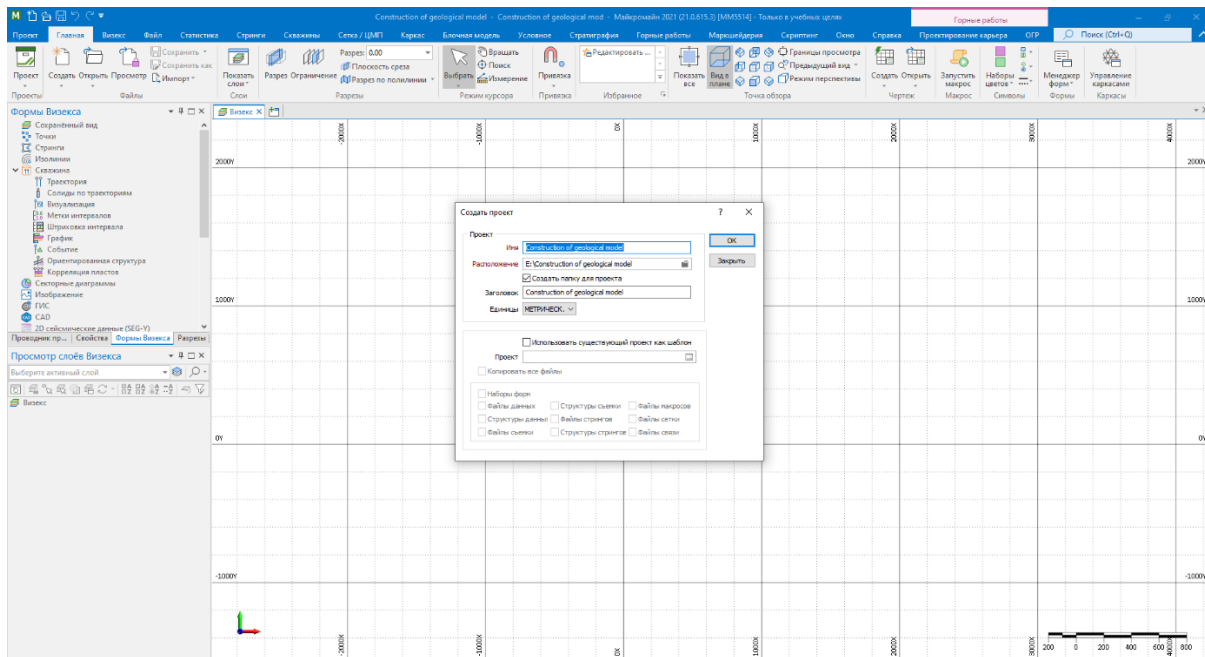


Construction of geological model by Micromine mining and geological information system

On ,the **Home** tab, in the **Projects** group select **File | Project | New** to create a project

In the Project New form, enter a name for the new project.

The
must



name
follow

Windows file naming conventions. The application supports the use of long filenames. Filenames cannot contain the following characters: \ / : * ? " < > |

Location

Click the ellipsis button to specify a location for the new project. The name of the new project will be the name you have entered.

Create directory for project

If the Create directory for project check box option IS selected, the project name will be appended to the location path and the project will be created in a new directory. If the Create directory for project check box option is NOT selected, the project will be created in the location specified.

Project Title

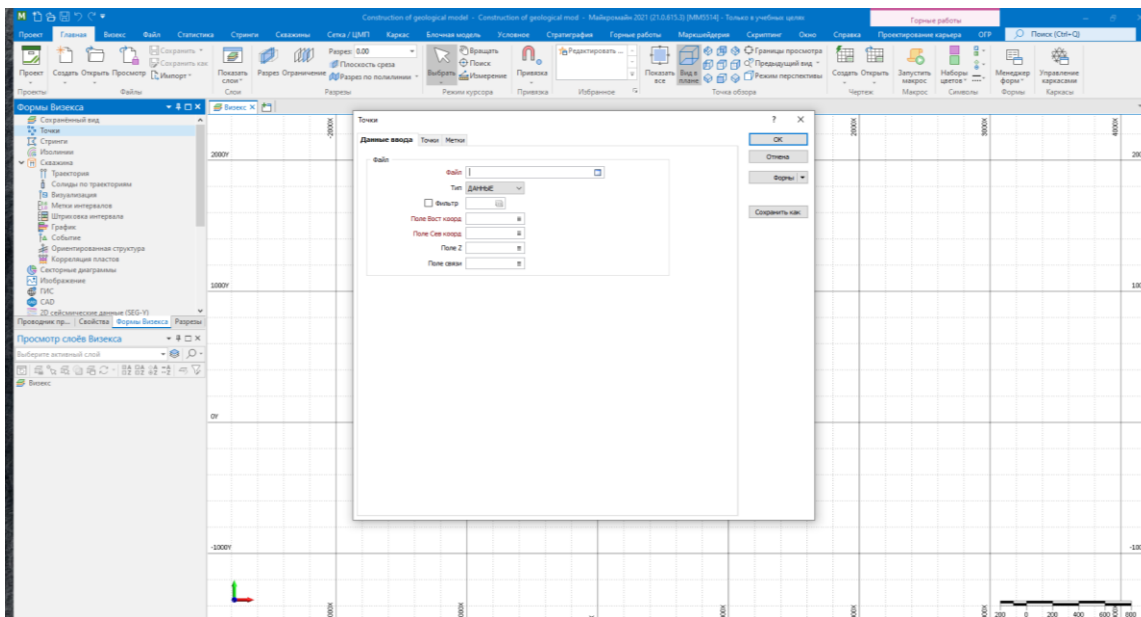
Optionally, enter a short but descriptive title for the project you are creating.

Units

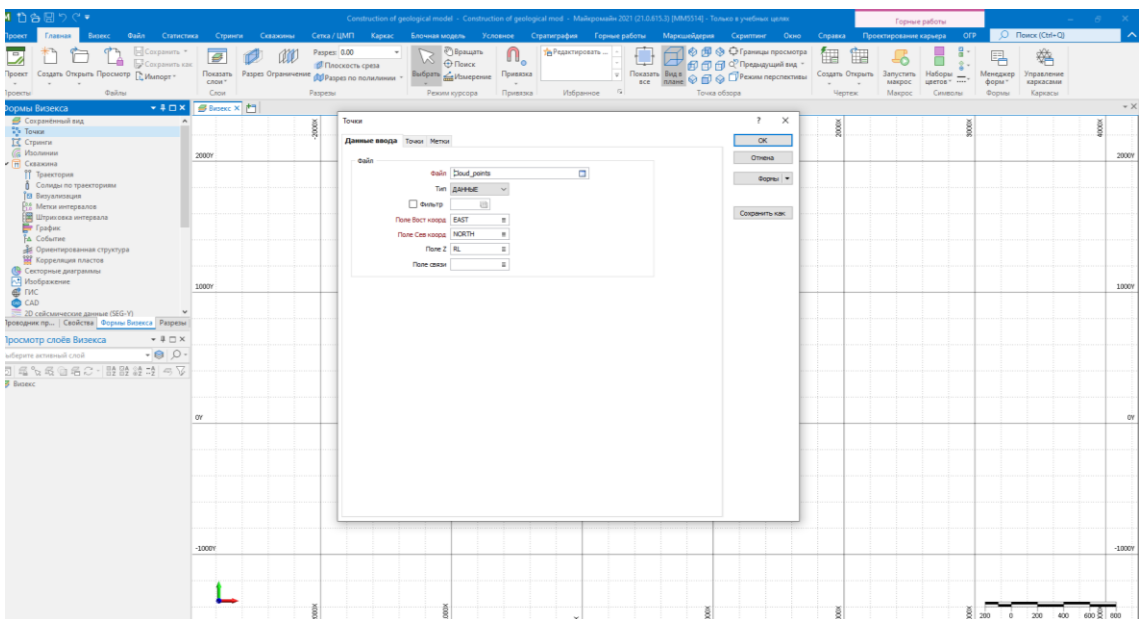
Measurement units must be constant throughout a project. Select METRIC or IMPERIAL from the drop-down list.

If units are IMPERIAL, grade calculations will use TONNAGE FACTOR instead of DENSITY, SHORT TONS instead of METRIC TONNES, and CUBIC FEET instead of CUBIC METRES.

On the Visex forms, double-click on the dots



From the list, select Cloud_points.DAT



New DTM

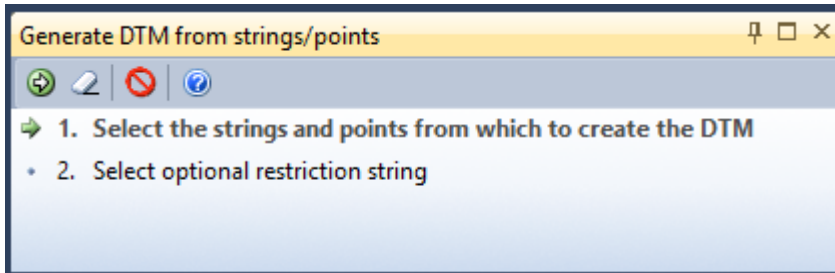
On the **Grid / DTM** tab, in the **DTM Create** group, click **Interactive New DTM** to generate a DTM from the strings and/or points selected in Vizex.



You can create a DTM from a layer that is not the active edit layer. This makes it possible, for example, to build a DTM from CAD/GIS data, without having to convert that data into Micromine String format.

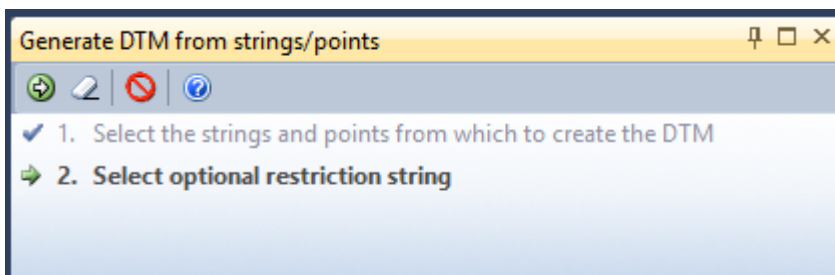
If no objects are selected the Selection Assistant will guide you through the process:

1. Select the strings or points to generate a DTM from.

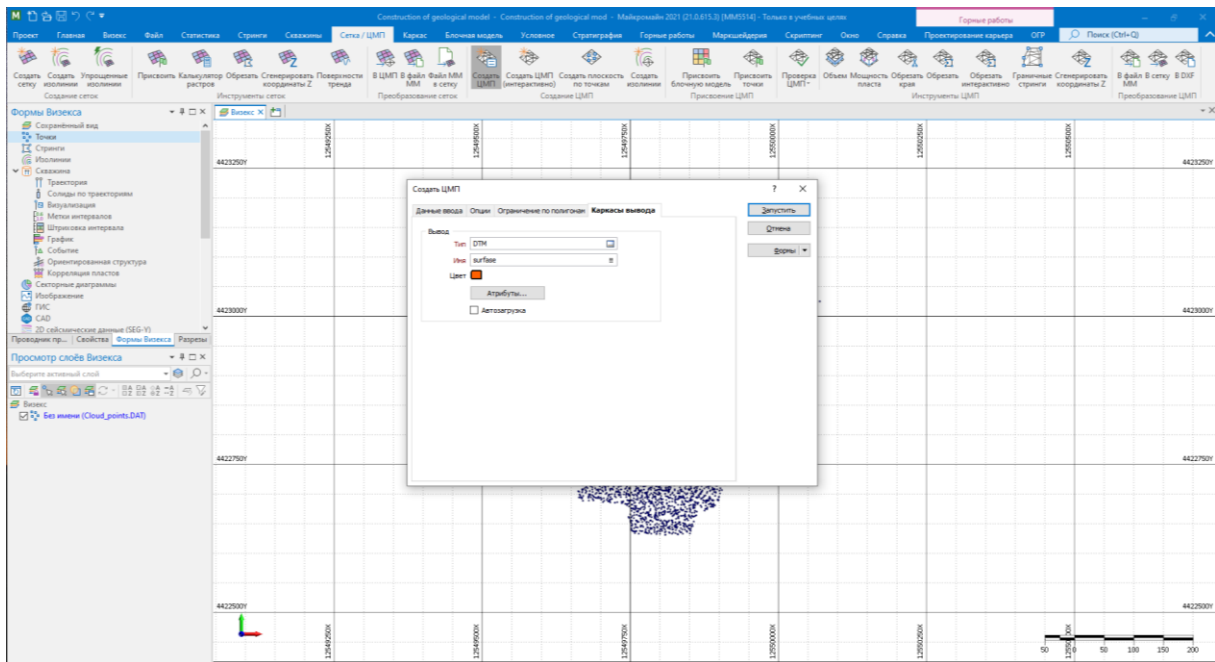


In the case of strings, you will generally select the top or outer string before generating a wireframe. This string is used as the boundary string and no triangles are generated outside it. However, if you select an intermediate string, this will be used as the boundary string.

2. Optionally, you can select a restriction string to limit the extent of the triangulation.



3. Accept the string selections to invoke the DTM Create form.
4. In the DTM Create form, specify the properties of the Output wireframe and set [Processing Options](#).



Type

Select one of the wireframe types in the current project by double-clicking in the Type input box. A wireframe type is a name used to categorise a wireframe. All wireframes, no matter what their type, are the same - collections of triangles, attributes and metadata.

Name

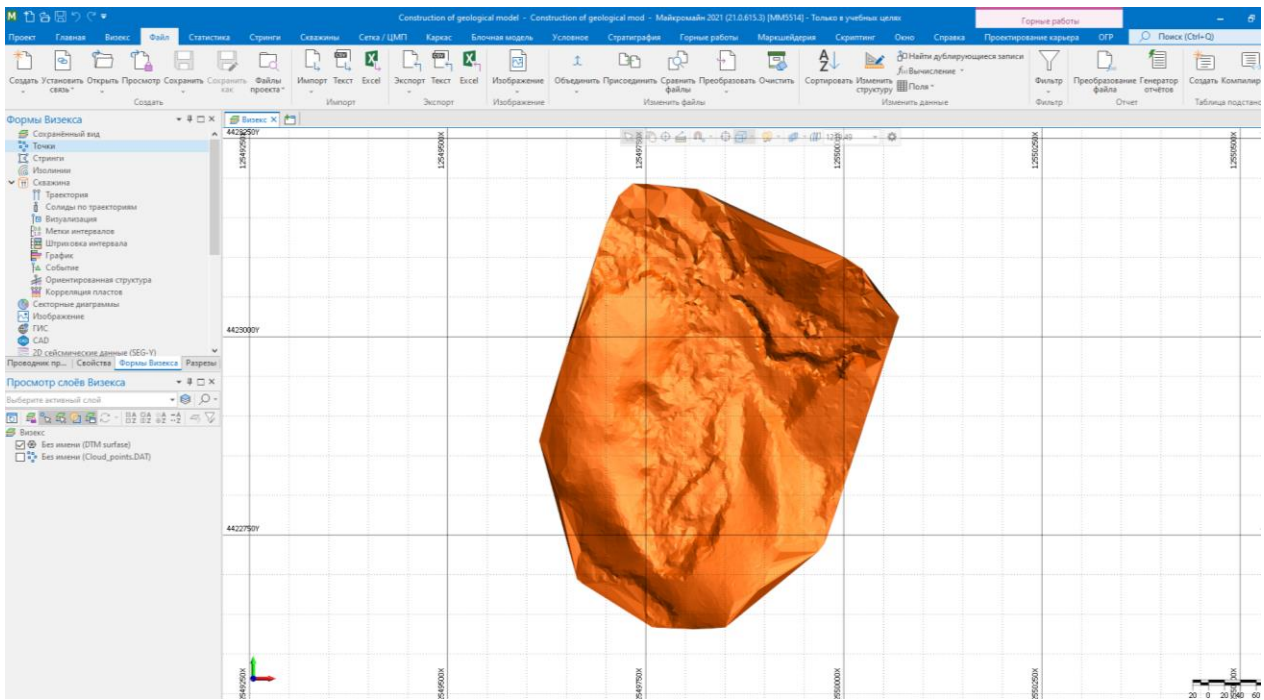
Double-click (or click on the Select icon) to select the name of the wireframe you want to use in the process.

Attributes

Click the **Attributes** button to set standard and user-defined attributes for the wireframe. When you save a wireframe, you should set standard attributes (colour and title settings etc.).

Auto load

Select this option to load the wireframe into the display once it has been created. If you do not select this option, you will need to double-click on the Wireframe Form Set Type in the Vizex Layer Types pane to display it.



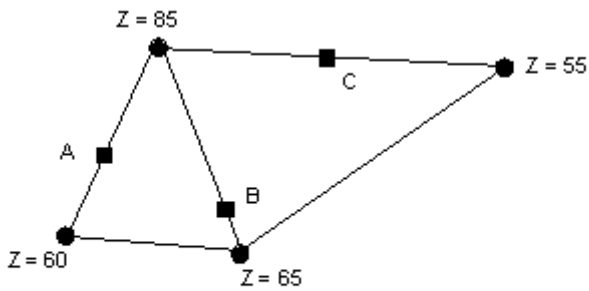
Contours

On the **Grid / DTM** tab, in the **DTM Create** group, click **Generate Contours** to generate a Contour file from a DTM.



Contours are lines connecting points of equal Z value. When you create a DTM, a Z field is defined and these Z values can be contoured using the Contours function. The DTM is a triangular mesh with each vertex being a point in the data file that was used to create the DTM. Each vertex has an associated Z value and each triangle side defines a constant slope between two known points.

So, if two vertices of the same triangle have Z values of 19m and 21m, then half way along the connecting side the Z value will be 20m. If a specific Z value “enters” a triangle by intersecting one of its sides, it must “leave” the triangle by one of the other two sides, thereby “entering” an adjacent triangle. A contour line is generated by tracking a specific Z value in this way.



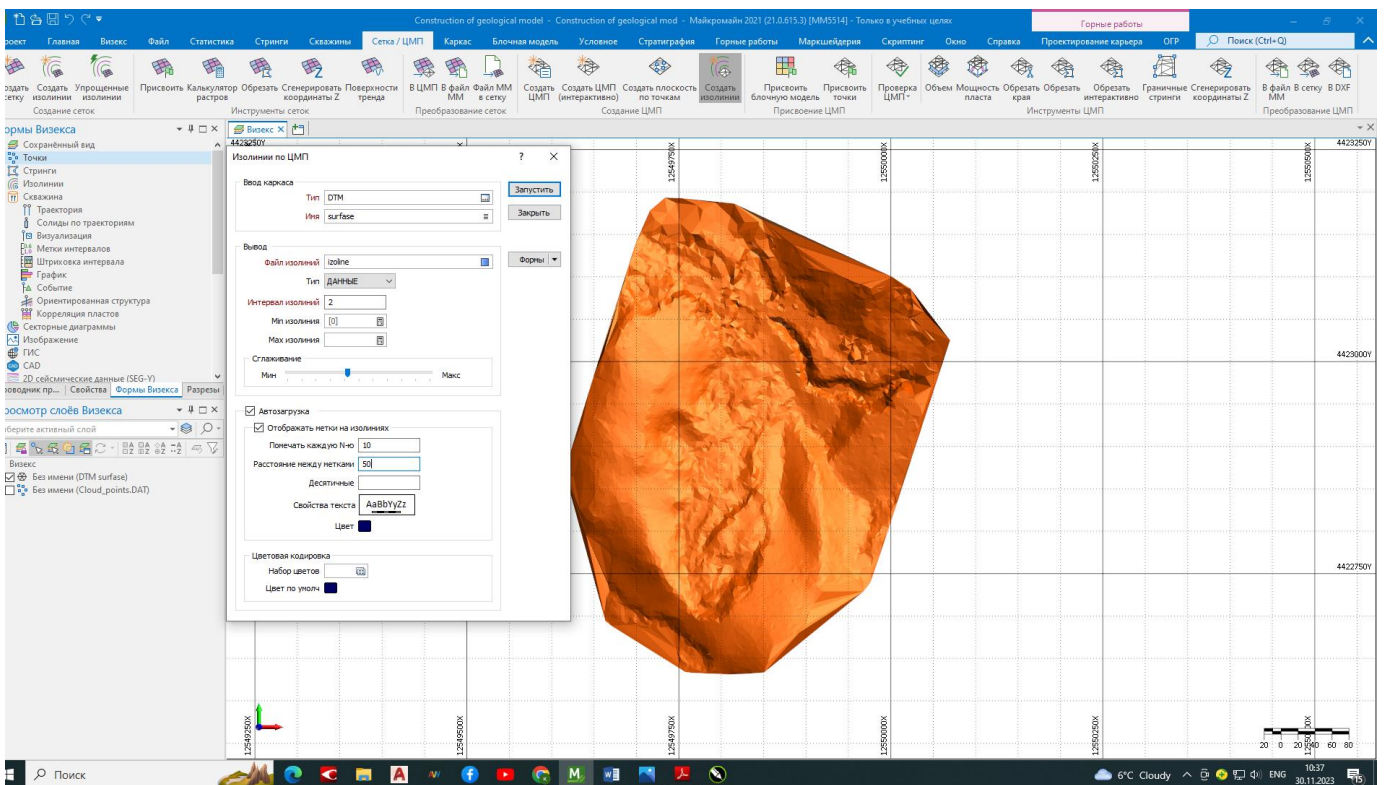
Contour 70 would pass through A, B and C

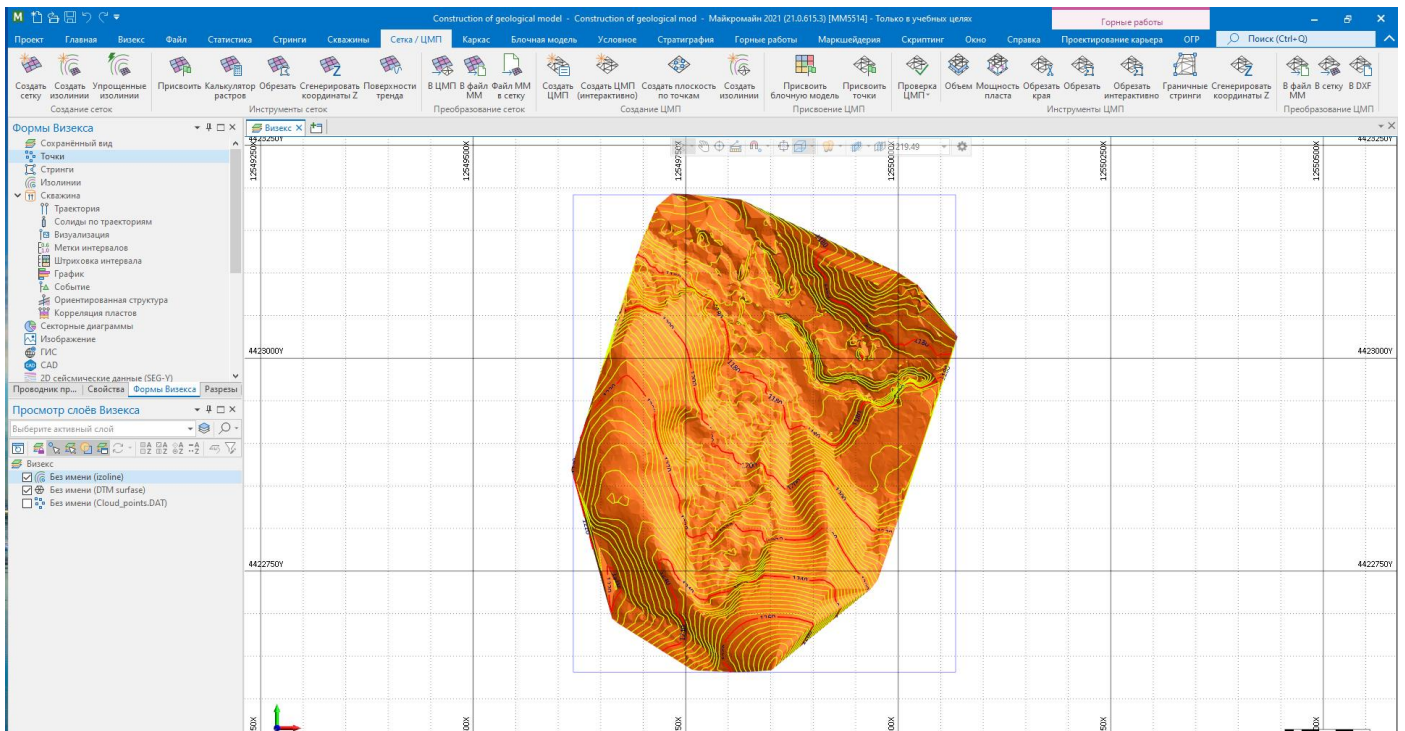
Interpolation of Z values

The Contours file can be loaded as a layer in Vizex. The Contour file has the fields X, Y, Z and JOIN. When displaying contours, Z is defined as the String field and JOIN as the Join field, to ensure that the contours display correctly. Labels are not written to the Contour file.

Wireframe Input

In the DTM Contour form, specify the **Type** and **Name** of the input wireframe:





Output

Contours written to the Output file are based on the Z values of the DTM.

Contour interval

Enter a Contour interval which defines the spacing between contour lines.

Contour minimum and maximum

Enter the Contour minimum and maximum values to restrict the range of contours.

Smoothing

If Smoothing is not applied, then the intercepts on the triangle sides will be connected by straight lines, often resulting in “jagged” contours. Use the **slider bar** to determine the degree of smoothing of the contour lines.

Straight line contours will never cross, however there is no guarantee that smoothed contours will not cross.

Label every Nth

The Contour interval parameter defines which contour lines will be labelled.

The *Label every Nth* value is defined in units of “contours”. A value of 5 will label every fifth contour starting from the lowest.

So if, for example, a data set is being contoured with a minimum contour of 0, a maximum of 100, and a contour interval of 5, setting *Label every Nth* to 5 would mean that contours are labelled at 0, 25, 50, 75 and 100. A *Label every Nth* value of 4 will put the contours at 0, 20, 40, 60, 80 and 100.

If no Contour minimum is set, a default Contour minimum is applied which ensures that labels are positioned on contours which are multiples of the Contour interval * the Label (every nth) spacing.

Distance apart

Type in a value in grid units for the Distance apart, that is, the distance between adjacent labels as measured along the contour line.

A small label distance will result in contour labels being closely spaced. A large distance value may result in no labels appearing on a contour line at all, particularly if the line is short. Labels are only drawn on a contour line if it is longer than the distance apart.

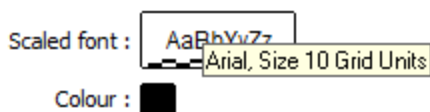
Decimals

Specify the number of Decimal places that will be displayed when labelling numeric values.

Scaled font

Double-click on the Scaled font preview box to select a scaled font for the labels.

The Font preview size in a form is always shown as 12 point, regardless of the font size chosen in the **Text Properties** dialog. However, if you hover the mouse over the Font preview box, details of the current font are shown. The size of the font is shown as "Grid Units".



Colour Coding

Double click (F3) to select the **Colour set** and the **Default colour** that will be used to control the display colour.

The colour set maps colours to text strings or numeric ranges. Right click (F4) to create or edit a colour set. The Default colour will be used when a colour set is not defined.

Excel

On the **File** tab, in the **Import** group, click **Excel** to import data directly from the worksheets in a Microsoft Excel workbook. The data source connection is managed by the application.

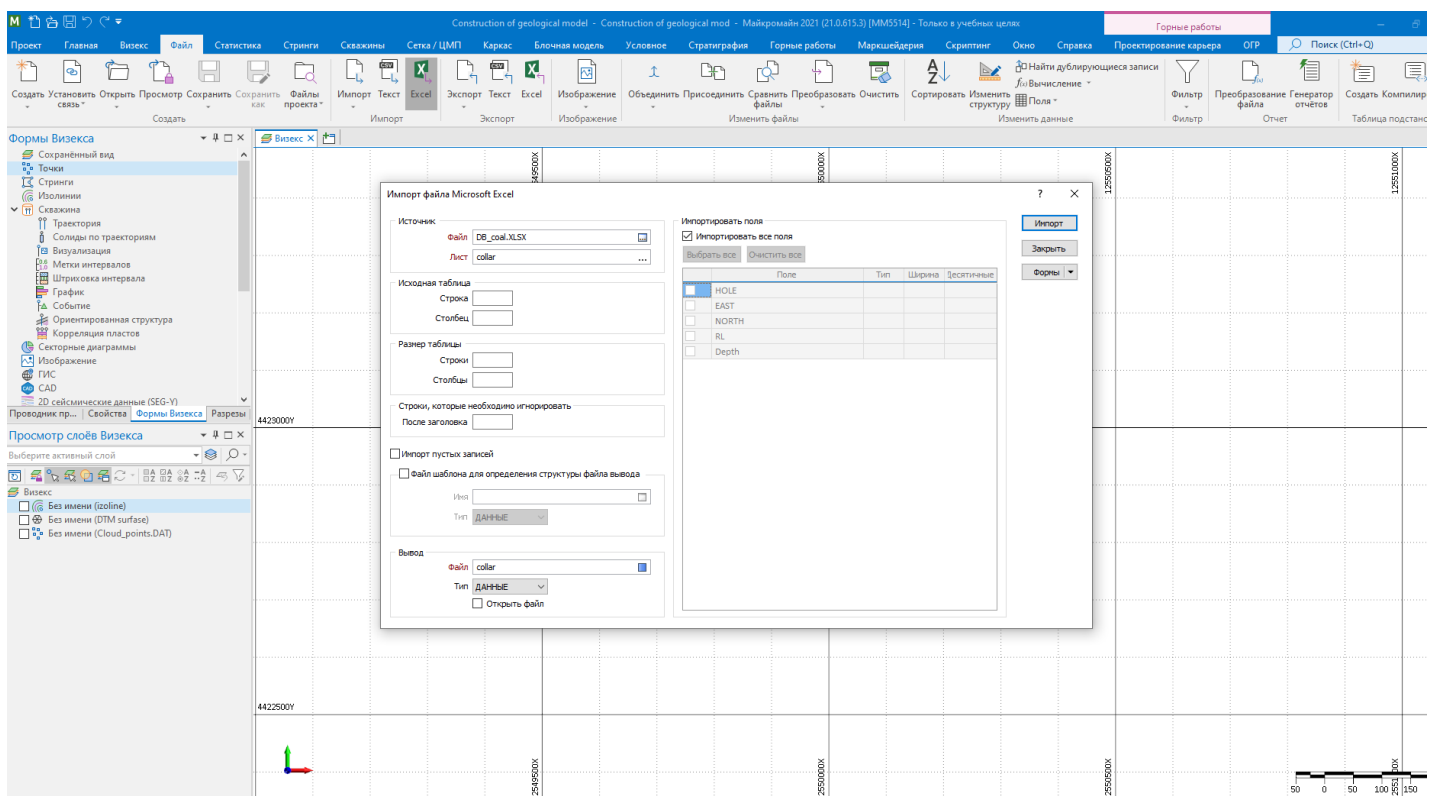


Use Template File to Define Output File Structure

If this option is selected, the data in the source file will be written to a new file which is based on the template file you have specified.

The width and decimal precision of each field is based upon the values encountered when the rows are scanned. Values in the Source file are mapped sequentially to the fields in the Output file and data types are converted where necessary.

If you know that the Type, Width or Decimal Precision properties of a field are incorrect, you can change them in the Fields to Import grid.



Output

File

Double-click (or click on the Select icon) to select the name of the Target file. The imported data will be saved in this file. If the file exists, you will be asked if you want to overwrite or append to it.

The process will only append to the selected fields if the data being imported is compatible with those fields.

Auto open file for editing

Select this option to open the file for editing in the File Editor once it has been imported.

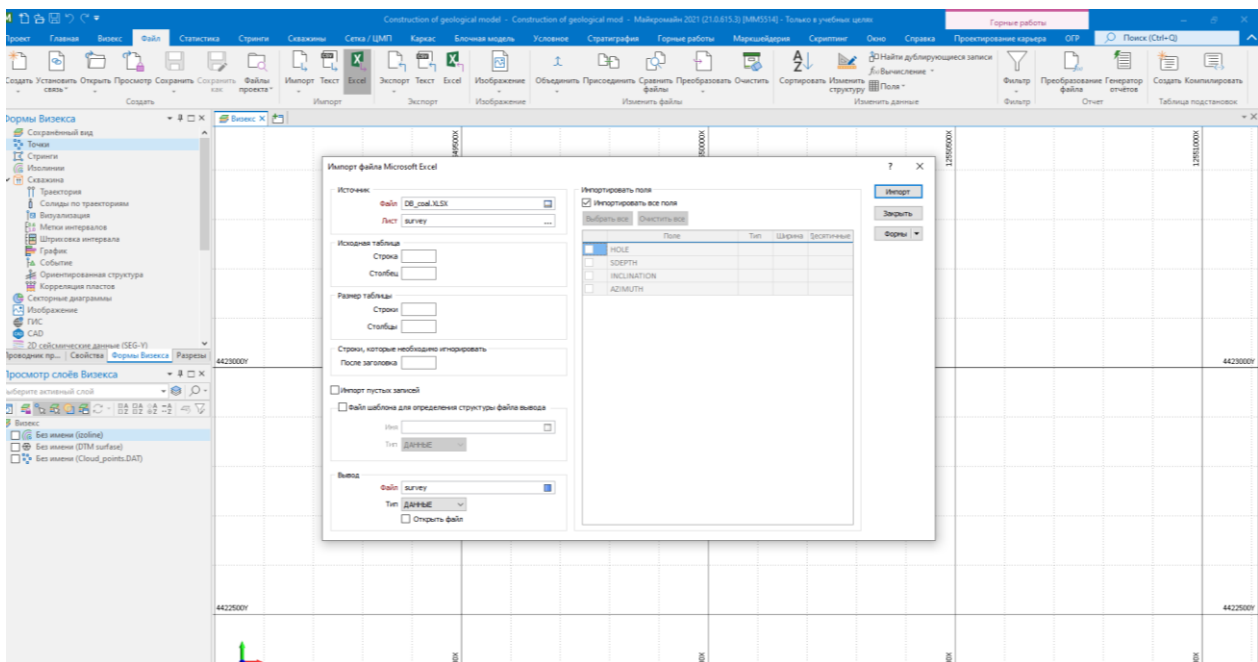
Fields to import

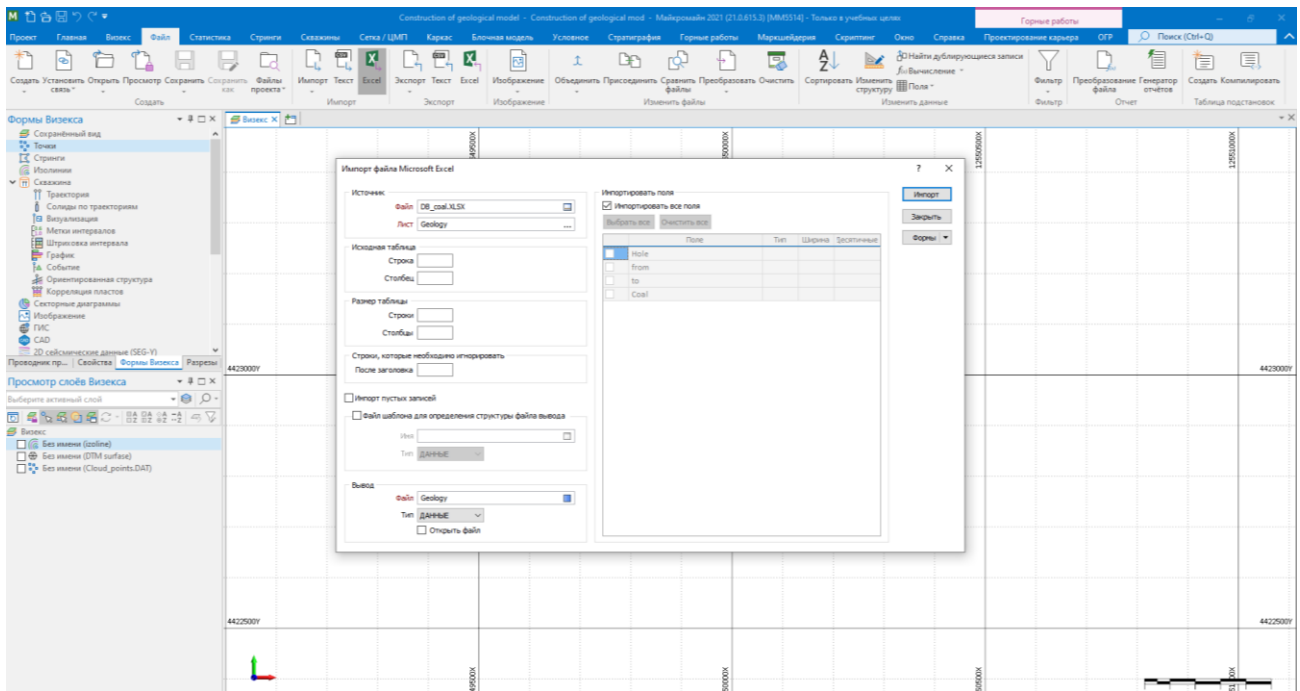
Click the *Import all fields* check box to import all fields.

Otherwise click the **Select all** button or the **Clear All** button prior to individually selecting the fields you want to import.

Import

Click **Import** to complete the process. A message will appear when the import is complete. You can right-click in the Target file box to view the imported data.





New Drillhole Database

On the **Drillhole** tab, in the **Database** group, click **New** to create a Drillhole Database by referencing a Collar file and (optional) associated Survey, Interval and Event files.



The Database Properties form is displayed whenever you create or edit a drillhole database:

Properties

If downhole data is being used, then data in a Collar file and (optionally) a Survey file will be imported and used to generate trace coordinates. External interval and Event files can also be associated with the database. The fields in these files can be used to classify and colour code the trace display.

Collar file

Specify a **Collar file**, enter field settings and optionally apply a **filter** to create a subset of the data in the file. The *Hole* field should be a field containing unique hole identifier values.

Hole ID fields 1, 2, 3

Double-click to select the name of the fields that uniquely identify each hole in the file. Up to 3 Hole ID fields can be specified.

This can be useful, for example, when the holes imported from disparate datasets are known to have duplicate keys. When more than one Hole ID is specified, a unique ID (a concatenation of the Hole IDs) is used to query the database.

East and North and Z fields

Specify the names of the fields in which Easting, Northing, and Z coordinates are stored in the trench file.

Total depth field

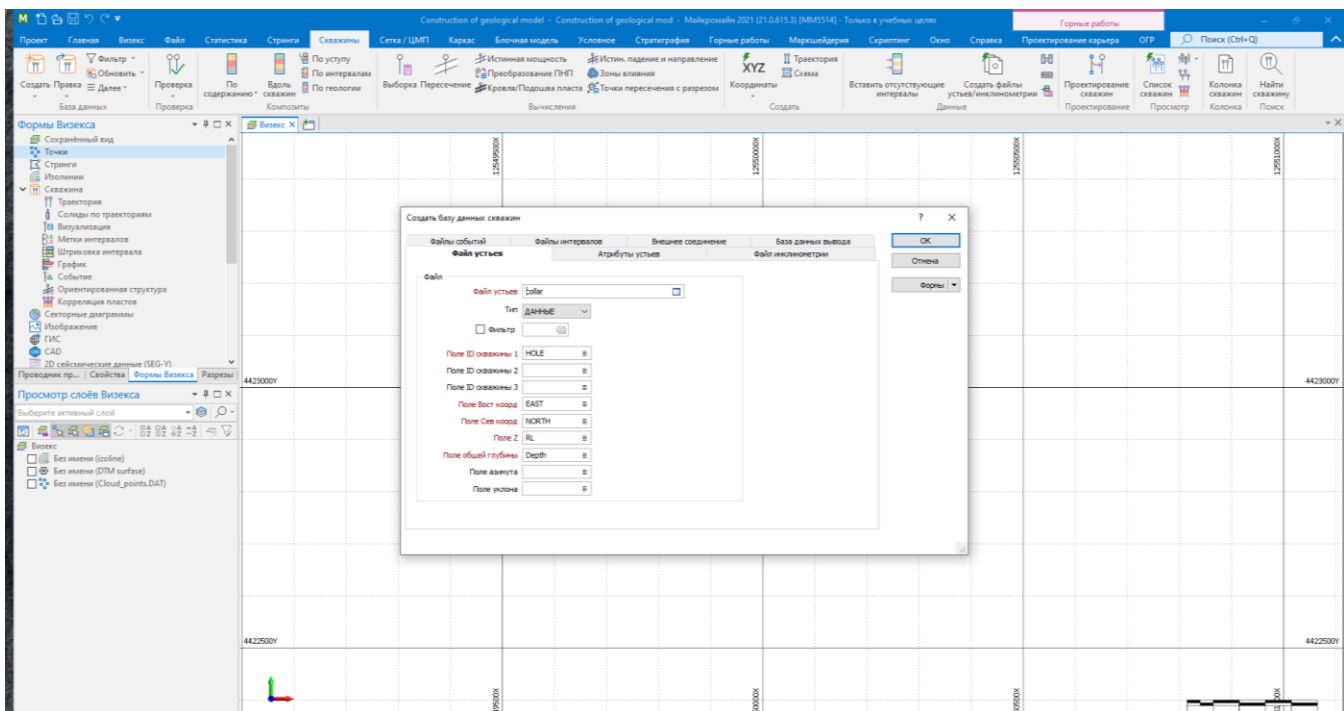
Enter the name of the field containing the total depth (length) of each drillhole.

Azimuth field (Collar)

Enter the name of the field containing drillhole collar azimuths. Azimuths are expressed in decimal degrees, with zero (0°) being grid North. Negative values and values greater than 360° are correctly evaluated.

Inclination field (Collar)

Enter the inclination field of the collar file. Inclinations are expressed in decimal degrees from -90 to 90, with negative values representing down holes.



Survey File

If downhole data is being used, then the data in a Collar file and (optionally) a Survey file can be imported and used to generate trace coordinates.

Downhole surveys

If survey data is to be used to generate the drillhole trace, select the *Downhole surveys* check box.

Survey file

Specify a Survey file, enter field settings and optionally apply a **filter** to create a subset of the data in the file.

Hole ID fields 1, 2, 3

Double-click to select the name of the fields that uniquely identify each hole in the file. Up to 3 Hole ID fields can be specified.

This can be useful, for example, when the holes imported from disparate datasets are known to have duplicate keys. When more than one Hole ID is specified, a unique ID (a concatenation of the Hole IDs) is used to query the database.

Survey depth field

Enter the name of the depth field in the Survey file. Survey depths must be in increasing order for each hole.

Azimuth field

Enter the name of the field containing drillhole azimuths. Azimuths are expressed in decimal degrees, with zero (0°) being grid North. Negative values and values greater than 360° are correctly evaluated.

Inclination field

Enter the inclination field of the Survey file. Inclinations are expressed in decimal degrees from -90 to 90, with negative values representing down holes.

Azimuth Correction

Optionally apply an [Azimuth Correction](#).

Apply to 1st azimuth

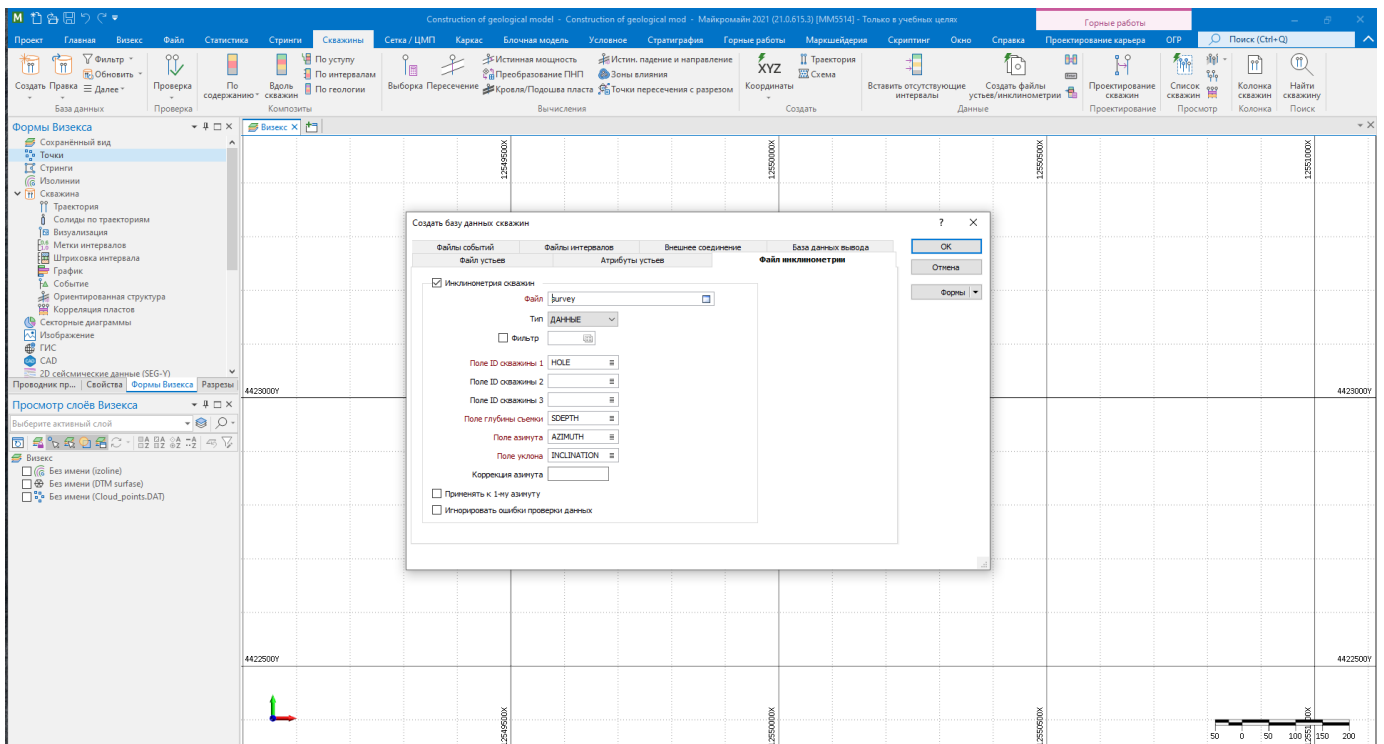
Select this option to apply the azimuth correction to the first and all subsequent azimuths.

Do not apply the azimuth correction to the first azimuth when collar surveys have an azimuth taken in terms of grid North and subsequent downhole azimuths are taken in terms of magnetic bearings.

Ignore validation errors

The state of the *Ignore validation errors* check box is saved in the database, indicating that you have opted not to check the validity of the file. Micromine will attempt to handle the file as well as possible without raising validation errors or warnings.

All drillhole validations that incorporate a drillhole database will honour the *Ignore validation errors* flag. If you wish to manually validate a file without clearing the flag, you can do so via the **Validate Drillhole** function on the **Drillhole** tab, in the **Validation** group. This is the only function that ignores these flags altogether.

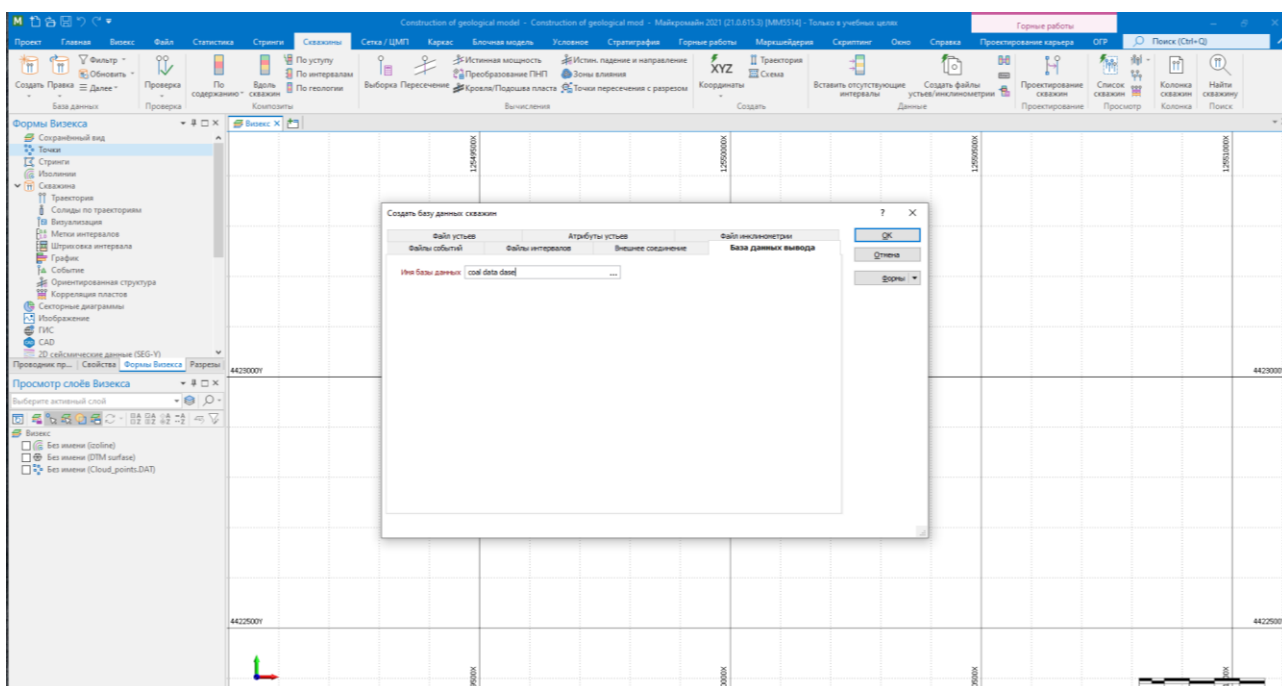
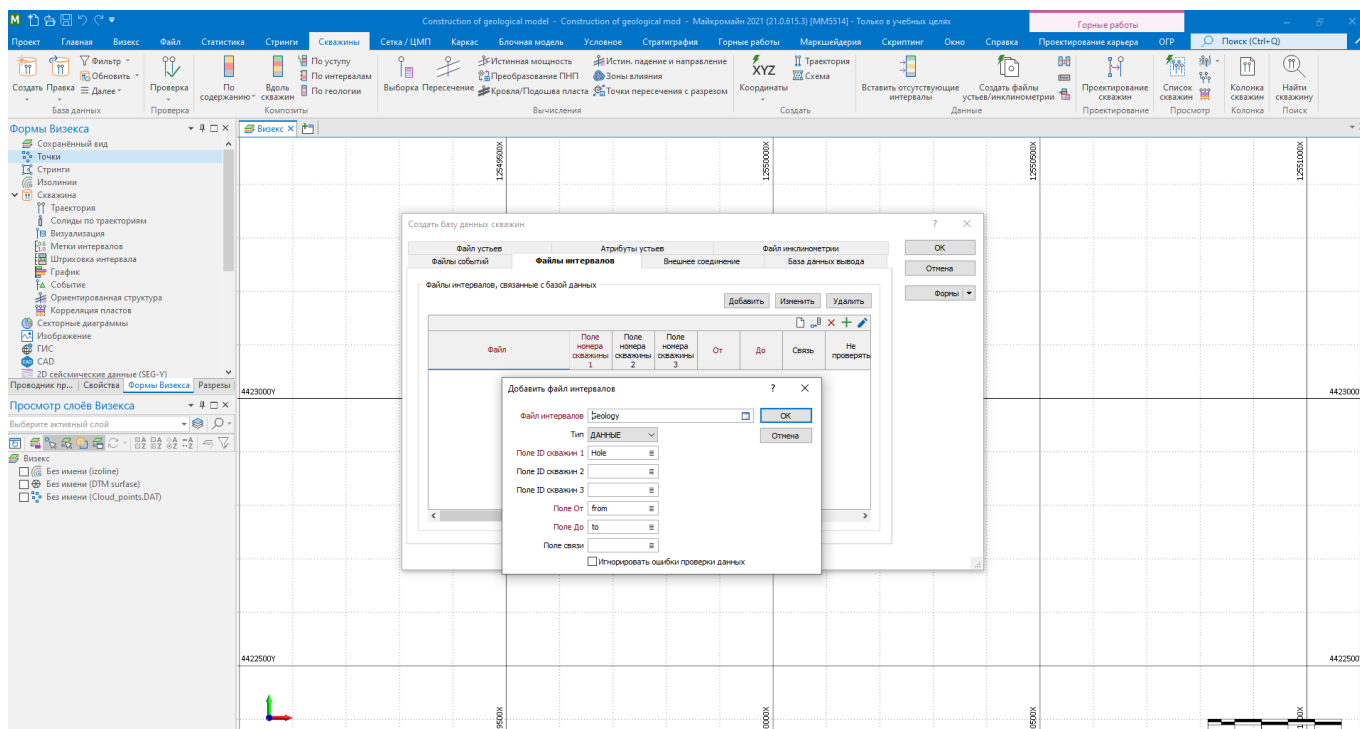


Event and Interval Files

Event files and *Interval* files can be associated with a drillhole or trench (DHDB) file. When you set the properties of a drillhole or trench database (or right-click on the name of a drillhole or trench database in a form) you can add or remove the files associated with that file.

You can also modify the field settings for an associated file.

If you click **Cancel** in the Database Properties form, any changes you may have made to the associated files are discarded.



Drillhole Trace

On the **Home** tab or the **Vizex** tab, in the **Layers** group, select **Drillhole | Trace** (or double-click Drillhole Trace in the Vizex Layer Types pane) to display a drillhole trace.

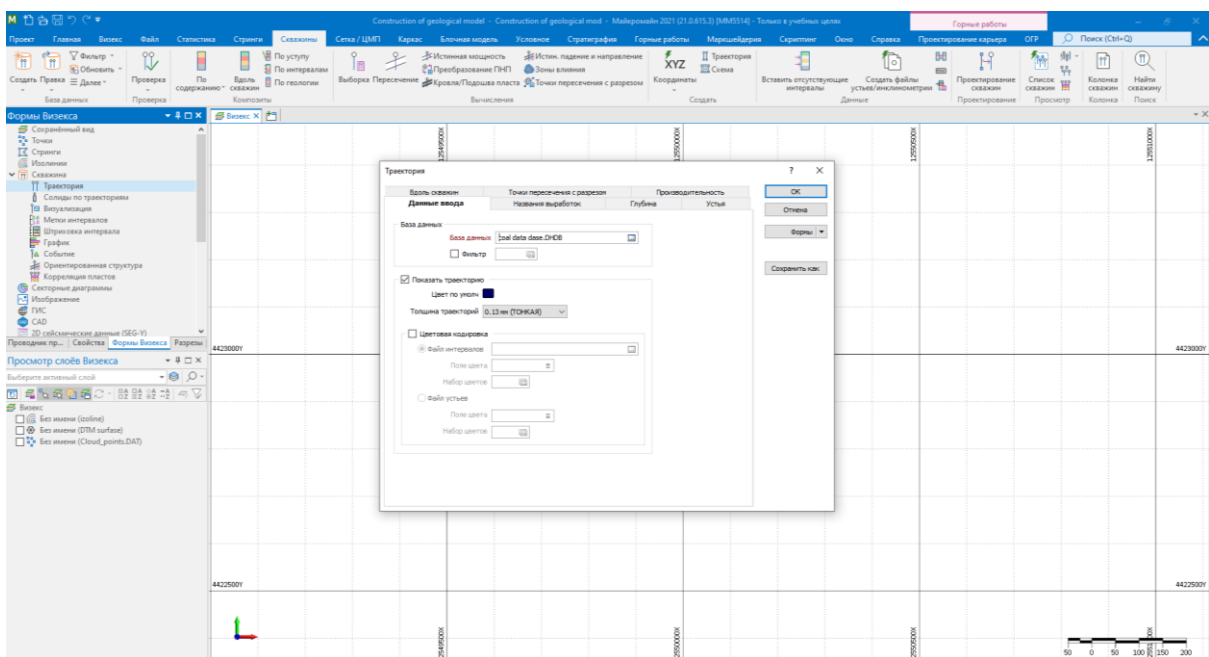


You can also specify the size and smoothness of the cylinders displayed along a [Solid Trace](#).

Database

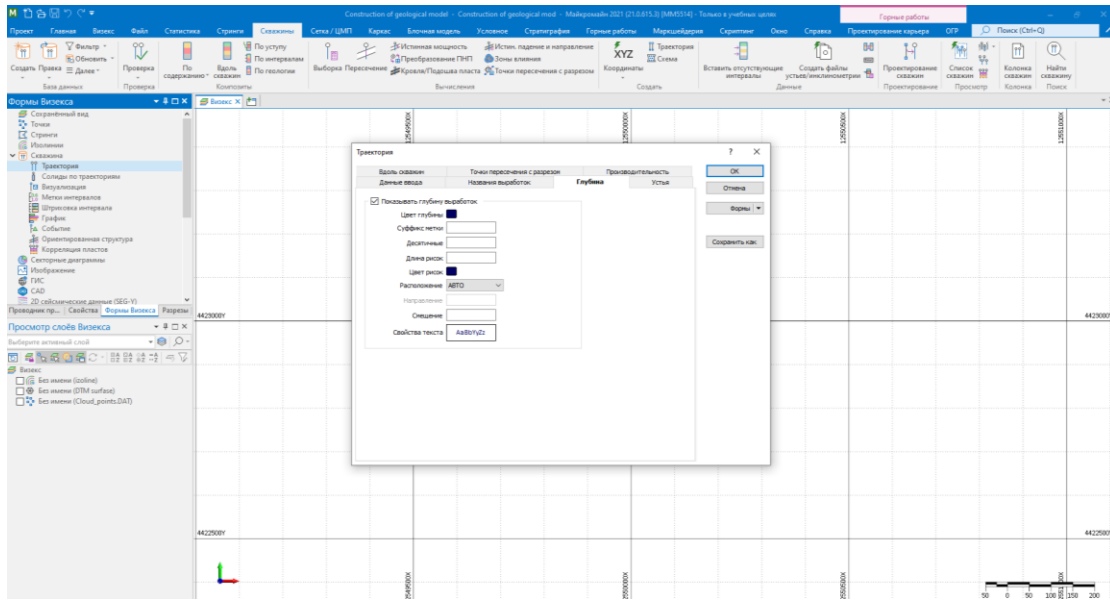
Trace coordinates are generated automatically when you create a Drillhole Database. The database contains the collar and (optionally) the survey data needed to generate the trace.

Double-click (F3) to select from a list of databases in the current project. Alternatively, you can right click and choose the **Select** option from the right-click menu. You can also **Edit** the properties of an existing database or create a **New** database.



Show Hole Depth

Select the Show Hole Depth check box option in the Hole Depth tab, if you want to display the hole depth at intervals along the drillhole trace.



Show Collar

Select the Show Collar check box option in the Collar tab, if you want to display the hole collar at the start of the drillhole trace.

Colour field

Specify the name of a field which contains the values that will be used to colour-code the display.

You can choose to select colour values directly from the Colour field without selecting a Colour set. In this case, the values in the Colour field must be valid RGB, Hex, or Integer colour definitions.

Format	Example
R,G,B	89,169,215
HTML Hex	#59A9D7
Hex	0x59a9d7
int	5876183

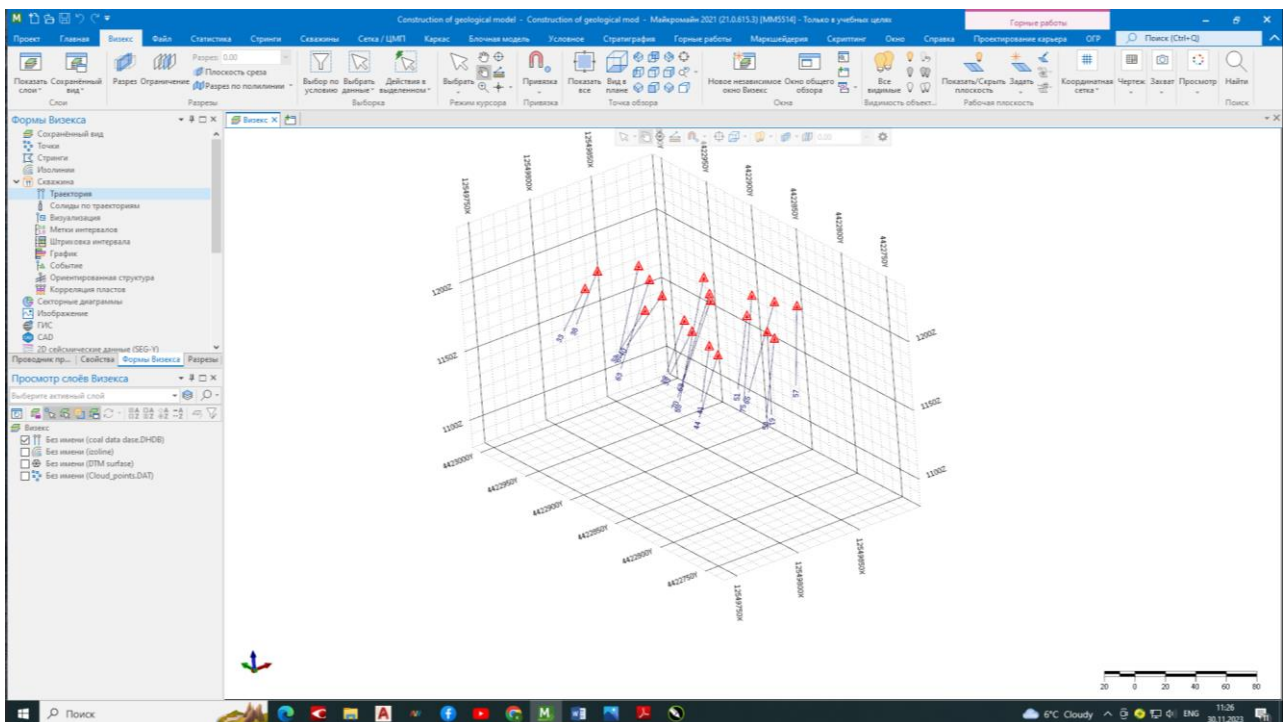
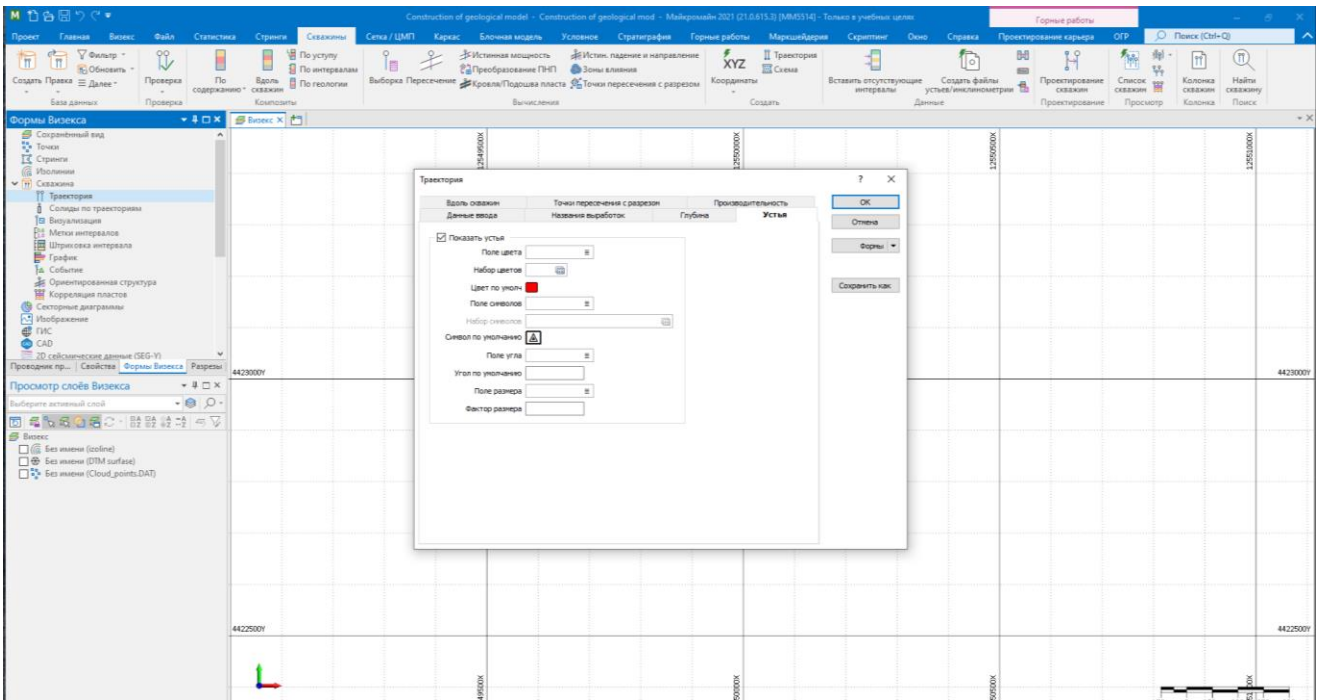
An **RGB(r,g,b)** function which lets you calculate RGB colour values on the fly, can be included in a Dynamic Field Expression. See: [Expression Editor](#)

Colour set

To map values in the Colour field to the colour values in a Colour set, double click (F3) to select the set that will be used to control the display colour. Right-click (F4) to create or edit a Colour set.

Default colour

Double-click (F3) to select the colour that will be used when a Colour field or a Colour set is not defined - or when a value in the Colour field is either not valid or is not mapped in the Colour set.



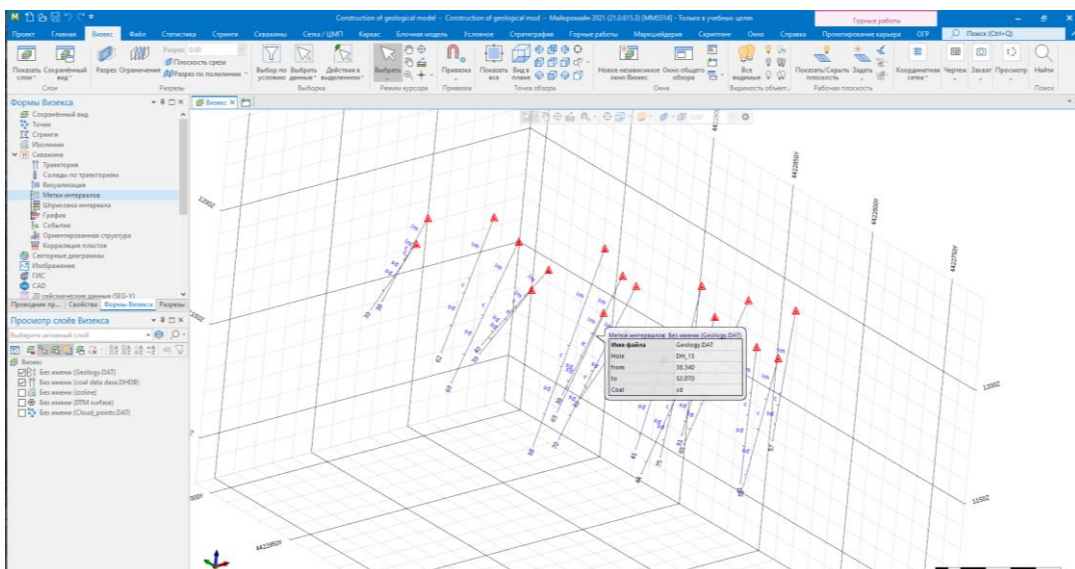
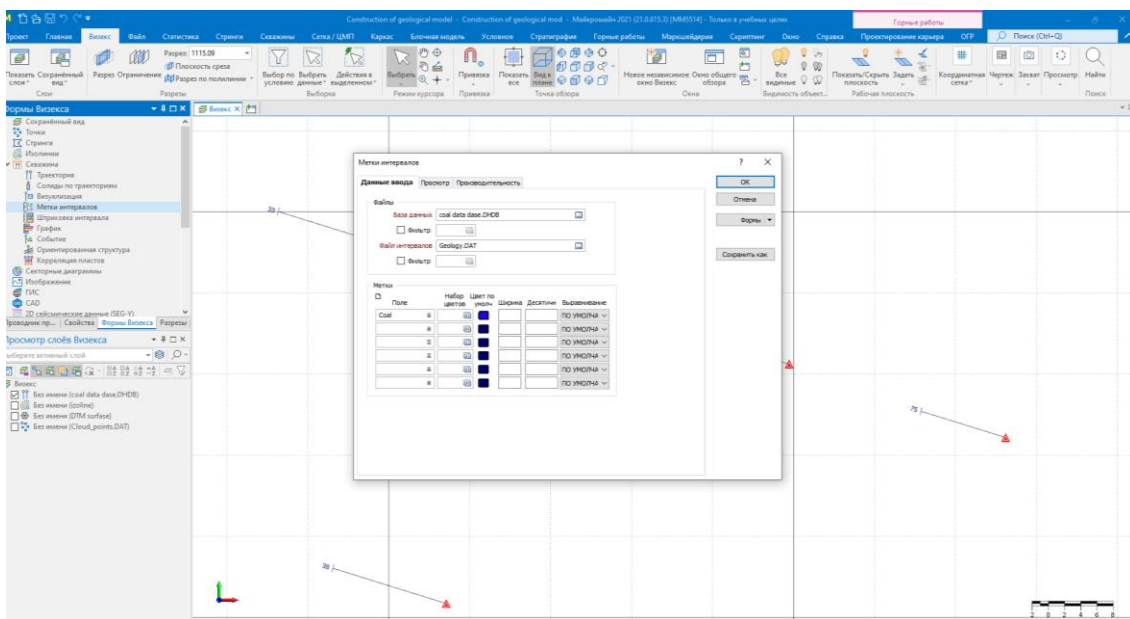
Метки интервалов

Во вкладке **Главная** или вкладке **Визекс**, в группе **Слои**, выберите **Скважины | Метки интервалов** (или дважды нажмите на Метки интервалов в панели Типы слоев Визекса), чтобы вывести в просмотр в Визексе метки интервалов.

0.6
1.0

Данные ввода
Файл

Настройте следующие параметры в окне Метка интервала:



Vein

On the **Implicit** tab, in the **Lithology** group, click **Vein** to model true-thickness lithology veins.



Input

Drillhole Database

Double-click (F3) to select from a list of databases in the current project. Alternatively, you can right click and choose the **Select** option from the right-click menu. You can also **Edit** the properties of an existing database or create a **New** database.

Select the **Filter** check box if you want to apply a filter to the database. Enter a filter number in the adjacent response. Double click (F3) to see a list of existing filters. Right click (F4) to open the dialog box where you can create a new filter.

Interval file

Double-click (F3) to select an Interval file that contains the grades you want to interpolate. You can optionally apply a Filter to the records in the file.

Lithology field

Double-click (F3) to select the name of a Character field that contains the lithology codes you want to include or exclude from the interpolation.

Numeric Exceptions

(Optional) Use the [Numeric Exceptions](#) group to control the way that non-numeric values are handled. Non-numeric values include characters, blanks, and values preceded by a less than sign (<).

Lithology to Model

When you select a valid Lithology field, the Exclude grid is populated with a full list of Lithology codes. Use the arrows between the *Include*, *Exclude* and *Ignore* grids to select the codes you want to include, exclude and ignore.

The points for each of the Lithology codes in the Include and Exclude columns will be assigned a positive or negative value respectively. This value will be the distance from the closest 0 (boundary points).

For Included codes, the values will be positive. For Excluded codes, the values will be negative. Ignored codes are excluded altogether from the interpolation.

Surface Control

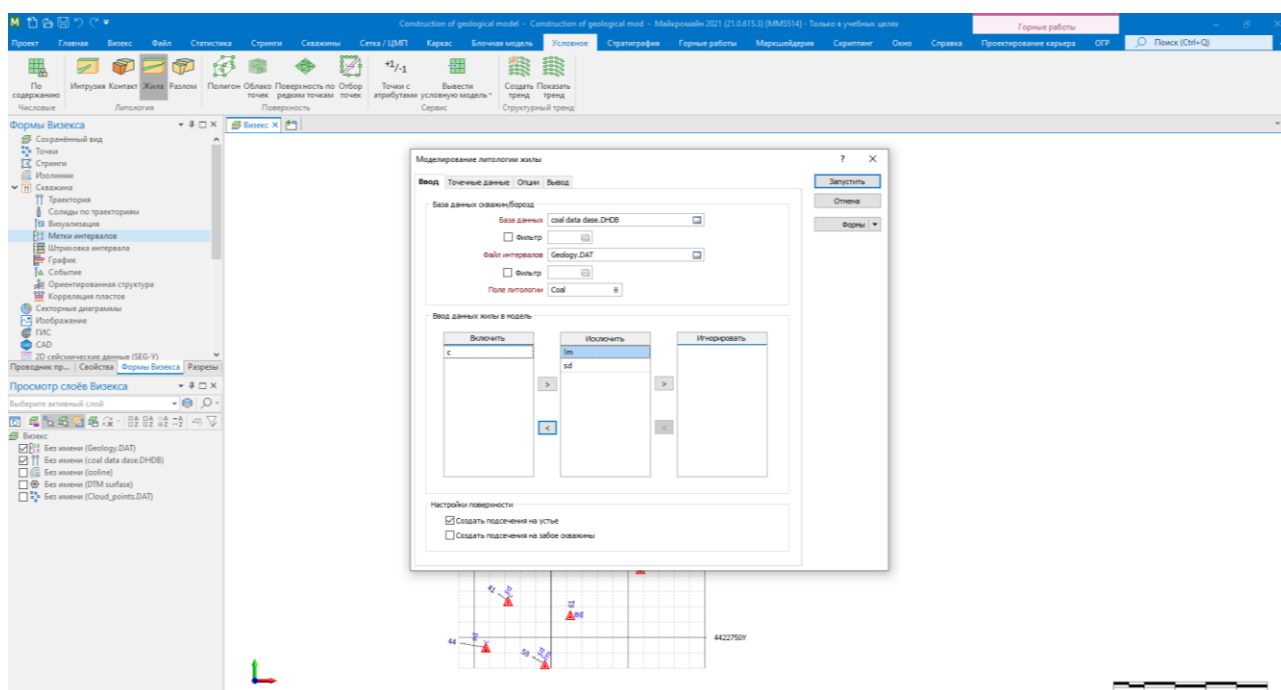
To constrain the model to your drillholes, you can choose to create intercepts at the collar and at the end of each hole.

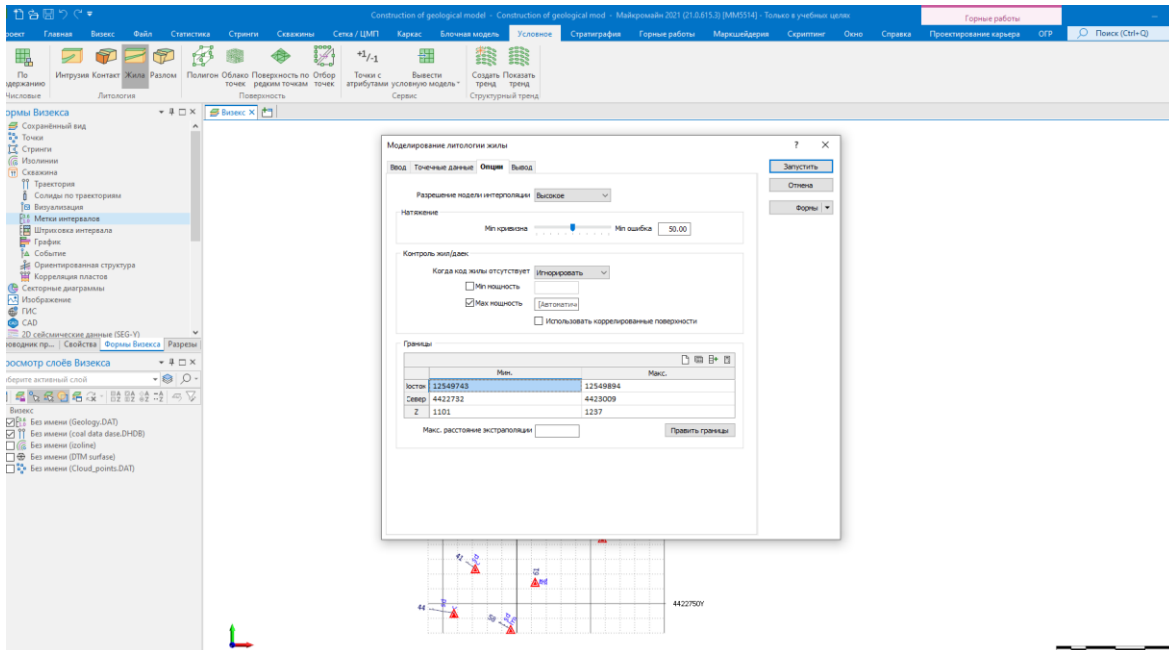
Create intercepts at collar

Enable this option to prevent the hanging wall from trending above the collar of the hole.

Create intercepts at end of hole

Enable this option to prevent the footwall wall from trending below the end of the hole.





Output

On the Output tab of the form, specify the type and name of an Output wireframe.

Wireframe

Type and Name

Double-click or click on the Select icon to set the Type and Name of the wireframe that will be generated from the input strings.

Tolerance

Specify a tolerance (in Grid units).

The surface extracted from the implicit model is done to the accuracy defined by the tolerance, such that the difference between the wireframe surface and the theoretically perfect mathematical model falls within the measured tolerance distance.

Quality

This setting provides a convenient way to control the quality and the speed of wireframe generation. There are five quality settings: Draft, Standard, High, Ultra and Custom.



Setting the quality to **Draft** allows the general shape of the output wireframes to be generated quickly.

Generate complement solid

If this option is selected, then a separate solid wireframe will be generated which encloses the space between the contact surface and the edge of the **ROI**, one that is external to the **INCLUDE** intervals specified on the Input tab.

The complement solid wireframe will have the same name as the primary wireframe, appended by a suffix “_complement”.

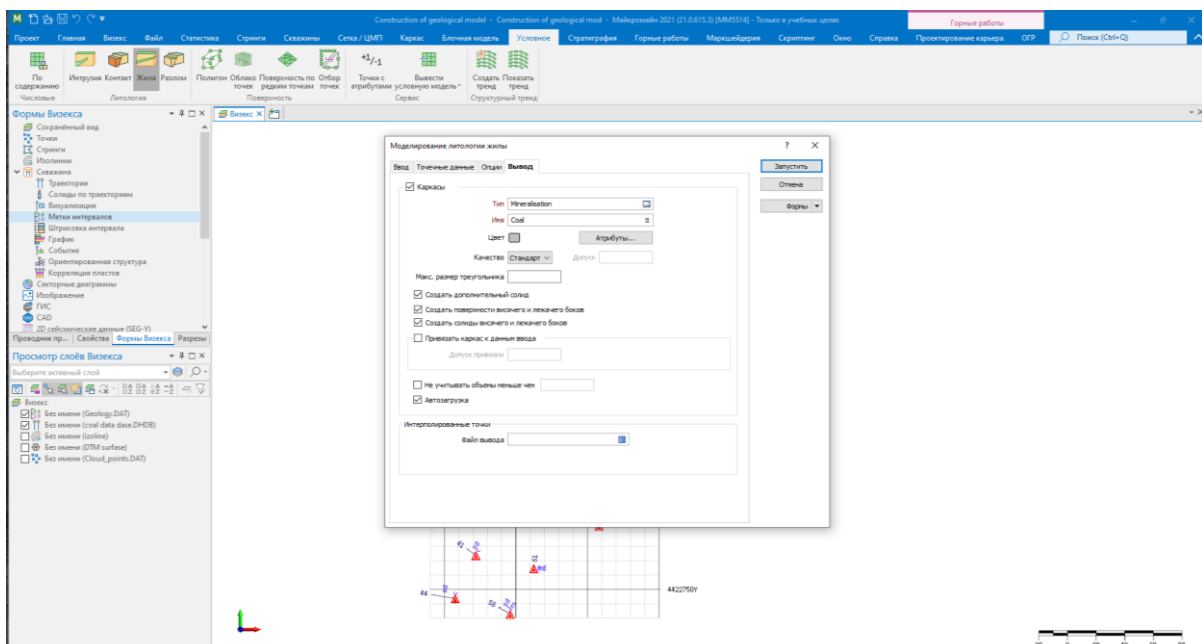
Generate surface wireframes

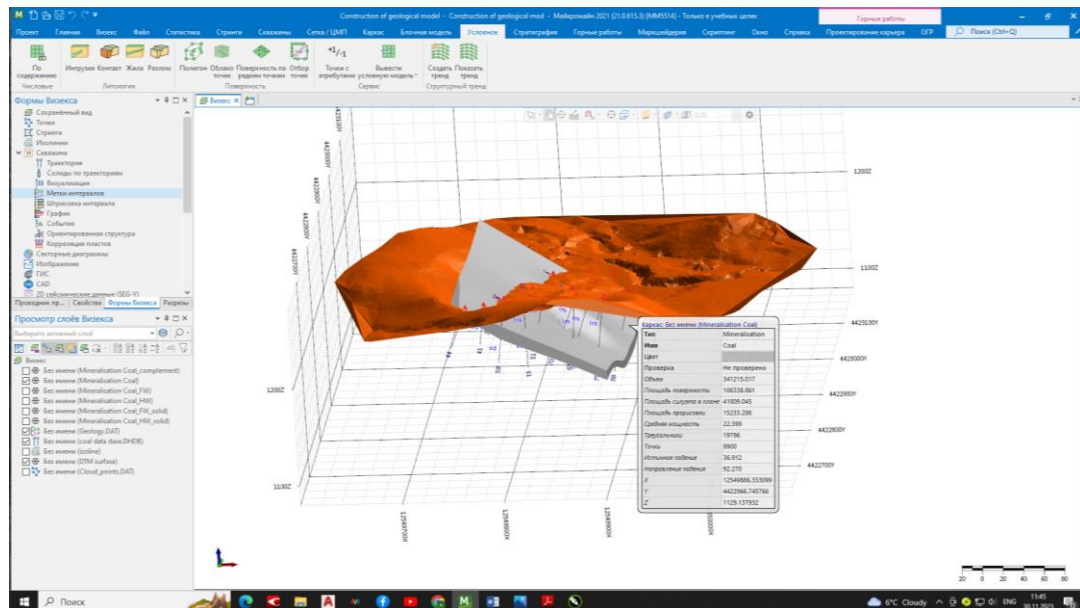
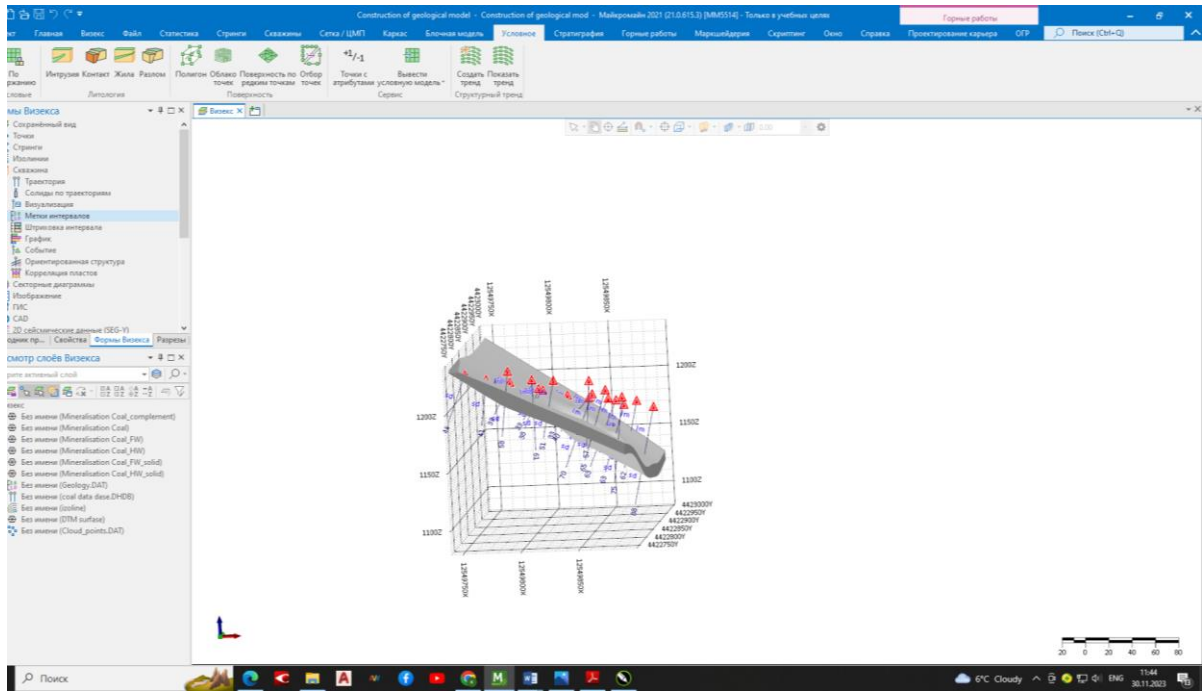
If this option is selected, two separate wireframe surfaces will be generated, one representing the top surface of the vein, and one representing the bottom surface of the vein.

These top and bottom vein surface wireframes will be given the same name as the primary solid vein, appended by the suffixes “_surface_1” and “_surface_2” respectively.

Auto load

Select this option to load the generated output in Vizex. The default draw style for an auto-loaded wireframe is 3D Shaded.





Validate

Use the tools on the **Wireframe** tab, in the **Validate** group, to:

- click **Validate Wireframe** to validate a wireframe in Vizex.

- check wireframes for self-intersection, closure, and invalid topology

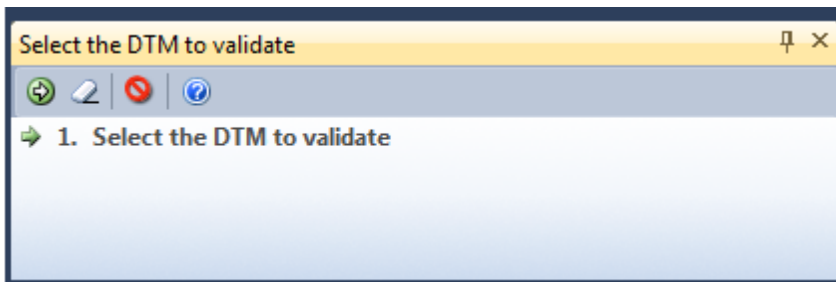


- click **Validate DTM** to run Wireframe Validation and check that each selected surface is a valid DTM.
- check DTMs for self-intersection, closure, invalid topology and surface integrity

On the **Wireframe** tab, in the **Validate** group, or the **Grid / DTM** tab, in the **DTM Tools** group, click **Validate DTM** to run Wireframe Validation and check that each selected surface is a valid DTM.



If no DTM is selected in Vizex, you will be prompted to select one.



Check self-intersection

When this validation option is selected, the Validate function will check for intersecting triangles. The highlight colour selected as part of your Vizex Options will be used to display self-intersecting triangles.

Check closure

When this validation option is selected, the Validate function will check that each surface is closed. If you do not need to check for closure, for example, when you have triangulated a fault, select the Check self-intersection option only. The highlight colour selected as part of your Vizex Options will be used to display a surface that is not closed.

If open sections or connection errors are detected during the validation process, wireframe validation strings are created. If those strings are coincident with the original construction strings, then they will be given priority when strings are selected during editing.

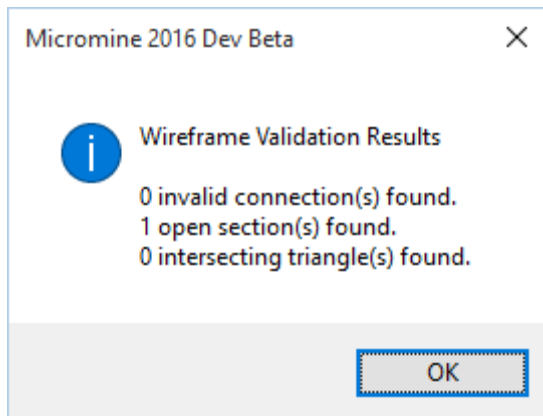
This makes it easier to correct the wireframe using both the validation strings and the construction strings.

Auto-Detect DTM Plane

When this validation option is selected, a check is made to ensure that the triangle faces of the DTM all point roughly in the same direction and have the same sign. Accept the default or select the DTM view plane that will be used for the validation.

OK

Review the results of validation and then click **OK** to close the [Results window](#). The colours you have set as part of your Wireframing options will be used to highlight any triangulation problems.



The following results are reported:

- [Invalid Connections](#). When these are found, the validation process will not be continued.
- Open sections. These may not necessarily be errors.
- Intersecting triangles. These must be corrected. There can be no intersecting triangles in a valid wireframe.
- Any triangles that fail the DTM test.

Unnecessary points and triangles are removed automatically as part of a [Clean \(form\)](#) operation.

You can toggle Validation options on and off via the **Wireframing** tab on the the **Options | Vizex (per project)** form.

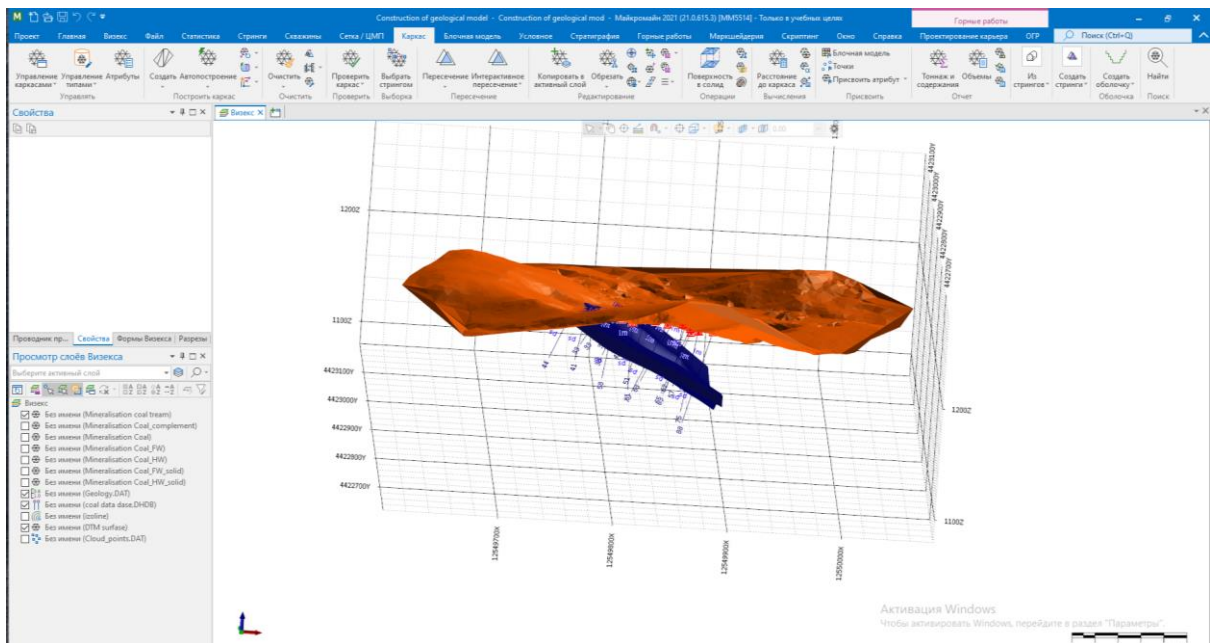
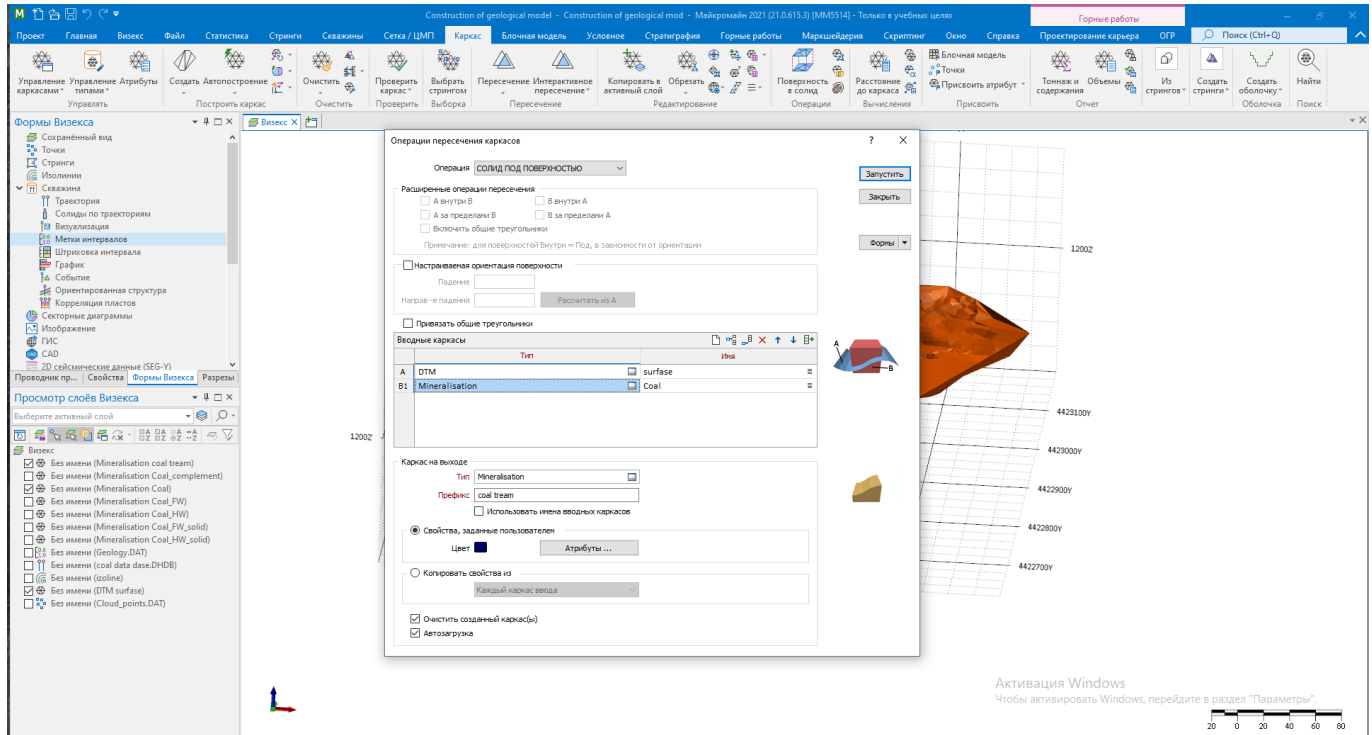
If you select Validate DTM as the validation option in the Wireframe Manager (see below) you can also check that a surface is a valid DTM.

On the **Wireframe** tab, in the **Boolean** group, click **Wireframe Boolean** to pass two wireframes (A and B) as arguments to a chosen Boolean operation.



Operation

Union, Intersection and Difference Operations can be applied to solids and **Difference and Merge Operations** can be applied to surfaces. When you select an operation from the drop-down list, an illustration of the inputs and output from that operation are illustrated on the *Boolean Operations* form.



The “Difference” set operations defined for solids and surfaces are mathematically different. In the case of surfaces, the difference operations perform a surface trim. The Merge operation is Difference (A-B) and Difference (B-A) merged together.

[Surface Merge Min/Max](#) operations are intended for XY surface triangulations only.

Advanced Boolean Operations

If you select ADVANCED, a number of [Advanced Boolean Operations](#) are enabled. These options apply to both solids and surfaces and can be used as an alternative to the operations provided in the main list. To get the exact output you want, you can select a combination of operations.

Include common triangles

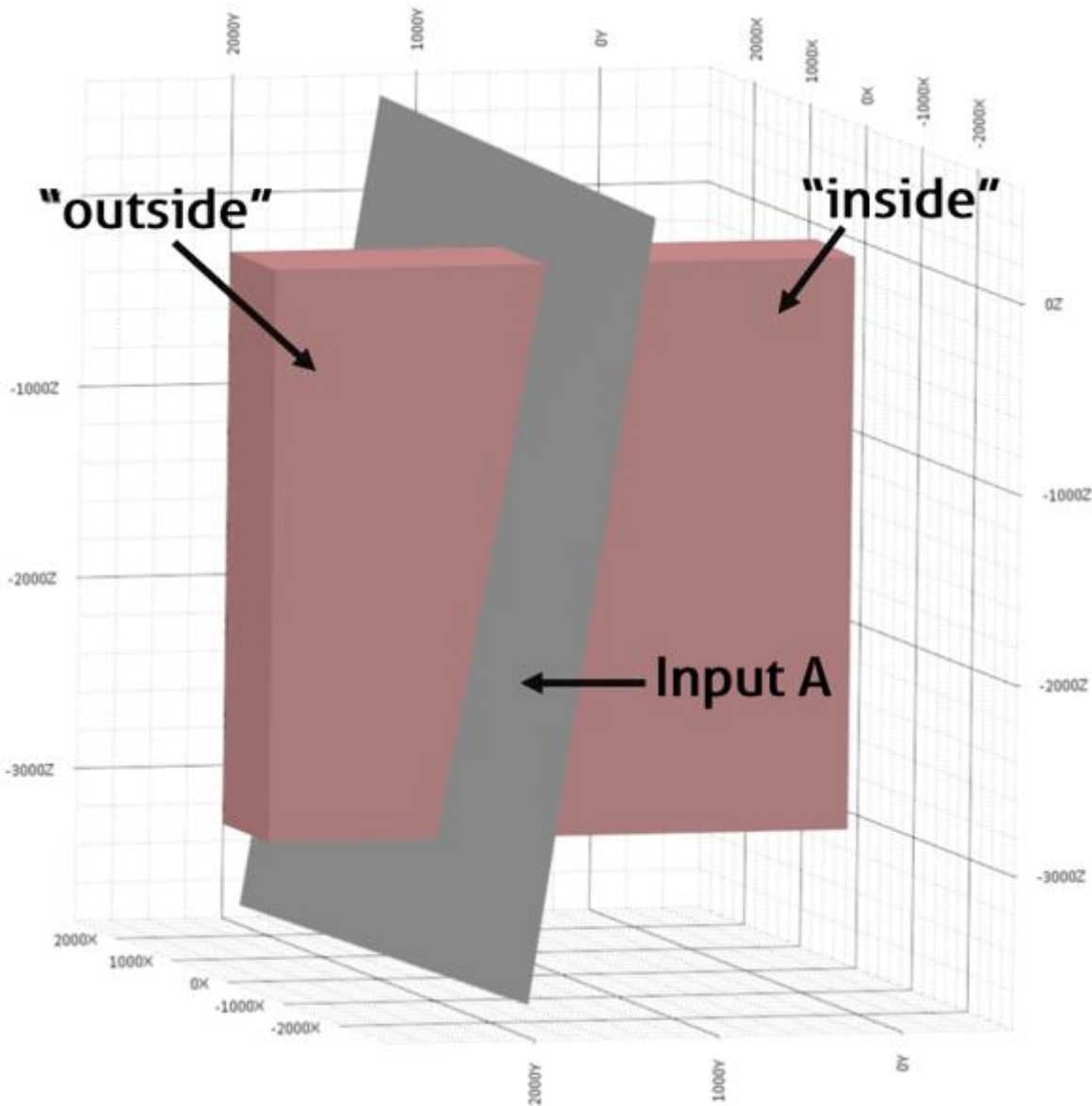
Select this check box to include triangles which are common to both wireframes A and B.

Custom Surface Orientation

Select this checkbox to define how Micromine treats the surface where “inside” and “outside” are defined during the following operations:

- SOLID ABOVE SURFACE
- SOLID BELOW SURFACE
- ADVANCED SURFACE AGAINST SOLID

The default methodology uses the z-value (horizontal). For example:



When the checkbox is selected, Micromine will automatically determine the surface orientation from input A, a manual override of **Dip** and **Dip Direction** values is available. Alternatively, click the **Calculate from A** button to set the orientation of the surface to match the orientation of input (A).

Custom Surface Orientation


Dip [Auto]

Dip direction [Auto] Calculate from A

Input Wireframes

Use the grid to specify the wireframes you want to perform a Union operation on. Use the buttons on the grid list toolbar to [Manage the rows in the list](#).

Autofill Wireframes

 Click the **Autofill** button to automatically generate benches based on the bench parameters you have specified in the [Autofill Wireframes](#) form.

Type

Double-click (or click on the Select icon) to select a wireframe type. You can also right-click in the grid and select **Edit Wireframe Types**.

Name

Double-click (or click on the List icon) to select a wireframe of the corresponding type.

You can also right-click in the grid and select **Edit Wireframe Attributes**.

To edit a wireframe type, right-click on a row and select **Edit Wireframe Type** from the right-click menu.

You cannot specify more than two input wireframes when the chosen operation is "Advanced".

See: [Wireframe Boolean Operations](#)

Snap common triangles

Select the *Snap common triangles* check box to remove any discrepancies that result from numerical precision problems on shared triangles.

Output

Type

Select the **Type** of the output wireframe(s).

Name

If there is only one output, enter (or double-click to select) the **Name** of the wireframe that will be created. If there are multiple output wireframes, the Name field is disabled and a Name prefix field is enabled instead.

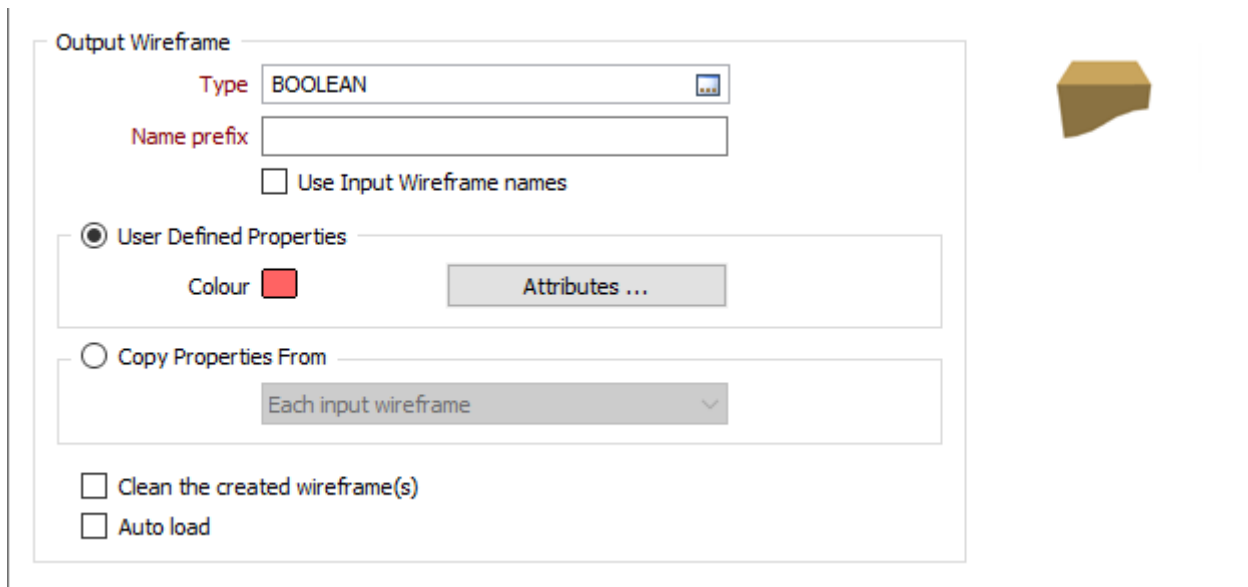
Name prefix

When there are multiple outputs and the output type is the same as the input type, or the *Use Input Wireframe names* check box is not selected, a **Name Prefix** may be used to uniquely name each wireframe.

If there are multiple output wireframes, an incrementing suffix is appended to the name prefix to make the names unique.

Use Input Wireframe names

If there are multiple outputs, all output types are the same, and the output type is different to the input type, then the original names of the input wireframes may be used to name the output wireframes.

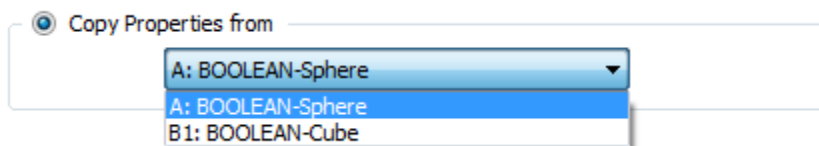


When writing **properties** to the output wireframe(s) you can choose whether to:

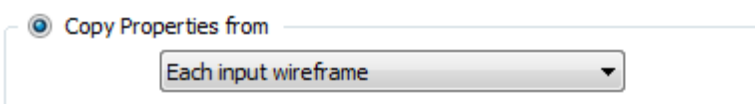
- copy properties and attributes
- set user-defined properties and attributes

Copy properties and attributes

If there is only one output, select an input wireframe to copy properties and attributes from:



If there are multiple inputs, properties and attributes are copied from each input wireframe:



User defined Properties

User defined properties are defined once and are applied to all outputs.

Standard [Attributes](#) such as Code, Colour and Title must always be defined, no matter what type of wireframe you are dealing with. You can also enter up to twenty different user-defined attributes (for example, Average Grades and Density) for each type of wireframe.

Clean the created wireframes

Select this option to clean the output wireframes.

Auto load

Select this option to load the generated output in Vizex. The default draw style for an auto-loaded wireframe is 3D Shaded.