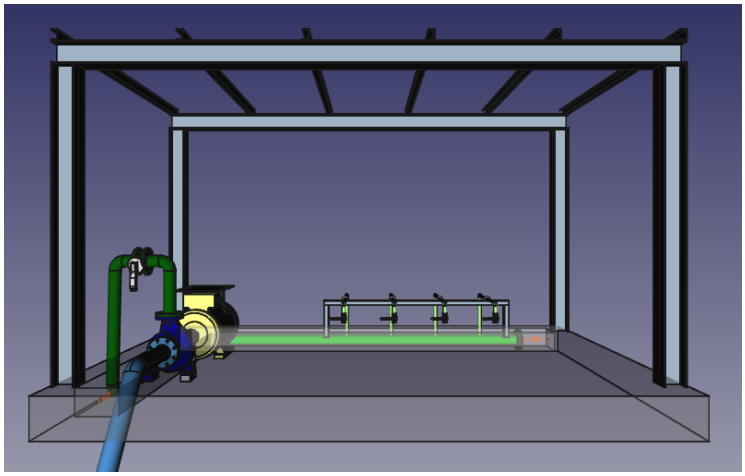
Next tutorial illustrates the latest features of the workbench.

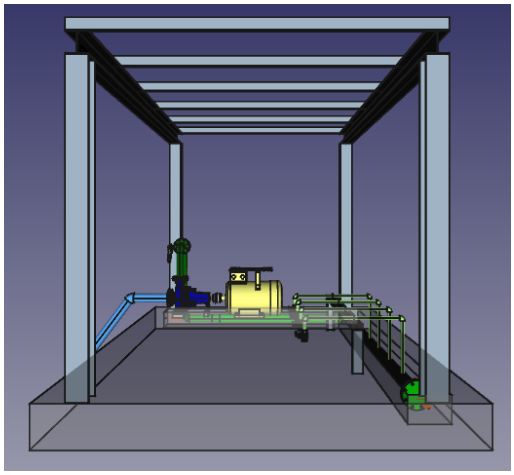
At the end of it, the appendix will describe more in detail tools' dialogs and options, so it should be considered as the overall documentation of Flamingo tools.

Most important improvements deal with drafting and easier managment of frame-lines / pype-lines .

TUTORIAL

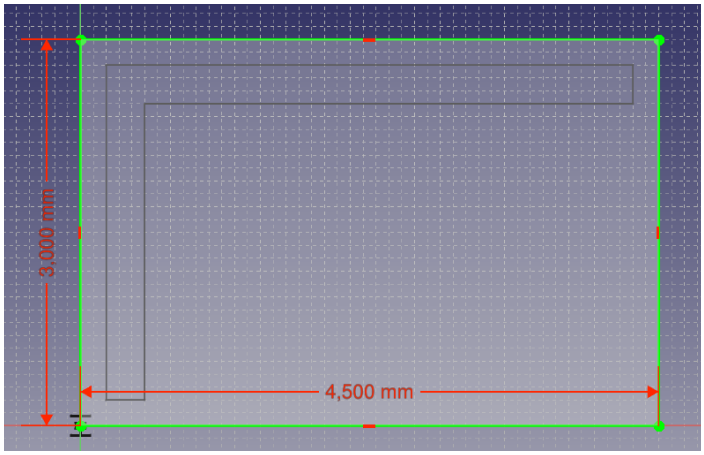
Task is to draw a kind of water pumping unit like the one below.





1. CONCRETE BASE

Create a sketch with Sketcher and draw the outline of a rectangular concrete basement, 4.5 mt wide and 3 mt high.

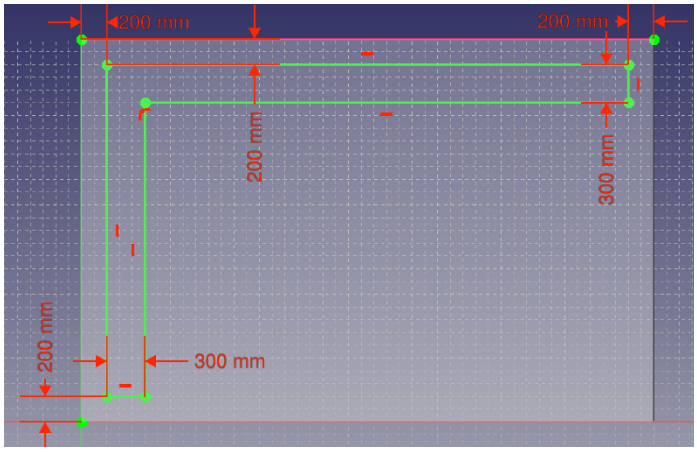


Then extrude it downwards for 300 mm.

Select the upper face and attach to it a new sketch for the channel (300 mm wide and 250 mm deep).

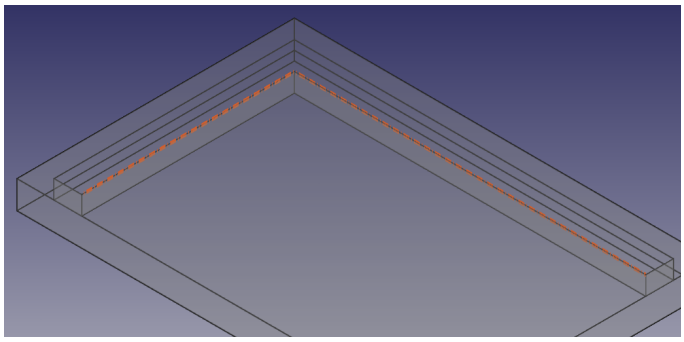
Create the pocket on the concrete.

To see better the following operations, set the transparency of this object to 80%.



2. MAIN HEADER

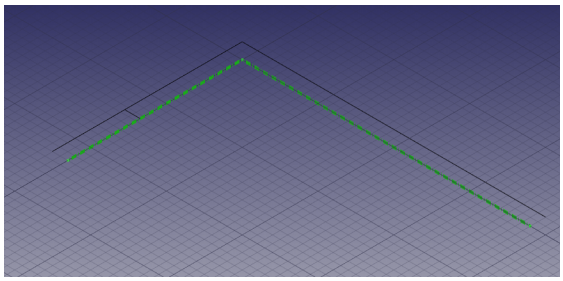
Select the two inner edges of the L-shaped channel, first the vertical, then the horizontal: that will orient the centerline in a convenient way for further operations.

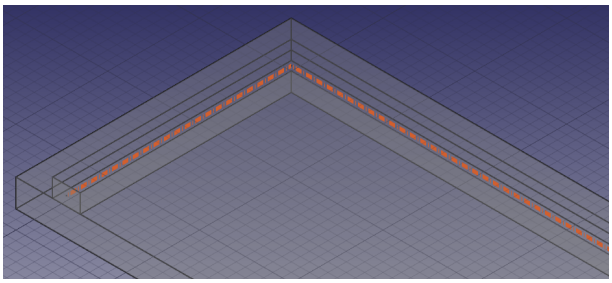


Press the "create path" button. This creates a nice orange and dash-dotted wire that will be used as the centerline of pipe to be created next.

Note: this step is not strictly necessary, because it's also possible to insert a pype-line directly from existing edges and the path is automatically created. Nevertheless for illustration purpose I deemed more clear to go in this way. Read the APPENDIX for more informations about single commands.

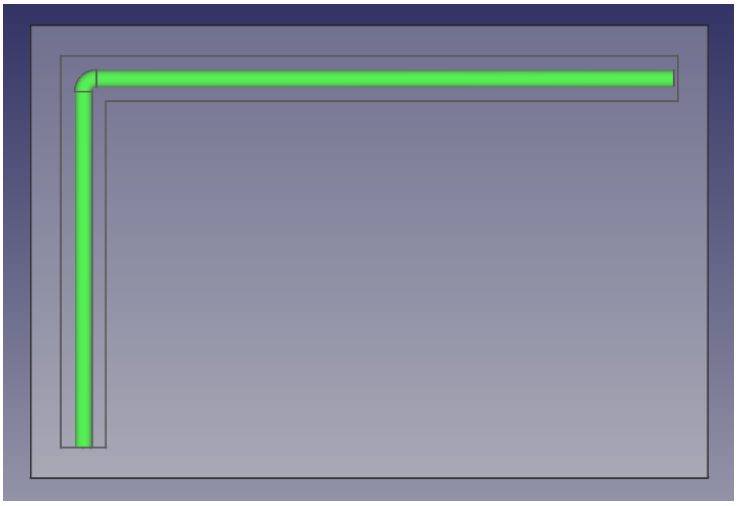
Use the Offset tool (150 mm outside) from the Draft workbench and the shiftTheBeam tool (100 mm down) from Flamingo workbench to center the DWire in the middle of the channel.

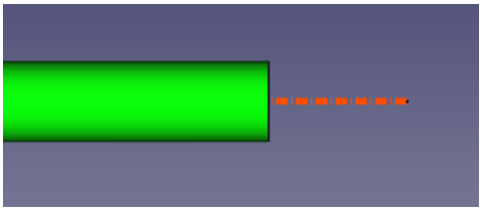




Select the centerline in the viewport and call the Pypeline manager from the "Pype tools" menu or from the corresponding toolbar.

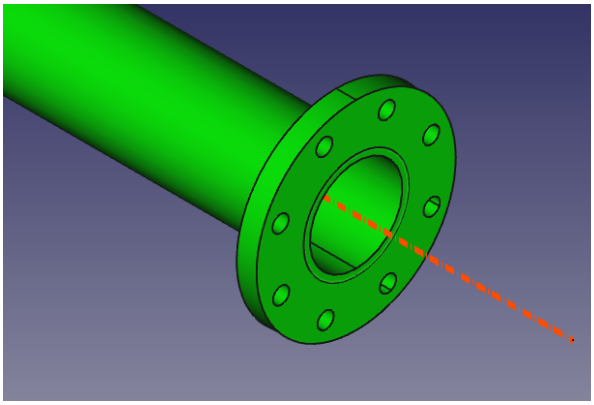
In the dialog that appears select the desired pipe rating (SCH-STD for instance), the desired nominal pipe size (DN100) and the color (I like green). Check in the combo that <new> is selected and optionally put in the text-box the name of the pypeline to be created (e.g. "main"). Finally the pypeline is created by pushing on "Insert".





Select the horizontal pipe and call the stretching tool. In the dialog that appears press the "Get length" button and round the value in the text-box to be more or less 300 mm shorter. Press OK to change the length of the pipe and then Cancel to exit from the dialog.

You might have also changed the length with the horizontal scroll-bar, if you wished to make it more roughly.



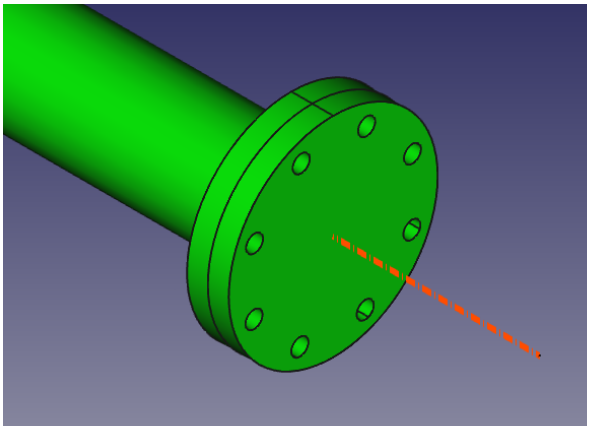
Add one flange to the pipe end.

Select the edge of the pipe and call the insert flange tool. In the dialog select the flange type and, from the combo, select the pypeline which the newly created flange is going to belong to.

The size of flange is automatically determined by the pipe selected while the color and the group are determined by the pypeline selected above in the combo.

"Insert" to create the flange.

To create the mating blind flange, select the outer edge of the flange depicted above: now it's necessary also to choose the size (DN100) because no pipe is selected in the viewport. Press "Insert" and a new flange is created placed concentric to the selected circular edge. To blind it, just set to 0 the "d" property (hole diameter).



Flanges may be overlayed: in that case select the outer circular edge of the blind flange and push the reverse orientation button.

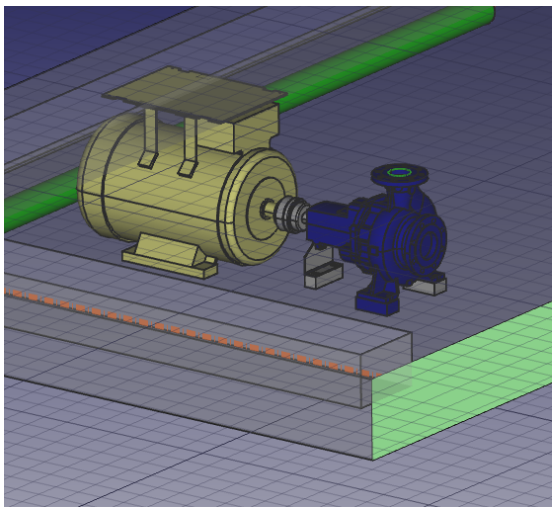
This tool has been specifically created to reverse orientation and position of pipes and beams: if an edge on one of the ends is selected, that edge is used as pivot.

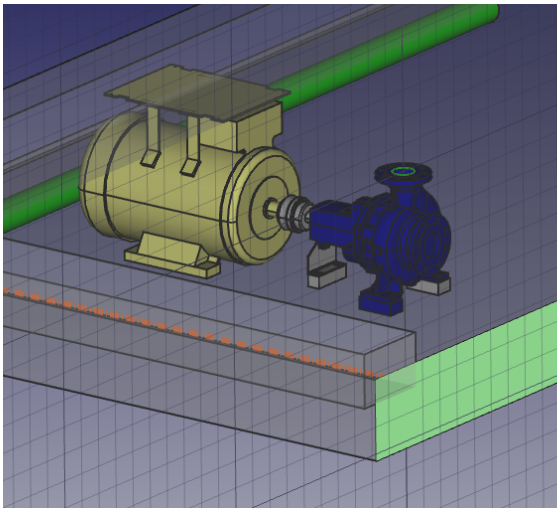
3. PUMP AND DELIVERY PIPING

As said at the beginning, improvements have been done to drafting tools: that can be seen in the Utils toolbar. As before it's possible to change the working plane according existing geometry (with some additional options) but now there's also the possibility to spin or offset the working plane. Also a little hack of the DWire creation tool has been added, to allow drawing more complex paths without interrupting the command to change working plane.

Merge the model of a centrifugal pump (I used a standard EN pump 100/80/200 taken from a repository of 3D models on the web) and move it next to the vertical branch of the channel like below. To do that the common tools of Frame tools menu are available.

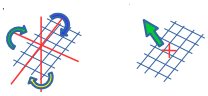
Select the circular edge of delivery flange of the pump and the vertical face of the concrete slab as shown below. Call the "align Workplane" command and WP will be centered to the circular edge center and oriented according the face selected.





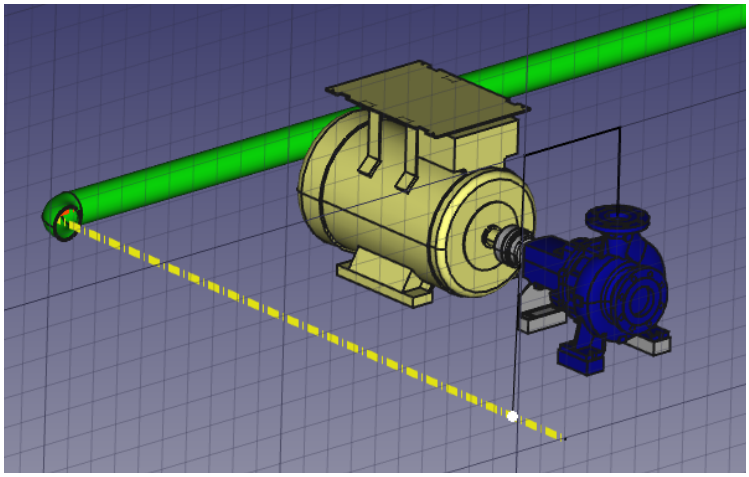
There are other combinations possible for this tool, among others:

- one vertex
- one face
- one vertex + one face
- two straight edges (not parallel)
- one curved edge



Beside that it's possible to rotate the WP around its axes or offset it along it's normal. In such cases a temporary green arrow is showed to indicate the direction of translation or rotation.

For convenience, hide the pipe that runs parallel to the pump.

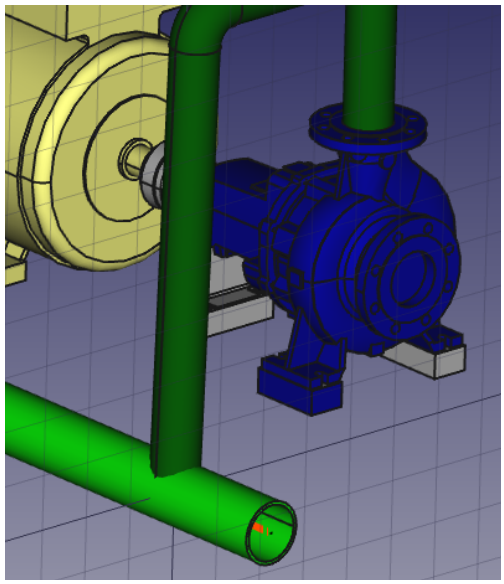


Then use the enhanced "Draw a wire" tool to make the path of the delivery pipe of the pump: start the line from the center of delivery flange and end it perpendicular to the axis of "main" pype-line as depicted at side. Use the suitable snapping tools for this.

As you may notice, each time a point is added to the DWire, the WP center is also moved to it: that makes easier to sketch down segments of known length using the snap-to-grid feature.

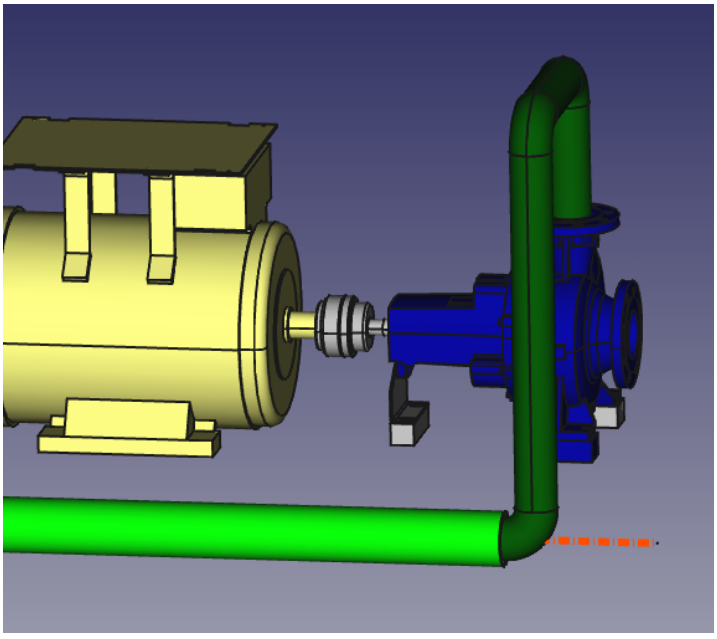
Also at the bottom of the dialog are added some additional buttons that allow to re-orient the WP without interrupting the DWire. You may call these features also with acceleration keys <Ctrl>+<Shift>+<Letter in brackets> (*note: the shortcut won't work if the viewport has lost focus*).

Create one new pype-line named "delivery" (SCH-STD, DN80, darker green) as done for the "main". →



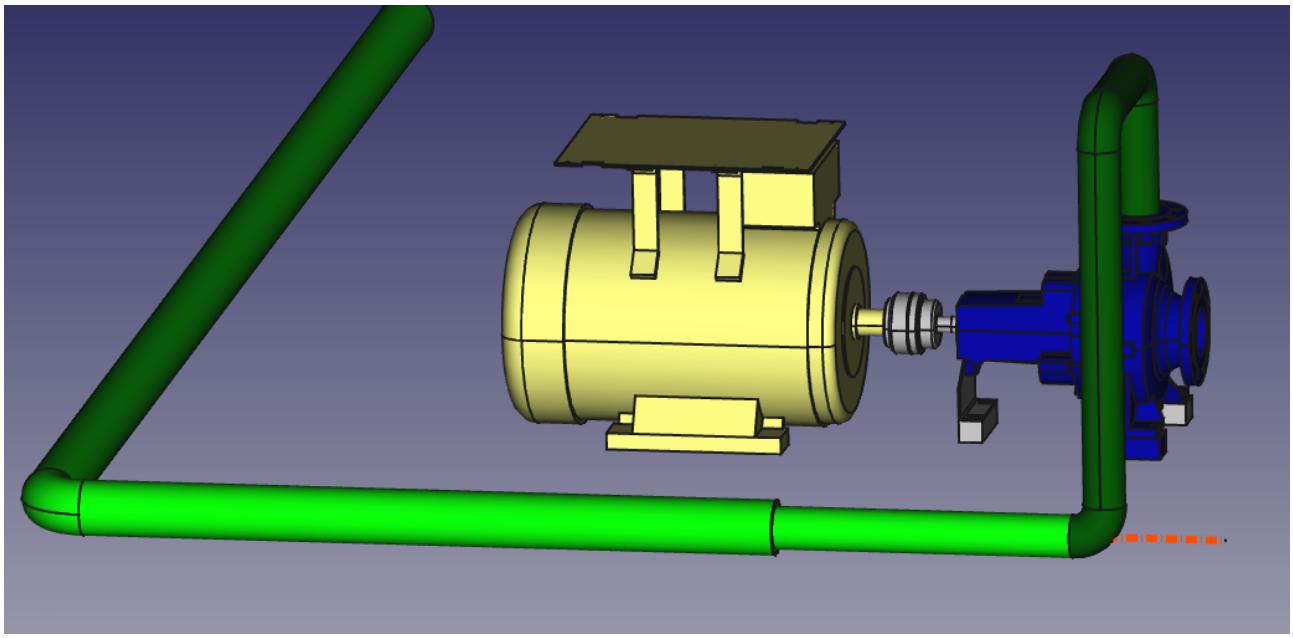
Connect the two pipes creating a curve between them: select first the DN80 pipe, so that the size will be automatically selected, and in the combo select "delivery", so that the curve will belong to that pype-line.

↓

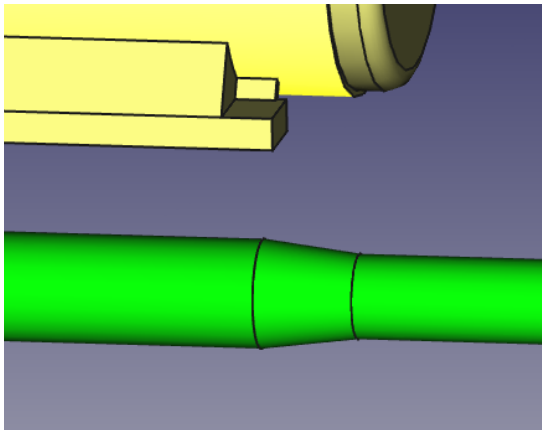


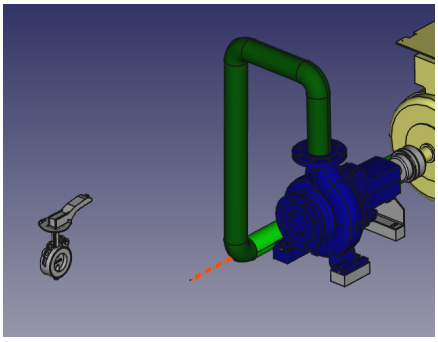
Break the pipe so to create two sections of pipe one after the other. As the length is not important, tap on the slide bar until "Point" value is set to 30%. As an alternative it is also possible to insert the value, absolute or in %, directly in the text-box. Care that the pypeline in the combo is "main".

Then use the "Insert pipe" tool to change the dimension of the first section of the pipe: select the pipe in the viewport, select DN80 in the dialog and push "Apply". Result shall be like below.



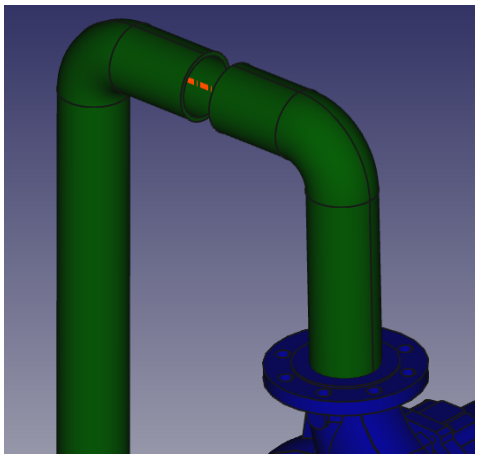
Select the two adiacent pipes of different diameter and create the reduction between them. In the dialog you shall only care about the pypeline selected in the combo and the type of reduction: the major and minor diameter are defined automatically by the pipes selected.

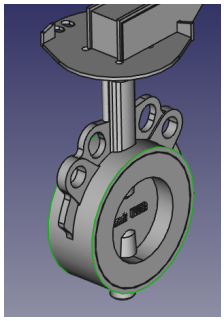




Merge into the drawing the model of a DN80 butterfly valve (I used a common wafer-type with lever taken from the same repository of the pump).

Use the same breaking tool used above but now define a gap equal to the length of the valve.

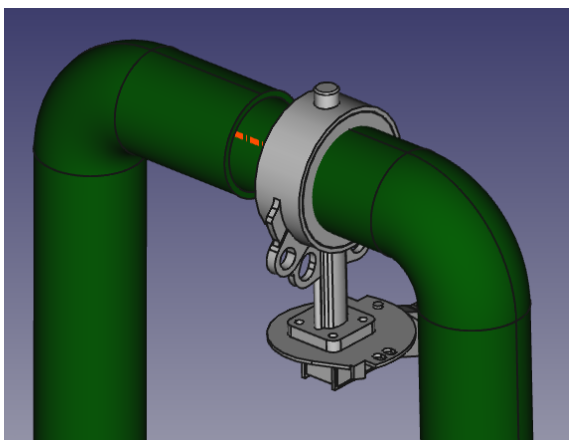


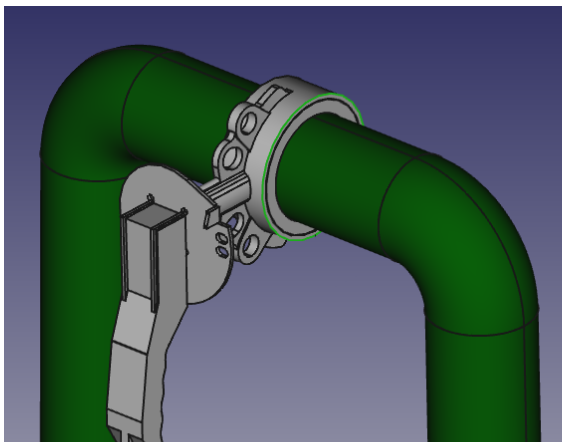


Beside the direct input in the relevant text-box, it is also possible to measure the gap from the geometry of the model; thus select the two opposite edges (or faces, as well) of the valve and press "Get gap" in the dialog. After that select the horizontal pipe segment of the delivery, set "Point" to 40% and press OK.

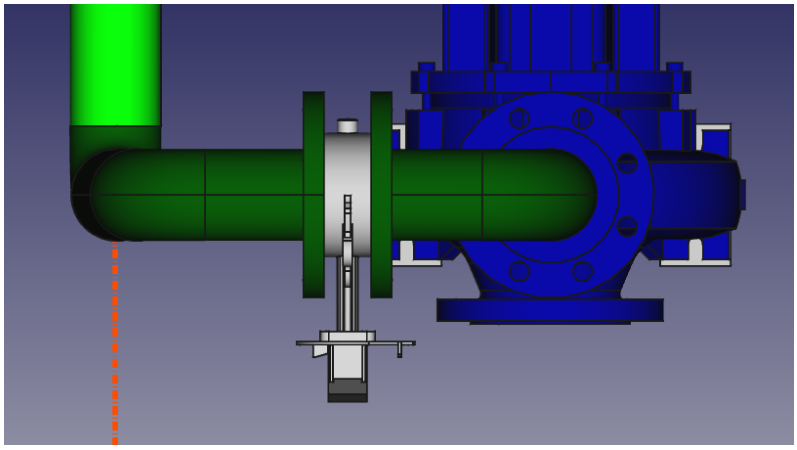
Use the tool "Mate pipes edges" to move the valve in the gap just created. It is a one-shot command without dialogs; so select one target circular edge on the pipe and the edge to be moved on the valve and push the button.

If the result is not satisfactory because the object is oriented the opposite way, for instance you have the chance to use the reversing tool as done for the blind flange earlier. Otherwise use alternative tools on the frame toolbar.



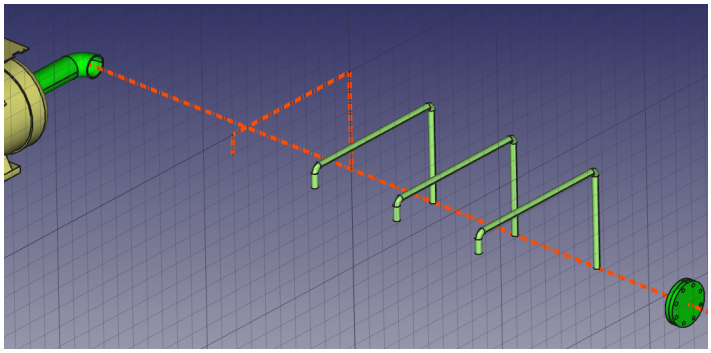


Use "Insert flange" to complete the piping with the missing flanges.



4. BRANCHES AND SUCTION

It's not worth to repeat the steps to create the remaining pipes. Just follow again, *mutatis mutandis*, the procedure done for the "delivery" pype-line.

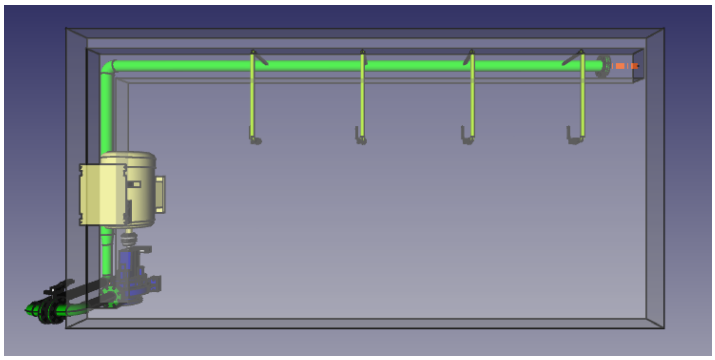


Only the main points:

- put the workplane orthogonal to the centerline of the horizontal pipe segment of "main"

- offset it to a suitable position

- draw approximately the centerline using the snap-to-grid feature

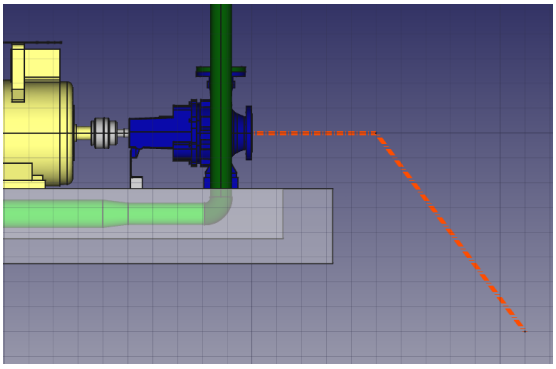


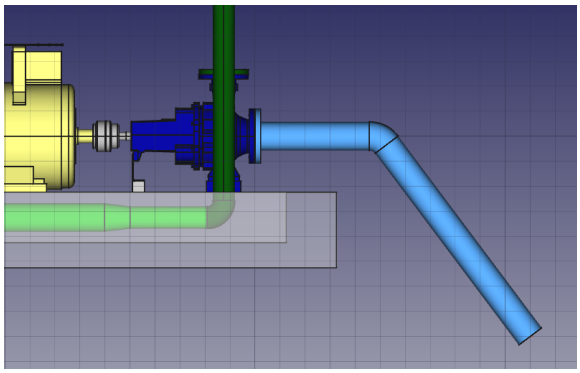
- copy the centerline 3 times along the x-axis with 500 mm distance

- create a pypeline (DN25, paler green) for each centerline

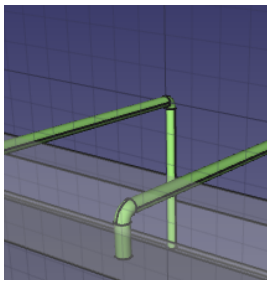
- import the model of a 1" threaded valve and place it at the free end of each branch

The same is valid for suction pipe (DN100, light blue).

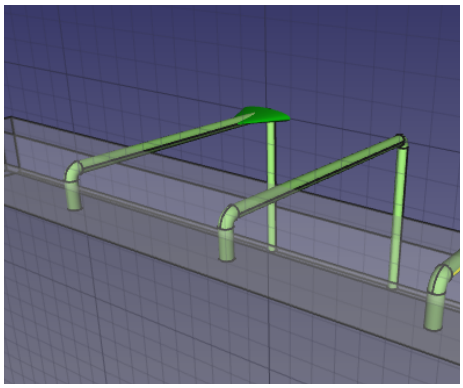




5. PIPE SUPPORT



For drawing the pipe support first change position of working draft plane so that it is orthogonal to the horizontal pipes of branches and more or less at the middle of their length.

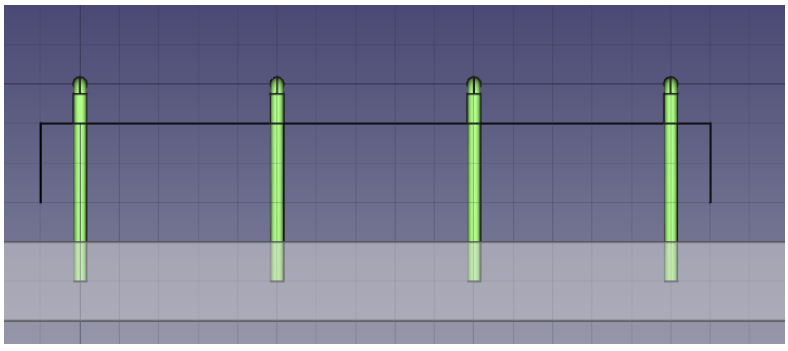


Select one circular edge at the end of the pipe and set there the WP.

Then offset it by an amount approx. half of the length of the pipe.

If in doubt how long is the pipe, query the model by selecting the tube and using the Query object tool.

Align conveniently the view and start to draft a kind of portal similar to the one shown to the right. It's not necessary that it touchs the basement or that it's tangent to the pipes. Just care that the vertical segments are approx. 100 mm outside the pipes' bounding box.

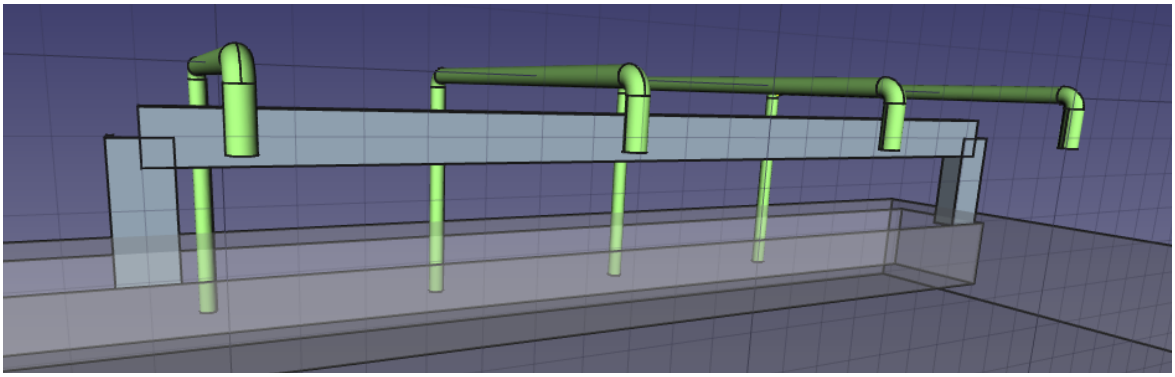


With "Insert Std. Sections" of the frame toolbar it is possible to insert in the model the standard 2D sections that will be used to create beams afterwards. Thus from the dialog select one UPN80, that is used for the pipe support, and one HEA160, that will be used next for the structure.

Hide the "Profiles_set" group that is created to store the prototypes of sections just inserted.

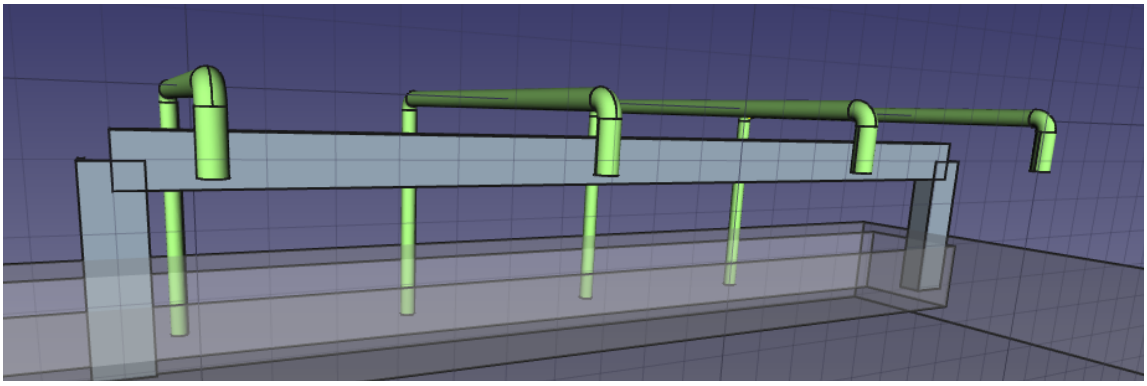
With Frame Manager create the frame-line object and group. In the dialog, from the list of available sections select UPN80; type a name (e.g. "support") in the text-box and press insert.

Notice that in the combo at top right corner the newly created frame-line is automatically selected after it's create. So go ahead, select the drafted portal structure and press "Get Path". Then press Redraw button and the beams of the specified section are drawn on the segments of the path automatically.

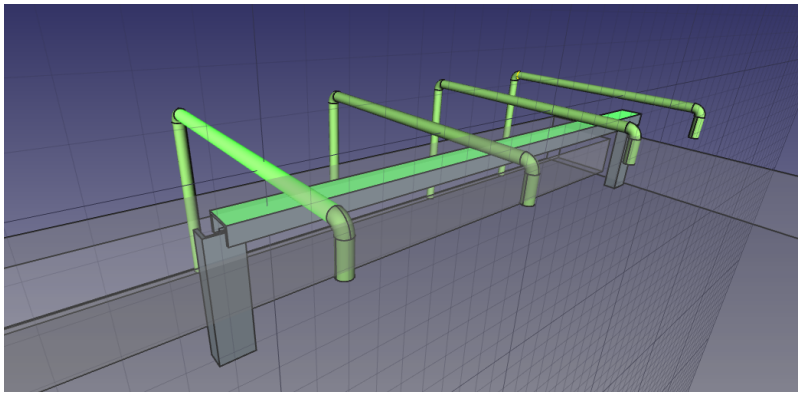


Extend the vertical beams to the ground: select first the face of the basement and then the two beams (use Ctrl key for multiple selection) then call the extendTheBeam tools.

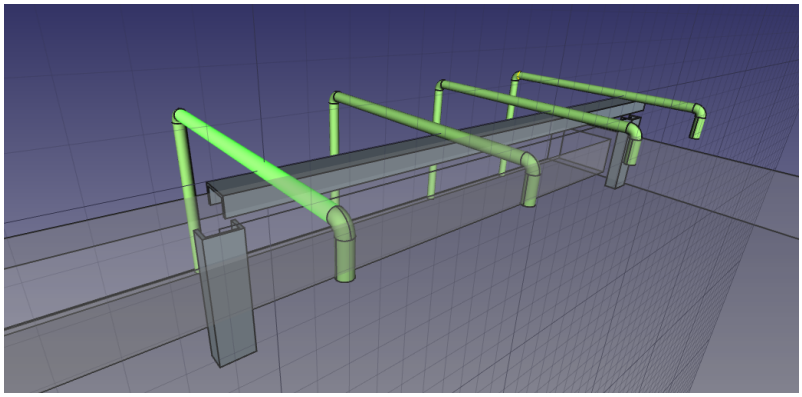
The face selected first is automatically de-selected and taken as the target to which to extend other objects: you see that also from the box in the dialog where it's written "*Object_name*:Face". So press OK to extend (or trim) the legs to the ground and press Cancel to close the dialog.



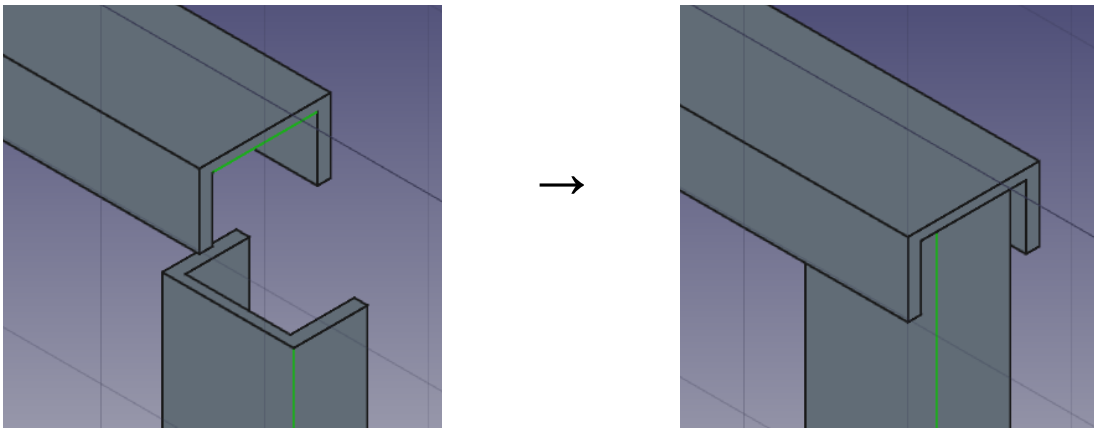
Select the horizontal beam and spin it until the wider flat face of it sees the pipes above it.



Make the pipes touch the face of the beam by selecting one of them and the face; then press the "Raise support" button.



Extend the beams where they connect with the adjustAngle tool, which extends reciprocally two beams placed at square angle by selecting in sequence their edges.



When done, always remember to press Esc to stop the command.

Note: as for pypelines, when frame-line is redrawn all these adjustments will be lost and the beams will be drawn on the original drafted path.

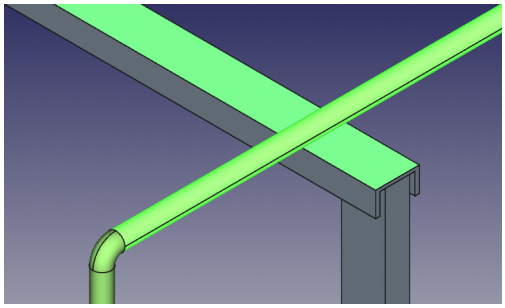
To avoid that it's possible to disconnect the frame-line object from a path by pressing "Get path" with a null selection. The same is valid for pype-lines.

Tie the pipes to support with U-bolts.

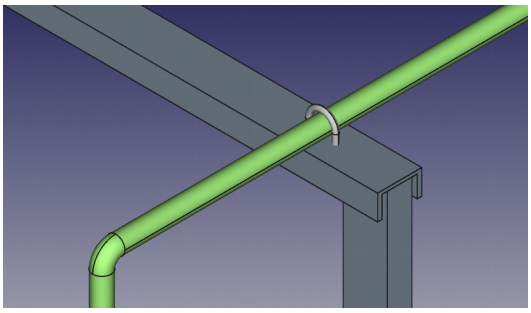
As done with the extension tool, it's possible to pre-select the reference face to orient the threaded ends of the U-bolt before running the command: you see the name of selected reference face in the bottom left corner of the dialog.

If pipes are selected, there is no need to specify the size of the bolt.

Only select the desired position checking the corresponding box (Head / **Middle** / Tail); select the pypeline that you want the bolt to belong (or <none>) and press Insert.

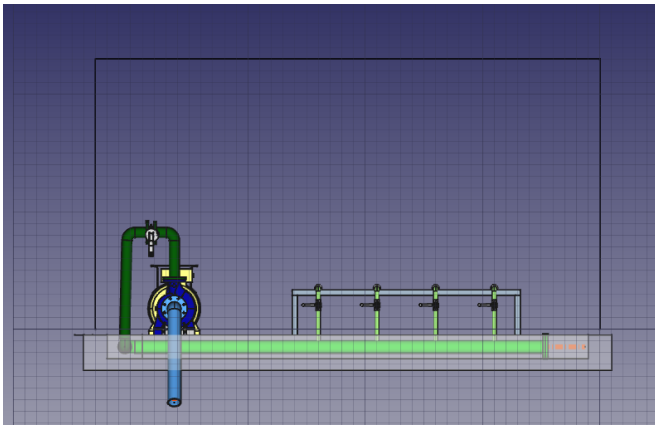


→



6. STRUCTURE

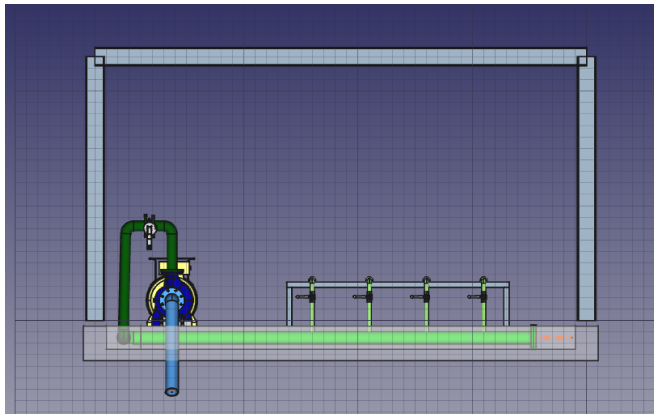
Finally draw the cover structure.



Begin with drafting a portal like did for the pipe support. For a change, remember you can also use a Sketch object as the path for frame-lines and pype-lines.

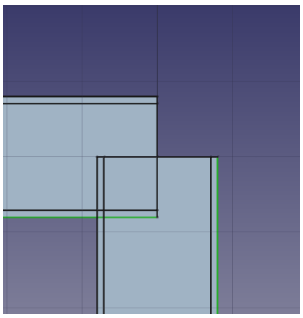
Create a new frame-line object, named "cover".

Get the path and Redraw it.

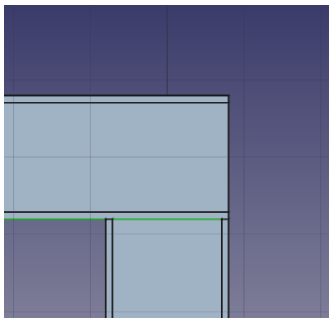


Then spin the beams around their axes to orient them as desired.

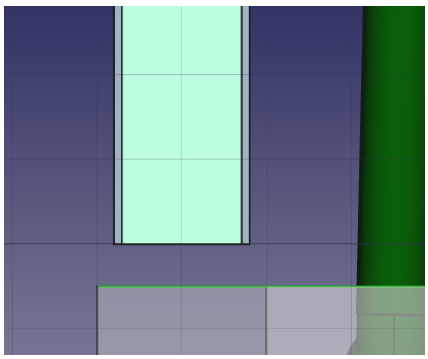
Adjust connections at the angles...



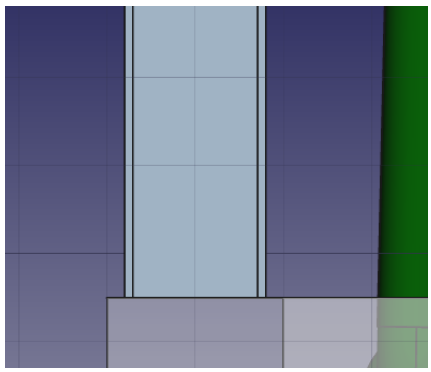
→



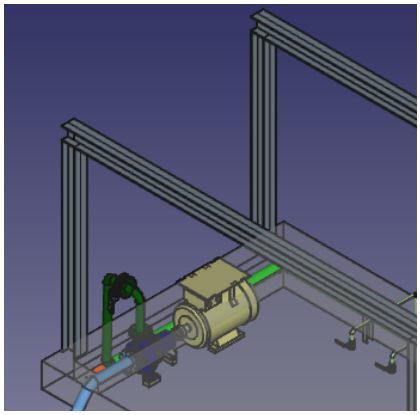
...and extend the columns to the ground.



→



Copy the beams to the other side of the basement.



When copying remember two things:

- the object that is moved is the original one, not the copy (this allow to create series of objects by applying the same command again and again);
- the copied objects are placed outside any pype-line or frame-line group where the originals were. So if needed you should drag and drop them back in the model tree.

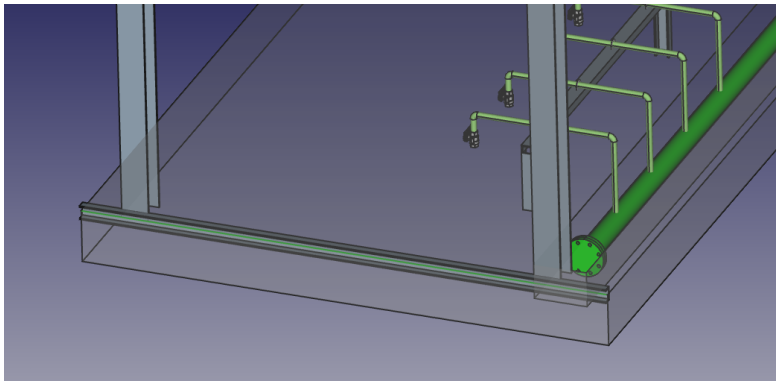
To create the final beams of the cover let's see an alternative way to add members to a frame-line.

As usual open the Frame-line Manager and in the combo-box select the already created "cover" frame-line.

Then select the section UPN80, which namely is not the same section assigned to "cover".

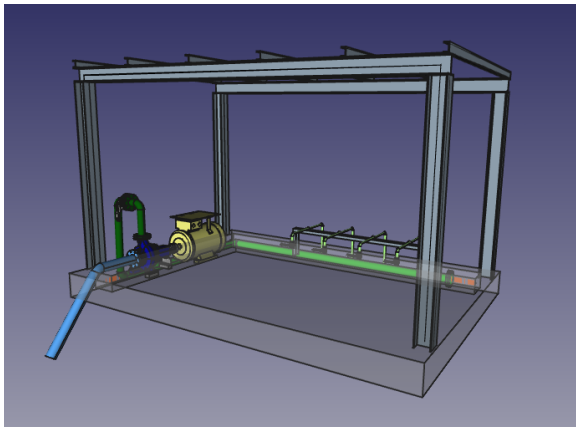
In viewport select any edge, for example the short edge of the basement.

When Insert is clicked a new beam with the section selected is drawn over the selected edge (or edges) and included inside the "cover" group.

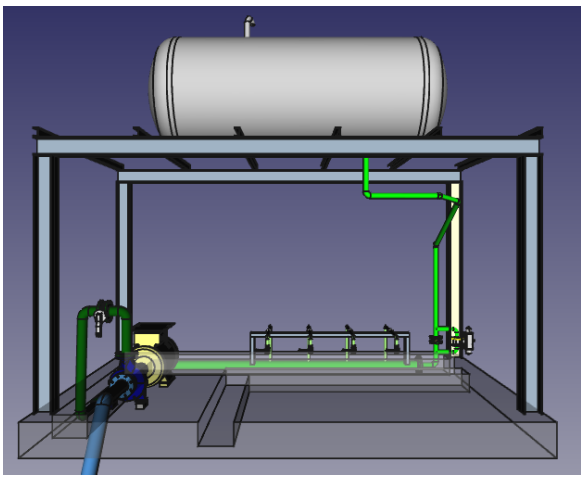


Note: the beams added in this way, that are not connected to the path, will be deleted during next "redrawing".

Now, to place the beam in the right place, trim it and copy it with the tools that we have learned to use should be an easy excercise...



For further practice you may also add a discharge funnel in the concrete and a cylindrical shaped reservoir (made with a pipe with customized diameter 1200 mm and 2 caps) on the top of the structure then connect it to the main distribution pipe with a check valve and a bypass automatic valve.



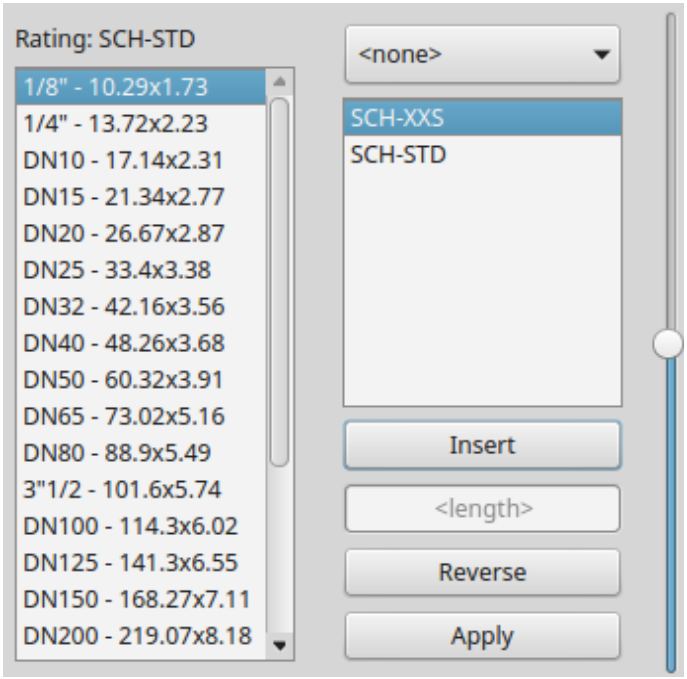
APPENDIX

Detail descriptions for each command in the toolbars can be found below.

PYPE TOOLS

1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19

1) Insert a tube



Opens a dialog to insert tubes.

All "*Insert a...*" dialogs have some common features. The top-right combo is one of those: it lists the pype-line objects defined in the current document: with this it's possible to select to which pype-line to assign the newly created pipes. You can also leave it to <none> so that the object is created on the root line of the model tree.

In the top-left corner is printed the currently selected pipe rating, taken from the listbox in the right column. Pipes dimensions for each pipe-rating are defined in .csv files, which is possible to add or modify, with few simple naming rules, according needs. *Curves*, *reductions* etc. have the same rules for definition of their tables of dimensions: see files in ../Mod/flamingo/Tables. Read also "tutorialPype.pdf" to know how to customize or create them.

To define position and orientation of pipes, following selections are possible:

- one or more faces (the center of it)
- one or more straight edges (along it)
- one or more curved edges (the center of curvature)
- one or more vertexes
- nothing; in this case the tube will be placed at origin.

If no length is specified, the default is 200 units (just a convenient length, in mm).

Other common features are:

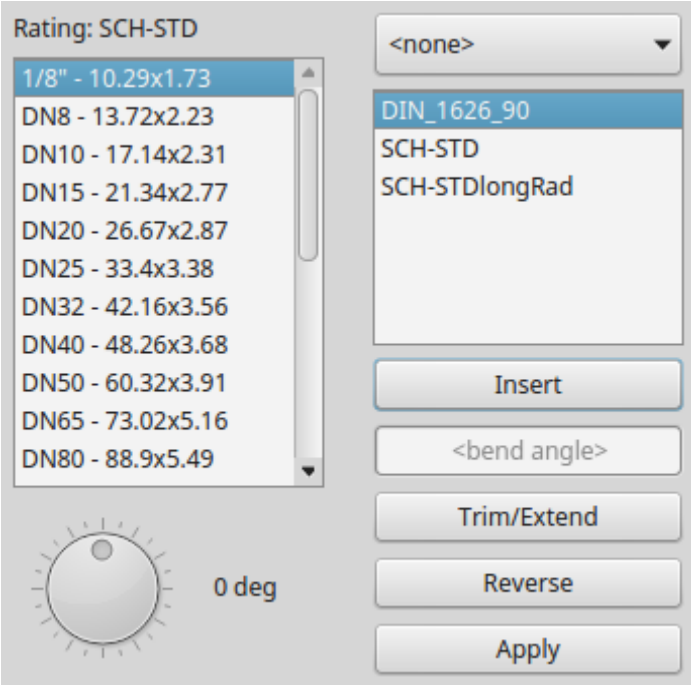
Reverse button, allow to rotate by 180° the last tube created or those currently selected.

Apply button, allow to apply a different lenght (if it's specified) or Nominal diameter to the tubes currently selected.

The side slider allow to change the length of last pipe between 1 and 200% after it is

created.

2) Insert a curve



Opens a dialog to insert one elbow.

Trim/Extend button allow to adjust the length of selected pipes to the selected edge of the curve.

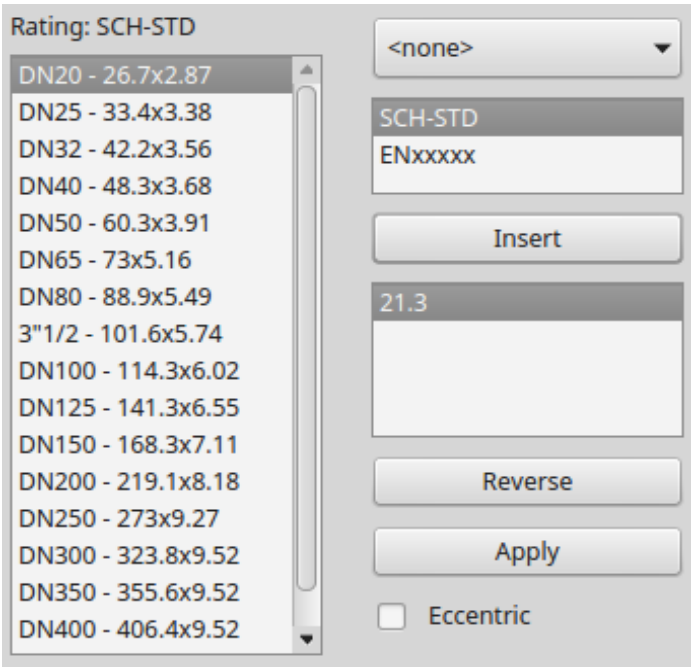
To define position and orientation following selections are possible:

- one vertex,
- one circular edge
- one pipe at one of its ends; in this case the curve's diameter and thickness will automatically fit those of the selected pipe
- a pair of edges or pipes or beams, also not contiguous but intersecting; in this case curve's properties will automatically fit to connect the two selected objects; also selected pipes will be automatically trimmed or extended to the curve's edges
- nothing; in this case the curve will be placed at origin.

If no angle is specified the default is 90 degrees.

The dial allow to regulate the orientation of elbow after it is created.

3) Insert a reduction



Opens a dialog to insert concentric reductions.

To define position and orientation following selections are possible:

- two pipes parallel (possibly co-linear)
- one pipe at one of its ends
- one pipe
- one circular edge
- one straight edge
- one vertex
- nothing (created at origin)

In case one pipe is selected, its properties are applied to the reduction.

In case two pipes are selected, the tool will try automatically to connect them with the right

major and minor diameter. Tick the checkbox to create an eccentric reduction.

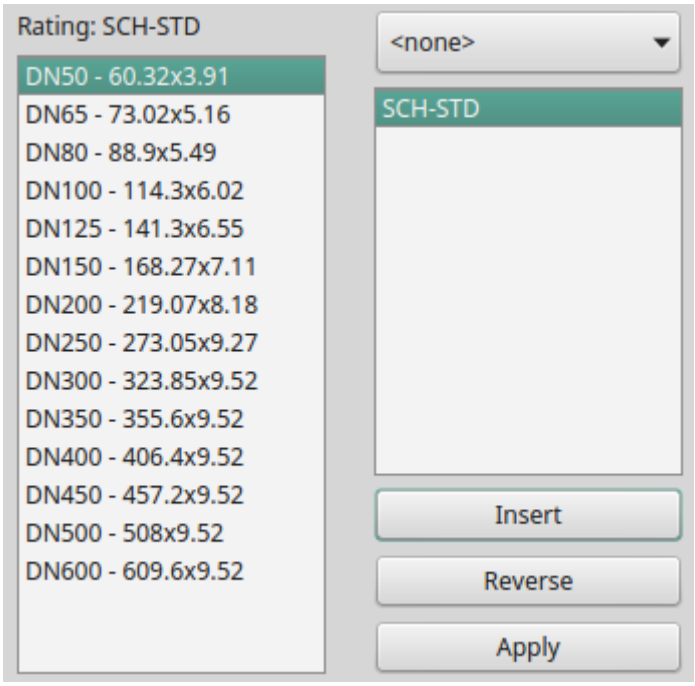
4) Insert a cap

Opens dialog to insert caps.

To define position and orientation following selections are possible:

- *one or more curved edges (axis and origin across the center)*
- *one or more vertexes*
- *nothing*

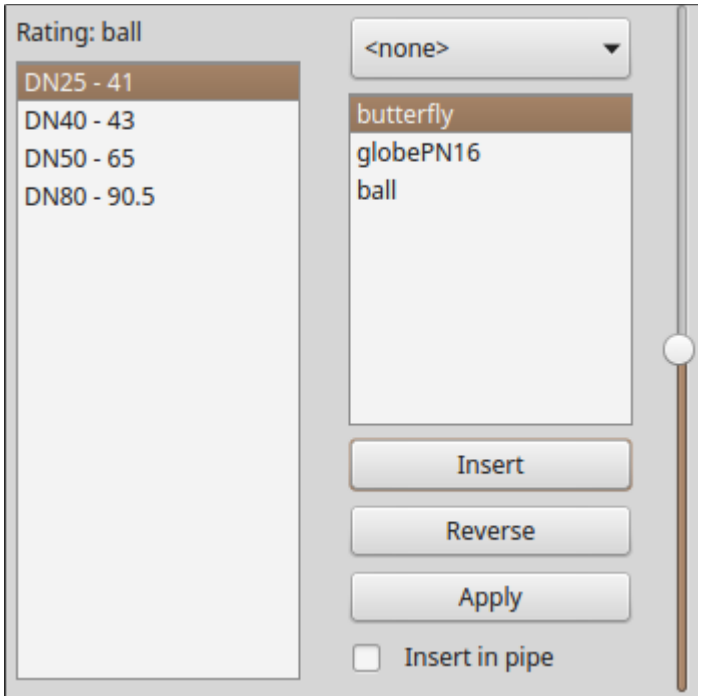
If a pipe edge is selected the caps' properties will automatically fit to those of the pipe.



5) Insert a valve

Create a "placeholder" of a valve from a .csv table like above. Beside the offset dimension, it's important because it defines also the Kv coefficient that will be used to calculate pressure losses with the relevant tool in "Utils" menu.

Note that the symbol of the placeholder changes according the type of the valve, if in its name is found one keyword among "ball", "butterfly" or "globe".



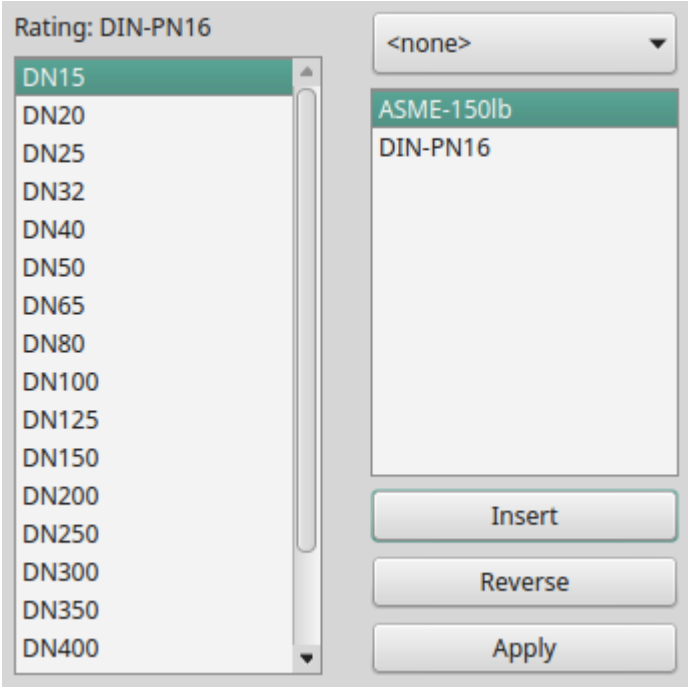
6) Insert a flange

Opens dialog to insert flanges.

To define position and orientation following selections are possible:

- *one or more circular edges,*
- *one or more vertexes,*
- *nothing.*

In case one pipe is selected, its properties are applied to the flange.



7) Insert a U-bolt

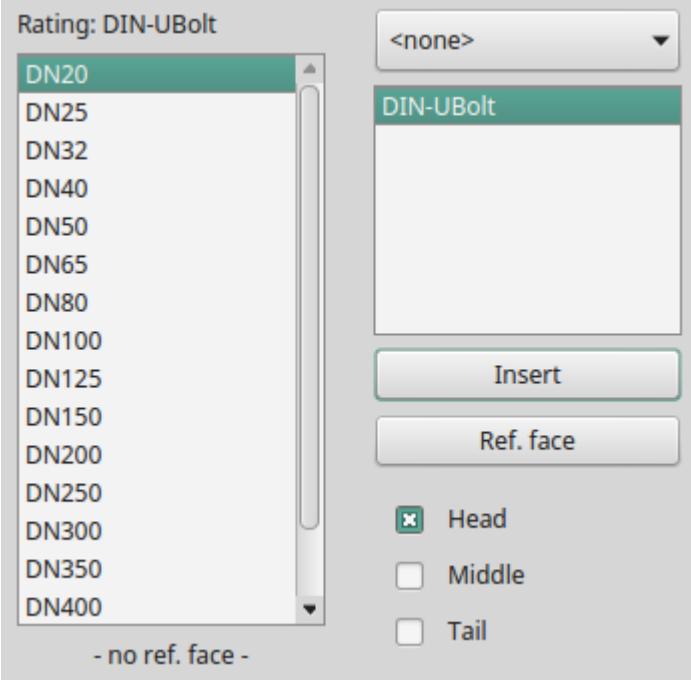
Opens dialog to insert U-bolts.

To define position and orientation following selections are possible:

- *one or more circular edges*
- *one or more pipes*
- *nothing.*

In case one pipe is selected, its properties are aplied to the U-bolt. Moreover it's possible to choose to place the U-bolt at the **Head** or **Tail** ends or in the **Middle** of the pipes by checking the relevant box.

With **Ref. face** button it's possible to select the face of the support to which to orient the U-bolt axis.



8) PypeLine Manager

Before talking about the dialog it's worth to recall what the pype-line object is in the context of Flamingo workbench. This object represent a collection of objects "PType" that are updated with the methods defined in the Python class itself.

At present time it creates, with the method "obj.Proxy.update(obj,[edges])", pipes and curves over the given edges and collect them in a group named according the object's obj.Label.

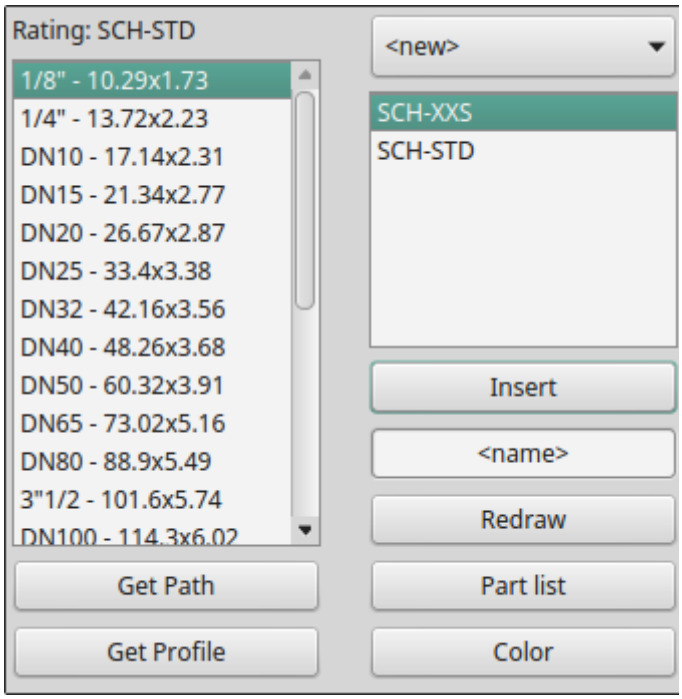
A standard bending radius "3D" (i.e. 1.5xO.D.) is applied for curves. The Bend Radius is a common property of object pype-line, thus it can be changed and then redrawn.

When the Label of the object pype-line is renamed, the name of its group is changed accordingly.

The class PypeLine2 has the optional attribute ".Base", which namely represent the centerline of the piping:

- If Base is None, PypeLine2 behaves like a bare container of objects, with possibility to group them automatically, assign one color and extract the part-list.
- .*Base* can be a Wire or a Sketch or any object which has edges in its Shape.
- Running "obj.Proxy.update(obj)", without any [edges], the class attempts to render the pypeline (*Pipe* and *Elbow* objects) on the "obj.Base" edges: for well defined geometries this usually leads to the desired result. If [edges] are given, pipes and curves will be drawn along them.
- Running "obj.Proxy.purge(obj)" deletes from the model all Pipes and Elbows that belongs to the pype-line.
- Remember that the object created outside the .Base won't be updated when the .Base is changed and the pypeline is redrawn and (except pipe and curves) won't be deleted if the pype-line is purged.

This understood, the command opens the dialog to create or modify one pype-line.



The dialog is very similar to those for insert other objects seen before.

The pipe ratings tables, where the O.D. and thickness are defined, are the same of those for tubes (e.g. Pipe_SCH-STD.csv).

When <new> is in the combo and **Insert** is pressed, a new pype-line object is created in the document with the relevant group.

It is possible to create one pypeline in three ways, according to the objects selected in the viewport when **Insert** is pushed:

- *nothing is selected*. One pype-line is created with property .Base = **None** and included in its group with the specified name and color (or default values). The piping objects to populate it can be created one-by-one with the commands seen above or alternatively a centerline can be selected afterwards with **Get Profile** and **Redraw** buttons.
- *one DWire object is selected*. It is automatically taken as Base and converted in a Path (orange, dash-dotted) and pipes and curves are drawn along it.
- *a set of edges are selected* (even not contiguous but anyway having intersections extending their ends). One Path is created connecting all the edges (see the Path tool in the Frame toolbar) and assigned as .Base to the newly created pype-line. Then pipes and curves are drawn on it as above.

After that it's still possible to add other objects (such as Flange, Reduct...) using the relevant insertion commands described above. When objects are created within a pype-line they are automatically included in the relevant group of the model and the common properties (i.e. O.D., thickness, color, bending radius etc.) are applied.

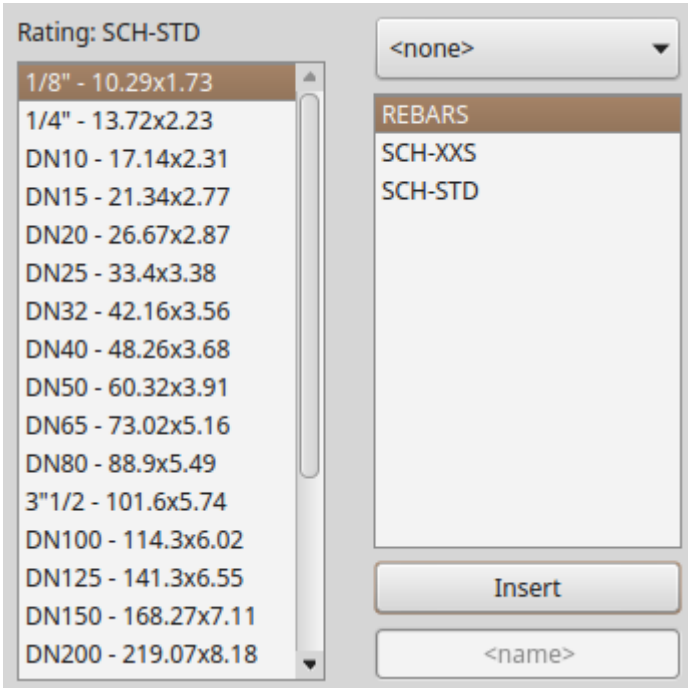
If at least one pype-line is already in the model, that can be selected from the combo-box: in this case, pushing Insert creates the pipes and curves like described above but, instead of creating a new pype-line object, it adds them to the selected existing pype-line. Beware that the piping created in this way will be deleted at next Redraw.

Get Path, Get Profile and **Color** allow to change the .Base property, the nominal size and color of the object respectively.

Redraw re-create tubes and curves along the .Base (if defined) after any modification to the path or the properties of the pype-line.

Part list generates a .csv file with the bill of material of the piping object included in the pype-line selected in the combo.

9) Insert a PypeBranch



This pype object behave like a PypeLine except it automatically updates whenever the Base (a DWire or a SketchObject) is modified: that includes changing the placement, stretching, moving, adding or deleting edges. It is mainly intended to represent the secondary branches of the PypeLine (see the dedicated tutorial) but it can also act as a stand-alone object.

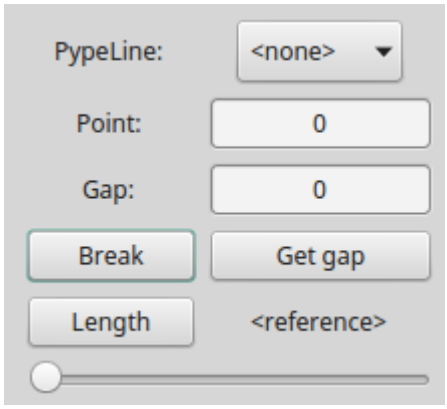
This is an important task that allow to change quickly the layout of pipes but, as a drawback, its geometry is more rigidly defined. In other words, pipes can not be splitted or resized independently because they will be eventually redrawn on the Base. Changin the OD, thk or BendRadius of the PypeBranch, instead, will apply on all tubes and curves of it.

10) Break the pipe

Opens a dialog to break one pipe at a defined point, optionally making a gap between the ends of the two parts. Multiple selection is possible.

Insert in the **Point** text-box the length where the pipe or pipes are going to break: this can be an absolute value or just a percentage of the length (a numeral followed by %). In some case it's quicker to use the slide-bar at the bottom to change this value. The **Length** button allow to measure the length of the selected pipe and use that as the reference of the slide-bar scale.

If it's needed just to break pipes in two, leave the **Gap** text-box to 0; otherwise define the length of the gap. If a reference length is choosen, also the gap can be defined as a percentage. As seen in the tutorial, it's possible to measure the gap from geometries in the model with the **Get gap** button: that's the distance between any geometric entity or even the length of a single edge.



Pushing on **Break** performs the action.

The Pypeline combo, as usual, allow to choose the group to which to assign the new objects created.

11) Mate pipes edges.

When two circular edges belonging to different objects are selected, pressing this button will make the second object move to make the edges concentric and coplanar.

This works not only with pipes.

It may happen that orientation of second object is not the desired one: in that case it's necessary to reverse it using other placing tools available in the workbench.

12) Join the pypes

Joins the Ports of different objects in a graphical way. It works only among pype-objects, also from different workbenches, where the Ports[] property is defined congruently.

13) Fit one elbow

Select 2 intersecting pipes + 1 elbow: executing this command, they will be joined. It works only among pype-objects, also from different workbenches.

14) Extend pipes to intersection

By selecting two pipes, this command extend them both to their intersection point, if exists.

15) Extend pipe to intersection

By selecting two pipes, this command extend the first to the intersection with the other, if exists.

16) Lay-down the pipes

By selecting one face and multiple pipes, this command translates the pipe along the normal of the face in order to make them lie on its plane.

17) Raise-up the support

Similar to the tool above but in this case is the support that is raised or lowered, so that the face is tangent to the pipe.

18) Attach to tube

Attaches a pype object (2, 3, 4, 5 or 6) rigidly to the nearest end of a pipe (1).

To detach, click on the button while the attached object is selected alone.

19) Create pipes point-to-point

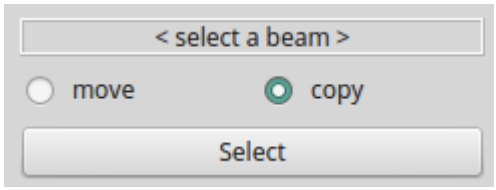
Opens a dialog similar to "Draw a DWire" together with the dialog of "Insert a pipe": this allows to draw a sequence of pipes, connected by curves, just selecting one point after the other. It is also allowed to change properties of the pipe and/or the pype-line on the fly.

FRAME TOOLS

1) Place one-beam over one-edge

Given a beam object and an edge in the model, this tool lay down the beam over the edge by selecting them one after the other until ESC is pressed.

2) Fill the frame



Dialog to create over multiple edges selected in the viewport the beams of the type of that previously chosen among those present in the model.

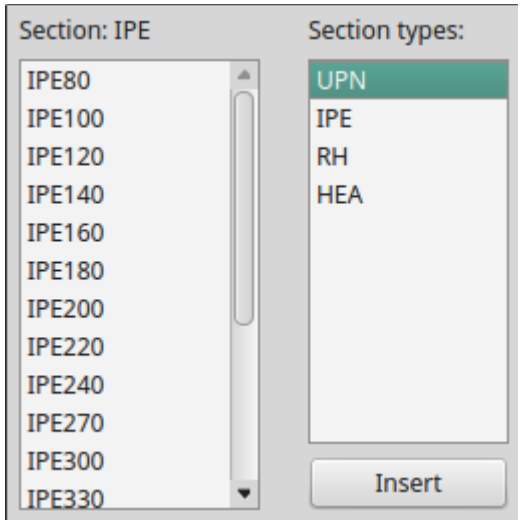
With the button **Select** it's possible to change the type of beam.

3) Insert a Path

Tool to create a continuous DWire over the path defined by the edges selected in the viewport, even if these are not touching or are intersecting in the middle or belongs to different objects. The only constraint is that exists the intersection between two subsequent edges, in the order that they were selected.

Also the DWire is given the view-properties of a center-line, i.e. orange and dash-dotted.

4) Insert Std. Sections



Dialog to create the set of profiles to be used in the model for object FrameLine.

- **Section** list: it includes all the sections defined in the .csv file corresponding to the selected section type.
- **Section types** list: the types of profiles defined with the .csv files included in the folder /tables
- **Insert** button: creates the group "Profiles_set", if not already existing, and adds to it the object of the selected profile.

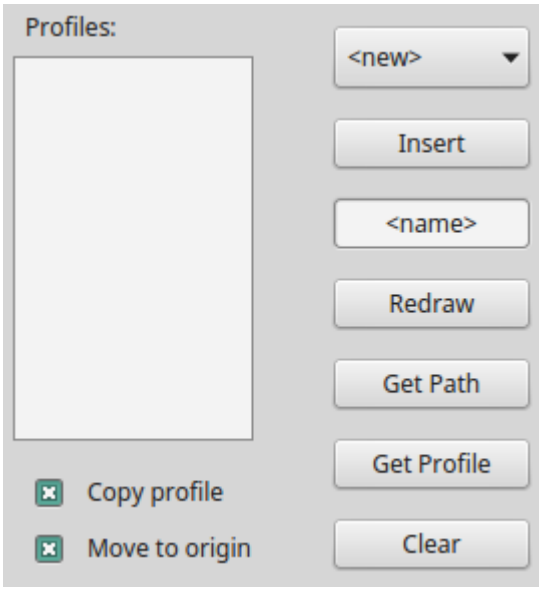
Other profiles tables can be created by adding the

relevant .csv file in the /tables folder. The rules to create or customize such tables are similar to those for pipe-lines.

Other profiles can be drafted in the model and dragged inside the group "Profiles_set".

The orientation of the DWires may influence the rendering of beams.

5) FrameLine Manager



Same as for "pype-line" objects, this is a dialog to create and change properties of "frame-line" objects.

Also similarly to what seen above, the frame-lines are objects that collects properties common to a set of beams (namely the beam's section) which are included in a common group in the tree of the model.

They also have an optional property ".Base", by default set to None, which is the centerline of the beams of the frame. After a path, alias .Base, is defined (a DWire or a Sketch) other beams can be added to the frame-line but they will be deleted when **Redraw** is invoked.

The dialog provides the following features:

- a list of beams' profiles previously included in the model by "Insert Std. Sections" dialog (read further);
- a combo-box to select the active FrameLine among those already created or <new> to create a new one;
- a text-box where to write the name of the FrameLine that is going to be created; if nothing or "<name>", the FrameLined will be named as default "Telaio00n";
- **Insert** button: creates a new FrameLine object or adds new members to the one selected in the combo-box if edges are selected in the active viewport.
- **Redraw** button: creates new beams and places them over the selected path. New beams will be collected inside the group of the FrameLine. Does not create or update beams added to the FrameLine outside its defined path.
- **Clear** button: deletes all beams in the FrameLine group. It applies also to beams added to the FrameLine outside its defined path.
- **Get Path** button: assigns the Dwire selected to the attribute Path of the FrameLine object.
- **Get Profile** button: changes the Profile attribute of the FrameLine object to the one of the beam selected in the viewport or the one selected in the list.
- **Copy profile** checkbox: if checked generates a new profile object for each beam in order to avoid multiple references in the model.
- **Move to origin** checkbox: if checked, moves the center-of-mass of the profile to the origin of coordinates system: that makes the centerline of the beam coincide with the c.o.m. of the profile.

If the name of a FrameLine object is modified, also the name of the relevant group will change automatically but not viceversa.

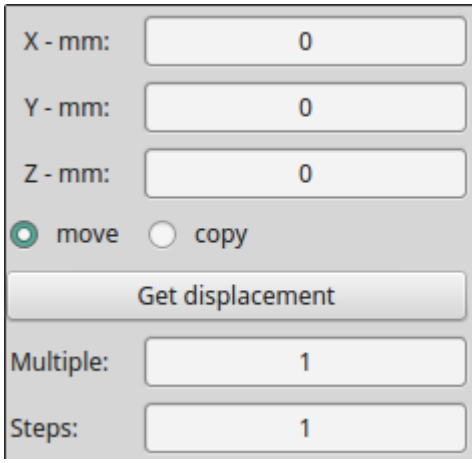
6) Spin the beam

Tool to spin one object around the "Z" axis of its shape by 45 degrees.

7) Reverse the beam

Tool to spin one object around the "X" axis of its shape by 180 degrees. If one edge of the object is selected, that is used as the pivot of rotation.

8) Shift the beam



Dialog to translate and copy objects.

X, **Y** and **Z** textboxes for direct input the amount of translation in each direction.

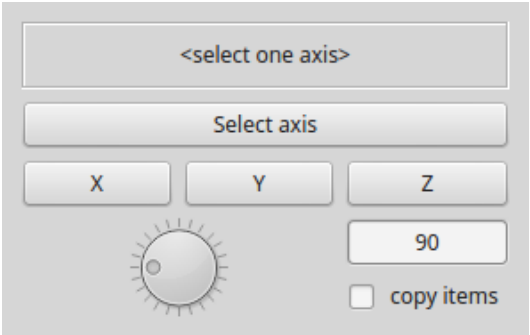
Multiple textbox is the multiple coefficient of the translation amount.

Steps textbox is the denominator of the translation amount. It's used when the amount of translation is to be covered in some steps.

Get displacement button to take the amount and direction of translation from the distance of selected entities (points, edges, faces) or even from a single edge. In the latter case, a green arrow is displayed to show the direction.

OK to perform the action and **Cancel** to close the dialog.

9) Pivot the beam



Dialog to rotate any object around any edge: the edge can be straight (axis of rotation) or curved (the axis of the curve becomes axis of rotation).

Angle textbox to insert the degree of rotation.

Reverse button to rotate in the opposite direction, if necessary.

X, Y, Z select the principal axis.

OK to perform the action and **Cancel** to close the dialog.

10) Flush the surfaces

Tool to flush the parallel faces of two objects.

Actually the command takes to the same level, respect the position and orientation of the first face selected, the center-of-mass of all faces selected. Thus it translates the objects even if the faces are not parallel.

11) Mate the edges

Tool to mate two parallel edges.

Actually the command moves the objects along the minimum distance of their selected edge to the first one. Thus it translates the object even if edges are not parallel and it's a good way to place objects in desired position.

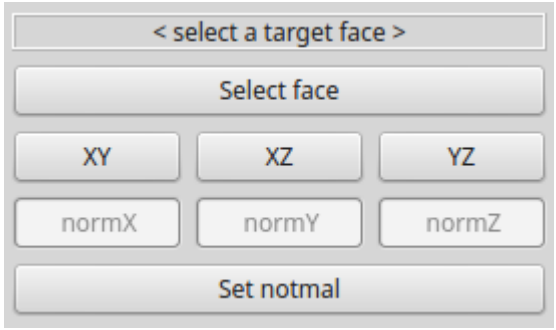
It is also possible to select two edges of the same objects. With this method is possible to move quickly one object by steps defined on its own geometry.

12) Rotate + mate the edges

Tool to translate and rotate the beams to mate two edges.

Same as above but it also makes the edges co-linear.

13) Align flanges



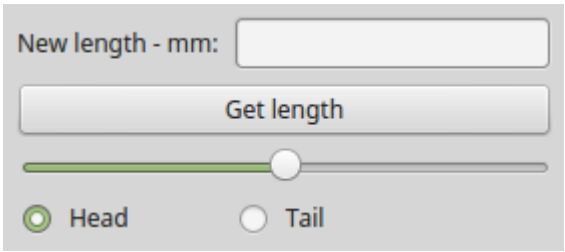
Tool to rotate beams so that their surfaces are parallel to one reference plane.

It's possible to preselect the reference face before invoking the command.

The three **XY**, **XZ** and **YZ** buttons allow to choose directly the orientation of principal planes as the reference.

Finally it's possible to enter directly the new orientation of faces by the three coordinates of the normal and the button **Set normal**.

14) Stretch the beam



Dialog to change the length of beams.

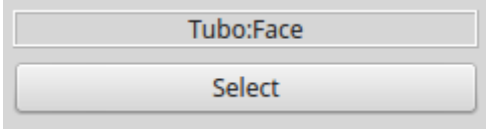
In the textbox write the new length that will be applied to selected beams or pipes.

Otherwise **Get Length** button takes the new length from the selected geometry (either the length of a beam or edge or the distance between geometric entities).

With the slider it's possible to change the length wirtten in the text-box from -100% to +100%.

Radio buttons **Head** and **Tail** allow to choose the side of the beam that will be changed.

15) Extend the beam



Dialog to extend one beam to one selected target.

If entities are preselected before calling this command, the first entity is automatically taken as target and the object attached to it is removed from the selection set. In any case it's possible to change the target object with the push-button **Select**.

16) Adjust frame angles

Tool to adjust the beams at square angles of frames. To understand at best how it works, refer to the previous tutorial.

UTILITIES

1) Make a polygon

The first two tools of utils are part of a separate project which aims to create an automatic scanner of rooms with a stepper motor and an ultrasonic distance meter.

This tool creates one regular polygon inside a sketch.

2) Polygon from file

Tool to create any polygon inside a sketch taking vertexes from a .csv file, where they are stored in polar coordinates.

3) Query the model

Tool to get various informations according to the object or objects selected. Beside length or distances, it is specifically suited to give informations related to beams and pipes (length, section, angle-between).

4) Align workplane

Tool to set the position and rotation of working plane according to the selected existing geometry.

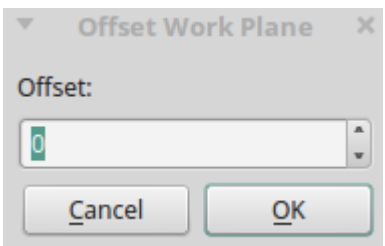
The normal of WP isdefined scanning the elements in the following order:

i. the normal of a face
ii. the normal of the plane of a curve
iii. the normal of the plane containing two segments

The origin of WP is defined (in order) by

i. one vertex
ii. the center of curvature of a line
iii. the intersection of two lines
iv. the center of an edge

5) Offset workplane

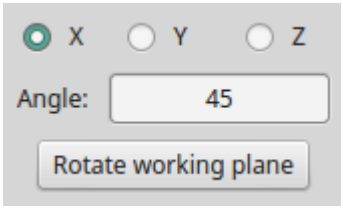


Shifts the WP along its normal vector.

To show the direction of offset, a temporary green arrow is displayed on the screen. Clearly also negative values are allowed.

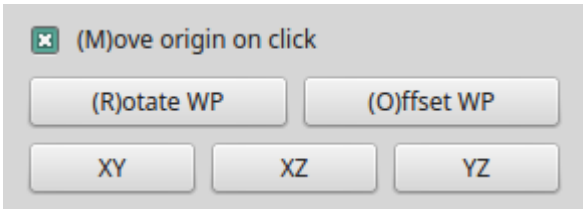
6) Rotate workplane

Rotates the WP around one of its axis. Also in this case a green arrow is displayed in the viewport to identify the present orientation of the WP: the arrow is pointed in the Z direction and the long base of the arrow is layed over the X direction.



7) Draw a DWire

This tool works exactly like the corresponding tool of Draft workbench but with few additional options at the end of the dialog.



As default, the origin of WP is redefined at each point added because this makes simpler to draw segments of known length and orientation using the snap-to-grid option.

Then two push-buttons, invoked also with the short-key Ctrl+Shift+(), allow to rotate and offset the WP as seen above without breaking the DWire object.

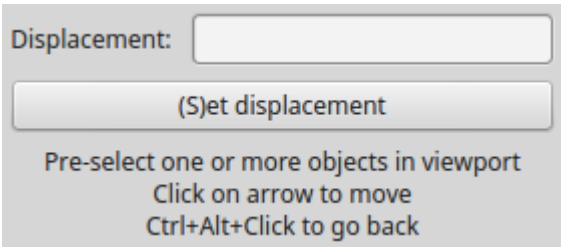
The last three buttons allow to quickly change the rotation of WP to be parallel to the principal planes.

8) Move Objects

Quickly move the parts selected by tapping on the arrow in the viewport.

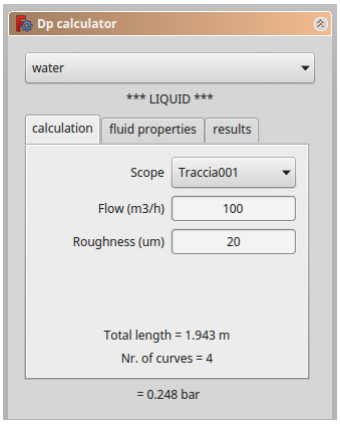
The direction of the arrow and amount of displacement (if not input directly in the textbox) are dependent by the geometry selected.

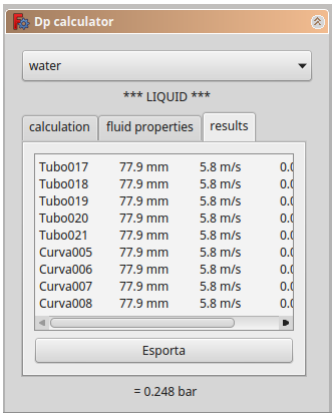
Hold down Ctrl+Alt key to reverse direction.

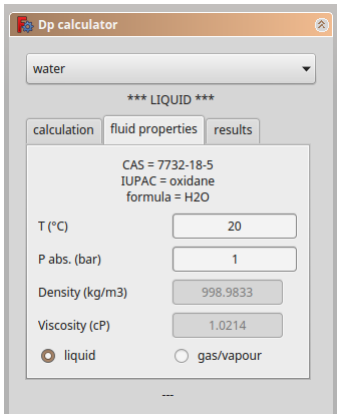


9) Calculate pressure losses

Thanks to the introduction of flow factors Kv and the use of external python modules, it is now possible to estimate pressure losses of various fluids across the pipes and valves directly from the 3D model.







See the dedicated tutorialPype3.pdf for details.