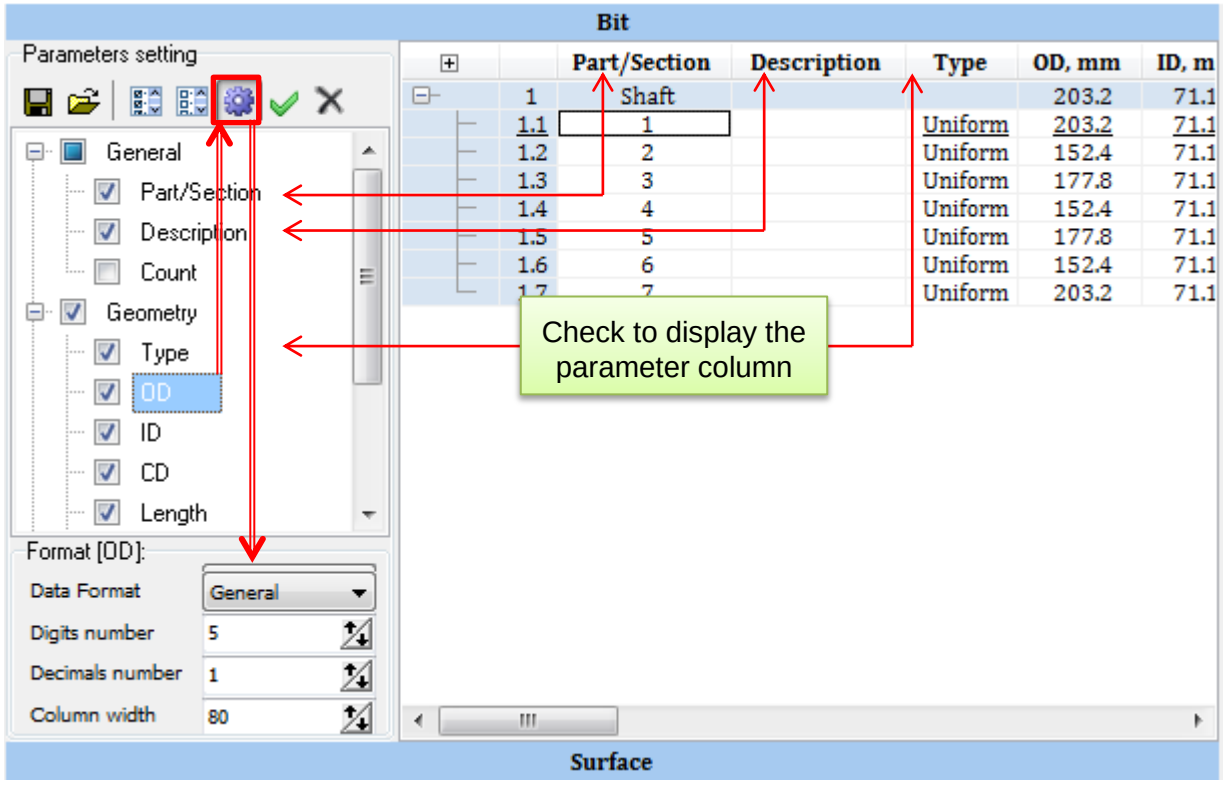


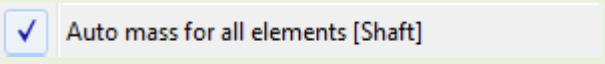
Table output can be customized by **Table settings** tools placed on the left vertical panel. GUI enables selection of parameters/columns to display in the table; click on **Table settings** button, or use the fast access buttons to select parameters for the table and turn on output format if necessary.



Parameters

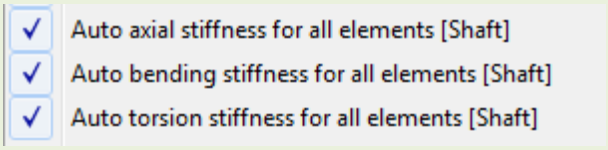
Part description contains wide list of parameters.

Parameter	Units		Description
	SI	Imperial	
General category - fast access button.			
Pipe/Section			Pipe / section caption; used for specifying point on the part in analyses.
Description			Pipe / section description.
Count			Part quantity. Not available in Part Editor; used in *Drillstring editor* only.
Geometry category - fast access button.			
Type			Section type: *Uniform* or *Blade*.
OD	mm	inch	Section: outer diameter of the pipe section. Pipe: maximal outer diameter of the pipe sections.
ID	mm	inch	Section: inner diameter of the pipe section. Pipe: minimal inner diameter of the pipe sections.
CD	mm	inch	Section: Contact diameter of the pipe section. Pipe: Maximal contact diameter of the pipe sections. **Note**: Contact diameter is equal to the outer diameter for *Uniform* sections and set automatically; contact diameter of *Blade* section is defined additionally, and can be equal or greater then outer diameter of blade section body.
Length	m	ft	Length of the section.
Alignment	m	ft	Parameter of multi-pipe parts description: distance from the lower end of *Shaft pipe* to the lower end of *Outer pipe*. **Note:** Alignment field is available for *Outer pipe* items only.

Inertia category - M fast access button.			
Material			Material of the section. If material is assigned for all pipe sections, the material caption is displayed in the pipe material field; the field is empty otherwise. The user can set material for the section, or for all pipe sections, or for all part sections by popup menu tools. **Note:** Material is selected from *Material Database*; one needs to edit the database if part material is not included in the database.
Lin. mass	kg/m	lbm/ft	Linear mass of the section/pipe. Linear mass of section is calculated automatically from material density and cross section geometry. Pipe linear mass is taken from the corresponded mass value divided by the length value.
Mass	kg	lbm	Mass of the section/pipe. Mass of section is calculated automatically from material density and section geometry. Pipe mass is taken from the masses of the sections. **Note:** *Linear mass* and *Mass* of section/pipe value can be set manually in **Advanced User** and **Developer** modes. One needs to select the corresponded row in the table and disable **Auto mass…** option by pop up menu. 
Summary category - Σ fast access button.			
Acc. Length	m	ft	Accumulated length reference value (read only): • for section: distance from lower point of the pipe to the top point of section. • for pipe: distance from lower point of the pipe to the top point of pipe = pipe length. **Note:** Value is available for the *Shaft pipe* only.
Acc. Mass	kg	lbm	Accumulated mass reference value (read only): • for section: mass of all sections from lower end of pipe to the current section inclusively. • for pipe: mass of all pipe sections. **Note:** Value is available for the *Shaft pipe* only.

<table>
<tr><td colspan="4">Stiffness category - fast access button.</td></tr>
<tr><td>Model type</td><td></td><td></td><td>Pipe model type:
Flexible – pipe sections are simulated by flexible uniform beams;
Rigid – pipe is modeled with absolutely rigid body of the prescribed geometry;
3D FEM model – pipe model is described by 3D finite element model (*.fss file), imported from FEM software in accordance to the modal finite element approach.</td></tr>
<tr><td>Lin. Axial. Stiff</td><td>kPa</td><td>ksi</td><td rowspan="3">Linear axial/bending/torsional stiffness of the section/pipe.

Linear axial stiffness of section is calculated automatically from material characteristics and cross section geometry.
Pipe axial stiffness is taken from the stiffness and geometry of the sections.

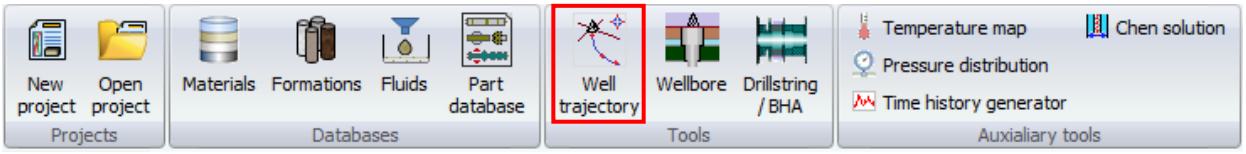
Note: Linear axial/bending/torsional stiffness of section/pipe value can be set manually in Advanced User and Developer modes. One needs to select the corresponded row in the table and disable Auto axial/bending/torsional stiffness… option by pop up menu.
  </td></tr>
<tr><td>Lin. Bend. Stiff</td><td>kN/m^2</td><td>kip/ft^2</td></tr>
<tr><td>Lin. Tors. Stiff</td><td>kN/deg</td><td>kip/ deg</td></tr>
<tr><td colspan="4">Safety category - [S] fast access button.</td></tr>
<tr><td>Max. T</td><td>kN*m</td><td>kip*ft</td><td rowspan="2">Value of the maximal admissible torque/force can be applied to the section/pipe.

Note: Parameters are available for the Shaft pipe only.

Note: Parameters are not used in the current version of the DSA software.</td></tr>
<tr><td>Max. F</td><td>kN</td><td>kip</td></tr>
</table>

1.4.3. Well Trajectory Editor

Use **Well Trajectory Editor** button of the head menu to generate or edit file of survey description.

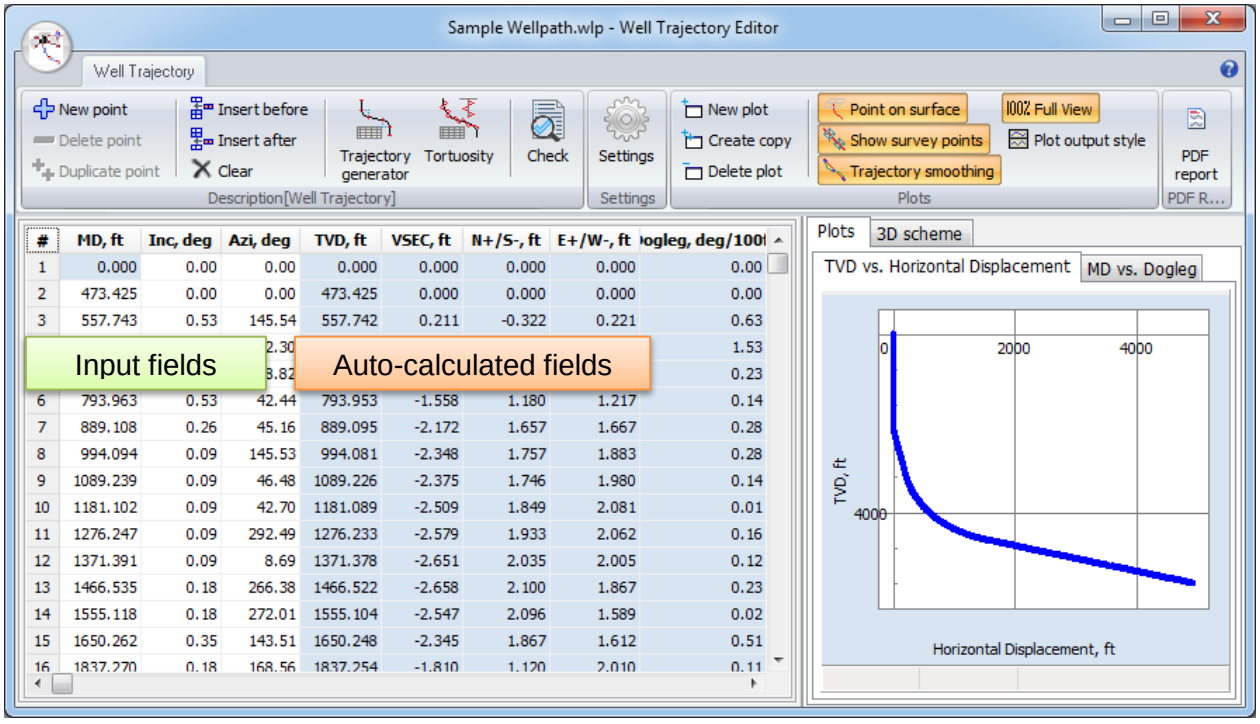


Well survey is described by set of survey points; *tortuosity* is described additionally and added to the basic survey if enabled.

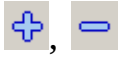
Trajectory can be saved in or loaded from xml file of special format (*Wellpath file *.wlp*). The files are saved and loaded from *Source directory\wellpaths* folder by default. The user can also **Paste** data from Excel table.

Survey station description

The GUI enables description of survey by point-by-point table input, or by set of constant build/turn rate intervals defined from surface or from the selected measuring depth.



#	MD, ft	Inc, deg	Azi, deg	TVD, ft	VSEC, ft	N+/S-, ft	E+/W-, ft	ogleg, deg/100
1	0.000	0.00	0.00	0.000	0.000	0.000	0.000	0.00
2	473.425	0.00	0.00	473.425	0.000	0.000	0.000	0.00
3	557.743	0.53	145.54	557.742	0.211	-0.322	0.221	0.63
4	2.30							1.53
5	3.82							0.23
6	793.963	0.53	42.44	793.953	-1.558	1.180	1.217	0.14
7	889.108	0.26	45.16	889.095	-2.172	1.657	1.667	0.28
8	994.094	0.09	145.53	994.081	-2.348	1.757	1.883	0.28
9	1089.239	0.09	46.48	1089.226	-2.375	1.746	1.980	0.14
10	1181.102	0.09	42.70	1181.089	-2.509	1.849	2.081	0.01
11	1276.247	0.09	292.49	1276.233	-2.579	1.933	2.062	0.16
12	1371.391	0.09	8.69	1371.378	-2.651	2.035	2.005	0.12
13	1466.535	0.18	266.38	1466.522	-2.658	2.100	1.867	0.23
14	1555.118	0.18	272.01	1555.104	-2.547	2.096	1.589	0.02
15	1650.262	0.35	143.51	1650.248	-2.345	1.867	1.612	0.51
16	1837.270	0.18	168.56	1837.254	-1.810	1.120	2.010	0.11


,
and 
buttons enable adding new, delete and copying survey points presented by lines in **Well Trajectory Editor** table.

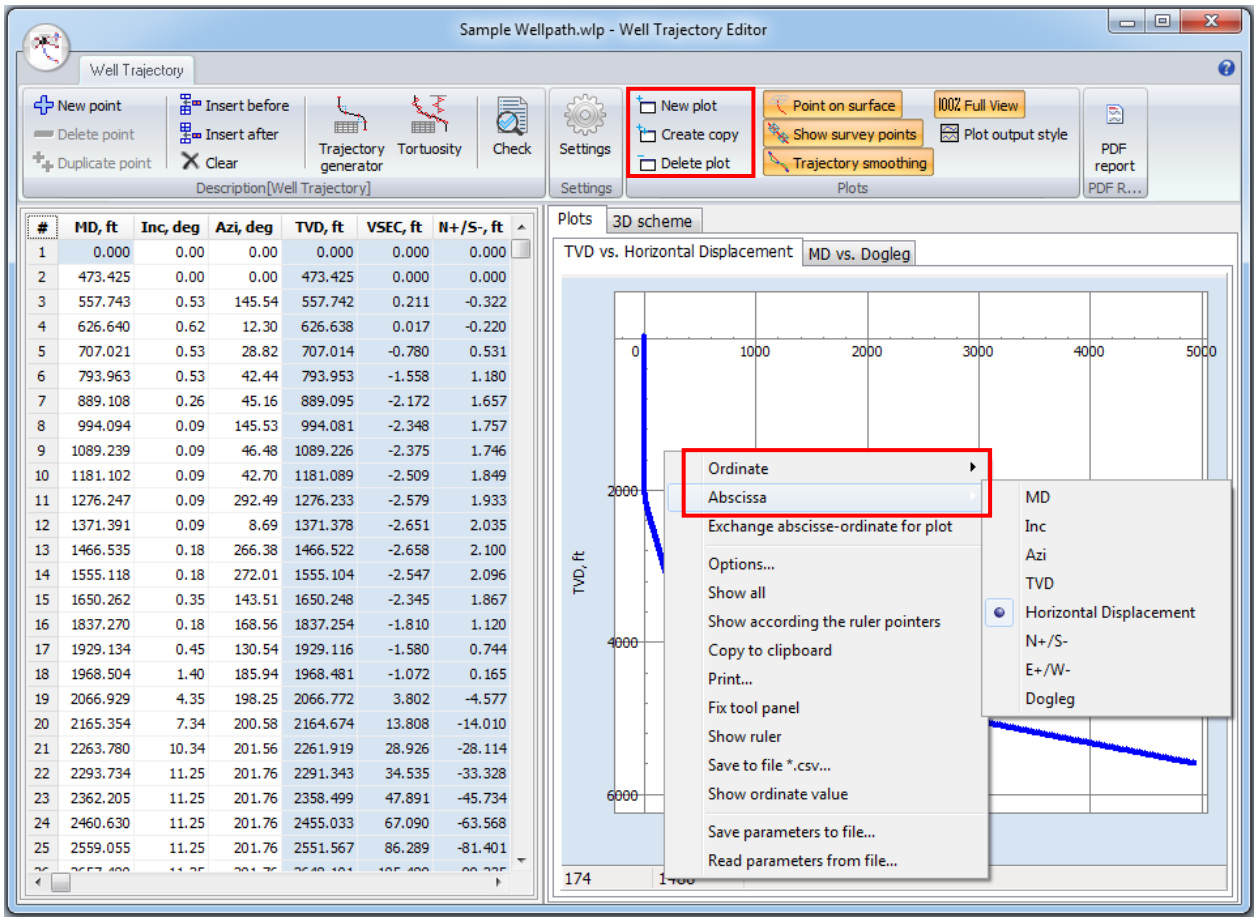
Trajectory is described by editing of **MD**, **Inclination** and **Azimuth** values in the lines of the table.

TVD, *VSEC*, *N+/S-*, *E+/W-* and *Dogleg* parameters are evaluated automatically from the input data. Editable fields of the table have white coloring, auto-calculated ones are marked with grey.

Input trajectory is visualized automatically by 2D plots and 3D view placed right to the input table:

➢ 2D plots

All the table parameters can be added on 2D plots; one can add (), delete () or duplicate () plot to the right panel, and customize the parameters on horizontal and vertical axes by pop up menu controls.

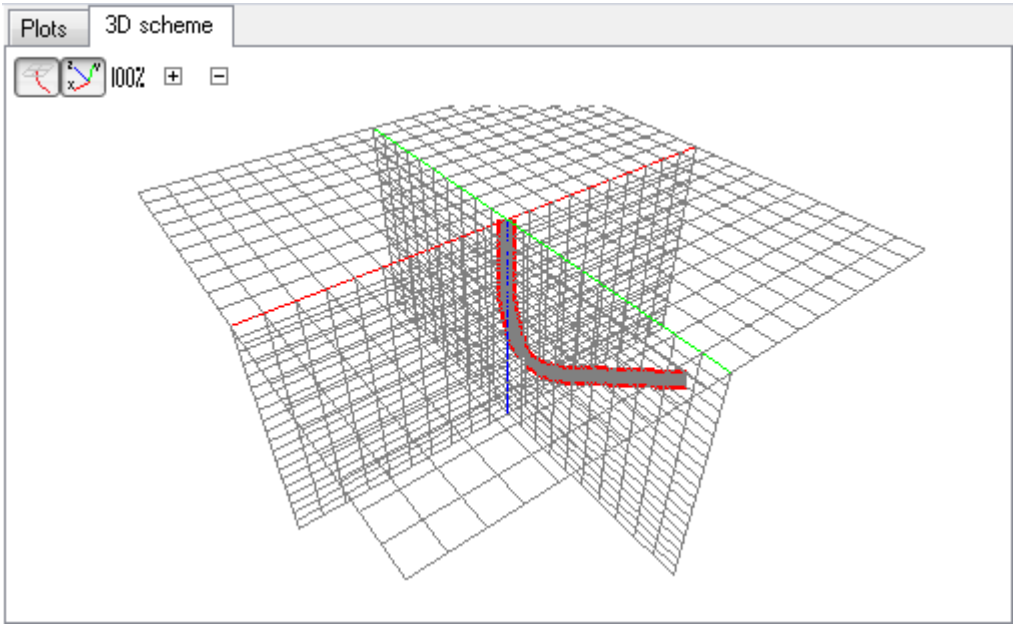


GUI enables the following plot output format settings:

- add surface survey point on plots (if not defined in the table);
- highlight survey point on plots;
- smooth plots by adding intermediate points between survey stations in accordance to minimal curvature equations;
- set full view for all plots;
- switch between 2D plot display modes: one plot for tab, or several plots on single panel.

➢ 3D view

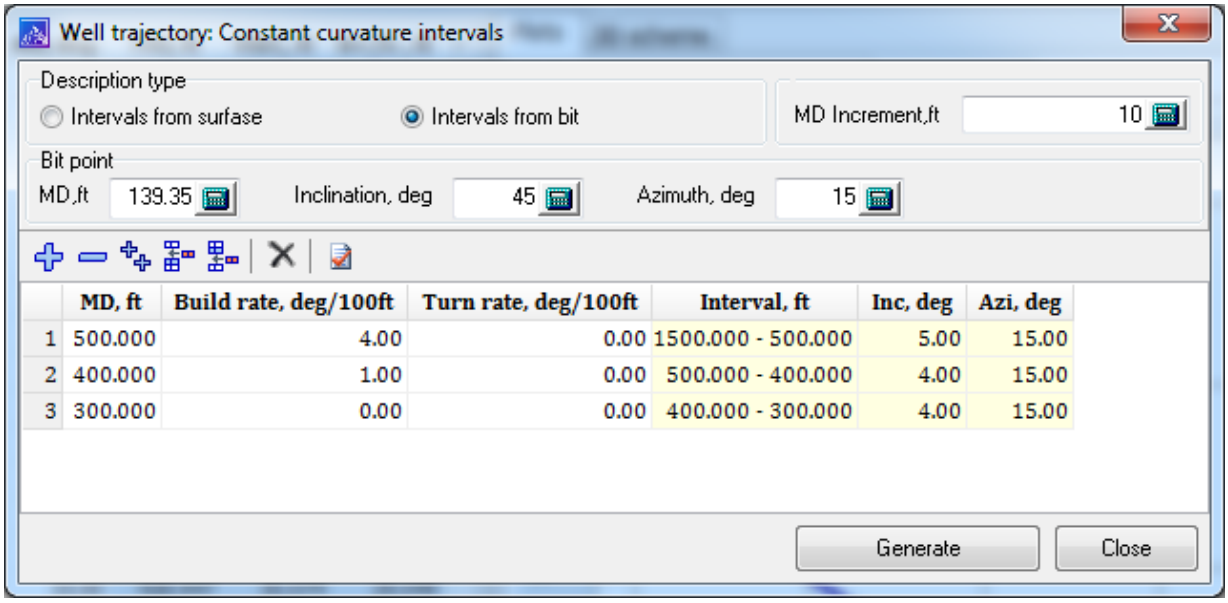
Spatial well trajectory view is available on **3D scheme** page of the right panel.



Trajectory intervals generation

The GUI enables generation of survey stations from the set of Turn Rate / Build Rate intervals defined from the end point or from the surface point of the well trajectory (end point).

One can use **Trajectory generator** wizard to define **Bit Point** measured depth, inclination and azimuth, step of generating survey stations (**MD Increment**) and list of intervals defined by end point measured depth, build rate and turn rate.

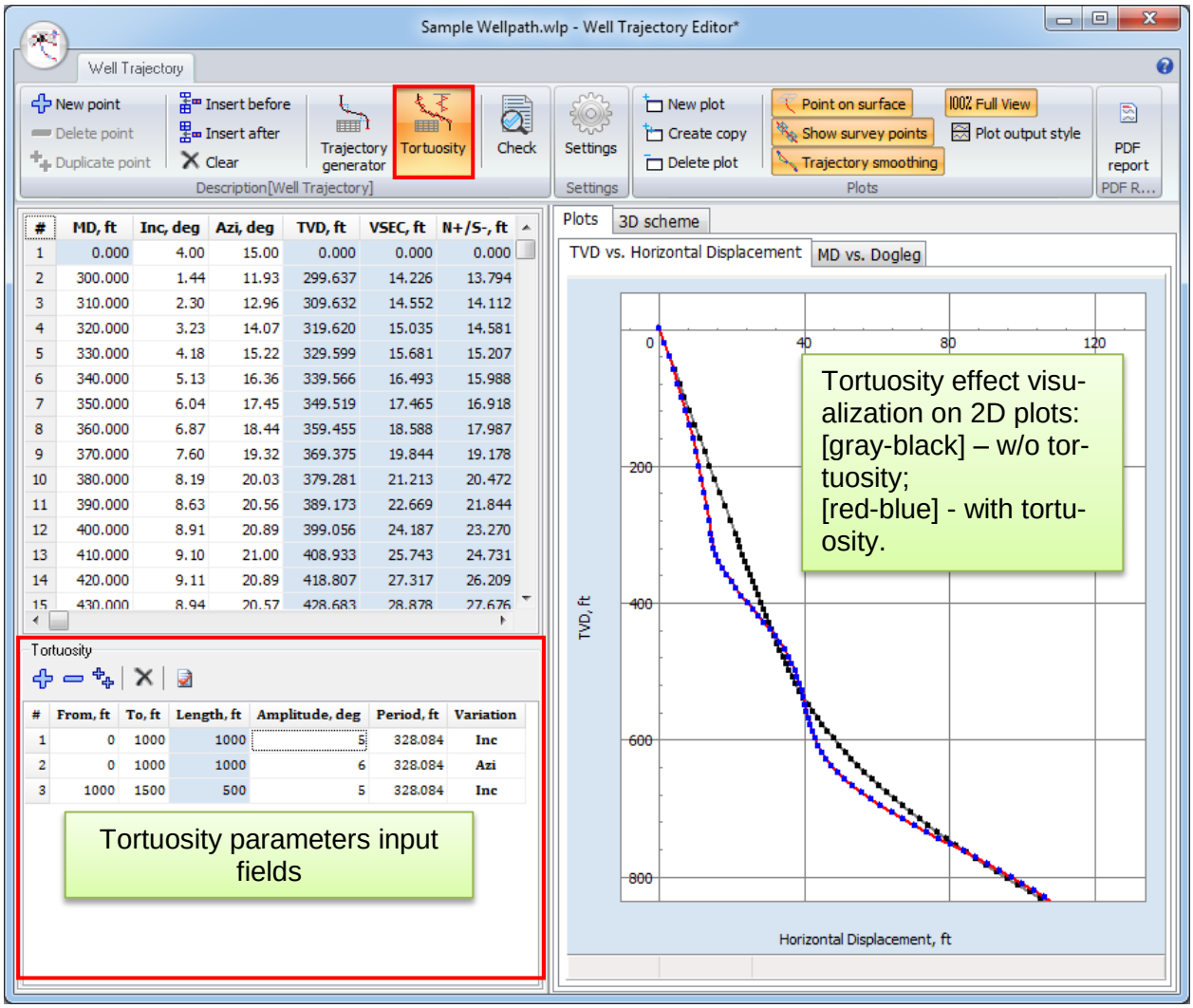


Click **Generate** button to create survey stations in accordance to the input data.

Well tortuosity description

GUI enables description of well trajectory as the combination of basic survey points defined in the table, and tortuosity intervals – intervals of harmonic variation of inclination and/or azimuth along the trajectory.

Click **Tortuosity** button on the head menu to take into account tortuosity intervals, which can be defined on the bottom table:



#	MD, ft	Inc, deg	Azi, deg	TVD, ft	VSEC, ft	N+/S-, ft
1	0.000	4.00	15.00	0.000	0.000	0.000
2	300.000	1.44	11.93	299.637	14.226	13.794
3	310.000	2.30	12.96	309.632	14.552	14.112
4	320.000	3.23	14.07	319.620	15.035	14.581
5	330.000	4.18	15.22	329.599	15.681	15.207
6	340.000	5.13	16.36	339.566	16.493	15.988
7	350.000	6.04	17.45	349.519	17.465	16.918
8	360.000	6.87	18.44	359.455	18.588	17.987
9	370.000	7.60	19.32	369.375	19.844	19.178
10	380.000	8.19	20.03	379.281	21.213	20.472
11	390.000	8.63	20.56	389.173	22.669	21.844
12	400.000	8.91	20.89	399.056	24.187	23.270
13	410.000	9.10	21.00	408.933	25.743	24.731
14	420.000	9.11	20.89	418.807	27.317	26.209
15	430.000	8.94	20.57	428.683	28.878	27.676

#	From, ft	To, ft	Length, ft	Amplitude, deg	Period, ft	Variation
1	0	1000	1000	5	328.084	Inc
2	0	1000	1000	6	328.084	Azi
3	1000	1500	500	5	328.084	Inc

Data verification

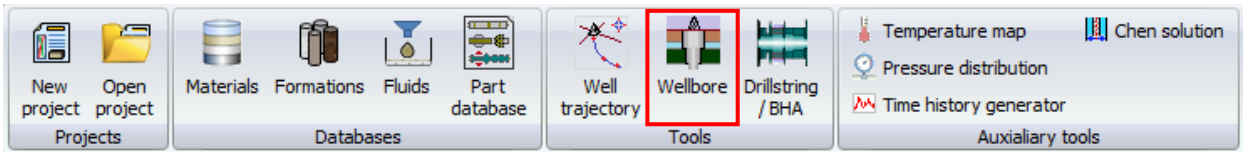
Well trajectory description can be verified with the **Verify Data** button placed on the top panel of the **Well Trajectory Editor** window.

Data reporting

Well trajectory description can be reported as PDF document with the button placed on the head menu of the editor window.

1.4.4. Wellbore Editor

Use **Wellbore Editor** button of the head menu to generate or edit file of wellbore construction description.

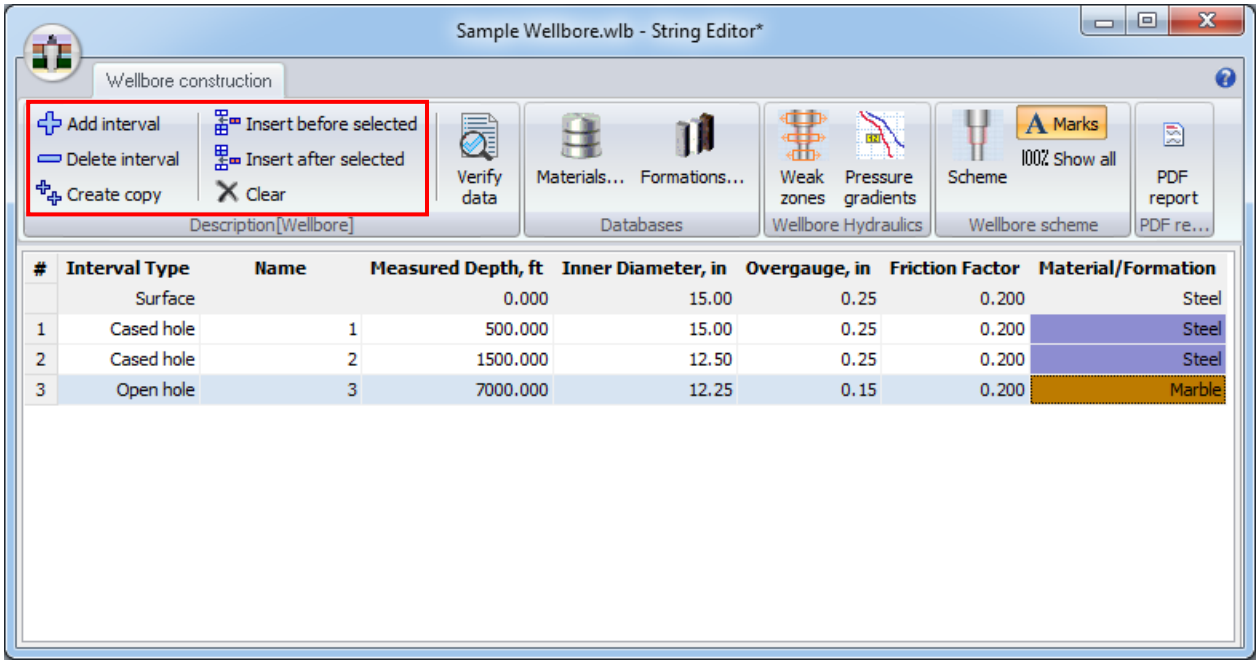


Wellbore construction is described by set of intervals defined by inner diameter vs. measured depth; hydraulics parameters of the wellbore can be defined optionally.

Wellbore data can be saved in or load from xml file of special format (*Wellbore file *.wlb*). The files are saved and loaded from *Source directory\wellbores* folder by default. The user can also **Paste** data from Excel table.

Wellbore construction description

, and buttons enable adding new, delete and copying wellbore intervals presented by lines in **Wellbore Intervals Editor** table.

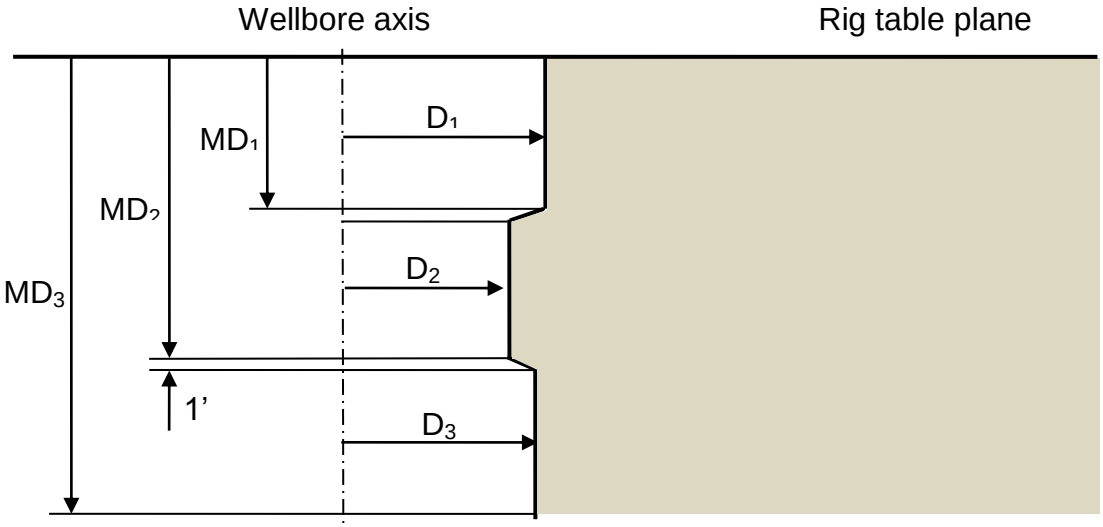


#	Interval Type	Name	Measured Depth, ft	Inner Diameter, in	Overgauge, in	Friction Factor	Material/Formation
	Surface		0.000	15.00	0.25	0.200	Steel
1	Cased hole	1	500.000	15.00	0.25	0.200	Steel
2	Cased hole	2	1500.000	12.50	0.25	0.200	Steel
3	Open hole	3	7000.000	12.25	0.15	0.200	Marble

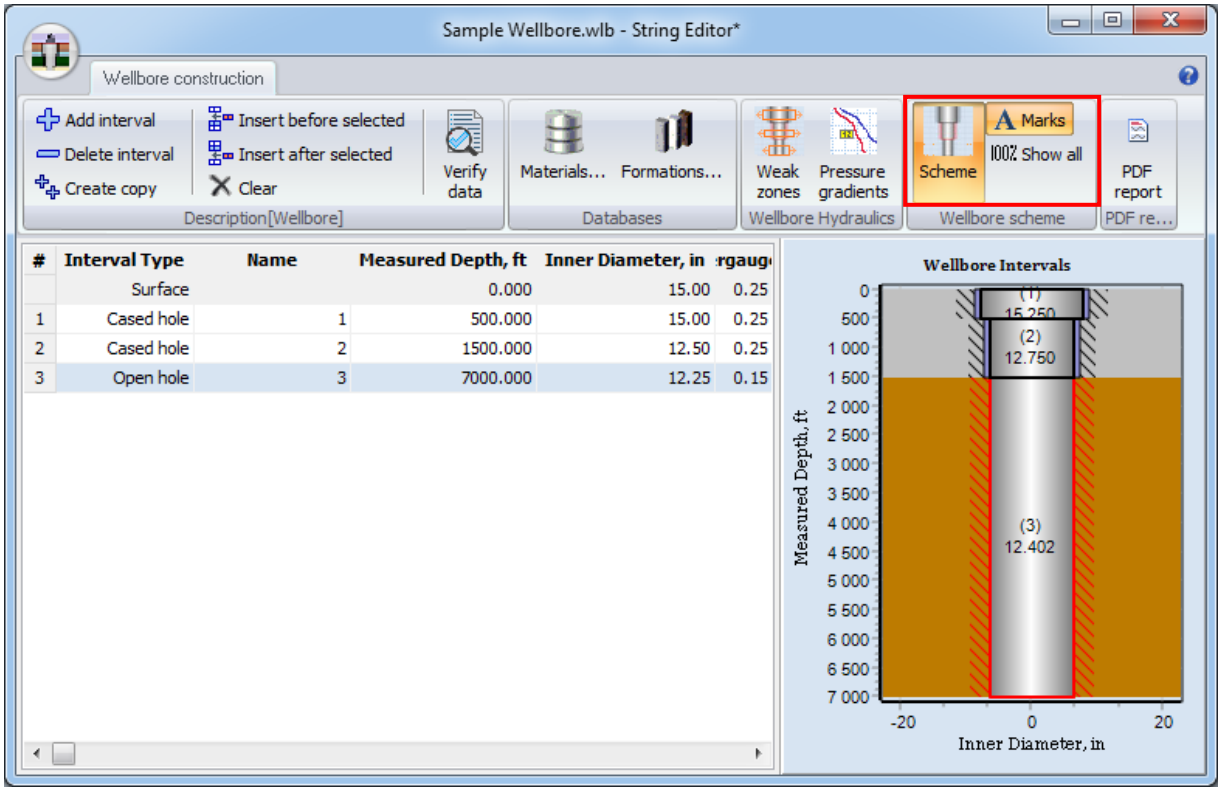
Wellbore interval is defined with *Measured Depth* of the end point, interval type (*Open hole* or *Cased hole*), hole diameter calculated from basic *Inner diameter* value and *Clearance* adding, and drillstring-wellbore walls *Friction factor*. *Material* is specified for *Cased* interval; *Formation* – for *Open hole* section.

Parameters of the intervals can be specified by editing of the values in the table lines.

Diameter of the hole for analysis is calculated by summation of the *Inner diameter* and *Clearance* values. One feet transient sections are used for smoothing of hole diameter between sibling intervals, see the scheme on figure below.



Input data is visualized automatically on the scheme placed right to the table.



Data verification

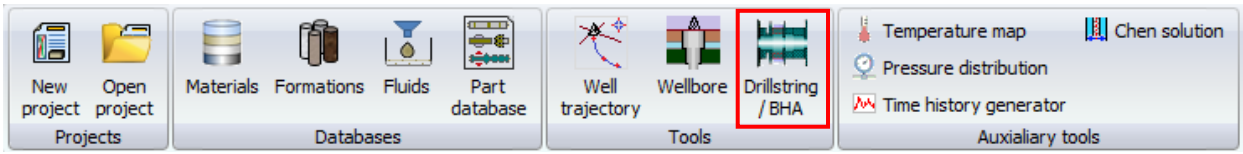
Wellbore description can be verified with the **Verify Data** button placed on the top panel of the **Wellbore Intervals Editor** window.

Data reporting

Wellbore description can be reported as PDF document with the button placed on the head menu of the editor window.

1.4.5. Drillstring / BHA Editor

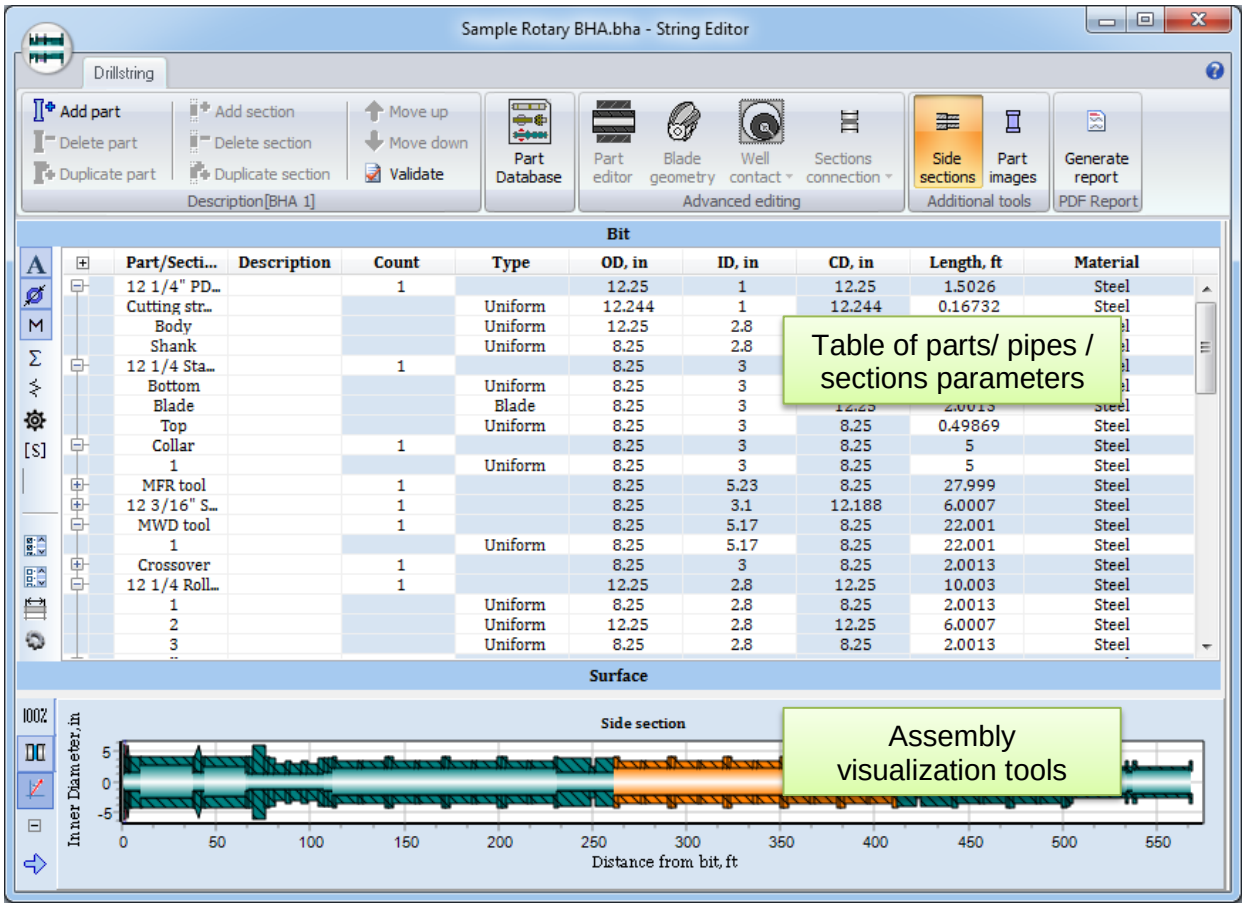
Use **Drillstring/BHA** button of the head menu to generate or edit file of drillstring or assembly description without starting new project.



Drillstring and bottom hole assembly design is described as the set of drillstring parts in "from bit to surface" order.

Division between drillstring and BHA description in DSA GUI is conventional; real drillstring is normally described in DSA GUI by *BHA* and the upper *drillstring*. Input GUI and internal format are identical, but *drillstring* description is stored in XML file with *.drs extension, *BHA* file has **.bha* extension. The files are saved and loaded from *Source directory\drillstrings* folder by default.

List of parts / pipes / pipe sections is stated in the main input table; **Side Section** and **Part Images** buttons on head menu enables visualization on the assembly during creation.

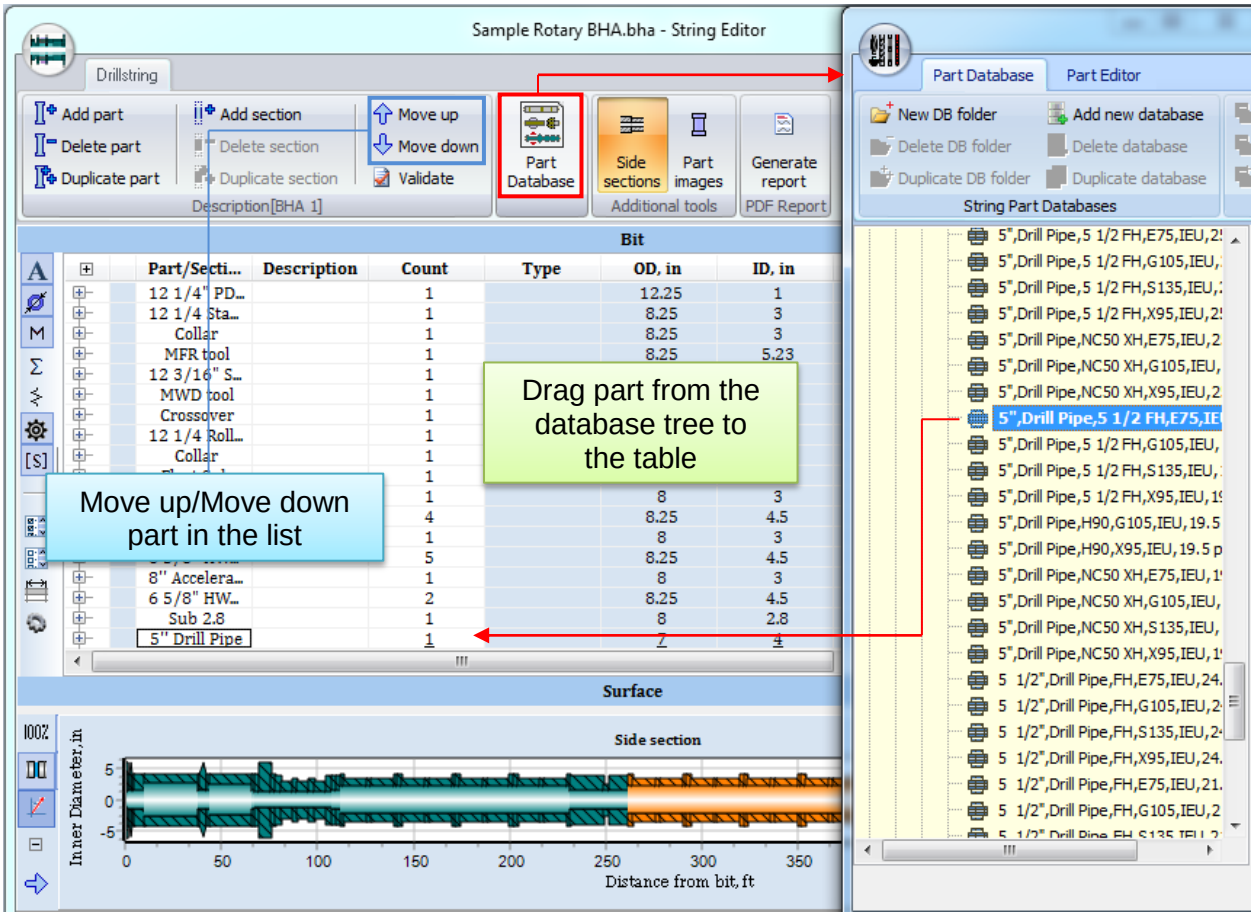


GUI enables several variants of the assembly description:

➢ Adding parts from *String Part Database*

One can open **String Part Database** window by main menu, or **Part Database** button on the head menu.

Drag-drop parts from the database tree to the editor table to add the part to the end of the list (top end of the assembly). Parameters of the part are displayed in the last lines of *String part table*. One can use **Move Up** ( ) or **Move Down** ( ) buttons on the top of the table to change the order of the parts in the list.

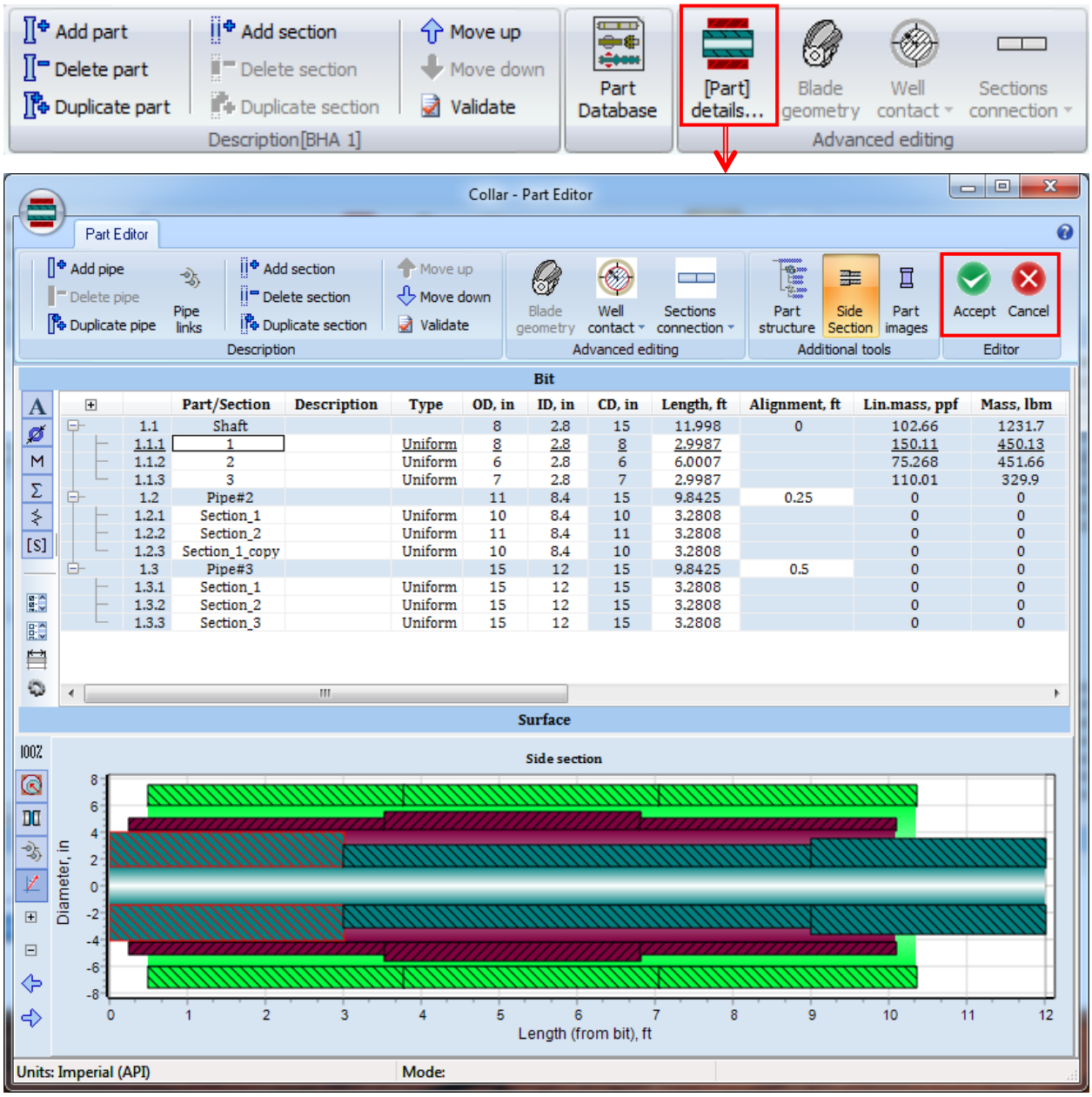


If it is necessary to add multiple instances of a part one can repeat drag-drop procedure several times, or just specify the number of repetitive parts in **Count** column of the table. The **Plural** part image is used for the part in string scheme if the count is more than one.

➢ New parts creation

Drillstring Editor GUI enables creation of new parts and modification of the units taken from *String Part Database*. One can use head menu buttons to add new part (), delete () or duplicate () the selected part.

Drillstring Editor parts description tools are based on *String Part Editor* interface (see Sect.1.4.2.1). Head menu controls enable description of simple "single shaft" parts. One can select a unit and click on **Part Editor** button to create/modify the multi-pipe component in separate *String Part Editor* window.

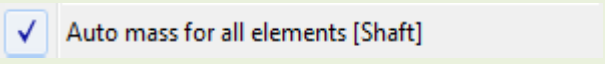


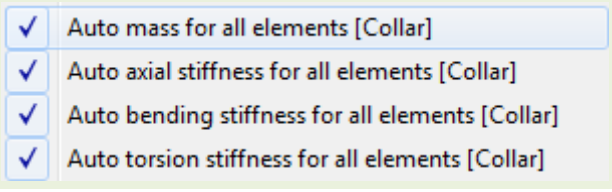
Click **Accept** to apply part description changing, or **Cancel** to reject the modification.

Parameters

Description of input table parameters specific for *part* rows are stated below.

Parameter	Units		Description
	SI	Imperial	
General category - fast access button.			
Part/Section			Part / section caption.
Description			Part description.
Count			Part quantity. Not available in Part Editor; used in *Drillstring editor* only.
Geometry category - fast access button.			
Type			Section type: *Uniform* or *Blade*.
OD	mm	inch	Maximal outer diameter of the part.
ID	mm	inch	Minimal inner diameter of the part.
CD	mm	inch	Maximal contact diameter of the part. **Note**: *Contact diameter* can be not equal to *outer diameter* for parts containing blade sections.
Length	m	ft	Length of the part, calculated from the length of *Shaft* pipe.
Alignment	m	ft	Parameter of multi-pipe parts description: distance from the lower end of *Shaft pipe* to the lower end of *Outer pipe*. **Note:** Alignment field is available for *Outer pipe* items only.
Inertia category - fast access button.			
Material			Material of the part elements (reference info field). If material is assigned for all sections of the part, the material caption is displayed in the pipe material field; the field is empty otherwise. The user can set material for the section, or for all pipe sections, or for all part sections by popup menu tools. **Note:** Material is selected from *Material Database*; one needs to edit the database if part material is not included in the database.
Lin. mass	kg/m	lbm/ft	Linear mass of the part. Part linear mass is taken from the corresponded mass value divided by the length value.

Mass	kg	lbm	Mass of the part. Mass of section is calculated automatically from material density and section geometry. **Note:** *Linear mass* and *Mass* of section/pipe value can be set manually in **Advanced User** and **Developer** modes. One needs to select the corresponded row in the table and disable **Auto mass…** option by pop up menu. 
Summary category - Σ fast access button.			
Acc. Length	m	ft	Accumulated length reference value (read only), taken with the account of part quantity. Value is calculated as the accumulated length of the previous part, plus *Shaft* pipe length multiplied on part quantity. **Note:** First part *Acc. length* value is equal to the part length; the last part value is equal to the total assembly length.
Acc. Mass	kg	lbm	Accumulated mass reference value (read only), taken with the account of part quantity. Value is calculated as the accumulated mass of the previous part, plus part sections mass multiplied on part quantity. **Note:** First part *Acc.mass* value is equal to the part mass; the last part value is equal to the total assembly mass.
Stiffness category - fast access button.			
Model type			*Part pipes* model type (reference value): • *Flexible* – all part sections are simulated by flexible uniform beams; • *Rigid* – all pipes are modeled with absolutely rigid bodies of the prescribed geometry; • *3D FEM model* – all pipe model are described by 3D finite element models (*.fss file), imported from FEM software in accordance to the modal finite element approach. • *Flexible + Rigid, etc.* – combinations of the upper stated variants

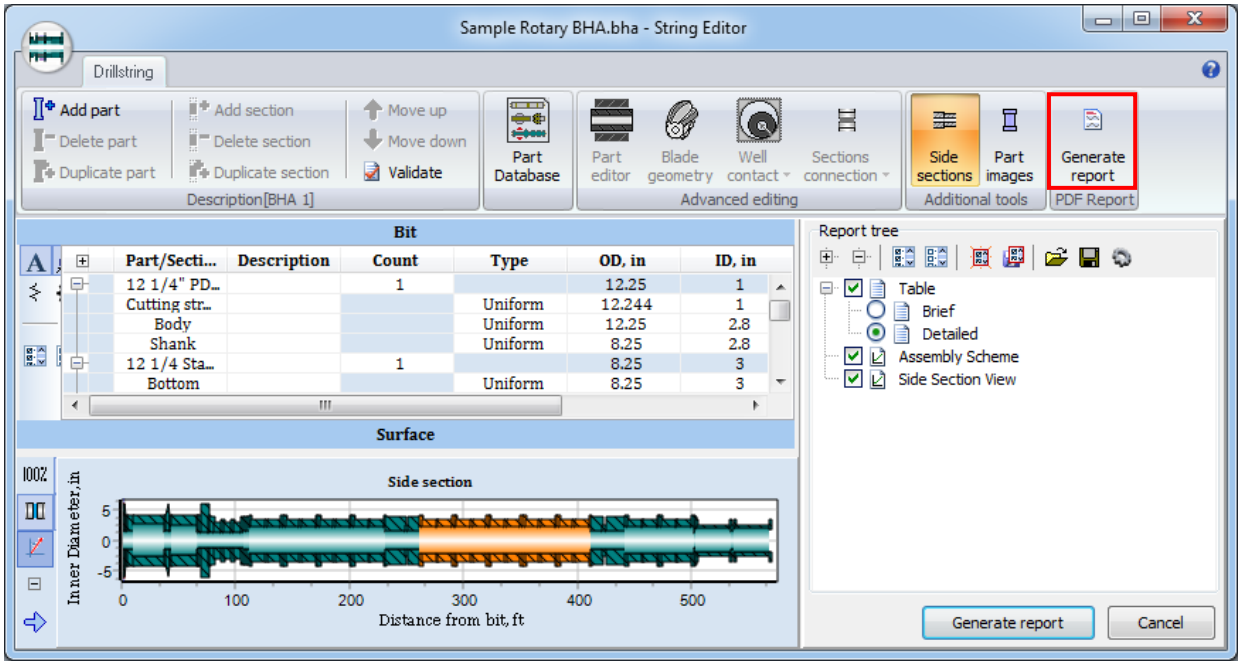
Lin. Axial. Stiff	kPa	ksi	Linear axial/bending/torsional stiffness of the part. Part axial stiffness is taken from the stiffness and geometry of the sections. **Note:** *Linear axial/bending/torsional stiffness* of part value can be set manually in **Advanced User** and **Developer** modes. One needs to select the corresponded row in the table and disable **Auto axial/bending/torsional stiffness…** option by pop up menu. 
Lin. Bend. Stiff	kN/m^2	kip/ft^2	
Lin. Tors. Stiff	kN/deg	kip/deg	
Safety category - [S] fast access button.			
Max. T	kN*m	kip*ft	Reference value of the maximal admissible torque/force can be applied to the part pipes. **Note:** Parameters are not used in the current version of the DSA software.
Max. F	kN	kip	

Data verification

String description can be verified with the  button placed on the head menu of the editor windows.

Data reporting

String description can be reported as PDF document with the  button placed on the head menu of the editor window.



1.5. GUI Drillstring Analysis Project

DSA Software enables solution of different types of problems related to drillstring dynamics. Problem formulation for different types of analysis requires diverse source data.

Project structure

To simplify the analysis procedure of for the user the DSA GUI units all the source data related to the different types of analysis, as well as analyses settings and calculated results in a Drillstring Analysis project (*Project*).

GUI enables run of single or several projects of the following content:

- *Drilling Information*

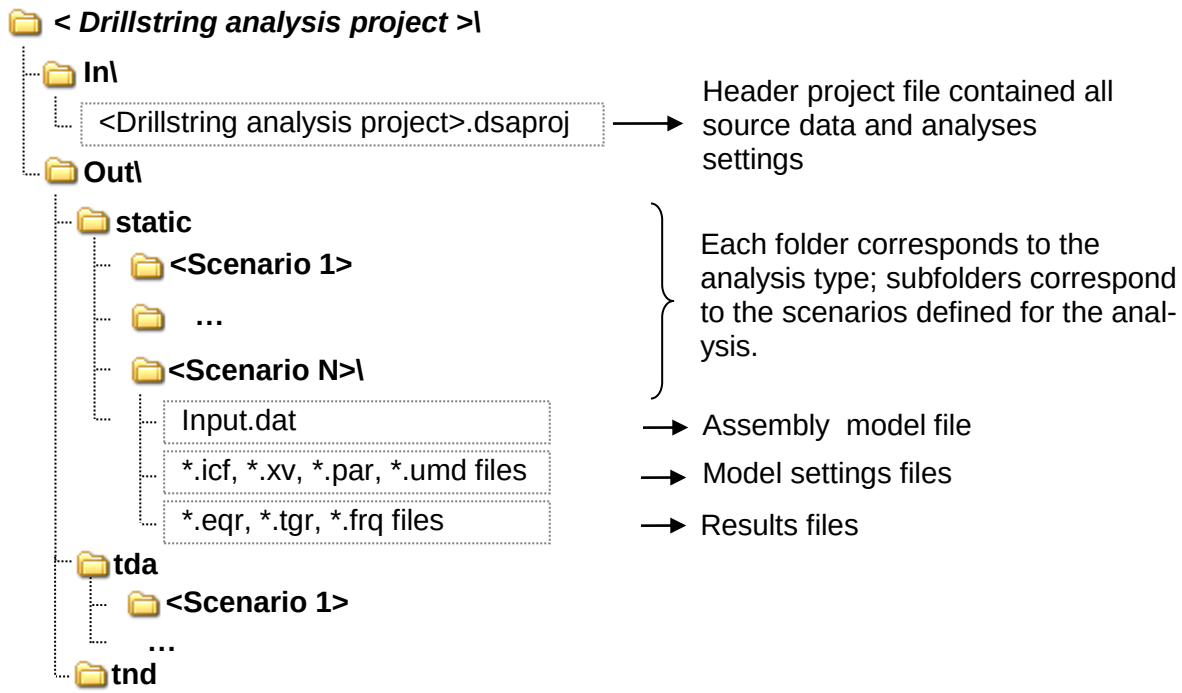
  Source data on well trajectory, wellbore intervals, BHA and drillstring design cases used in the project.

- *Analysis*

  Each type of analysis within the *Project* can contain any number of parameter sets – *scenarios* – that can be run in parallel threads or one by another. *Scenario* corresponds to the definite position of the drillstring/BHA in the well, set of operational settings and options specific for the analysis.

Project data storage

The following catalogues/files structure is used for project data storage:

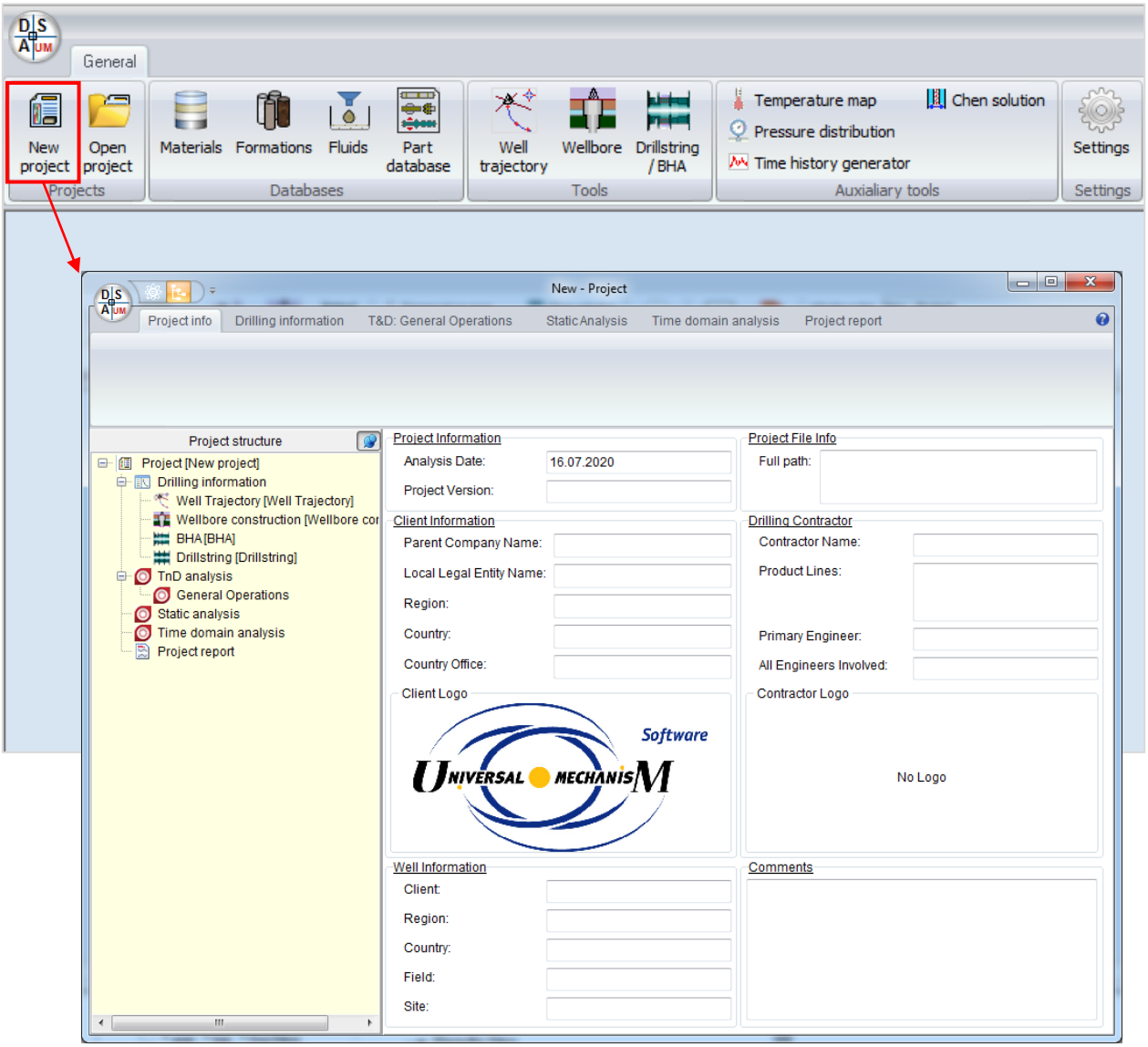


Projects files are stored in *Source directory\projects* folder by default.

1.5.1. Start New Project

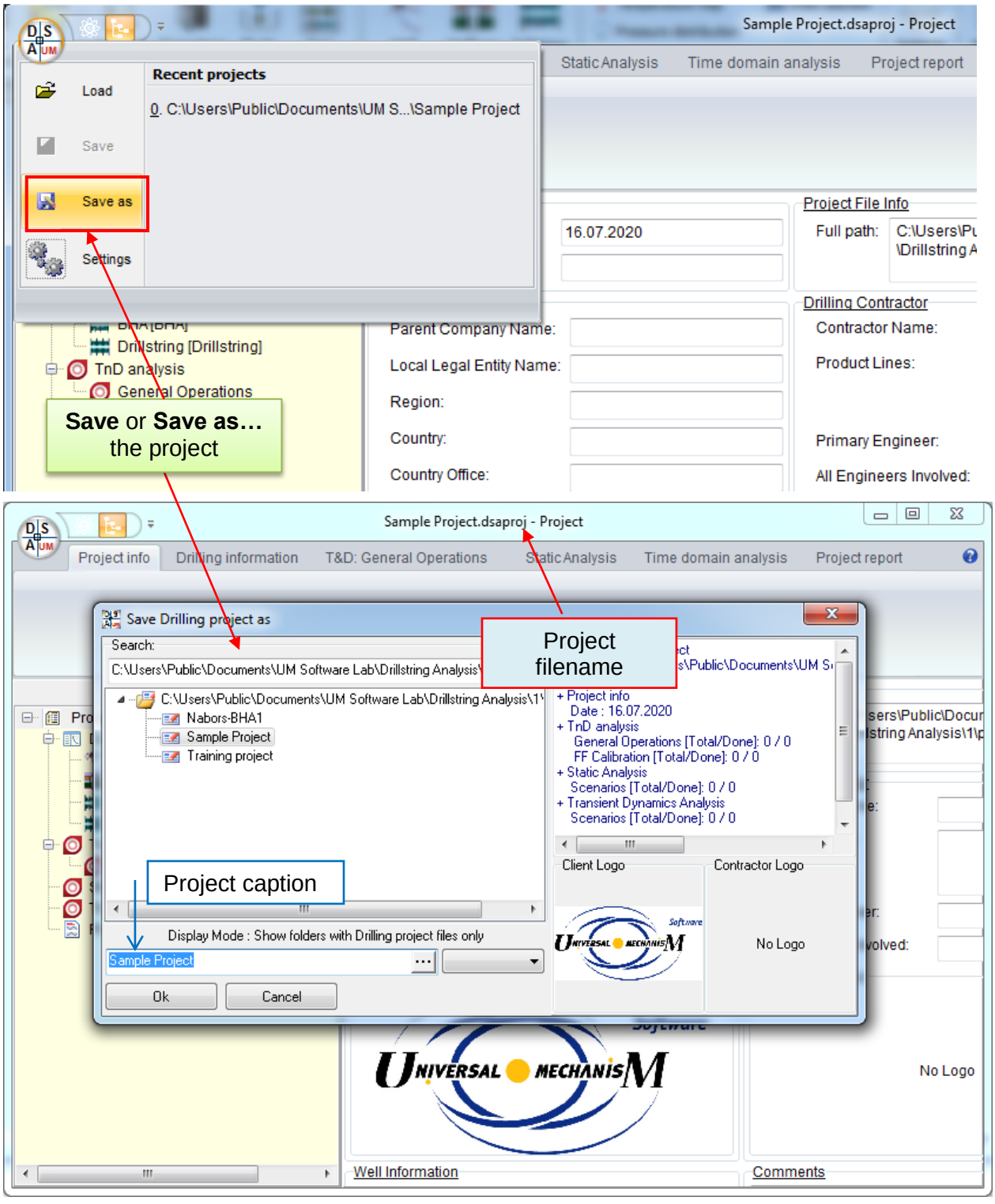
Click  button on the **Main window** top panel to start a new project. New **Project window** will be opened. The user can **Save** or **Save as** the project with the buttons placed on the head of **Project** window. One can also use buttons of the **Main window** head menu to save or save as the active project.

The structure of the window pages is described in details in Sect.1.5.4.



1.5.2. Save Project

New project is not saved to hard drive automatically. Click **Save as…** button and specify the new project location in **Save drilling project** dialogue window to save the project to disk.



The project name is displayed on the **Project window** header.

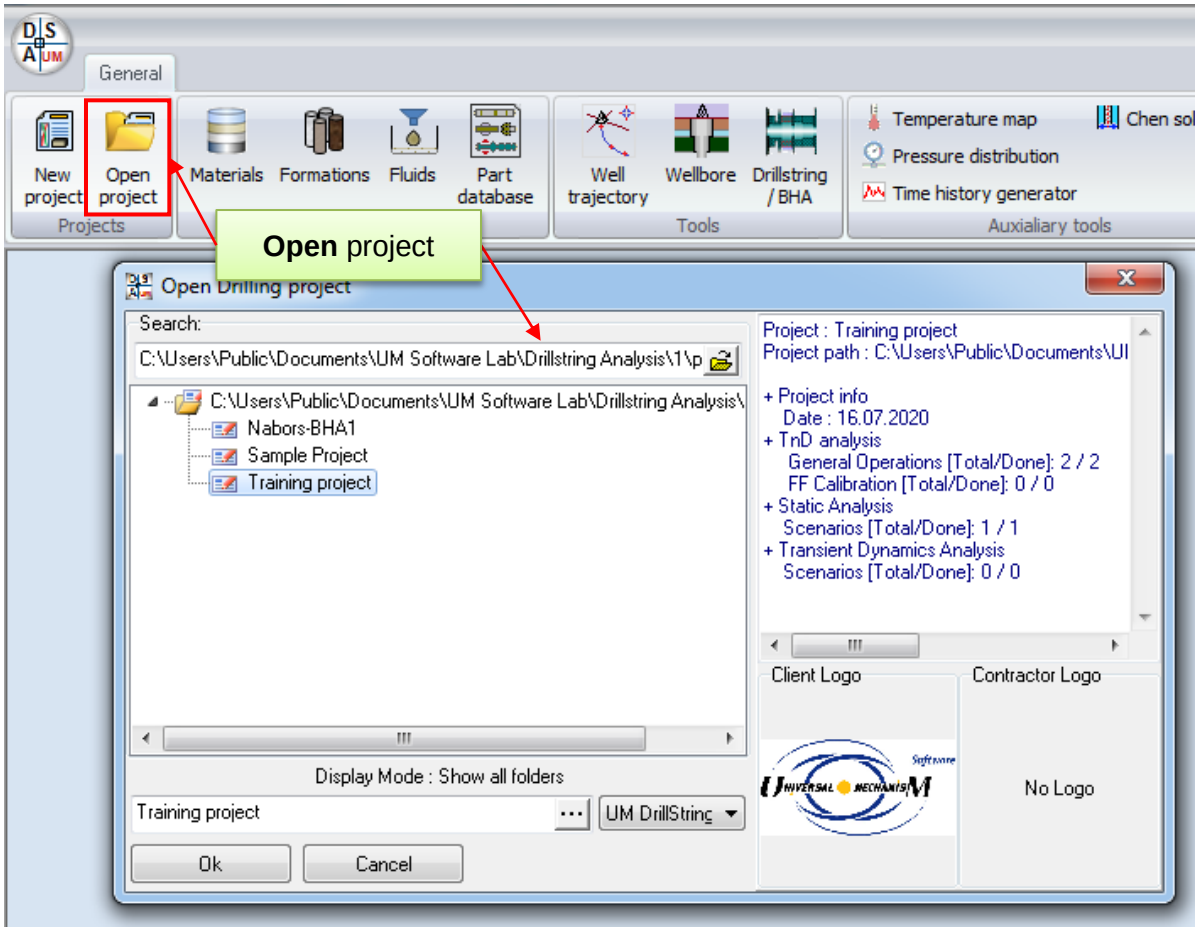
Save button is enabled and ‘ * ’ postfix is added to the project name while any changing are made in the project.

1.5.3. Open Project

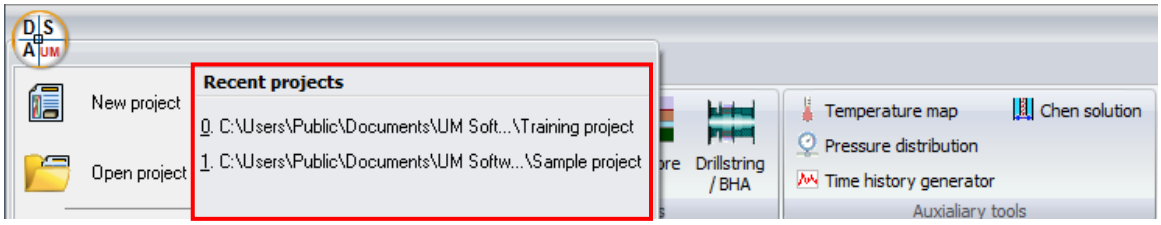
One can load the previously created drilling project with **Load** button on the head menu.

Program scans the **Source directory** automatically for available drillstring analysis projects. The list of projects lying in the **Source directory** or its sub-catalogues is displayed on the left panel of the **Open drilling project** window. The right panel contains the brief data on the selected project.

Click **Ok** to load the selected project in the new **Project window**. Click **Cancel** to close the dialog without project loading.



One can use **Recent projects** list of application menu to load on of the recently opened projects.

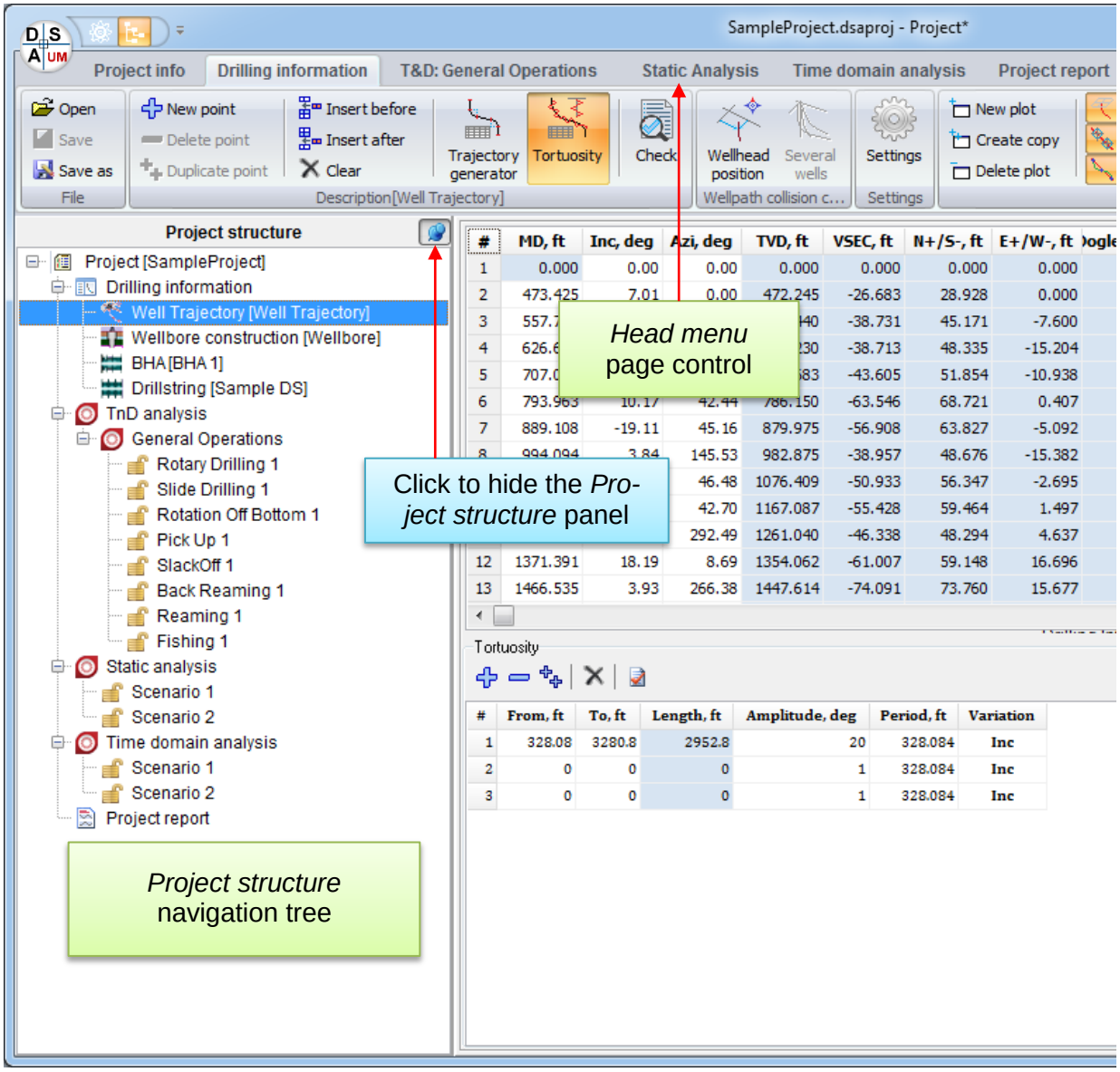


1.5.4. Project window

DSA *Project* window interface enables access to the *General project info*, *Drilling information*, and *Static*, *T&D* and *Time Domain* analyses data.

1.5.4.1. Project navigation tools

Navigation on project items is implemented by **Project structure** tree placed on the left panel, and partially duplicated by the head mean pages.



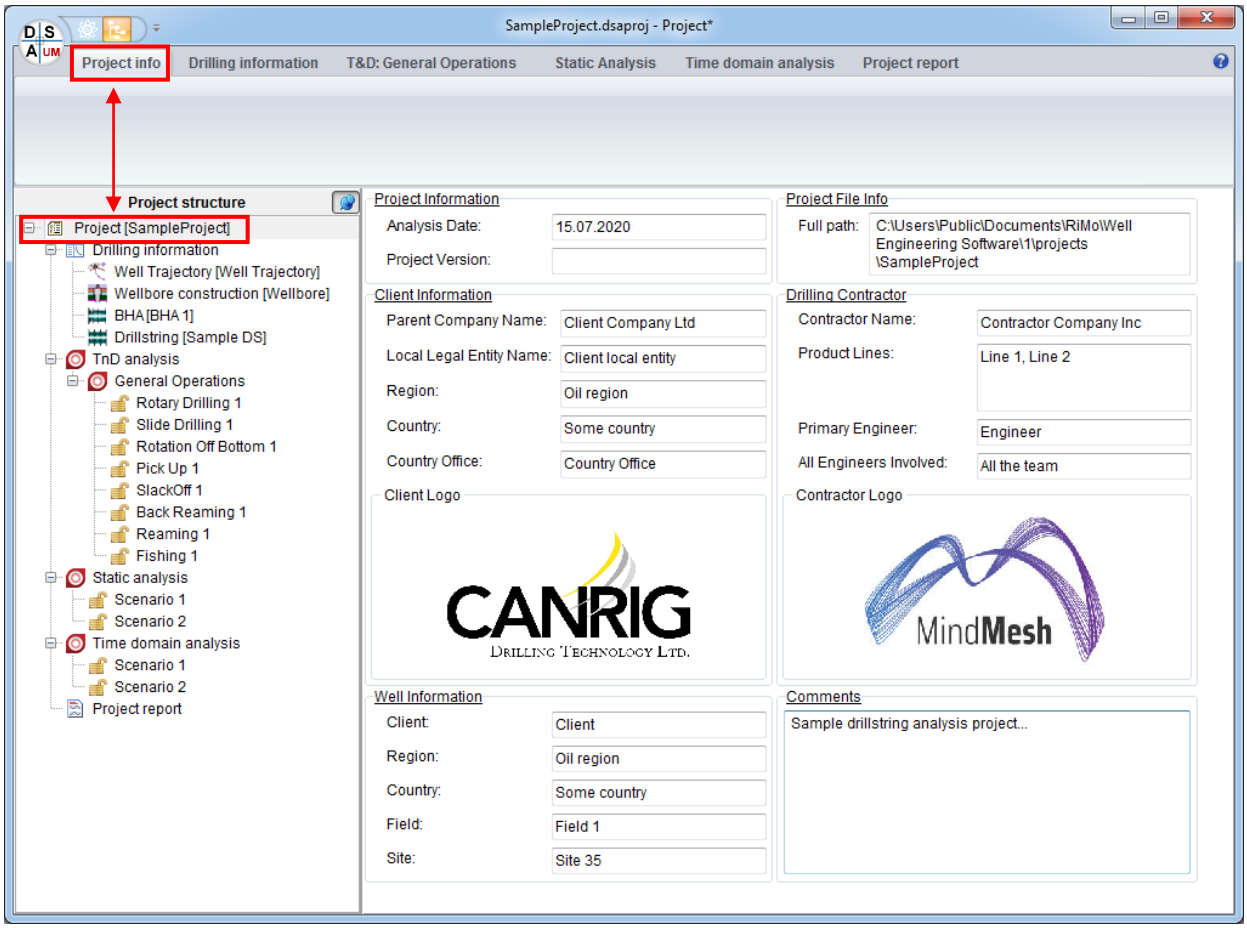
1.5.4.2. General project info page

Project Info page of the head menu contains general description of the project.

- ✓ Project & project file information
  - Analysis date - date of the last run of analysis or the project creation date.
  - Project version
  - Project file full path (with direct access to project folder by pop up)
- ✓ Client and Drilling Contractor information

Note: Double click on the **Client Logo** / **Contractor Logo** field to upload the logo from file. Default set of logo image files is placed in *Source directory\logos* folder. Select **Clear logo** menu item from the pop up menu to erase the logo.

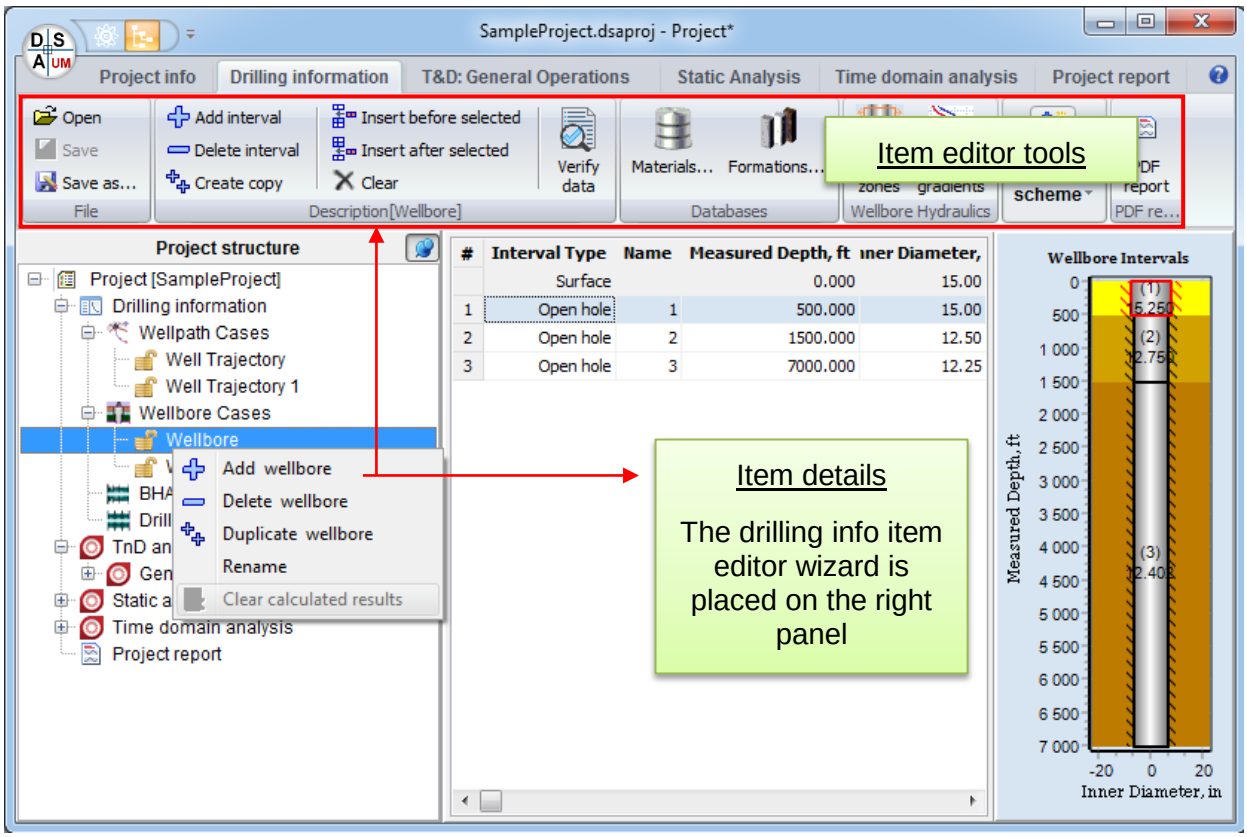
- ✓ Well information
- ✓ Project comments



1.5.4.3. Drilling Information page

Access to the *Drilling information* items of the project – well trajectories, wellbore cases, BHA and drillstring cases – is realized by *Project structure* tree.

Project drilling info is displayed in the tree under **Drilling Information** node as the sub nodes, grouped under **Well Trajectories**, **Wellbores**, **BHA Cases**, **Drillstring Cases** nodes.



One can select a node to display/edit the item details on the right panel; head menu **Drilling Information** page is initialized by the corresponded item editor tools.

The pop up menu enables the following operations with the selected item:

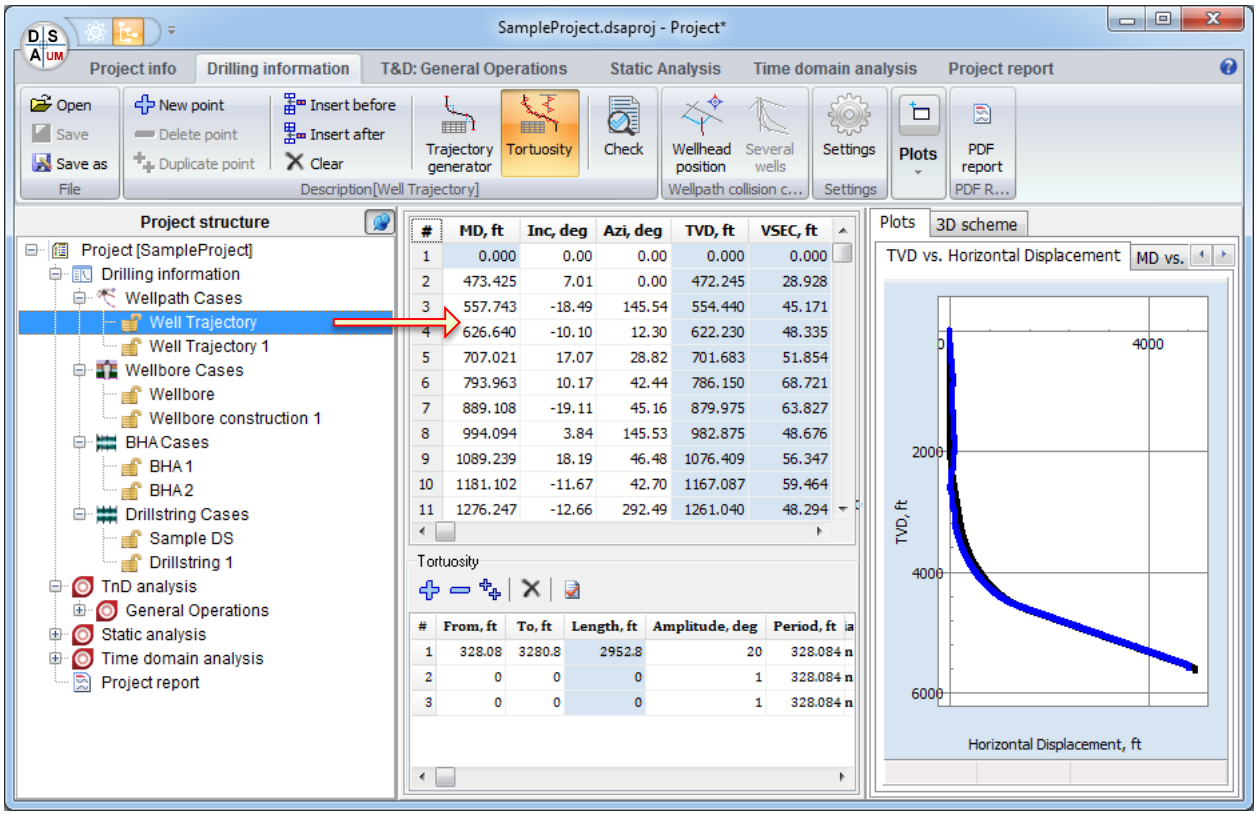
- add new item;
- delete selected item;
- duplicate selected item;
- clear calculated results (delete results of all scenarios the item is used).

Note: The item data is read-only if the item is used in one or several calculated scenarios. The icon is used for read-only items in the *Project structure* tree.

If one will try to delete the read-only item the request on **Clear calculated results** will be generated automatically.

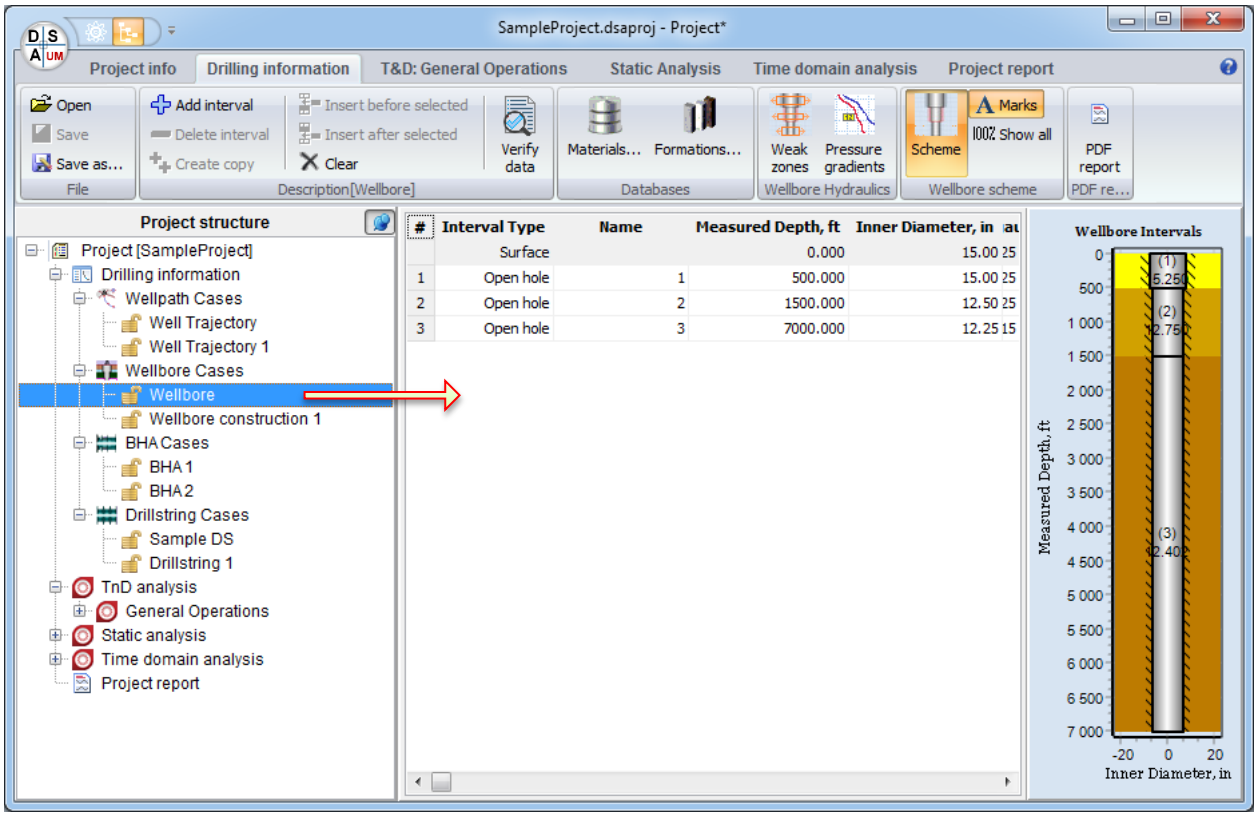
Well trajectory cases

Well trajectory item description functional is described in Sect.1.4.3.



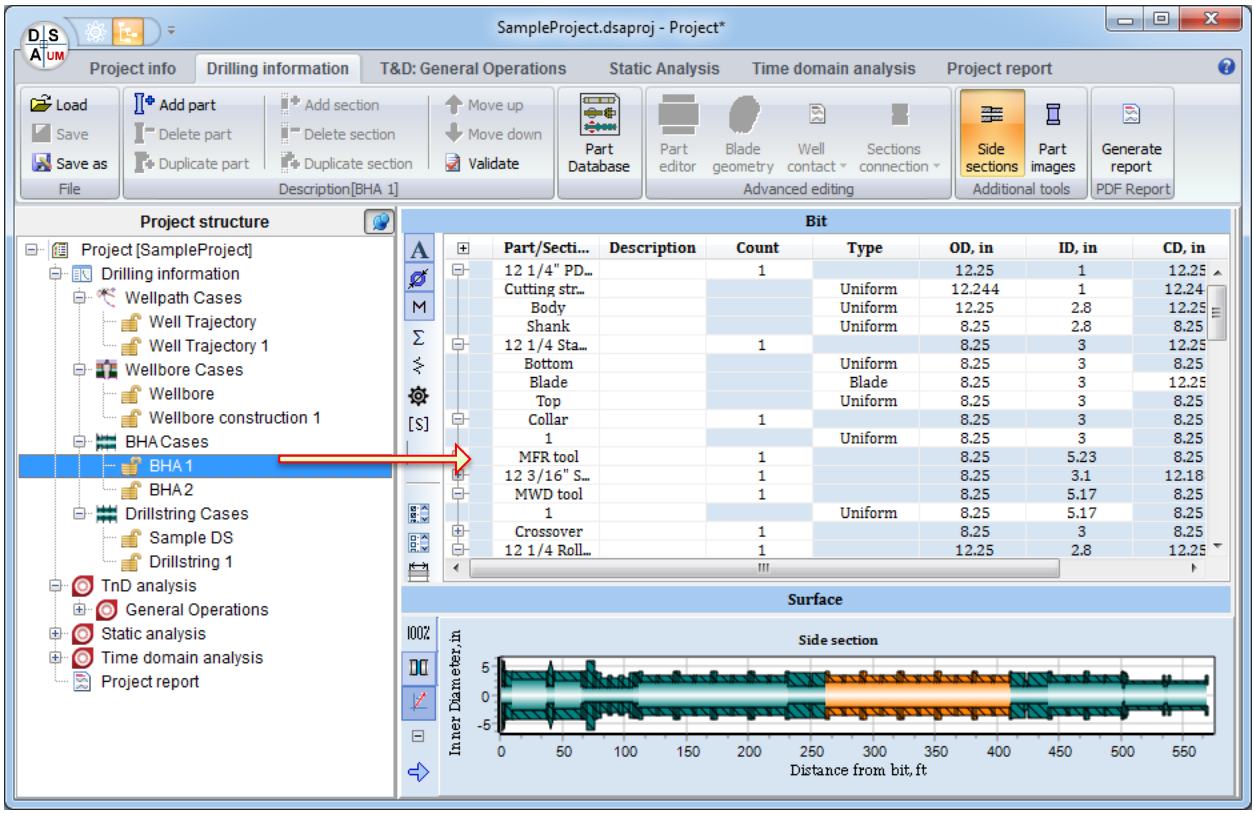
Wellbore cases description

Wellbore item description functional is described in Sect.1.4.4.



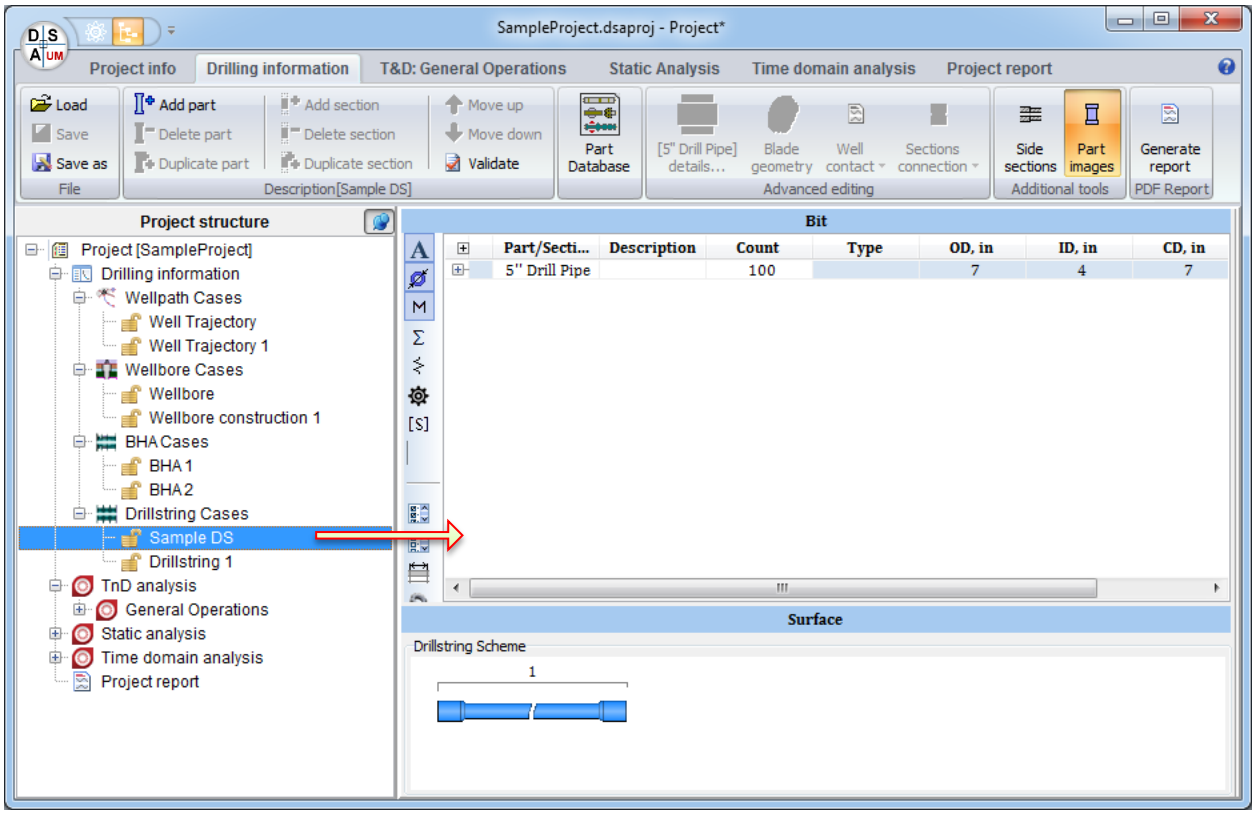
BHA cases description

Assembly description functional is described in Sect.1.4.5.



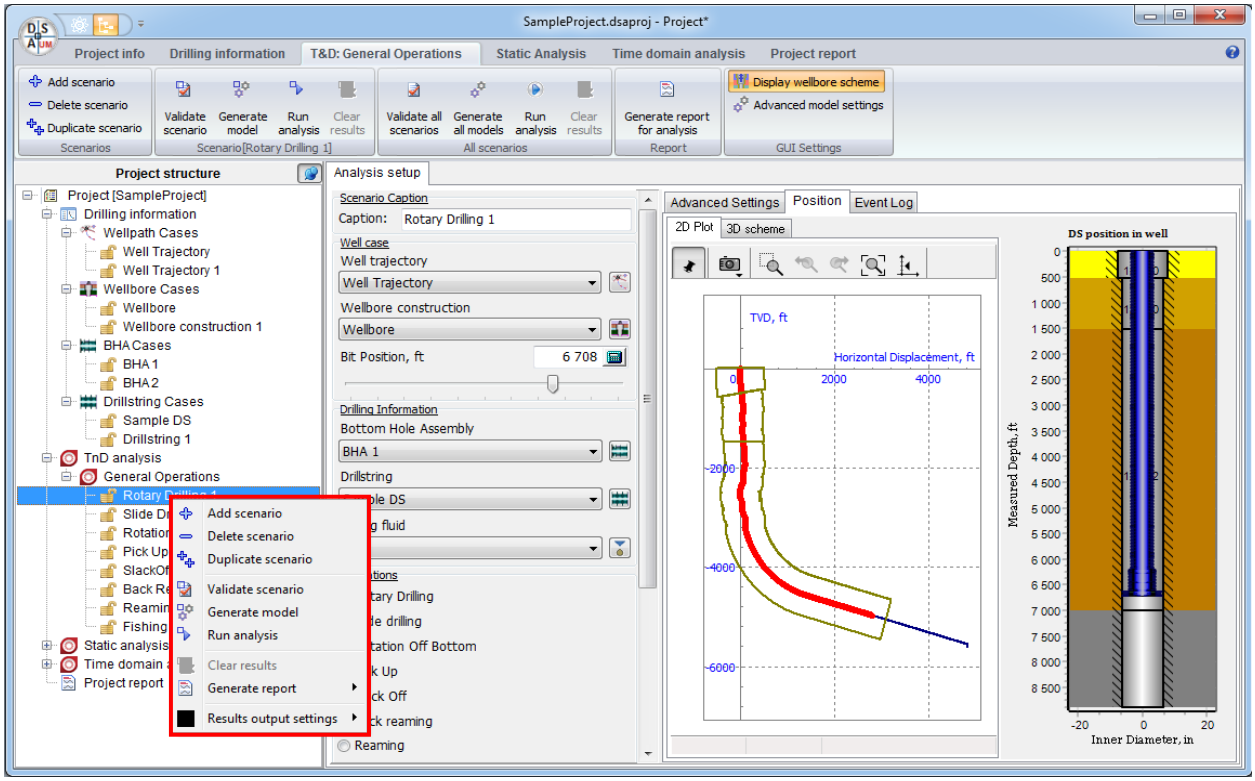
Drillstring cases description

Assembly description functional is described in Sect.1.4.5.



1.5.4.4. Analysis GUI

Drillstring analysis project can include any number of *scenarios* – blocks of input data, analysis settings and calculated results. List of scenario nodes is placed under the analysis node in the *Project structure* tree.



DSA Analysis GUI enables description of source data and analysis settings, validation of the inputs, generation of internal dynamic models, running the analysis and results output for single scenario, as well as for all scenarios for an analysis.

The pop up menu enables the following operations with the selected analysis or scenario nodes:

- add new scenario
- delete selected scenario;
- duplicate selected scenario;

✓ Operations available for single scenario, or all the scenarios:

- validate the scenario;
- generate model;
- run analysis;
- clear calculated results;
- generate PDF report for single scenario or an analysis (see Sect.);
- select 2D plot line color for scenario.

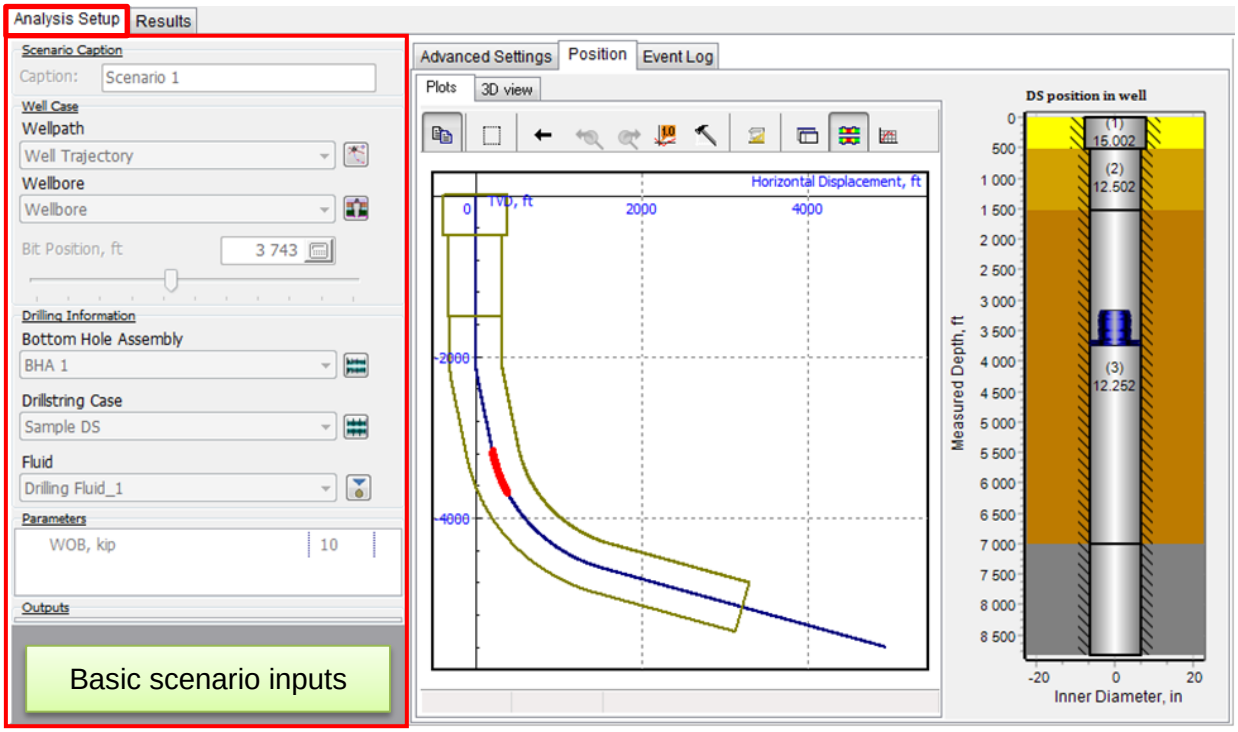
One can select a scenario node to display scenario data on the right panel:

✓ Analysis Setup

Analysis setup page contains tools for basic and advanced scenario parameters description, visualization of the assembly position in the well, as well as the analysis run and evaluation progress control.

Basic parameters

Set of basic drilling scenario inputs placed on the middle panel: information (well trajectory, wellbore, bit position, assembly, drilling fluid, etc.), operation settings and solver settings.



Advanced settings

Advanced settings of dynamic model of the assembly and fine turning of the solver parameters are available in **Advanced User** and **Developer** mode. Content of the page depends on the analysis type.

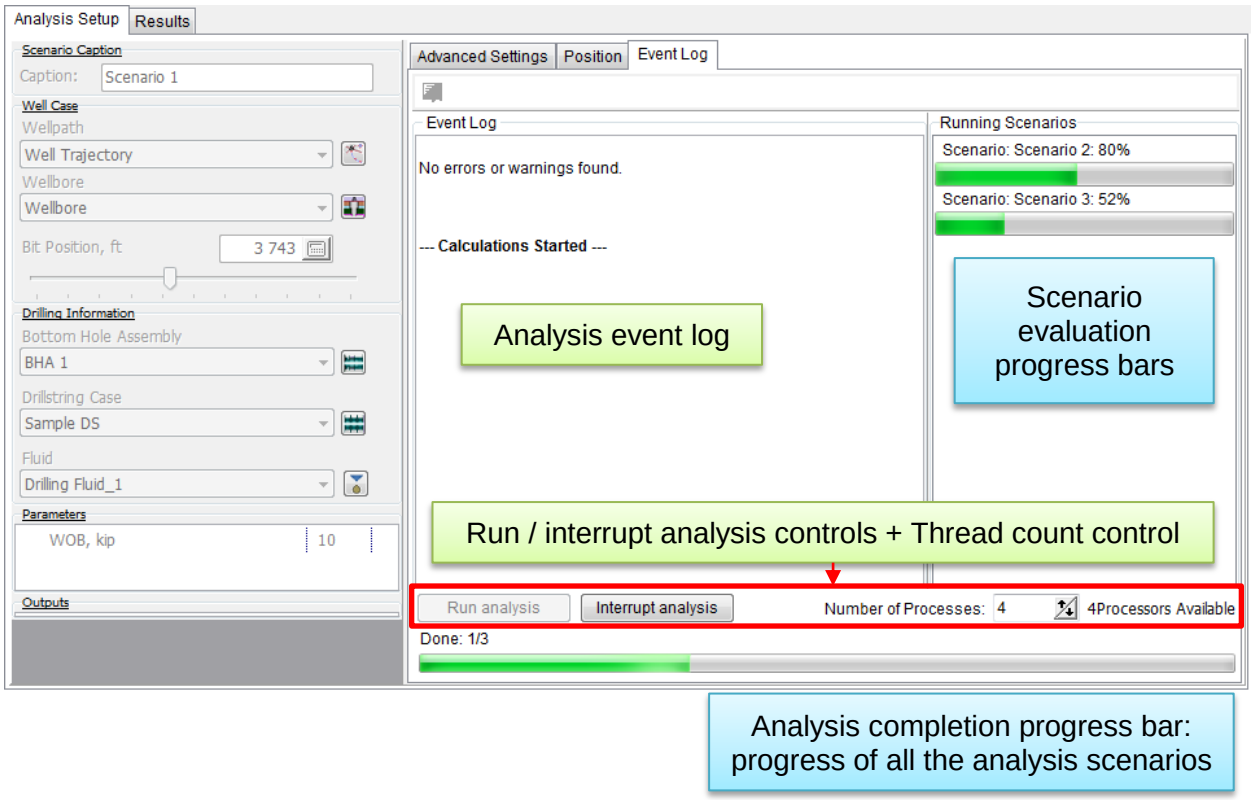
Note: Setup parameters are read-only if the scenario is calculated, or running, or already calculated. The icon is used for read-only items in the *Project structure* tree. One needs to **Clear calculated results** to change the inputs.

Visualization of the assembly position in the well

Position of the assembly in the wellbore is displayed on **Position** page by the – *Horizontal displacement vs. TVD* plot, *Well trajectory* 3D view and *Wellbore scheme* (optionally).

Analysis run and evaluation progress control

Event Log page for control of analysis procedure and warning / error messages output.



Click **Run analysis** button to start evaluation of all the scenarios.

Analysis calculations are processing in parallel threads – one thread per scenario; the user can specify the maximal number of threads involved in the analysis by **Number of Processes** control on the bottom panel before or during simulation.

Individual progress bars are added to the right panel for each of the proceeding scenarios during the analysis.

Interrupt analysis control enables breaking of evaluation procedure.

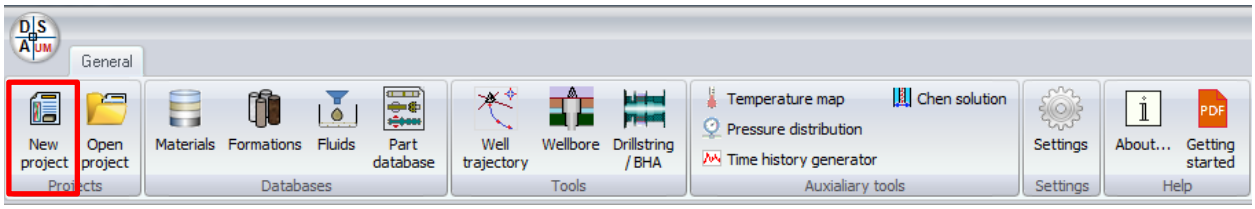
✓ **Results**

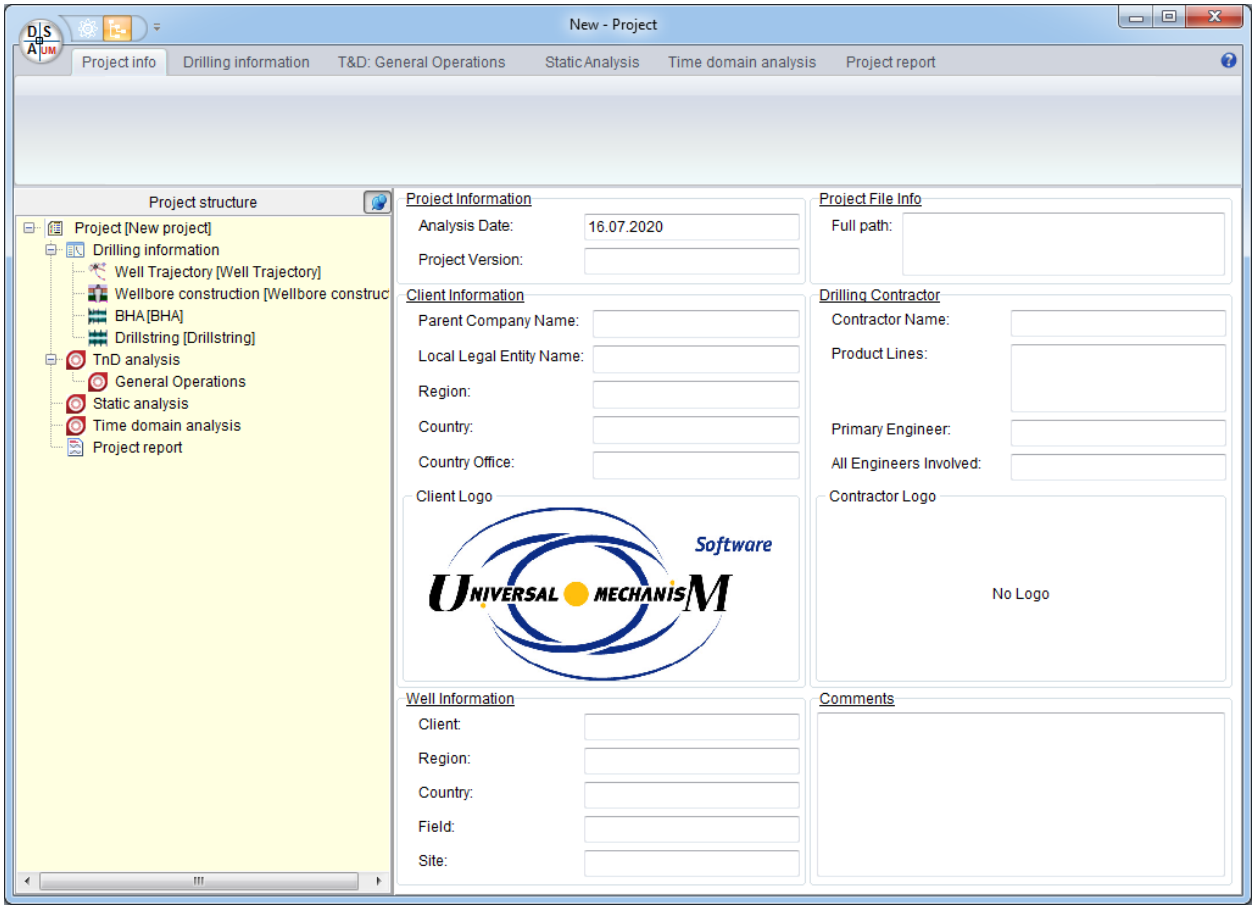
Results page contains set of tools for calculated results output for single scenario, or selected set of scenarios.

2. Training project

This section contains the step-by-step description of the procedure of the training project creation, drilling information input, Static and Transient analysis scenarios description, analyses run, evaluation control and results post-processing within DSA application GUI.

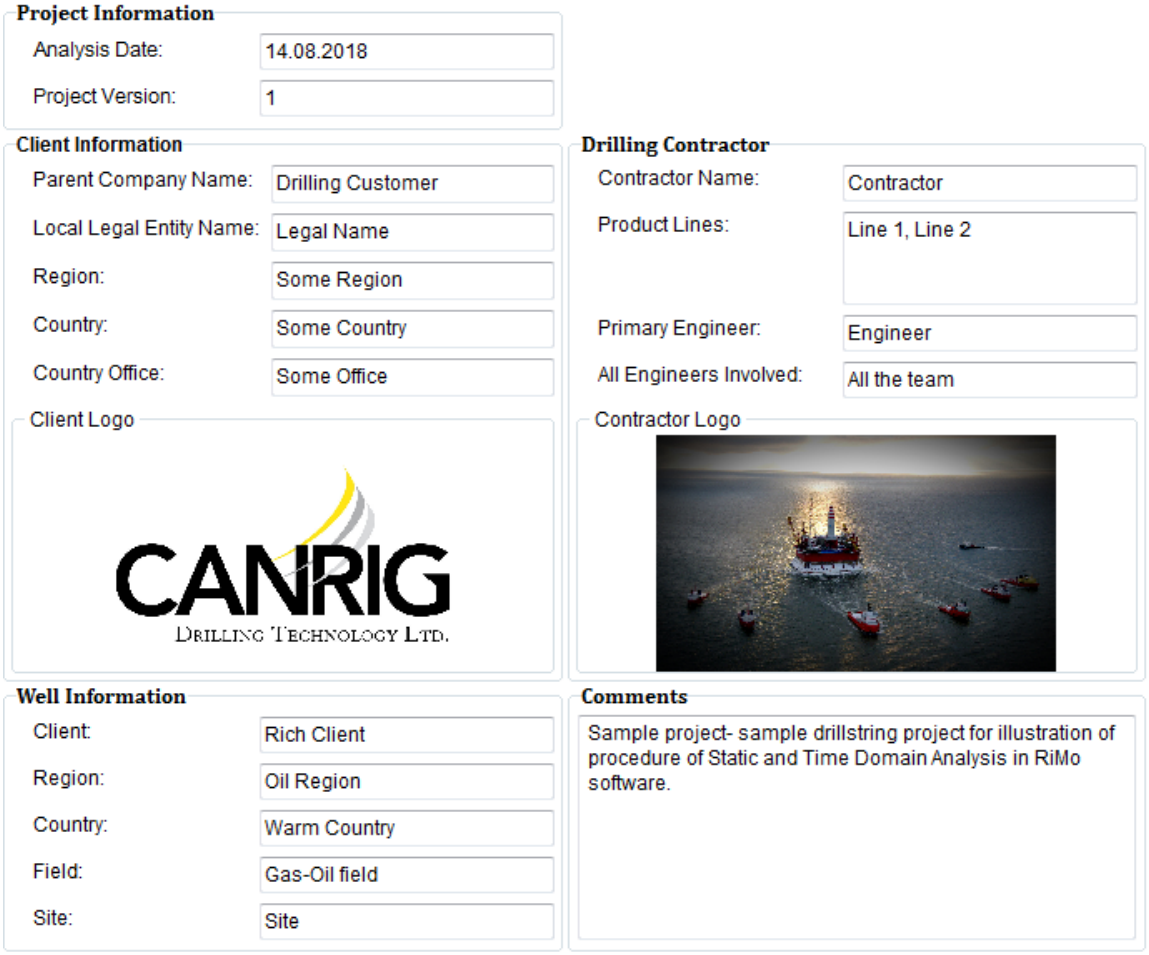
Step 1.1. Create **New project** by the head menu button; the project window will be opened.



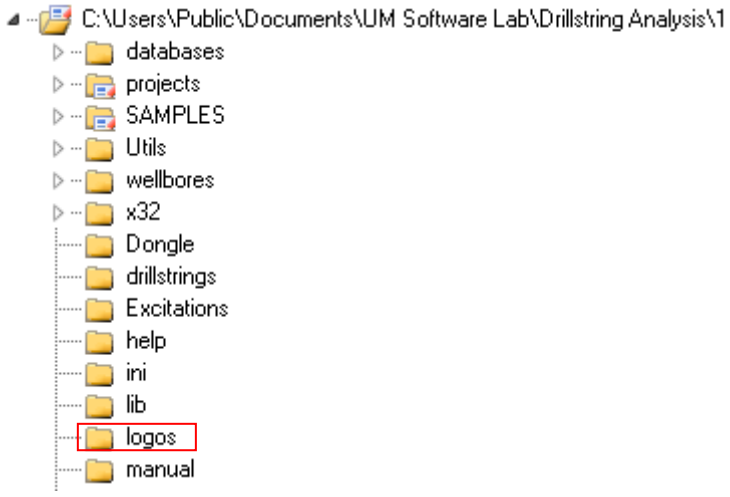


2.1. General project data input

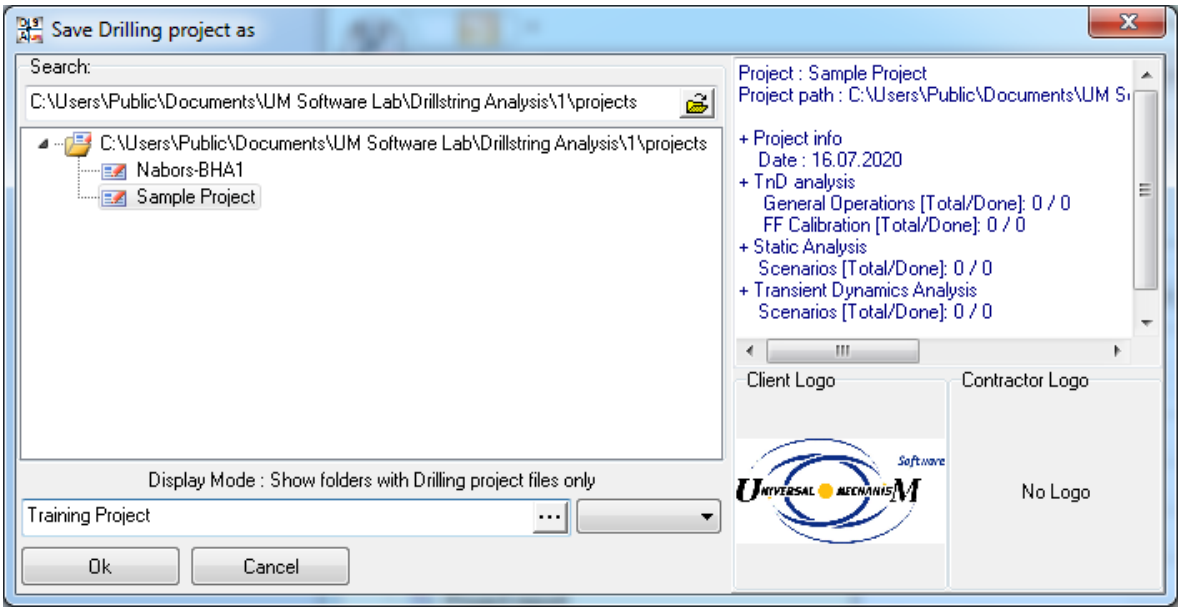
Step 2.1. Use **Project Info** tab to input general project information like project date and version, client and drilling contractor information, location data, well information and some comments on the project.



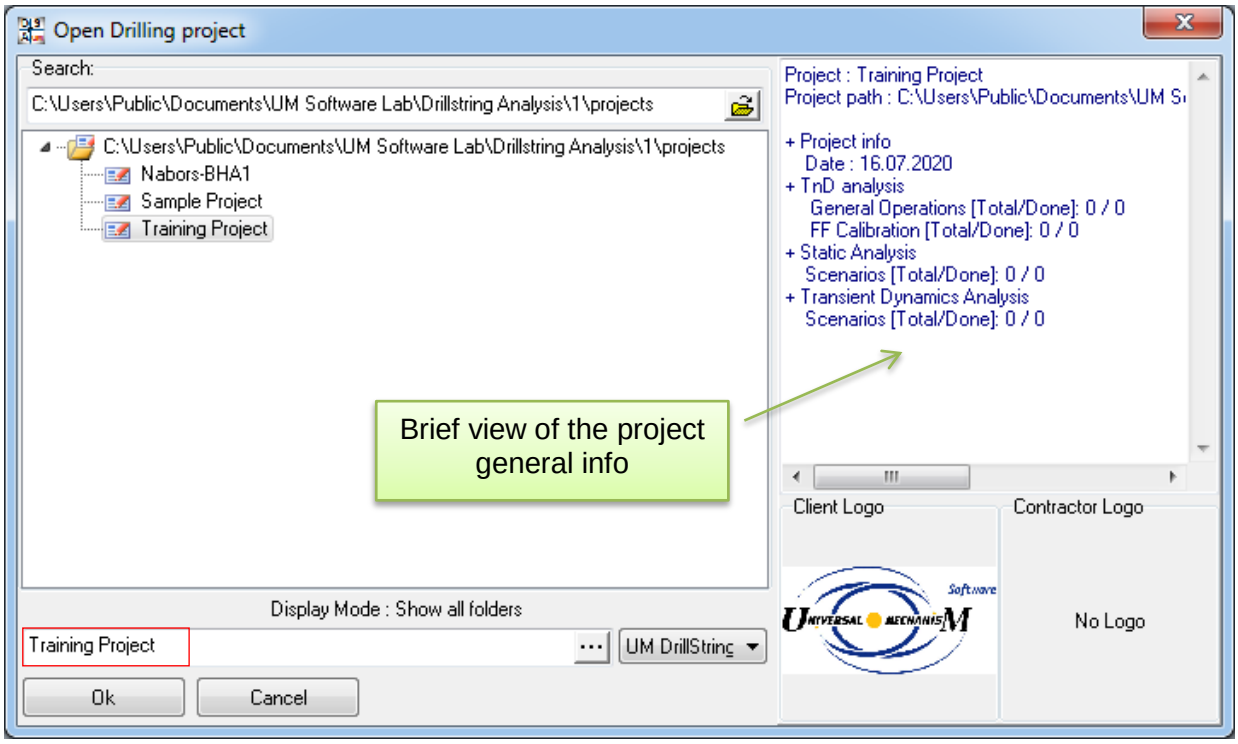
You can add client and drilling contractor logos to make project more recognizable. Double click on **Client Logo** and **Contractor logo** panel to load of the logos from *Source directory\logos* default folder, or any other location.



Step 2.2. Save modified project as **Training project**. The name will be added to the **Project window** caption.



Step 2.3. Click the **Save project as** bottom again and select the project from the list. You can see that the input data is stated on the right panel of the **Open/Save project** dialog window.
Close the dialog by clicking **Cancel**.



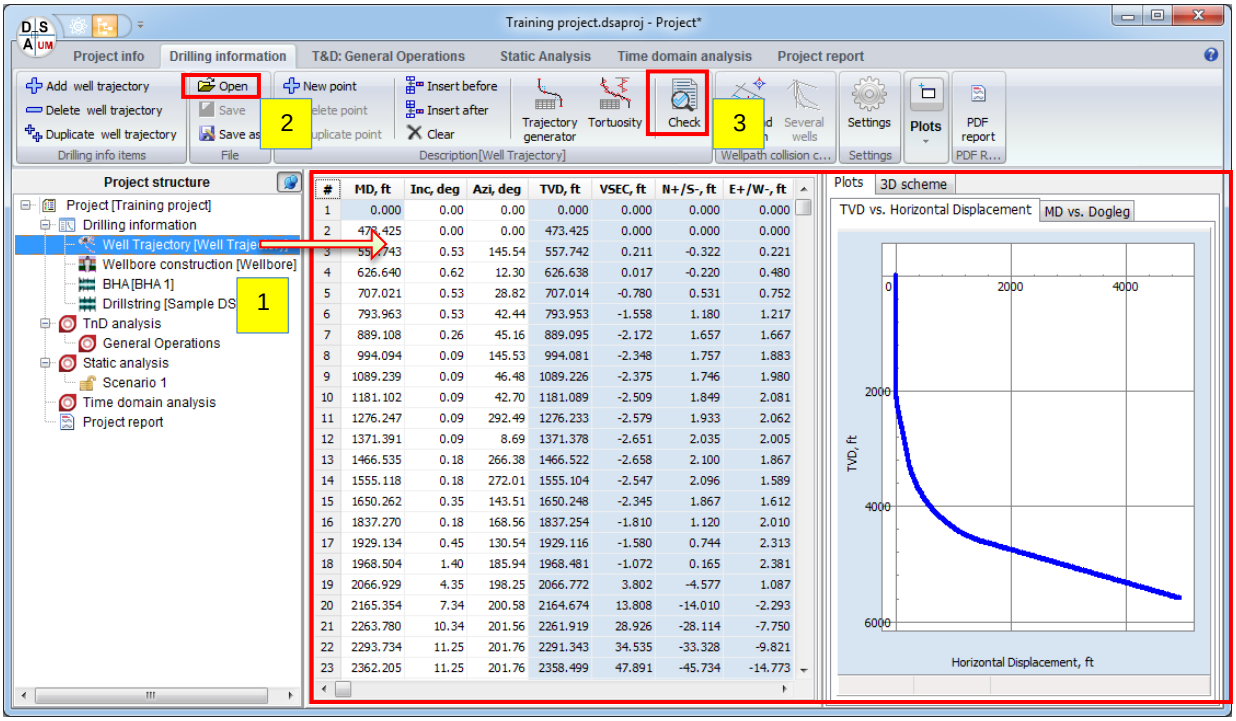
2.2. Drilling information description

In this section we will path through the input of source data general for Static, Torque&Drag and Time Domain analyses.

2.2.1. Well trajectory

Step 3.1. Select the **Drilling Information | Well trajectory** *Project tree* node to input survey data.

The interface of **Well Trajectory Editor** placed on the page and program capabilities in survey data description are described in Sect.1.4.3 Well Trajectory Editor.



Step 3.2. Click the **Open** button on the head menu to load survey data from *Source directory\wellpaths\ Sample Wellpath.wlp* file. Loaded data will be displayed in the table and on the right hand plots.

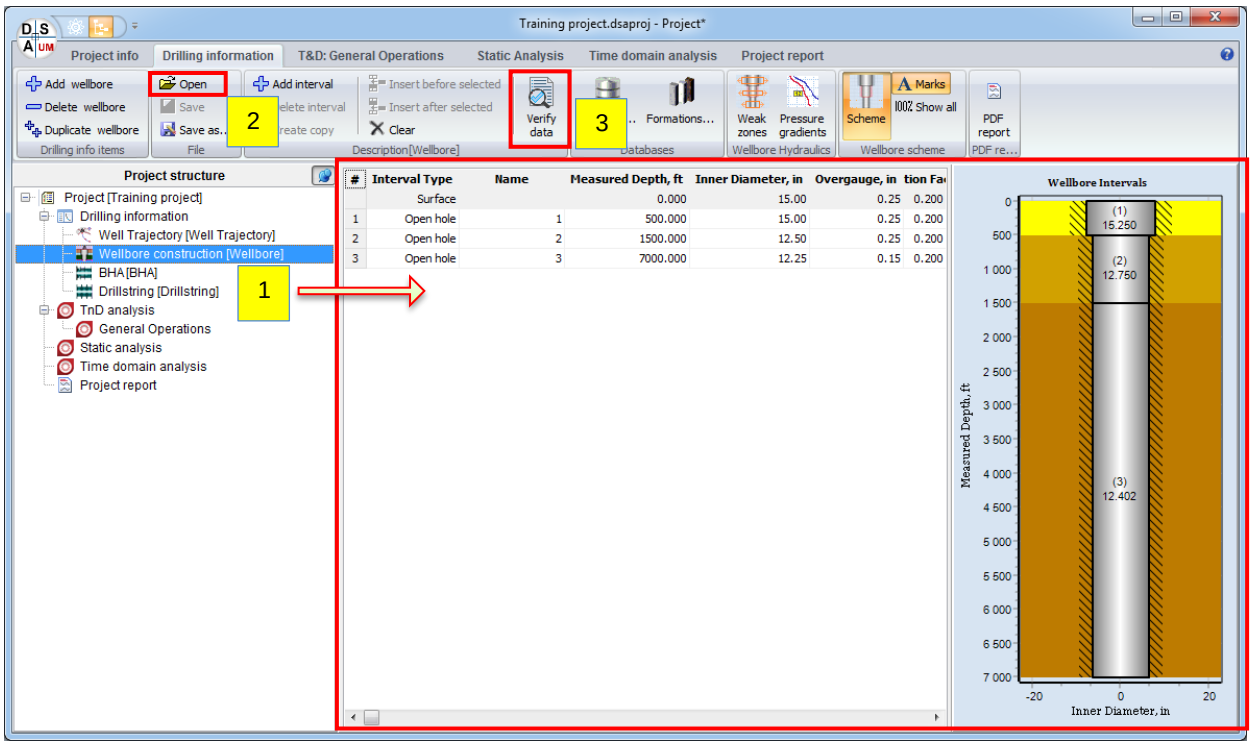
Note. If you've already input some data to the table program will ask you to save it before loading.

Step 3.3. Click the button placed on the head menu. The 'No errors or warnings found' message will appear if the input data is correct.

2.2.2. Wellbore construction

Step 3.4. Select the **Drilling Information | Wellbore construction** *Project tree* node to describe the wellbore.

The interface of **Wellbore Editor** placed on the page and the internal description of wellbore cross section are described in Sect. 1.4.4 Wellbore Editor.



Step 3.5. Click the **Open** button on the head menu to load wellbore description from *Source directory\wellbores\Sample Wellbore.wlb* file. Loaded data will be displayed in the table and on the right hand plots.

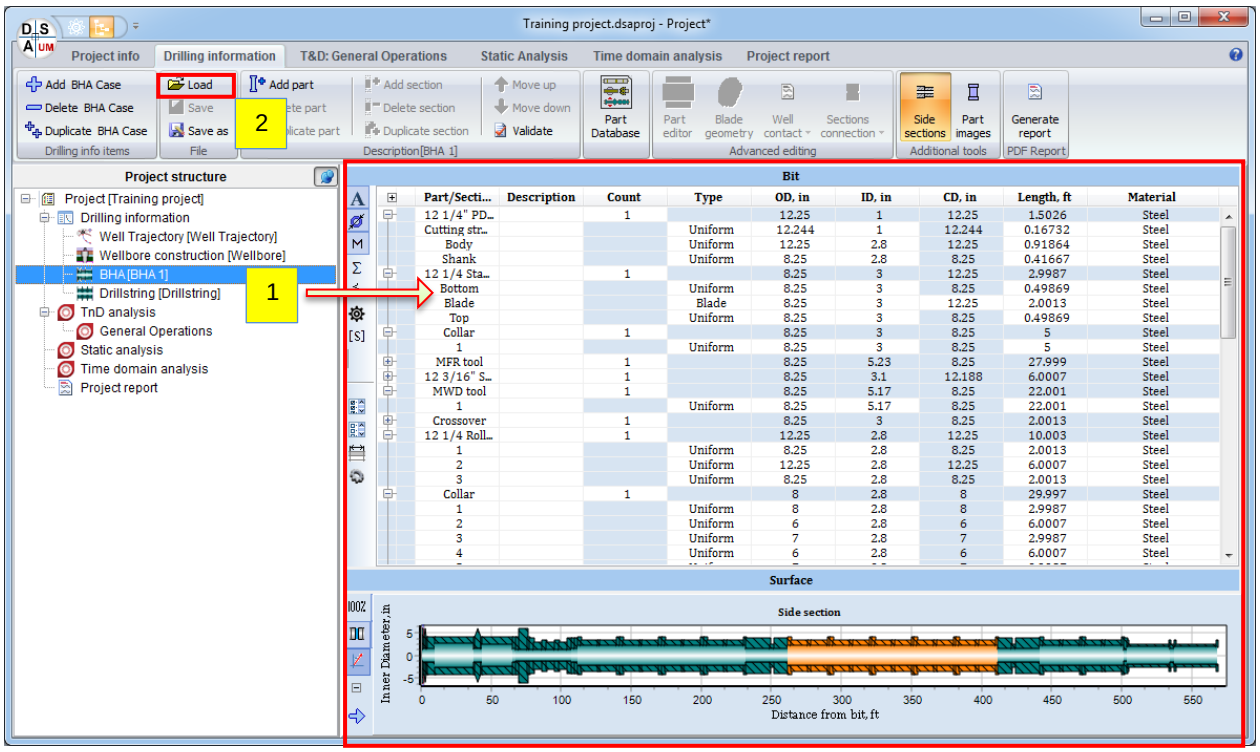
Note. If you've already input some data to the table program will ask you to save it before loading.

Step 3.6. Click the button placed on the head menu. The 'No errors or warnings found' message will appear if the input data is correct.

2.2.3. BHA Cases

Step 3.7. Select the **Drilling Information | BHA** *Project tree* node to describe the bottom hole assembly design.

The interface of **BHA Editor** placed on the page and the internal description of assembly design are described in Sect. 1.4.5 Drillstring / BHA Editor.



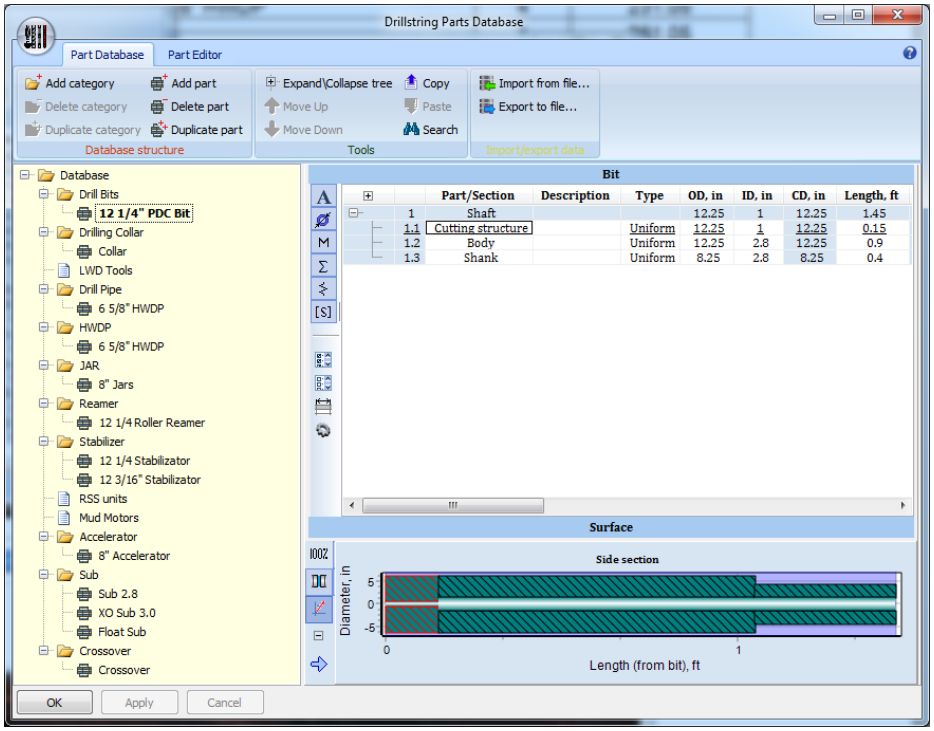
Note. One can load the Sample Rotary BHA model from the *Source directory\drillstrings\Sample Rotary BHA.bha* file with the **Open** button on the head menu, ignore **Steps 3.8 - 3.13** and come to page 72.

2.2.3.1. Rotary BHA description

Let's create rotary assembly consisted of the components listed in the table below:

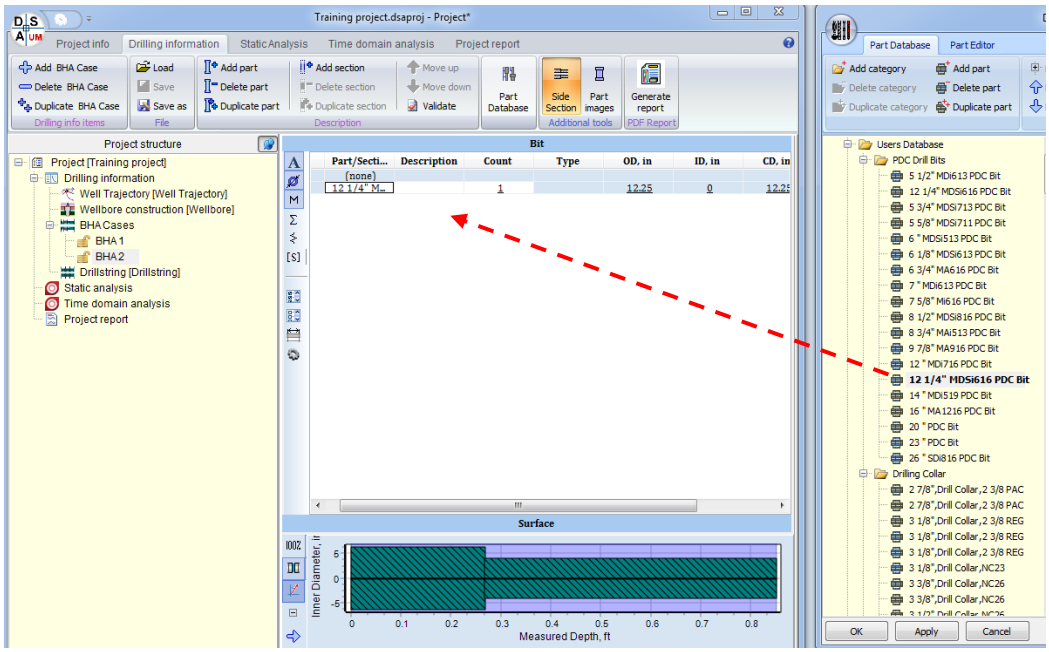
	Description	Count	Acc. length, ft
1	12 1/4" PDC Bit	1	1.45
2	12 Stabilizer	1	4.55
3	Collar	1	9.45
4	MFR tool	1	37.45
5	12 3/16" Stabilizer	1	43.45
6	MWD tool	1	65.45
7	Crossover	1	67.45
8	12 1/4" Roller Reamer	1	77.45
9	Collar	1	107.45
10	Float Sub	1	109.45
11	XO Sub 3.0	1	111.45
12	4 x 6 5/8" HWDP	4	231.05
13	8" Jars	1	261.05
14	5 x 6 5/8" HWDP	5	410.05
15	Accelerator	1	440.06
16	2 x 6 5/8" HWDP	2	500.4
17	XO Sub 2.8	1	502.4
18	5" Drill Pipe	1	533.4
19	Dart Sub	1	536.4
20	5 " Drill Pipe	1	567.4

Step 3.8. Click the **Part Database** button on the head menu to open the database wizard. The delivered database contains all the elements of the assembly.



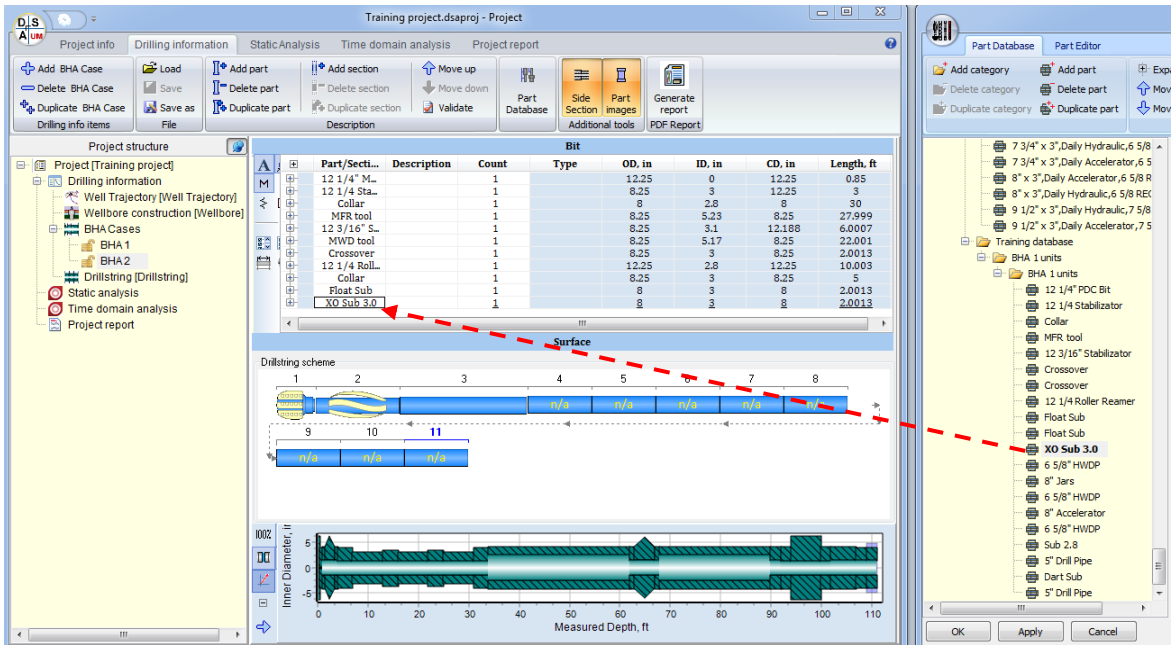
Step 3.9. In **String Part Database** window select **12 ¼ PDC Bit** part from the Drill Bits category and drop it to the *String Editor* table.

The new component will be added to the Rotary BHA case string, PDC **Bit** figure will be shown on the bottom.

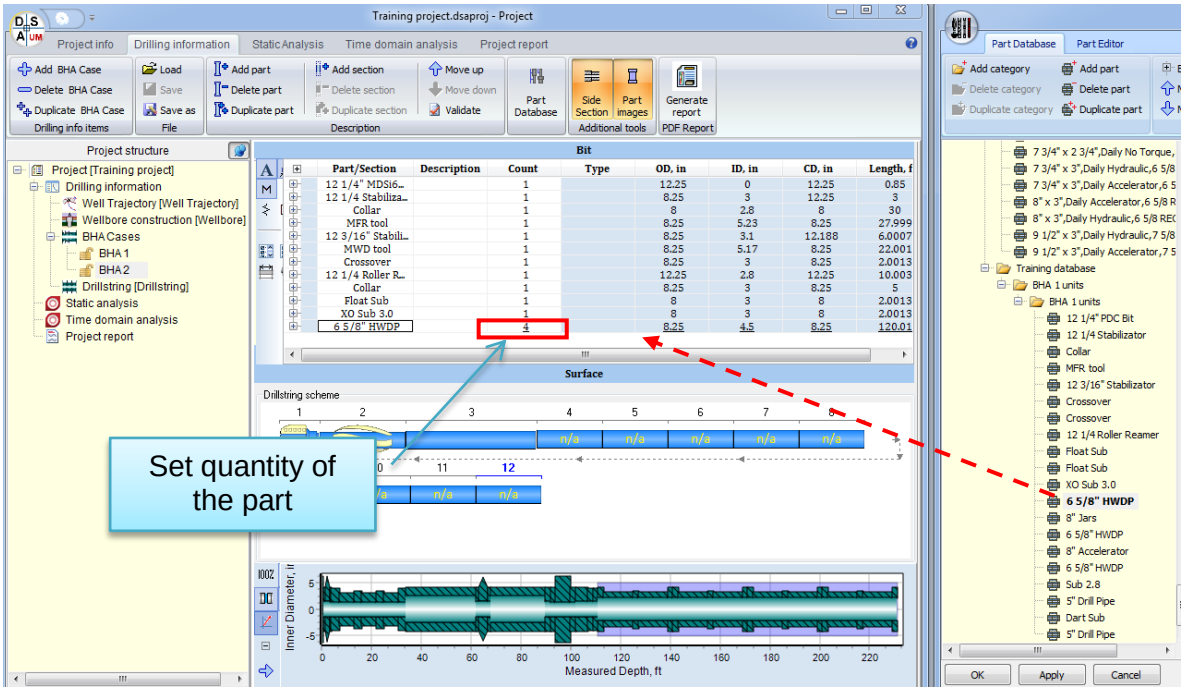


Step 3.10. Add sequentially in the same manner the following components **12¼" Stabilizer**, **Collar**, **MFR tool**, **12 3/16" Stabilizer**, **MWD tool**, **Crossover**, **12 1/4" Roller Reamer**, **Collar**, **Float Sub** and **XO Sub 3.0**.

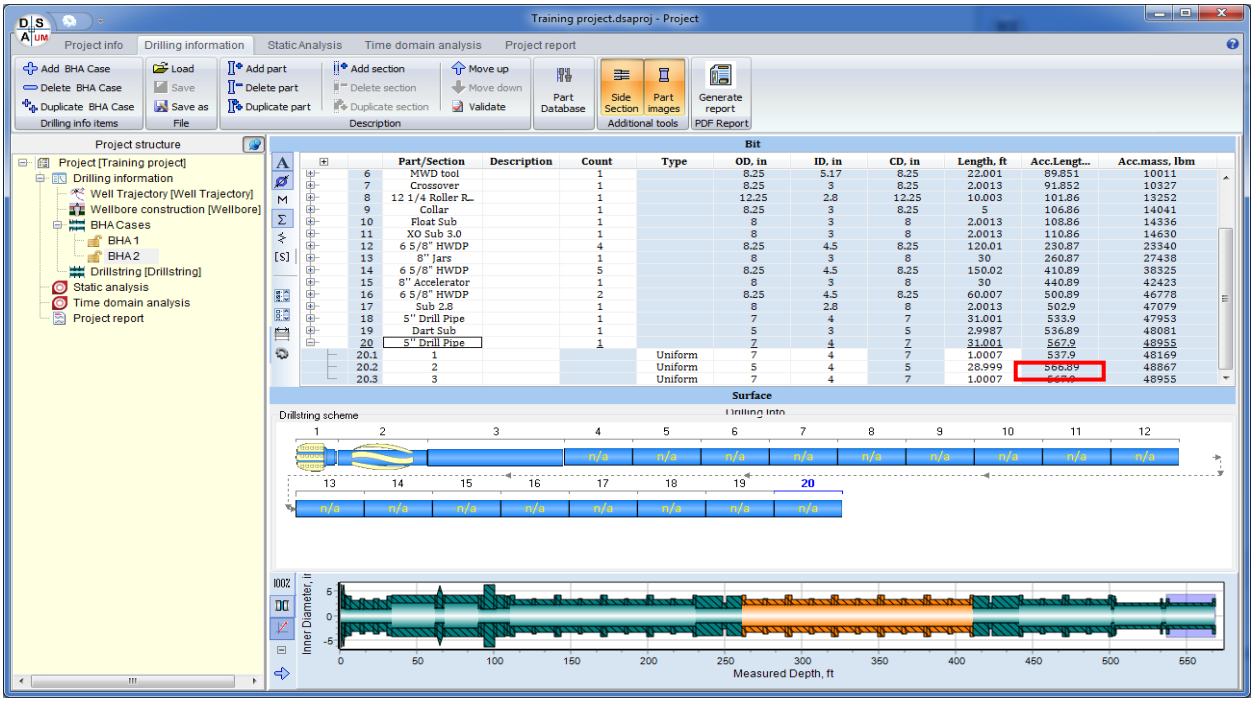
New components are added to the end of the list. Use  buttons on the head menu to change the order of the components if necessary.



Step 3.11. Now we need to add four sections of **6 5/8" HWDP**. We can drag the HWDP component to the table four times, but the easiest way is to add the part single time and set **Count** parameter (four) in the table row.



Step 3.12. Add sequentially the rest components: **8" Jars, 5 x 6 5/8" HWDP, Accelerator, 2 x 6 5/8" HWDP, XO Sub 2.8, 5" Drill Pipe, Dart Sub** and **5"Drill Pipe.**



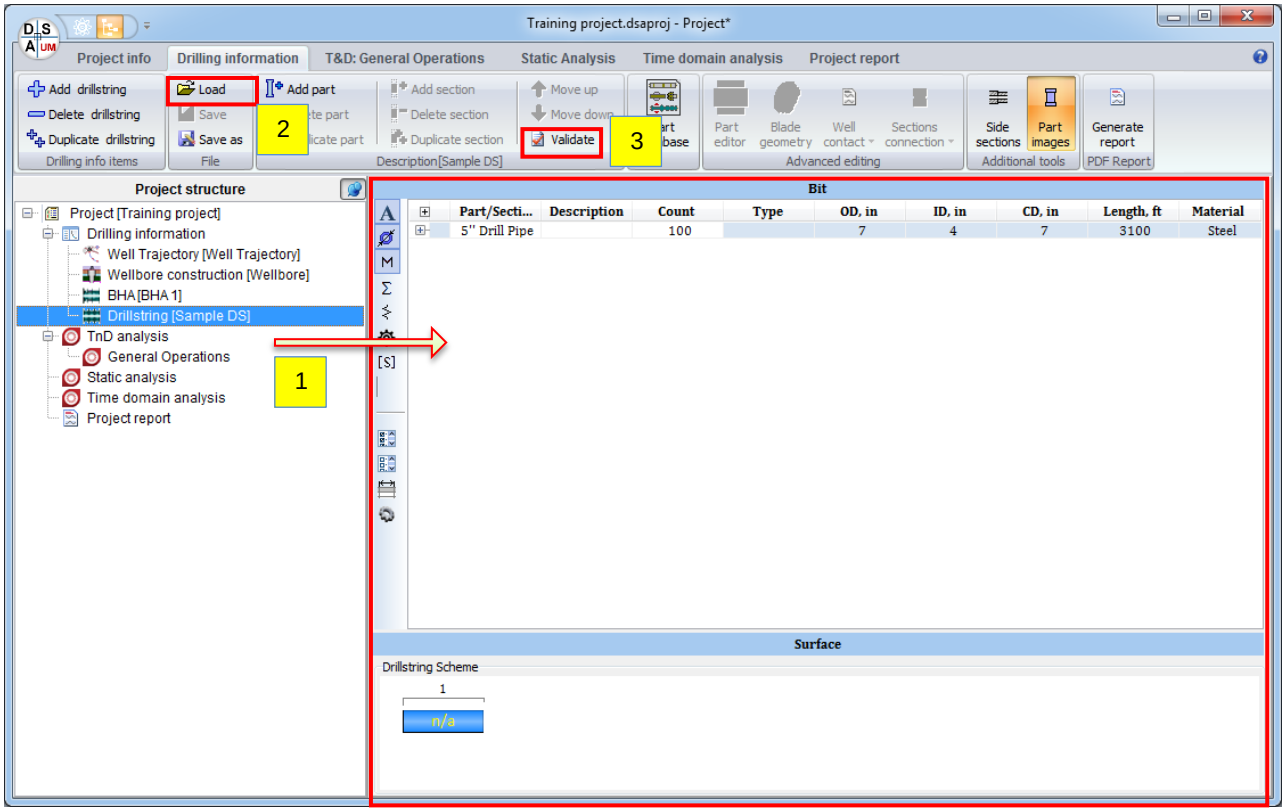
Finally check, that the total length of the Rotary assembly is equal to 567.4 feet.

Step 3.13. Click the button placed on the head menu. The 'No errors or warnings found' message will appear if the input data is correct.

2.2.4. Drillstring Cases

Step 3.14. Select the **Drilling Information | Drillstring** *Project tree* node to describe the bottom hole assembly design.

The interface of **String Editor** placed on the page and the internal description of assembly design are described in Sect. 1.4.5 Drillstring / BHA Editor.



Step 3.15. Click the **Open** button on the head menu to load the drillstring description from *Source directory\drillstrings\Sample Drillstring.drs* file. Loaded data will be displayed on the right hand panel.

Note. If you've already input some data to the table program will ask you to save it before loading.

Step 3.16. Click the button placed on the head menu. The 'No errors or warnings found' message will appear if the input data is correct.

Description of drilling info items for Static and Time Domain Analysis completed.

Step 3.17. Save the Project.

2.3. Static Analysis

The section contains brief overview of the Static Analysis concept, and step-by-step description of Static analysis of the Rotary BHA assembly in the sample well trajectory.

Static analysis overview

DSA application provides set of tools for the study of the equilibrium state of the assembly for the definite position in the well. Time domain simulation is used for evaluation of equilibrium state of the assembly:

- Assembly is placed in the wellbore along the well axis in accordance to the user-defined bit position
- Bit/top constraints and forces/torques are applied to the assembly model.

  Default loading scheme is the following: axial motion of the bit is constraint, axial force – defined as "Weight-on-Bit" – is applied to the top point of the assembly.

  Note: Static scenario boundaries (bit and top constraints of the assembly model) can be modified in **Advanced User** and **Developer** mode.

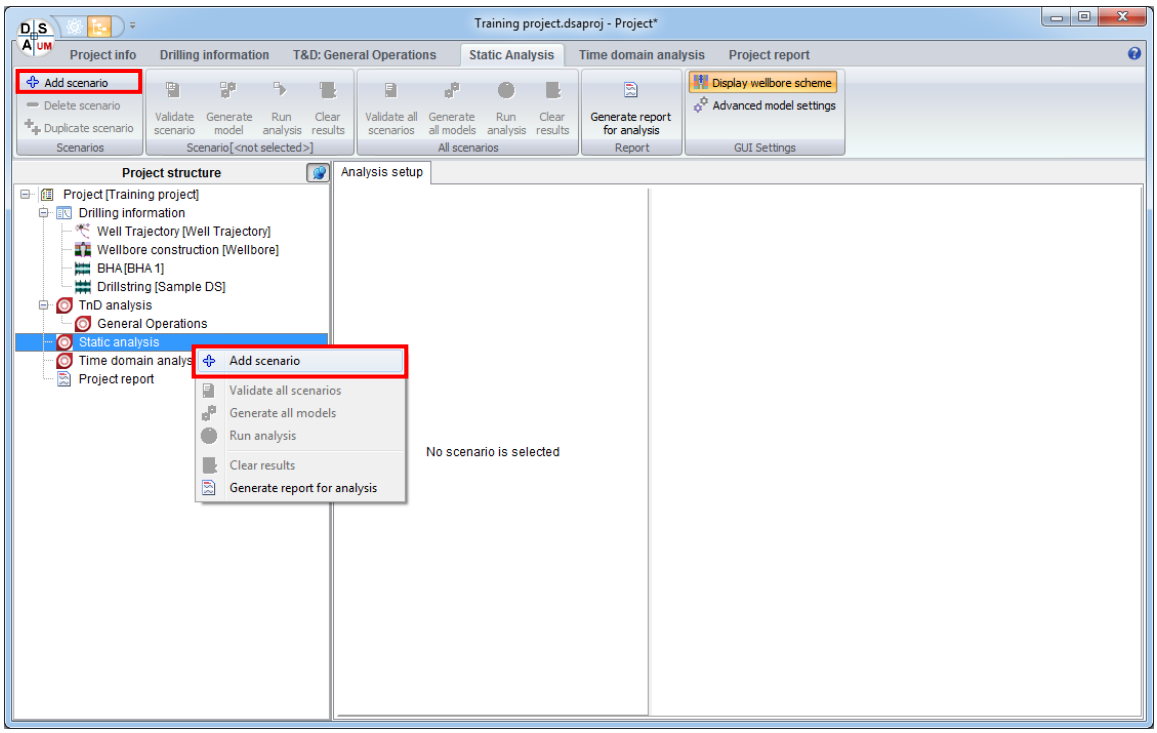
- Time domain simulation of the model motion is started; the model reaches an equilibrium state under action of applied forces and boundaries, contact interaction with wellbore walls, gravity forces and internal/external damping forces.
- Equilibrium state is fixed by the attenuation of kinetic energy of the system.
- Deformed shape of the assembly, distribution of assembly-wellbore contact forces, internal force factors, and stress state of the assembly units at the equilibrium state are evaluated and saved to binary result files.

Static Analysis GUI enables study of any number of test cases – *scenarios* – within single project. Static Analysis scenario (or *Static scenario*) is defined by the following basic inputs:

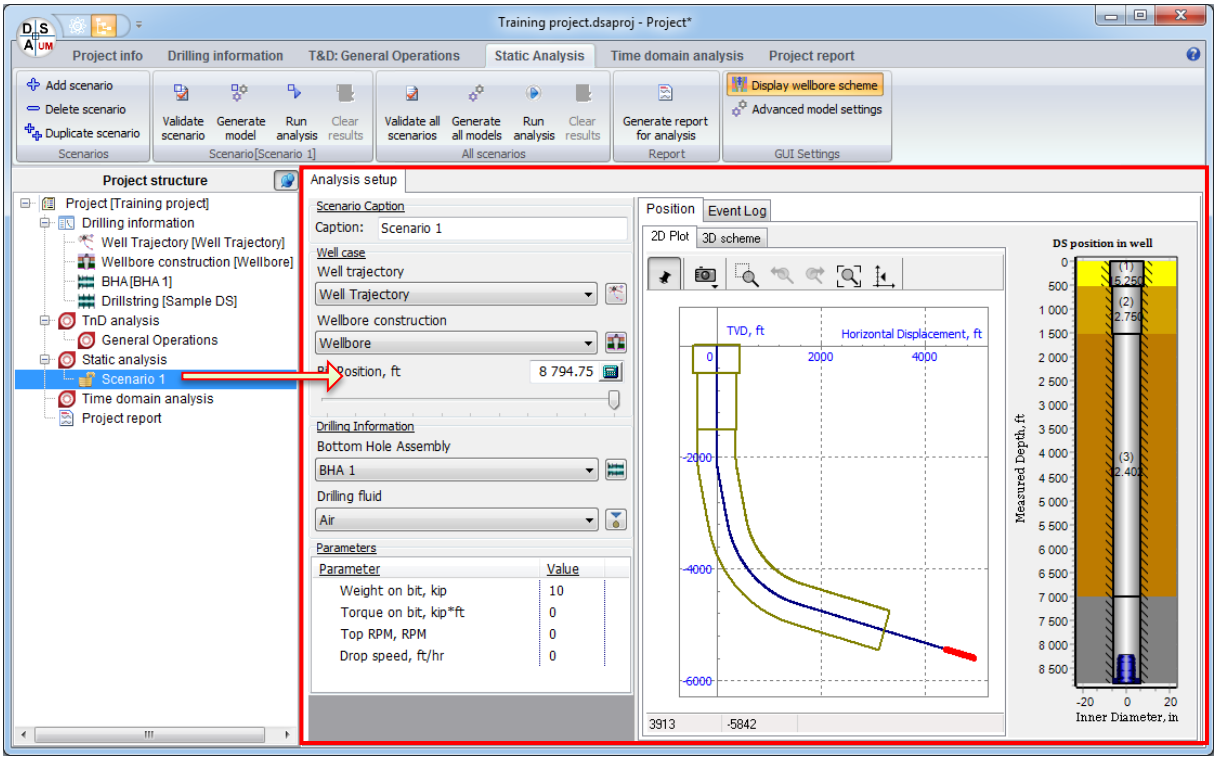
- Well case info: well trajectory case, wellbore case and bit position.
- Drilling Info: BHA case and drilling fluid.
- Operational settings: Weight-on-Bit value, corresponded (by default) to the axial force at the top point of the assembly.
- Set of advanced settings available for the **Advanced User** or **Developer**.

2.3.1. Scenario description

Step 4.1. Select the **Static Analysis** *Project tree* node to access the Static Analysis GUI, and **Add scenario** by popup menu or head menu button.



Scenario 1 test case will be added to the Static Analysis scenario list; the corresponded node will be added to the *Project structure* tree and focused by default; scenario data will be displayed on the right panel, **Analysis Setup** page.



Set the basic parameters of the new scenario by the middle panel controls.

Step 4.2. Set **Training scenario** caption for the scenario. The scenario node name will be update in the *Project tree* automatically.

Step 4.3. Select the **Well trajectory** item from the **Wellpath** list; the list contains all the well trajectories defined in the project.

Note: One can click on button to come to the **Well trajectory** item description page.

Step 4.4. Select the **Wellbore** item from the **Wellbore** list; the list contains all the wellbore designs defined in the project.

Note: One can click on button to come to the **Wellbore** item description page.

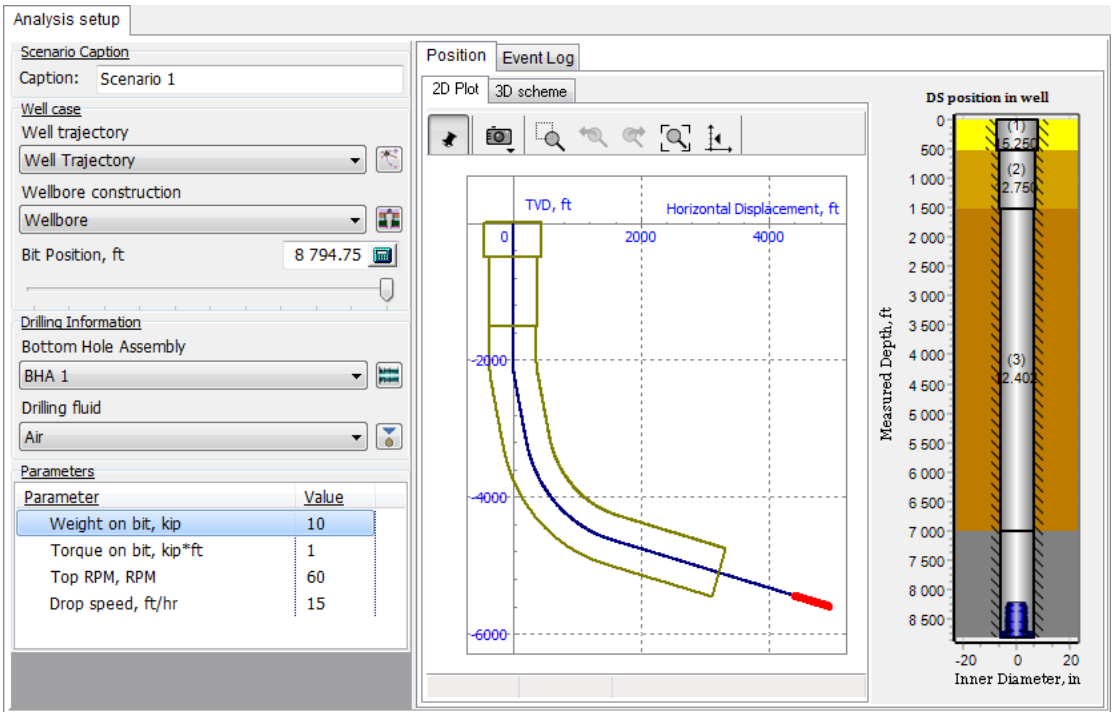
Step 4.5. Set **Bit Position** value equal to 6500 ft.

Step 4.6. Select the **BHA 1** item from the **Bottom Hole Assembly** list; the list contains all the bottom hole assembly designs defined in the project.

Note: One can click on button to come to the **BHA 1** item description page.

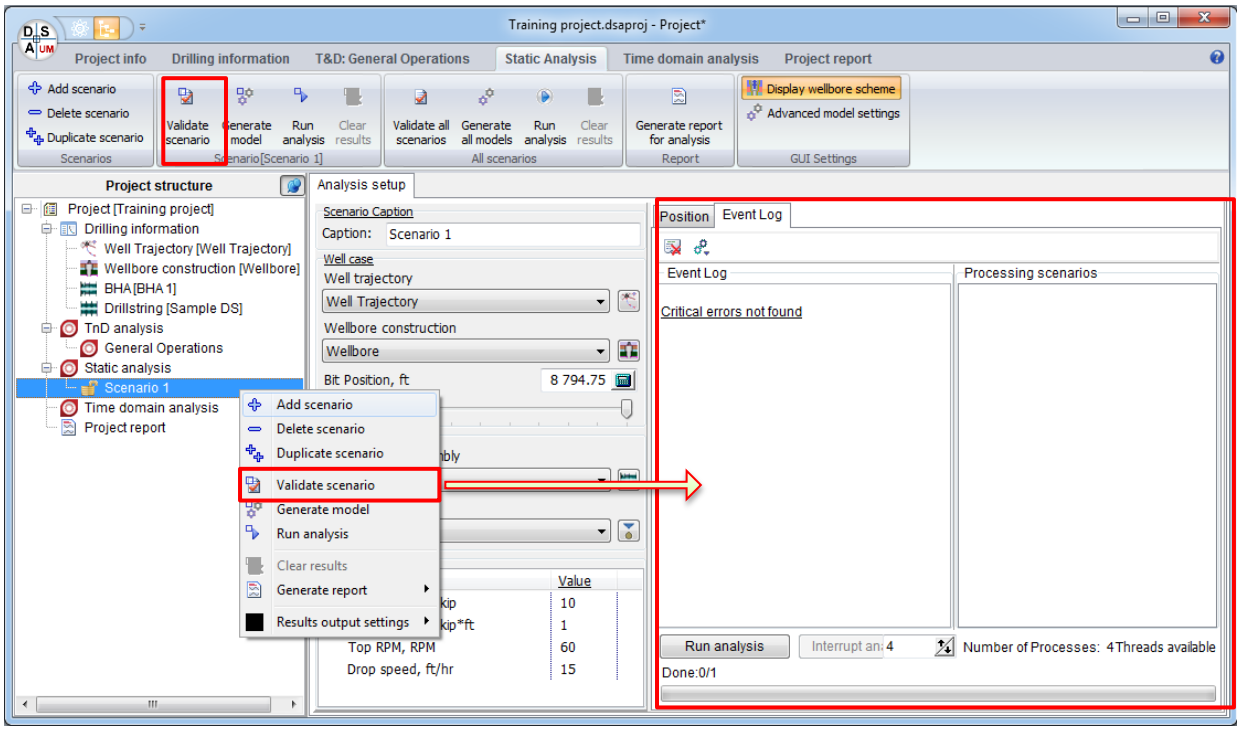
Step 4.7. Select the **Drilling fluid 1** item from the **Fluid** list; the list contains all the fluids from the Fluids Database (see Sect.1.4.1.3).

Step 4.8. Set **Weight-on-Bit (WOB)** value equal to **10 kip**; **Torque on bit** – 1 kip*ft; **Top RPM** – 60rpm; **Drop speed** – 15 ft/hr.



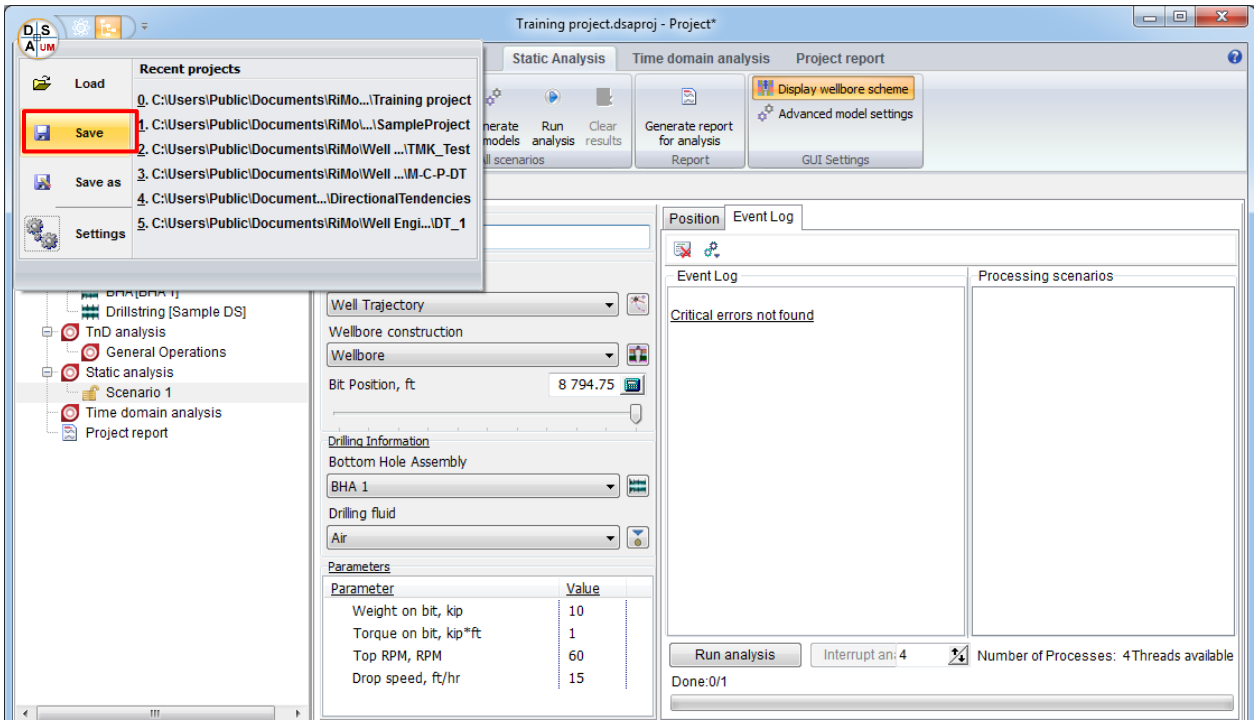
Step 4.9. Click on **Validate Scenario** button in the *Project tree* popup menu, or the head menu to validate the description of the scenario.

Event Log page will be displayed automatically on the right panel; the 'No errors or warnings found' message will appear if the input data is correct.



Training Scenario is described, validated and ready for running now.

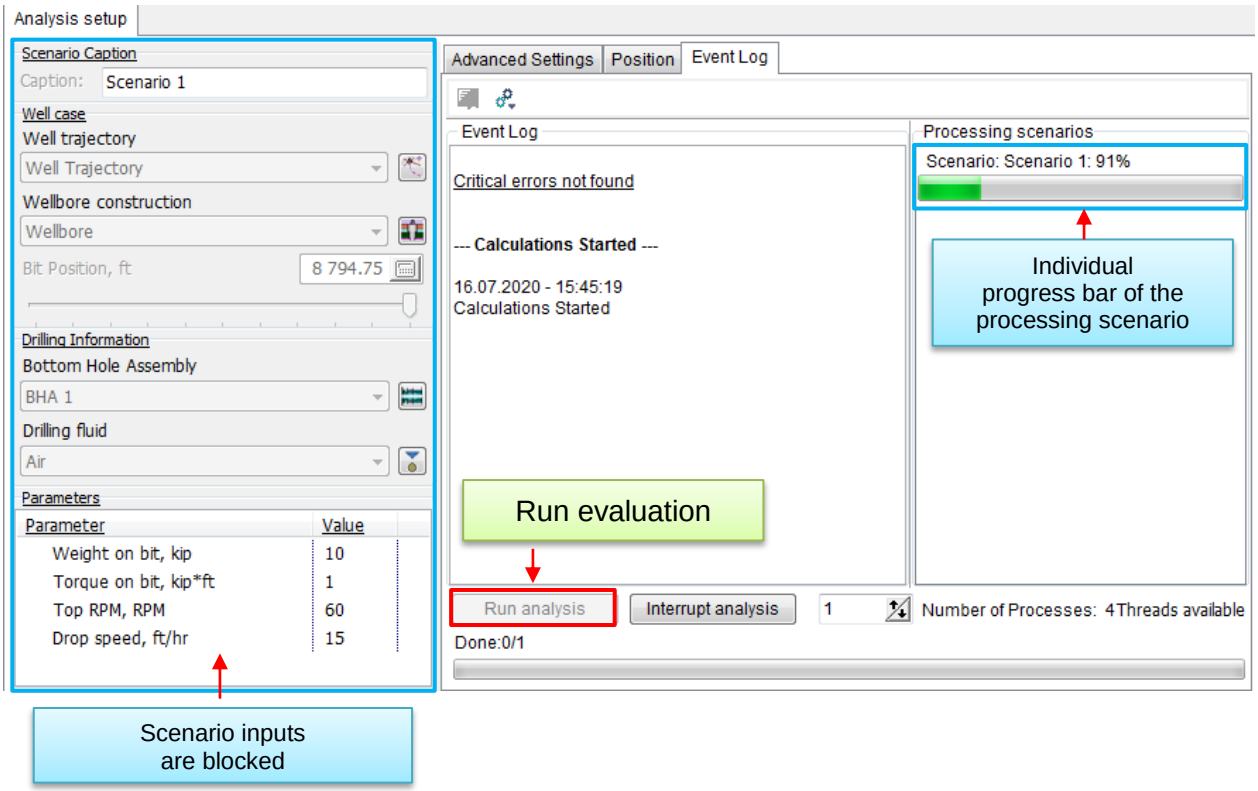
Step 4.10. Save the **project**.



2.3.2. Analysis run

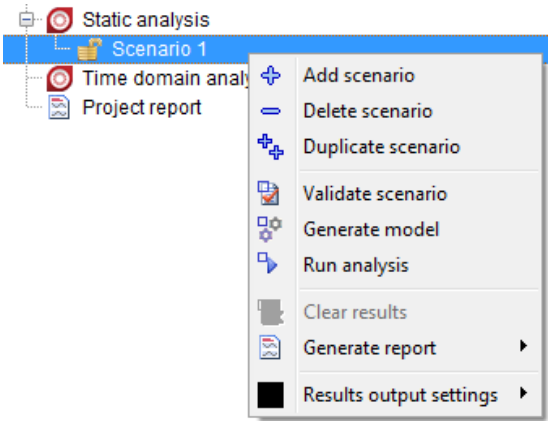
Step 4.11. Select **Event Log** page, and click on **Run Analysis** button to start the evaluation of the described scenario.

Individual progress bar will be added to the right **Running Scenarios** panel; **"--- Calculations Started ---"** record will be added to the **Event Log**; scenario inputs will be blocked automatically.

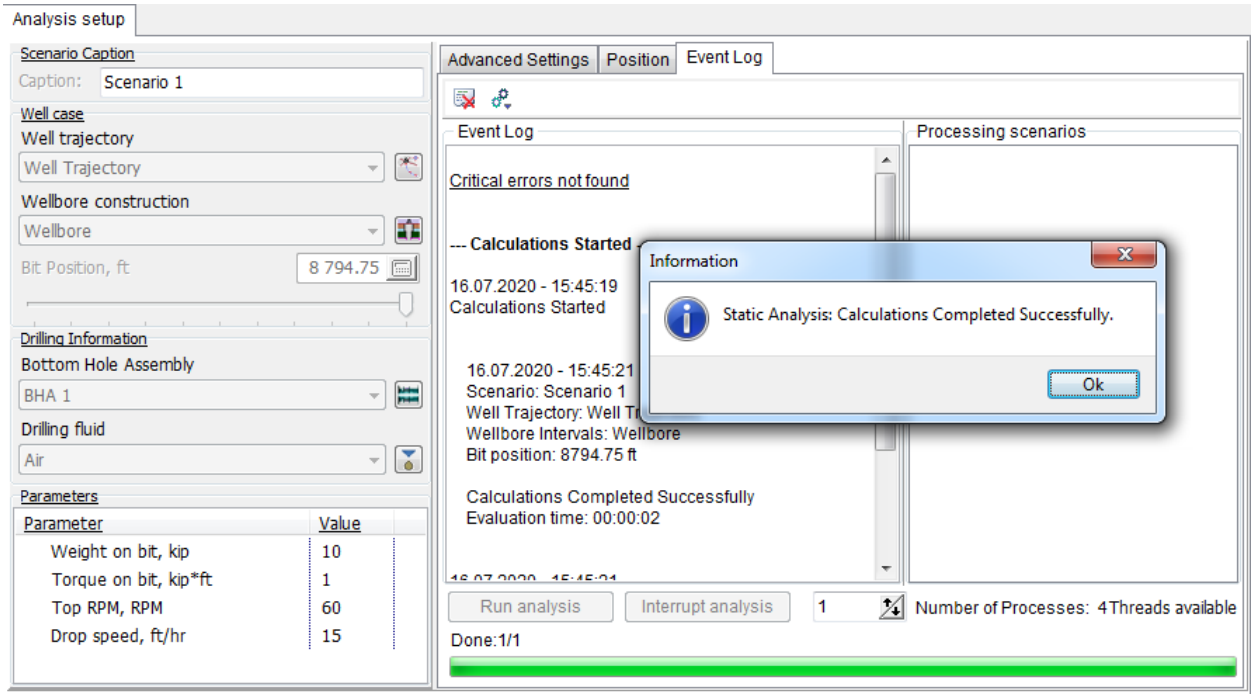


Note: Input data validation, and project saving is carried out automatically before process is running.

Note: One can use **Run all scenarios** button from the *Project tree* popup menu or head menu to start the evaluation; the **Run scenario** button enables run of the selected scenario only.



Scenario progress bar displays the current progress of the evaluation procedure. The progress is closed after completion of the scenario calculation; the scenario analysis details are added to the **Event Log**.

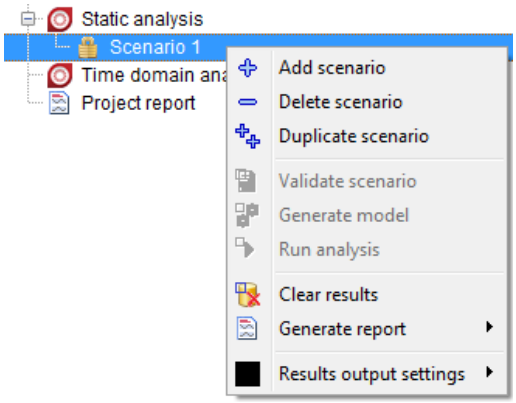


'Static Analysis Calculations Completed Successfully' message will be generated after the completion of all running scenarios (single scenario in our case). Normally, calculation of the scenario should take about 3-10 seconds.

Step 4.12. Click **Ok** to close the message.

Results for the scenario has been calculated and stored in the scenario folder on hard drive; calculated scenarios are marked by icon in the Project tree.

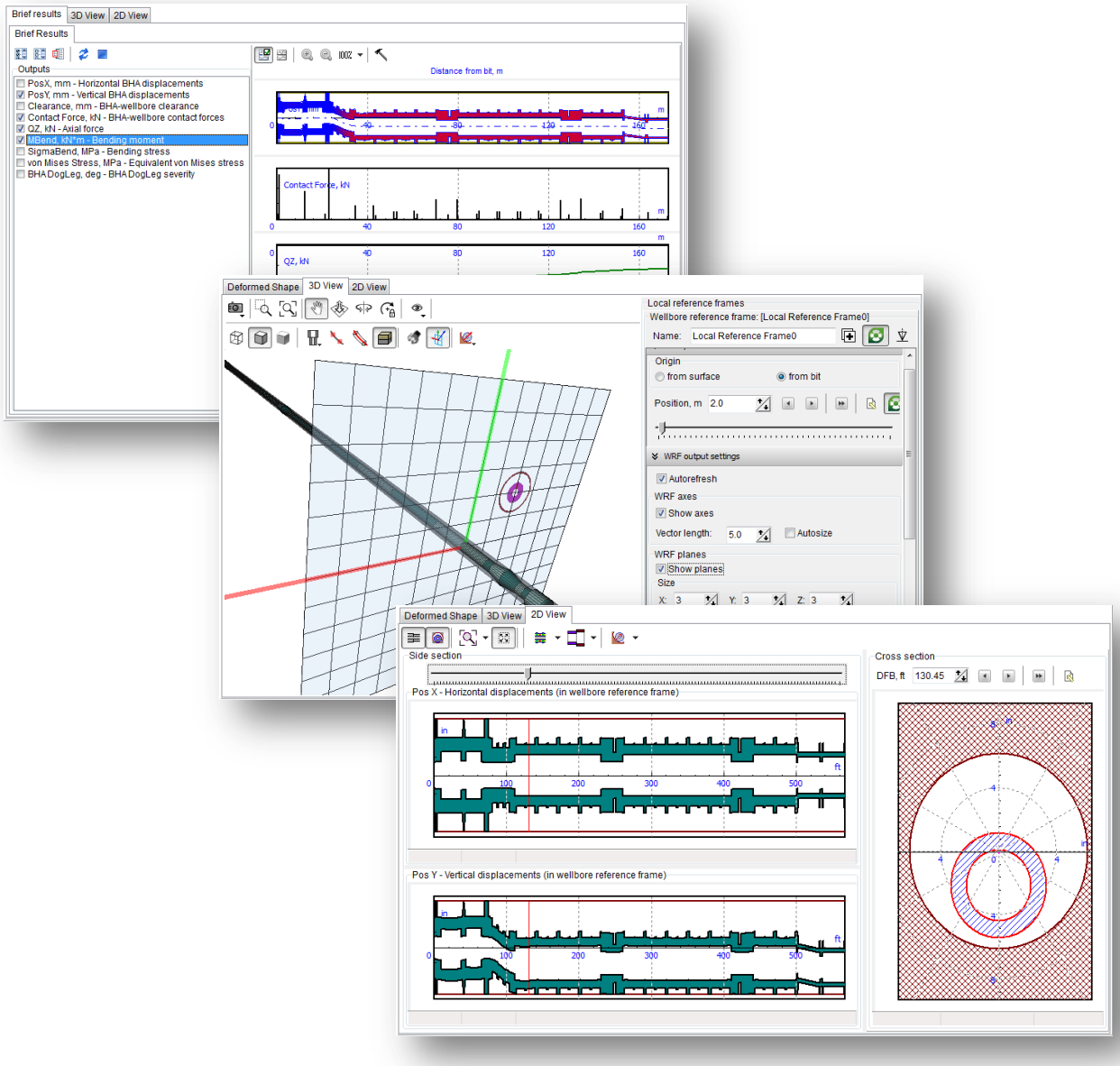
Step 4.13. Select the **Training Scenario** node in the Project tree, and open pop up menu. The **Clear results** option is available now; one needs to clear the results to modify description of the scenario. Close the pop up.



2.3.3. Results output

DSA GUI provides the various tools for 2D and 3D graphical output of the calculated deformed shape of the assembly, distributions of contact forces, internal force factors and stresses along the assembly:

- Brief results: set of 2D plots of the various result items on the single page.
- 3D View: 3D visualization of the assembly model in the wellbore.
- 2D View: 2D straightened side section / cross section view of the deformed shape of the assembly in the hole.

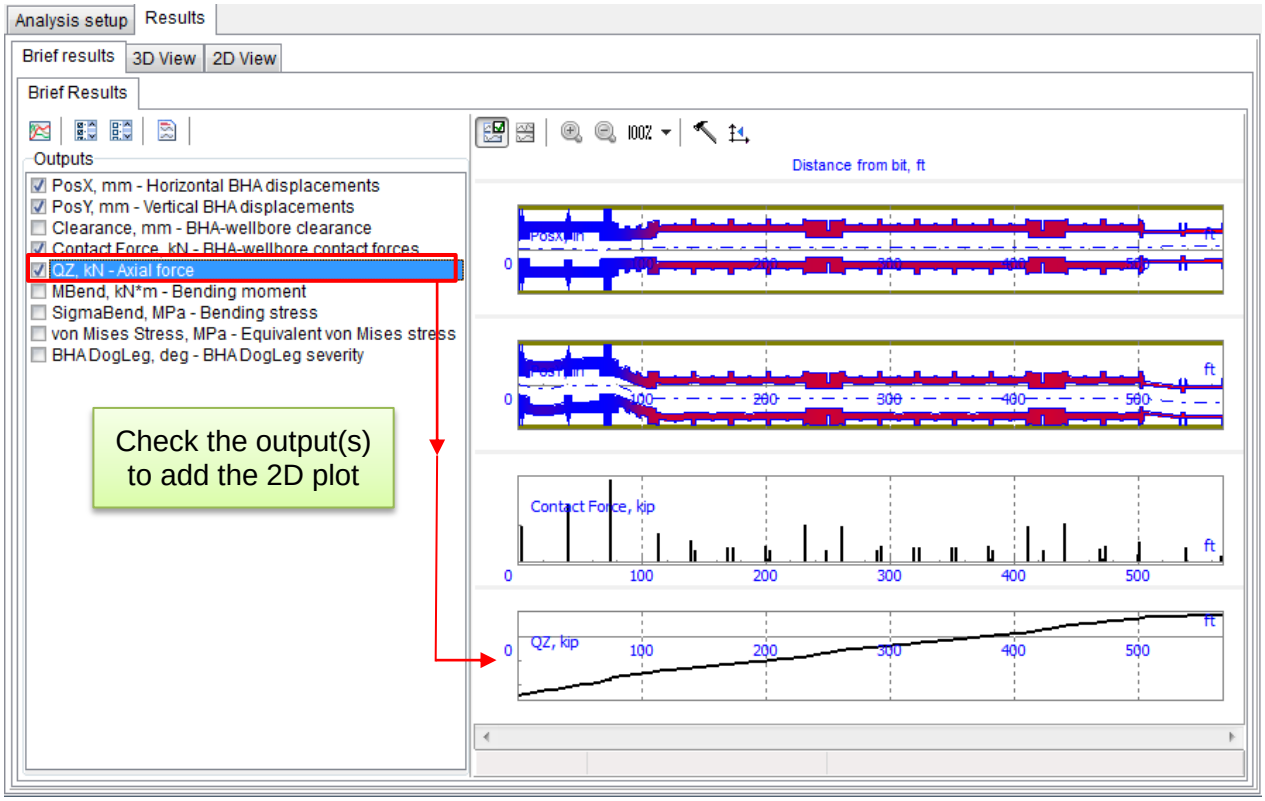


Step 4.14. Select the calculated scenario node (**Training scenario**) in the *Project tree* to make the **Results** page on the right page control visible, and go to the page.

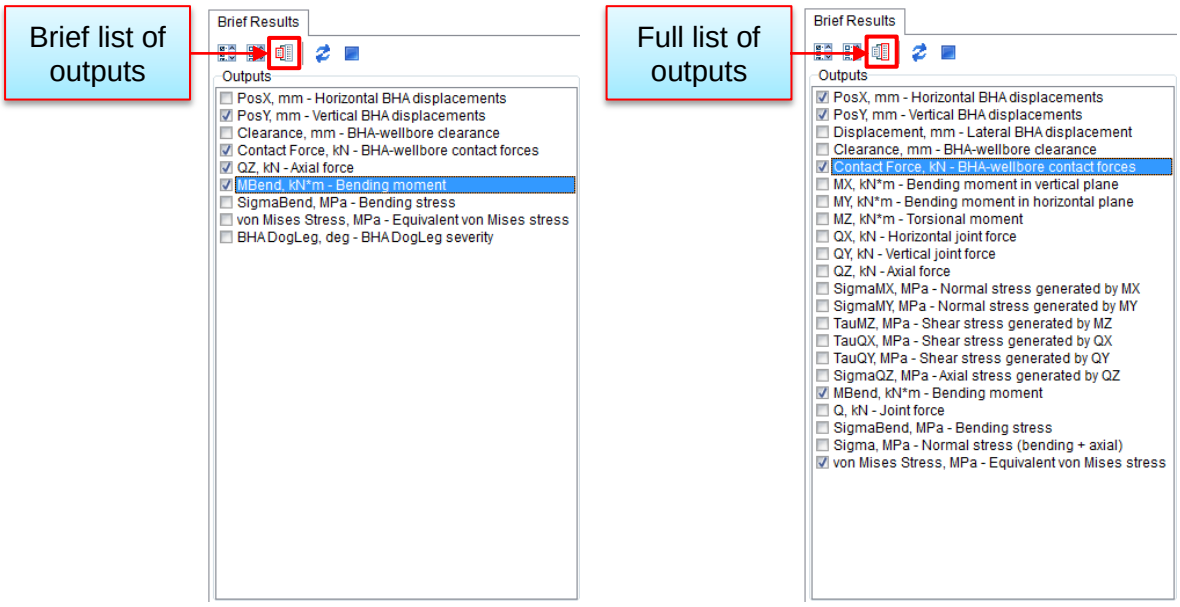
2.3.3.1. Brief results

Step 4.15. Select **Results** | **Brief Results** page to display the set of 2D plots of the various result items.

One can select an output from the check box list – the corresponded dependency of the selected output vs. distance from bit will be displayed on the 2D plot on the right panel.

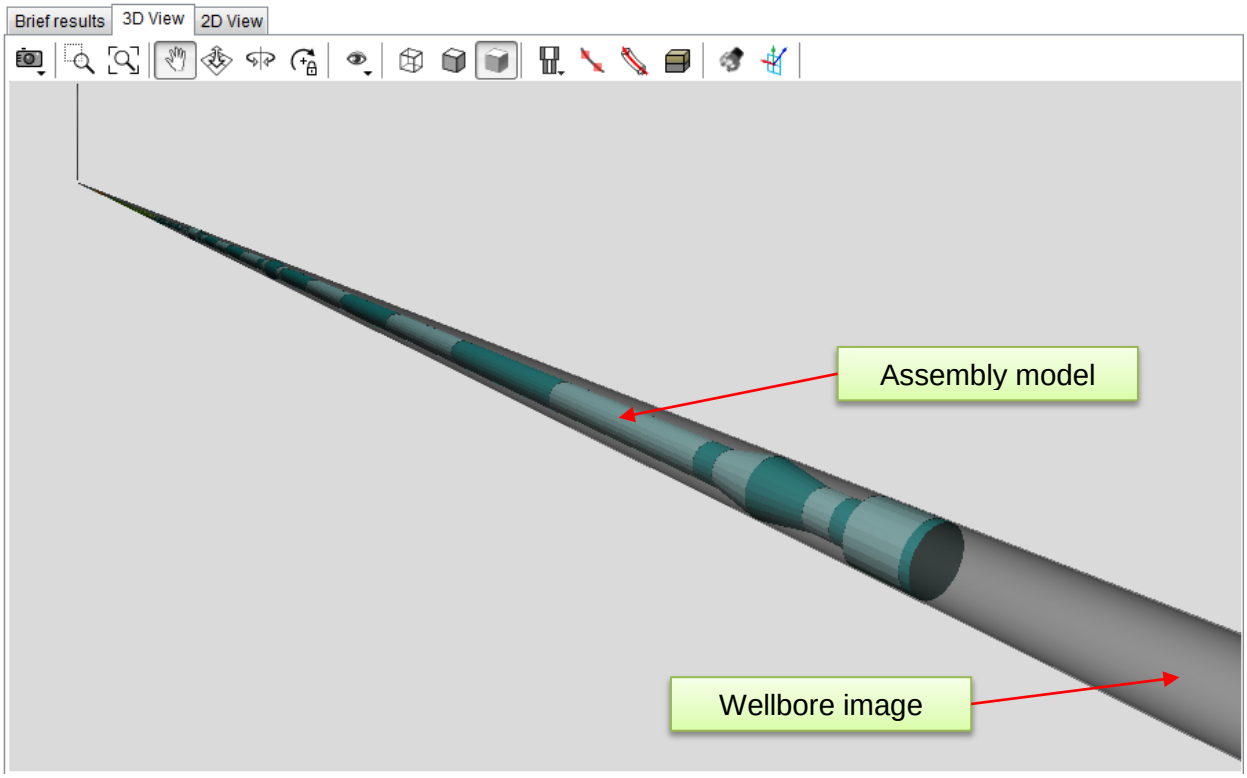


Step 4.16. Click on  button to switch between brief and full list of outputs.



2.3.3.2. 3D View

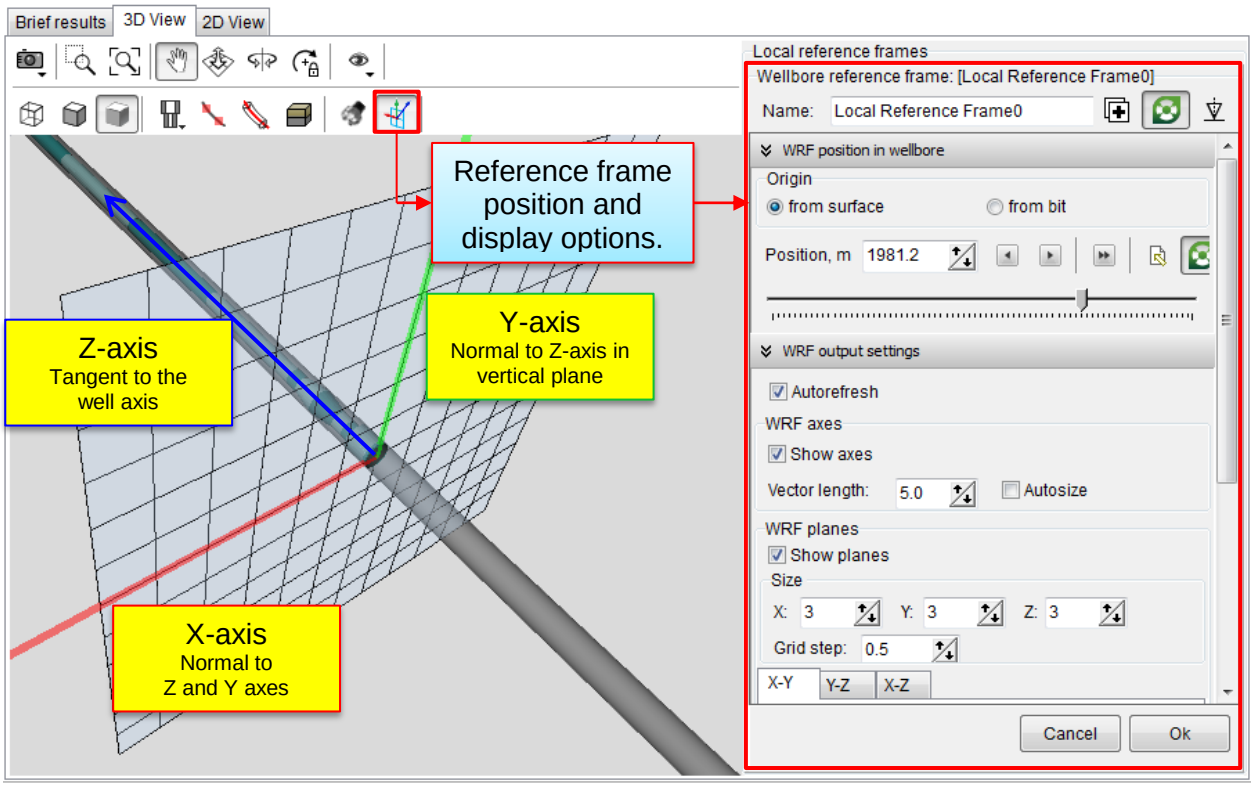
Step 4.17. Select **Results** | **3D view** page to study the 3D position of the assembly in the well.



On can turn on and save the 3D View by the set of options:

- - make the screen short of the 3D view;
- - zoom by rectangle (by mouse);
- - set full view;
- - move camera by mouse;
- - scale by mouse up/down move;
- - rotate by mouse;
- - fix rotation point;
- - select one of the standard views;
- - display edges only;
- - display edges and faces;
- - display faces only;
- - select the wellbore display mode;
- - display survey stations;
- - display well axis line;
- - display geo data;
- - set camera on bit;
- - display the wellbore reference frames;

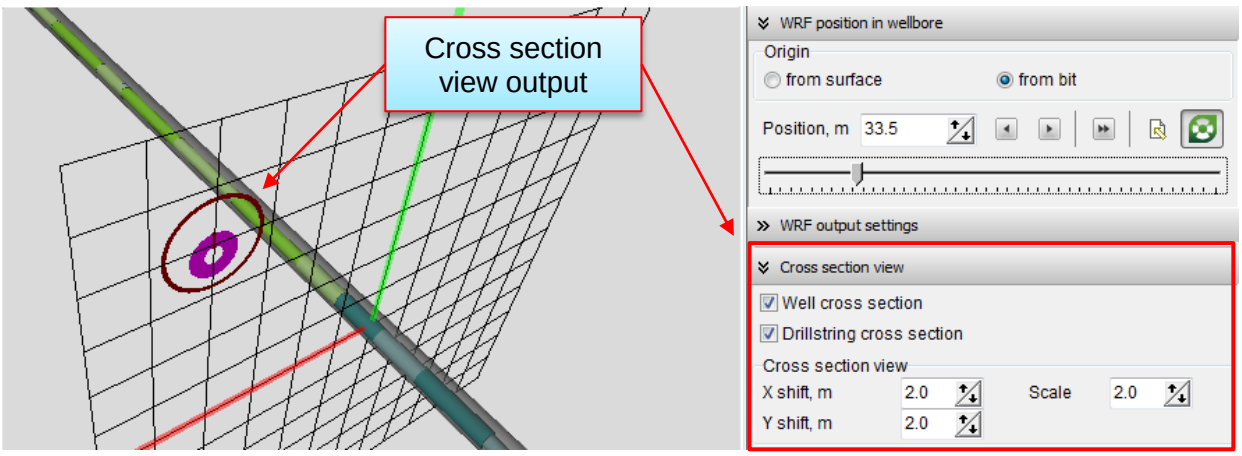
Step 4.18. Click on button to display the frame image on the 3D view and add the wellbore reference frame wizard on the right of the page.



The **Local reference frames** tool enables specifying on any number of local reference frames along the hole/assembly, and wide range of display settings of the frames.

The frame position can be defined by **Distance from surface** or **Distance from bit**; the user can show/hide the frame axes, grids and planes, select its size and coloring. One can collapse the unused panel from the wizard.

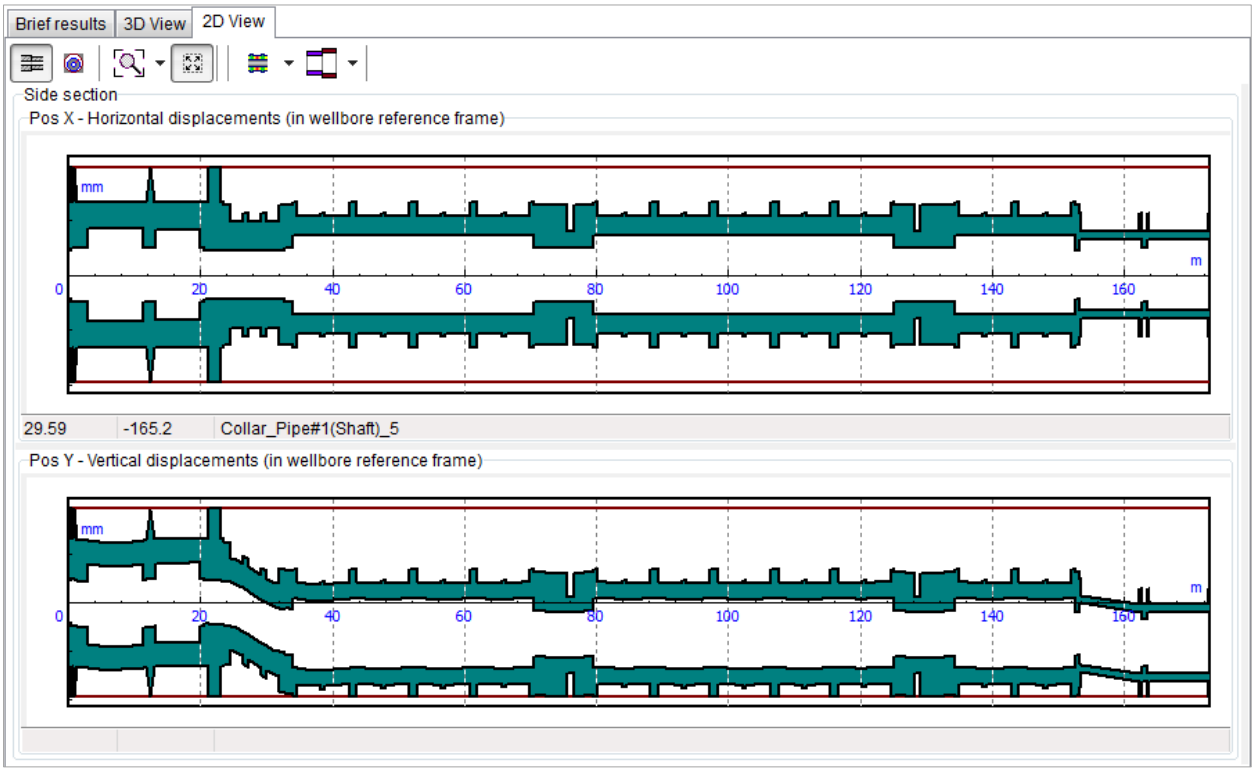
Step 4.19. Select the **Cross section view** panel, enable **Well cross section** and **Drillstring cross section** to display the scaled cross section view; change the **Position** to study local position of the assembly in the well.



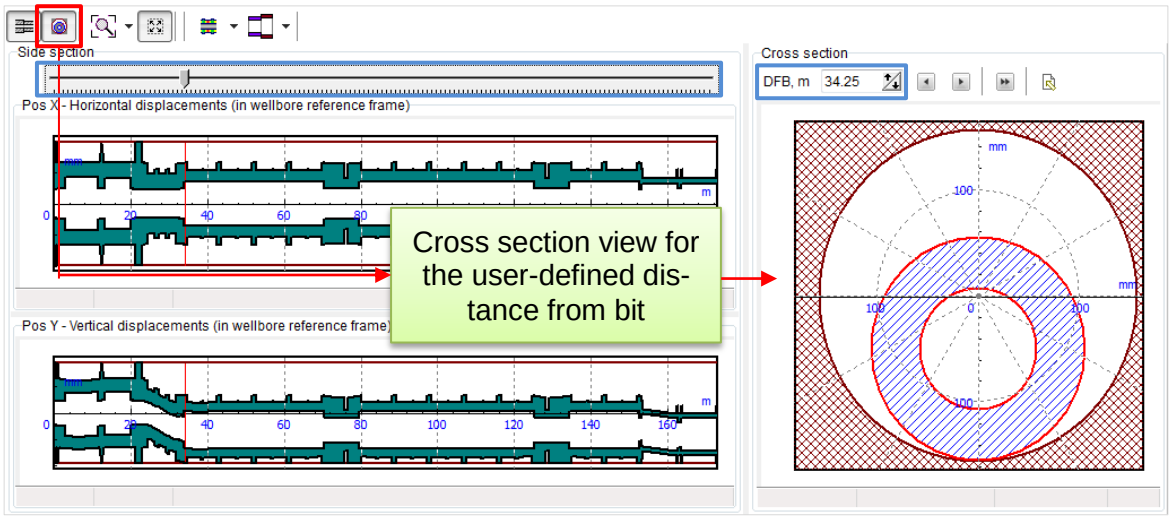
2.3.3.1. 2D View

Step 4.20. Select **Results | 2D view** page to study the 2D Side / cross section view of the assembly in the well.

The well axis is straightened; **PosX** plot corresponds to the displacement of the assembly points in the Z-X wellbore reference plane (reference frame orientation is calculated at each point) – and can be considered as lateral-horizontal displacement of the assembly in the well; **PosY** - corresponds to the lateral-vertical displacement of the assembly in Z-Y wellbore reference frame.



Step 4.21. Click the button to display the cross section of the well on the right of **PosX**, **PosY** plots. Use slider or input **Distance from Bit** manually to define the cross section position.



2.4. Torque & Drag Analysis

The section contains brief overview of the Torque & Drag Analysis concept, and step-by-step description of T&D analysis of the assembly combined from Rotary BHA and Sample DS assemblies.

Torque & Drag analysis overview

DSA application provides set of tools for the estimation of drag forces and resistance torque distribution along the drillstring for standard set of technical operations on rig within soft-string and stiff-string model of an assembly.

- ✓ **Soft-string** analysis is the fast solution based on assumption of positioning of assembly along the well trajectory.
  - Drillstring state is estimated analytically by numerical integration of flexible curved beam equilibrium state differential equations; by fact the procedure can be described considered as sequential solution of equilibrium state problem for short drillstring sections from bit to surface.
  - Boundary conditions – axial force and torque – are defined at bit point in accordance to selected operation type.
  - Contact forces are estimated from the bending of the drillstring section taken from the well curvature at point; friction forces and resistance torque on section are estimated from assembly-wellbore friction coefficient and the considering combination of tripping and section rotation.
  - Axial force at the top point of section is evaluated from the balance of gravity forces and friction forces.
  - Increasing of resistance forces by sinusoidal and helical buckling effects are considered from the conventional analytical equations.
  - Drag forces and resistance torque, contact forces, internal force factors and stress state of the assembly components, as well as elongation of the assembly are estimated.
  - Effects on temperature elongation of the assembly units can be considered optionally by account of temperature map.
  - Internal / external pressure effects on the resultant stress state can be considered optionally by account of pressure distribution.

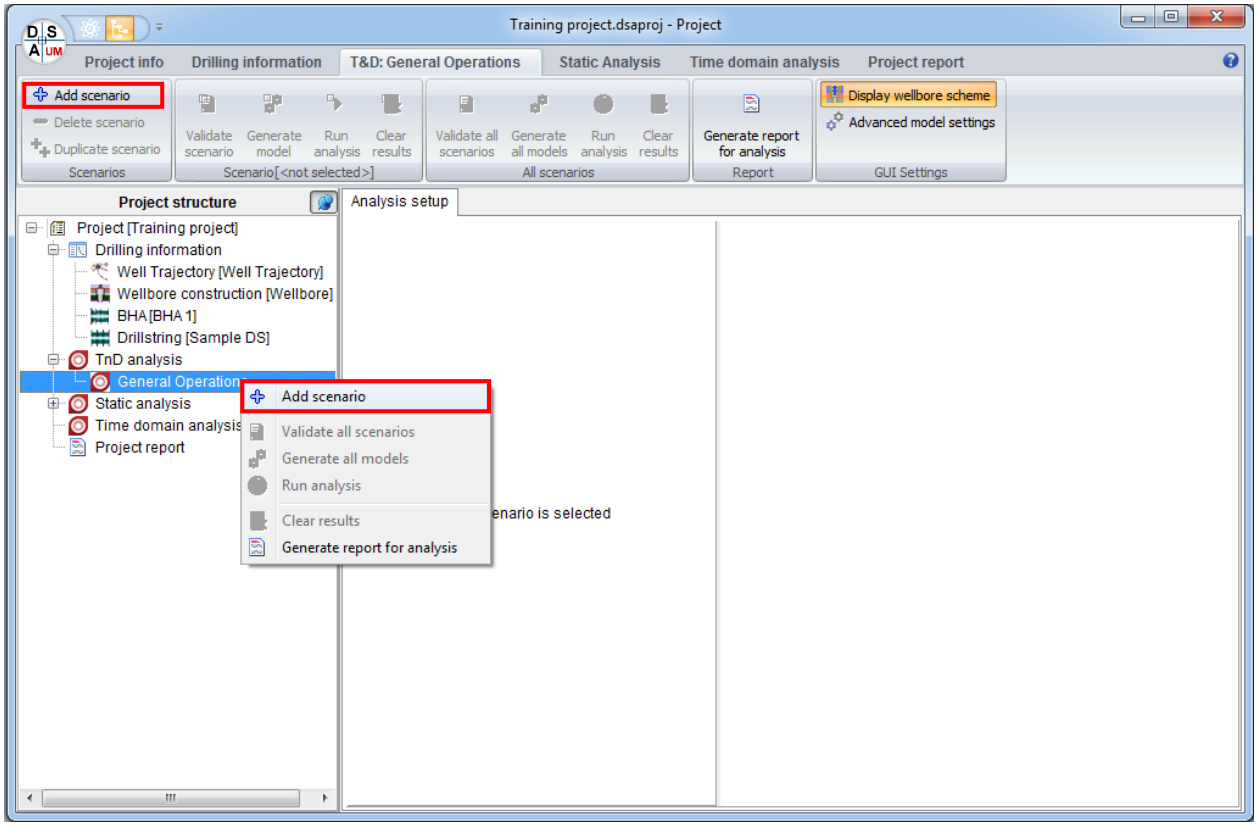
- ✓ **Stiff-string** analysis is the detailed analysis considering local deformations and lateral displacements of assembly units in the borehole; application of Static Analysis approach (Sect.2.3) for the whole drillstring.

Torque & Drag Analysis GUI enables study of any number of test cases – *scenarios* – within single project. Torque & Drag Analysis scenario (or *T&D scenario*) is defined by the following basic inputs:

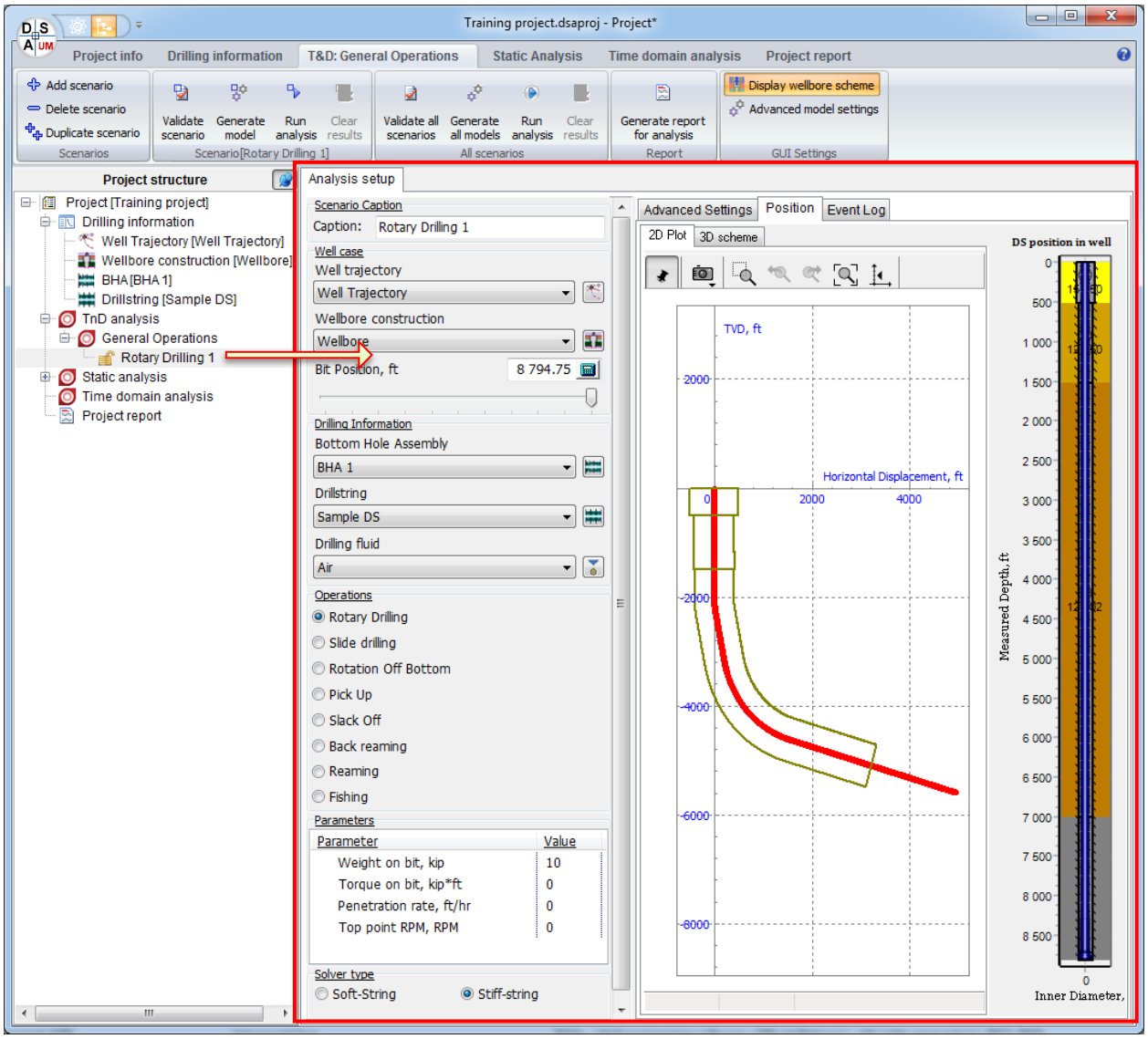
- Well case info: well trajectory case, wellbore case and bit position.
- Drilling Info: BHA case, Drillstring case and drilling fluid.
- Operational settings: type of technical operation and corresponded parameters (Weight On Bit, Torque on Bit, Penetration rate, Surface RPM, etc.).
- Set of advanced settings: Temperature map and Pressure distribution data.
- Advanced model settings available for the **Advanced User** or **Developer**.

2.4.1. Scenario description

Step 5.1. Select the **TnD Analysis | General Operations** *Project tree* node to access the analysis GUI, and **Add scenario** by popup menu or head menu button.



Rotary drilling 1 test case (default naming is taken from operation caption) will be added to the TnD Analysis | General Operations scenario list; the corresponded node will be added to the *Project structure* tree and focused by default; scenario data will be displayed on the right panel, **Analysis Setup** page.



Set the basic parameters of the new scenario by the middle panel controls.

Step 5.2. Select the **Well trajectory** item from the **Wellpath** list; the list contains all the well trajectories defined in the project.

Note: One can click on  button to come to the **Well trajectory** item description page.

Step 5.3. Select the **Wellbore** item from the **Wellbore** list; the list contains all the wellbore designs defined in the project.

Note: One can click on  button to come to the **Wellbore** item description page.

Step 5.4. Set **Bit Position** value equal to 6500 ft.

Step 5.5. Select the **BHA 1** item from the **Bottom Hole Assembly** list; the list contains all the bottom hole assembly designs defined in the project.

Note: One can click on  button to come to the **BHA 1** item description page.

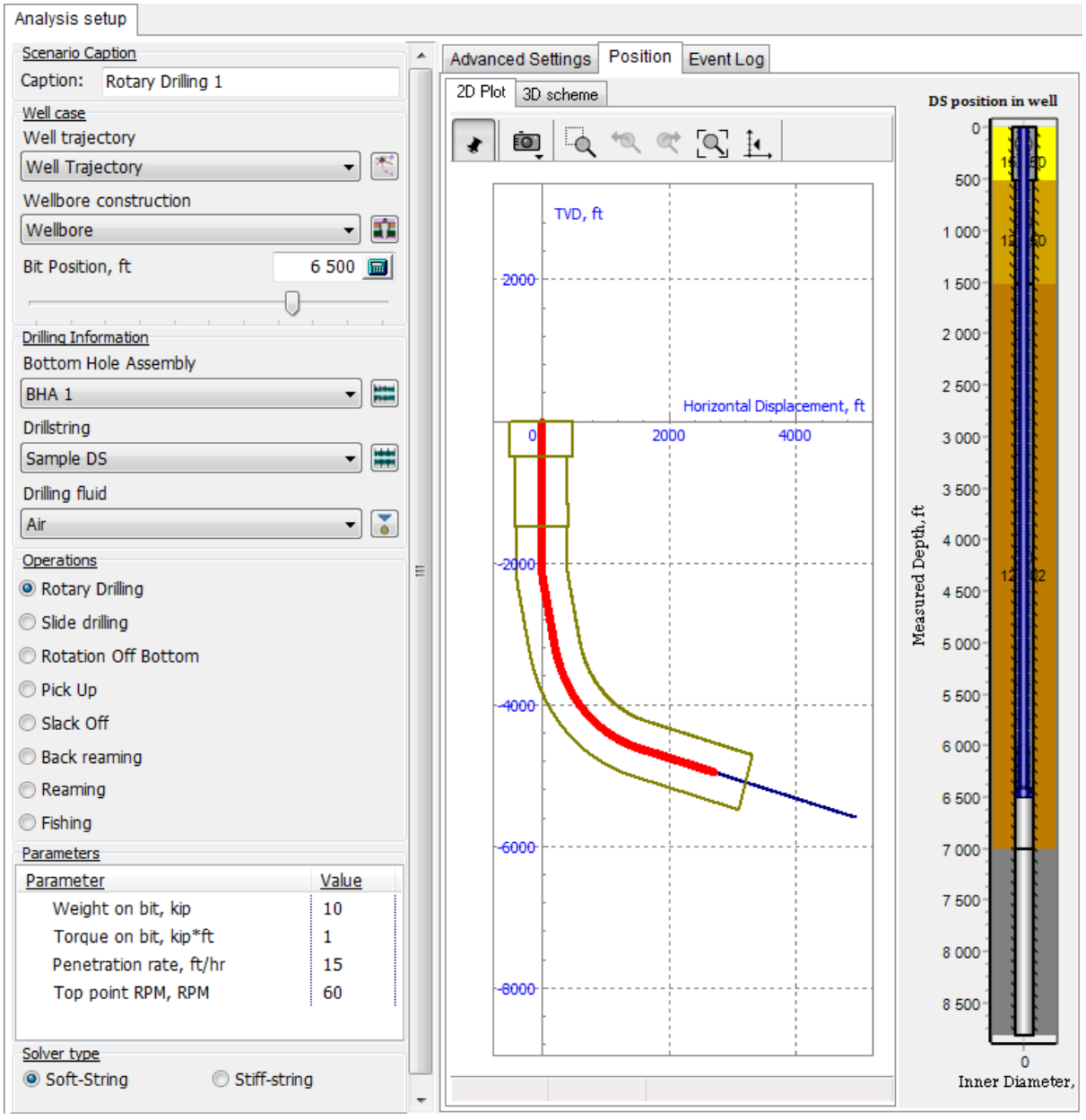
Step 5.6. Select the **Sample DS** item from the **Drillstring** list; the list contains all the Drillstring cases designs defined in the project.

Note: One can click on button to come to the **Sample DS** item description page.

Step 5.7. Select the **Drilling fluid 1** item from the **Fluid** list; the list contains all the fluids from the Fluids Database (see Sect.1.4.1.3).

Step 5.8. Select Rotary Operation scenario, and define parameter values: **Weight-on-Bit - 10 kip**; **Torque on bit** – 1 kip*ft; **Top RPM** – 60rpm; **Penetration Rate** – 15 ft/hr.

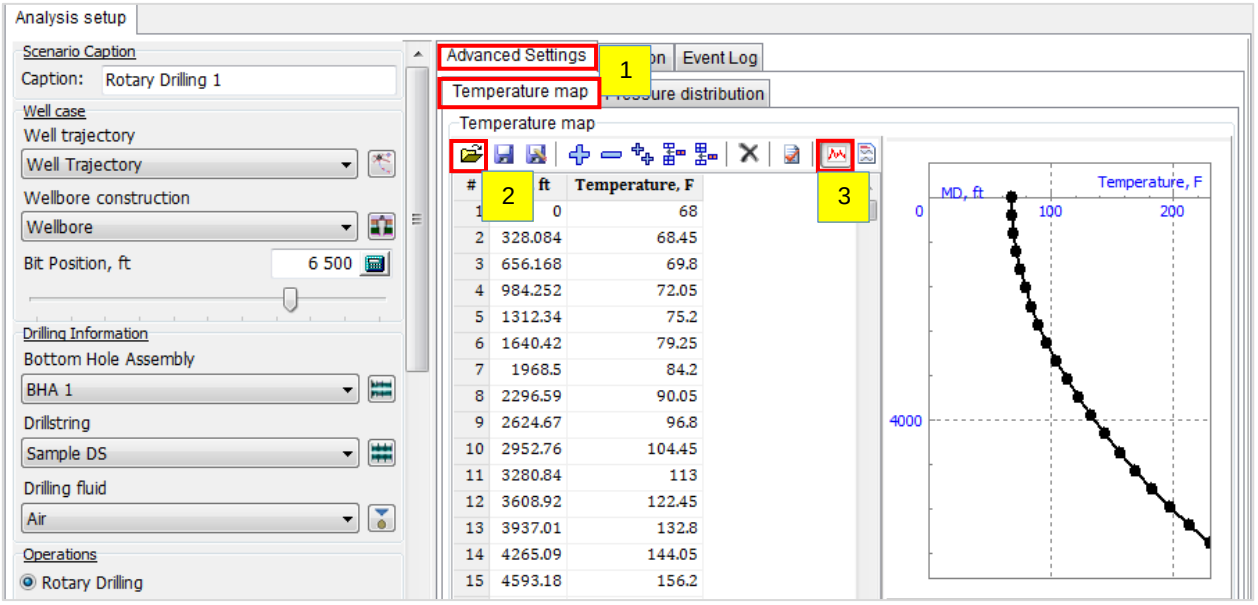
Step 5.9. Select **Soft-string** solver type.



Step 5.10. Select **Advanced Settings** | **Temperature map** page to describe temperature map distribution for the well on the right panel.

Click the **Open** button on the head menu of the *Temperature map editor* to load the sample temperature distribution from *Source directory\temperatures\ Sample temperature map.tmf* file.

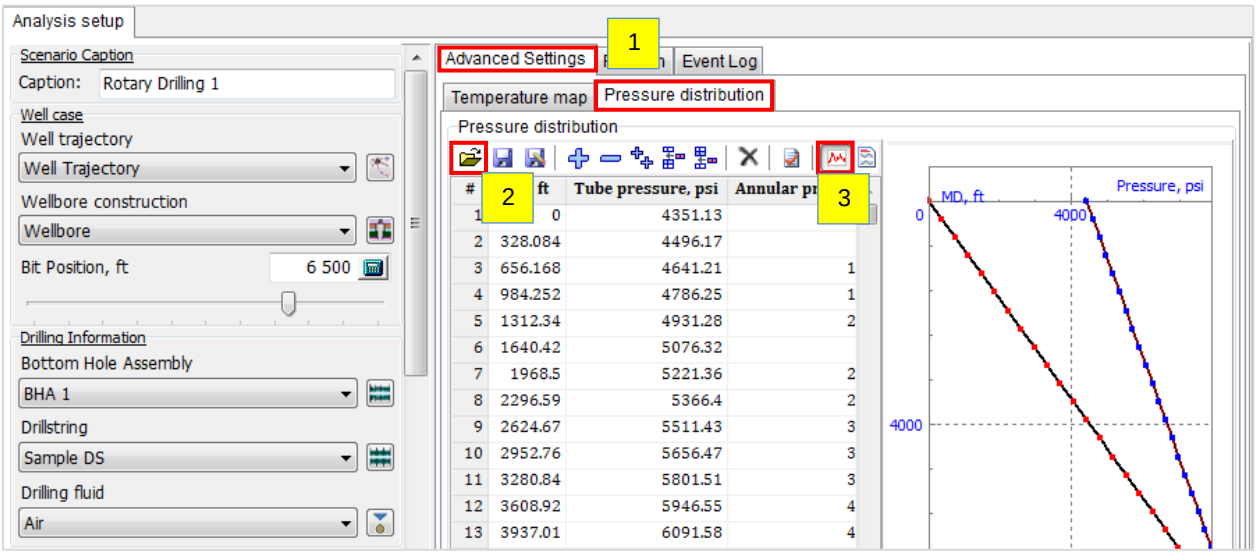
Click the button to visualize the distribution on plot.



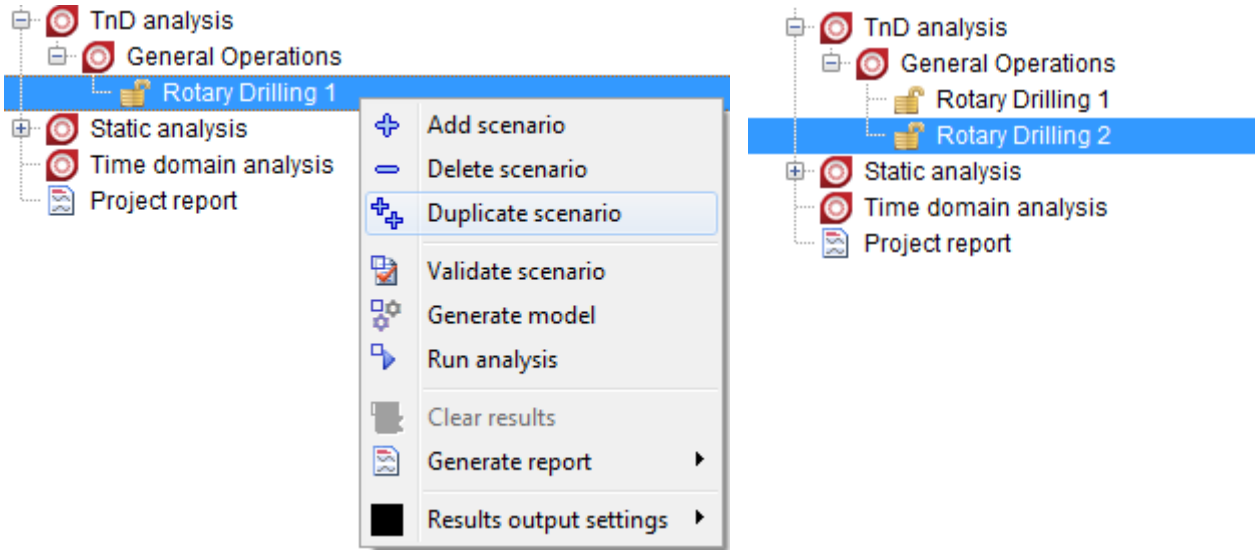
Step 5.11. Select **Advanced Settings** | **Pressure distribution** page to describe internal/external pressure distribution for the drillstring on the right panel.

Click the **Open** button on the head menu of the *TPressure distribution editor* to load the sample from *Source directory\pressures\ Sample pressure map.pmf* file.

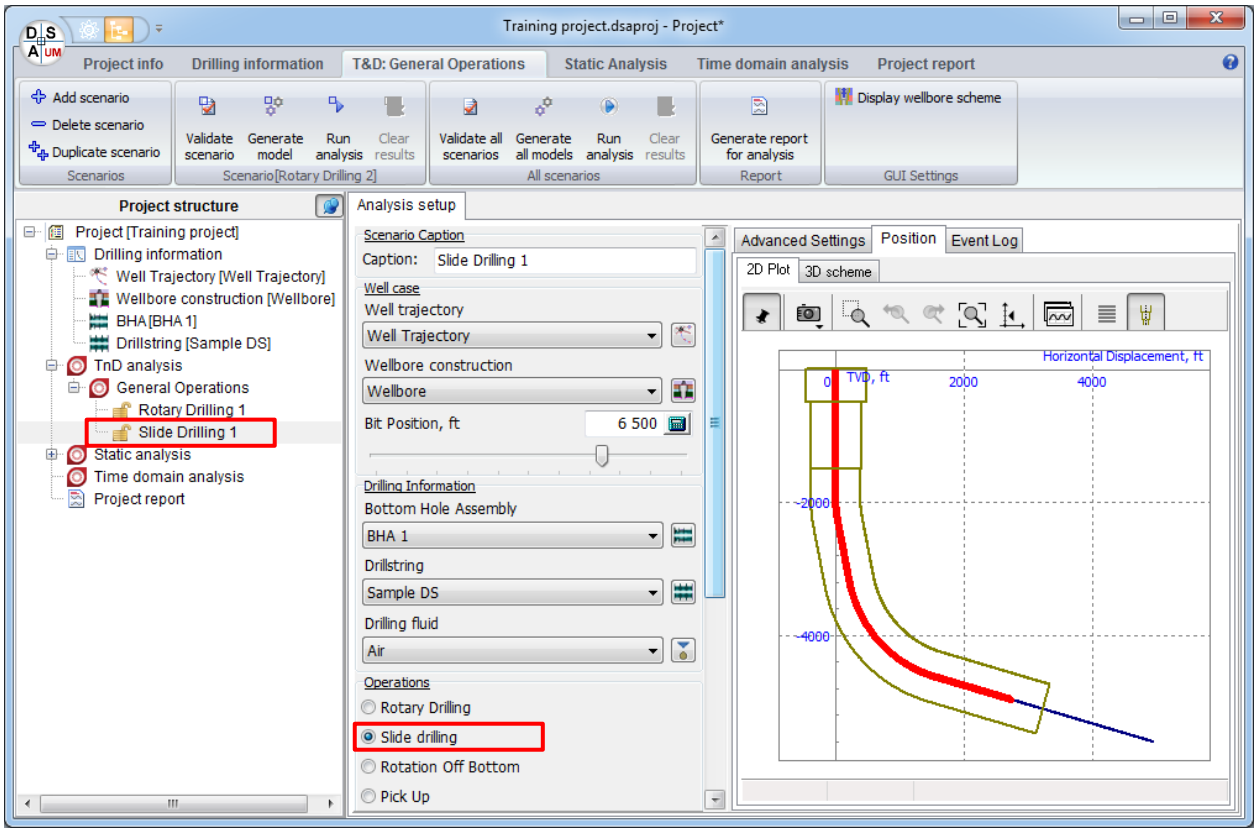
Click the button to visualize the distribution on plot.



Step 5.12. Select the *Rotary Drilling 1* scenario node in the *Project tree* and click *Duplicate scenario*.



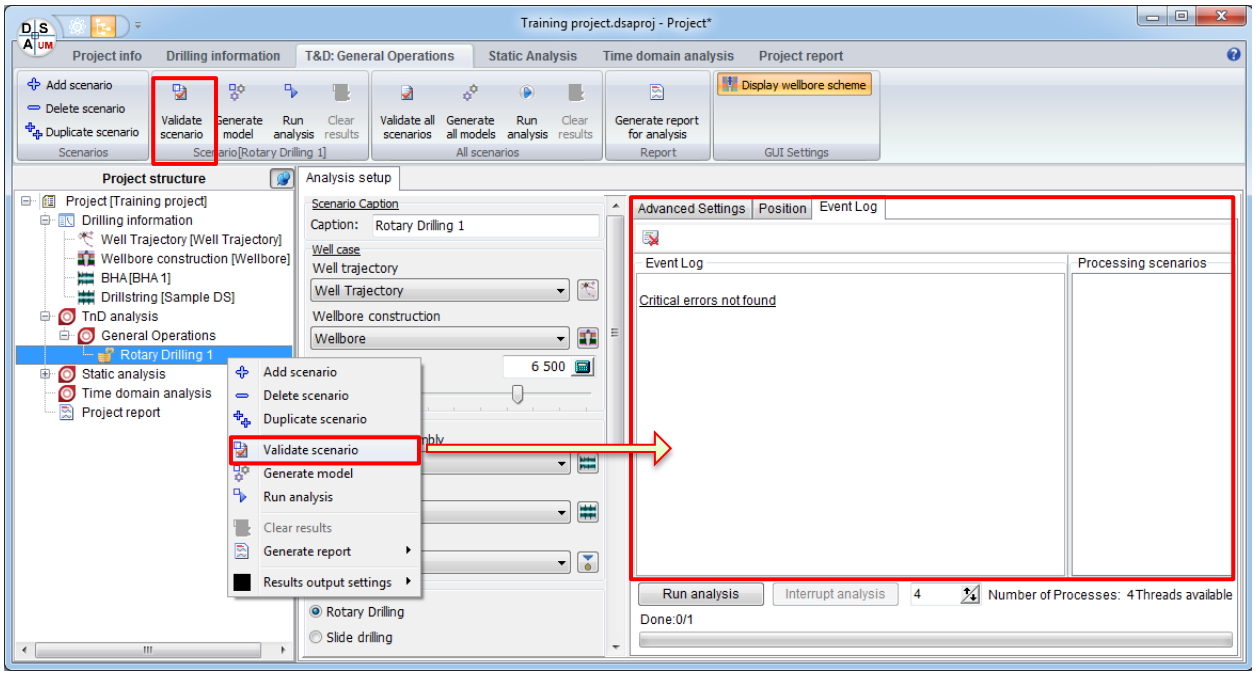
Step 5.13. Change the *Rotary Drilling 2* scenario and change the operation to Slide drilling. The scenario caption will be changed to *Slide drilling* automatically.



Now our project contains 2 T&D scenarios that we can compute and compare the results.

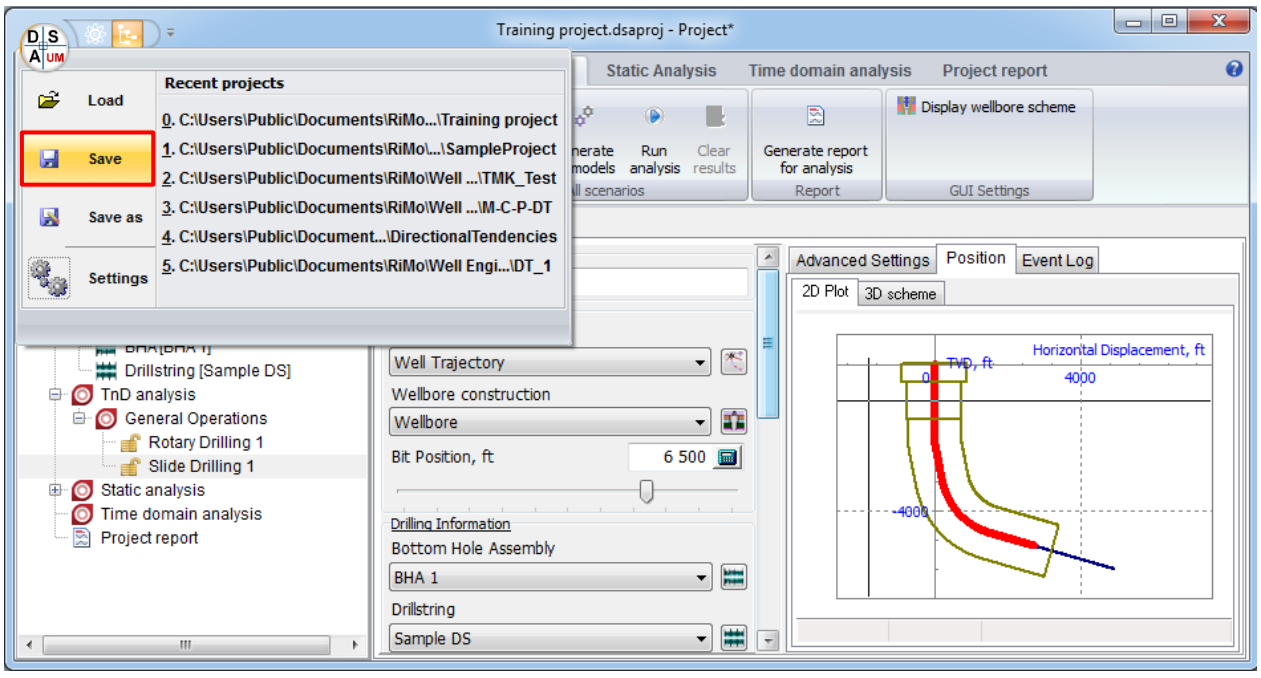
Step 5.14. Click on **Validate Scenario** button in the *Project tree* popup menu, or the head menu to validate the description of the scenario.

Event Log page will be displayed automatically on the right panel; the 'No errors or warnings found' message will appear if the input data is correct.



Rotary Drilling 1 and *Slide Drilling 1* scenario are described, validated and ready for running now.

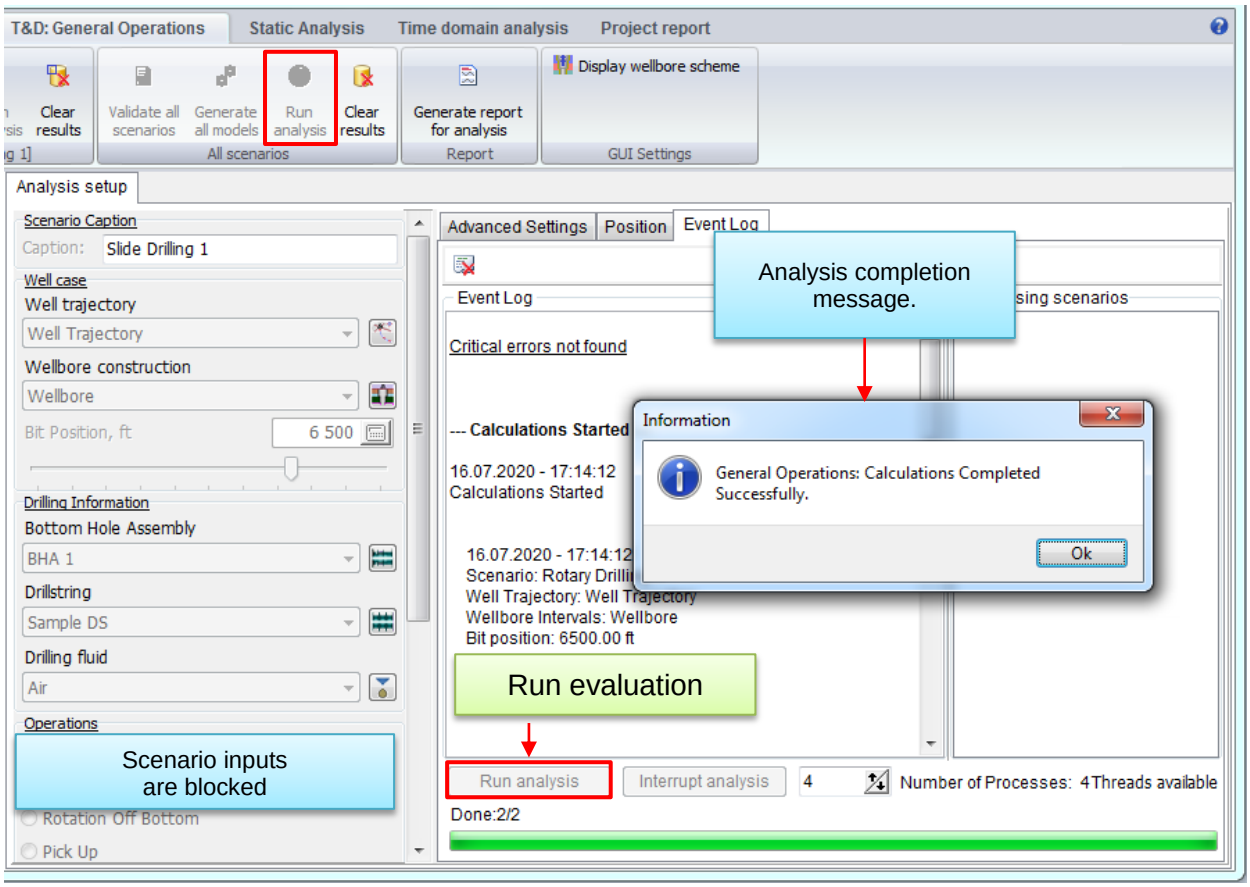
Step 5.15. Save the **project**.



2.4.2. Analysis run

Step 5.16. Select **Event Log** page, and click on **Run Analysis** button to start the evaluation of the described scenario.

Individual progress bar will be added to the right **Running Scenarios** panel; **"--- Calculations Started ---"** record will be added to the **Event Log**; scenario inputs will be blocked automatically.



Note: Input data validation, and project saving is carried out automatically before process is running.

Note: One can use **Run all scenarios** button from the *Project tree* popup menu or head menu to start the evaluation; the **Run scenario** button enables run of the selected scenario only.

'General Operations: Calculations Completed Successfully' message will be generated after the completion of all running scenarios (single scenario in our case). Normally, calculation of the Soft-string scenario should take less than 1 second.

Step 5.17. Click **Ok** to close the message.

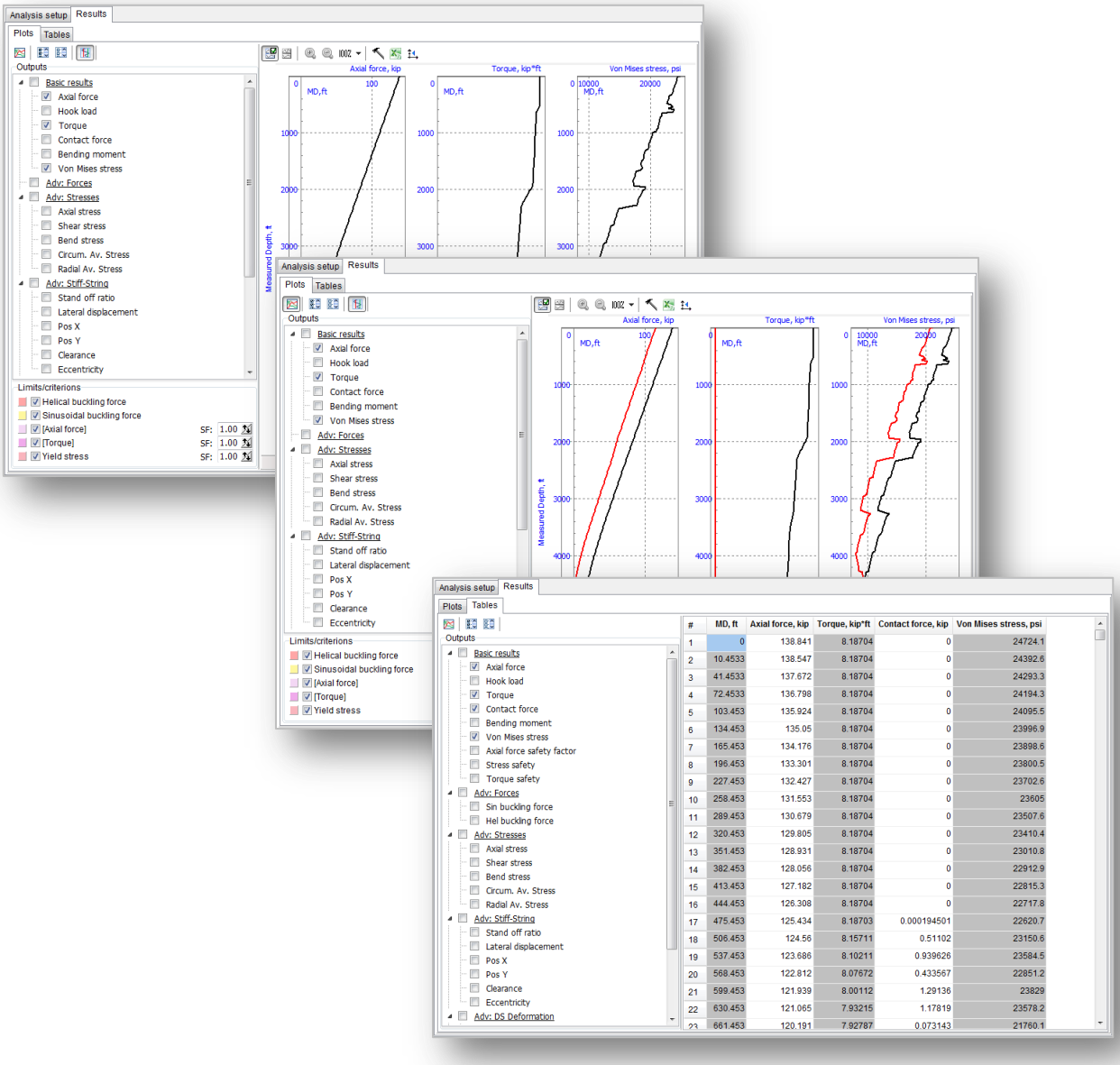
Results for the scenario has been calculated and stored in the scenario folder on hard drive; calculated scenarios are marked by icon in the *Project tree*.

2.4.3. Results output

DSA GUI provides the output of the calculated T&D results in 2D plot and Table formats:

- 2D plots: set of 2D plots of the various result items on the single page.
- Table output: table view of the calculated result items.

GUI enables output of the results of single (selected) scenario, as well as the output of the multiple scenarios results for comparison.



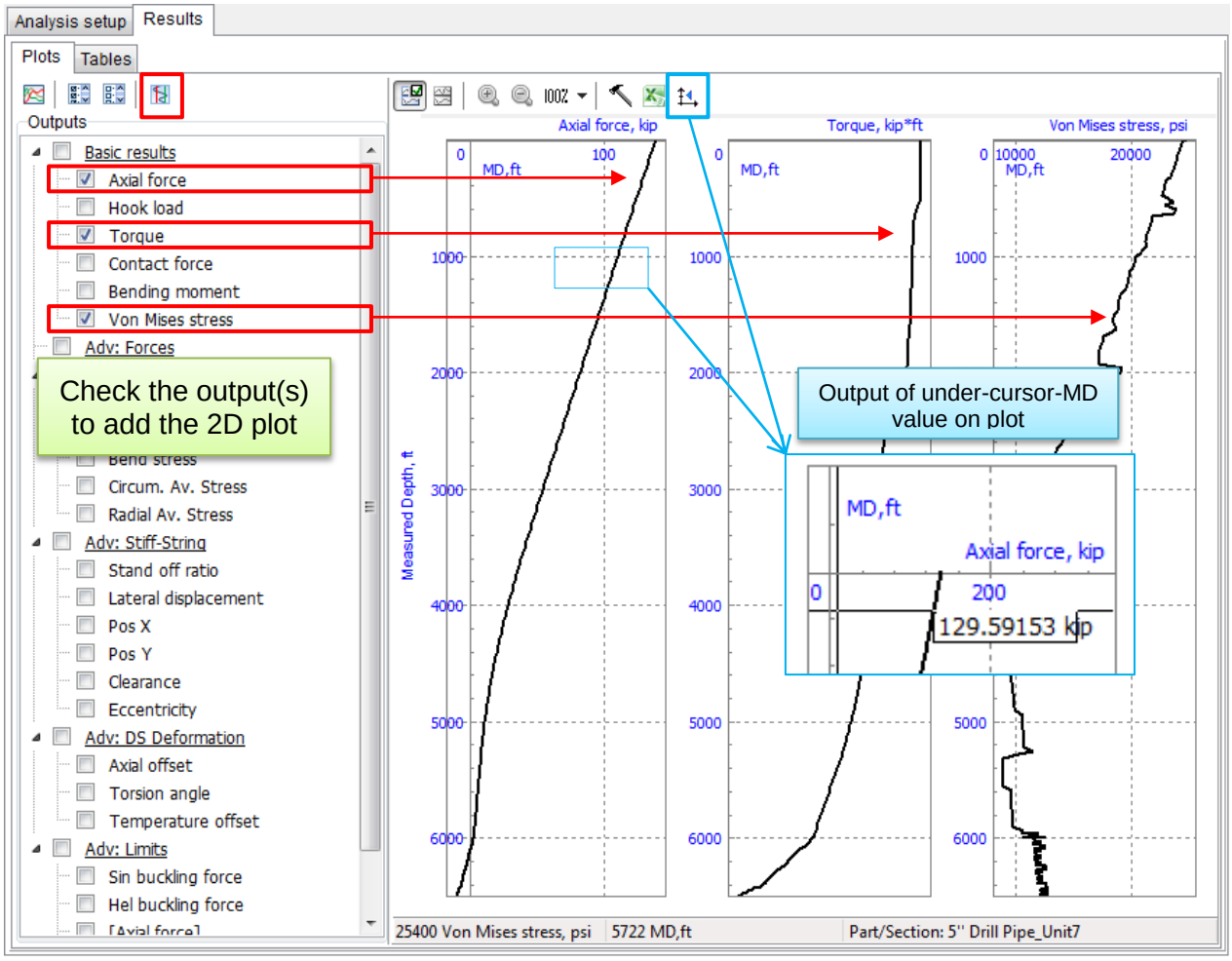
Step 5.18. Select the calculated scenario node (**Training scenario**) in the *Project tree* to make the **Results** page on the right page control visible, and go to the page.

2.4.3.1. 2D plots

Single scenario results output

Step 5.19. Select **Results** | **Plots** page to display the set of 2D plots of the various result items.

One can select an output from the check box list – the corresponded dependency of the selected output vs. measured depth will be displayed on the 2D plot on the right panel.



The following 2D Plot display options are available:

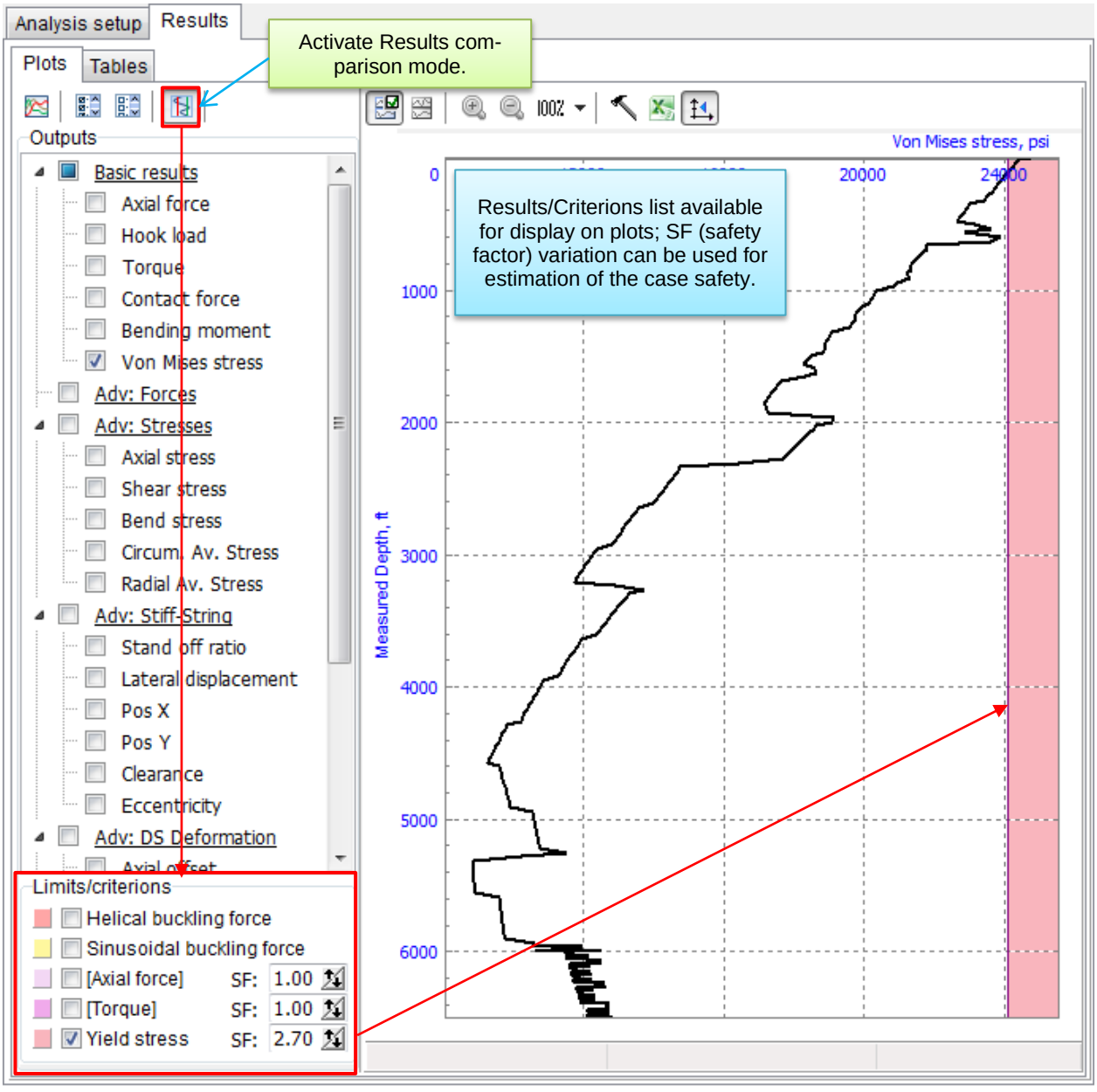
- 100% - Fit all plots;
- - Turn on advanced 2D plot settings;
- - Export plot data to MS Excel;
- - Display plot value corresponded to under-cursor-MD in hint window.

Results safety estimation

Safety estimation of the calculated results is available by visualization of **Limits/Criterions** areas on plots.

Limit/Criterion	Description
Helical/Sinusoidal buckling force limit	Estimation of critical helical/sinusoidal buckling force for drillstring section by analytical solution
[Axial force]	Max allowable force defined for the drillstring parts
[Torque]	Max allowable torque defined for the drillstring parts
Yield strength	Yield strength of the drillstring parts (from the part material properties)

Step 5.20. Check on the [Yield stress] criterion, and vary the SF parameter value to estimate the safety factor for the considering case.

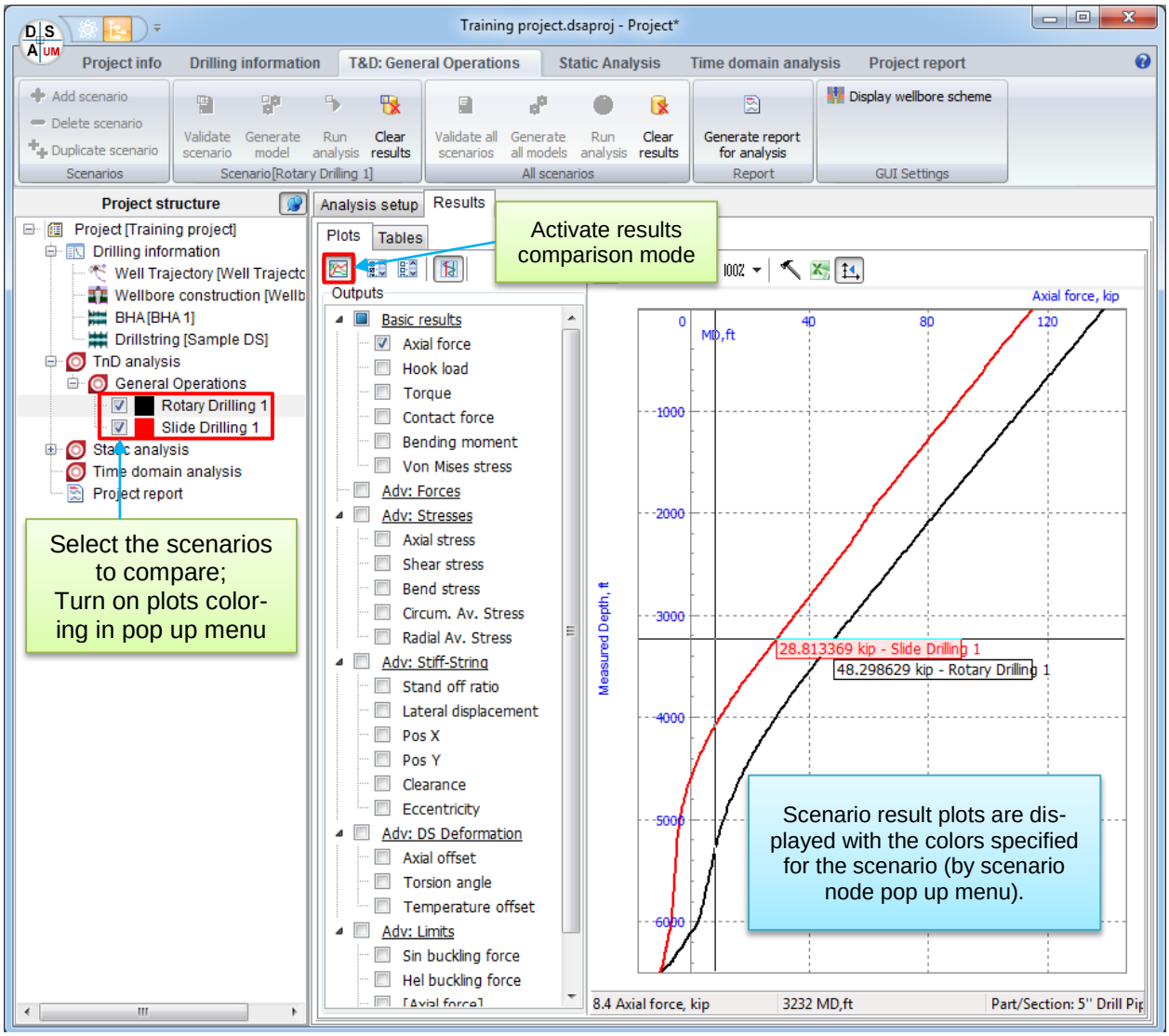


Results comparison mode

Step 5.21. Click on the button above the output list top activate the results comparison mode.

Checkboxes will appear left to the calculated scenarios nodes in *Project Tree*; the user can select the scenarios, which results should be displayed on one plot.

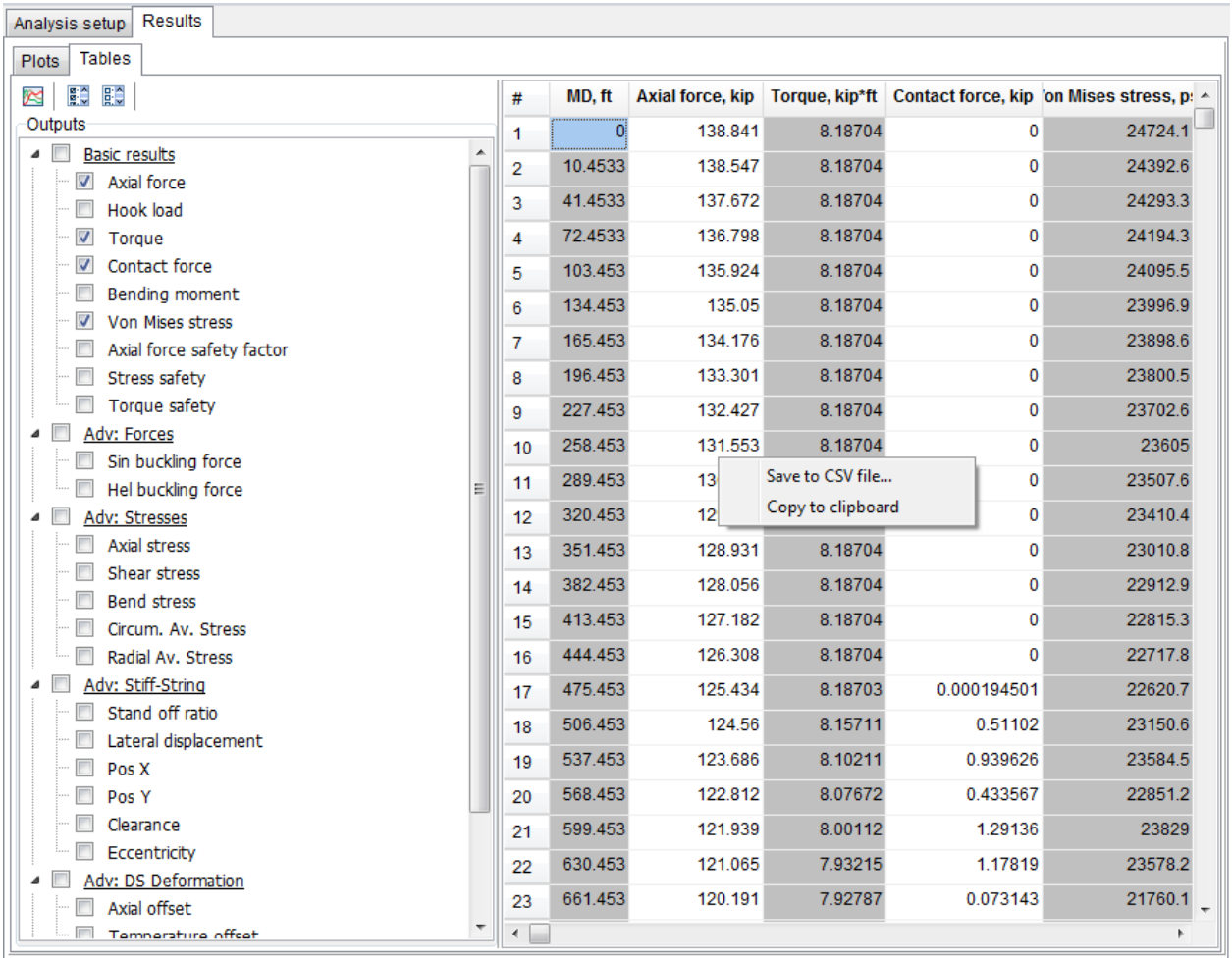
Step 5.22. Select the *Rotary Drilling 1* and *Slide Drilling 1* scenarios in the Project tree to display the comparison plots.



2.4.3.2. Table output

Step 5.23. Select **Results** | **Tables** page to access to numerical results data, export it to CSV file, or copy to clipboard.

One can select an output from the check box list – the corresponded dependency of the selected output vs. measured depth will be displayed as a column of the table.



#	MD, ft	Axial force, kip	Torque, kip*ft	Contact force, kip	on Mises stress, p
1	0	138.841	8.18704	0	24724.1
2	10.4533	138.547	8.18704	0	24392.6
3	41.4533	137.672	8.18704	0	24293.3
4	72.4533	136.798	8.18704	0	24194.3
5	103.453	135.924	8.18704	0	24095.5
6	134.453	135.05	8.18704	0	23996.9
7	165.453	134.176	8.18704	0	23898.6
8	196.453	133.301	8.18704	0	23800.5
9	227.453	132.427	8.18704	0	23702.6
10	258.453	131.553	8.18704	0	23605
11	289.453	13		0	23507.6
12	320.453	12		0	23410.4
13	351.453	128.931	8.18704	0	23010.8
14	382.453	128.056	8.18704	0	22912.9
15	413.453	127.182	8.18704	0	22815.3
16	444.453	126.308	8.18704	0	22717.8
17	475.453	125.434	8.18703	0.000194501	22620.7
18	506.453	124.56	8.15711	0.51102	23150.6
19	537.453	123.686	8.10211	0.939626	23584.5
20	568.453	122.812	8.07672	0.433567	22851.2
21	599.453	121.939	8.00112	1.29136	23829
22	630.453	121.065	7.93215	1.17819	23578.2
23	661.453	120.191	7.92787	0.073143	21760.1

Note: MD column is common for all the outputs.

2.5. Time Domain Analysis

The section contains brief overview of the Time Domain Analysis concept, and step-by-step description of Time Domain analysis of the Rotary BHA assembly in the sample well trajectory.

Time Domain analysis overview

DSA application provides advanced tools for close-to-real time simulation of an assembly motion in the well with optional account of operational loads, assembly-wellbore side contact, and bit-rock interaction.

Time Domain analysis can be run in multi-scenario mode (like Static Analysis) with result post-processing, or in *Simulation Desk* mode – animation of model motion and results output during simulation.

Simulation includes the following stages:

- Initial state evaluation:

  Time domain simulation is started from the equilibrium state of the assembly to minimize unrealistic transient effects.

  Equilibrium state is evaluated in accordance to the procedure described in Sect.2.3.

  Axial force defined for the time domain scenario is applied to the top of the assembly; bit point axial motion is constrained.

  Note: Constraints on lateral motion of top/bit point of the assembly can be defined in **Advanced User** and **Developer** mode.

- Time domain simulation

  Assembly model boundaries are redefined – constraint on axial motion of the top point is added; axial force – calculated at the initial state evaluation stage is applied to the bit point (used for initialization of bit-rock interaction force model, if active).

  Rotary speed is provided by control torque applied to the top point of the assembly; rotary speed is set equal to zero, model is speeding up under action of the control torque.

  Assembly kinematics and set of force characteristics are stored during simulation automatically, and can be replayed after completion of the analysis; the user can visually control the process by animation window and various 2D plots during simulation in *Simulation Desk* mode.

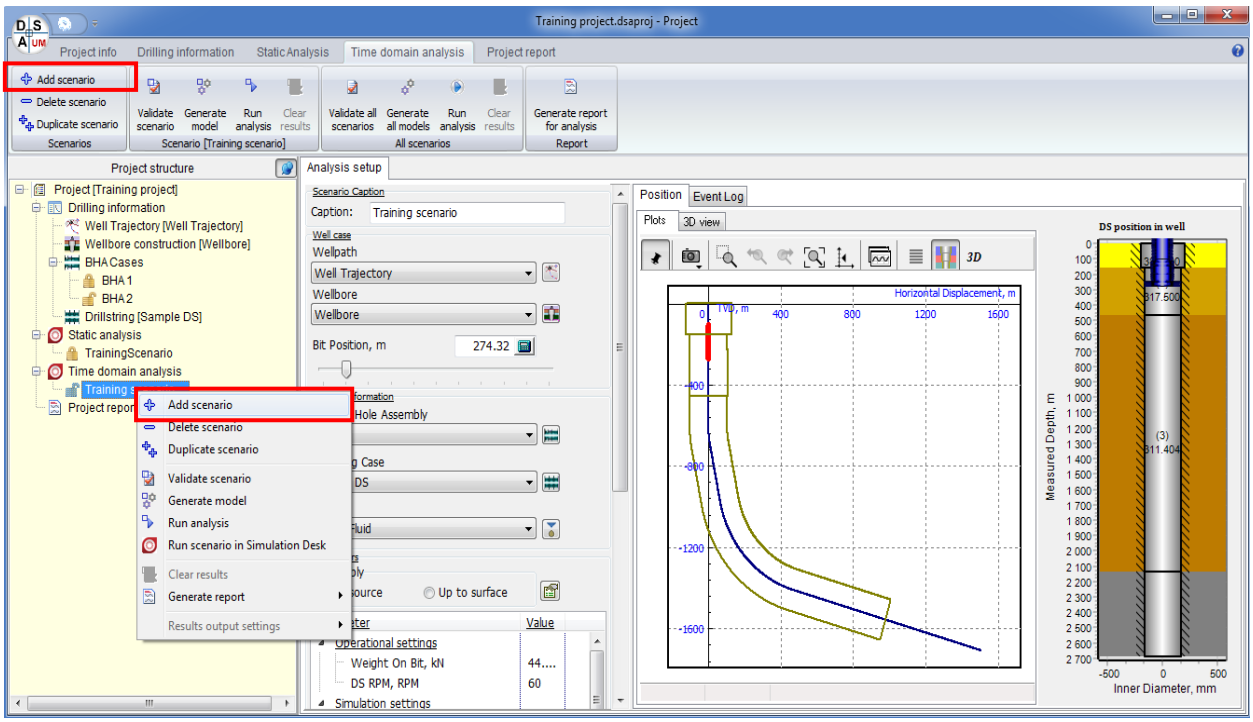
Time Domain Analysis GUI enables study of any number of test cases – *scenarios* – within single project. Time Domain Analysis scenario (or *TDA scenario*) is defined by the following basic inputs:

- Well case info: well trajectory case, wellbore case and bit position.
- Drilling Info: BHA case, drillstring case and drilling fluid.
- Operational settings: Drillstring weight, block weight, drillstring RPM.
- Simulation settings: simulation time, cut-off-time.
- Bit-rock interaction options.
- Harmonic excitations options.
- Set of advanced settings available for the **Advanced User** or **Developer**.

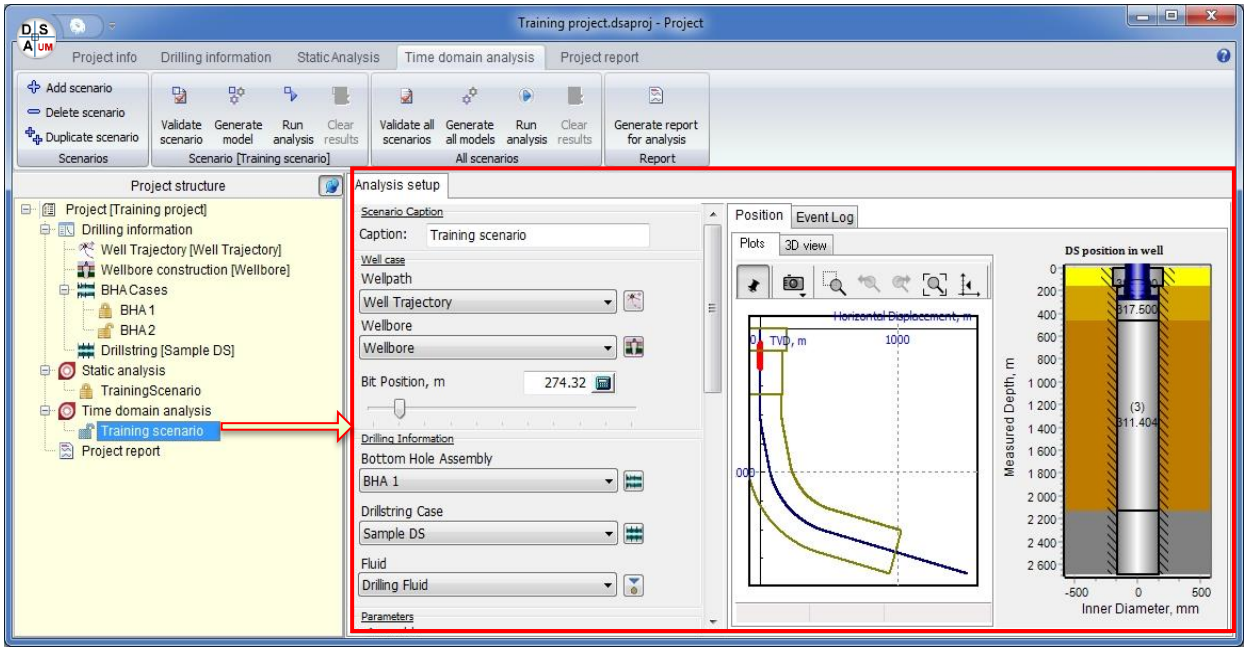
Single scenario corresponds to simulation of the motion of the whole drillstring, or its bottom part, described with BHA case and drillstring case; axial force applied to the top point of the assembly is evaluated as **Drillstring weight** minus **Travelling Assembly Weight** value.

2.5.1. Scenario description

Step 6.1. Select the **Time Domain Analysis** *Project tree* node to access the Static Analysis GUI, and **Add scenario** by popup menu or the head menu.



Scenario 1 test case will be added to the TDA scenario list; the corresponded node will be added to the *Project structure* tree and focused by default; scenario data will be displayed on the right panel, **Analysis Setup** page.



Set the basic parameters of the new scenario by the middle panel controls.

Step 6.2. Set **Training scenario** caption for the scenario. The scenario node name will be update in the *Project tree* automatically.

Step 6.3. Select the **Well trajectory** item from the **Wellpath** list; the list contains all the well trajectories defined in the project.

Note: One can click on button to come to the **Well trajectory** item description page.

Step 6.4. Select the **Wellbore** item from the **Wellbore** list; the list contains all the wellbore designs defined in the project.

Note: One can click on button to come to the **Wellbore** item description page.

Step 6.5. Set **Bit Position** value equal to **900 ft**.

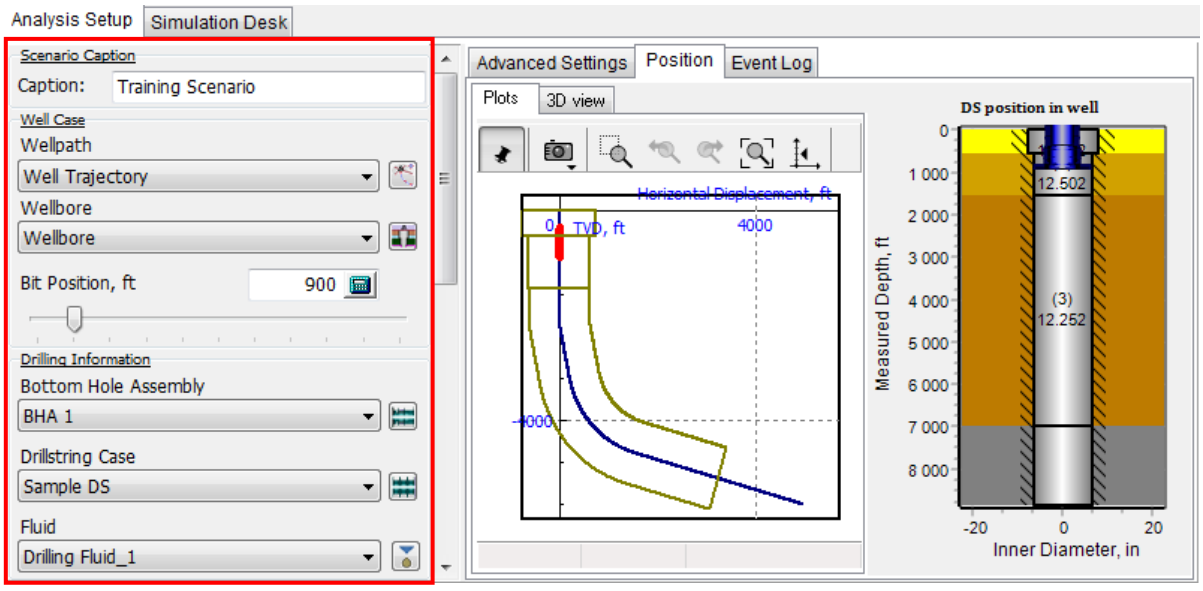
Step 6.6. Select the **BHA 1** item from the **Bottom Hole Assembly** list; the list contains all the bottom hole assembly designs defined in the project.

Note: One can click on button to come to the **BHA 1** item description page.

Step 6.7. Select the **Sample DS** item from the **Drillstring** list; the list contains all the drillstring designs defined in the project.

Note: One can click on button to come to the **Sample DS** item description page.

Step 6.8. Select the **Drilling fluid** item from the **Fluid** list; the list contains all the fluids from the Fluids Database (see Sect.1.4.1.3).

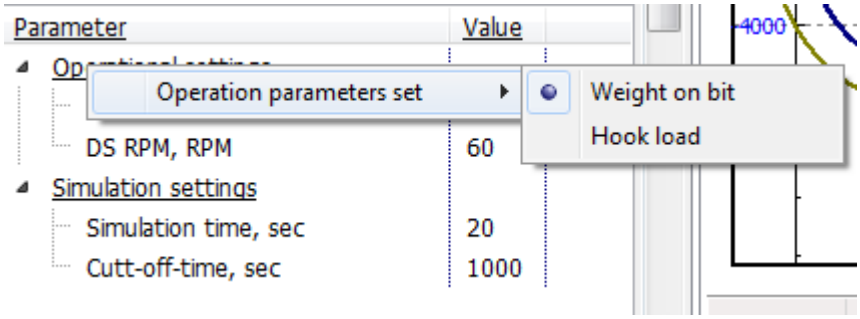


Step 6.9. Set operational and simulation parameters:

➢ *Operational settings*

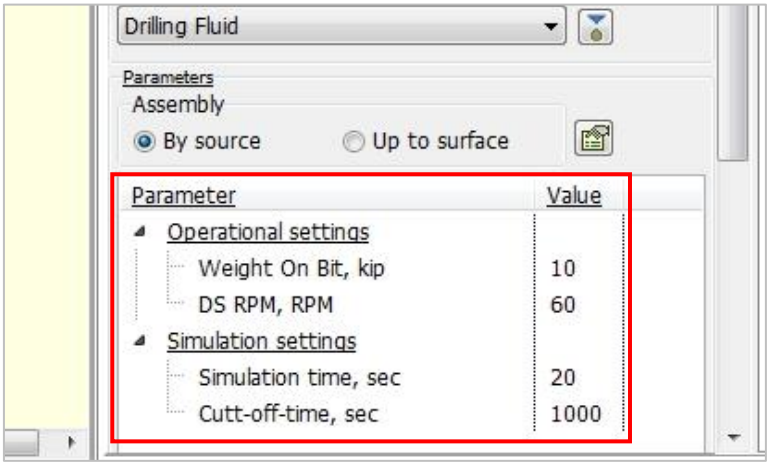
- **Weight on bit = 10 kip**;
- **DS RPM = 60rpm**

  Right click on the parameter value cell and select the option from the popup menu to choose operation parameters set (weight on bit).

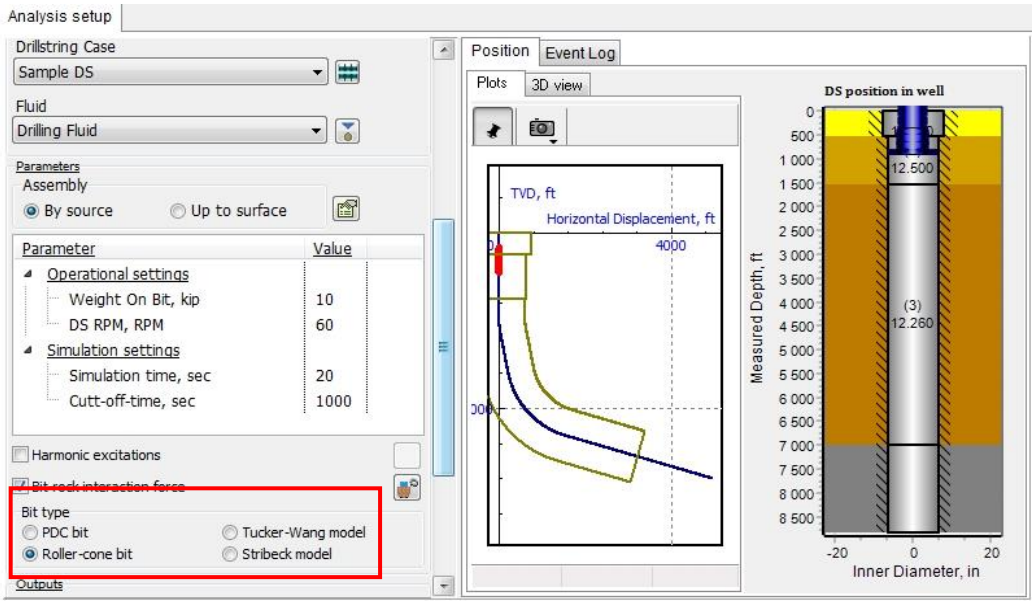


➢ *Simulation settings*:

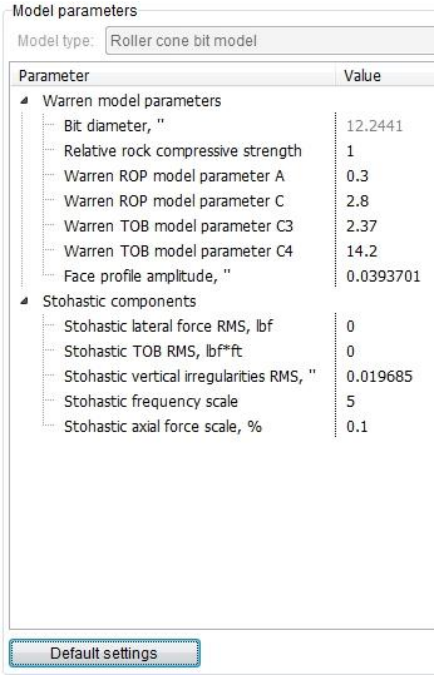
- **Simulation time = 20 sec**
- **Cut-off-time = 1000 sec.**



Step 6.10. Activate **Bit-rock interaction force** control and select **Bit Type**: **Roller cone bit**



Step 6.11. Press on button  and look on the BRI parameters. By default were accepted such settings:



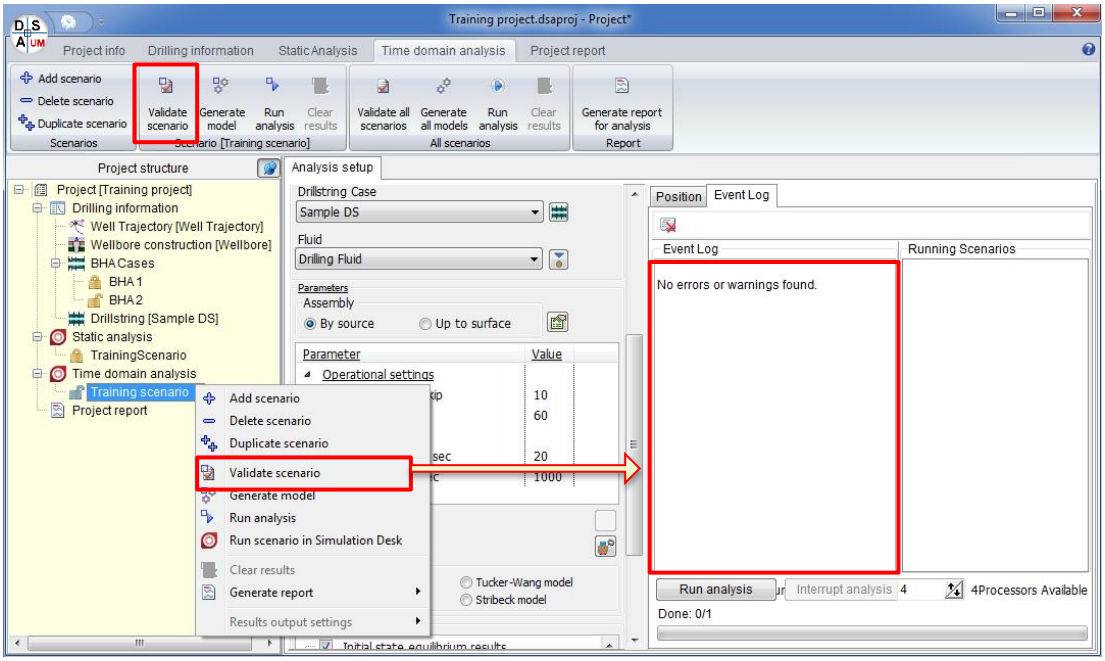
Bit diameter is equal of the bit diameter, stochastic components are not equal zero by default.

Step 6.12. Look on Top drive tab. By default were accepted such settings: Gain = 1.5e5, iGain = 2e7.

Step 6.13. Click on **Validate Scenario** button in the *Project tree* popup menu, or the head menu to validate the description of the scenario.

The information message on mismatch of the scenario bit position with the assembly length will be shown again (see above).

'No errors or warnings found' message will added to the **Event Log**, if the input data is correct.



Training Scenario is described, validate and ready for running now.

Step 6.14. Save the **project**.

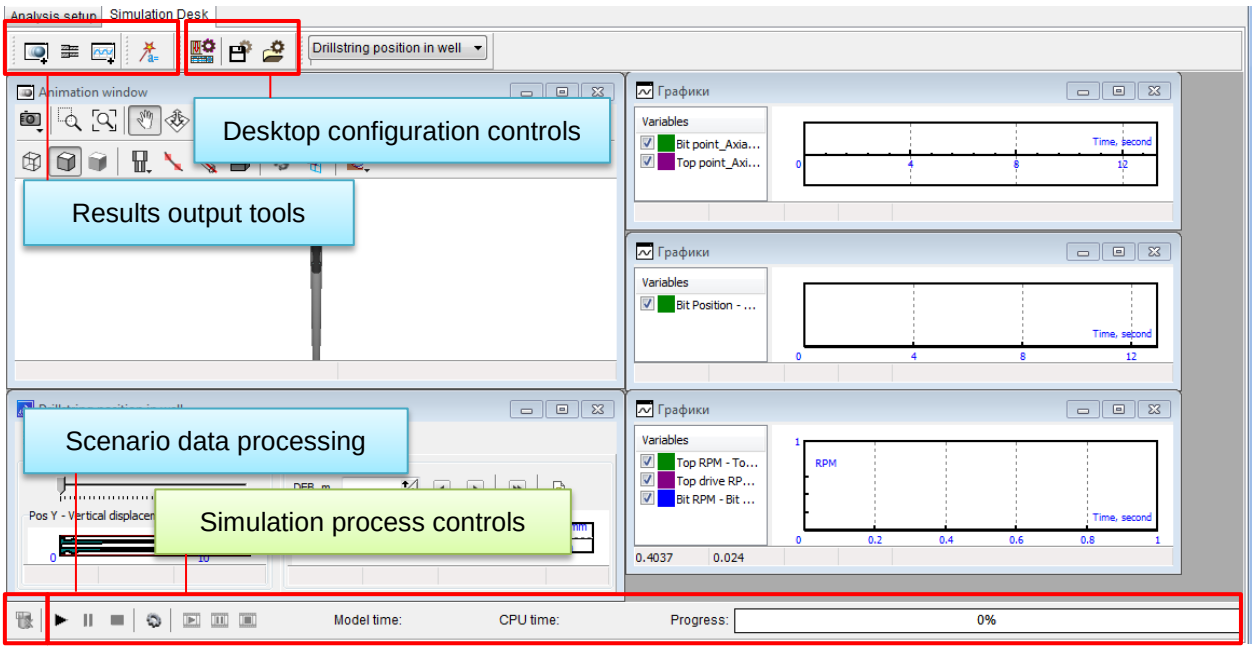
2.5.2. Run scenario by Simulation Desk

DSA GUI enables two variants of Time Domain Analysis processing: *Simulation Desk* and multi-scenario solver. In this section the step-by-step description of the scenario processing in *Simulation Desk* mode is considered.

Important! *Simulation Desk* is initialized by the scenario model files; one needs to generate the model (by special command or during the multi-scenarios run) to make the **Simulation Desk** page visible.

Simulation Desk GUI

Simulation Desk interface includes controls for the scenario data processing, simulation process controls, and set of tools for 3D/2D results output.



Scenario data processing:

- clear scenario results.

Simulation process controls:

- start or continue simulation;
- pause simulation;
- stop simulation and clear results.
- start replaying;
- pause replaying;
- stop replaying.

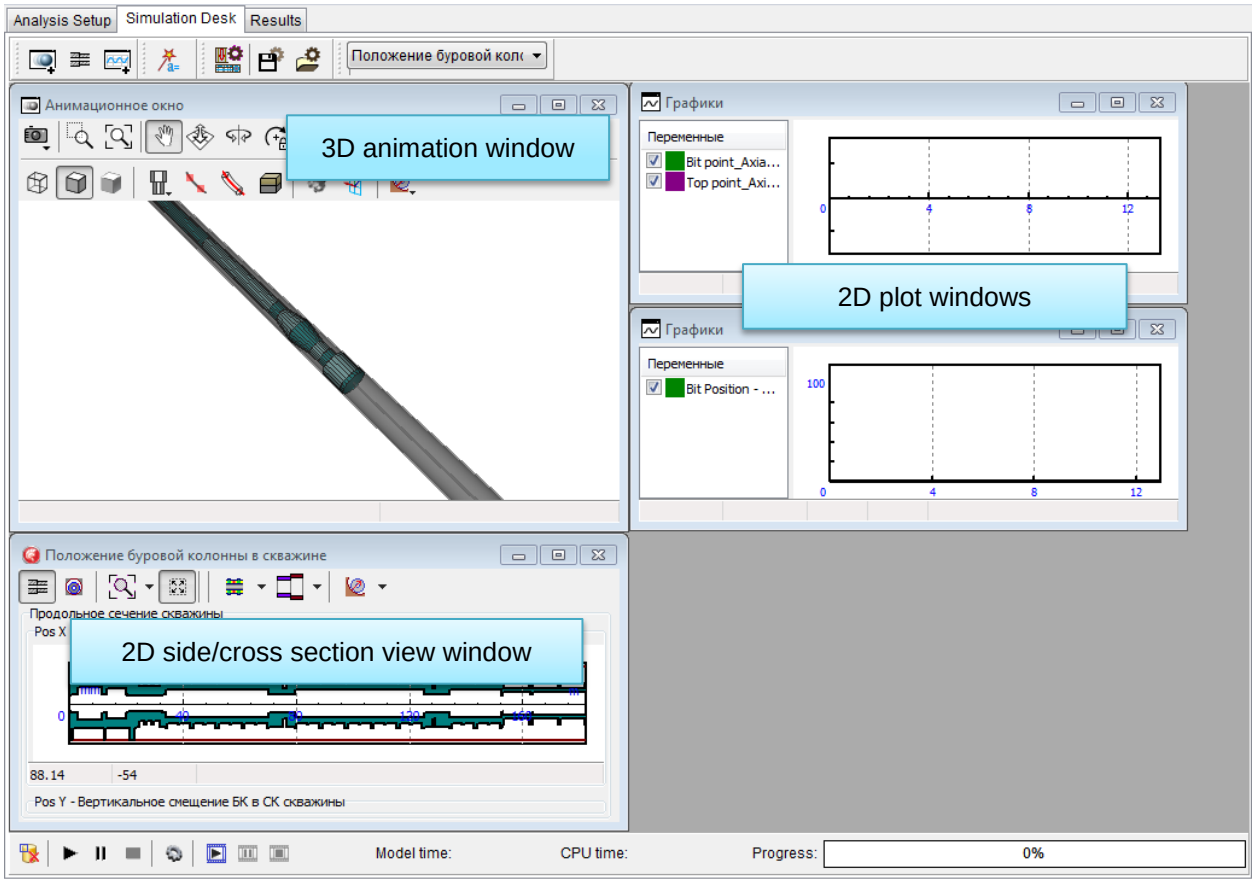
Results output tools:

- add 3D animation window;
- add 2D straightened side/cross section view;
- add 2D plot;
- show/hide *Wizard of variables*.

The default desktop configuration for the scenario is loaded automatically during the first loading of the scenario model, or after regeneration of the model files.

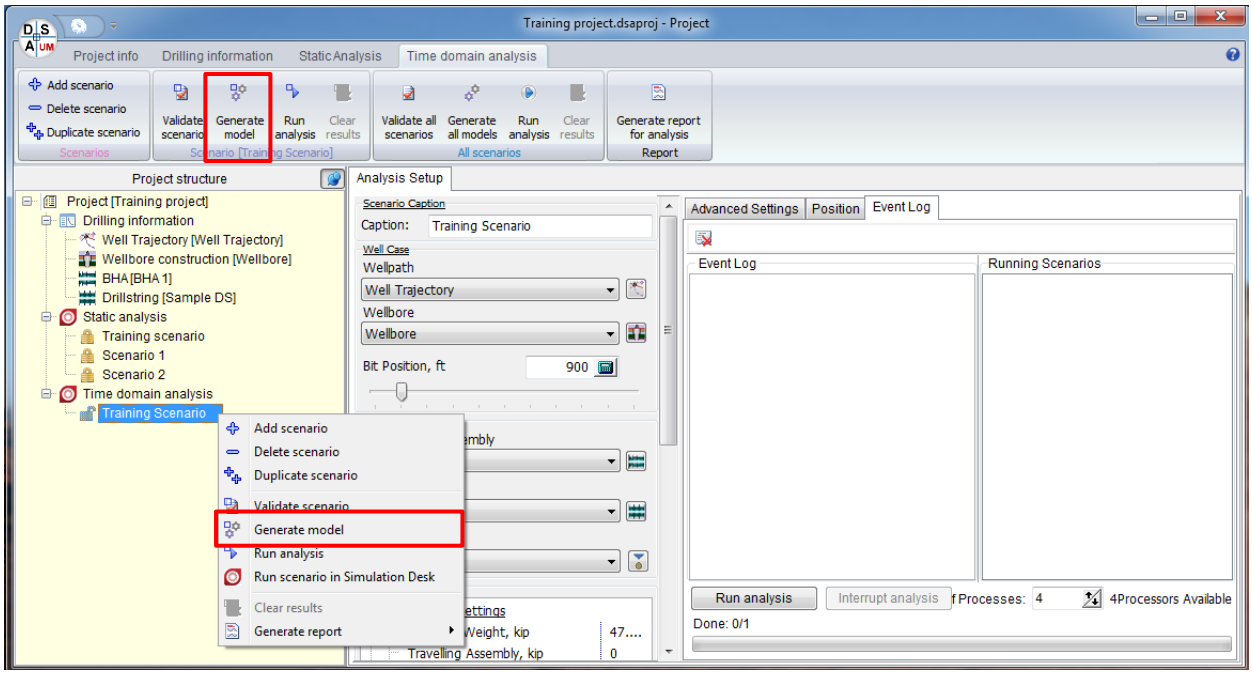
Default desktop includes:

- 3D animation window;
- 2D side/cross section view window;
- two 2D plot windows: Bit/Top point axial G-forces, and Bit Position vs. Time dependencies.



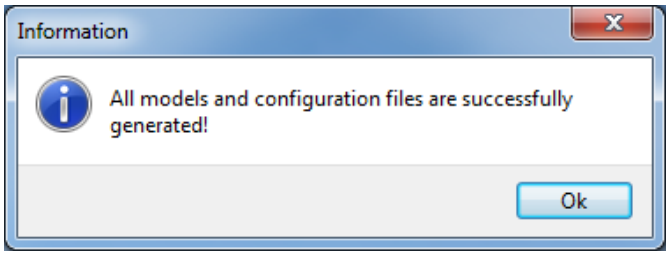
One can click button to regenerate and upload the default desktop.

Step 6.15. Select the **Training scenario** node in the *Project tree*, and click on **Generate Model** option of the pop up menu of the head menu.



Scenario inputs validation will be started automatically before model file creation.

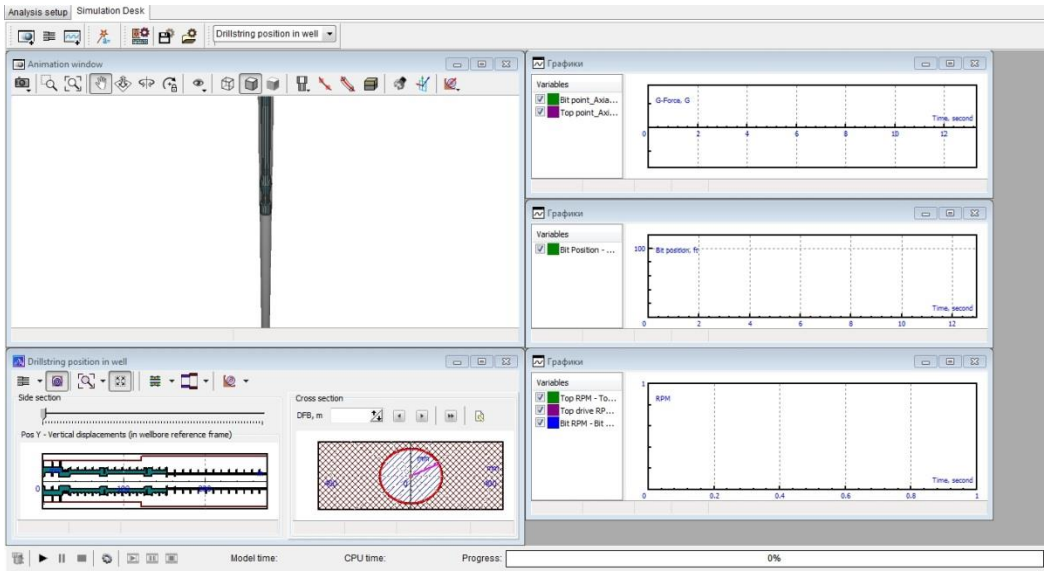
The success model generation message will be shown finally.



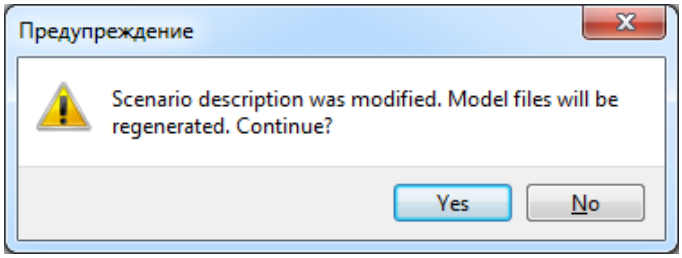
The model files have been created successfully, the model is ready for loading/running by *Simulation Desk GUI*.

Step 6.16. Select **Simulation Desk** page on the right panel.

Default desktop contains many windows with plots.



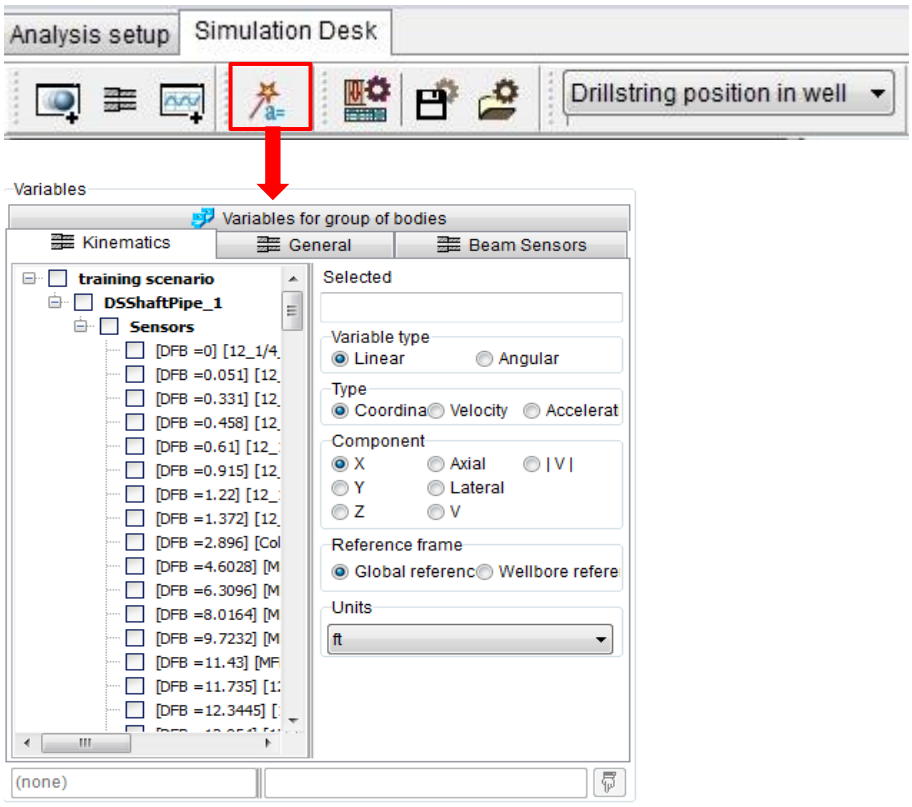
Note: The request on the model files generation will be output automatically, if the model files are not actual to the current settings of the scenario.



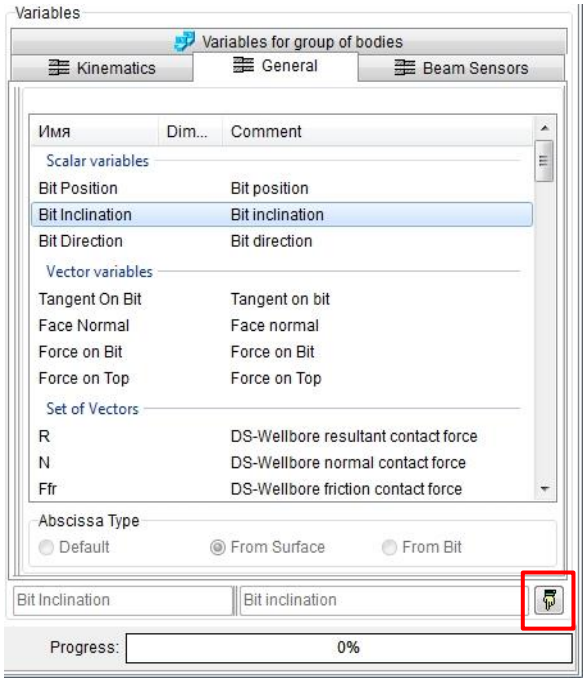
If one will click **No**, the empty **Simulation Desk** page will be opened; the request will be repeated when one will try to start the simulation process.

2.5.3. Wizard of variables

Step 6.17. Click on Wizard of Variables.



To add a variable to the graphics window, you need to click on the button “hand”;



The following types of variables are available used in DSA GUI:

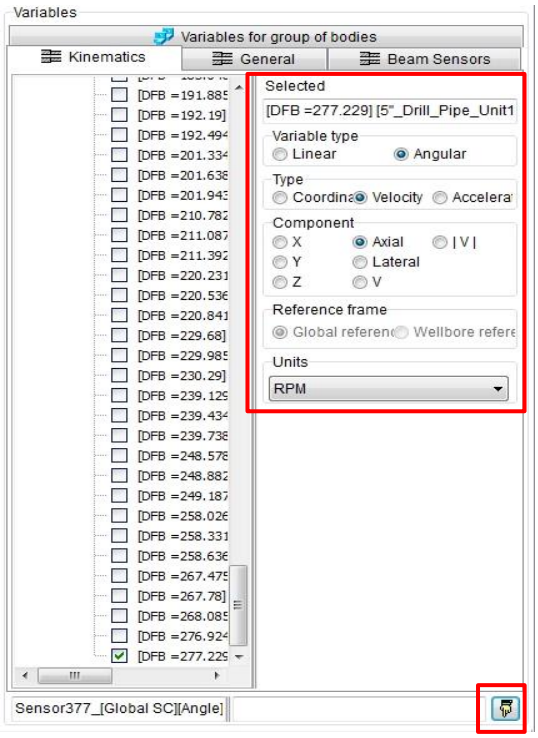
- Scalar variables (Bit position, Inc, Direction, etc.) are displayed in graphical window as a vs. time plot;
- Vector variables, Set of Vectors (Tangent on bit, Ffr, Axial, etc.) are displayed in animation window.
- Plot variables (Well Inc, MBend, etc.) are used for output of parameters distribution along the assembly.

Wizard of variable consist several group.

Step 6.18. Choose tab ‘General’, click ‘Force on bit’ at the ‘Vector variables’ group and click button ‘Hand’. Variable wall be added in 3D window.

Step 6.19. Choose tab ‘General’, choose ‘PosX’ at the ‘Plot variables’ group and click button ‘Hand’.

Step 6.20. Choose tab ‘Kinematic’, choose the last sensor left, click checkboxes like on picture and click button ‘Hand’ to output Bit angular velocity on separate graphical window.

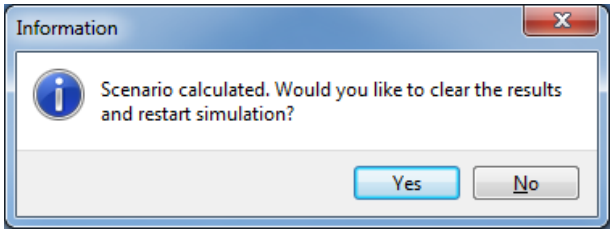


Step 6.21. Choose tab ‘Beam Sensors’, choose the first sensor left, choose ‘Qz’ and click button ‘Hand’.

Step 6.22. If necessary, **arrange the windows as you like**. Those windows that you do not need to calculate, you can collapse or close.

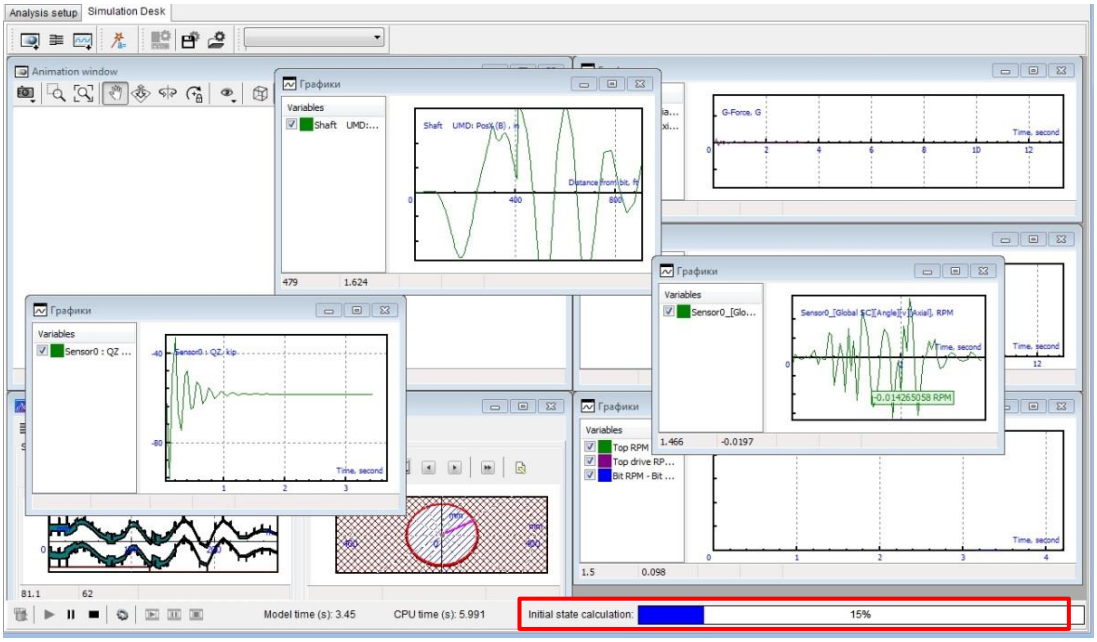
Step 6.23. Click on **Start Simulation** button 

Note: If the scenario already has calculated results (the scenario was done), the additional request will be generated to continue the procedure or cancel it.



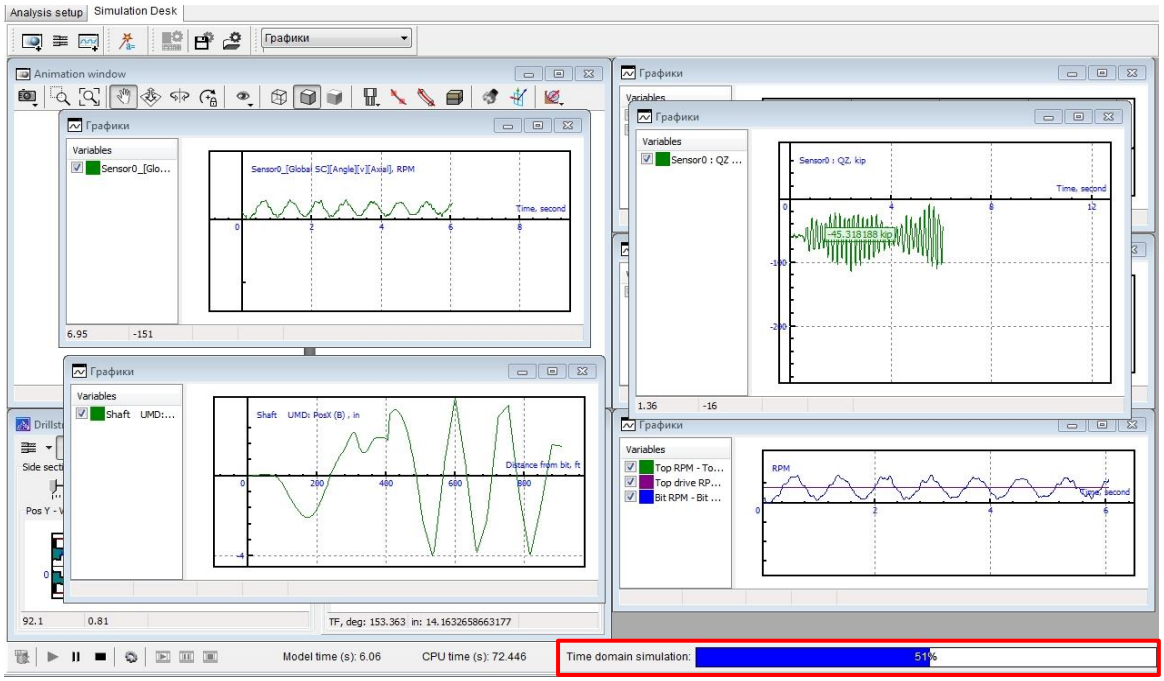
Click **Yes** to continue.

Simulation is run. The initial state calculation – evaluation of the equilibrium state for the user-defined boundaries – is processed at the first stage of scenario simulation. The progress bar label indicates the current stage.



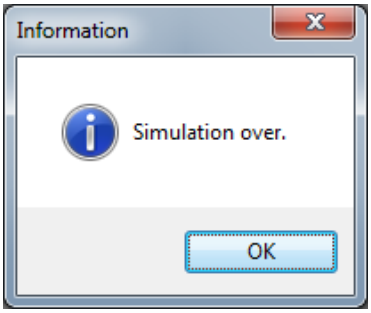
Note: If the scenario has been already started, and the input parameters are still actual to the previous run - the initial state will be taken from the saved *Initial state* results; equilibrium state evaluation stage will be missed.

After the equilibrium state evaluation (30% progress, by default), the time domain analysis of the rotating assembly is started. The progress bar label indicates the stage.



'Simulation over' message will be generated after any of the simulation end conditions will be reached (simulation time, or cut-of-time).

Note: Cut-of-Time is applied to the *Time domain simulation* stage only.



Step 6.24. Click **Ok** to close the message.

Results for the scenario has been calculated and stored in the scenario folder on hard drive; calculated scenario is marked by icon in the *Project tree*.

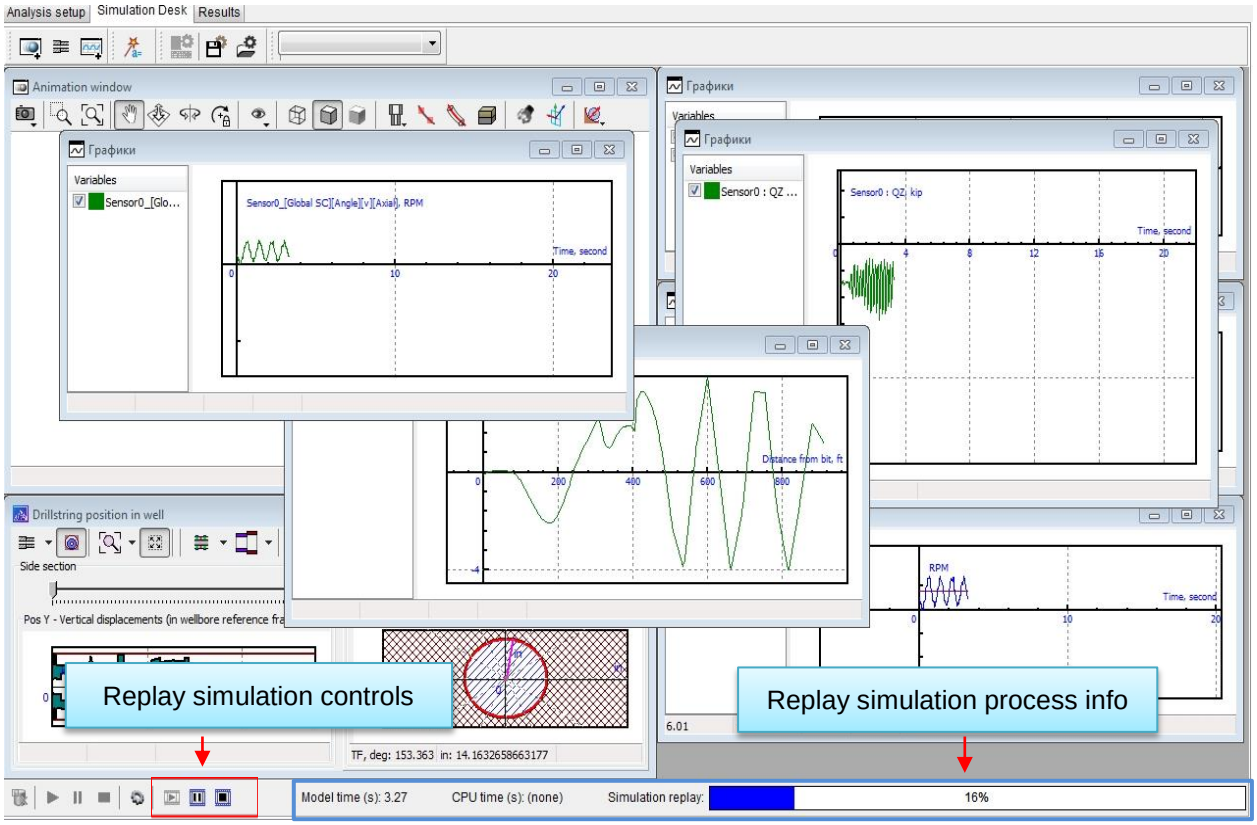
2.5.4. Replay calculated scenario by Simulation Desk

Simulation Desk GUI enables replaying of the assembly motion for the calculated scenarios. One can replay the scenario case without starting time domain simulation – 2D/3D animation and variables output are provided on the base of the records of kinematical characteristics stored during previous analysis.

Note: Some outputs – related to bit-rock interaction force models, etc. – cannot be restored from the instant kinematics of the model; these outputs have zero values in *Replay* mode. One needs to rerun Simulation, or use **Variables** and **Plots** pages to study the direct records of the outputs.

Simulation replay controls are placed on the bottom panel, left to the simulation process controls.

Step 6.25. Click the **Start replay** button left on the bottom panel to start the process.



Current position on the record is output on the progress bar; **'Simulation replay'** label marks the GUI mode.

Note: Scenario replay functionality is available for the calculated scenarios only. Output step cannot be changed in the *replay mode*. The simulation controls are blocked in the *replay mode*; one needs to stop the player to clear the scenario results or restart the scenario.

Assembly motion record will be output on 2D plots, side section view window and animation window. One can add any new variables/plots to control additional parameters by standard Simulation Desk GUI tools.

Step 6.26. Click the **Pause replay**  button left to pause the process.

The outputs are not cleared in the pause mode; one can turn on the outputs arbitrary and continue the process or interrupt it.

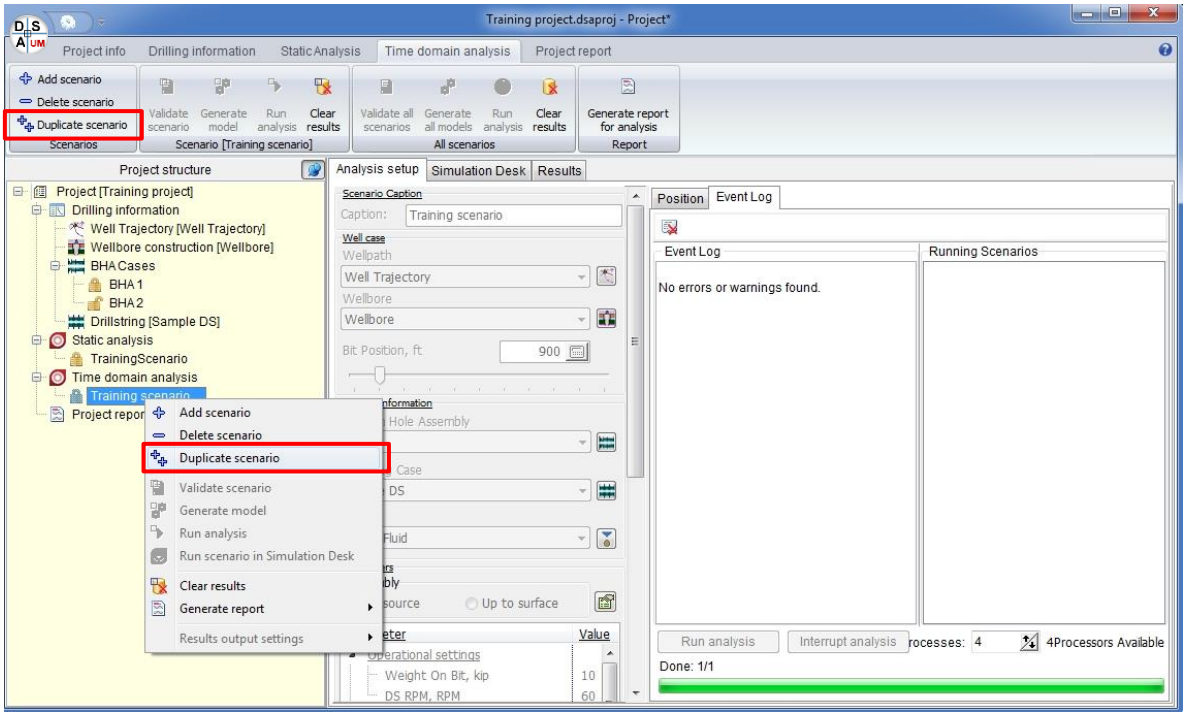
Step 6.27. Click the **Stop replay**  button left to interrupt the player. The outputs will be cleared, the assembly position will be restored to the initial.

Note: The process is paused automatically after reach of the end of the record. One needs to **Stop replay** to interrupt the process finally.

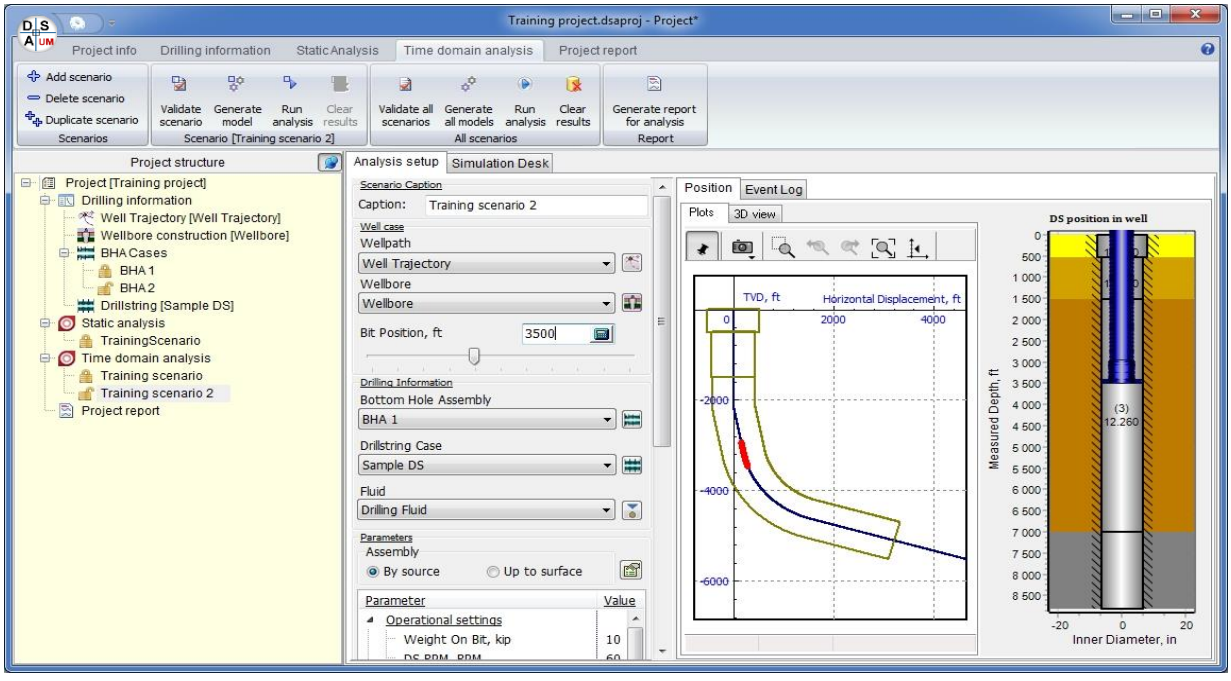
2.5.5. Multi-scenario run

DSA GUI enables two variants of Time Domain Analysis processing: *Simulation Desk* and multi-scenario solver. In this section the step-by-step description of the multi-scenario processing is considered.

Step 6.28. Select the **Analysis setup** page on the right panel, and duplicate the **Training scenario** by popup menu or head menu command.

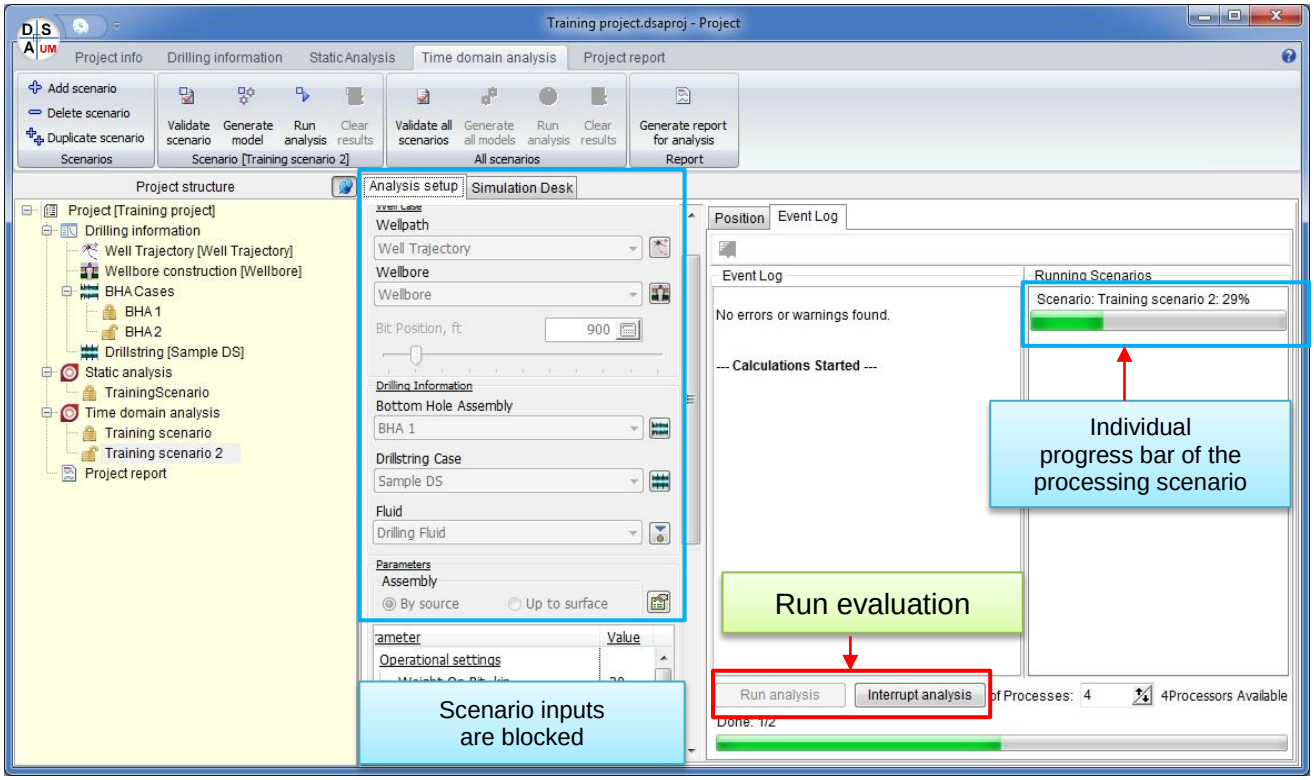


Step 6.29. Select and rename the new scenario; set scenario Caption equal to **Training Scenario 2.** Change WOB to 20 kip.



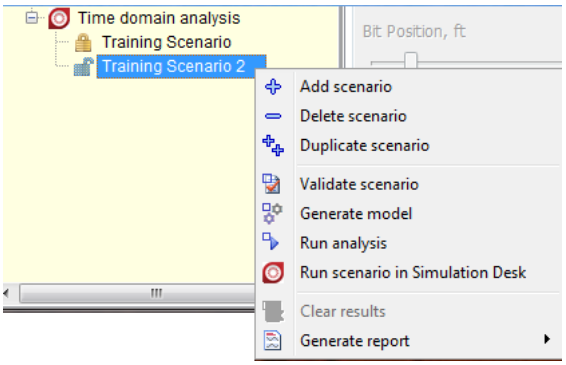
Step 6.30. Select **Event Log** page, and click on **Run Analysis** button to start the evaluation of the described scenario.

Individual progress bar will be added to the right **Running Scenarios** panel; **"--- Calculations Started ---"** record will be added to the **Event Log**; scenario inputs will be blocked automatically.

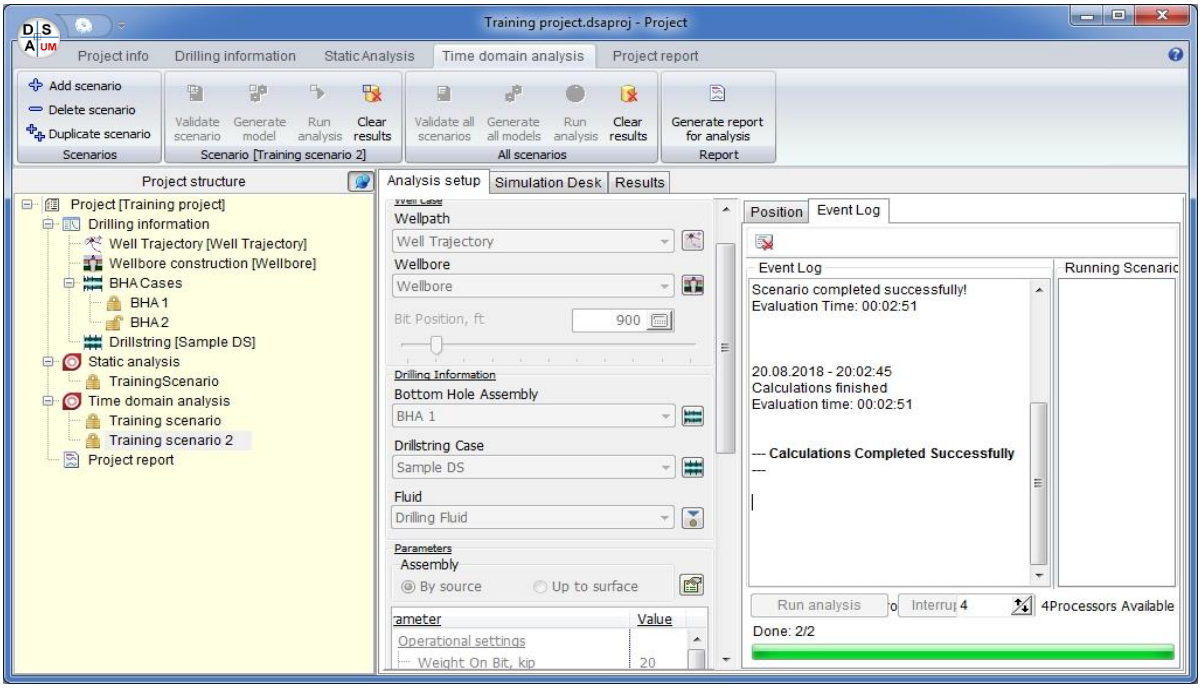


Note: Input data validation, and project saving is carried out automatically before process is running.

Note: One can use **Run all scenarios** button from the *Project tree* popup menu or head menu to start the evaluation; the **Run scenario** button enables run of the selected scenario only.



Scenario progress bar displays the current progress of the evaluation procedure. The progress is closed after completion of the scenario calculation; the scenario analysis details are added to the **Event Log**.

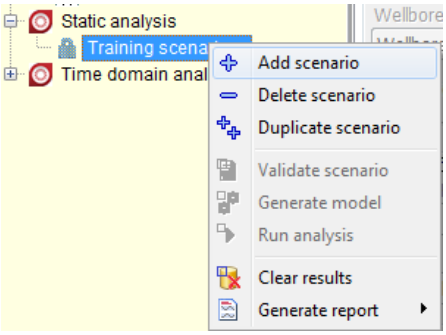


'Time Domain Analysis Calculations Completed Successfully' message will be generated after the completion of all running scenarios (single scenario in our case). Normally, calculation of the scenario should take about 60 seconds.

Step 6.31. Click **Ok** to close the message.

Results for the scenario has been calculated and stored in the scenario folder on hard drive; calculated scenarios are marked by icon in the Project tree.

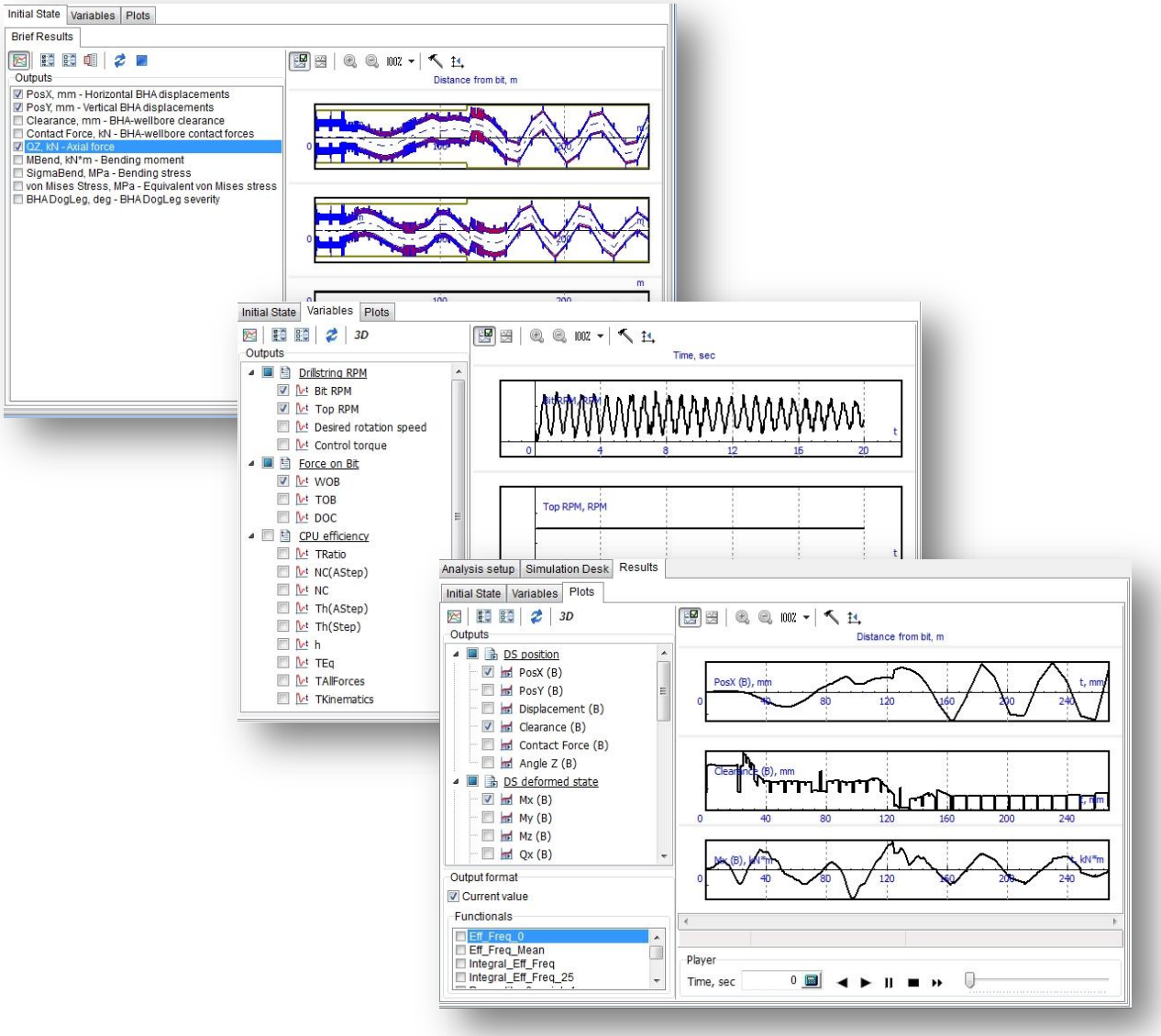
Step 6.32. Select the **Training Scenario** node in the Project tree, and open pop up menu. The **Clear results** option is available now; one needs to clear the results to modify description of the scenario. Close the pop up.



2.5.6. Results output

DSA GUI provides the various tools for 2D graphical output of the calculated time histories of kinematical characteristics, forces and stresses in the assembly parts:

- Initial state: set of 2D plots of the various result items on the single page for the initial equilibrium state of the model.
- Variables: 2D plots of variables vs. model time.
- Plots: 2D plots of output distribution along the assembly.

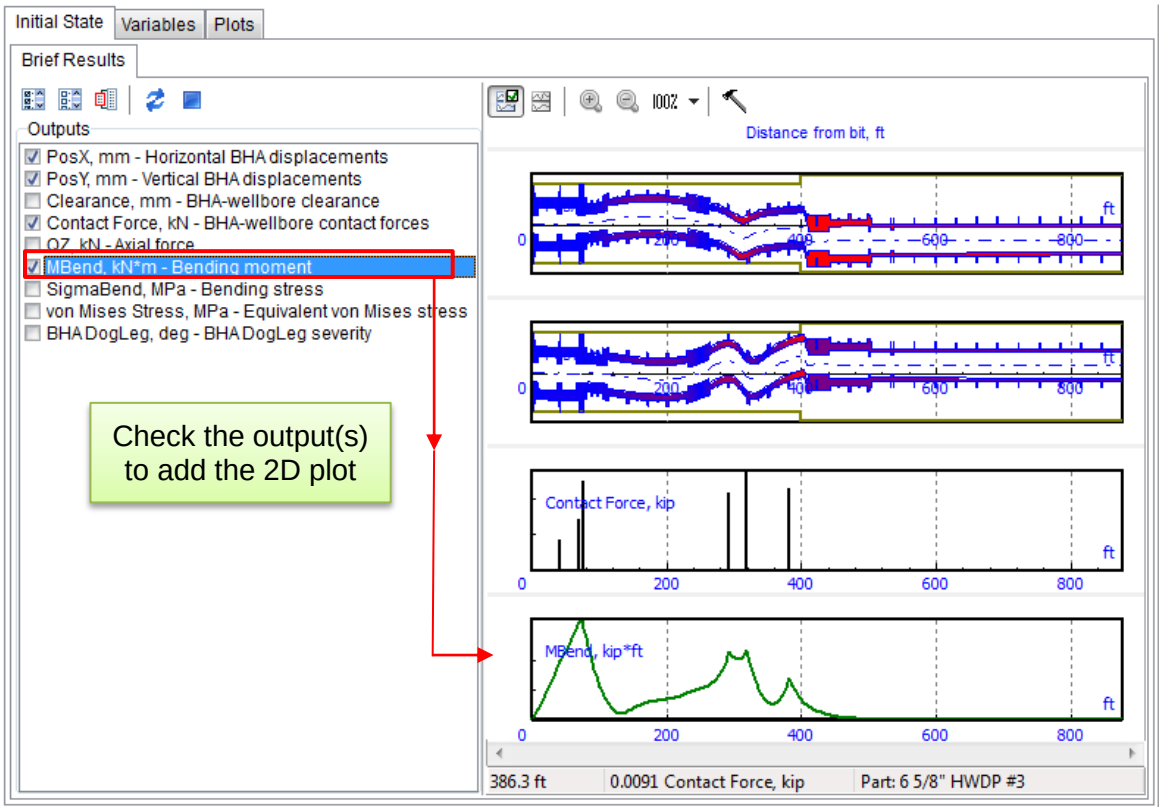


Step 6.33. Select the calculated scenario node (**Training scenario 2**) in the *Project tree* to make the **Results** page on the right page control visible, and go to the page.

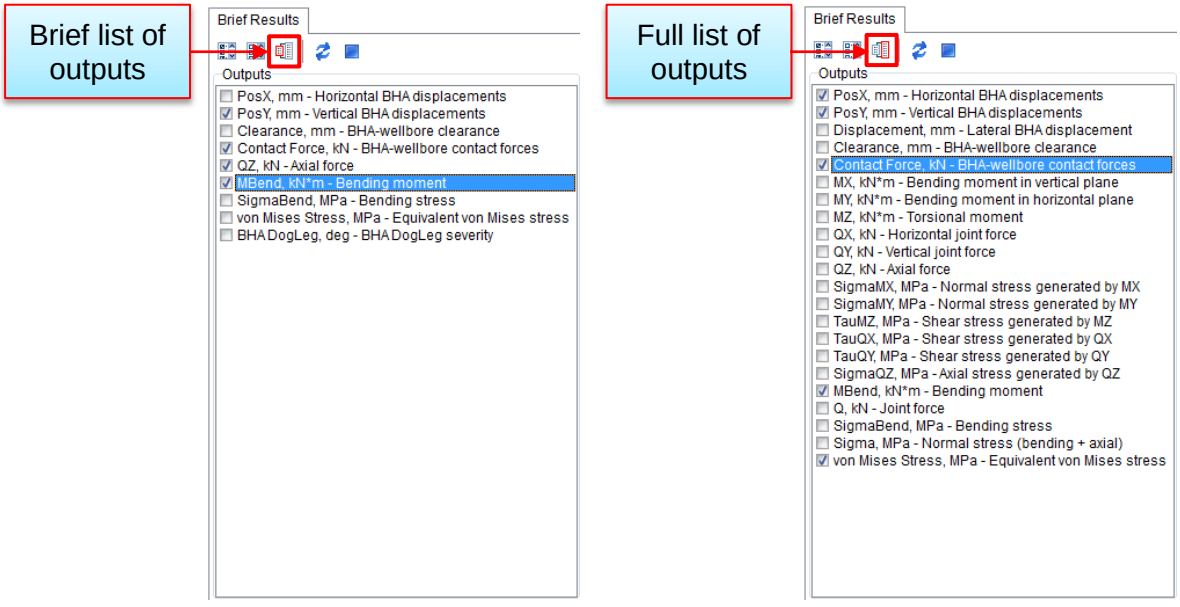
2.5.6.1. Initial State

Step 6.34. Select **Results** | **Initial State** page to display the set of 2D plots of the various result items for the initial state the time domain simulation was started.

One can select an output from the check box list – the corresponded dependency of the selected output vs. distance from bit will be displayed on the 2D plot on the right panel.

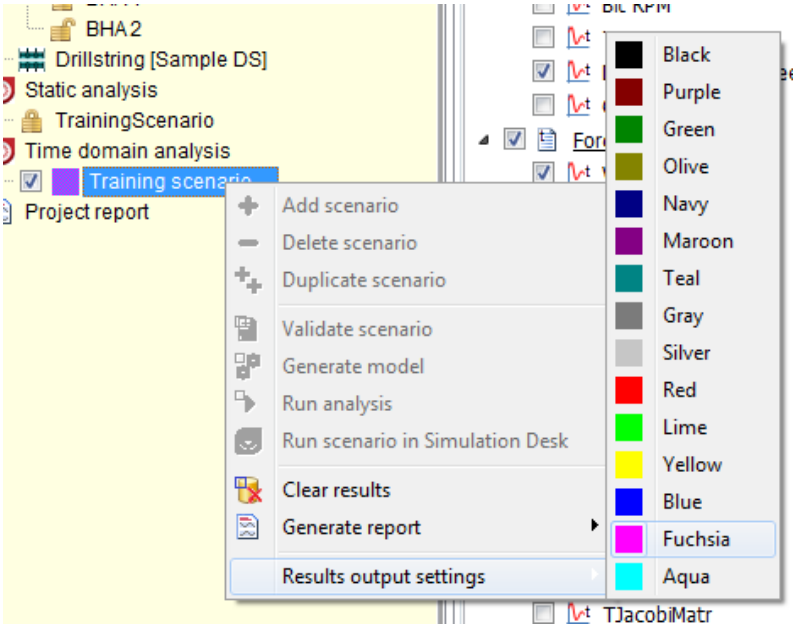


Step 6.35. Click on button to switch between brief and full list of outputs.



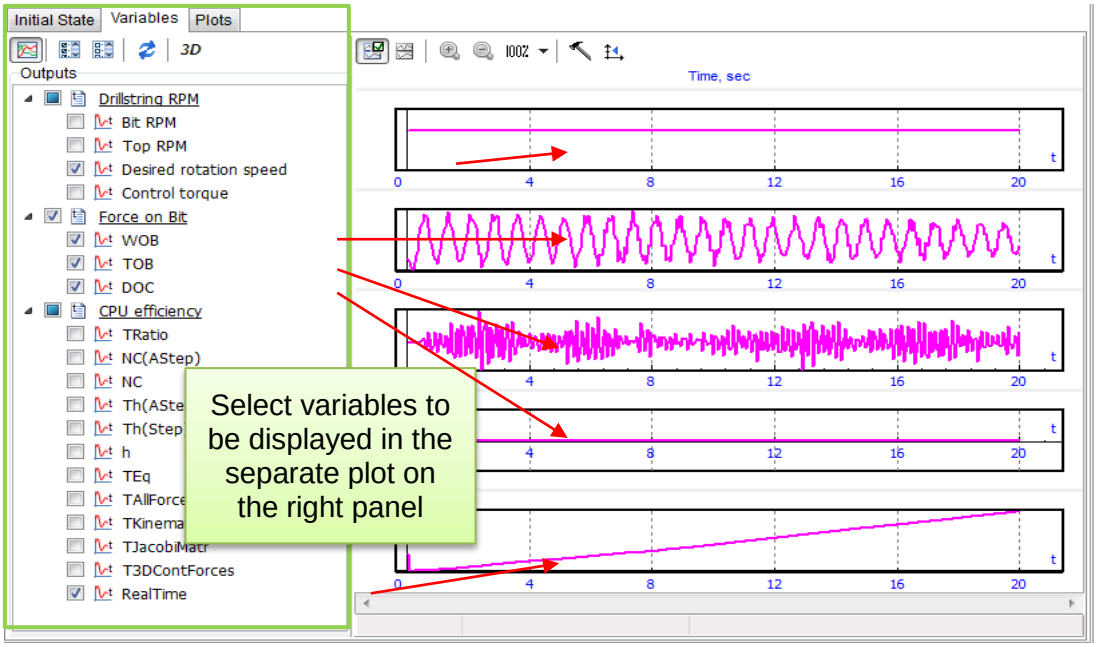
2.5.6.2. Variables

Step 6.36. Change color of scenario plots:



Step 6.37. Select **Results** | **Variables** page to study the time domain dependencies of scalar variables: RPM controls, Force on bit.

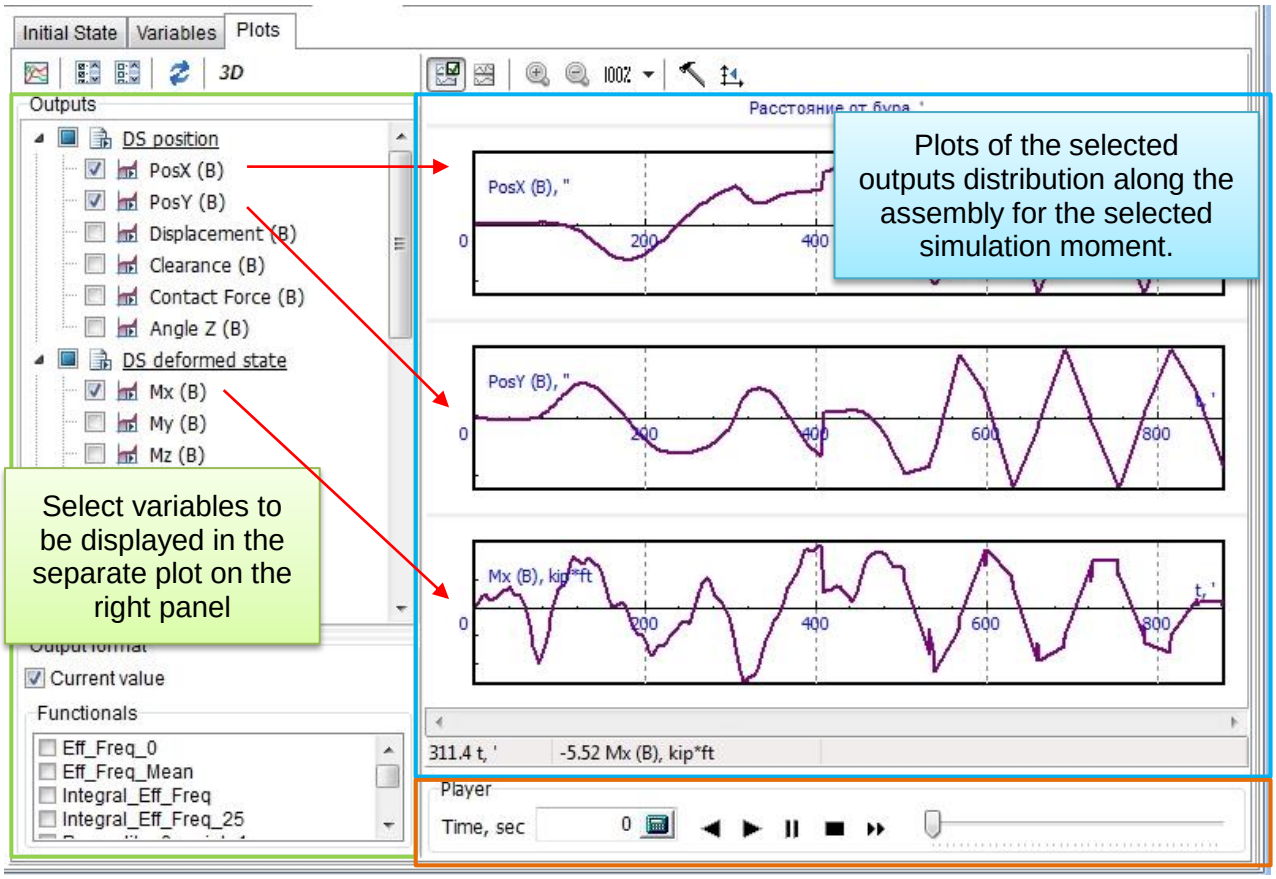
Step 6.38. Select several outputs in the list to show results of scenario simulation. The list of calculated scalar variables is displayed on the left panel; one can select any of the variables to display them on separate plots on the right panel.



2.5.6.3. Plots

Step 6.39. Select **Results** | **Plots** page to study the distributions of displacements, internal forces and stresses, etc. along the assembly in the time domain.

The list of calculated distributions (2D plot) is displayed on the left panel; one can select any of the variables to display them on separate plots on the right panel.



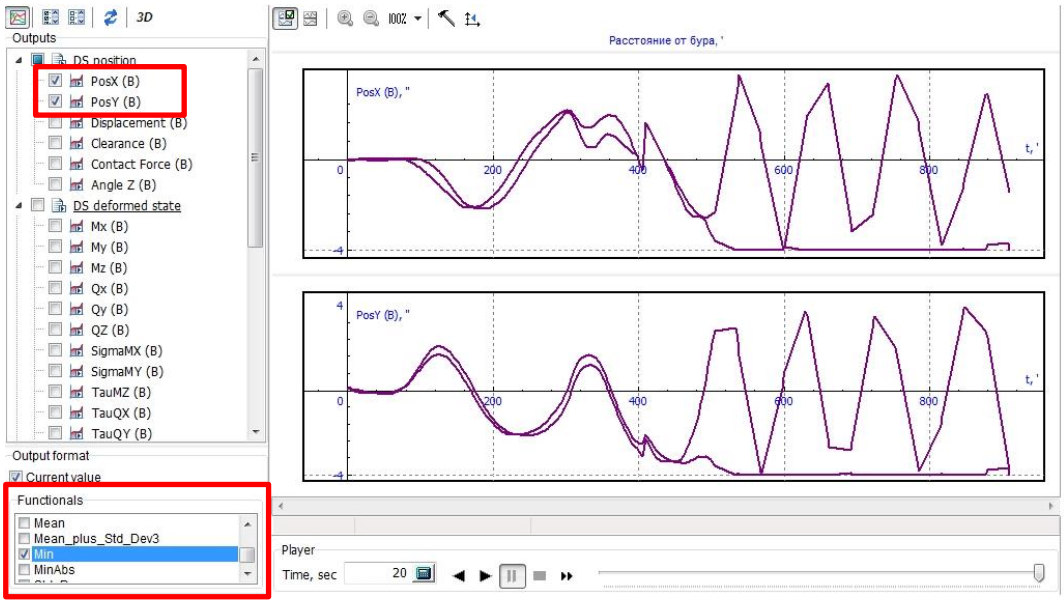
Step 6.40. Select **PosX**, **PosY** and **Mx** outputs in the list to display distributions of lateral displacements in wellbore reference frame, and bending moment along the assembly.

Step 6.41. Click on **Run player** button to animate the output variation histories fixed during simulation.

Step 6.42. Click on **Stop player** to interrupt the animation.

2.5.6.4. Functionals

Step 6.43. Look at the **Functionals** and choose ‘Min’. On plots will appear min values of output for simulation time.



2.5.7. Excitations

DSA TDA Analysis GUI enables description of various forced excitations for the assembly model (available in **Advanced User** mode):

- Inertial, force, or torque excitations.
- Harmonic excitations or excitation records from text (*.csv) file.

Use the tool which allows you to specify a harmonic or force excitations to any part of BHA.

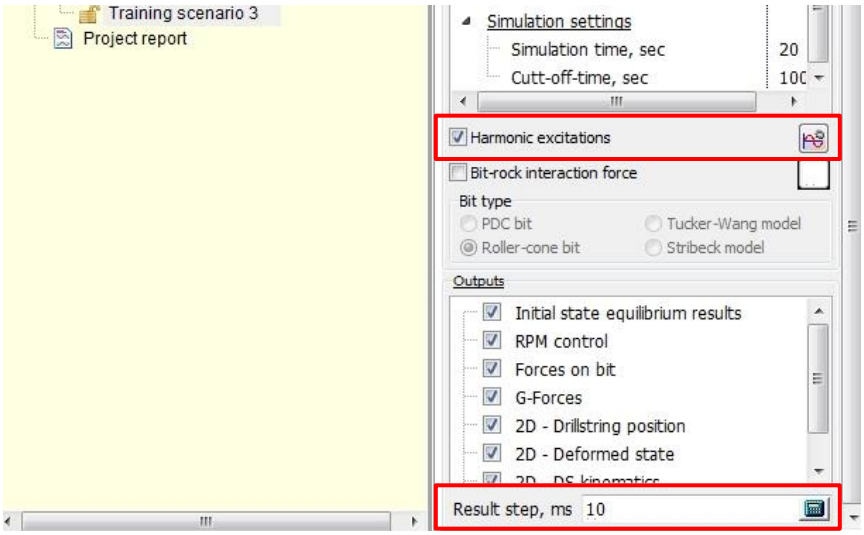
Step 6.44. Duplicate 'Training scenario' and rename it to '**Training scenario 3**'

Step 6.45. Enable **Harmonic excitations** and disable **Bit-rock interaction** force.

Step 6.46. Change **Result step** to 10 ms.

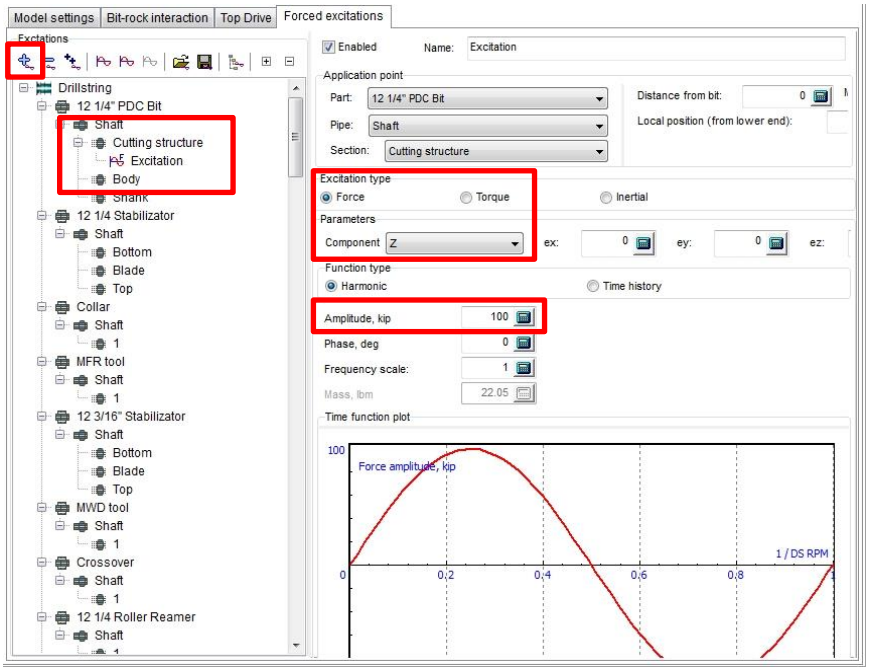
Step 6.47. Open tab Advanced settings -> **Harmonic Excitations** (available in **Advanced User** mode).

Step 6.48. Choose **12 ¼ Pdc-bit** (the first detail) and click button . New excitation parameters will be displayed on the right panel.



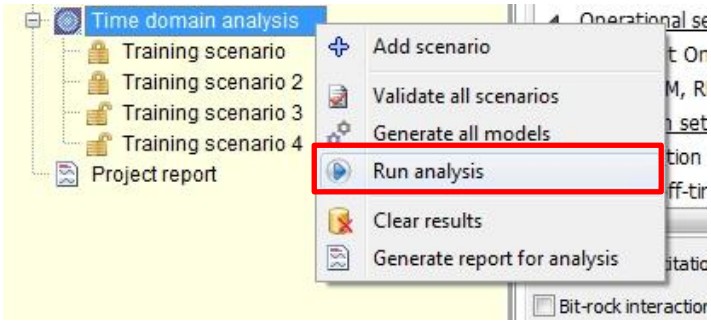
Step 6.49. Set the following parameters for the excitation:

- Excitation type: Force
- Component: Z (along the axis of the bore hole).
- Amplitude: 100 kip



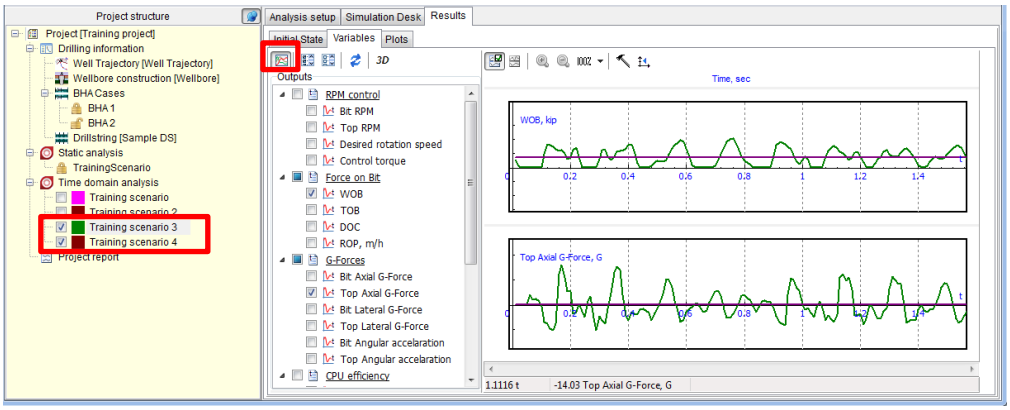
Step 6.50. Duplicate **Training scenario 3** and disabled **Harmonic excitations**.

Run calculation for last two scenarios.



2.5.8. Comparison mode

Step 6.51. After calculation go to tab **Results -> Variables**. Enable **comparison** mode  .
When comparing two last scenarios we can see influence of axial excitations to WOB and Axial G-forces.



Step 6.52. Copy the first scenario (Training scenario). Go to bit-rock interaction tab. Change to zero next stochastic parameters :

Stohastic components	
Stohastic lateral force RMS, lbf	0
Stohastic TOB RMS, lbf*ft	0
Stohastic vertical irregularities RMS, "	0
Stohastic frequency scale	5
Stohastic axial force scale, %	0

Run scenario and look at the **Results -> Variables** after calculation.

To compare several outputs for **Training Scenario** (with irregularities) and **current** scenario (without).