FAST AS FLUID

USER GUIDE

VERSION M01

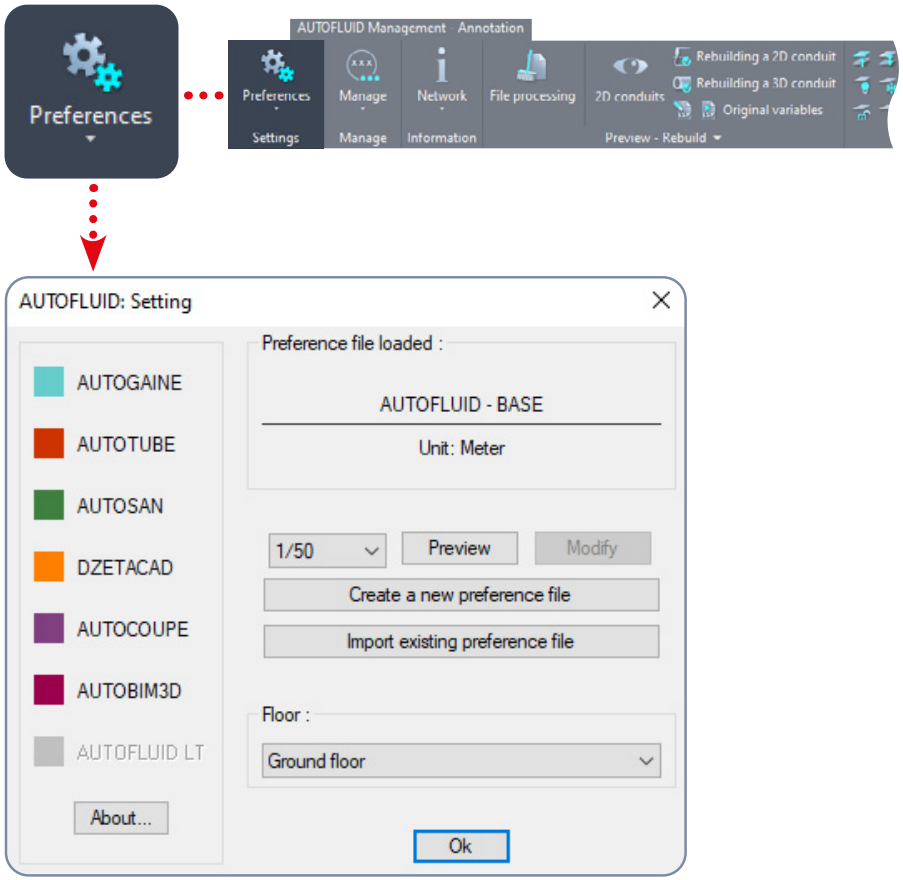
Content: Settings and 2D drawing

Content : Pressure losses, Cross Sections, 3D BIM

Basic settings

Set up AUTOFLUID for the floor plan you are going to work on.

Two parameters must be set first:



1 - The working unit of the architect's drawing

To find this value you can measure a simple door with the command «DIST» in your CAD software.

If the value reads approximately:

0.80	then the unit is METER
80.0	then the unit is CENTIMETER
800.0	then the unit is MILLIMETER

Do not use or take into account the value given by the «UNIT» command of your CAD software as it isn't related to the unit of the architest's drawing.

2 - The scale of the drawing

This is the scale that you will specify in the title block of your plan. Here is a few examples of the parameters AUTOFLUID can adjust, thanks to these 2 values:

- The size of texts
- The appearance of dimensioning
- The appearance of frames and leader lines
- The calculation of levels. And more...

Basic settings

Other parameters can be set.

For instance:

- The list of layers
- Colors, types of lines and thicknesses
- The graphic style of the network
- Texts
- Units
- Etc...

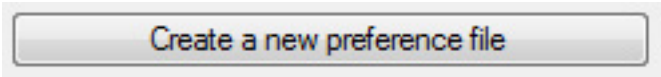
To change these parameters you must create a new preference file.

The preference file contains all the settings which run the AUTOFLUID package.

The file extension is «PREF» settings.

You will use these as a basis to create your OWN preference.

AUTOFLUID runs with default BASE file.

Click on 

and name the new file. Change it according to your needs by navigating the preference settings.

Once the changes are made the file can be imported by other users.

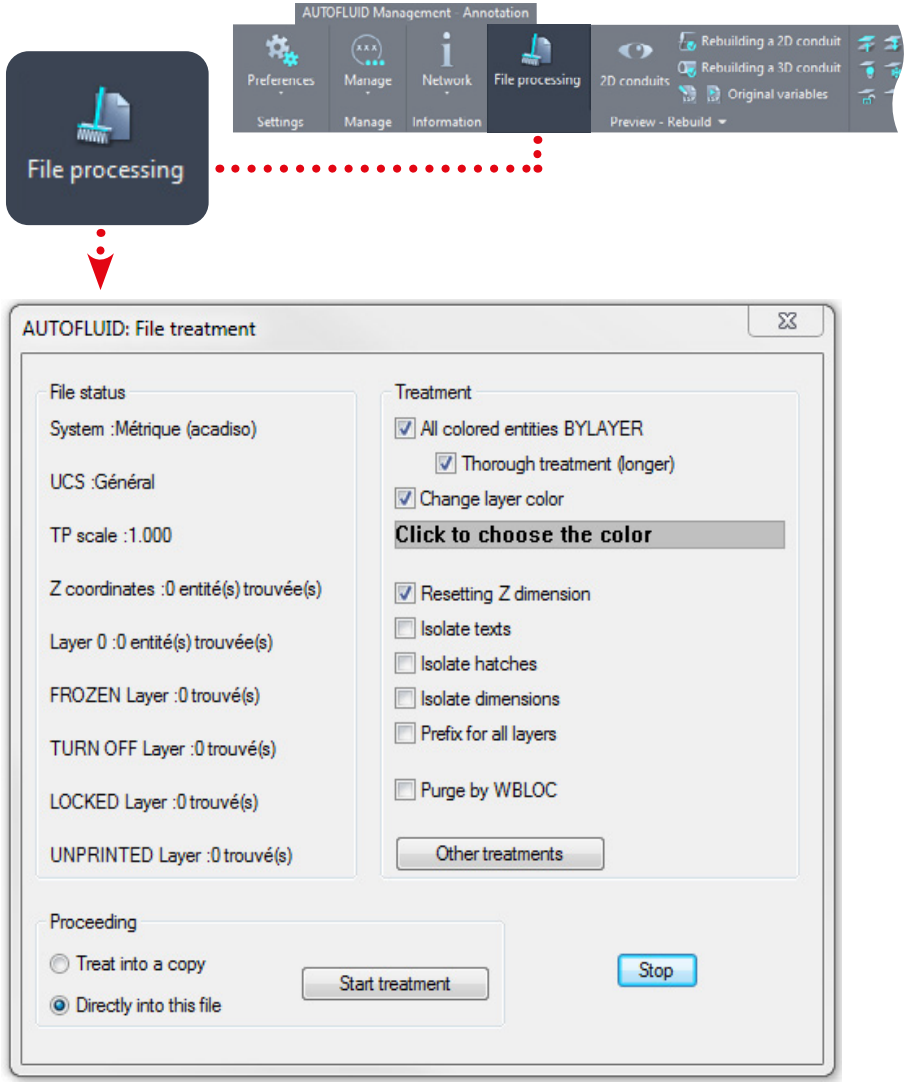
It is possible to come back and modify your preferences further at a later stage.

Changes made on a .PREF file are not automatically updated in the DWG.

Change the list of layers

Create a new preference file

Treatment of architectural files



This module allows you to amend the structure of a file.

It is possible to change colours, to handle the Z coordinate of entities, to isolate texts, hatches, dimensioning, etc...

File Treatment

File to work on: C:\archi\floor3.dwg

1. Open the file to treat. «C:\archi\floor3.dwg»
2. Launch the command
3. Choose options
4. Launch treatment.

At the end of the operation, verify that the file «C:\archi\Files treated by AUTOFLUID\floor3.dwg» is correct and save it.

NB: The « Purge by WBLOCK » option will save the file under the same name.

Calculation of air network duct sizes

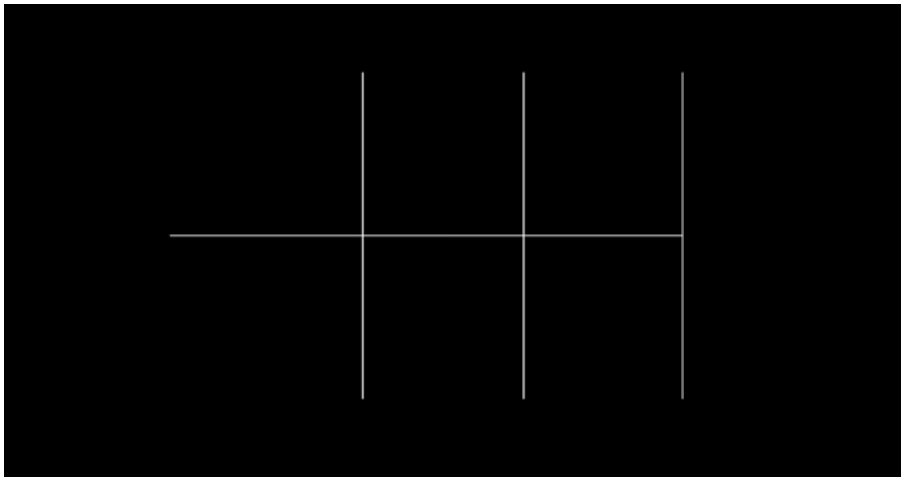
Using the «Air network calculator»

This calculation method works out the section sizes of the different segments of your network and writes the information on a diagram. It is a convinient way to find your duct sizes when drawing your networks in 2D using AUTOFLUID's routing commands. Note that there's no dynamic link between the network calculator and the drawing of the network in 2D.

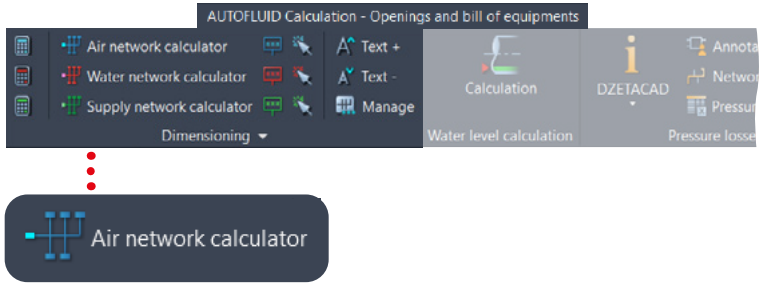
Here is a guide in 3 steps:

1- Draw the network diagram

Use the line command in your CAD software.
(Avoid overlapping lines).

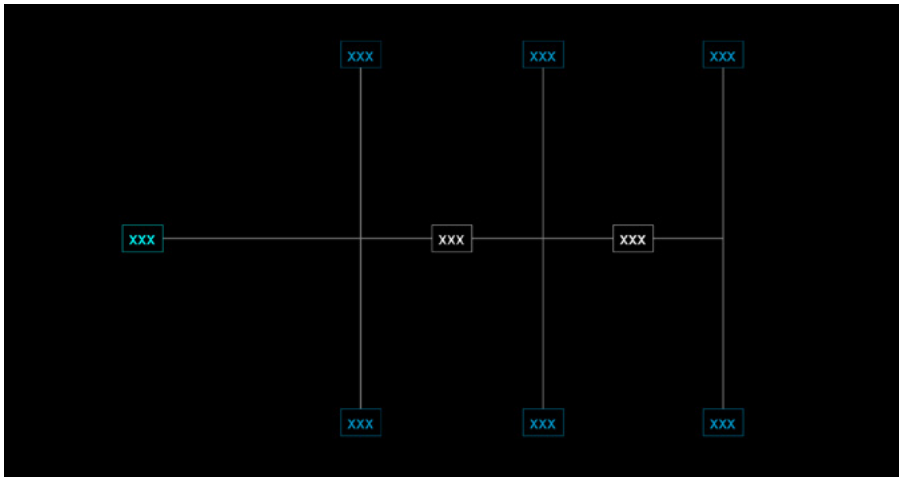


Dynamic air network calculation



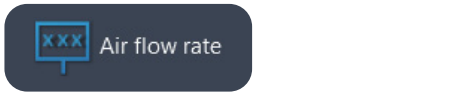
2 - Capture your diagram

Specify the calculation values and select the lines with the "Air network calculator" command. The diagram transforms into a dynamic tree structure. Texts (in dark blue) on the end branches await flow rate information.

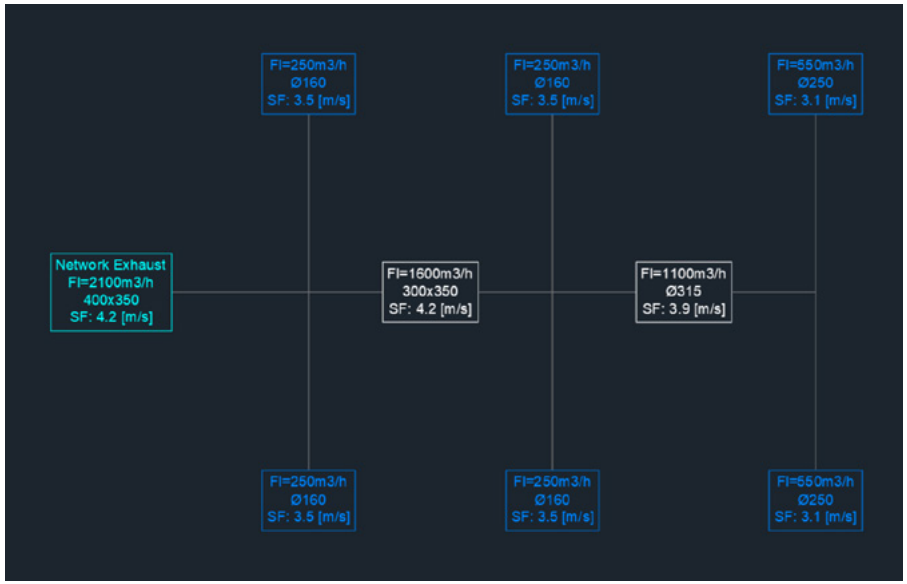


Calculation of air network duct sizes

3 - Insert the flow rate for each end branch



Use the command «Air flow rate» on each branch.



All the relevent branches are updated to display the type of information you selected.

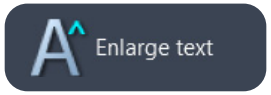
Good to know

Text displays in 3 different colours (that can be set in preferences)

- Dark blue: end branches
- White: Middle branches
- Light blue: Main branches

Size of text

The size of text can be adjusted with the following commands:



Calculation of air network duct sizes

Modifying an existing air network calculation

Change the velocity or the geometric constraint on one or several branches.

Change an air network variable

Using the command «Change an air network variable», enter the new values and select the relevent branche(s): all the texts will uptdate.

Adding or deleting a branch

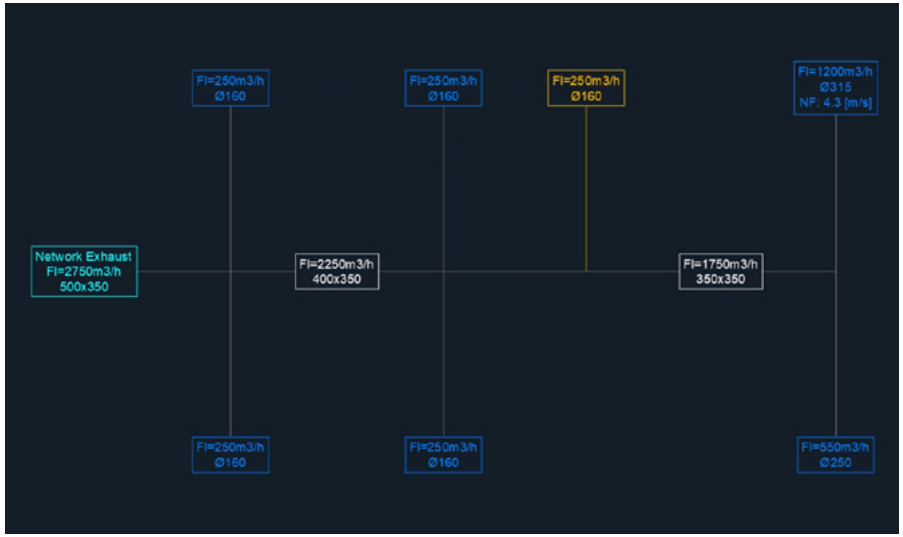
Use CAD commands to copy, move, stretch or delete... Or revise your network and then capture the whole diagram again.

Editing an air network calculation

Good to know

When you add a new branch, using your CAD mirror command for exemple, it will appear in orange by default.

Remember: every modification of the network diagram structure must be followed by a new capture to update the dynamic calculations.



Calculation of water network duct sizes

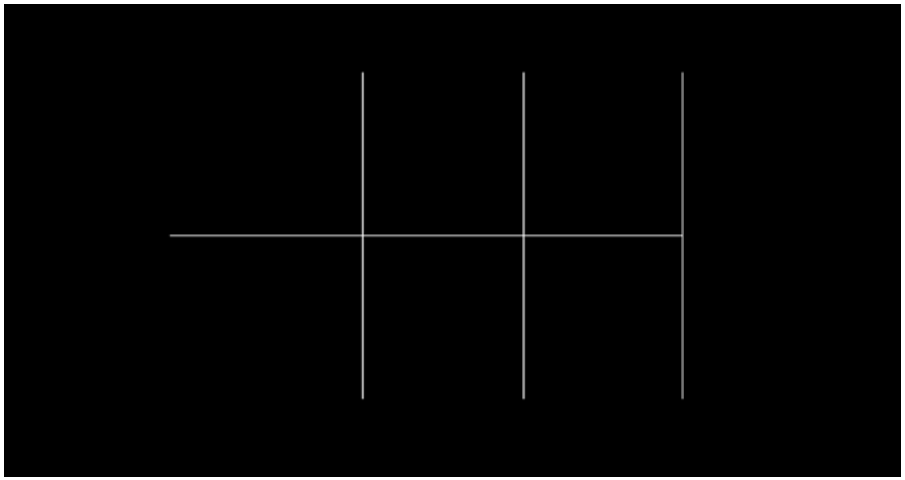
Using the «Water network calculator»

This calculation method works out the section sizes of the different segments of your network and writes the information on a diagram. It is a convinient way to find your duct sizes when drawing your networks in 2D using AUTOFLUID's routing commands. Note that there's no dynamic link between the network calculator and the drawing of the network in 2D.

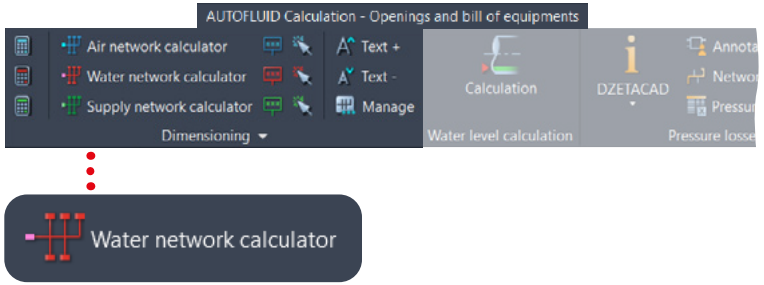
Here is a guide in 3 steps:

1- Draw the network diagram

Use the line command in your CAD software.
(Avoid overlapping lines).

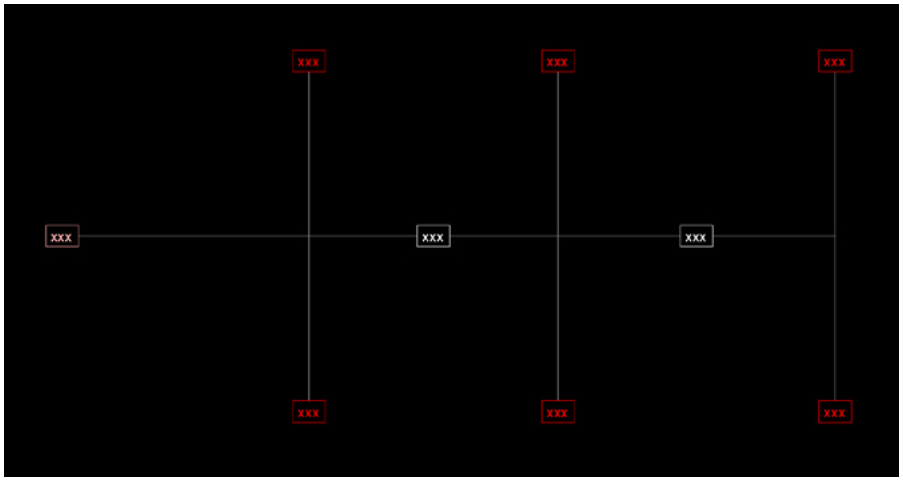


Dynamic water network calculation



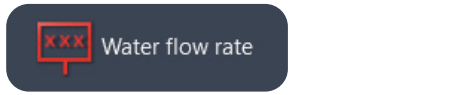
2 - Capture your diagram

Specify the calculation values and select the lines with the "water network calculator" command. The diagram transforms into a dynamic tree structure. Texts (in dark red) on the end branches await flow rate information.

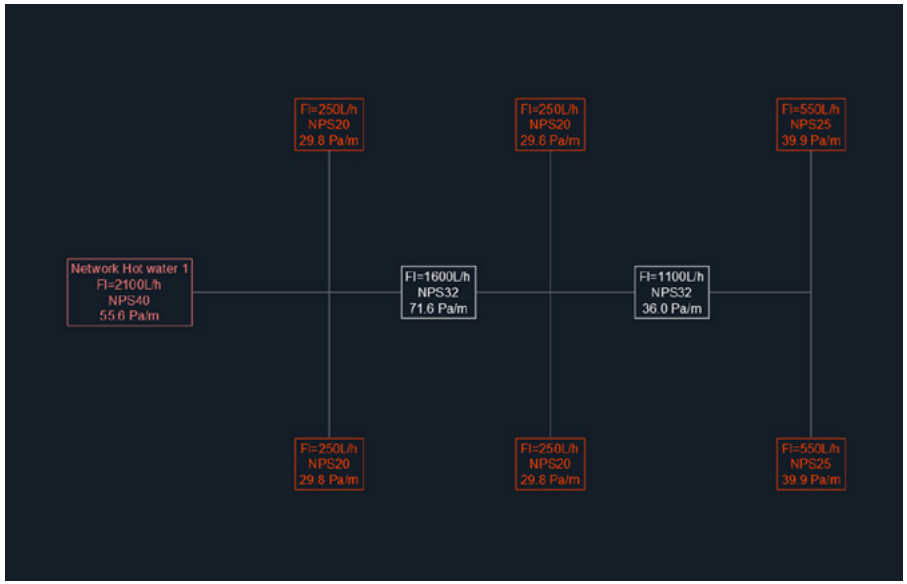


Calculation of water network duct sizes

3 - Insert the flow rate for each end branch



Use the command «Water flow rate» on each branch.



All the relevent branches are updated to display the type of information you selected.

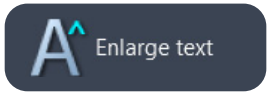
Good to know

Text displays in 3 different colours (that can be set in preferences)

- Dark red: end branches
- White: Middle branches
- Light red: Main branches

Size of text

The size of text can be adjusted with the following commands:



Calculation of water network duct sizes

Modifying an existing water network calculation

Change the velocity or the geometric constraint on one or several branches.

Change a water network variable

Using the command «Change a water network variable», enter the new values and select the relevent branche(s): all the texts will uptdate.

Adding or deleting a branch

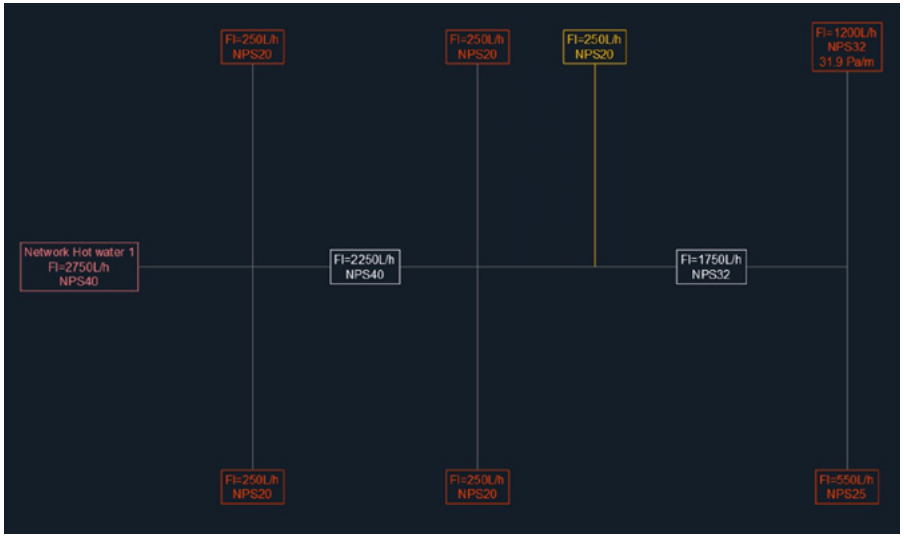
Use CAD commands to copy, move, stretch or delete... Or revise your network and then capture the whole diagram again.

Editing a water network calculation

Good to know

When you add a new branch, using your CAD mirror command for exemple, it will appear in orange by default.

Remember: every modification of the network diagram structure must be followed by a new capture to update the dynamic calculations.



Calculation of supply network duct sizes

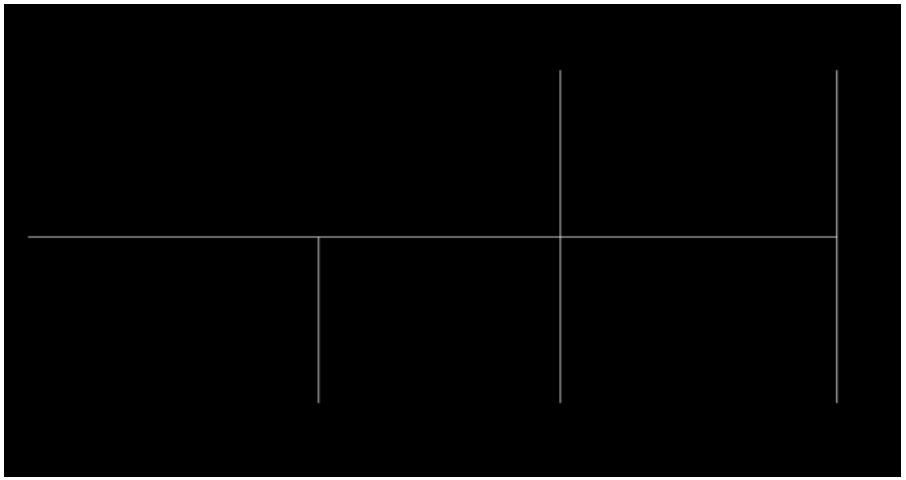
Using the «Supply network calculator»

This calculation method works out the section sizes of the different segments of your network and writes the information on a diagram. It is a convinient way to find your duct sizes when drawing your networks in 2D using AUTOFLUID's routing commands. Note that there's no dynamic link between the network calculator and the drawing of the network in 2D.

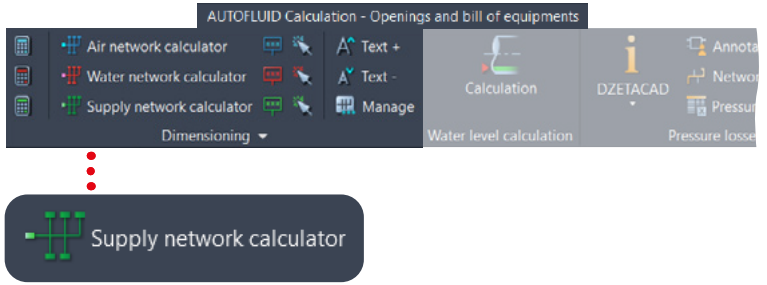
Here is a guide in 3 steps:

1- Draw the network diagram

Use the line command in your CAD software.
(Avoid overlapping lines)

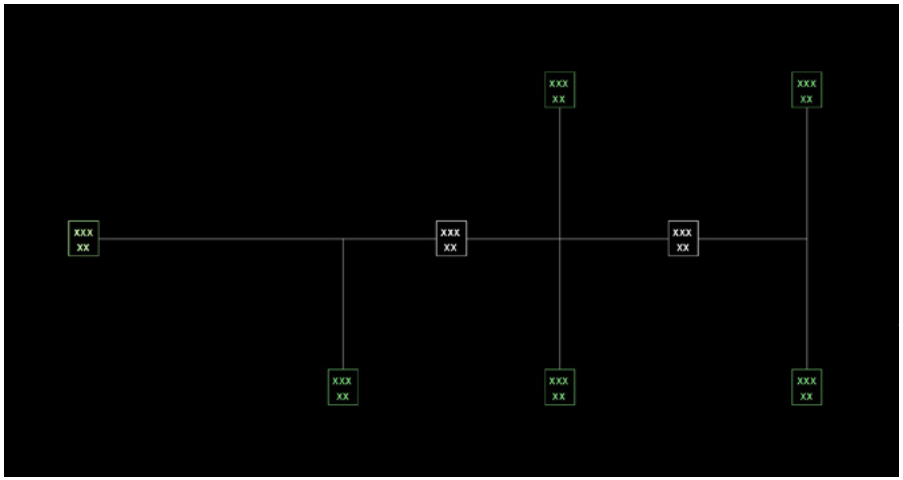


Dynamic supply network calculation



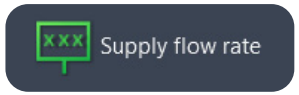
2 - Capture your diagram

Specify the calculation values and select the lines with the "Supply network calculator" command. The diagram transforms into a dynamic tree structure. Texts (in dark green) on the end branches await flow rate information.



Calculation of supply network duct sizes

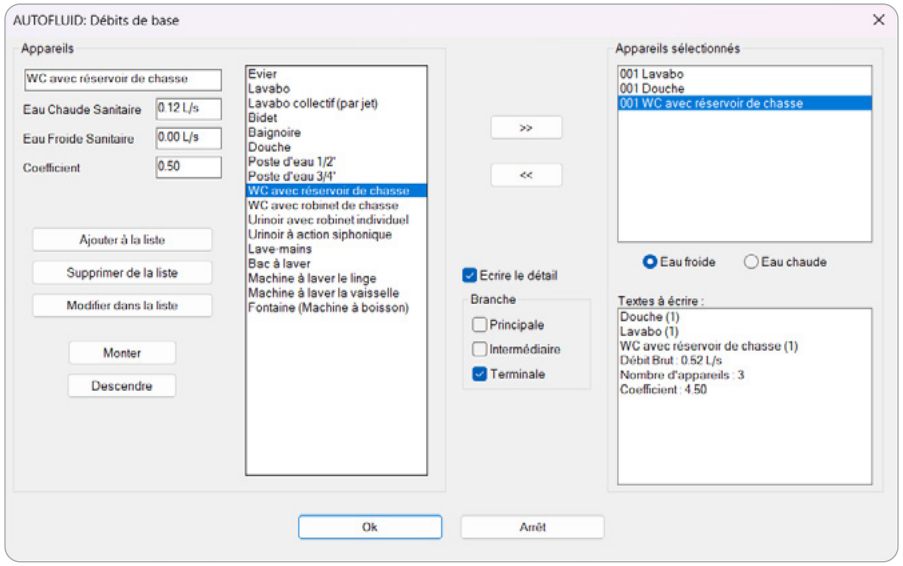
3 - Insert the flow rate for each end branch



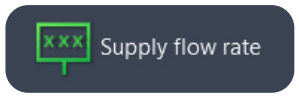
The «Supply flow rate» command lets you specify equipment to get their flow rate. Text is pre-formatted with all the caracteristics of the branch:

- Name of the pieces of equipment
- Cumulated flow rates
- The addition of the pieces of equipment
- Select the texts to write.

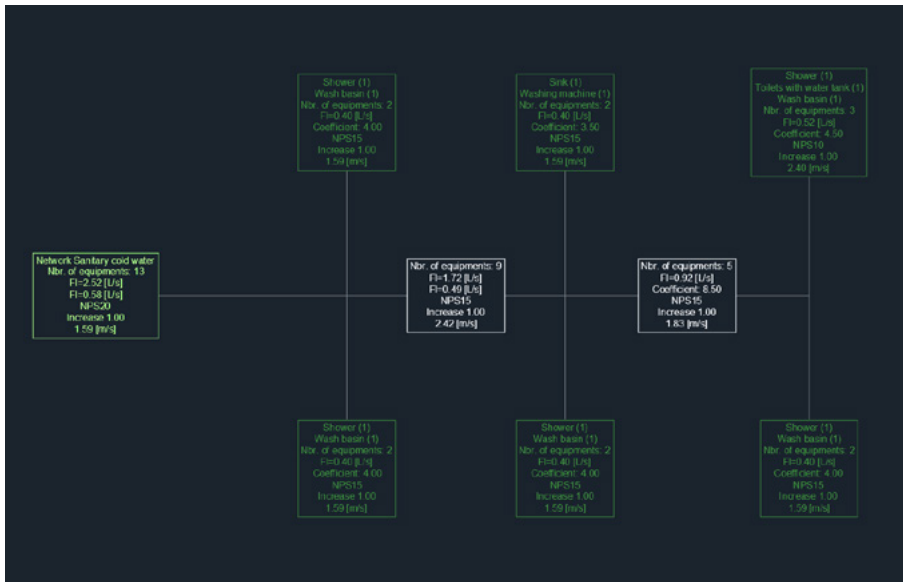
This command lets you manage the list of the equiments you most frequently draw.



Calculation of supply network duct sizes



Use the «Supply flow rate» command on each branch.



All the relevent branches are updated to display the type of information you selected.

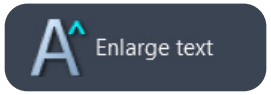
Good to know

Text displays in 3 different colours (that can be set in preferences)

- Dark green: end branches
- White: Middle branches
- Light green: Main branches

Size of text

The size of text can be adjusted with the following commands:



Calculation of supply network duct sizes

Modifying an existing water network calculation

Change the type of fluid, the velocity, the increase coefficient velocity or the geometric constraint on one or several branches.

Change a supply network variable

Using the command «Change a supply network variable», enter the new values and select the relevent branche(s): all the texts will uptdate.

Adding or deleting a branch

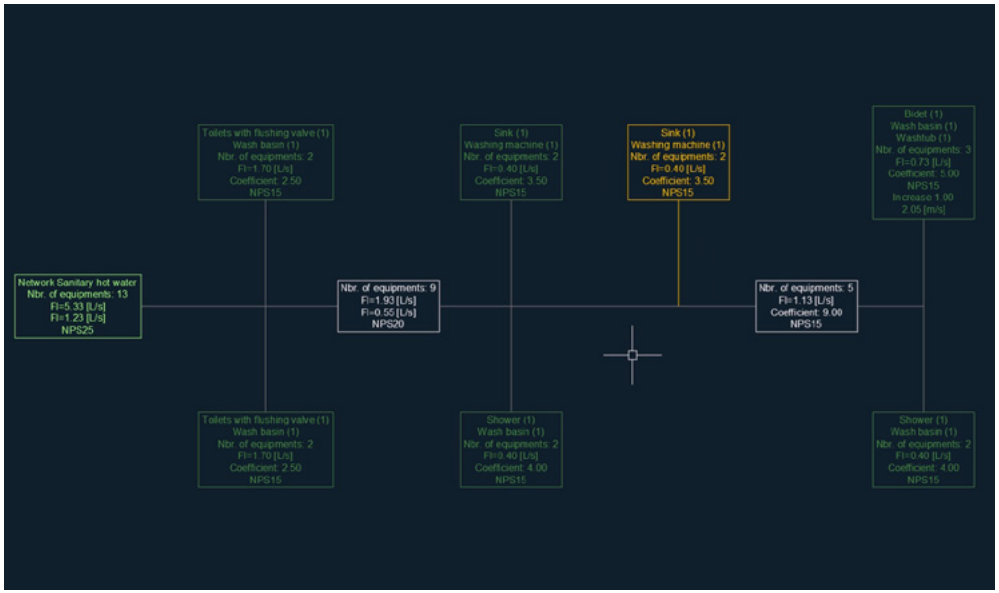
Use CAD commands to copy, move, stretch or delete... Or revise your network and then capture the whole diagram again.

Editing a supply network calculation

Good to know

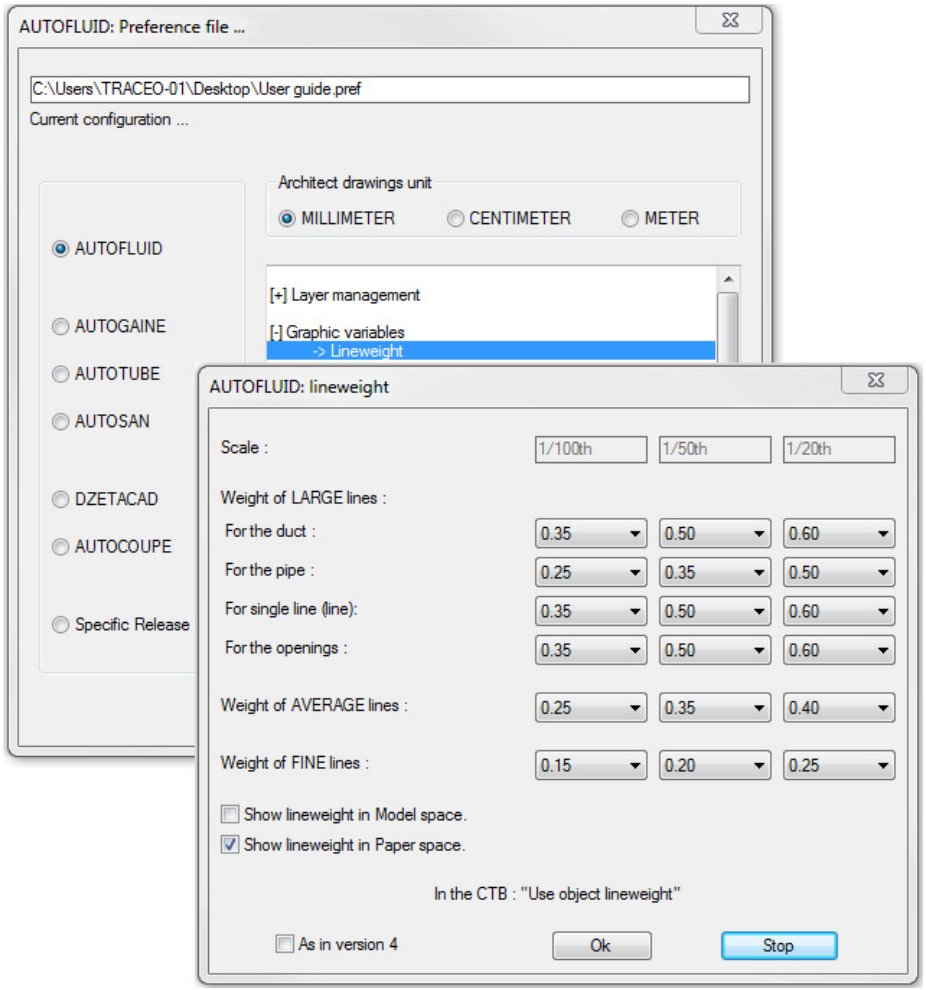
When you add a new branch, using your CAD mirror command for exemple, it will appear in orange by default.

Remember: every modification of the network diagram structure must be followed by a new capture to update the dynamic calculations.



Managing line thicknesses

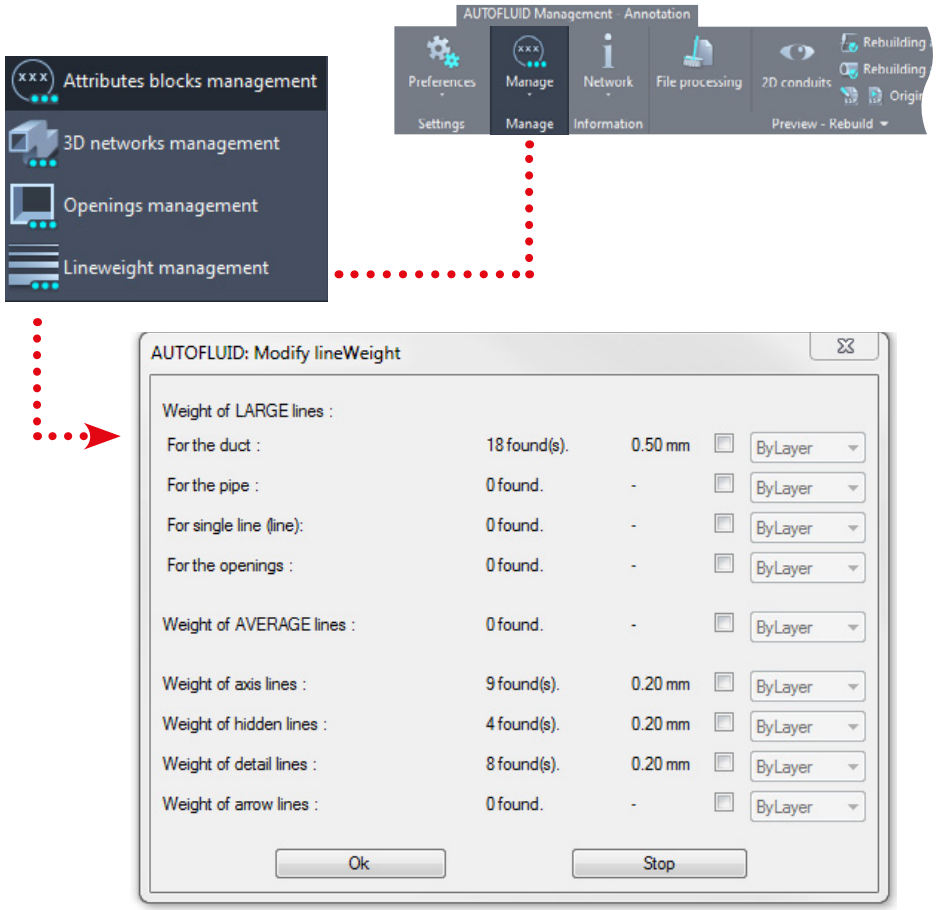
The thickness of each entity drafted with AUTOFLUID is automatically managed to be compatible with basic CTB files (acad.ctb or monochrome.ctb.)



Whether in color or not, the relief of your drawing will be preserved.

The above settings must be adjusted BEFORE drawing.

If you would like to change thicknesses AFTER drawing you can modify them by using the following command:



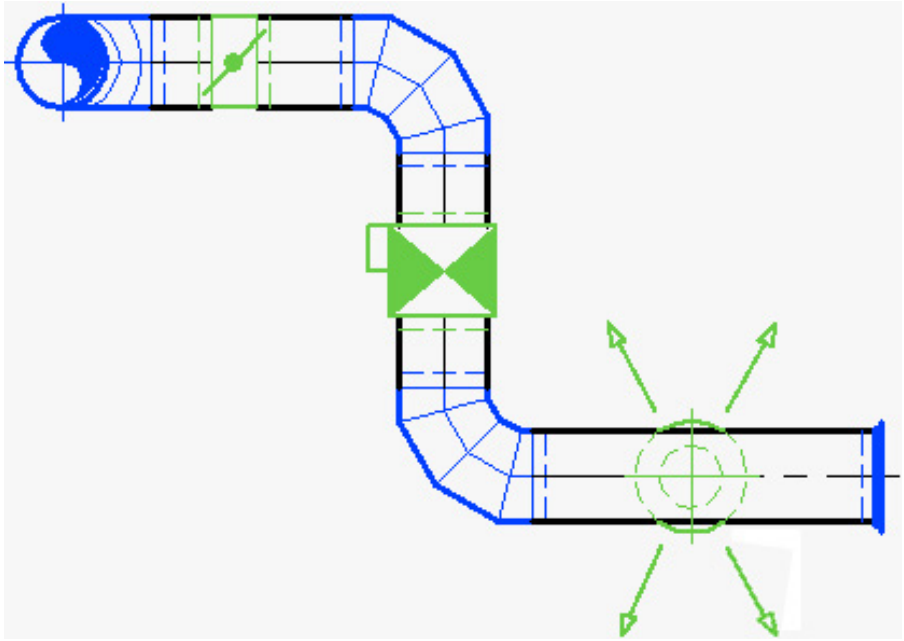
Introduction to double line drawing

Graphic structure of a network drafted with AUTOFLUID

Each entity created belongs to only one object.

There are 3 types of objects:

- Conduits (colored black)
- Parts (colored blue)
- Equipment (colored green)



Each object contains information serving the following purposes:

Modifications on a network

Modifications on text

Network bill

Calculation of pressure drops

Each part must be created using the appropriate command.

Otherwise the part may seem right graphically but the bill will be wrong and the commands for quick modifications may not work correctly.

Structure of a network

Structure of a conduit

There are 3 types of conduits:

- 3 lines with 1 axis (circular duct or tube)
- 2 lines (rectangular duct)
- 1 line or 1 polyline (single line)
- 4 lines of which 2 are dotted (Smoke exhaust conduit)

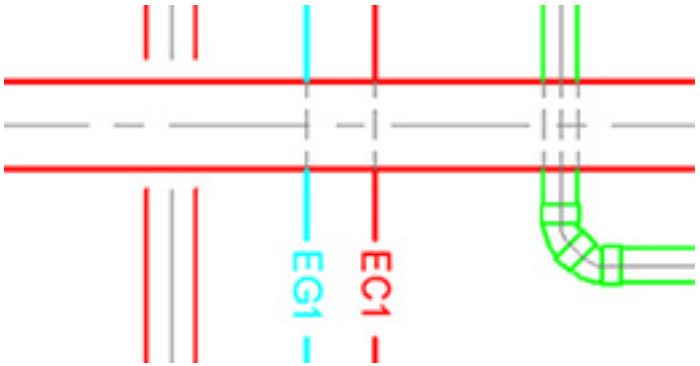
When a circular conduit contains no additional information, it will only be composed of 3 independent lines.

In AUTOFLUID, each line of the conduit acknowledges the 2 others.

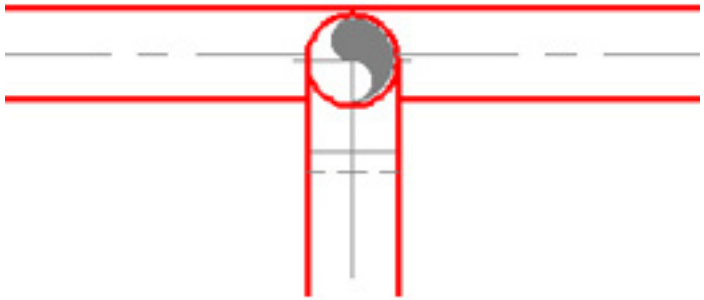
For this reason all conduits must be properly structured.

Factors that can alter the structure of a conduit:

- Crossings



- The «ADJUST» and the «BREAK» commands in your CAD software



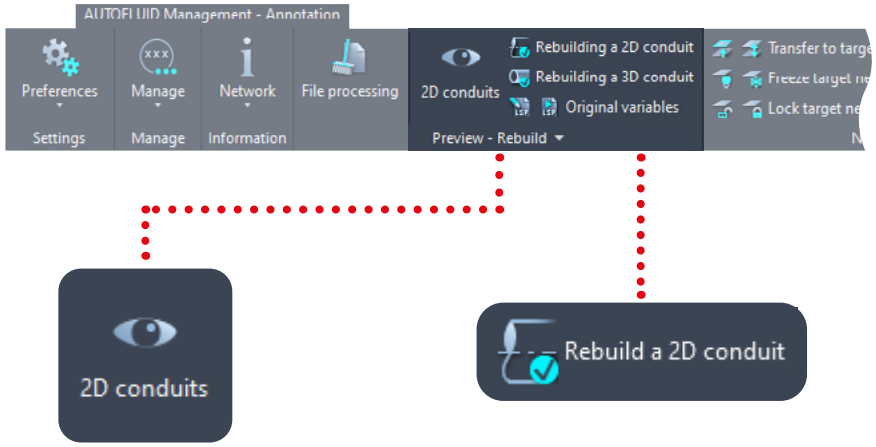
A partially cut conduit (2 out of 3 lines) will generate 2 conduits composed of 3 lines.

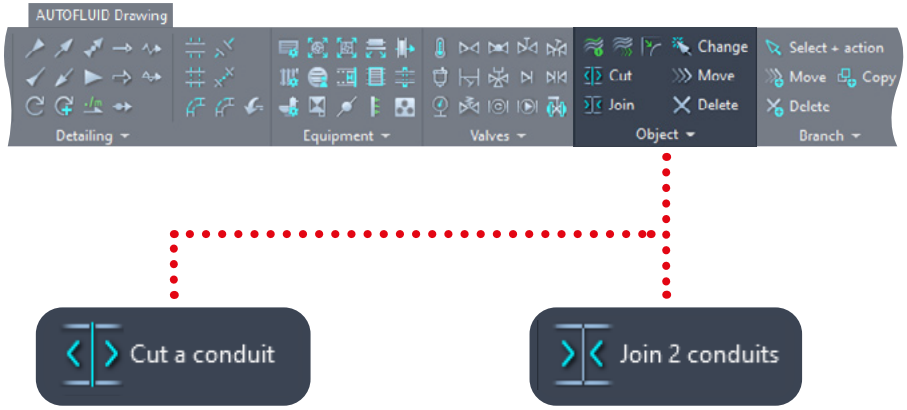
- Text on a line

If only the centre line of a conduit is cut with the “BREAK” command in your CAD software then AUTOFLUID will generate 2 conduits composed of 3 lines.

Structure of a conduit

Operations on conduits





2D Conduits

This allows all the conduits to turn into one colour and all the parts into another colour. If a conduit contains wrong information or if it is badly structured then it will become red. When this happens you must re-structure the conduit.

Rebuilding a conduit

Select the lines that form the conduit and specify the pipe size.
This information will be updated and the conduit will be acknowledged by all the commands.

Cut a conduit

Selct a conduit and split it in two piece without gap between them.

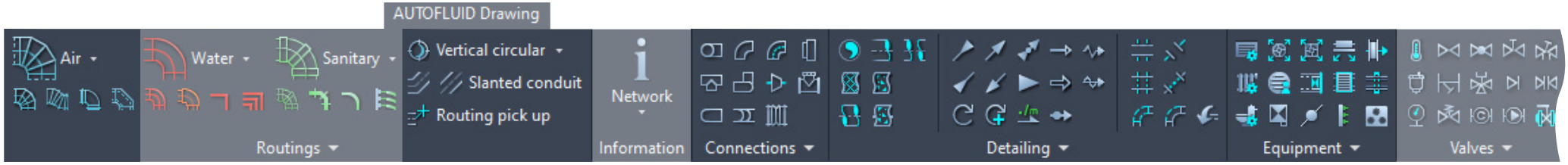
Join 2 conduits

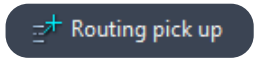
Select two conduits with a similar size and form one out of the two.

Operations on conduits

Drafting double line air networks

There are 3 types of drawing commands in this ribbon:





The routing command lets you draft circular ducts regardless of the shape of the network.

It contains many options (elbows, reducers, etc.) to model ducts while building them.

The command also takes into account layer management when drafting and allows the insertion of text relating to the drawing.

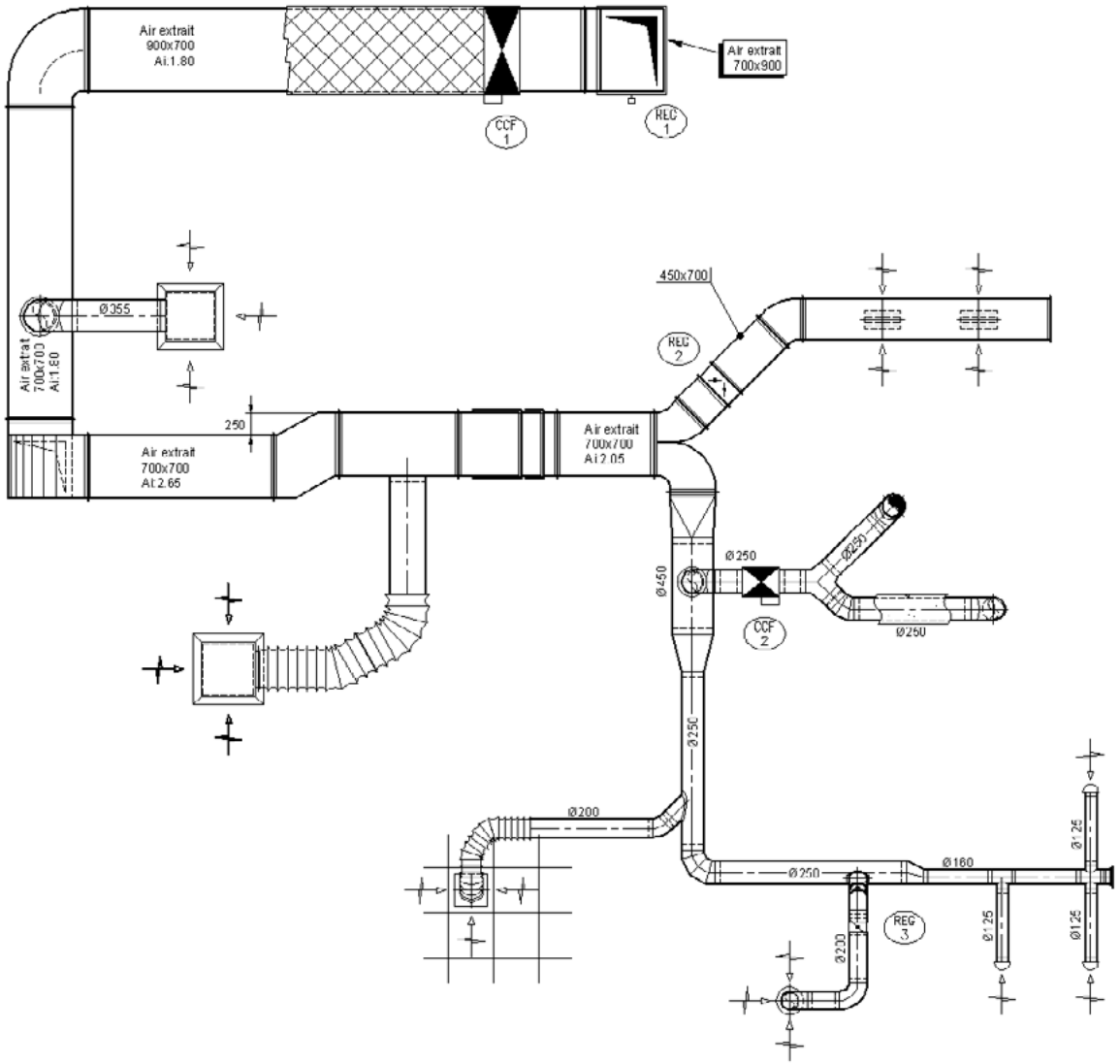
The pick up commands allow you to «hook» onto a pipe you've already drawn and then continue building on it.

All the other commands are intermittent or dressing commands.
For examples: Elbow, Tee, Break...
Damper, Insulation, Fire protection...

Drafting double line air networks

Exercise: drafting an air duct network

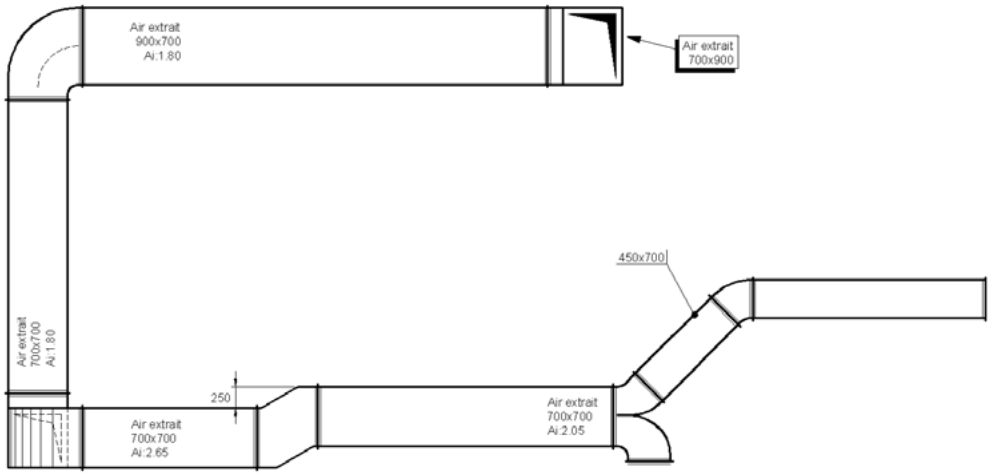
The duct beside can be drafted in 4 steps.



Drafting double line air networks

Step nº1

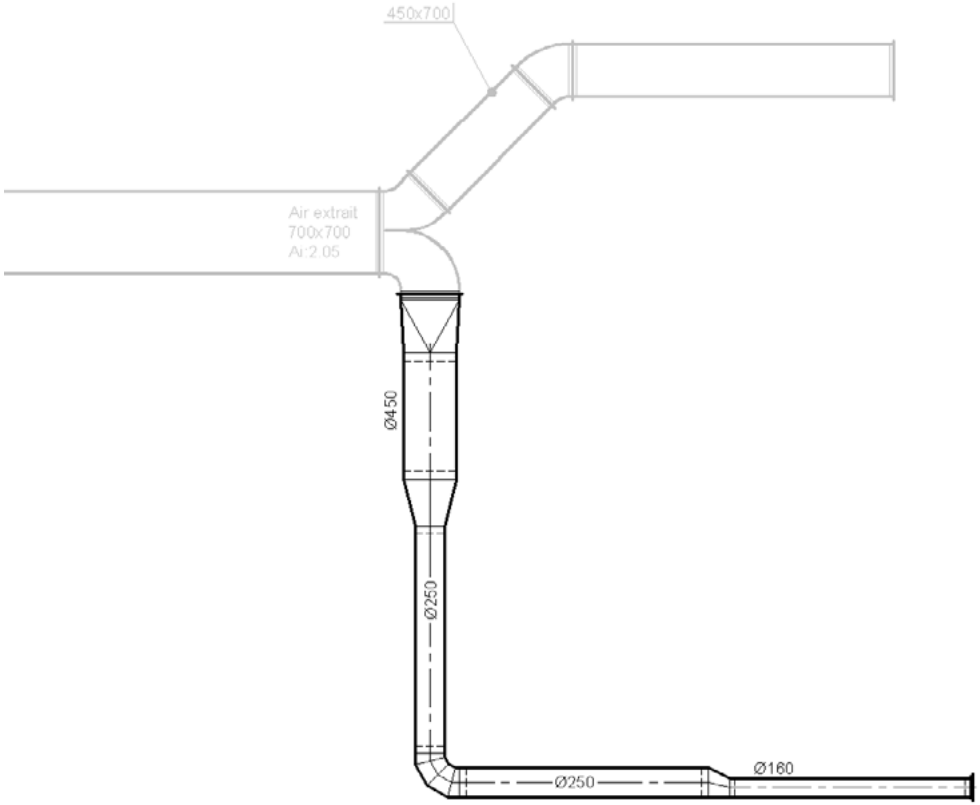
Start with the longest section and continue to the end of a branch (All the way to the cap).



Double line air networks drafting - Step 1

Step nº2

Pick up again from a diverging piece and as in part 1, continue until the end of the branch.



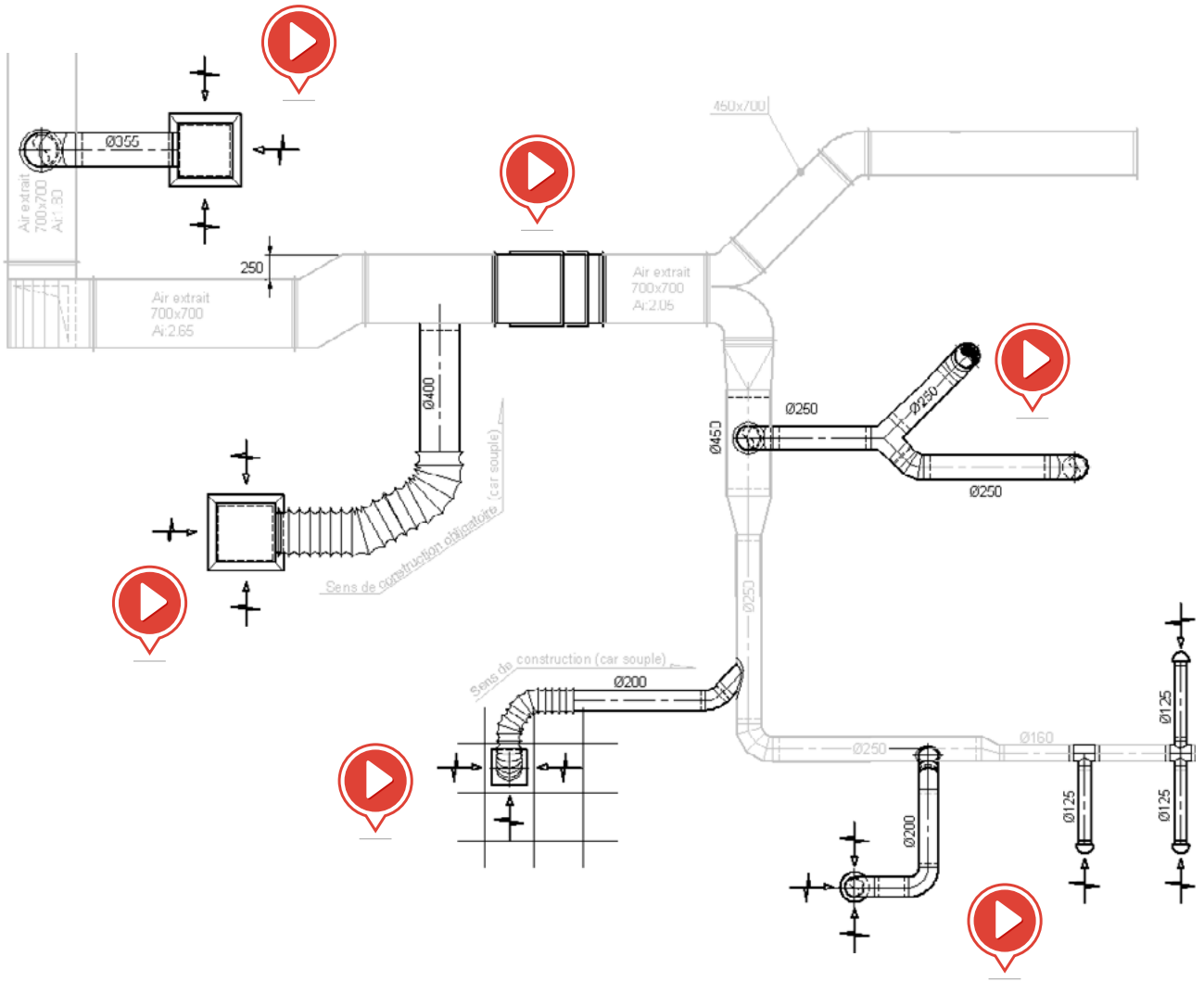
Double line air networks drafting - Step 2

Drafting double line air networks

Step 3

Create each of the branches - This can be done in 2 ways:

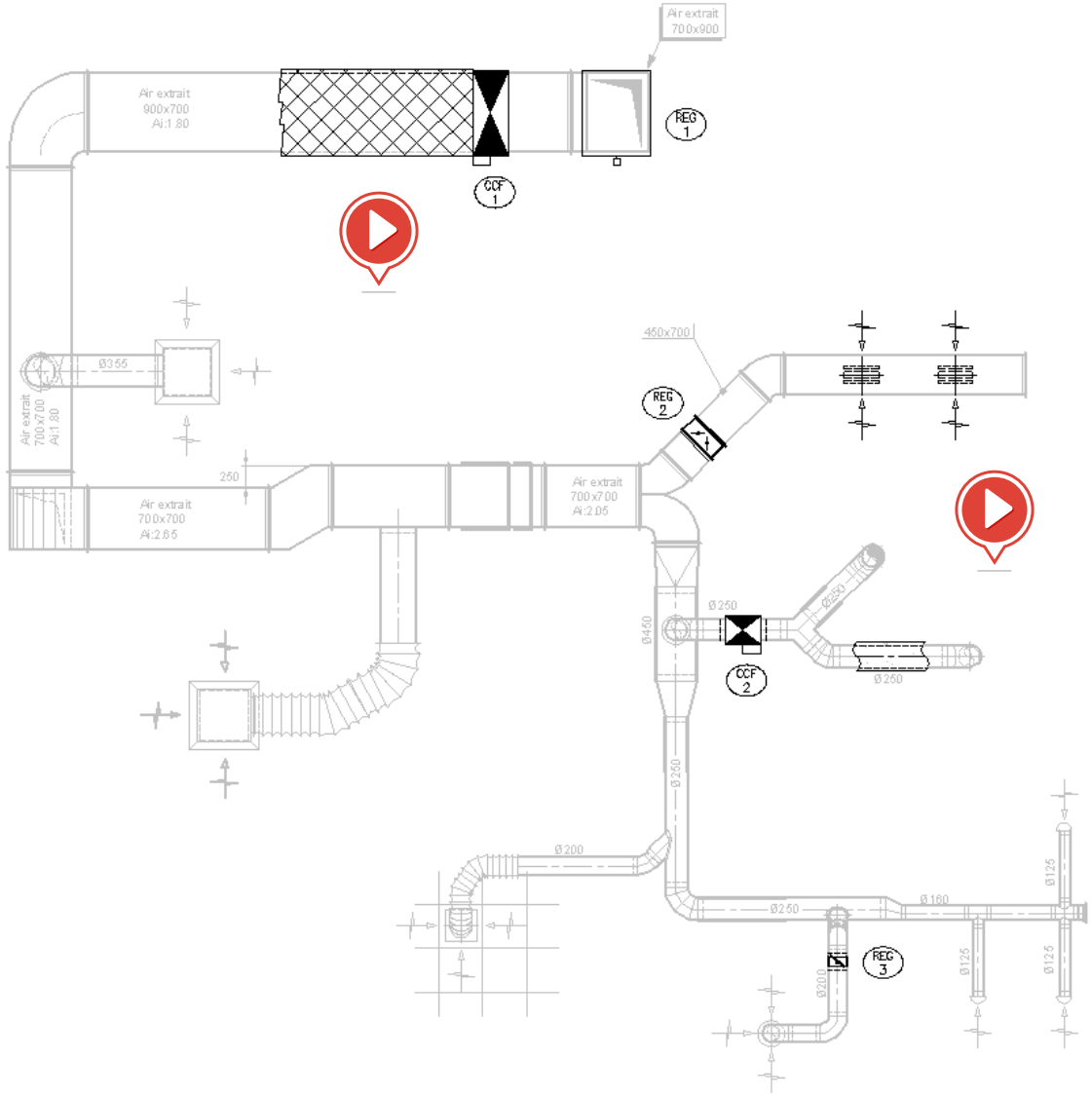
1. Start from the register and work your way towards the main duct (it is compulsory in the case of a flexible duct).
2. Start from the main duct and work your way towards the register.



Drafting double line air networks

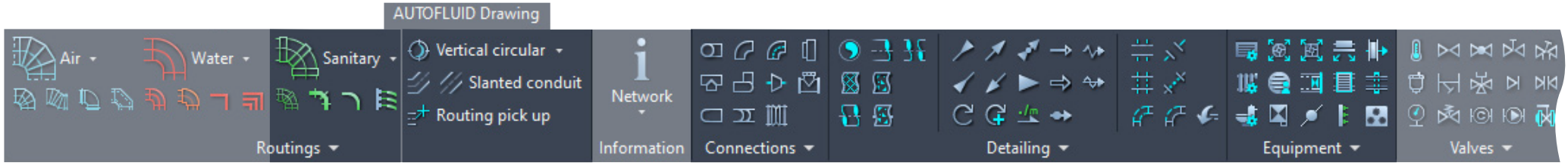
Step n°4

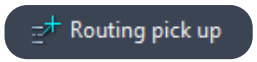
Dress the duct with equipment: valve, damper, fire proofing, registers on networks...



Drafting double and single line drains

There are 3 types of commands in this toolbar:





The routing command lets you draft circular ducts whatever the shape of the network.

It contains many options (elbows, reducers, etc.) to model ducts while building them.

The command also takes into account layer management when drafting and allows the insertion of text relating to the drawing.

The pick up commands allow you to «hook» onto a pipe you've already drawn and then continue building on it.

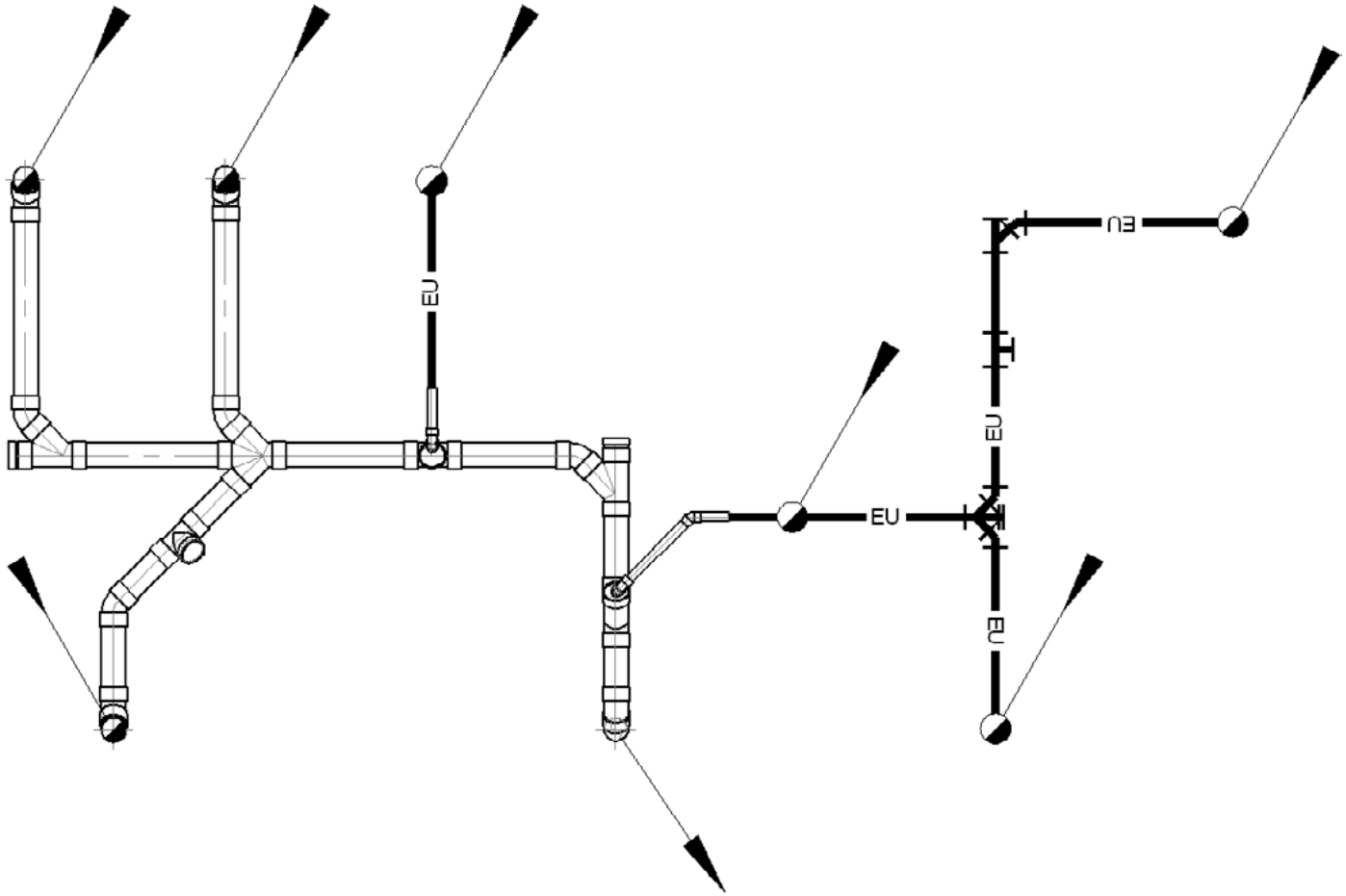
All the other commands are intermittent or dressing commands.

For example: Elbow, Reducer...
Inspection plug...

Drafting double and single line drains

Exercise: drafting a double line drains network

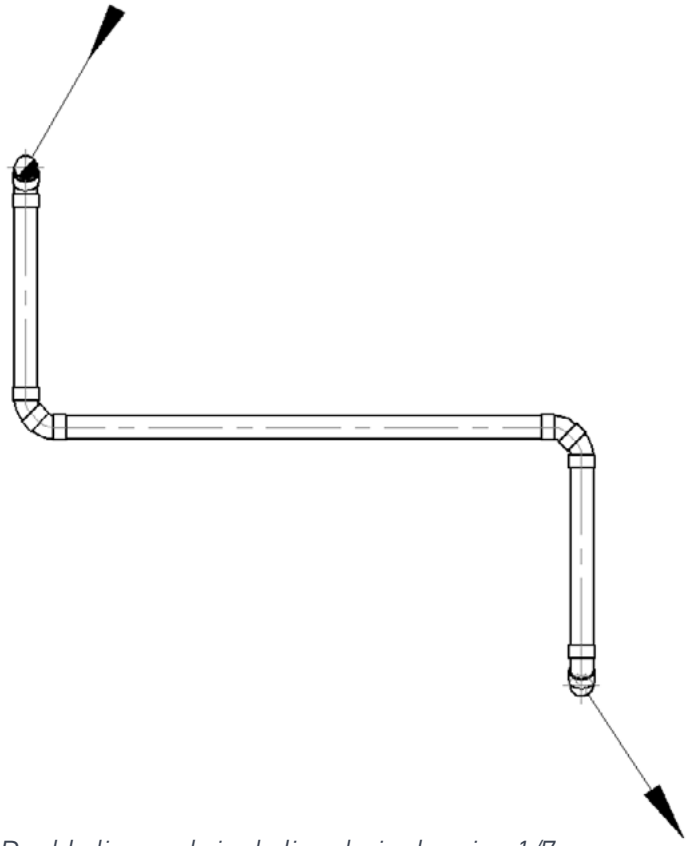
Several steps are necessary to draw the network below:



Drafting double and single line drains

Step nº1

Start from the furthest point and continue towards the end of the network.

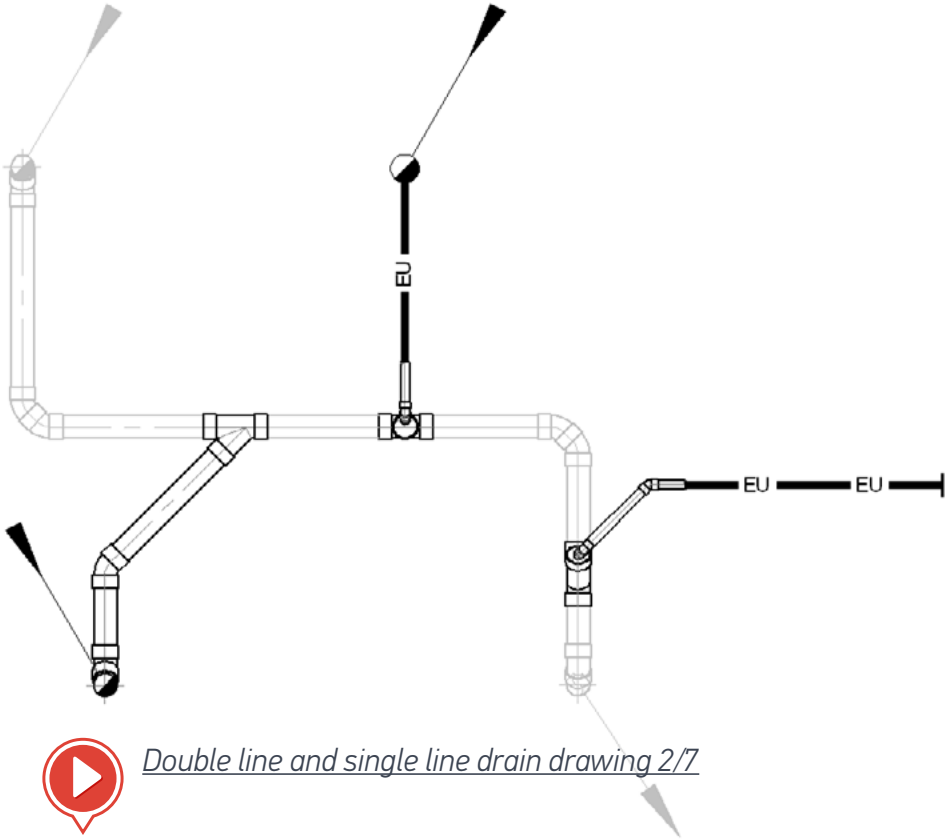


Double line and single line drain drawing 1/7

Step nº2

Make each connection - This can be done in 2 ways:

1. Start from a symbol and continue towards the main sewer.
2. Start from the main sewer and continue towards the symbol.

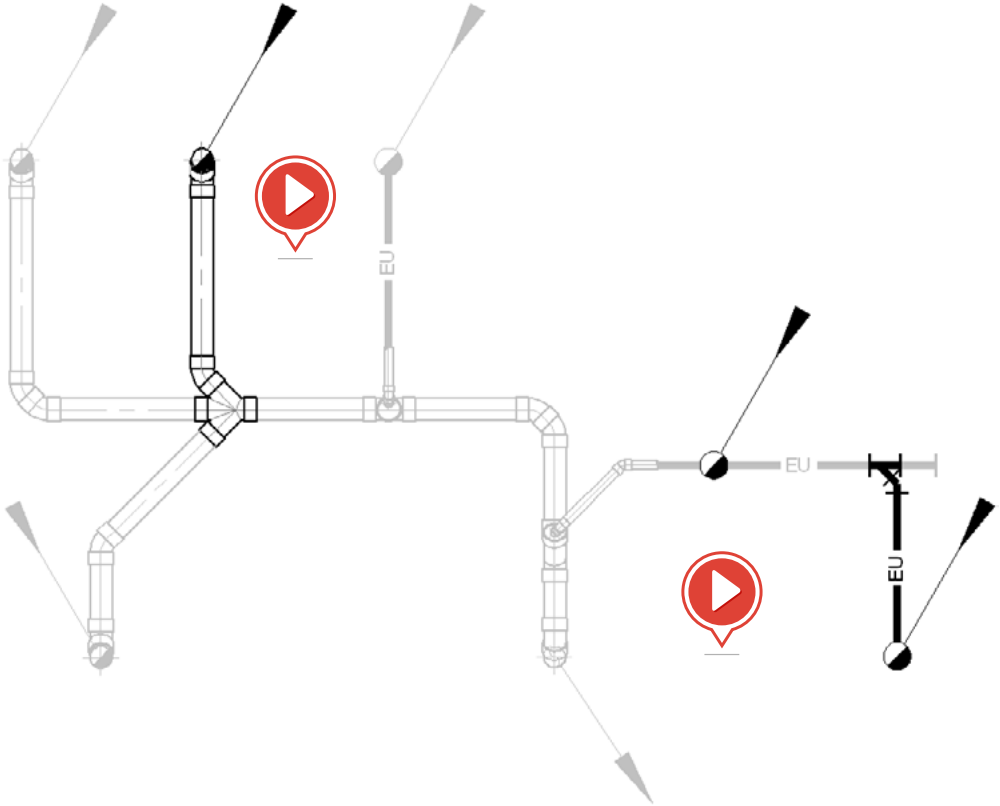


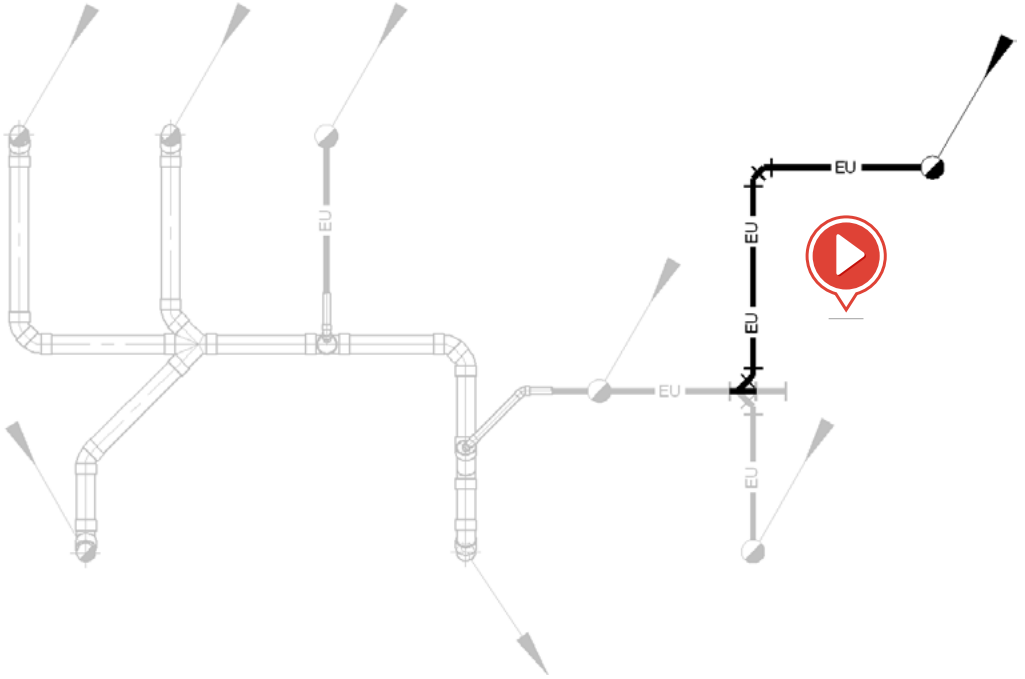
Double line and single line drain drawing 2/7

Drafting double and single line drains

Step nº3

Add the other connections.

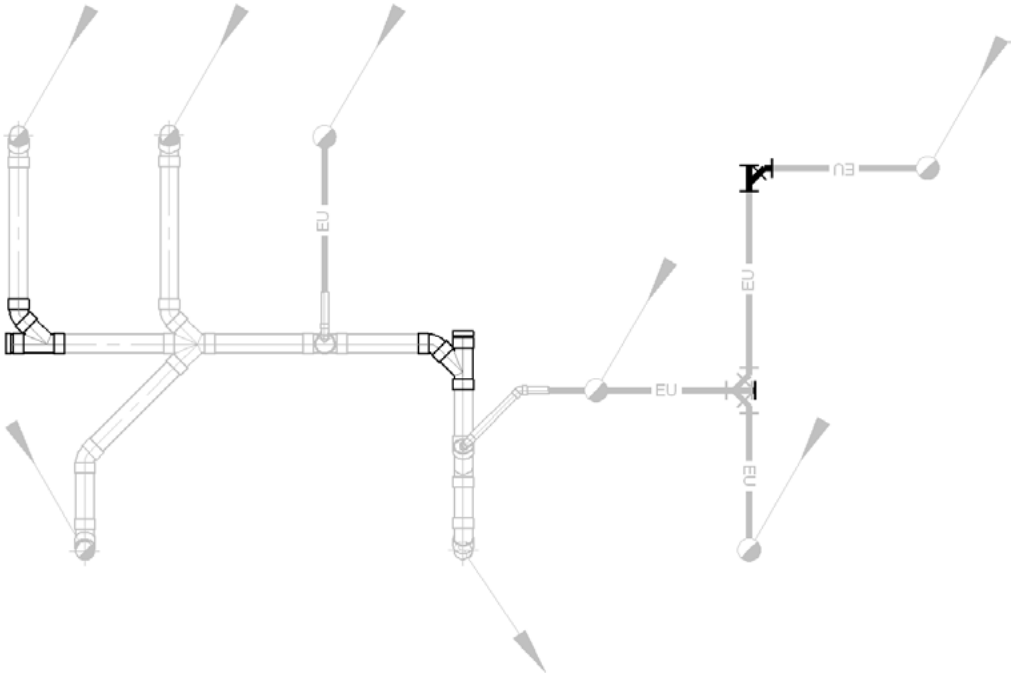




Drafting double and single line drains

Step nº4

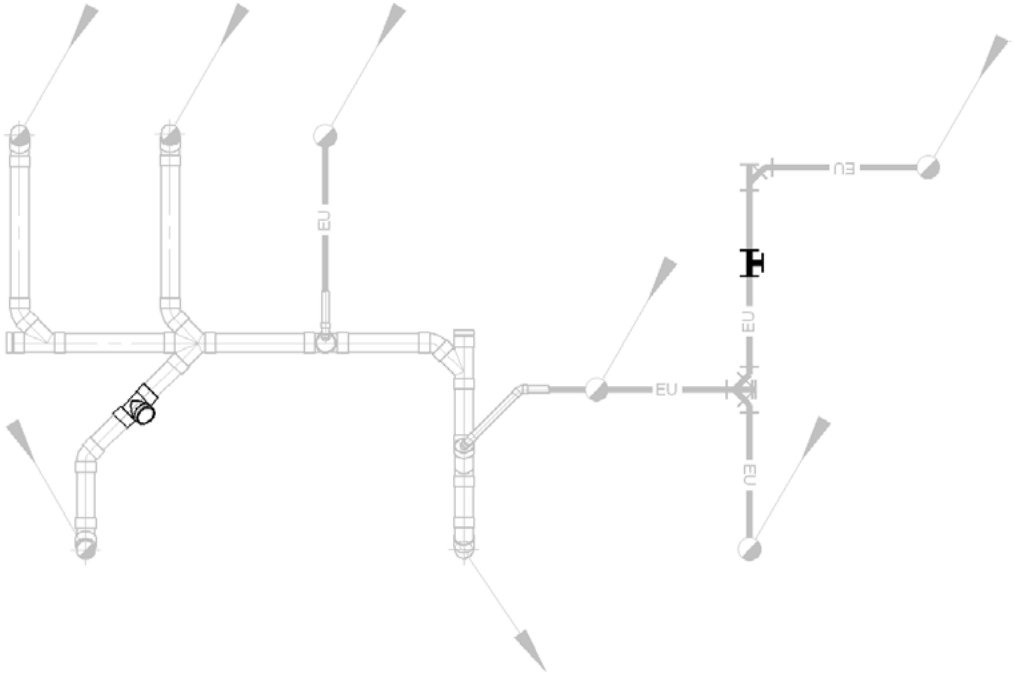
Use the «MODIFY» command to adjust the network.



Double line and single line drain drawing 6/7

Step nº5

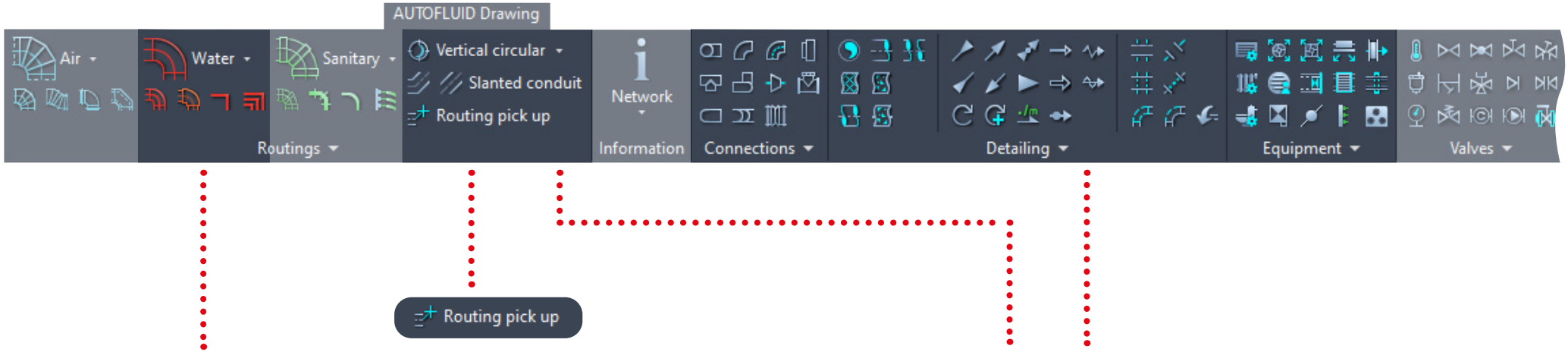
Add the equipment.



Double line and single line drain drawing 7/7

Drafting a single line set of pipes

There are 3 types of commands in this toolbar:



The routing command lets you draft circular ducts regardless of the shape of the network.

It contains many options (elbows, reducers, etc.) to model ducts while building them.
The command also takes into account layer management when drafting and allows the insertion of text relating to the drawing.

The pick up commands allow you to «hook» onto a pipe you've already drawn and then continue building onto it.

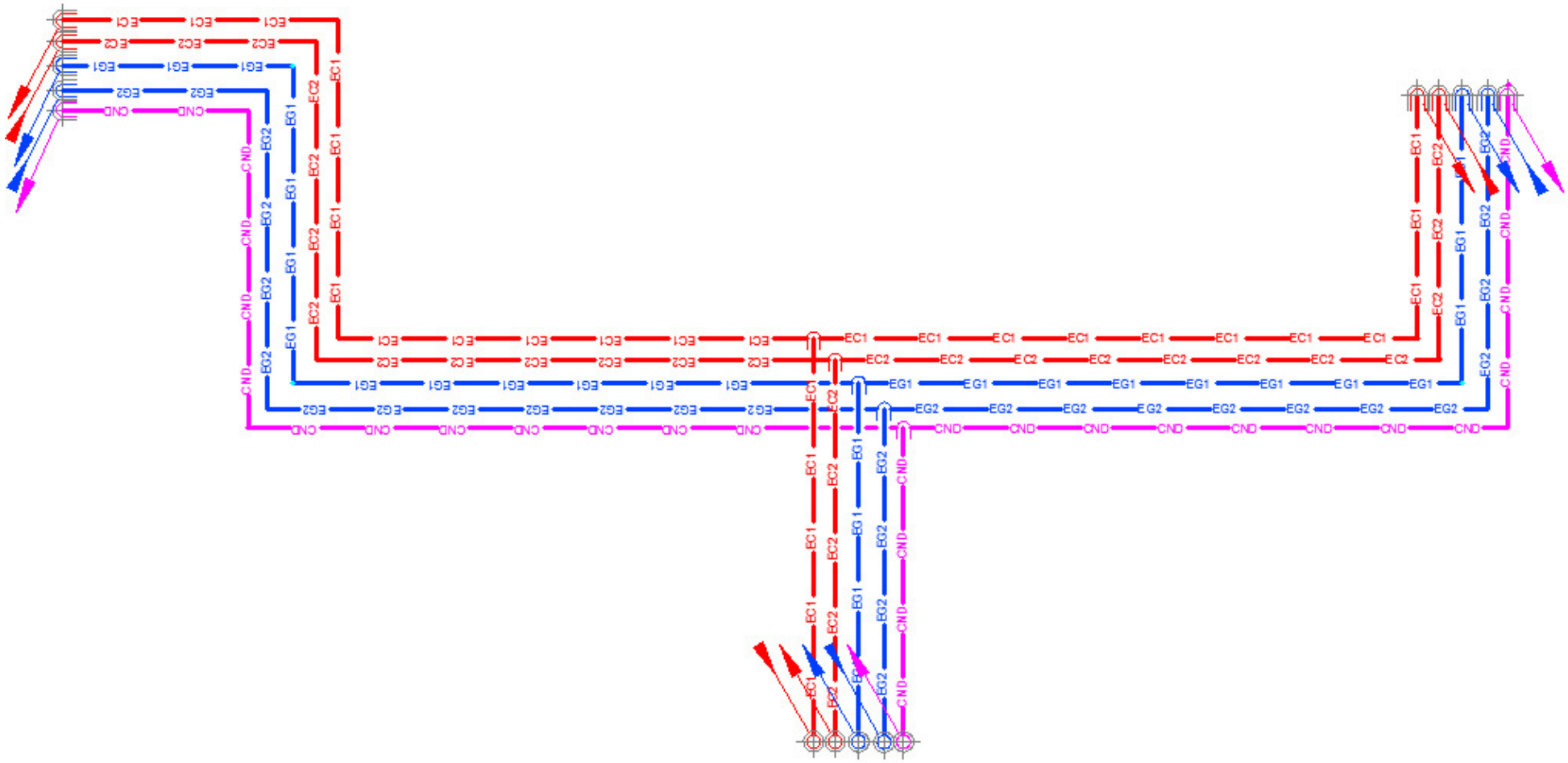
All the other commands are intermittent or dressing commands.

Examples: Elbow, Reducer...
Gates, Thermometers...

Drafting a single line set of pipes

Exercise: drafting a single line set of pipes

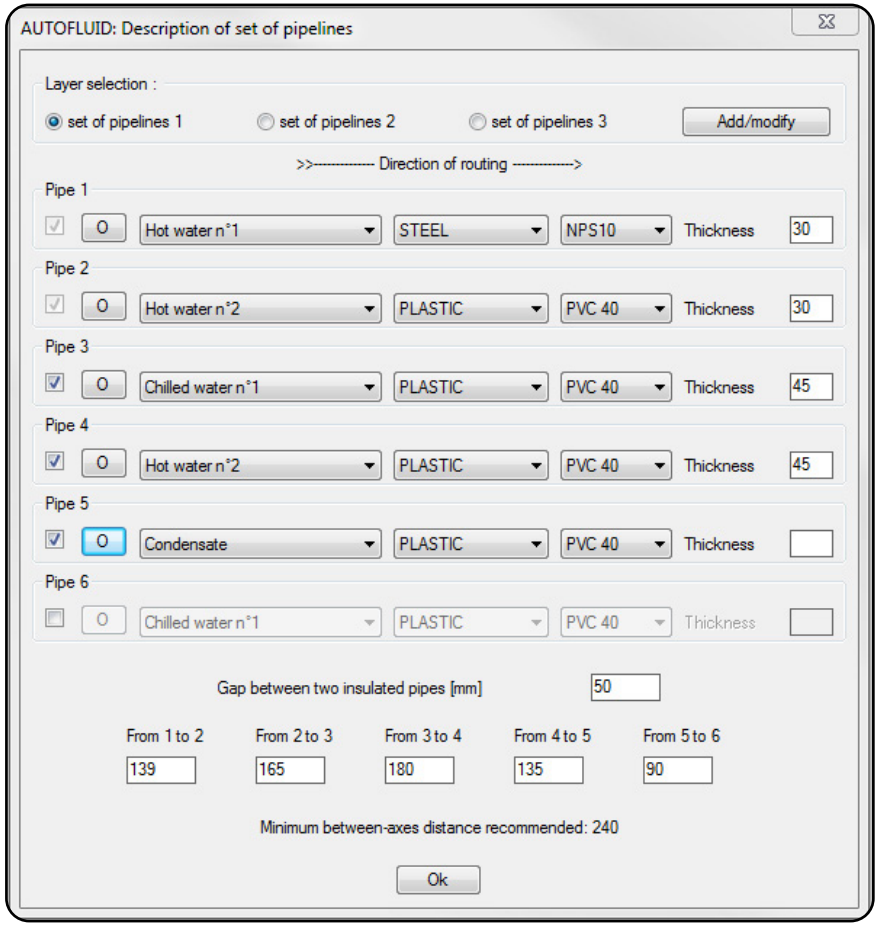
Several steps are necessary to draw the set of pipes below:



Drafting a single line set of pipes

Step nº1

Define the sets of pipes



You must specify:

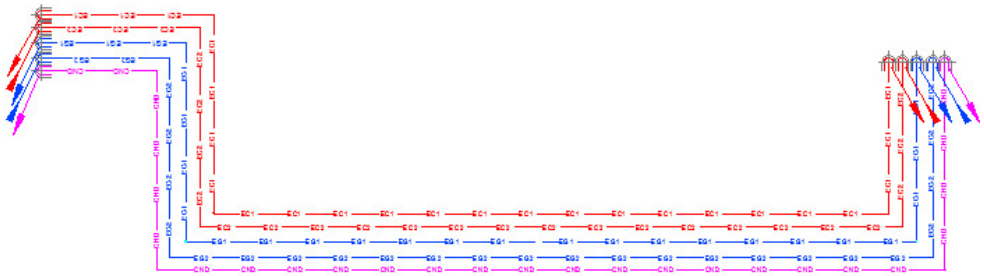
1. The number of tubes
2. And for each tube:
   - the name of the network (layer)
   - the specification
   - the pipe size
   - the thickness of insulation (if applicable)
3. The distance between each tube.

Sets of pipes Step 1

Drafting a single line set of pipes

Step nº2

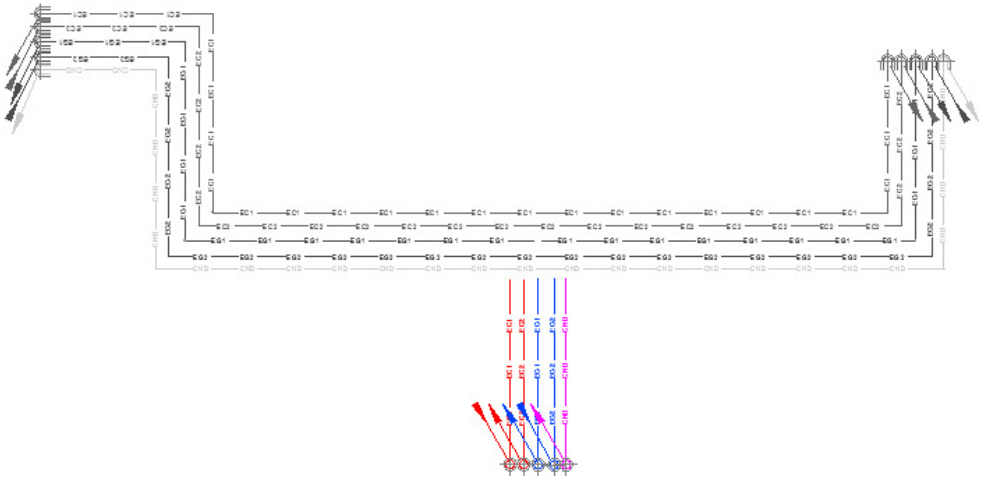
Layout the main path.



Set of pipes Step 2

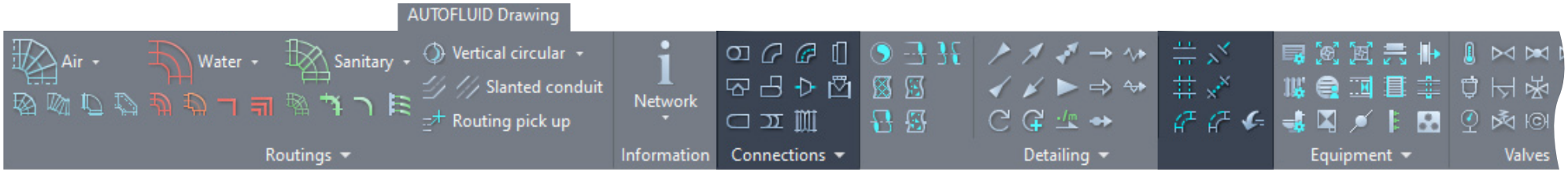
Step nº3

Add the additional branch without creating a connection.



Set of pipes Step 3

Drafting a single line set of pipes

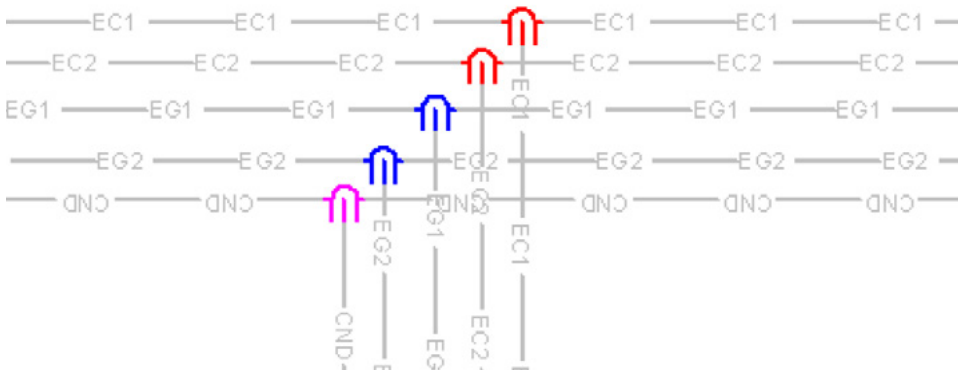


Connection, tee, Y branch

Cut crossings

Step nº4

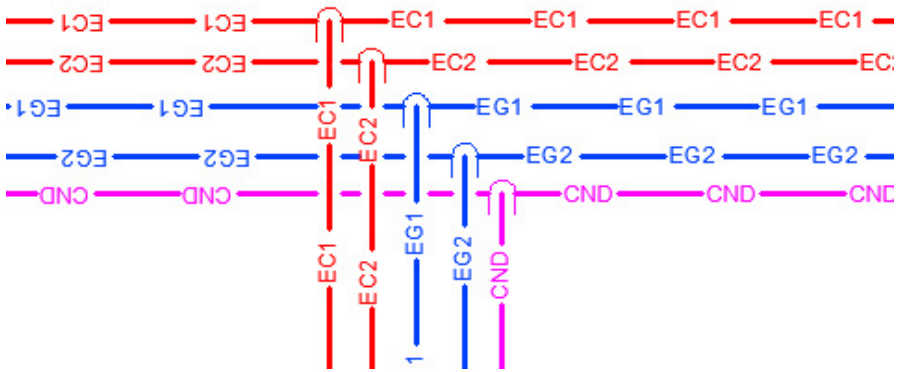
Create intermittent connections using commands in the «single line» toolbar:



Set of pipes Step 4

Step nº5

Manage crossings.

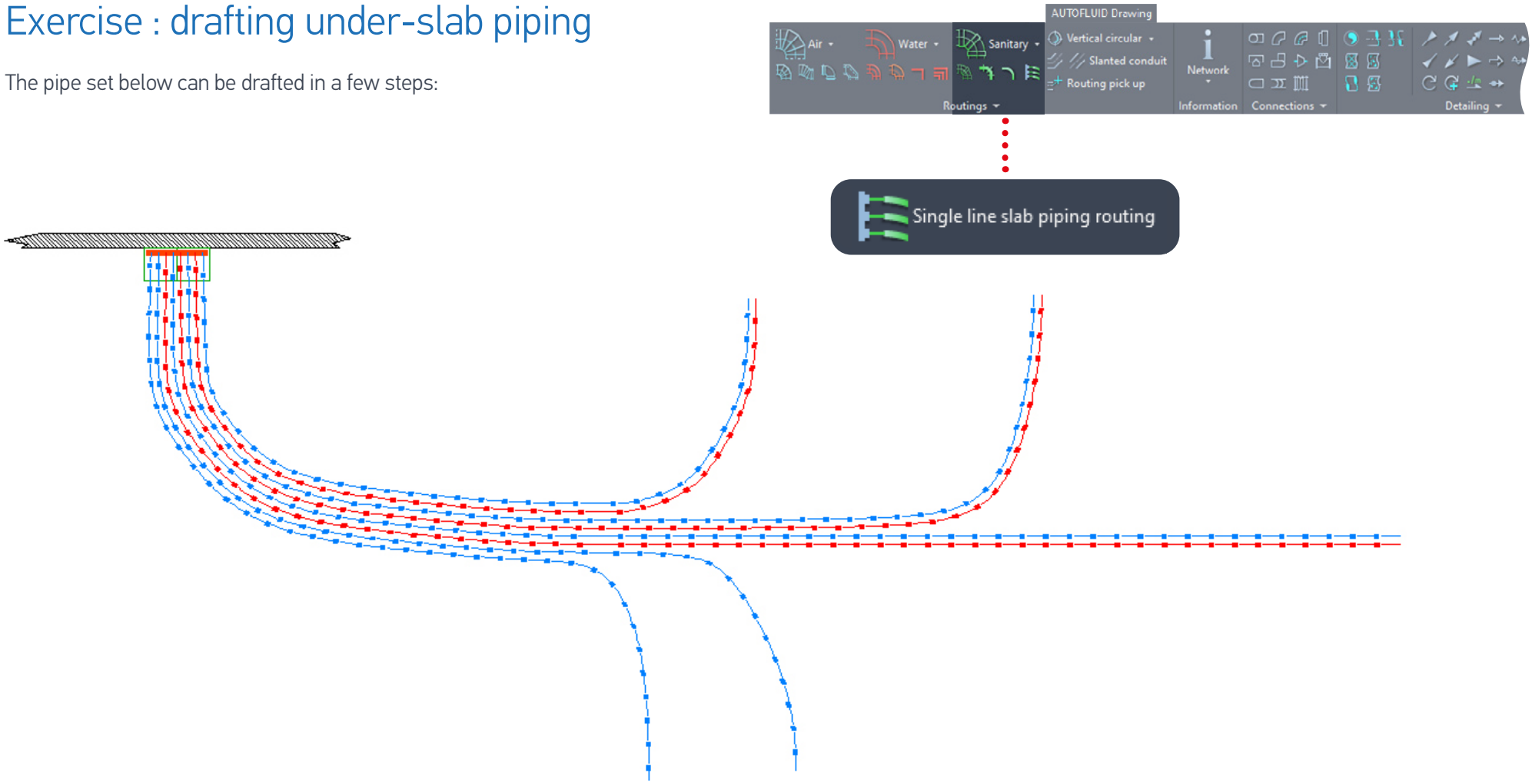


Set of pipes Step 5

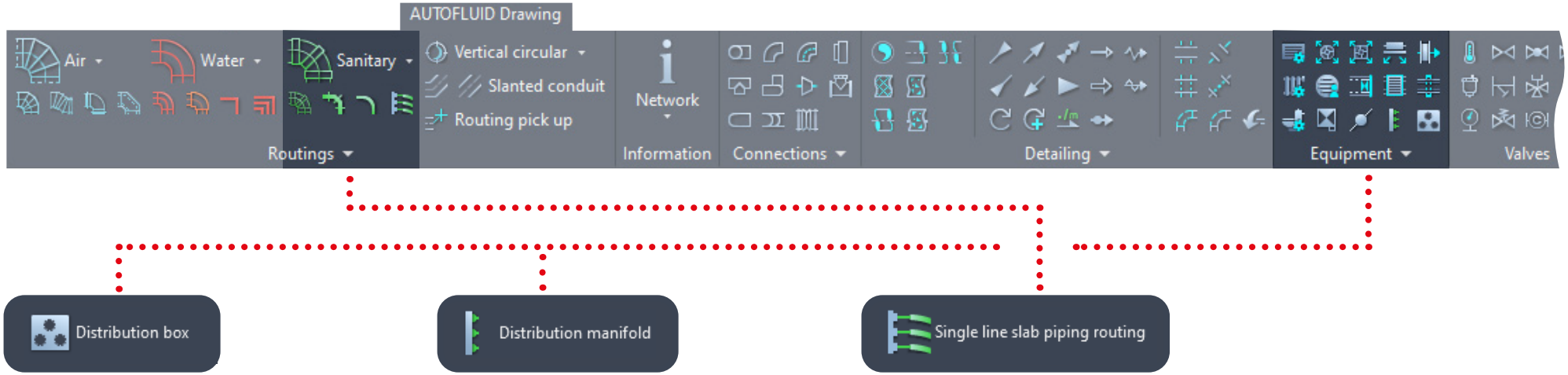
Drafting under-slab piping

Exercise : drafting under-slab piping

The pipe set below can be drafted in a few steps:



Drafting under-slab piping



Step nº1

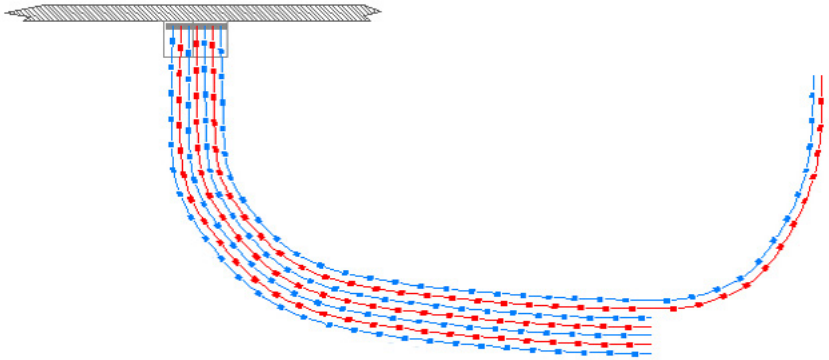
Position the distribution boxes.

Step nº2

Position the distribution manifold.

Step nº3

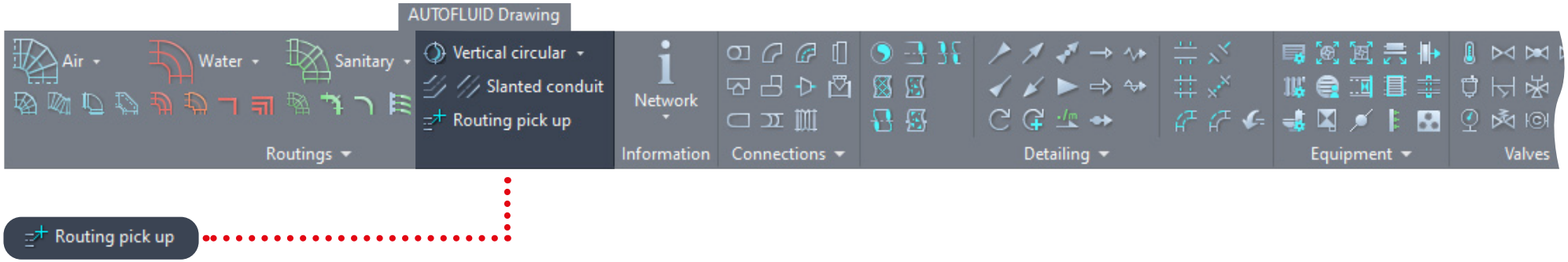
Start to draw from the manifold towards one of the entry points.



Under slab piping steps 1 and 2

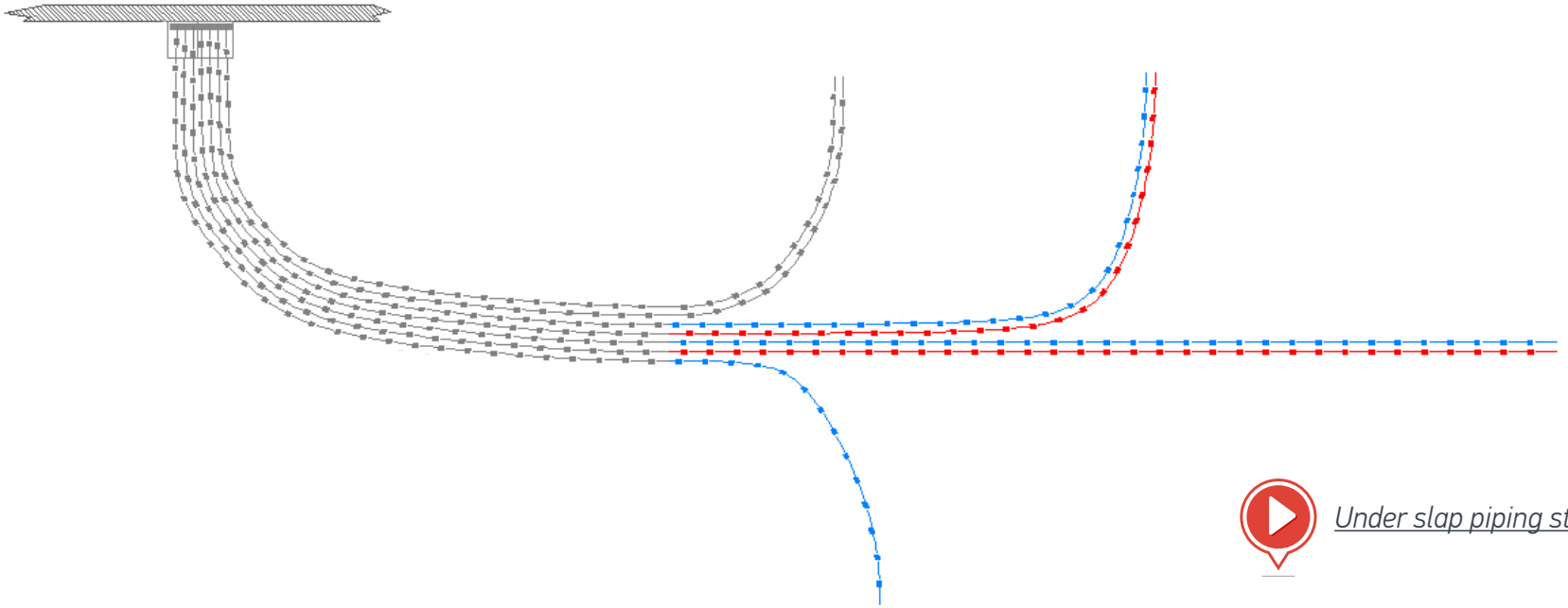
Under slab piping step 3

Drafting under-slab piping



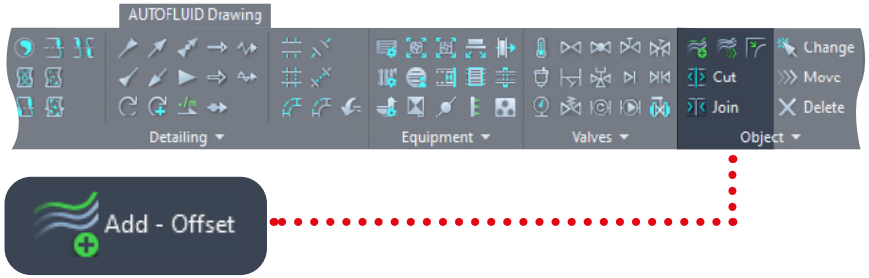
Step nº4

Feed each entry point using the pick up commands.



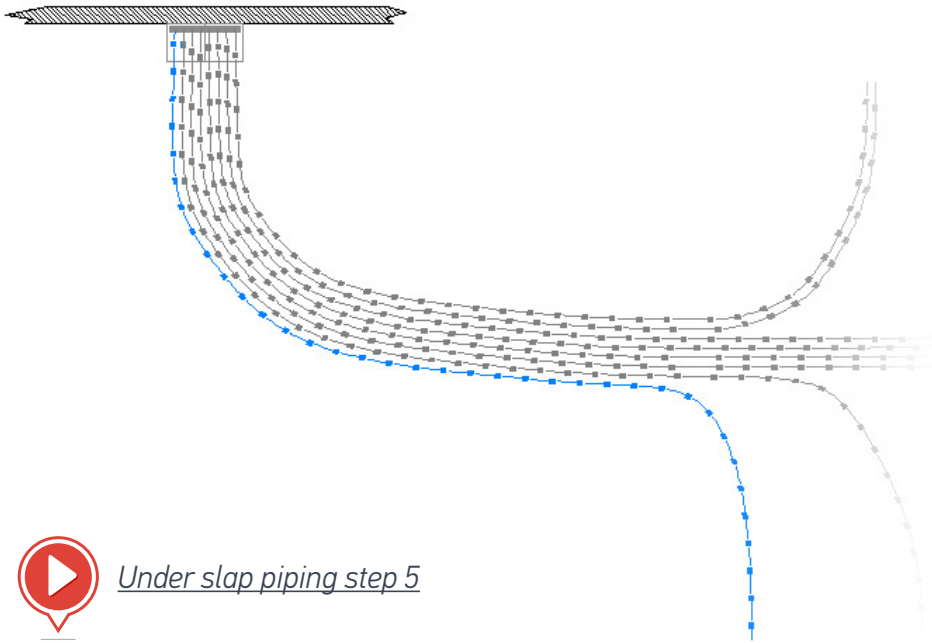
Under slap piping step 4

Drafting under-slab piping



Step nº5

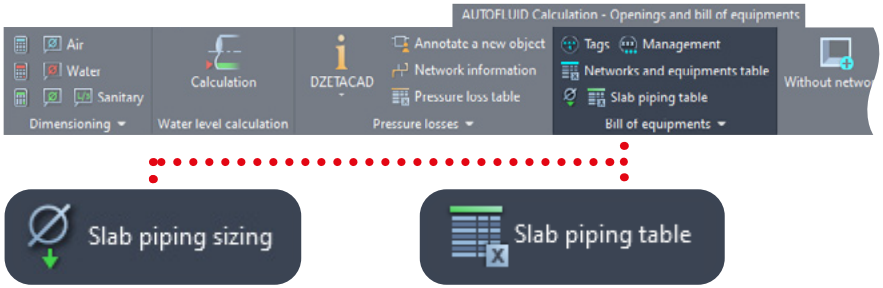
Add un under slab pipe to the pipe set.



Under slap piping step 5

A drafted network can be repositioned.

Reposition under slab pipes



Once the network is designed, specify the diameters.

Now the bill of materials can be produced.

NETWORK QUANTITY

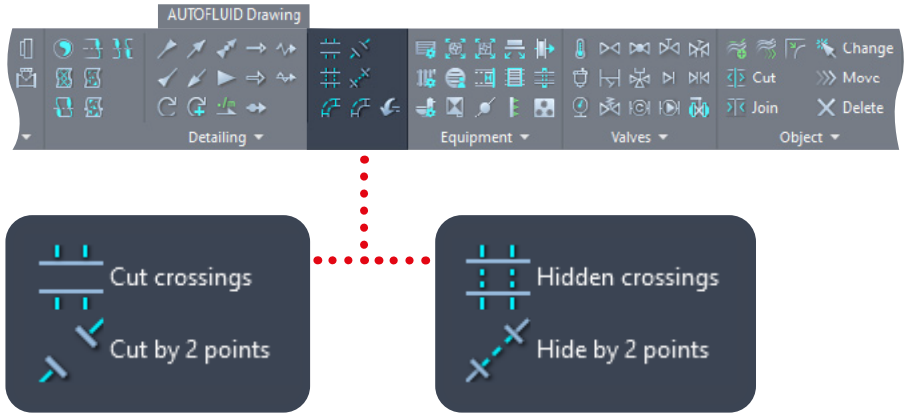
Defined zones: ALL

MULTI-LAYER PIPE

Naming	Dim.1	Dim.2	Dim.3	Quantity [m]	Area [m²]	weight[Kg]
CONDUIT	20			41.40	-	-
CONDUIT	25			119.68	-	-

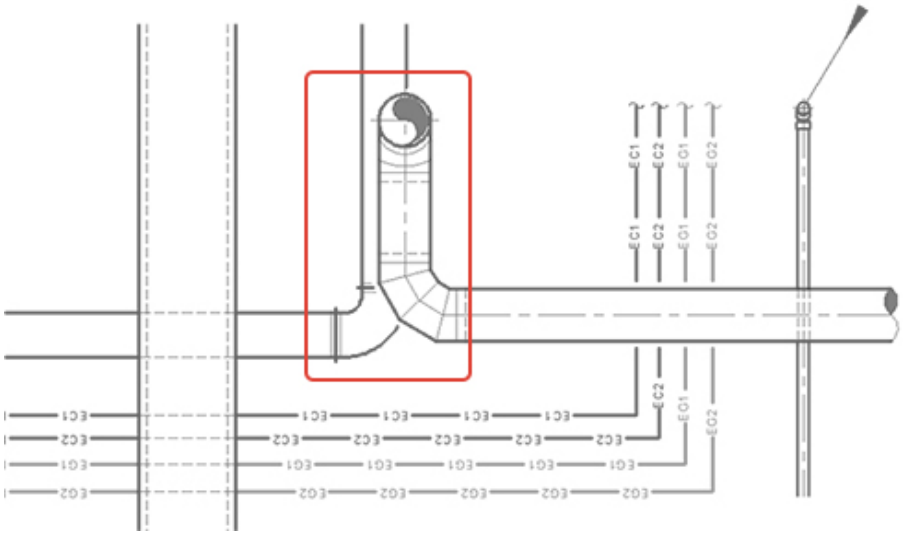
Under slab piping BOM

Network crossings



There are 2 possible modes:

- With dotted lines
- Cuts with gaps



There are 2 ways to operate (regardless of the chosen mode):

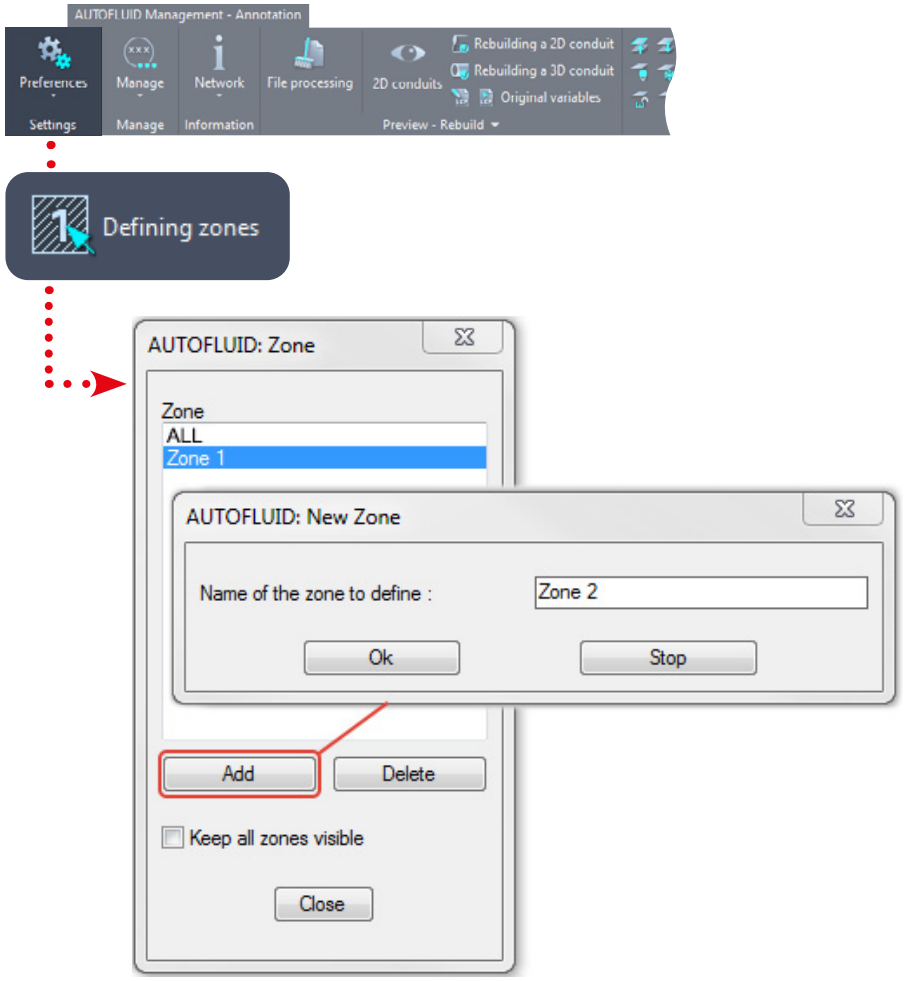
- Simple case: one conduit crosses another
  - Select the conduit to modify (the one below).
  - Select the boundary conduit (the one above).

- Other cases (circled red in the image)
  - Select the conduit to modify (the one below).
  - Confirm (to modify the selected entity).
  - Click on the first and then the second intersection.

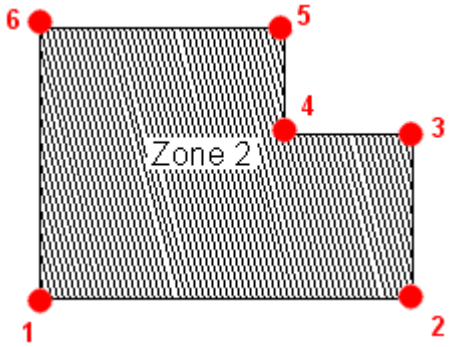
Crossings

Defining zones

Zones will be used to quantify networks and equipment. Bills can be based on layers (frozen or not) and zones. To create a zone:

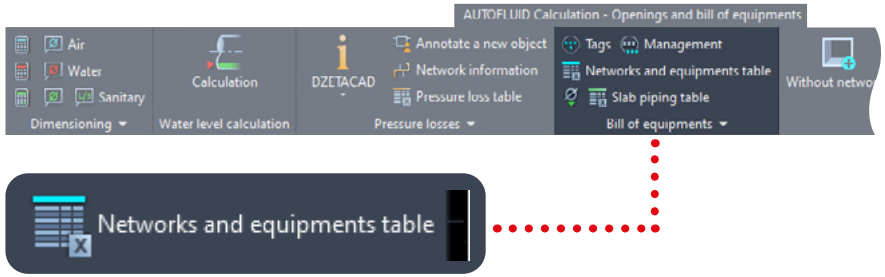


Once the new zone is named, select the points forming the outline of this zone. If the last point clicked is different to the first, then the zone will close automatically.



Defining zones

Network bills



Objects drawn on frozen layers won't be taken into account.

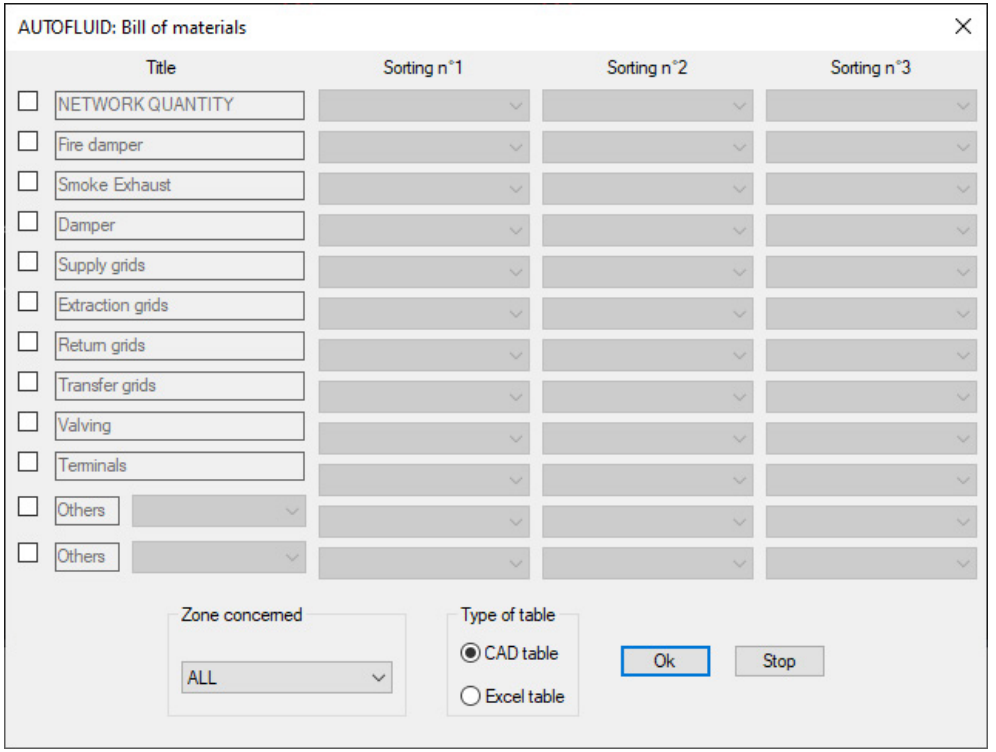
You can create a bill based on zones or networks (via layers).

Once the drawing is complete you can create a table listing all the different objects created in the «Model space». Click on «NETWORK QUANTITY» :

1. Click on «NETWORK QUANTITY»
2. Choose the appropriate zone
3. Choose the type of table (in the paper space of your CAD software or in Excel)
4. Click OK and place the table.

The table will not automatically update with new changes to your drawing.

If you have to make amends to your drawing, delete the existing table and create a new one.



Network bills

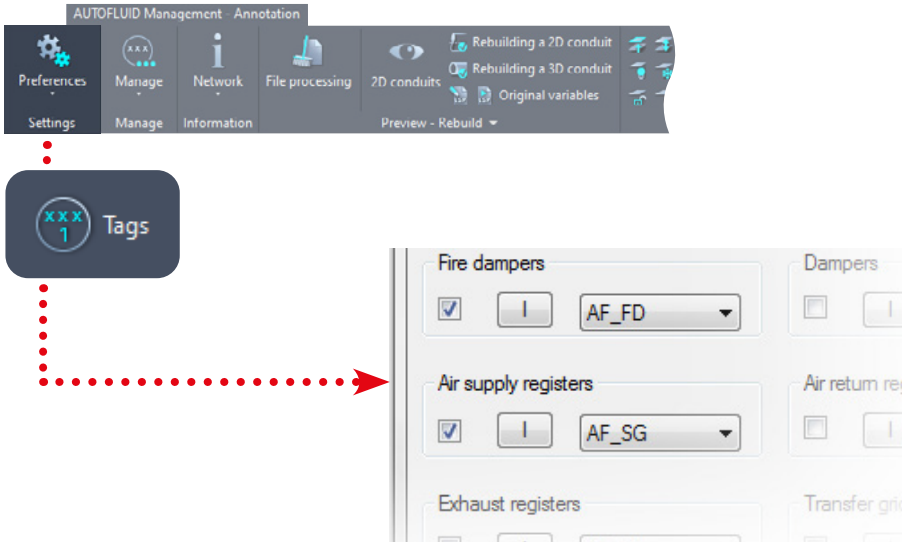
Equipment bills

To create an equipment bill (registers, valves, dampers, etc.) you must tag the equipment first.

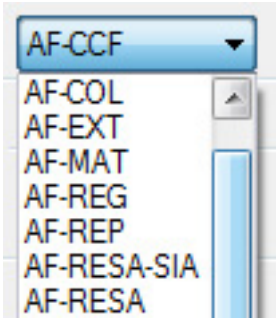
In order to do this you need to use assigned blocks.

A few ready-made assigned blocks come with AUTOFLUID. You can use them as they are or use them as a basis to create new ones.

To tag equipment while drawing you must assign blocks to equipment pieces:



Select the assigned block with which to tag the equipment:

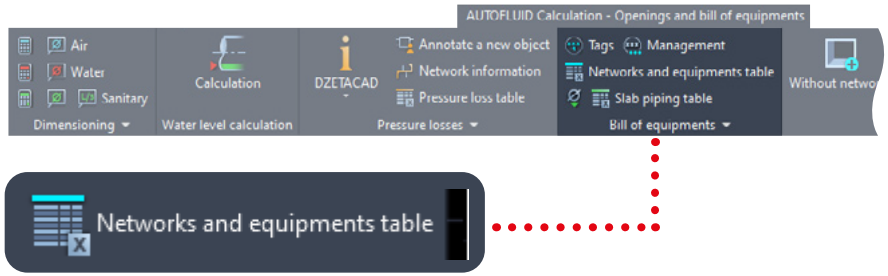


If the box ☑ is ticked, then the tag will be inserted whilst the equipment is added to the drawing.

The button I lets you insert blocks once the equipment has been added to the drawing.

Equipment bills 1/2

Equipment bills



1. Select the equipment you wish to list
2. Choose the zone affected
3. Choose the type of table (In the paper space of your CAD software or in Excel)
4. Click OK and place your table.

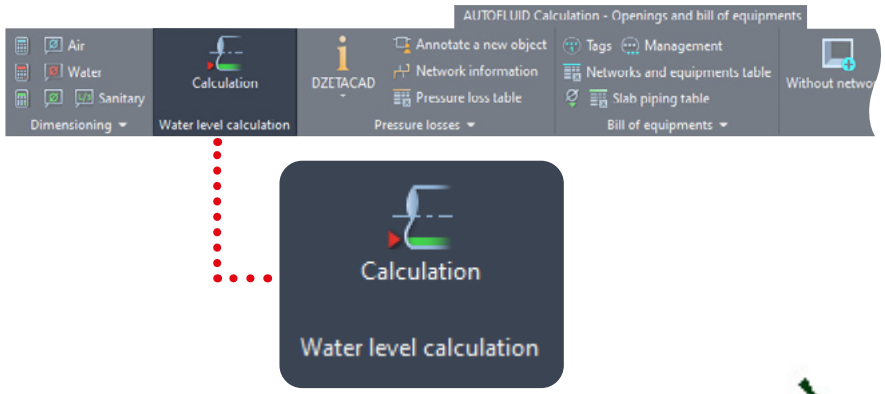
Objects drawn onto a FROZEN layer will not be taken into account, therefore you can create a bill according to zones and networks (via layers).

The table will not automatically update with new changes to your drawing.

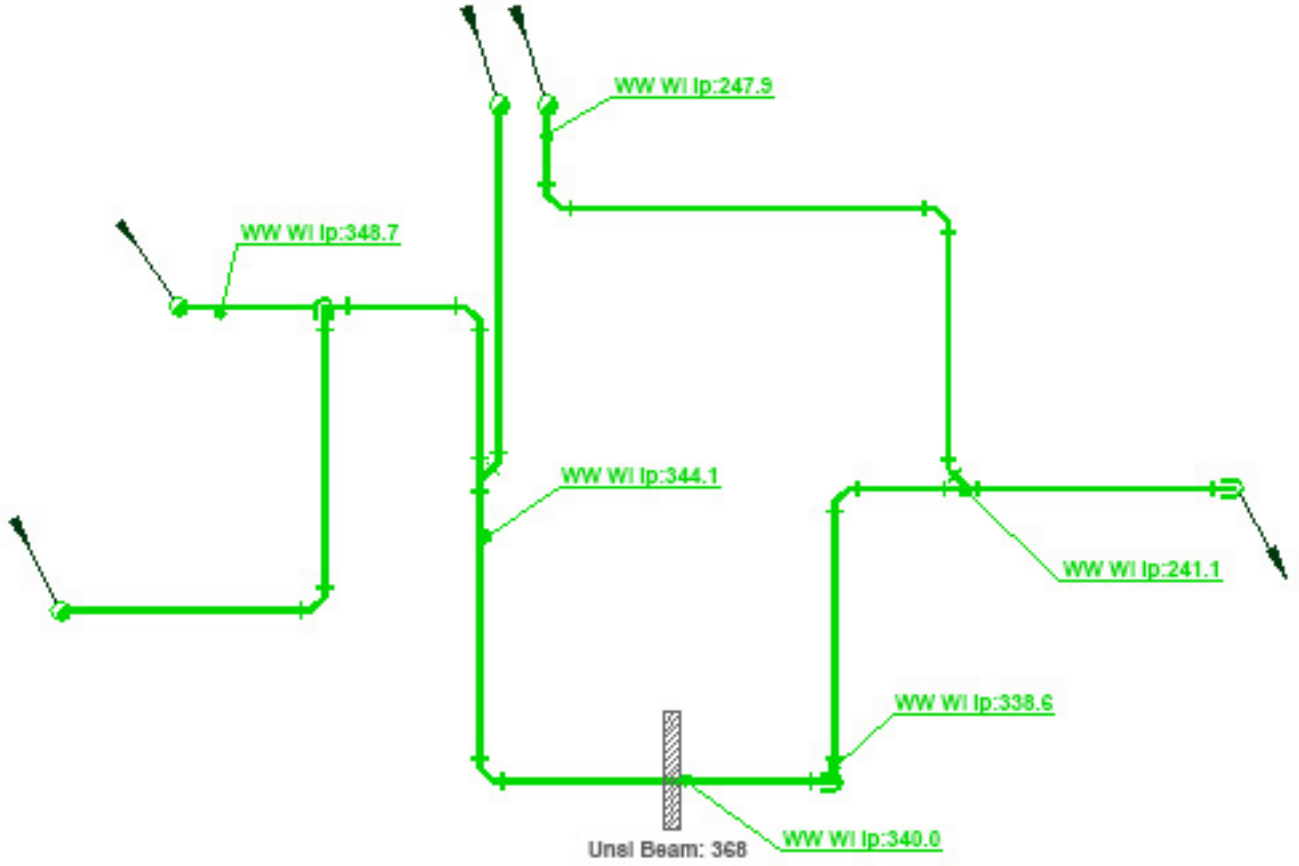
If you have to make amends to your drawing, delete the existing table and create a new one.

Equipment bills 2/2

Calculations and texts for water levels

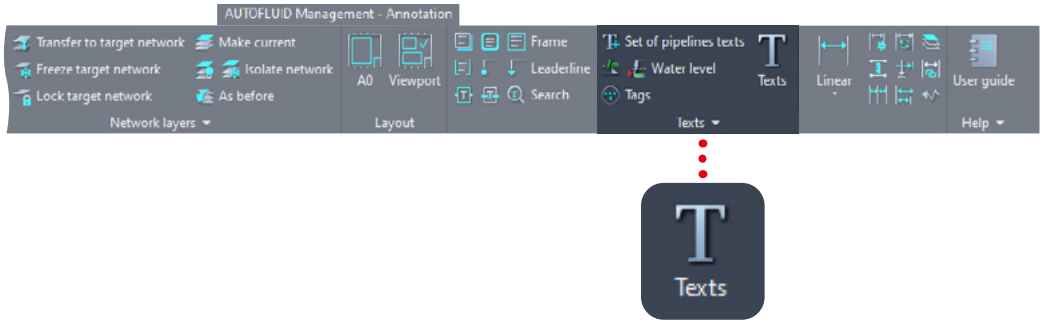


This command lets you compute different levels of a drain network. Indicate a few key points in order to calculate the new water level in relation to the network's segments.

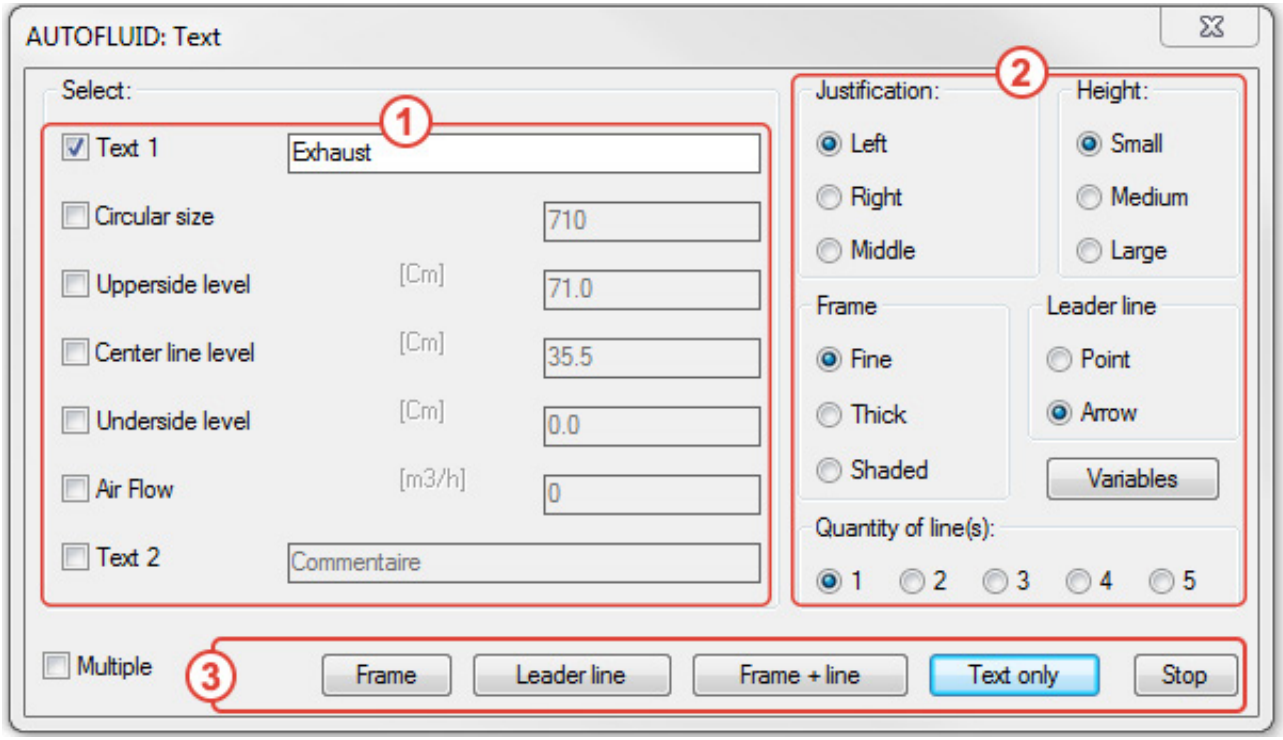


Water levels

Texts



Let's use the example of a circular duct: select the relevant entity. The dialogue box below opens. It is pre-populated with the information already available.



Texts

1. Choose texts to write: tick the box beside the text you would like to include.

2. Choose the text appearance:

- Height and justification
- Type of frame
- Type of leader line
- Number of lines.

3. Choose options of presentation:

- Text only
- Framed text
- Underlined text with leader line
- Framed text with leader line.

Texts

N.B.

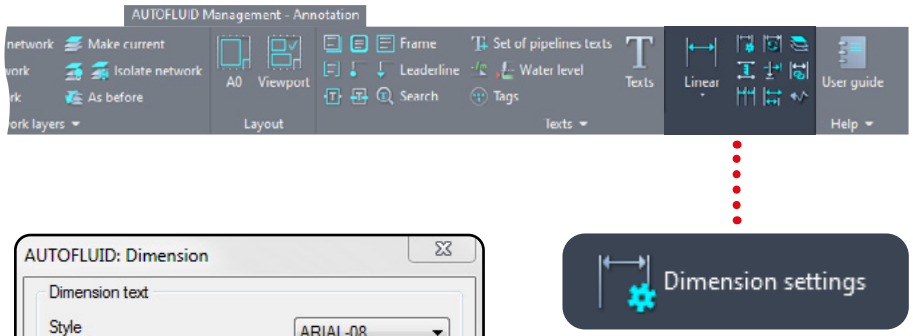
When calculating one level in relation to another level and according to the pipe size, check the level's unit in the preferences panel (calculation variables) to ensure the calculation is right.

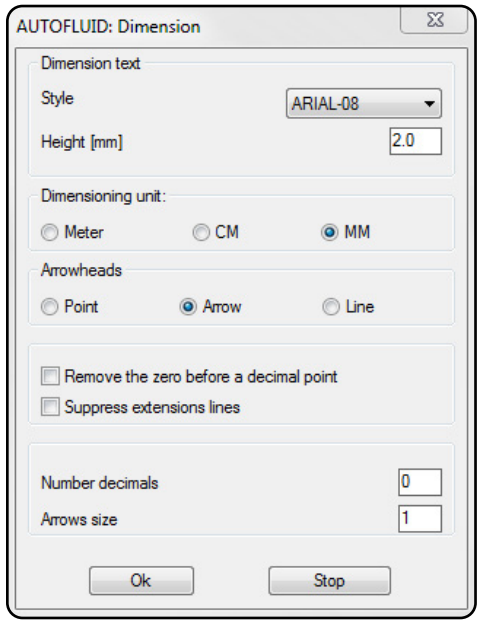
Types of default values and values that can be changed in the preferences panel (or by clicking: Variables) :

- Text style
- Type of text (simple or paragraph)
- Height and justification
- Pipe size format
- Prefixes and suffixes
- Type of frame.

Text variables

Dimensioning





AUTOFLUID manages most of the dimenisioning variables featured in your CAD software.

The following variables can be changed:

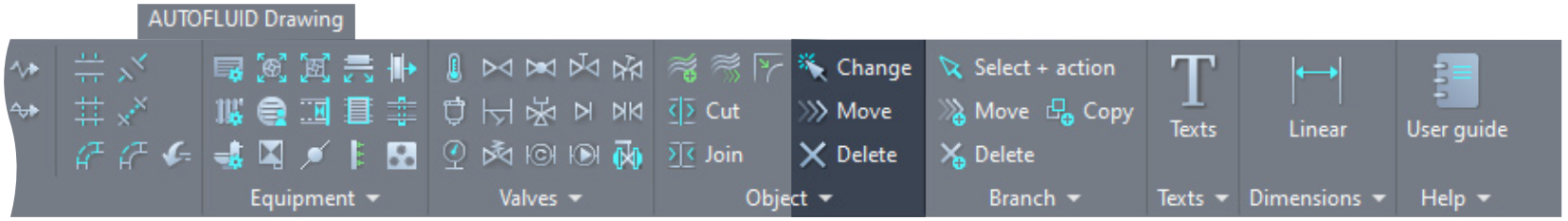
- Style of dimensioning text
- Height of text
- Unit of dimensioning text
- Appearance of tags
- The dimensions of tags
- The zero before a decimal point
- The display of extension lines.

N.B.

To add a new style to the list simply create a new style with the «STYLE» command of your CAD software.
(Height must be 0).

The height of the text will stay the same on screen as on the printed drawing since it is related to the UNIT and the SCALE of the drawing.

Modification commands



Change an object

Move an object

Delete an object

Modify

- Swapping one object with another.
- Changing pipe sizes:
  - On a duct
  - On a part.
- Modify texts:
  - Change text with automatic update of the frame and/or the leader line
  - Move text with automatic update of the lead er line
  - Move a leader line.

Move

- Move an object on a conduit (valve, reducer, insulation...) and adjust surrounding graphics.
- Move a conduit between two objects and adjust surrounding graphics.

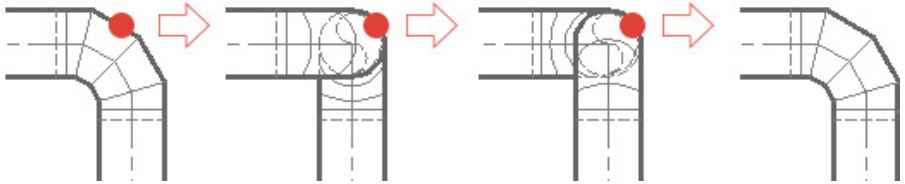
Delete

- Delete an object (elbow, conduit or valve) and adjust surrounding graphics.

Modification commands

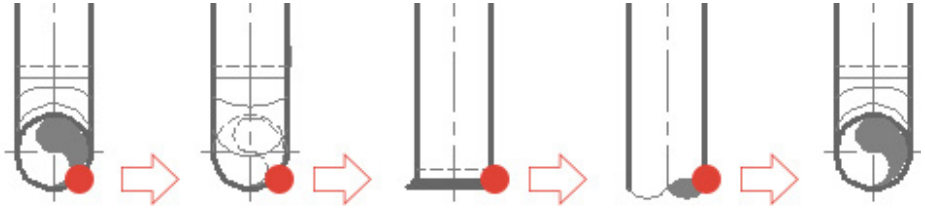
Change an object

swapping one object with another



Click on:	It will change into:
An elbow	A descent
A descent	An ascent
An ascent	An elbow

Other Examples:

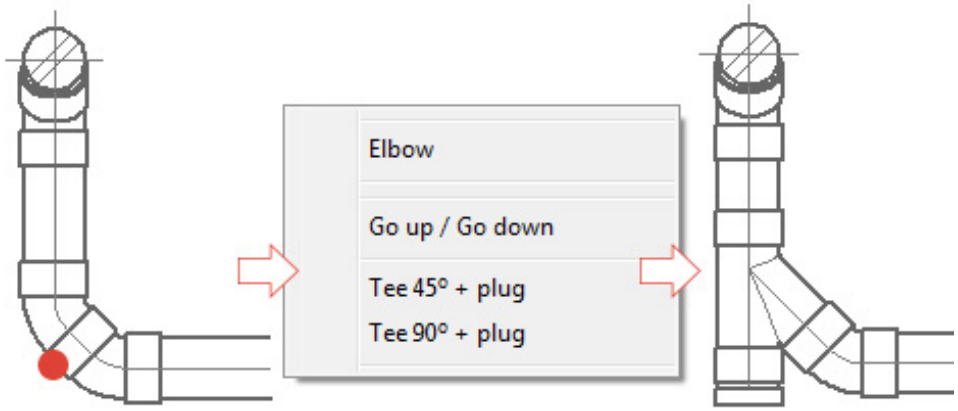


Click on:	It will change into:
An elbow going upward	An elbow going downward
An elbow going downward	A cap
A cap	An end
An end	An elbow going upward

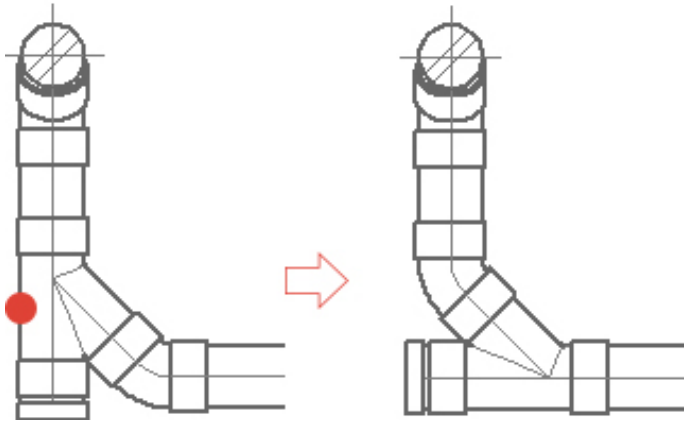
This command works on a wide range of AUTOFLUID objects and on all types of routing commands (circular, rectangular, tube, drain,as well as single line drawings).

Modification commands

It operates in two ways: in a loop as already described or via drop down menus:



In this particular example the angle of the Tee is wrong. Simply click on it to change its direction:



Another example:

Modify 1

Modification commands

Change an object

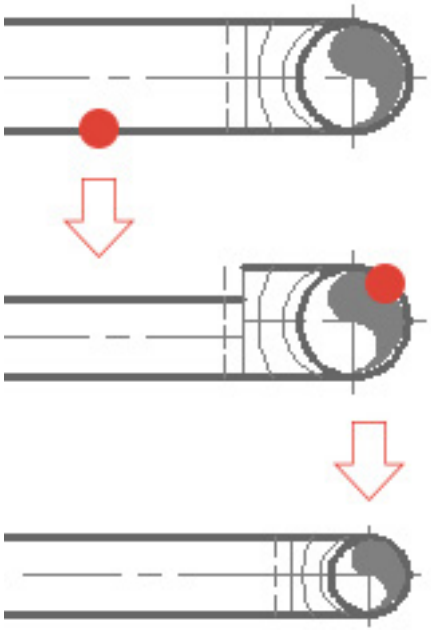
Modify Pipe sizes

Change a pipe sizes directly on the conduit. The parts will then adapt to it.

It is not possible to change the pipe size of a part directly without changing the size of the conduit

1. First click on a conduit, choose the new pipe size and then the modification will take place.
   The clicked line doesn't move. Instead the two others adapt.

2. Then click on the objects adjacent to the already modified conduits to change them.



Modify 2

Modification commands

Change an object

Edit a text

Depending on where you click, suitable actions will be triggered:

Amend the text with the modify command in your CAD software.
The frame and the leader line adjust automatically.

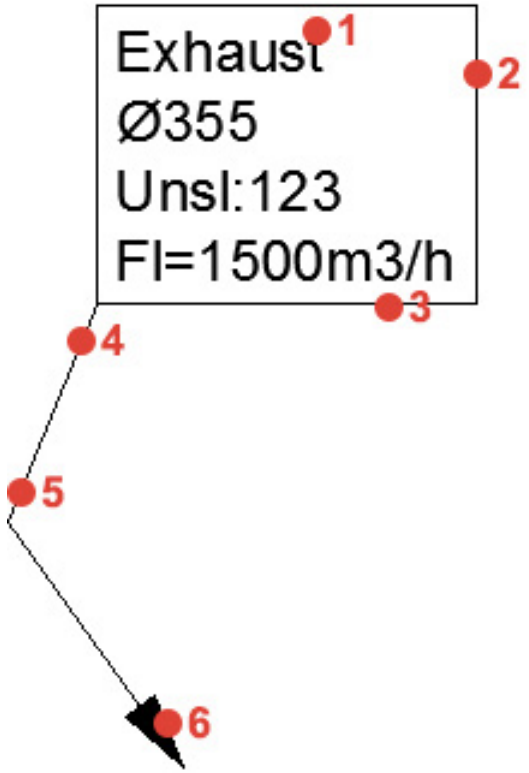
Vertical edge of the frame.
Lets you drag the whole content. (Text, frame, leader line).

Horizontal edge of the frame.
Lets you drag the text and frame.
The leader line adjusts automatically.

Lets you move the start of the leader line without changing the other segments.

Lets you move the intersection of the two segments of the leader line without changing neither the start nor the tip.

Allows you to move the tip of the leader line without changing the other segments.



Modify 3

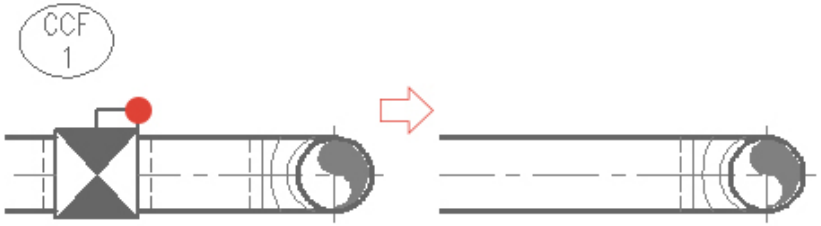
Modification commands

Delete an object

Deleting an object

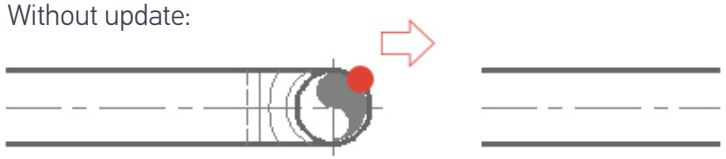
Allows you to delete an AUTOFLUID object by clicking only on one of the entities that form the object. The remaining entities will automatically update.

With update:



Erase all the entities of the valve and then close the conduit.

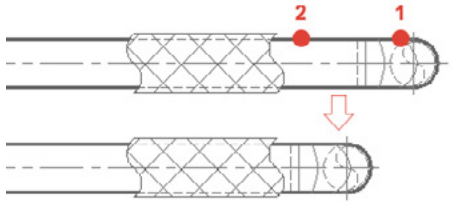
Without update:



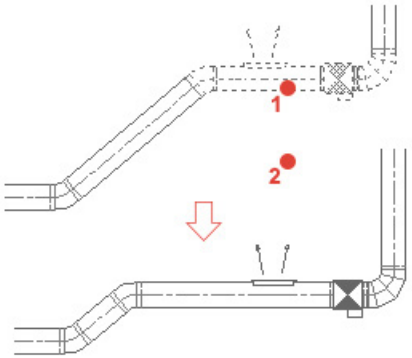
Delete

Move an object

Move an object on a conduit

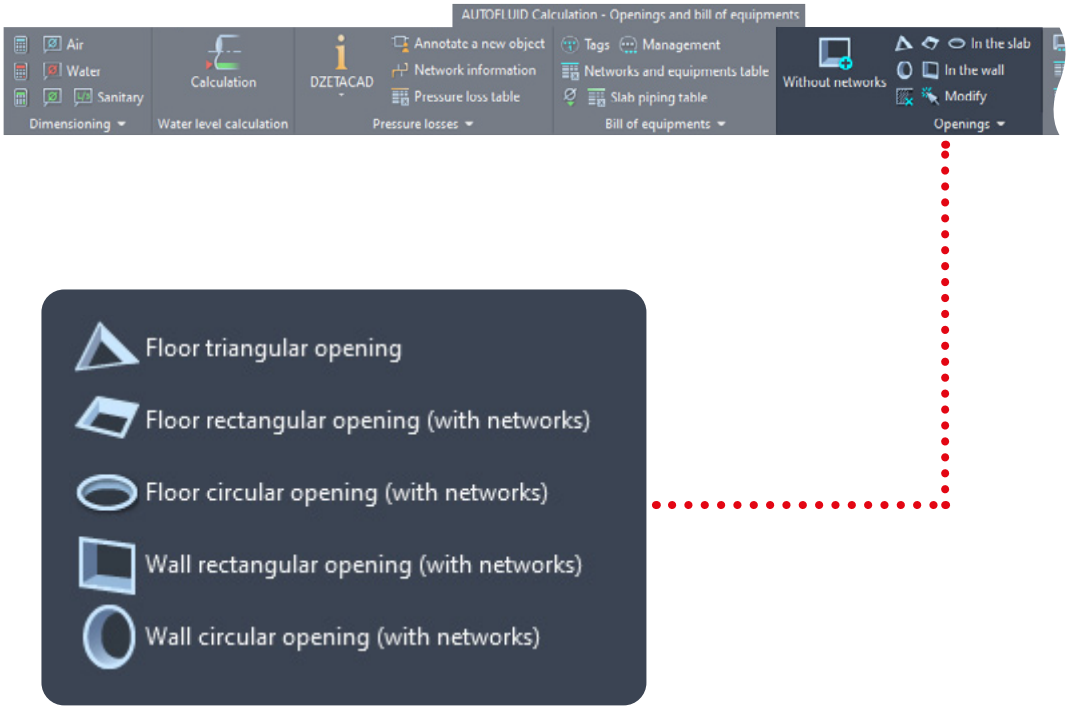


Move a conduit between two objects

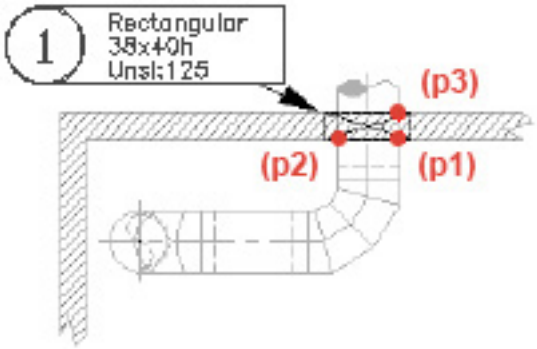


Move on object

Openings



Openings with networks

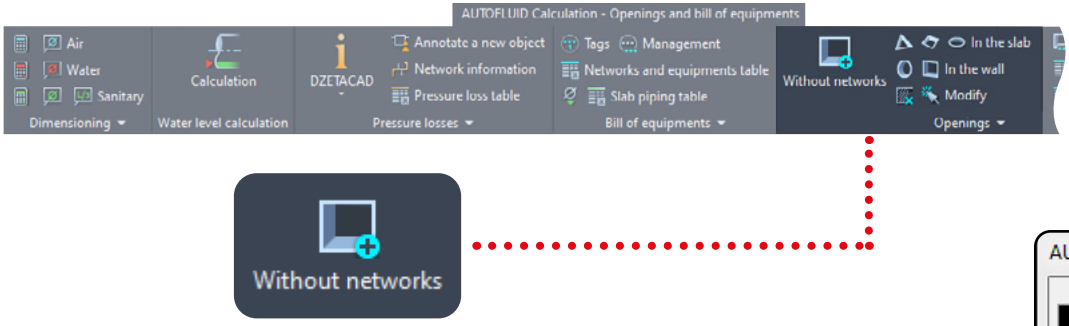


First define the distance between the conduit and the edge of the opening (in the preferences panel). Then simply select 3 points that belong to the conduit and to the wall. The command will automatically calculate the dimensions of the opening, draw it, and tag it.

Tagging allows you to list the openings in an Excel or in a table in your CAD software.

Openings with networks

Openings

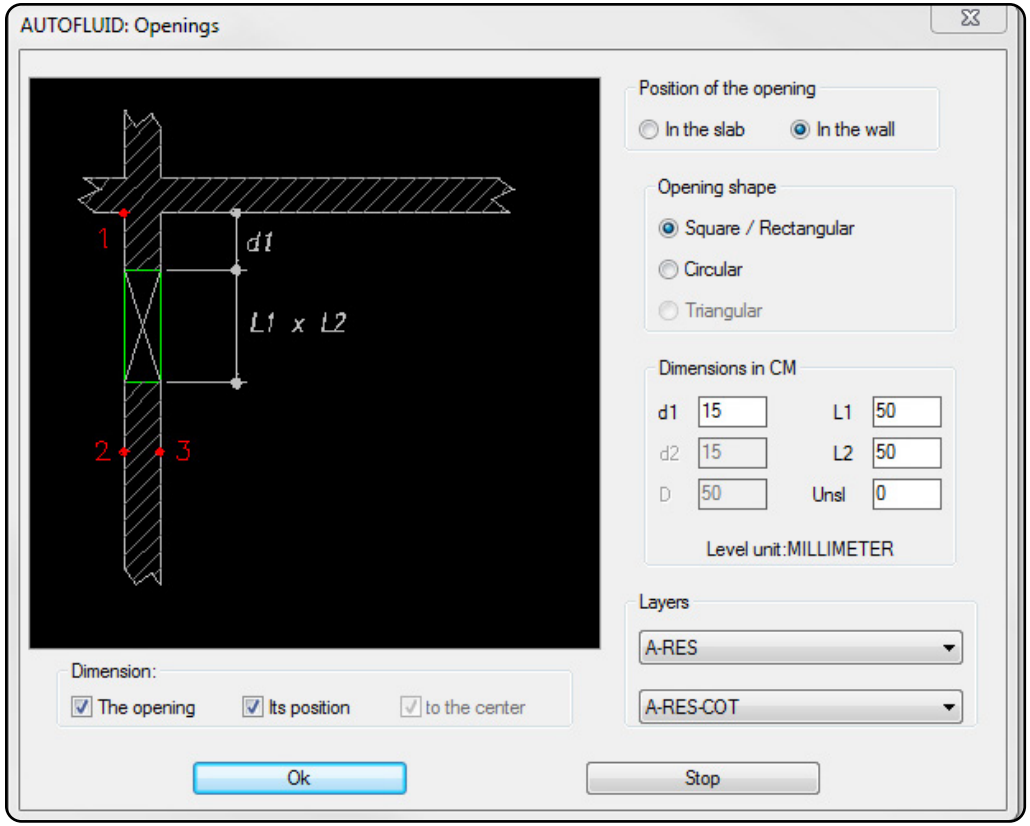


Free openings (without networks)

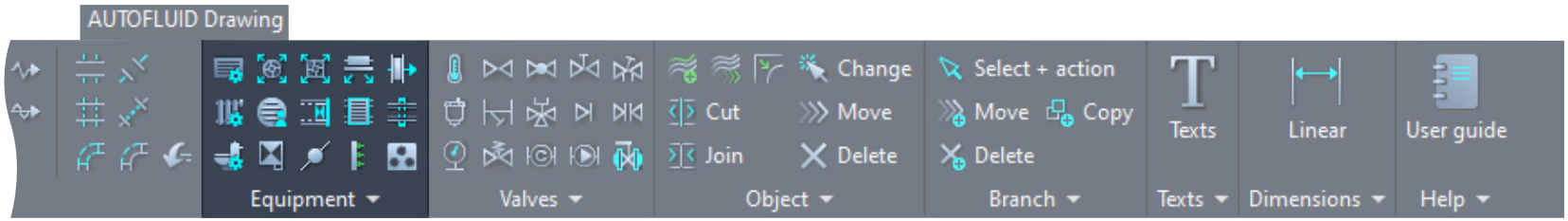
This opening is drafted and tagged in the same way as the 'openings with networks'.

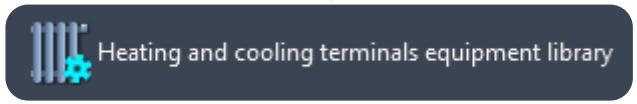
Tagging allows you to list the openings in Excel or in a table in your CAD software.

Free openings



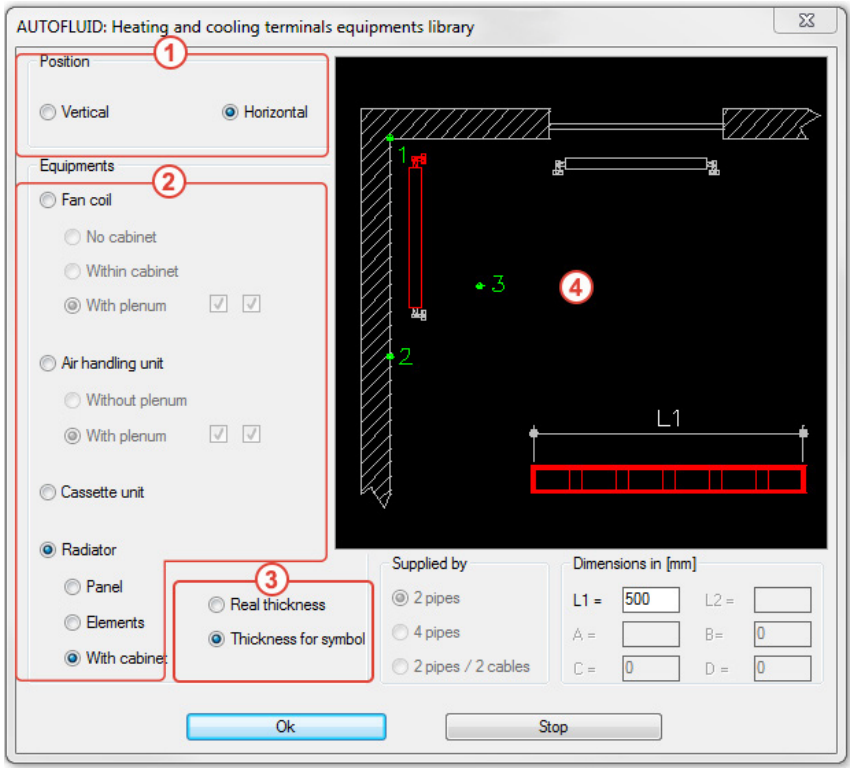
Terminals





1. Indicate the position of the equipment part
2. Choose the type of equipment
3. Choose the desired thickness

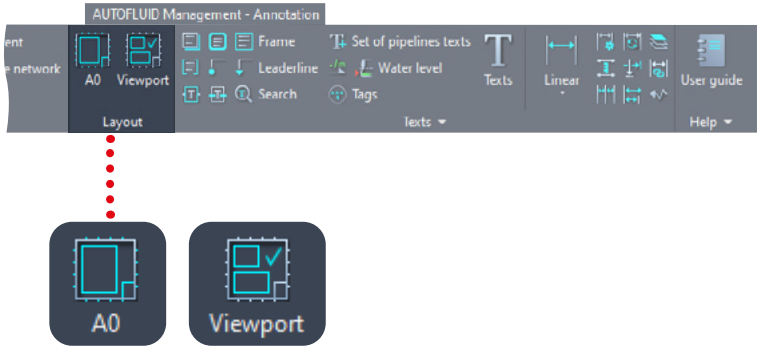
Click on the image to position the element. The element's position is indicated (centered between 2 points or in a corner): this will guide you to place the 3 points correctly.



Terminals

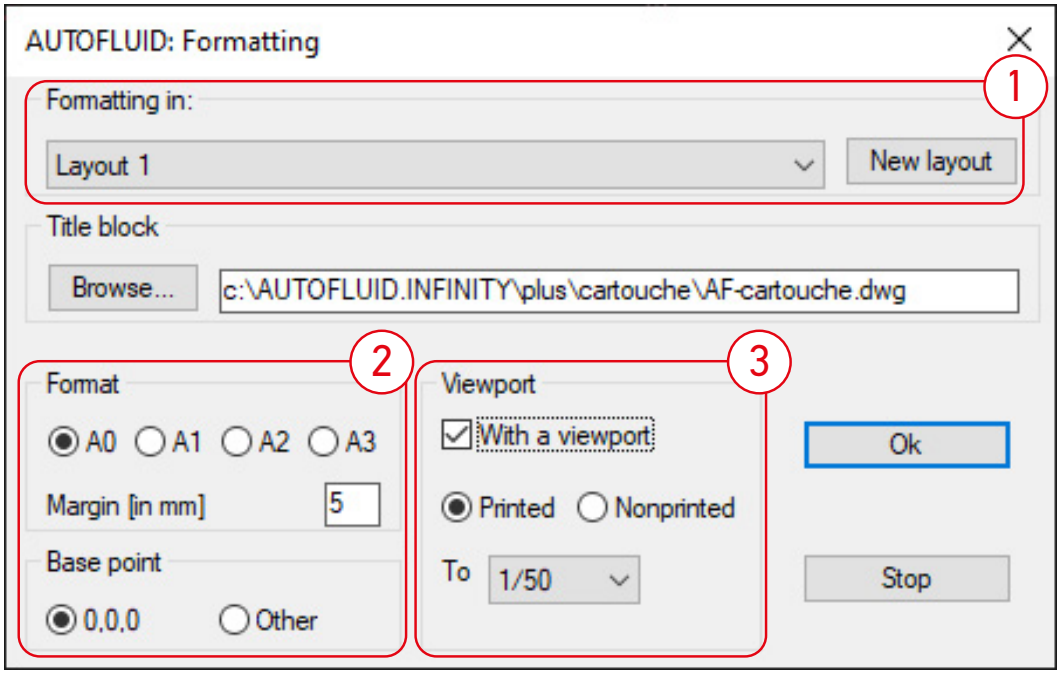
Page layout

1. Choose a presentation or create a new one.
2. Choose a format.
3. Choose 'to print' or 'not print' the window (on a layer that won't print).



You can create a quick layout with the following settings:

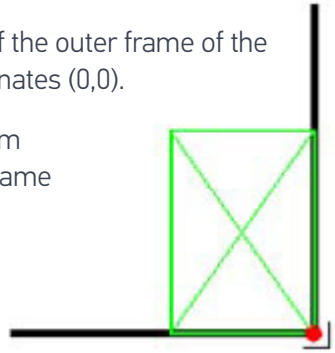
- double frame
- window (automatic scaling)
- title block (if the name is specified).



N.B. The title block has to be made as a block.

The bottom right hand corner of the outer frame of the title block must have the coordinates (0,0).

This point will overlap the bottom right hand corner of the inner frame of the page.



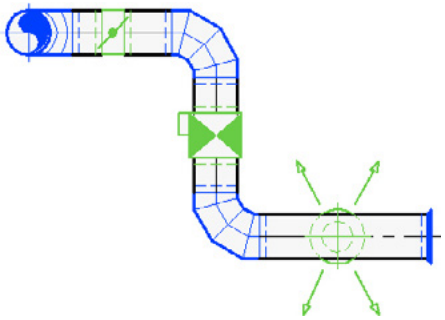
Understanding and exploiting a 2D network

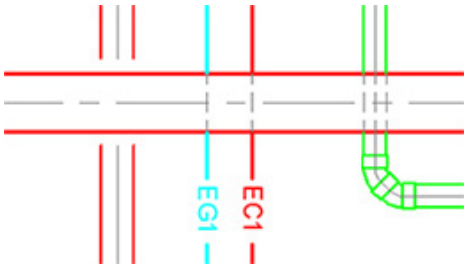
Quality of a 2D network

To ensure AUTOBIM3D functions properly, the 2D network structure must be correct.

See sections :

- INTRODUCTION TO DOUBLE LINE DRAWING
- STRUCTURE OF A CONDUIT
- OPERATIONS ON CONDUITS





N.B.

- The following parts must be drafted with AUTOFLUID c.12 patch version and later to be compatible with AUTOBIM3D or AUTOCOUPE:
  - Vertical dampers
  - Vertical fire dampers
  - Flocking and insulation
  - Tap fittings
  - Sanitary connections.

- All parts must be surrounded by their conduits.
  For example:
  - A Tee must be surrounded by 3 conduits
  - A 'flat' Elbow > 2 conduits, etc...
  - The oblique conduits must be linked to at least 1 horizontal conduit.

Understanding and exploiting a 2D network

Definition of terms

- The network
  It is composed of graphical objects such as conduits, elbows, reducers, connections...

- 'On line' equipment
  These are the small pieces linked to the networks: fire dampers, dampers, grids, tap fittings...
  This type of equipment is 'made' on demand while routing the networks, in order to be drafted quickly and to fit any network sizes.
  It looks graphically simple, or even schematic, however it is sized at the required dimensions. It can be automatically replaced by more detailed graphic blocks if needed.

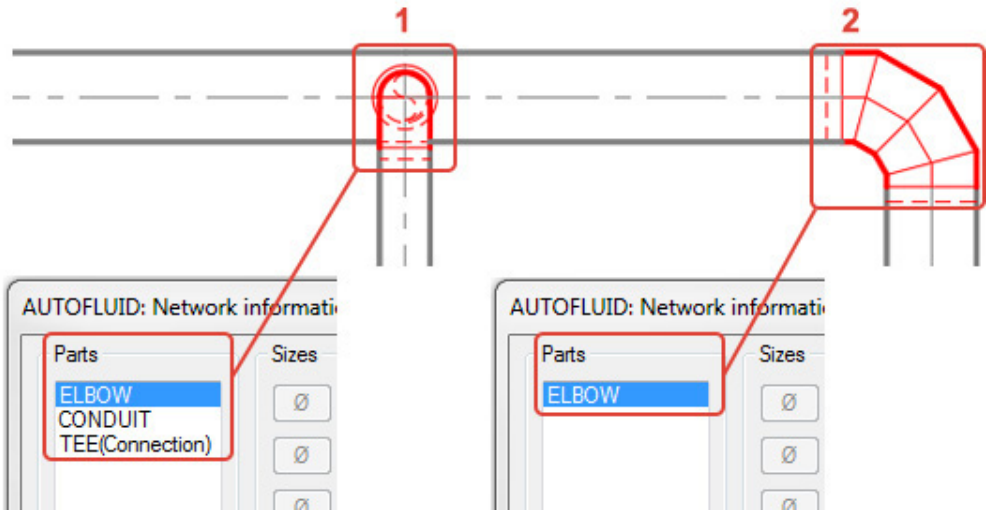
- Main equipment
  Main equipment blocks aren't provided with AUTOFLUID. Many users already have their own block libraries. Nowadays manufacturers themselves provide the needed elements in different formats (DWG, RFA, IFC...) for integration of their pieces into your plans and models.

- Graphical objects
  They represent the network routing.
  For example: a 'flat' elbow, an elbow going 'through upper floor', a connection...
  One graphical object may contain several elements.

Example below:

1. The 'topside connection' graphical object contains 3 elements: ELBOW / CONDUIT / TEE

2. The 'flat elbow' graphical object contains 1element: ELBOW



Elements

See example above.
They contain information that allows:

- Quick modifications of graphical objects
- Pressure drop calculation
- 3D model creation.

Understanding and exploiting a 2D network

- Horizontal conduits — Represented by:

Circular	> 3 lines
Rectangular	> 2 lines
Smoke exhaust rectangular	> 4 lines
Single line	> 1 line (according to the routing command used: line/polyline)

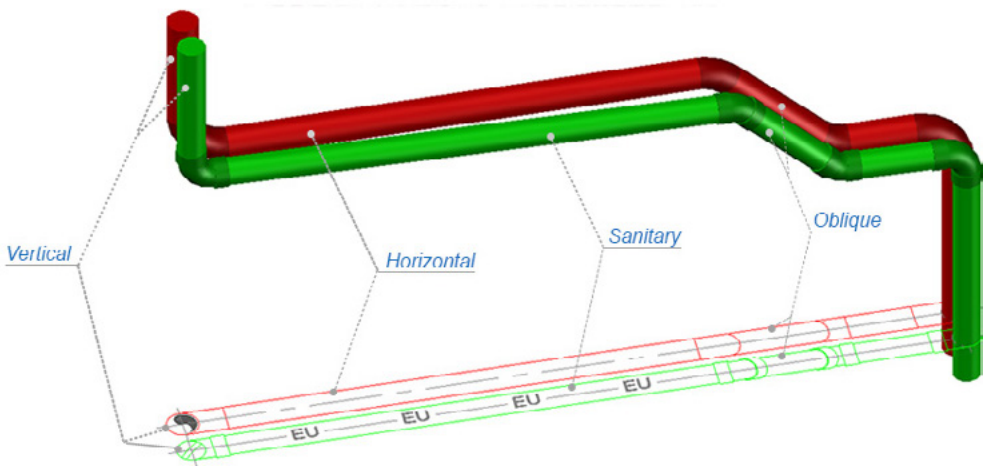
- Vertical conduits — Represented by:

Circular	> 1 circle
Rectangular	> 1 square/1 rectangle
Smoke exhaust rectangular	> 1 doubled square/rectangle
Single line	> 1 circle

- Sanitary conduits (drain)

The drain conduits are by definition slightly oblique.
The drain networks have always been graphically drafted as the horizontal ones.
The slight slope generates ellipses on connection pieces that are so thin that they are ignored (elbows, wye branches...).

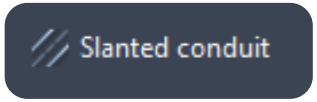
The angle of a slightly sloping conduit needs not be given.



- Slanted conduits

They are represented as the horizontal ones plus the angle information.

There are 2 possibilities to draw slanted conduits:
- Using the «Break» one off commands or routing commands option
- Transforming a horizontal conduit



In order to create them using an already existing conduit, use the «SLANTED CONDUIT» command.

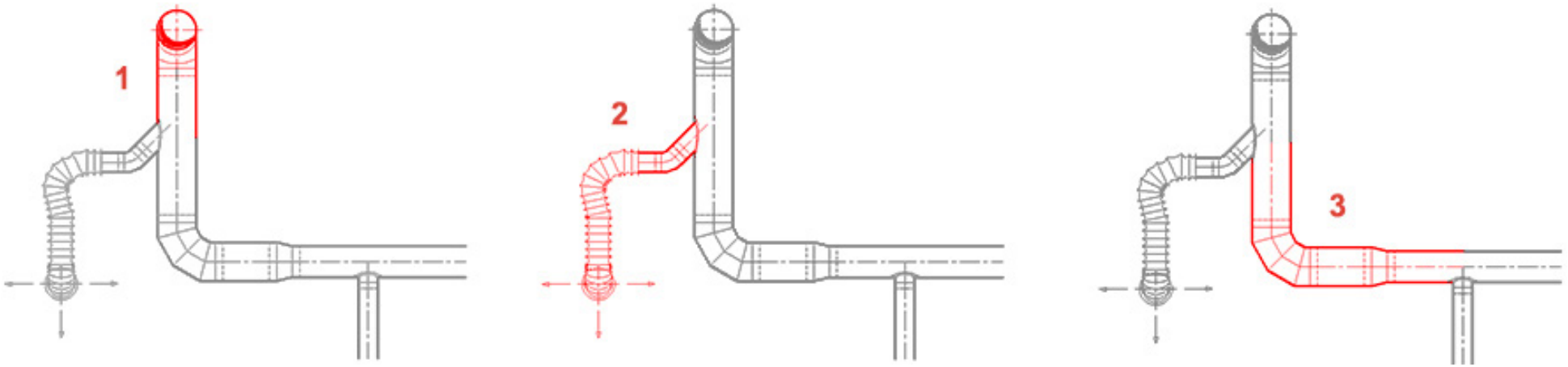
Then, if needed, adapt the adjacent pieces with the command «modify 1 object» (See Exercise : Create 3D).

Understanding and exploiting a 2D network

- Branches

  A branch is a segment in a network. It is defined by its extremities: a diversion, a terminal or a level breakage.
  Example: a Tee to a Grid, a Wye branch to an Elbow 'Through lower floor'.

  In the example drawing below, 3 branches can be identified:



- Spread

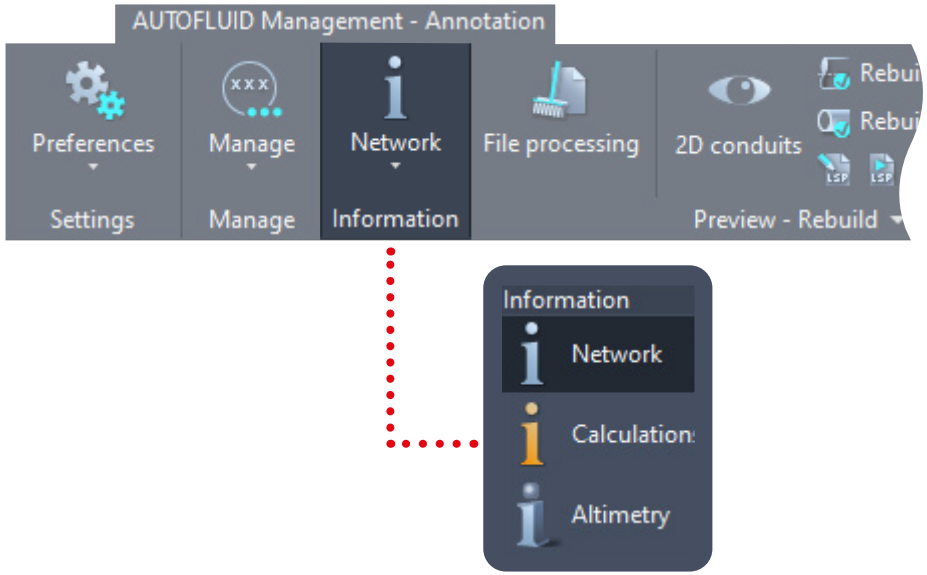
  It means giving an information to a graphical object and then spread this information to all the branch's graphical objects.
  Each branch must be annotated: the information does not automatically go from one branch to another.

Understanding and exploiting a 2D network

Information and texts

Be careful to differentiate the «TEXT» command from the «Information» commands.

The «Information» commands allow you to read or assign some information to the elements composing the «graphical objects» .
The «TEXT» command lets you retrieve information and allows you to complete it with more text. Texts can be framed and/or can come with a leader line.



The «Information» commands:

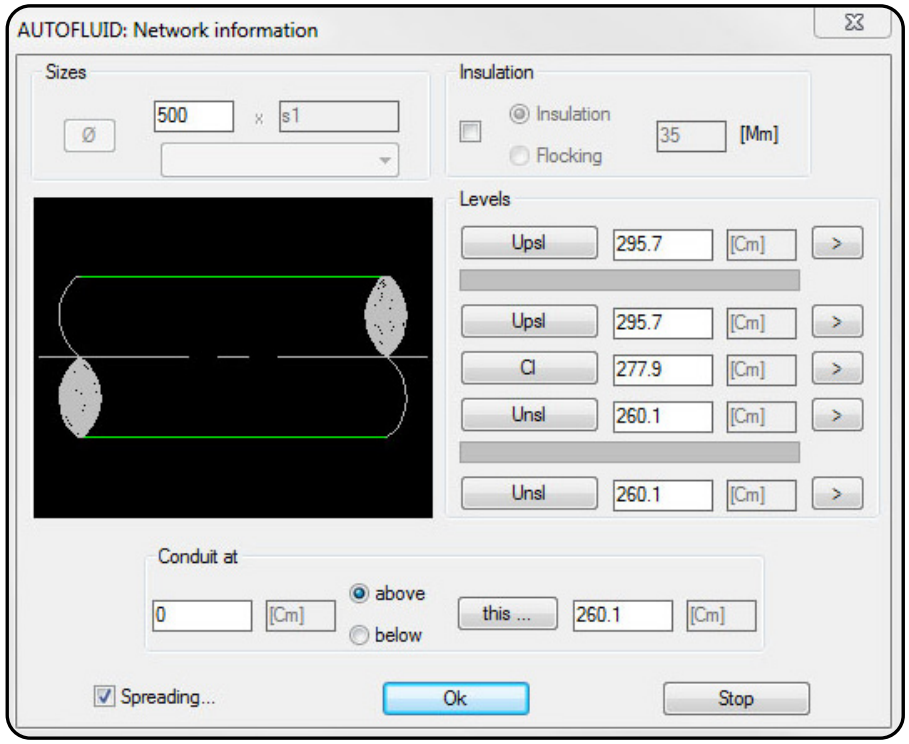
3 commands will allow you to get some information or assign some information to a network:

These 4 dialog boxes are adapted to the corresponding modules:
As far as DZETACAD is concerned, the main information is Flow, whereas with AUTOCOUPE and AUTOBIM3D, it is the Level and the Water level that matter the most.

A value information that was confirmed in one of the dialog boxes is valid for the others.

Specifying 2D networks levels

Specifying levels



Altimetry

The levels are given branch by branch.

The selected branches levels can be quickly specified, either by typing the level value directly, or by retrieving the level from another conduit.

The level information must mainly be given to the horizontal conduits. The levels of the other graphical objects will automatically derive from the adjacent conduits level.
Example: spreading the level along a network that contains a level breakage.

The missing or false levels may be automatically replaced by levels allowing a correct 3D modeling.

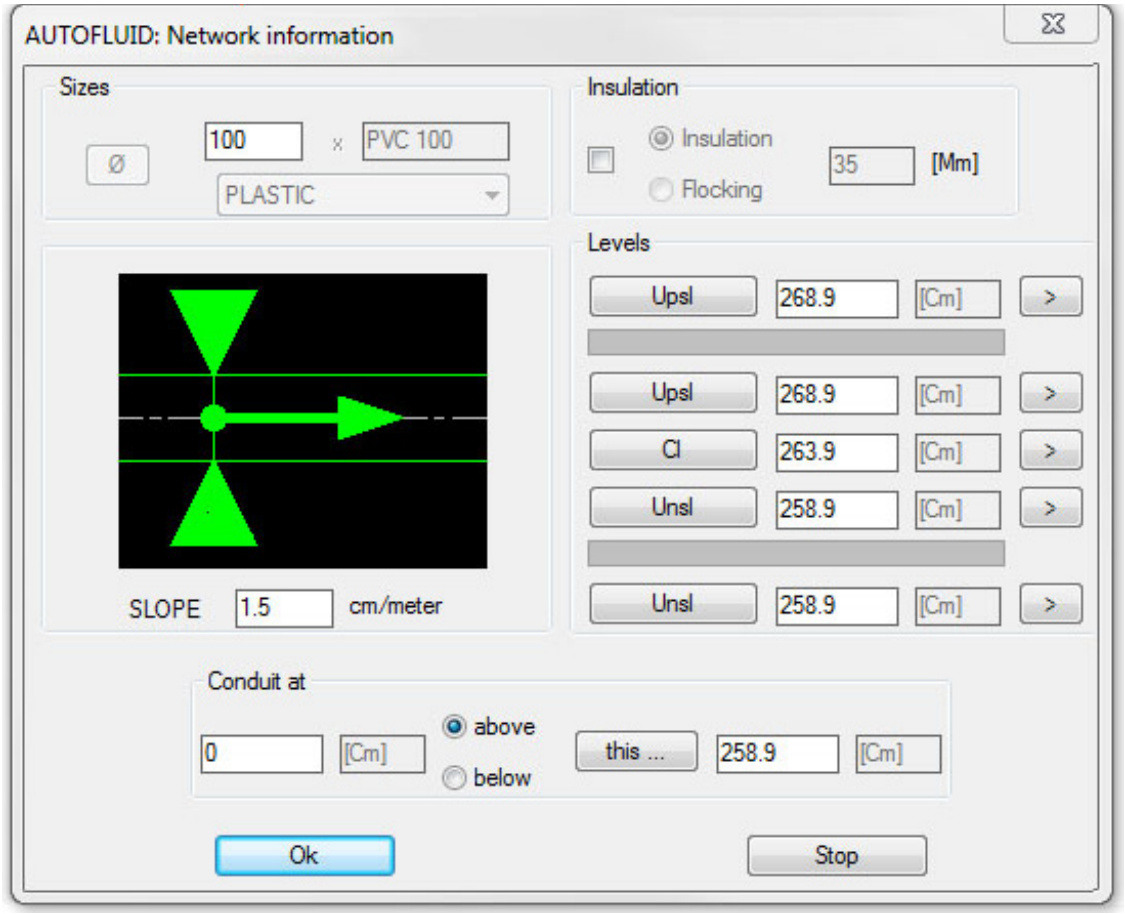
The 2D insulation drafted (Insulation / Flocking) will not generate a 3D insulation.

Check the « INSULATION » box if you wish to integrate it into your 3D model.

Specifying levels

Specifying 2D networks levels

Specifying the 2D network water levels



Altimetry

In order to assign the levels to sanitary conduits, run the same « NETWORK INFORMATION » command as for the other networks.

Using this command, you will quiclkly specify the water levels:

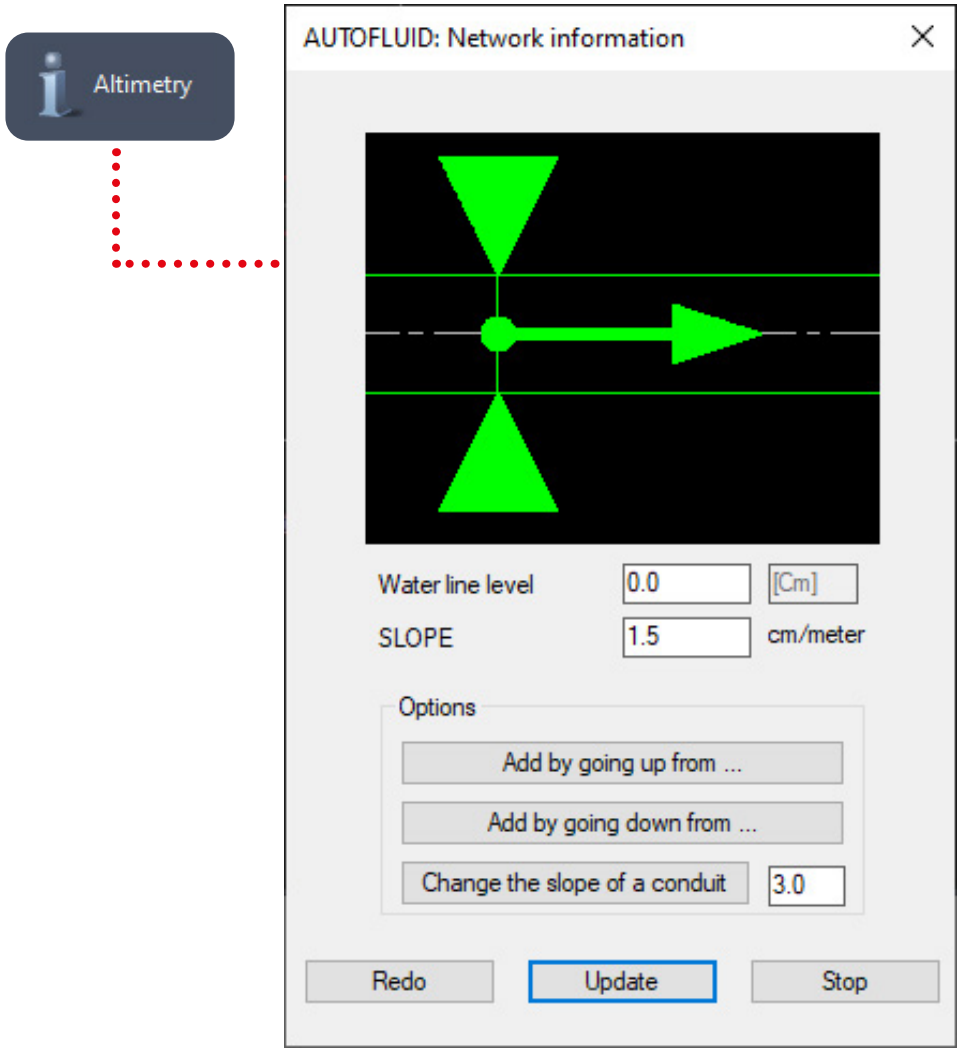
Choose the starting point for the water level and type its value as well as the slope value, then click progressively downwards or upwards along the branch. The adjacent water levels are thus automatically calculated.

The water level settings of a network can be modified. See section «Changing water level settings».

Specifying the water levels

Specifying 2D networks levels

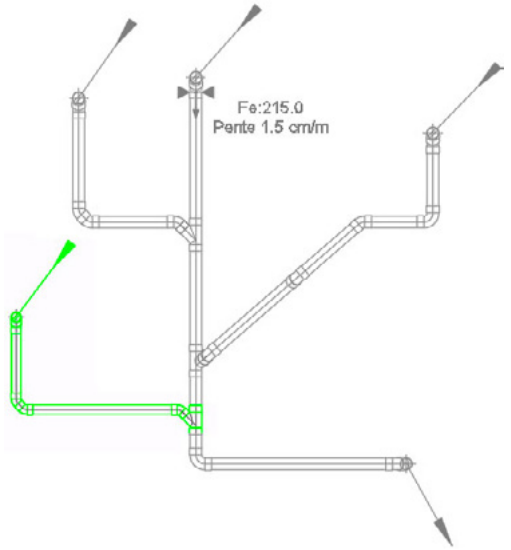
Changing water level settings of a network



When network graphics are modified, corresponding water level settings can also be changed.

Click on the logo/text of the water level in order to:

- Redefine the starting point of the water level
- Redefine the slope
- Add/delete a branch
- Redefine the slope of a segment.



Water level modification

Specifying floor levels

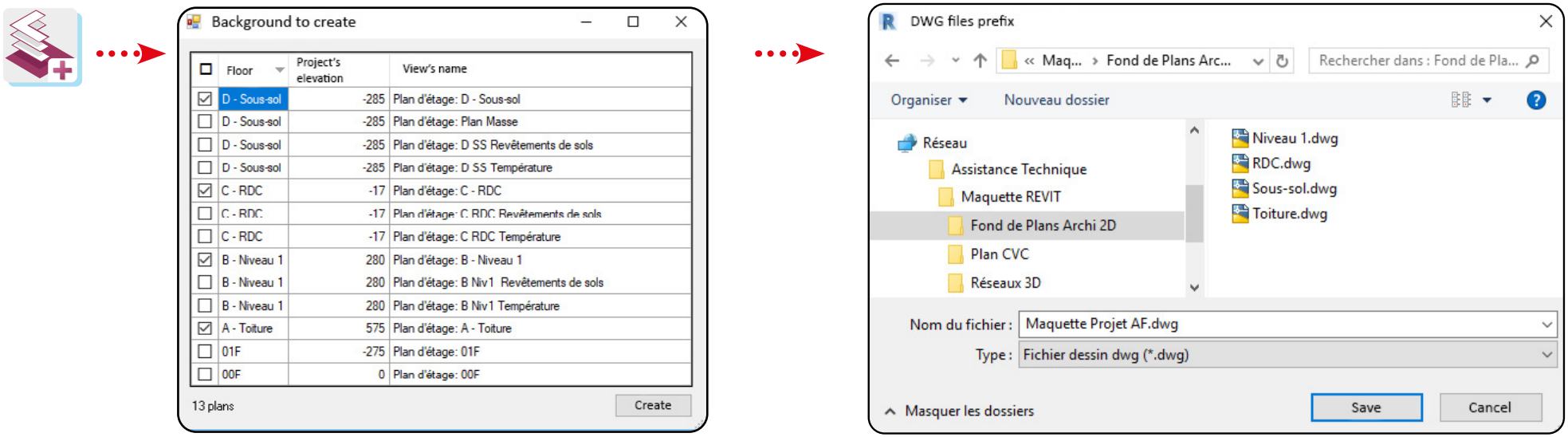
If the architect provides a REVIT model

Thanks to the RVT-Connect plug-in, it is possible to read and extract the information of each level in the architect model.

Step n°1: Create DWG backgrounds

From the model, you can generate 2D plans of each floors then integrate their level into AUTOFLUID. With the AUTOFLUID toolbar in REVIT, ensure the compatibility between your CAD software to the model.

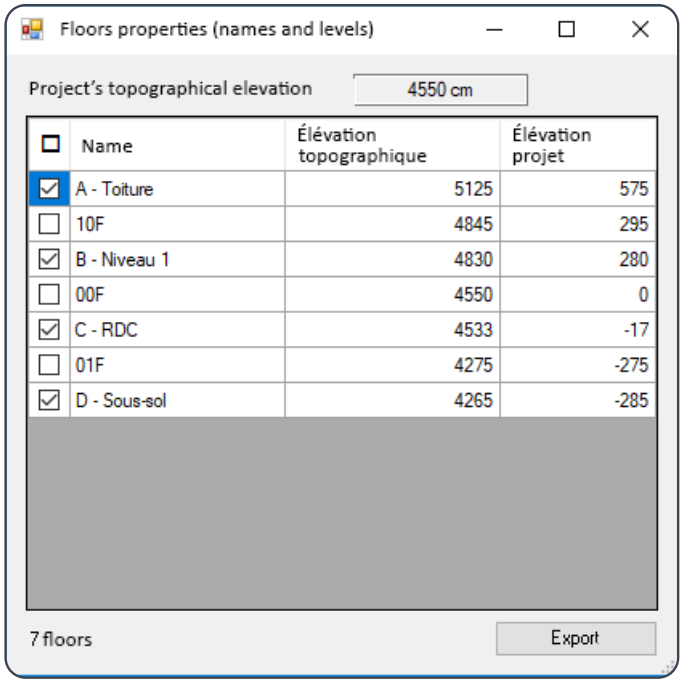
Click on 'Create DWG' and select the 2D plans you wish to extract. Click on 'Save' and select a destination directory.

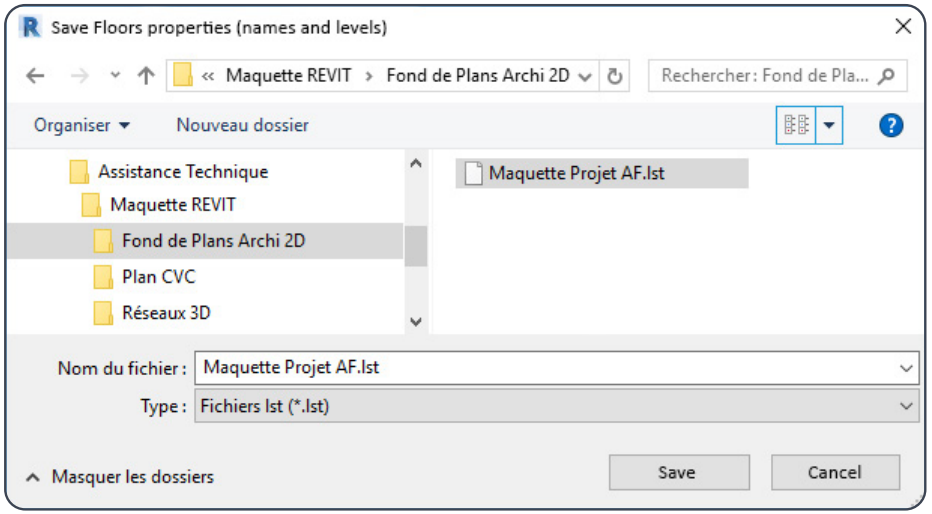


Specifying floor levels

Step n°2 : Extract floor properties

This tool lets you export all the informations about the levels in a .LST file. Click on 'Floors properties', select the levels. Click on 'Export' then choose a folder.

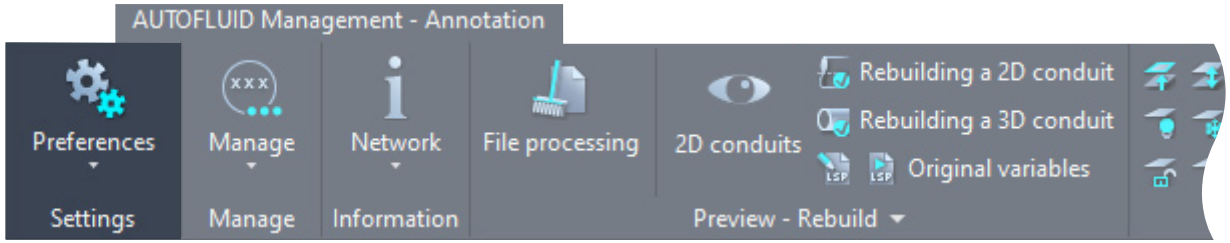


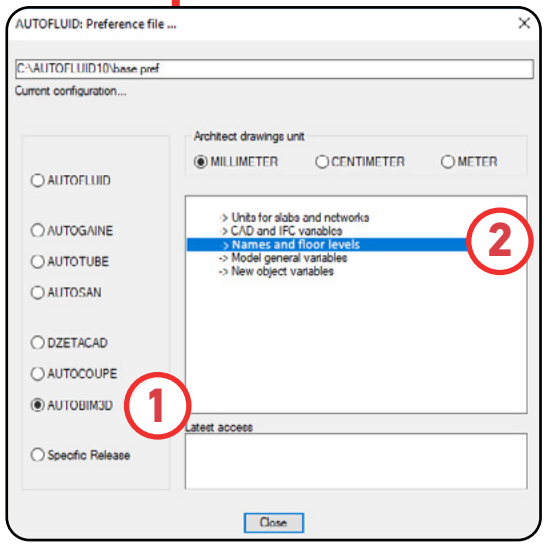


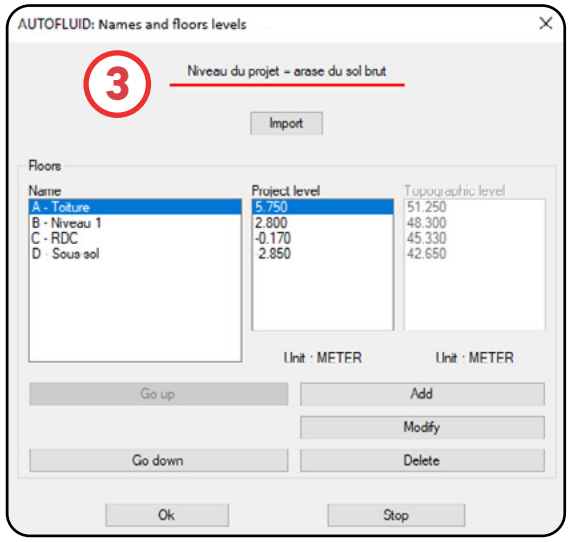
Specifying floor levels

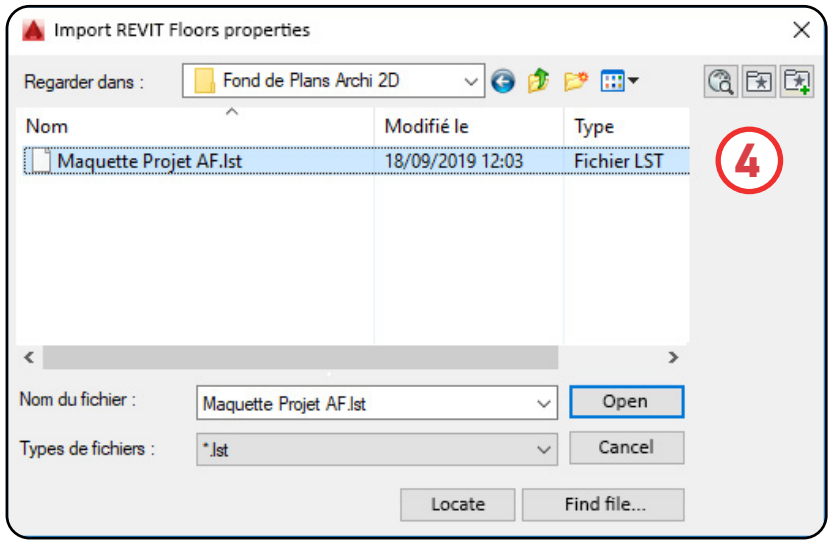
Step n°3 : Import the floor properties

Open your CAD program and then import the LST file, that your created previously in REVIT, into your AUTOFLUID preferences :









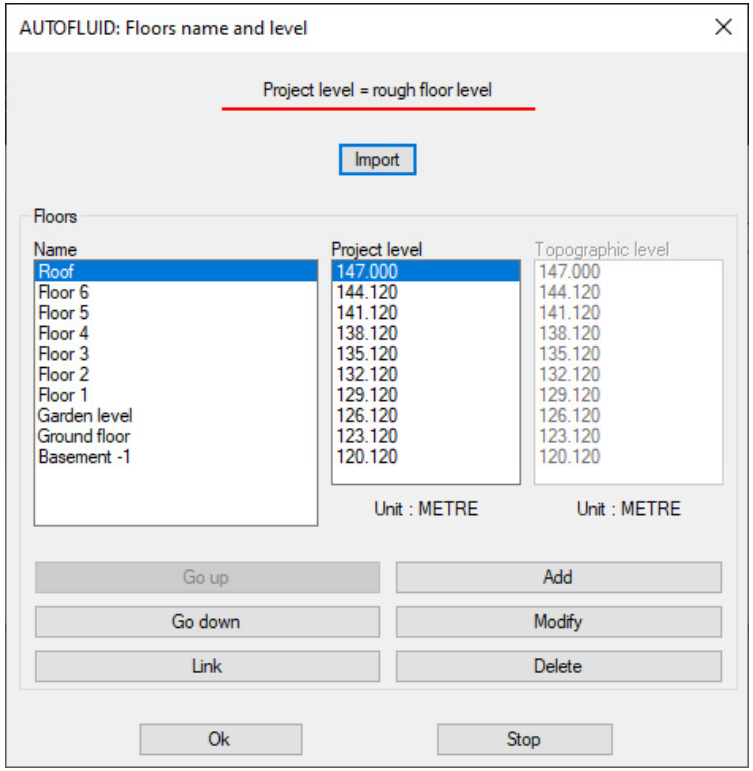
A connection with the model is created. Use the 2D drawing for design background, as usual: import it as a Xref (recommended), as a block or by opening it.

If needed, you can clean it up using file treatment (see section «Treatment of architectural files») as you would on a regular architectural plan. Now you can design your networks using all the commands of AUTOFLUID.

Specifying floor levels

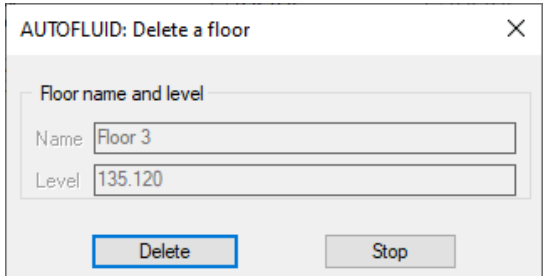
If the architect provides 2D plans

You can specify the structure of the building in this dialog box. Since you're working from a series of 2D plans rather than from a 3D model, RVT-Connect won't compile the floors information from REVIT for you. Therefore you will have to specify it yourself.

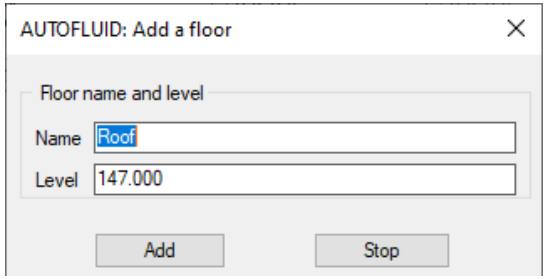


Specifying floor levels

1. To delete a redundant floors: select the floor, click «delete» and confirm



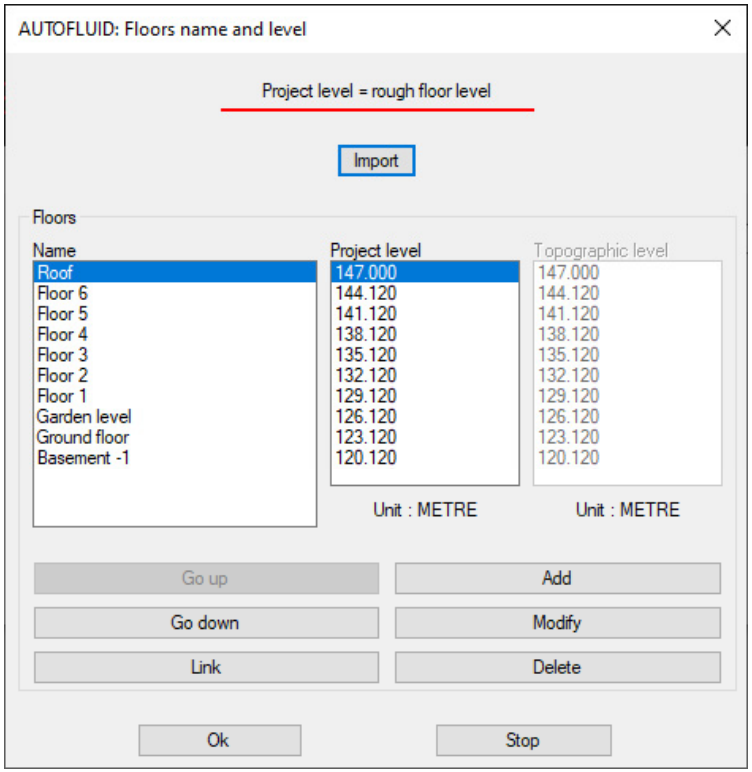
2. Add the floors of your building: click on «add» in the «Floors name and level» dialog box.



- Insert the floor name
- Insert the floor height level (the rough floor level)
- Then click on «Add» - your floor appears in the list.
- Repeat the operation for each floor.

3. In the «Floors name and level» dialog box, you can use the «up» and «down» buttons to position your floors in the right order.

You can also «Link» two floors such as in the case of a mezzanine. For example: your floor 5 would be positionned directly above floor 3 since floor 4 is a mezzanine, which isn't a complete floor.

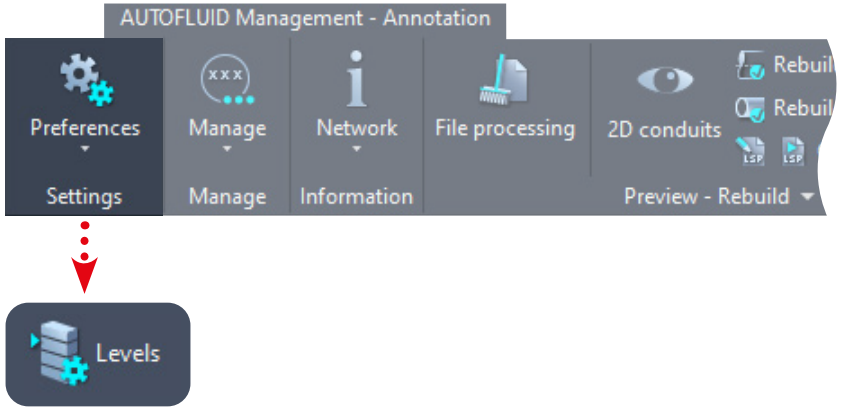


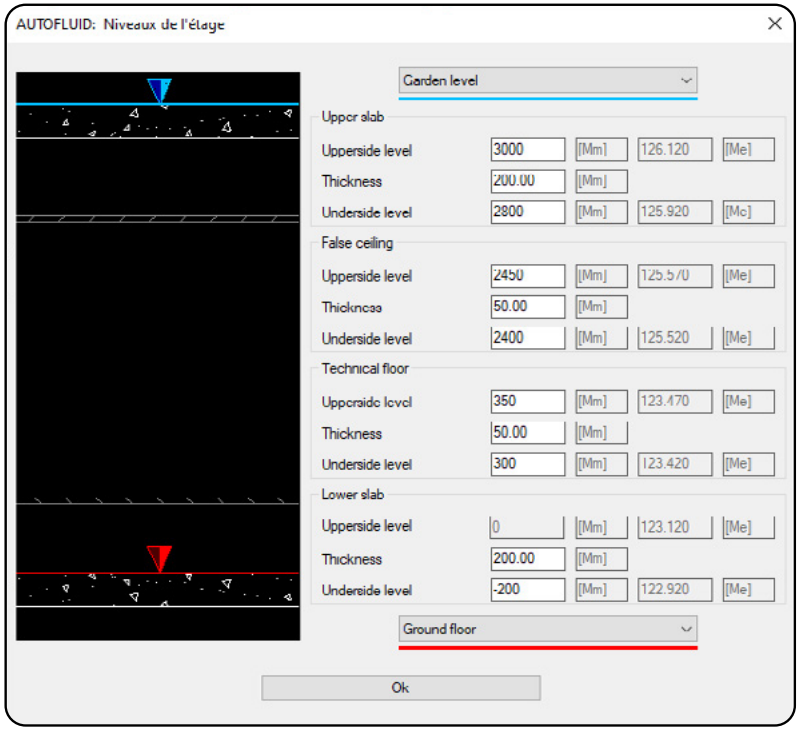
Specifying floor levels

Level and thickness of floor elements

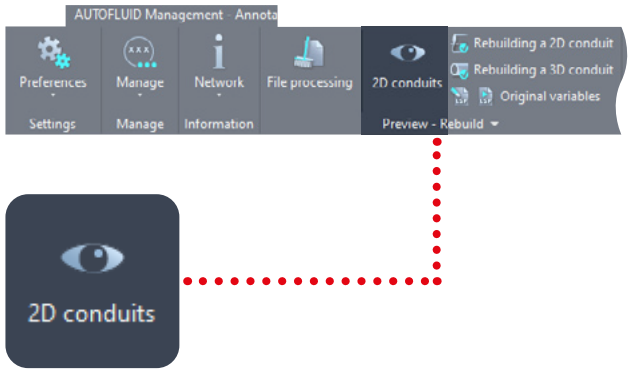
ONce your list of floors is specified, you can adjust the thickness of each floor and the level of suspended ceiling and raised floors.

This operation should be performed using the «Levels» command rather than in the preferences.





Calculation of pressure losses



Step n°1 : Check the connections

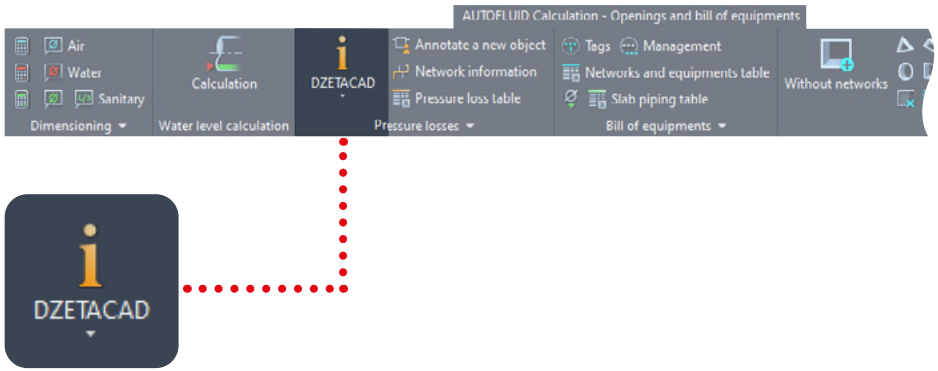
1. Save your fil under a new name

2. Isolate the network that you would like to compute

- Check the connections:
- Check the conduits (colored white).
  If they appear in red then re-build them.
- Rebuild the conduits that might have been cut for display reasons.
- Delete flocking and insulation...

General principle

Inform tees, grids...



Step n°2 : Inform the network

After having drafted the network you must add any missing information.

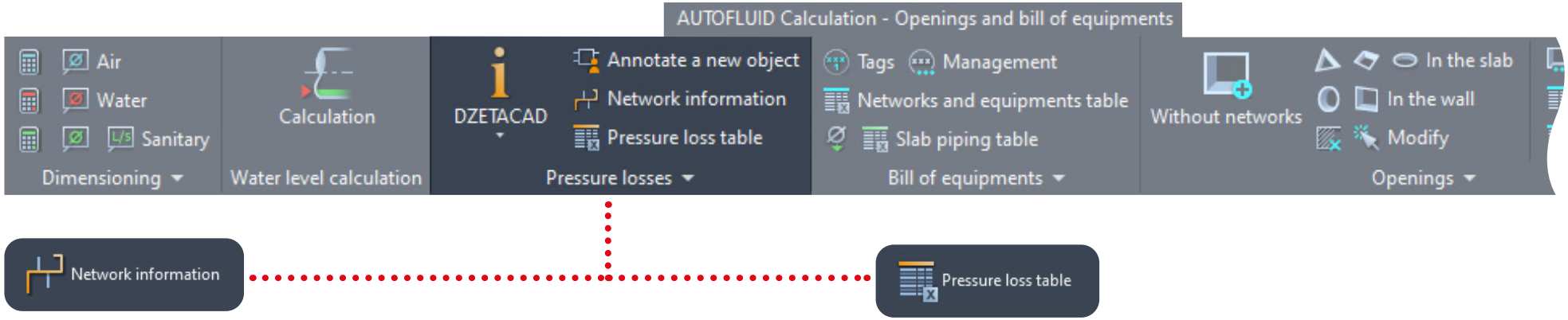
1. Flow rate: click on any element of a branch (duct, elbow,...). BEFORE clicking on an intersection (tee, cross, wye) all adjacent branches must be informed.

2. Flow direction: in intersections, reductions, transformations.

3. Vertical duct lengths: for descents, through floors, or in topside and under-side connections...

4. The manufacturer data for the equipment: valves, dampers ...

Calculation of pressure losses



Step n°3 : Indicate the parts of the network to calculate

1. Select the network branch by branch from the terminal to the source or in the opposite direction.
2. A check up table lets you highlight any missing part in the previous step.
3. You can name your network in this table.
4. The described network will be redrawn in a presentation with the same name.

Step n°4 : Export to Excel

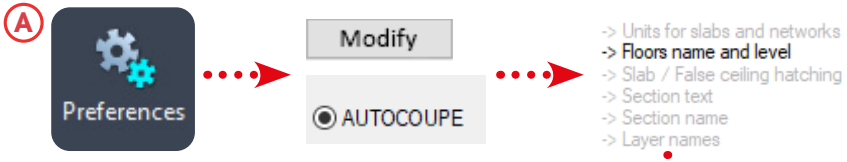
1. Export the calculations to Excel from the presentation. If Excel requests that you activate macros when it starts automatically, please do so.
2. The macro will run an automatic formatting of the table.
3. You will then be able to edit all the parts of this file as if you had created it yourself.

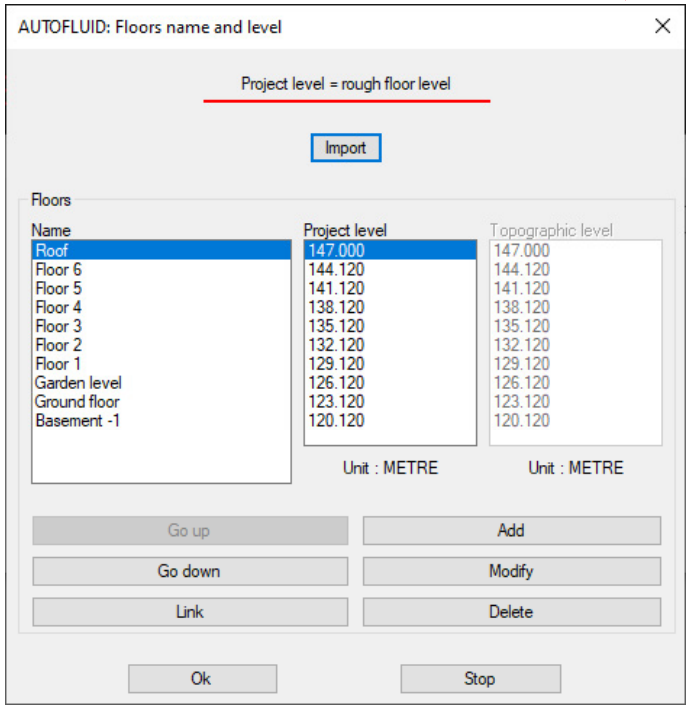
Limits of the software (tips to get around them)

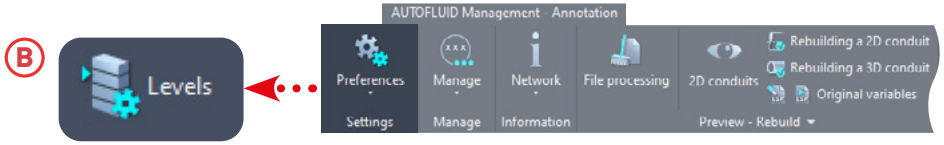
Generating automatic cross sections

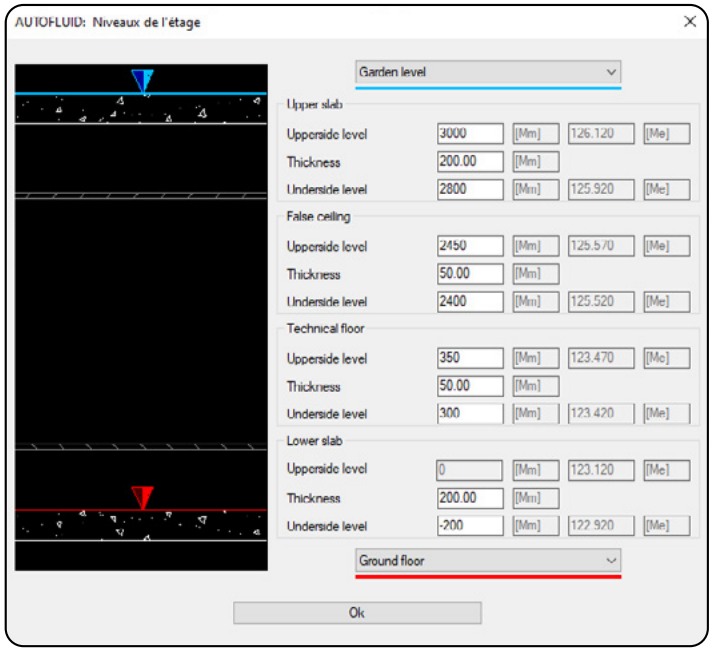
1. Check the altimetry settings of the floor level

To check an correct the altimetry settings of a floor level, open the «Name and level of floors» window. Open the preference pane (A) and then click on «Floor level» (B).

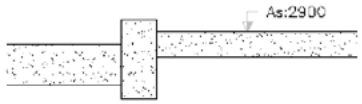








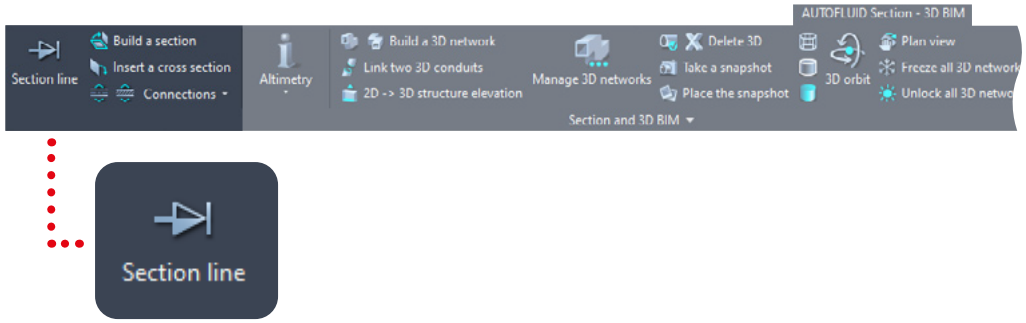
Thickness and level must be specified for each object. If your project features several levels, indicate the current one. You can adjust the others later.



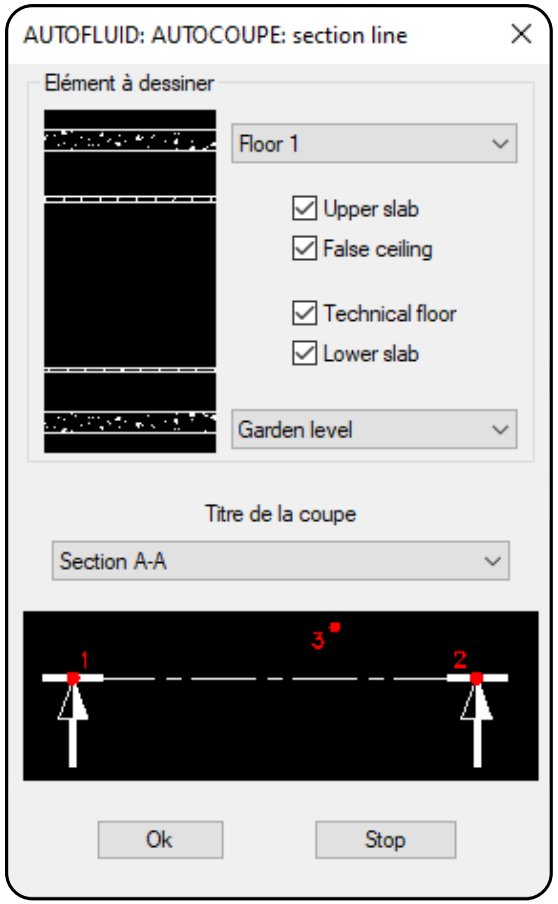
If you have neither technical floor nor suspended ceiling, you can ignore these boxes.

Generating automatic cross sections

2. Indicate and place the cutting plane



1. Choose the architectural objects that you would like to see in your cross section.
2. Choose a name for your cross section
3. Click ok and position the 3 points.



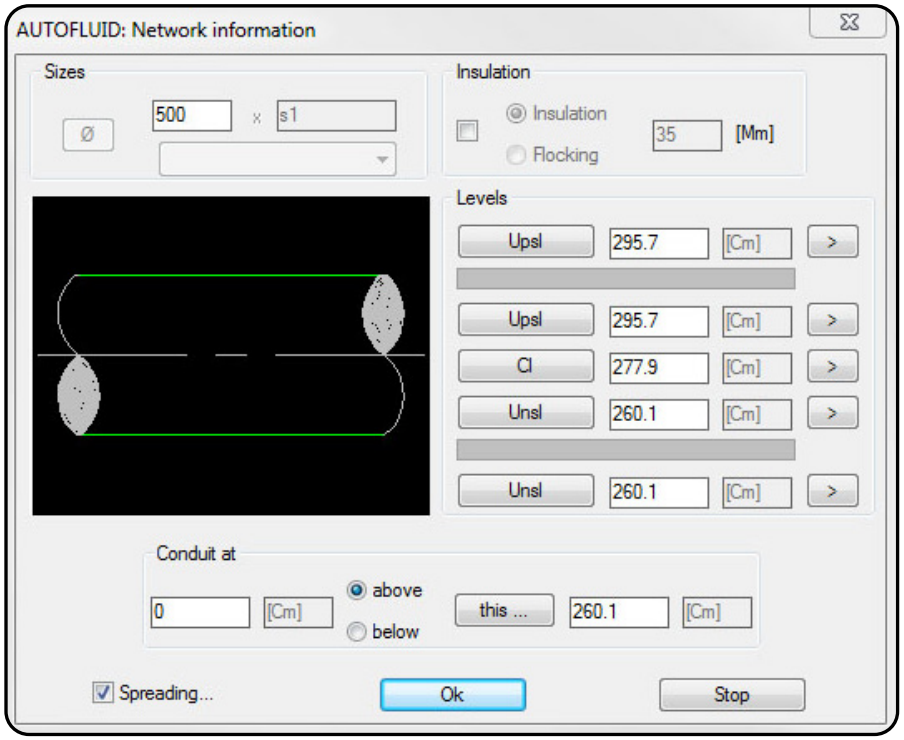
Generating automatic cross sections

3. Specify the level of the networks in the plan view

Use this command to specify the network level 

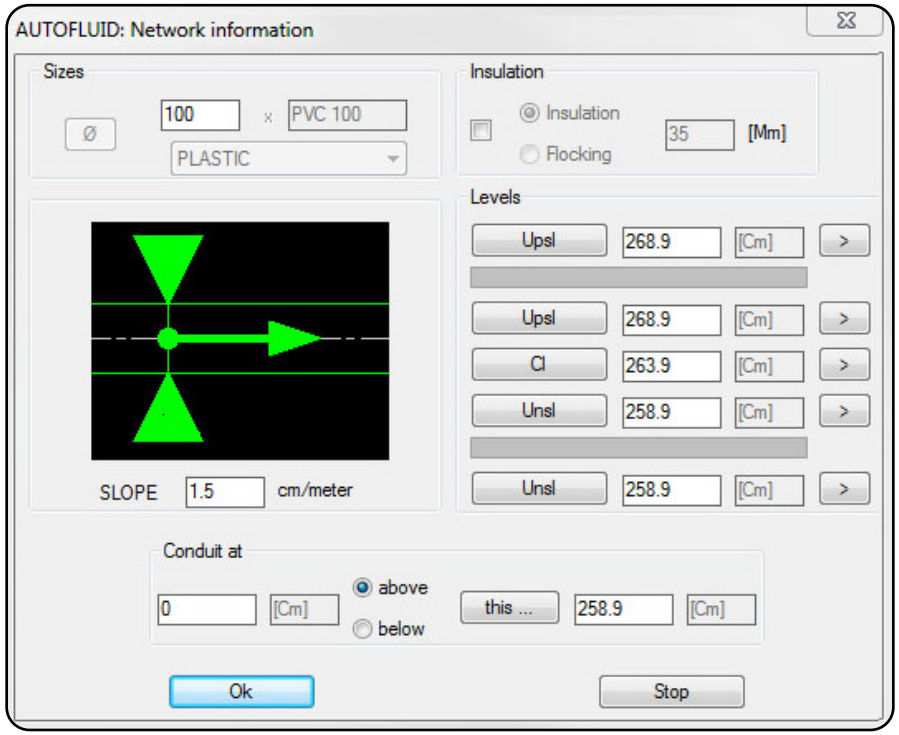
Select a conduit and, depending on the type of network, one of the two dialog boxes below will open.

For an air or water network



Find out more here : Specifying levels

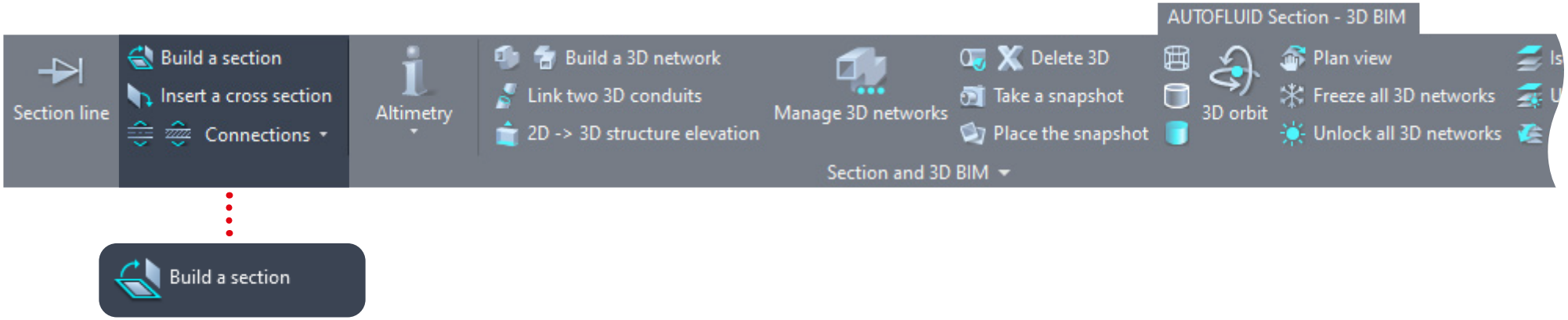
For a drain network



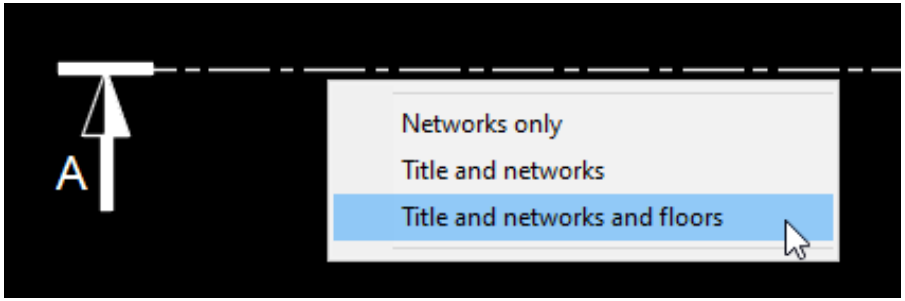
Find out more here : Specifying the 2D network water levels

Generating automatic cross sections

4. Generate and place a cross section

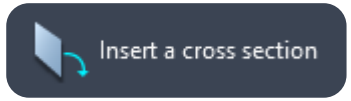


1- Select the cutting line and choose which objects you would like to display.



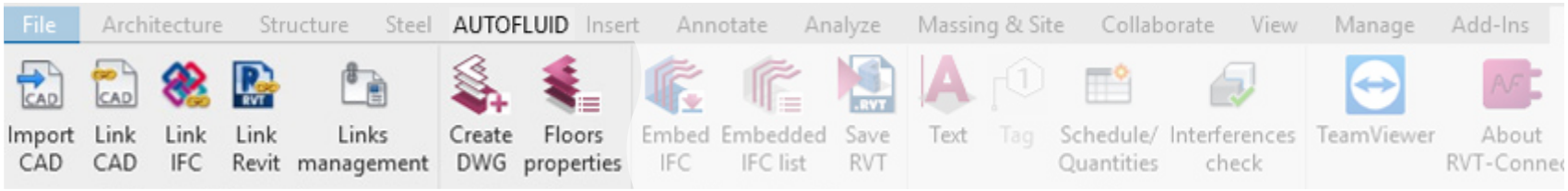
2- Select the relevent 2D networks in the plan view and confirm. When the cross section appears, choose between two possibilities :

- Position it directly where you want it to be displayed in the layout.
- Press «esc» and position it later (in a differernt UCS for example) by using the «Insert Cross Section» command:



RVT-Connect : description

The RVT-Connect ribbon in REVIT



Import a DWG 3D duct network into a current model. It will be imported as a 'block' which cannot be modified inside the current model.

Link a .DWG 3D duct network into the current model. It will be embedded as an 'external reference', which cannot be modified. However, it will automatically update if the linked .IFC is modified.

Link a .IFC 3D duct network into the current model. It will be embedded as an 'external reference', which cannot be modified. However, it will automatically update if the linked .IFC file is modified.

Link a .RVT 3D duct network into the current model. It will be embedded as an 'external reference', which cannot be modified. However, it will automatically update if the linked .RVT file is modified.

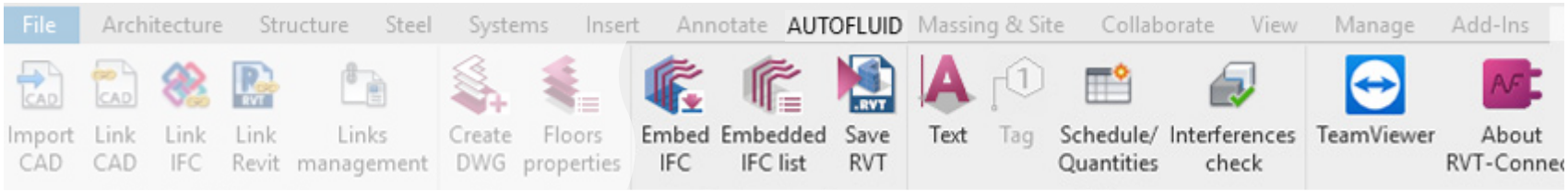
Manage external linked or imported files (.DWG - .IFC - .RVT - ...)

Generate 2D plans from the model. This command lists all the model views and levels. Select the views to generate and then choose a destination directory where the new .DWG files will be created.

Generate a linked file that feeds into the AUTOFLUID preference file. This command lists all the model floors. Select the floor(s) you need and then choose a destination directory where the new .LST file will be created. Next, import this file in your AUTOFLUID preference file, in your CAD software. AUTOFLUID will know the names and levels of the floors of your project's model. The communication bridge between the 2 software is created.

RVT-Connect : description

The RVT-Connect ribbon in REVIT



Once your 2D plans and duct network are designed in your CAD software, export them in IFC format and use this command to incorporate them in the model. They will be automatically located at the right place with the correct height.

List and select embedded duct network with the 'embed IFC' command.

To send .RVT files to your clients, embed your duct network into a blank model and then save it in .RVT format.

Create a text using the information provided in each duct network object.

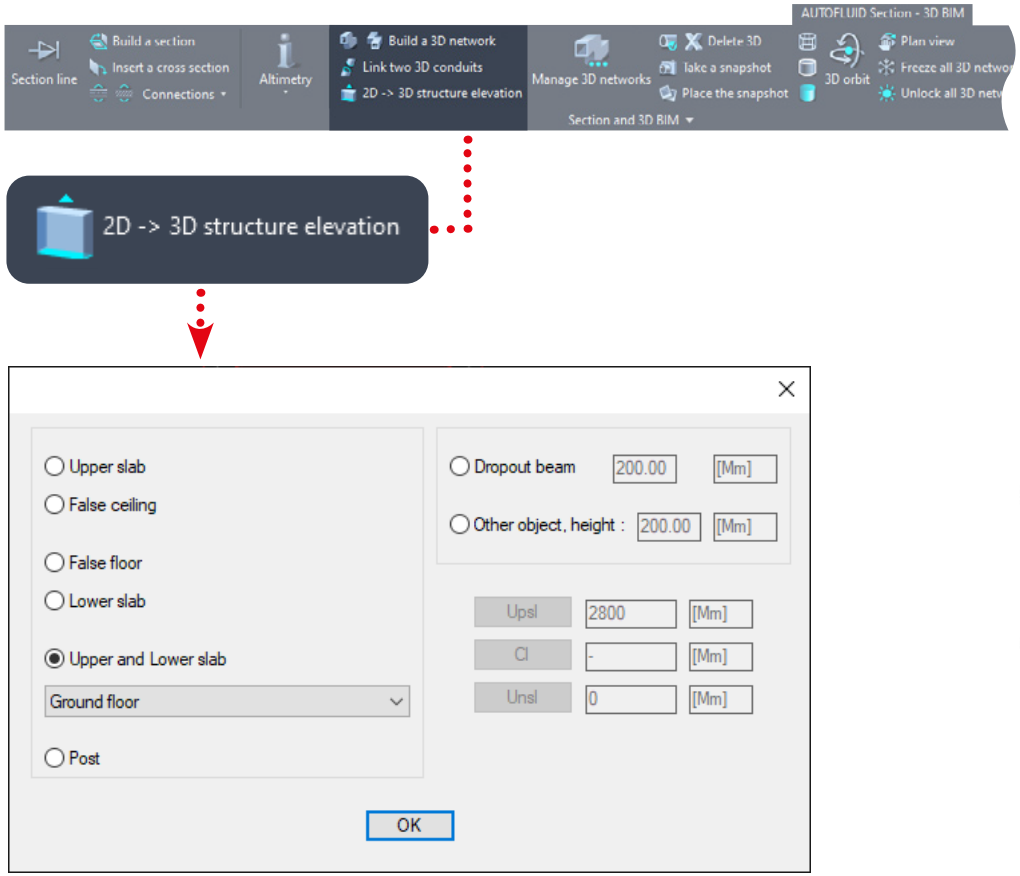
Generate detailed bills of materials from your .IFC duct network.

Show possible interferences between the AUTOFLUID .IFC duct network and the architecture or any other object in the model.

TeamViewer module for hotline service assistance.

Information about the software.

Creating 3D architectural elements

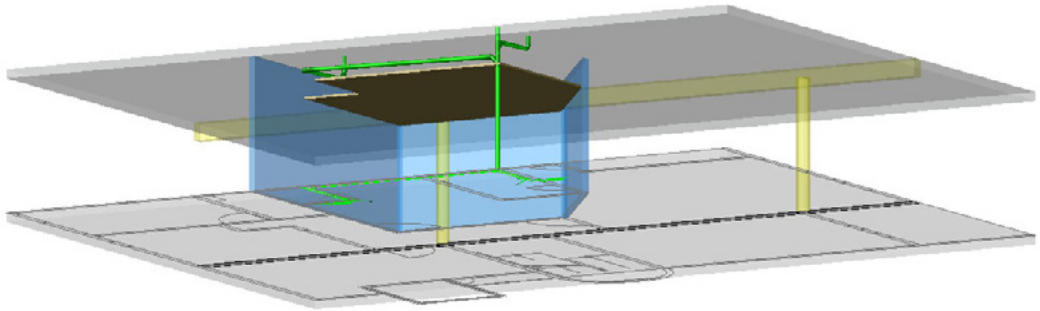


This is not an architectural design software, but a tool that will enable you to make 3D elevations of the most common architectural elements drafted in 2D.

You will thus be able to visualize the networks in their environment without having to transfer them into the 3D model. This tool can also be used to design the structure of machine rooms.

The transparency of architectural elements can be set in the Preferences file.

An example of a bottom view:

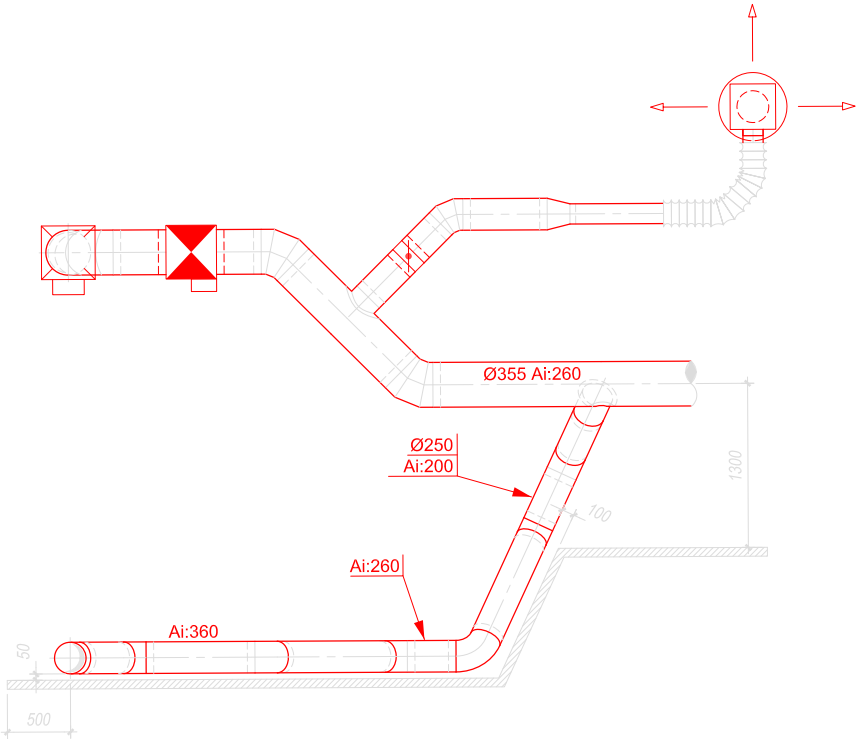


3D Architecture

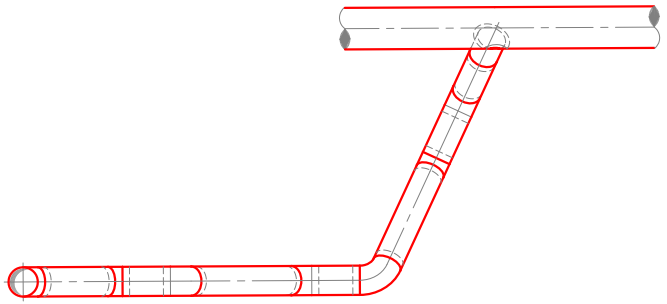
Creating a 3D network

Exercise in 3D création

This exercise contains all of the difficulties you may face when specifying the information of a 2D network.



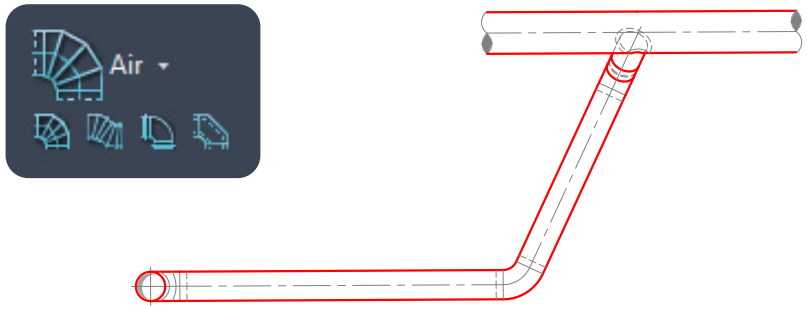
The branch below contains slanted conduits. This part of the exercise shows how to create 2D slanted conduits and adjust adjacent pieces (breaks, elbows...) and therefore how to get a 2D network that can be 'transformed' into 3D.



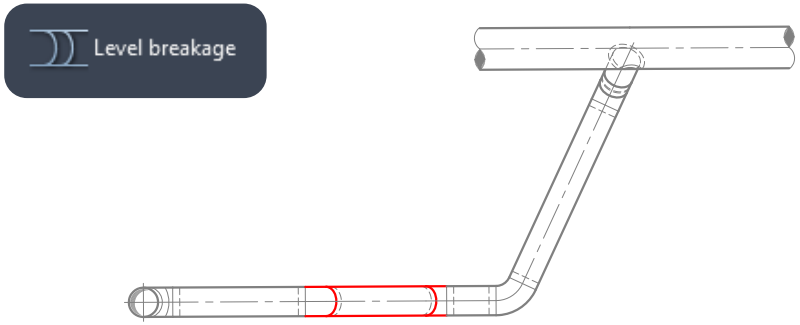
To get a good understanding of the process involved in drawing these conduits, follow the coming 6 steps.

Creating a 3D network

1. Draft the main network and then the connected branch

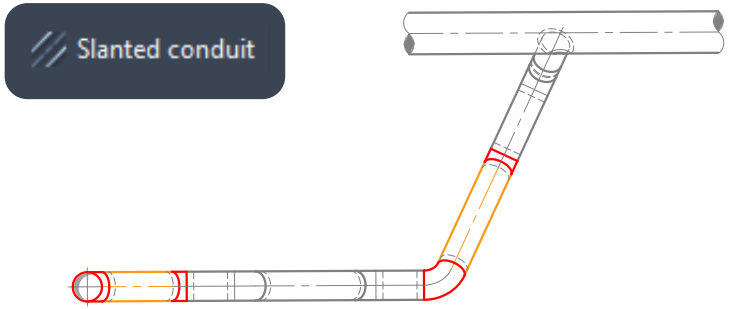


2. Add the break using the one-off command : «BREAK»



3D Transformation : steps 1-2

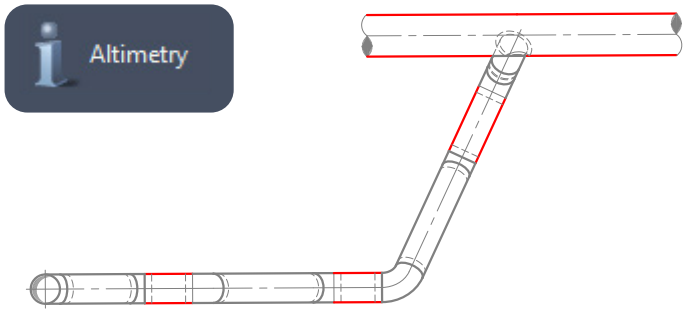
3. Cut the horizontal conduits to create the slanted conduits



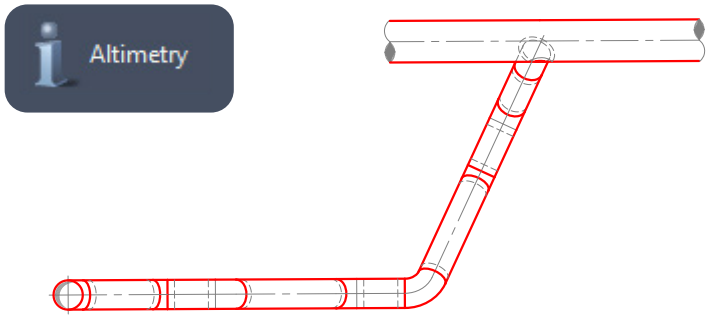
3D Transformation : step 3

Creating a 3D network

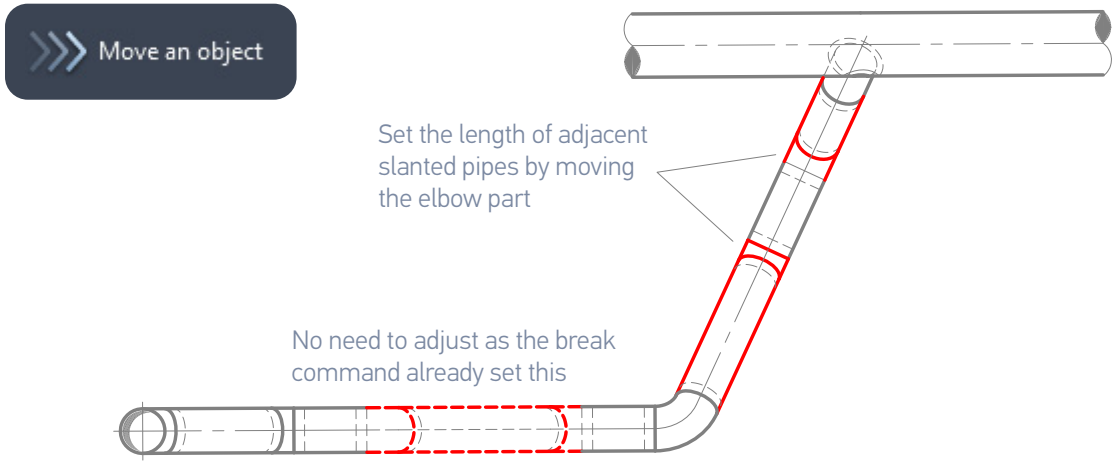
4. Specify the levels of the horizontal conduits without spreading the information to the rest of the network.



6. Spread the information to the rest of the network.



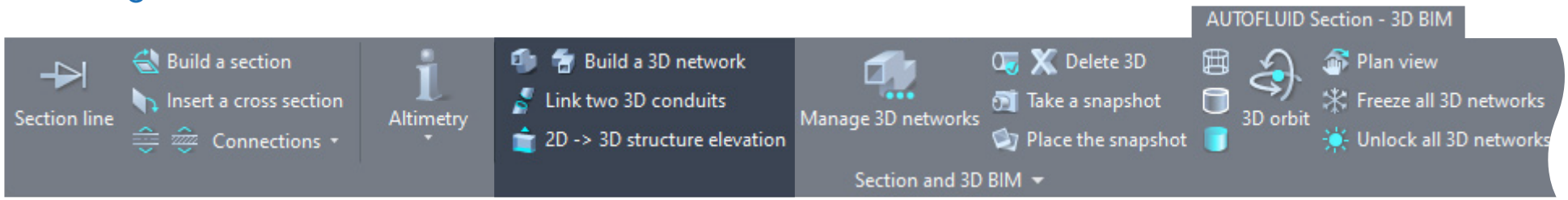
5. Adjust the position of the elbows.



3D Transformation : steps 4-5-6

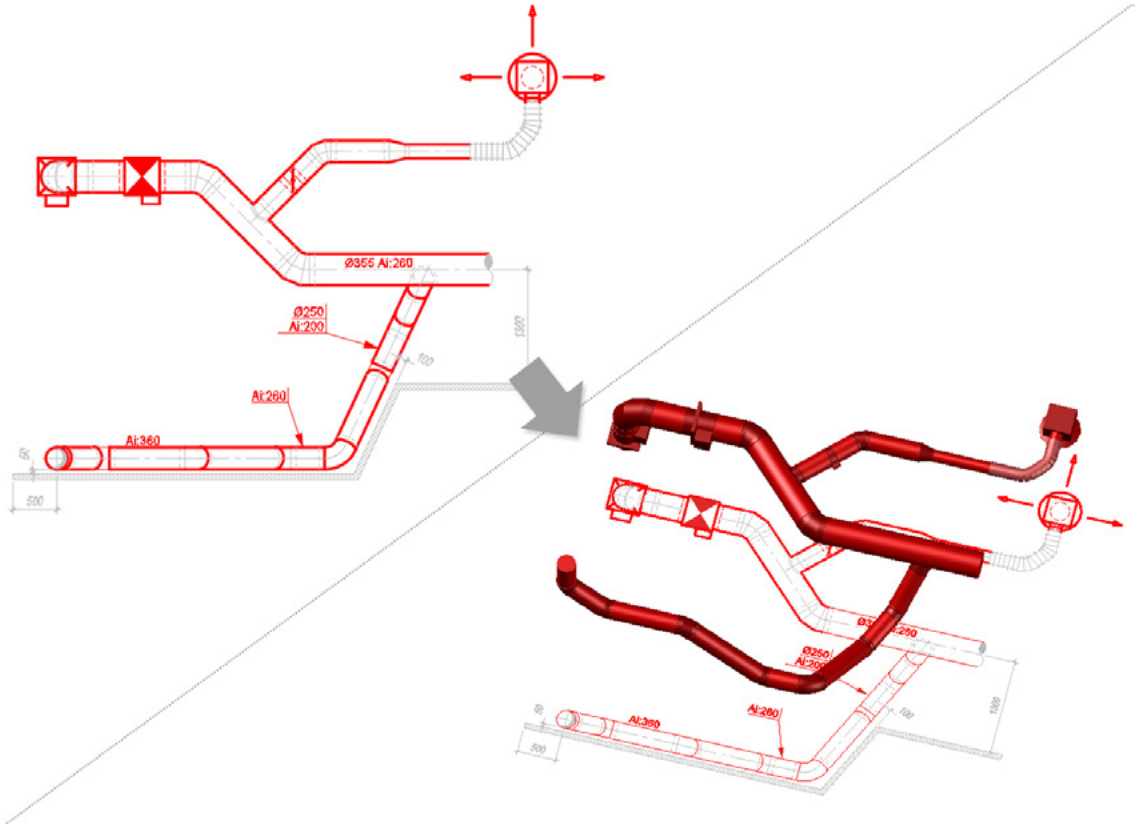
Creating a 3D network

Creating a 3D air network



Capturing

Simply select the 2D network to create the 3D network.



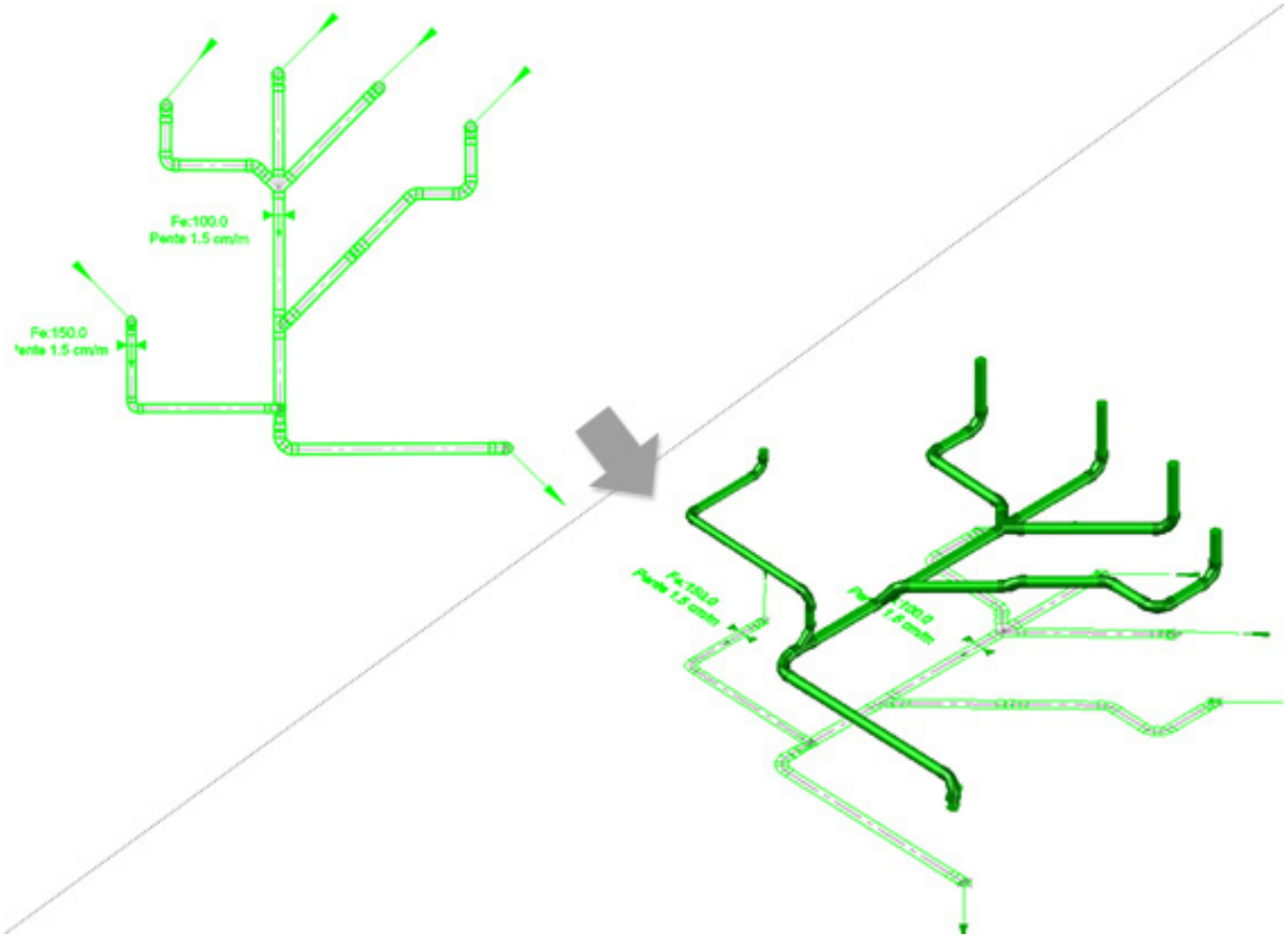
Creation of 3D network

Creating a 3D network

Creating of a 3D drain network

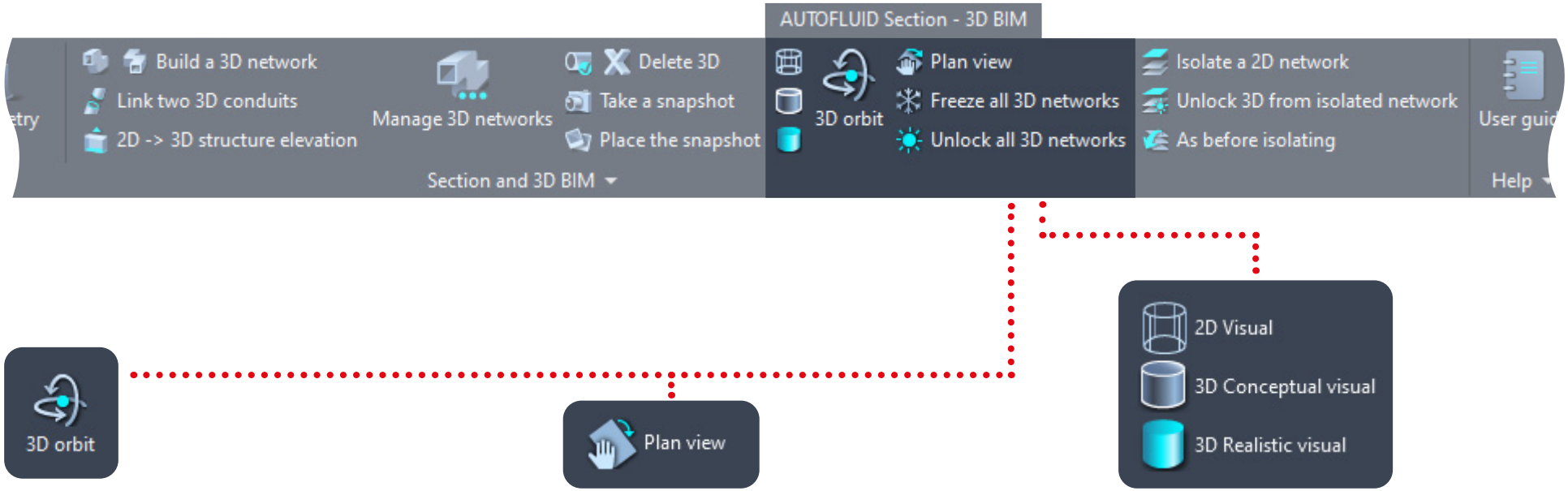
It isn't necessary to select the whole network at one time. It is recommended to break up the process and select one part of the network at a time.

See section 'Modeling -3D networks management' to Delete, Merge, Rename, or Export networks.



Creating a 3D network

Previewing the model



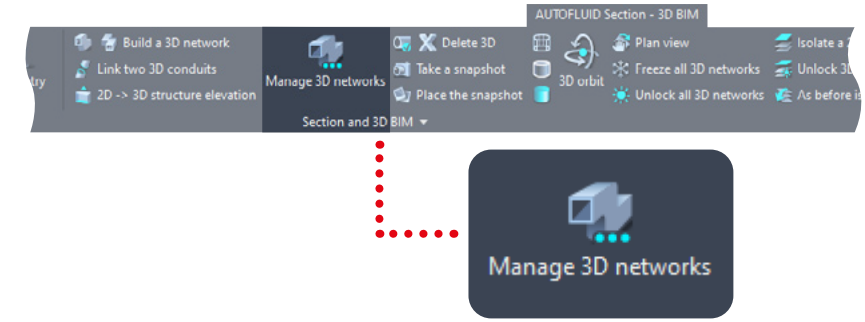
Lets you orbit in any direction around your network. It is the native command of the CAD software you use (see your CAD software User Guide).

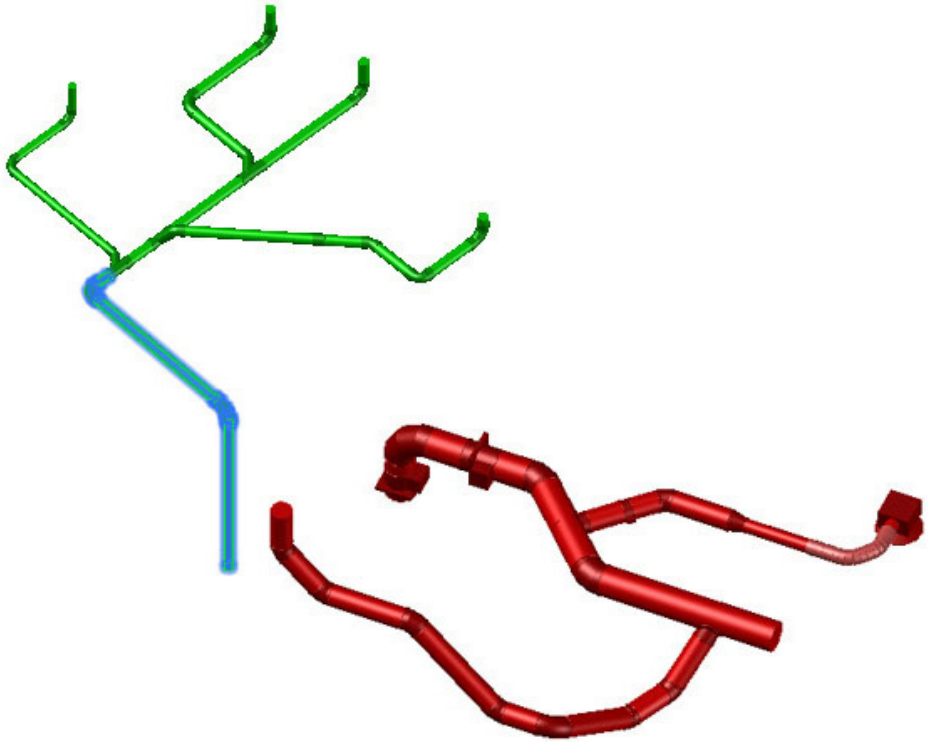
Allows going back to plan view with a zoom onto the selected entity (prevents the Zoom Extents activation of your CAD software command).

Preview styles :

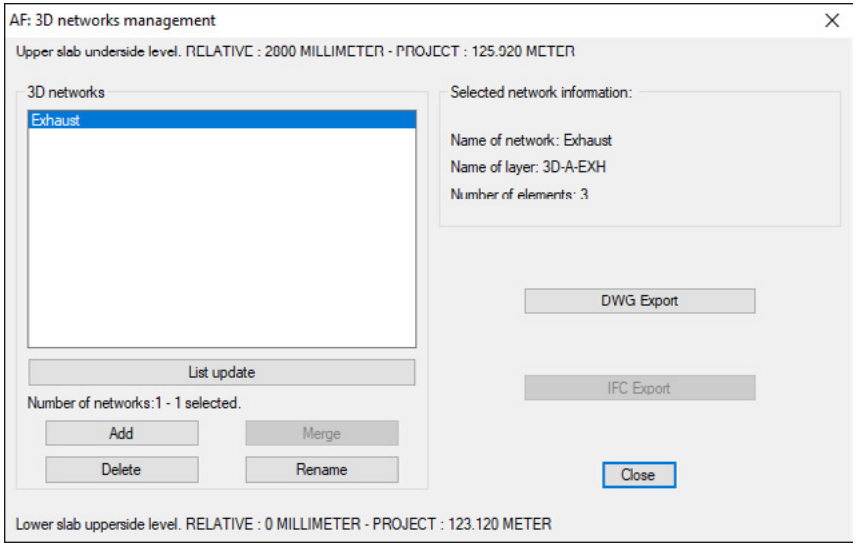
- 2D Wireframe
- Conceptual style, with visible edges
- Realistic style: this is the most appropriate aspect for creating 3D snapshots.

Creating a 3D network





Modeling -3D networks management



This command will allow you to merge the different segments of a network (from the same layer). Networks can also be renamed or deleted. Lastly, select the desired networks and click on «DWG Export». The selected networks will be automatically combined into one DWG or IFC file.

When saving, a default folder and file name are suggested - they can be modified. The resulting IFC can then be imported into a modelling software like Revit.

Creating a 3D network

RVT-Connect : Import 3D networks into REVIT

When your duct networks are completed, import them in REVIT.
There are several ways they can be imported:

1. Import or link your 3D duct network in a .DWG or .IFC format
2. Embed your IFC files into the model
3. Create a .RVT file that will be 'linked' inside the model

The purpose of AUTOBIM3D is to use your 2D plans as a key part of the production process. All changes to the 3D model will have to be done through the 2D plan first in order to be reflected in the modified 3D model part.

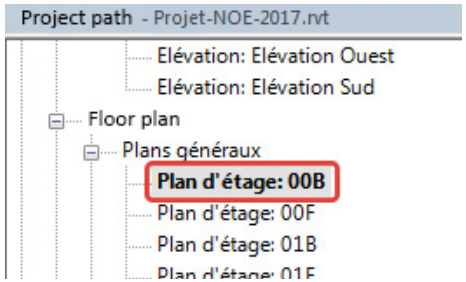
Link CAD | Link IFC | Link Revit

If you use the 'Link...' tools, REVIT will automatically update the model.

1. How to import or link your 3D duct network in a .DWG format

Open the project (.RVT file)

In the project path window, select the floor plan matching the AUTOBIM3D model to import (the positioning is automatic)



In the AUTOFLUID toolbar

Link CAD | Link IFC | Link Revit

- «Import CAD» DWG file
- «Link CAD» .DWG file
- «Link IFC» .IFC file

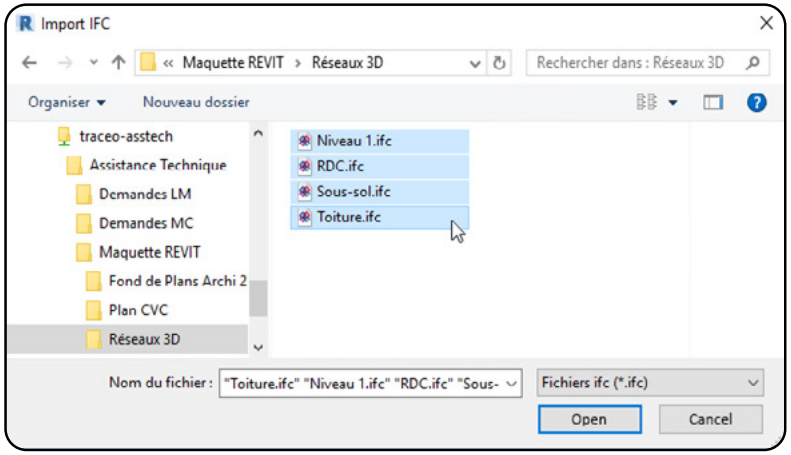
The "Link" command in REVIT is similar to the "Xref" command in AUTOCAD.

Now you can view your networks, however they can't be edited in the model.

Creating a 3D network

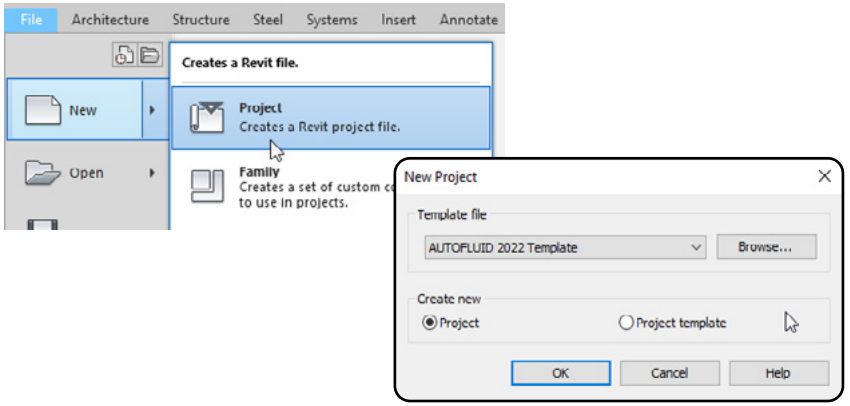
2. Embed your .IFC files into the model

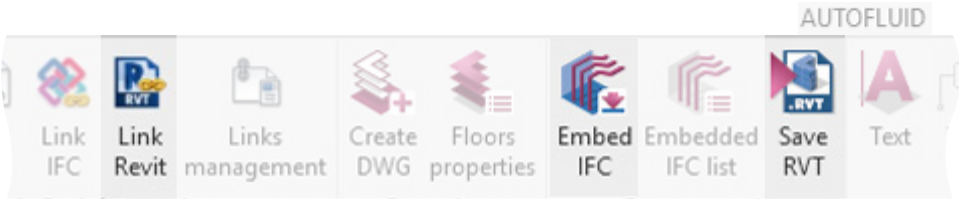
- Open the architect's model
- Embed your .IFC files
- Select your files (multiple selection is possible)
- Click 'Open'



-> Your duct networks can be accessed and edited in REVIT.
They embed all the BIM standard's data.

3. Create an .RVT to 'Link' it into the model. Open a new project using the 'AUTOFLUID 20xx.rte' template



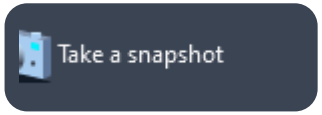


Compile the different floors of your IFC duct networks

Save your compilation as an .RVT file

Link your .RVT file

3D snapshots - creation and placing

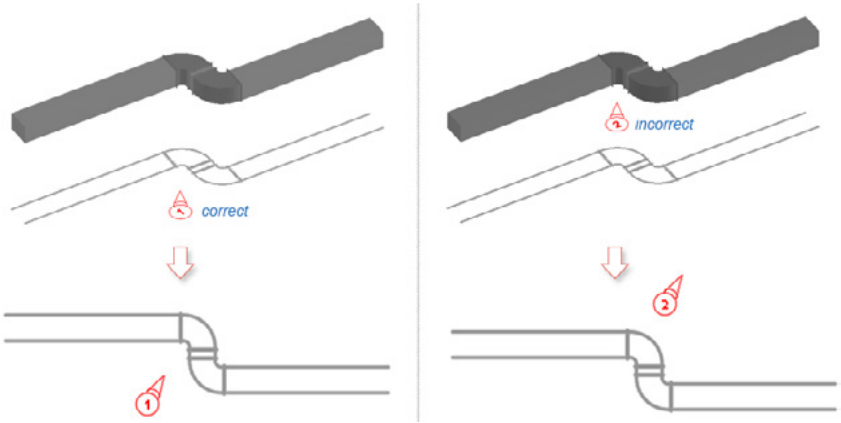


3D snapshots - Creation

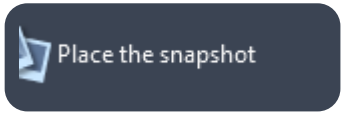
The « SNAPSHOT » command allows you to enrich your 2D plan presentation. It is not part of the modeling process. After choosing the view using the visualisation tools (positioning and aspect), run the « TAKE A SNAPSHOT » command, position the marker and select the 3D elements.

- The orientation of the marker number corresponds to the initial UCS

- Make sure to position the marker accordingly to the plan view: it must indicate the 2D views elements.

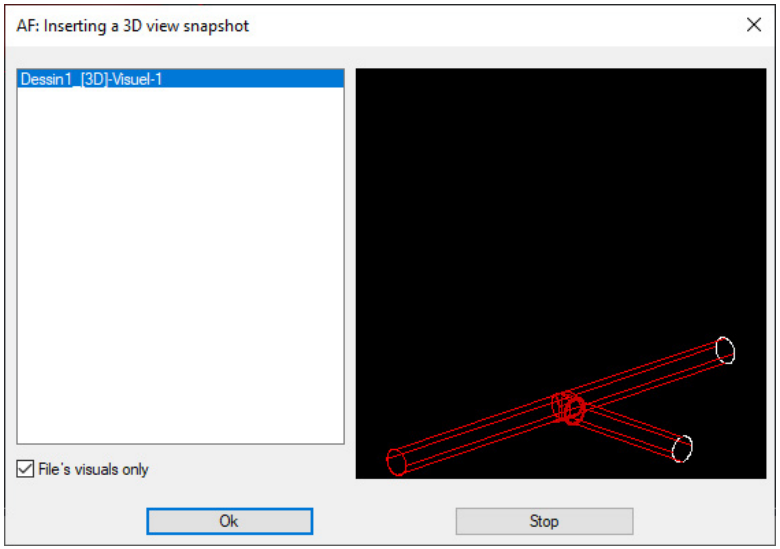


The marker « 2 » below is incorrect as it points to the 3D network in Orbit view, but it is placed behind the 2D network in Plan view.



3D snapshots - Placing

Position the 3D view snapshot in Model or Layout space.



Contacts : Support, Training, Video tutorials

Technical support on mobile and PC

If you didn't find the answers you were looking for in this user guide, or if you encountered a problem you can't solve, you can also find information here :

- See our FAQs page
- Contact the Hotline

Join the AUTOFLUID community

Subscribe to the AUTOFLUID Newsletter to receive all the tips, new releases, or special offers to get the best out of your AUTOFLUID suite!

More videos online

Navigate and discover all our video tutorials, methods and new features on mobile and PC at www.autofluid.fr/en/tutorials-and-videos.

YouTube Channel

Subscribe to the Tracéocad channe 

updates. You won't miss any of our tips and tutorials !

Training

Make sure to get the most out of your AUTOFLUID suite with training specifically designed around your needs.

Our programms are aimed at beginners as well as advanced users. They can be held as one to one sessions, from a distance in video conferencing, at your premises or even as multi-business sessions at our offices.

Tracéocad instructors are professionnals in the field of CAD for fluids whom skills stretch largely beyond mere product demonstration.

To speak to an adviser and to receive a quote for training, please contact us via e-mail contact@traceocad.fr or telephone : **+33 (0)4 86 79 20 00.**

What do you think ?

Your opinion and suggestions as a fluids professionnal are a precious contribution to the quality of the AUTOFLUID suite. It helps us making constant improvements and contribute to your productivity in the best way. Let's talk !

Index of videos 1/2

Index of videos 2/2

TraCéo
FLUID SOFTWARE

contact@traceocad.fr • Tel: +33 (0)4 86 79 20 00
www.autofluid.fr • www.traceocad.fr