

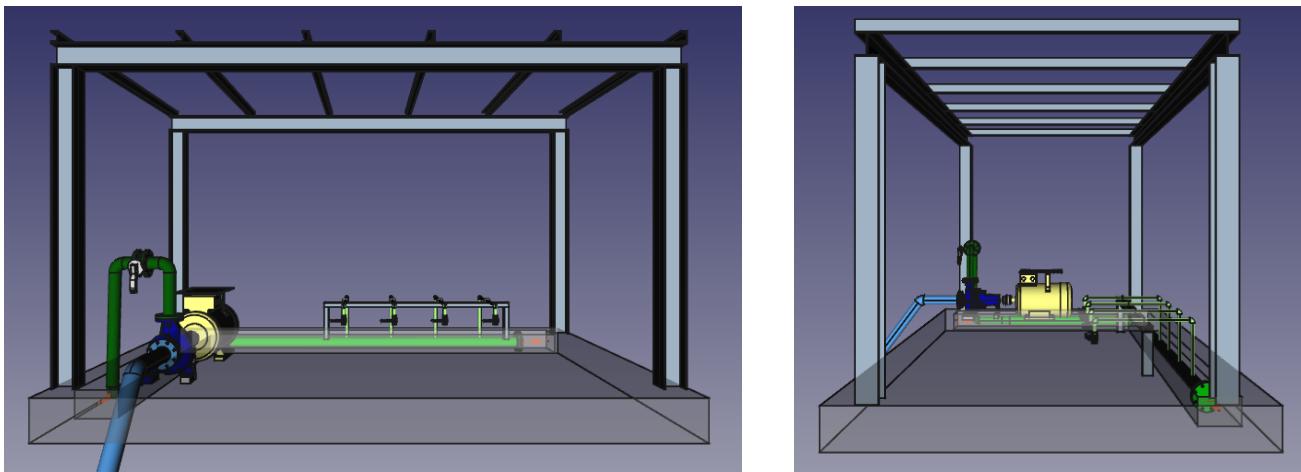
Next tutorial illustrates the latest features of the workbench.

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 management of frame-lines / pipe-lines .

TUTORIAL

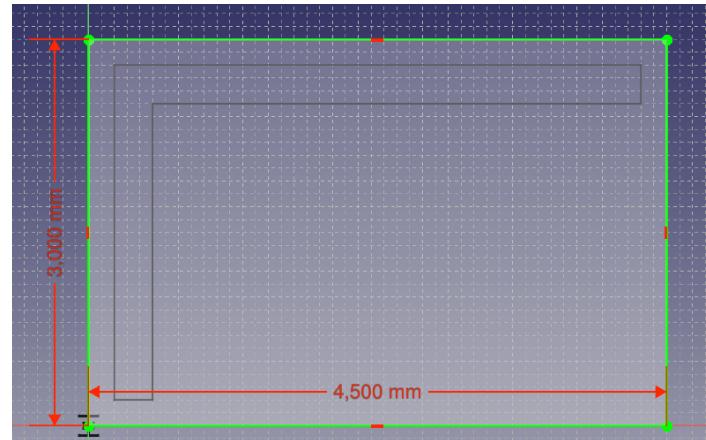
Task is to draw a kind of water pumping unit like the one below.



1. CONCRETE BASE

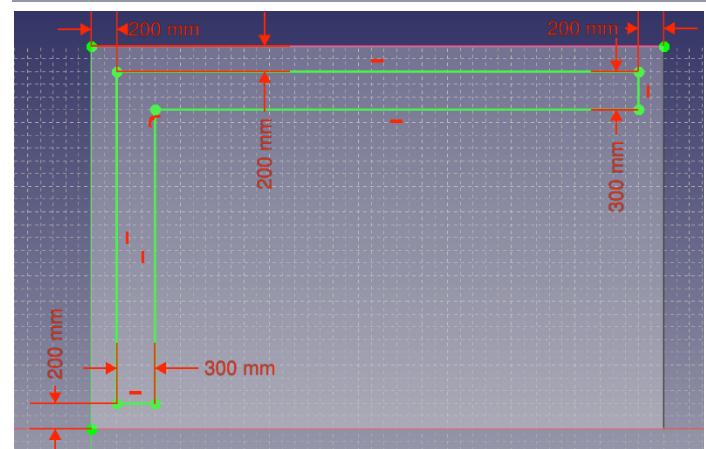
Create a sketch with Sketcher and draw the outline of a rectangular concrete basement, 4.5 mt wide and 3 mt high.

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.

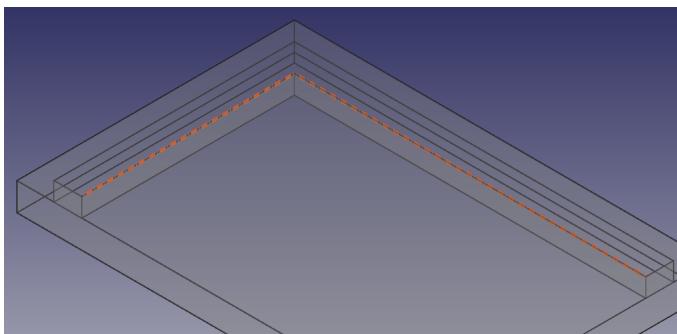


To see better the following operations, set the transparency of this object to 80%.

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.



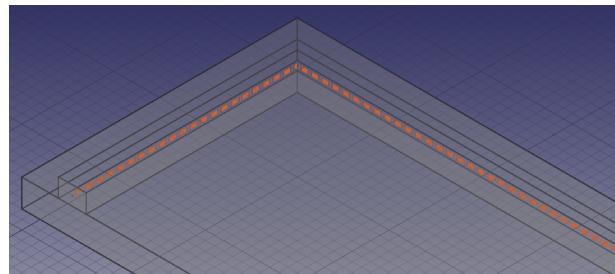
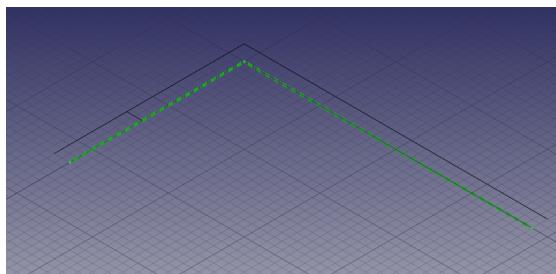
Press the "create path" button. This creates a nice orange and dash-dotted wire that will be used as the centerline of pipe to be created next.

Note: this step is not strictly necessary, because it's also possible to insert a pipe-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.

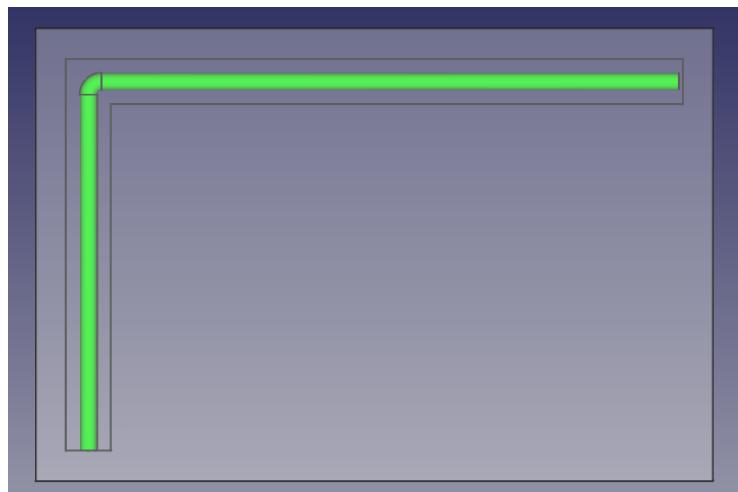


Use the Offset tool (150 mm outside) from the Draft workbench and the shiftTheBeam tool (100 mm down) from Flamingo workbench to center the DWire in the middle of the channel.



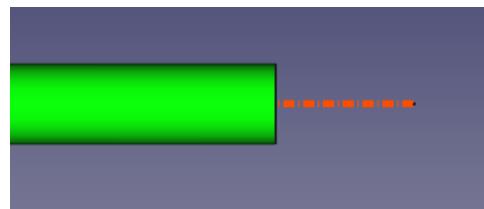
Select the centerline in the viewport and call the Pipeline manager from the "Pype tools" menu or from the corresponding toolbar.

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 pipeline to be created (e.g. "main"). Finally the pipeline 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.



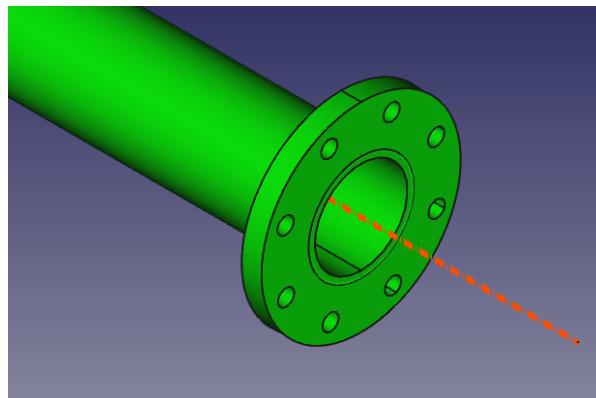
You might have also changed the length with the horizontal scroll-bar, if you wished to make it more roughly.



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 pipeline 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 pipeline selected above in the combo.

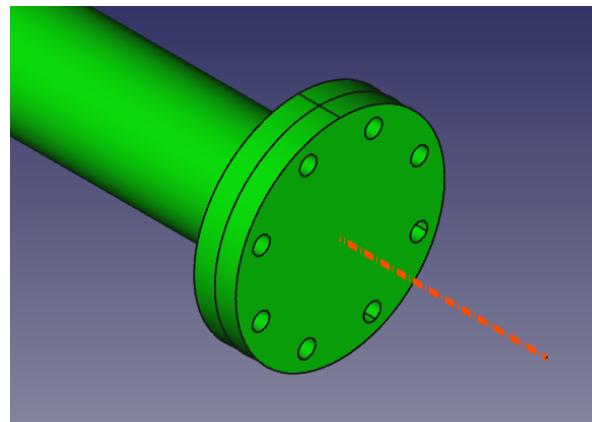


"Insert" to create the flange.

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

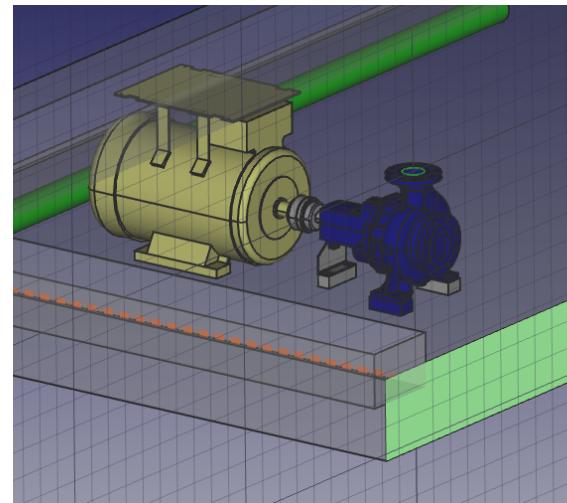
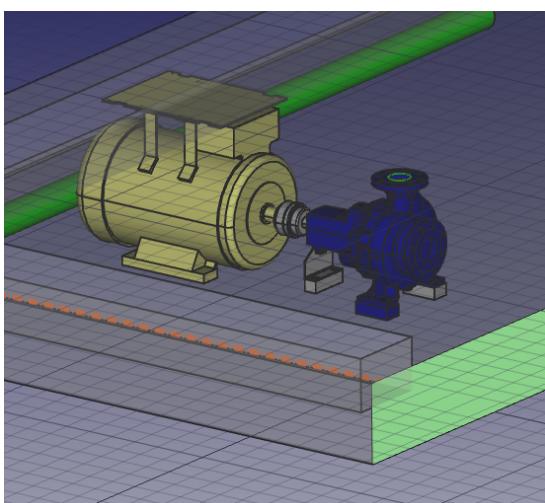
As said at the beginning, improvements have been done to drafting tools: that can be seen in the Utils toolbar. As before it's possible to change the working plane according existing geometry (with some additional options) but now there's also the possibility to spin or offset the working plane. Also 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.

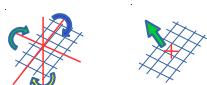


Select the circular edge of delivery flange of the pump and the vertical face of the concrete slab as shown below. Call the "align Workplane" command and WP will be centered to the circular edge center and oriented according the face selected.



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

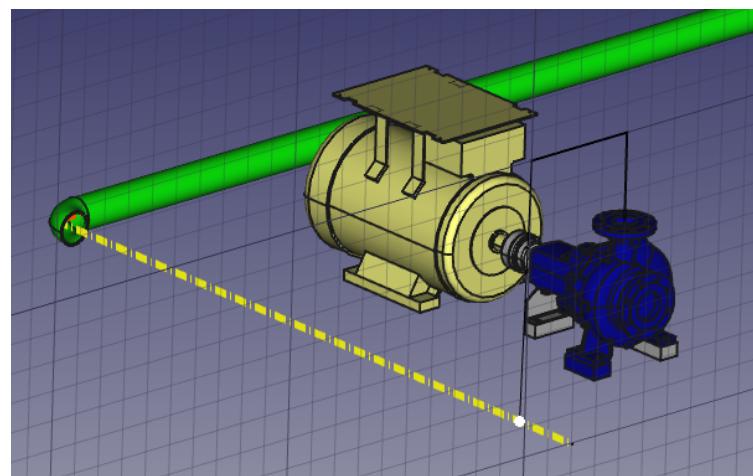


Beside that it's possible to rotate the WP around its axes or offset it along its normal. In such cases a temporary green arrow is showed to indicate the direction of translation or rotation.



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" pipe-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.

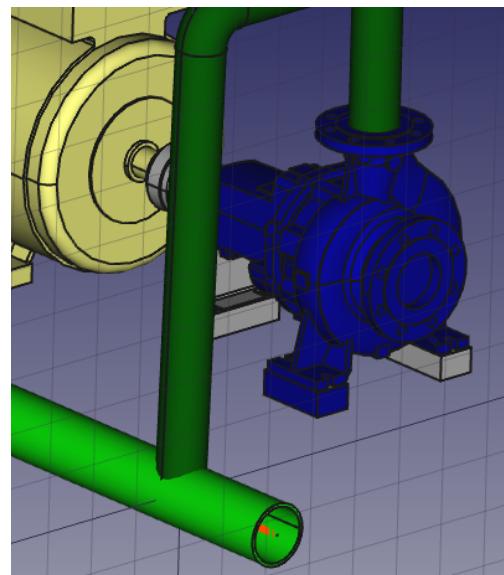
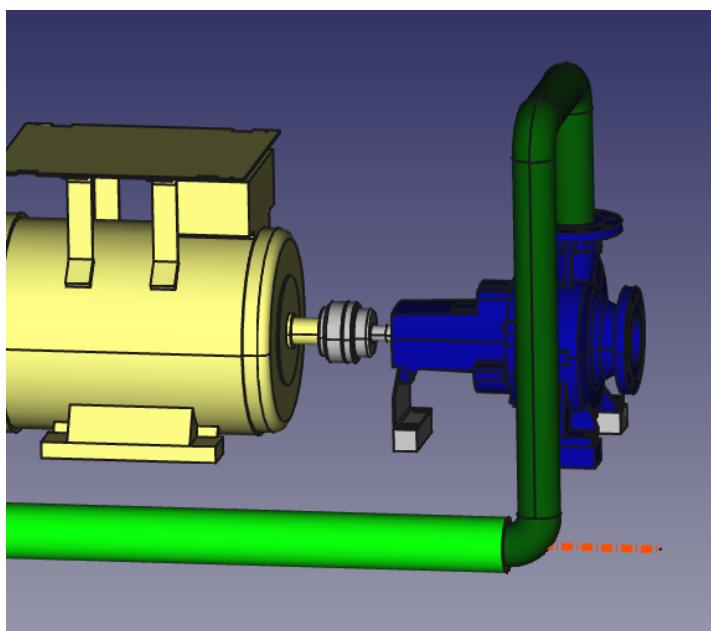
Also at the bottom of the dialog are added some additional buttons that allow to re-orient the WP without interrupting the DWire. You may call these features also with acceleration keys `<Ctrl>+<Shift>+<Letter in brackets>` (*note: the shortcut won't work if the viewport has lost focus*).



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



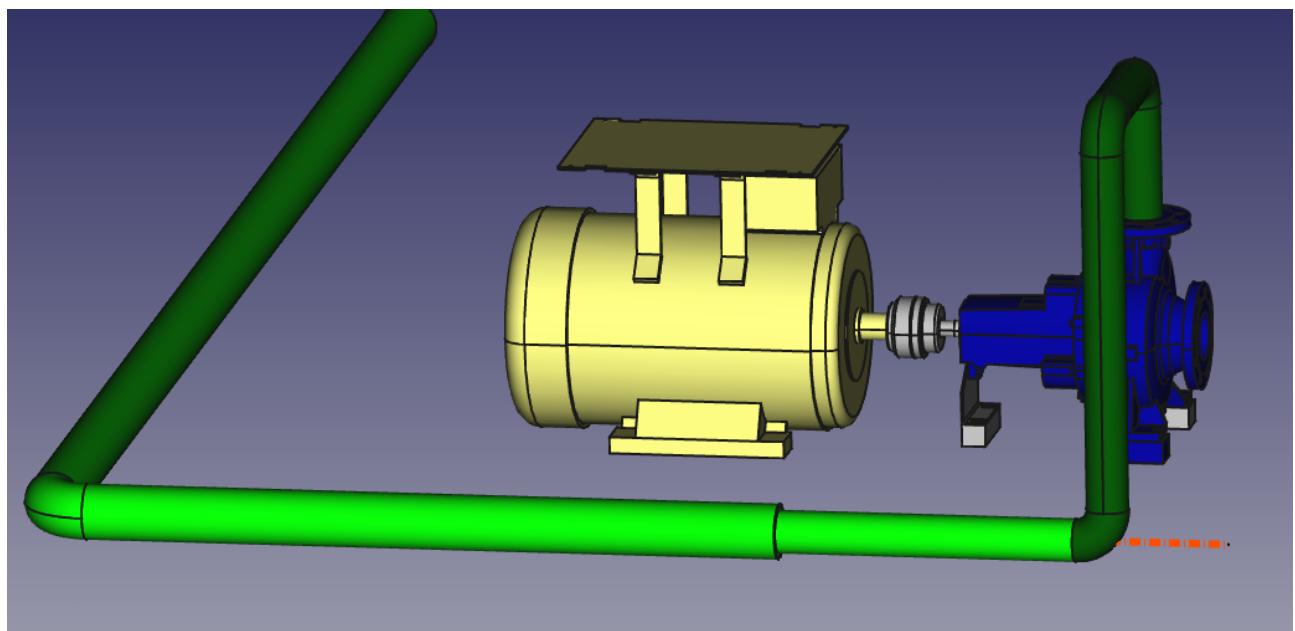
Connect the two pipes creating a curve between them: select first the DN80 pipe, so that the size will be automatically selected, and in the combo select "delivery", so that the curve will belong to that pipe-line.



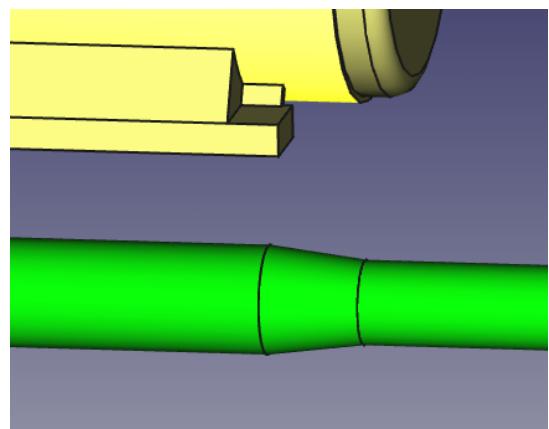
 Break the pipe so to create two sections of pipe one after the other. As the length is not important, tap on the slide bar until "Point" value is set to 30%. As an alternative it is also possible to insert the value, absolute or in %, directly in the text-box. Care that the pipeline in the combo is "main".

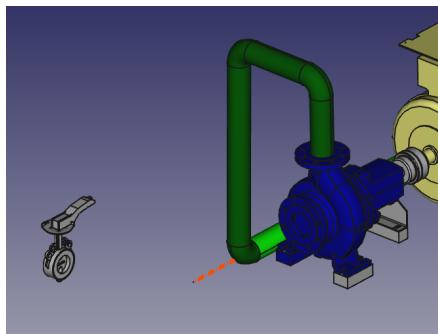


Then use the "Insert pipe" tool to change the dimension of the first section of the pipe: select the pipe in the viewport, select DN80 in the dialog and push "Apply". Result shall be like below.



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

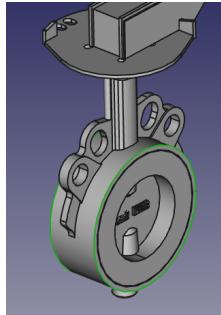




Merge into the drawing the model of a DN80 butterfly valve (I used a common wafer-type with lever taken from the same repository of the pump).

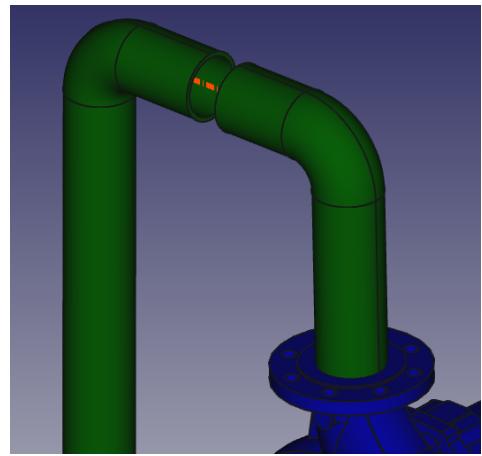


Use the same breaking tool used above but now define a gap equal to the length of the valve.



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

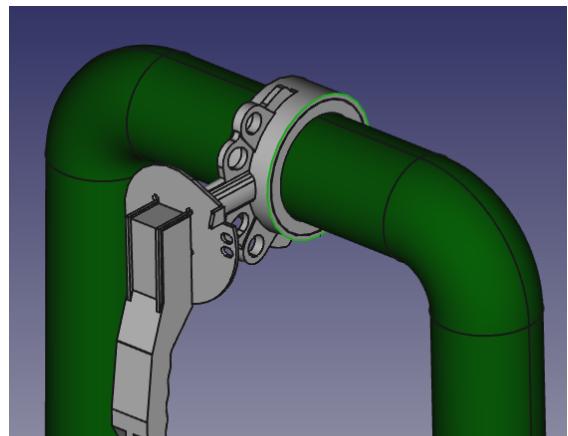
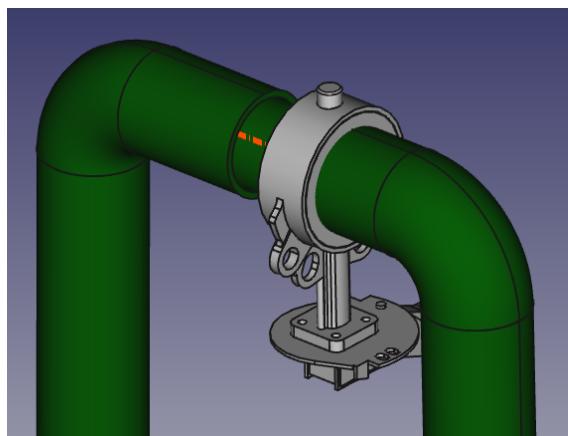
press OK.



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.

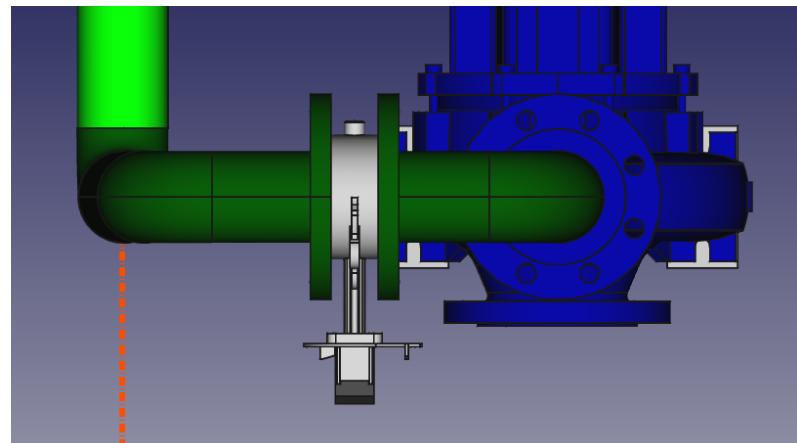


If the result is not satisfactory because the object is oriented the opposite way, for instance you have the chance to use the reversing tool as done for the blind flange earlier. Otherwise use alternative tools on the frame toolbar.





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

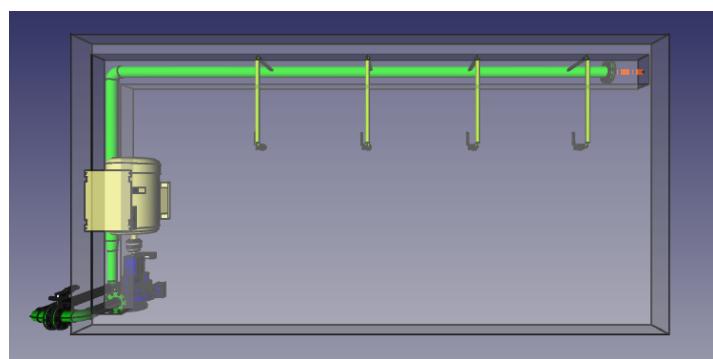
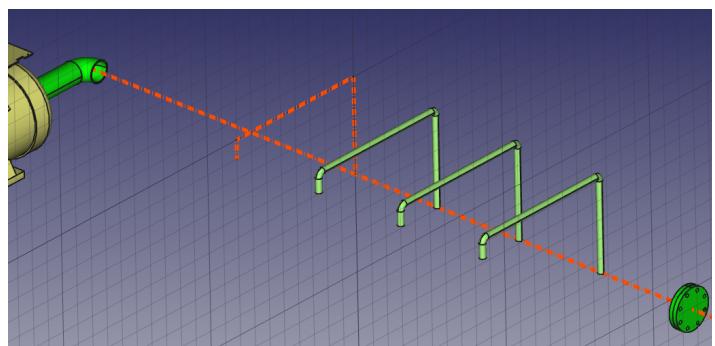


4. BRANCHES AND SUCTION

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

Only the main points:

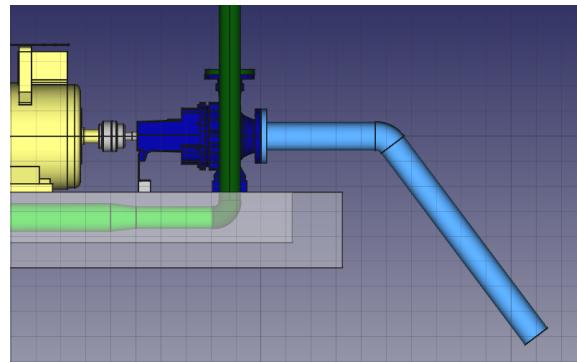
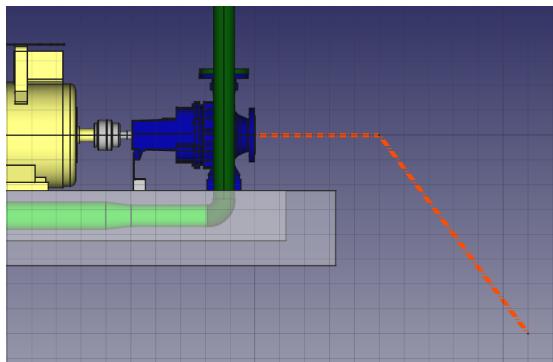
- put the workplane orthogonal to the centerline of the horizontal pipe segment of "main"
- offset it to a suitable position
- draw approximately the centerline using the snap-to-grid feature



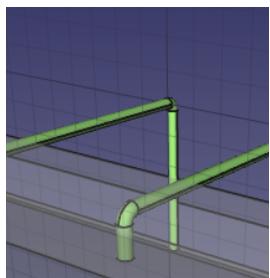
- copy the centerline 3 times along the x-axis with 500 mm distance
- create a pipeline (DN25, paler green) for each centerline
- import the model of a 1" threaded valve and place it at the free end of each branch



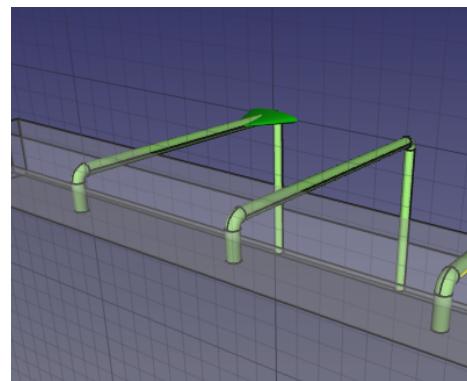
The same is valid for suction pipe (DN100, light blue).



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.

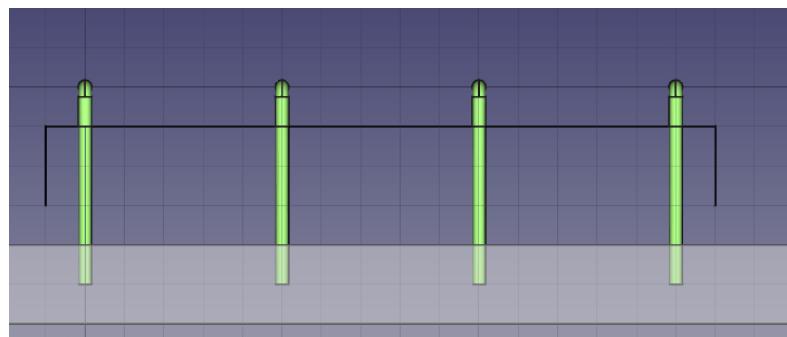


Then offset it by an amount approx. half of the length of the pipe.



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 touches 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.

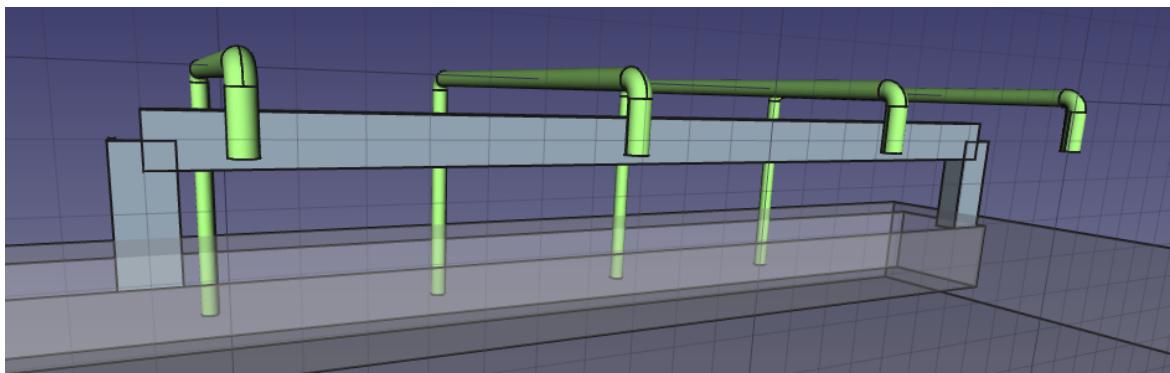


 With "Insert Std. Sections" of the frame toolbar it is possible to insert in the model the standard 2D sections that will be used to create beams afterwards. Thus from the dialog select one UPN80, that is used for the pipe support, and one HEA160, that will be used next for the structure.

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

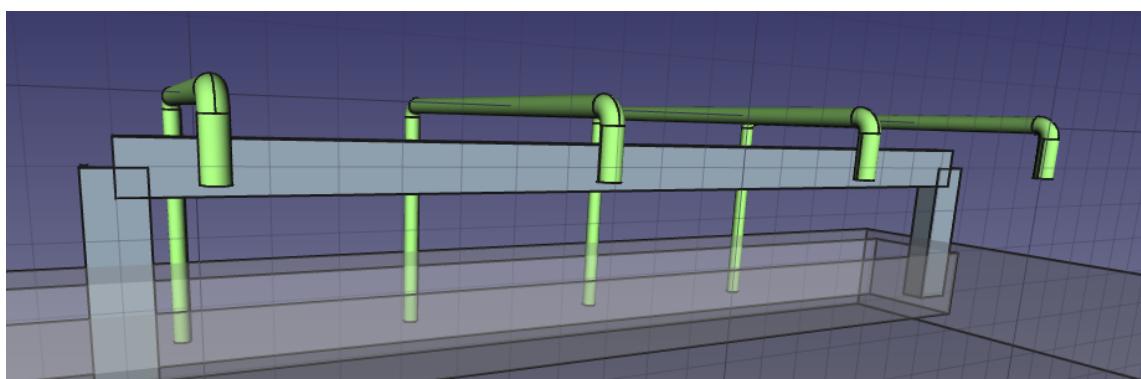
 With Frame Manager create the frame-line object and group. In the dialog, from the list of available sections select UPN80; type a name (e.g. "support") in the text-box and press insert.

Notice that in the combo at top right corner the newly created frame-line is automatically selected after it's created. 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.

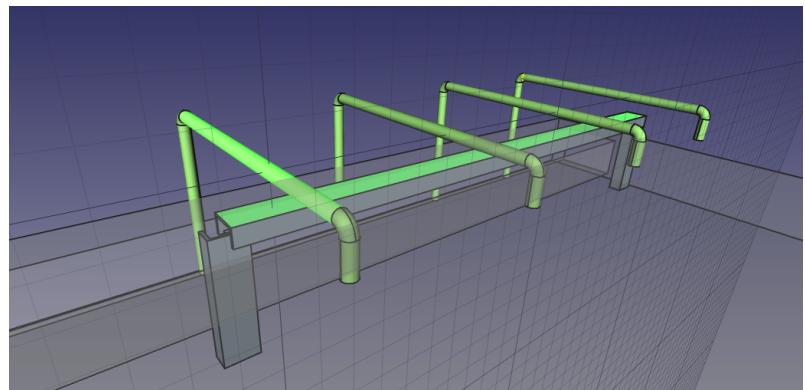
The face selected first is automatically de-selected and taken as the target to which to extend other objects: you see that also from the box in the dialog where it's written "*Object_name:Face*". So press OK to extend (or trim) the legs to the ground and press Cancel to close the dialog.



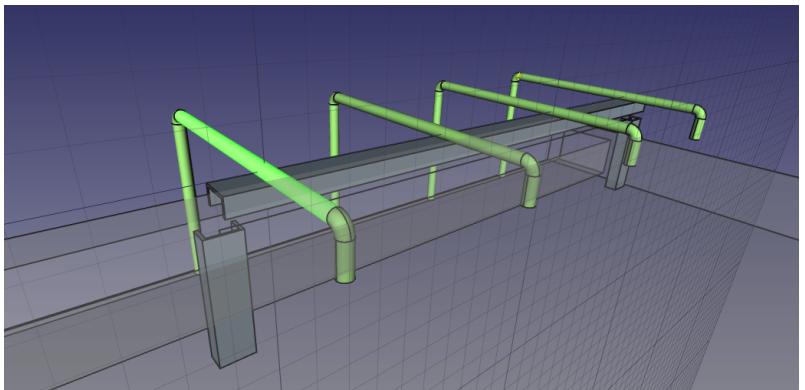
printed 1 lug 17



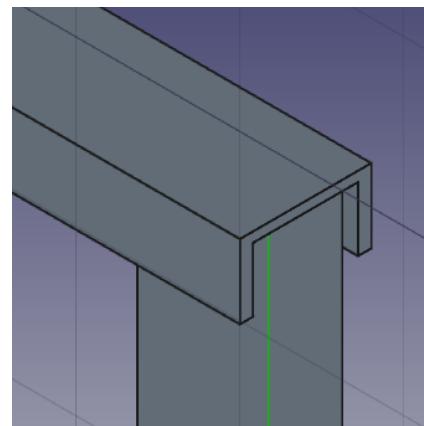
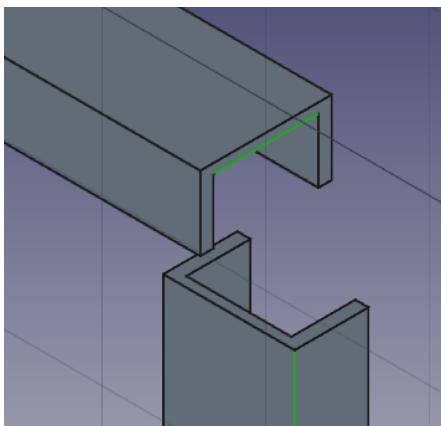
Select the horizontal beam and spin it until the wider flat face of it sees the pipes above it.



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 pipelines, 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 pipe-lines.

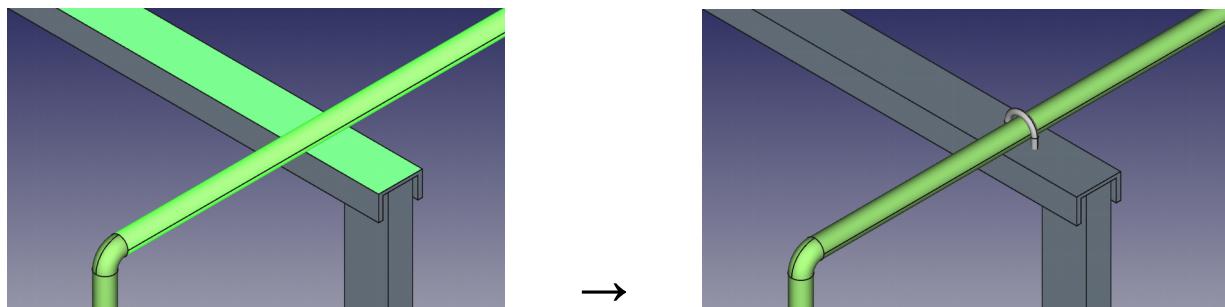


Tie the pipes to support with U-bolts.

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 pipeline that you want the bolt to belong (or <none>) and press Insert.



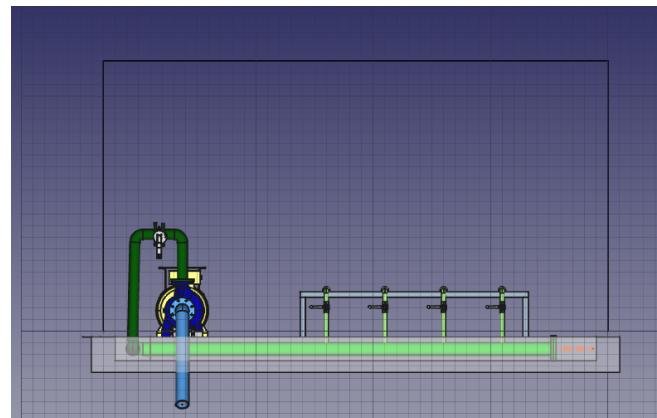
6. STRUCTURE

Finally draw the cover structure.



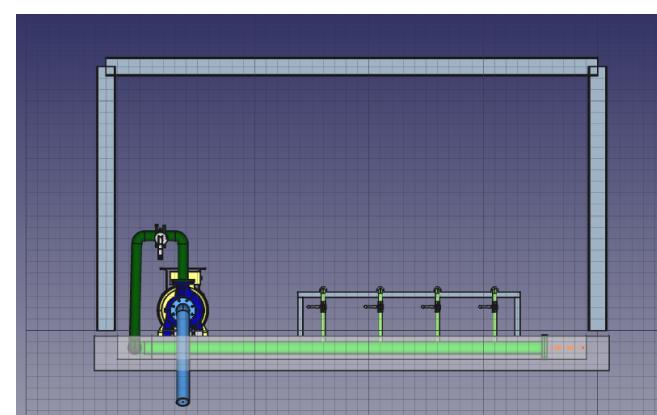
Begin with drafting a portal like did for the pipe support.

For a change, remember you can also use a Sketch object as the path for frame-lines and pipe-lines.



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

Get the path and Redraw it.

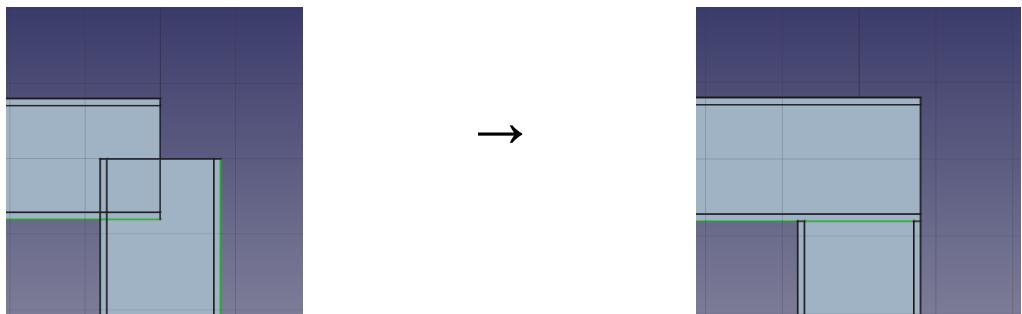


Then spin the beams around their axes to orient them as desired.

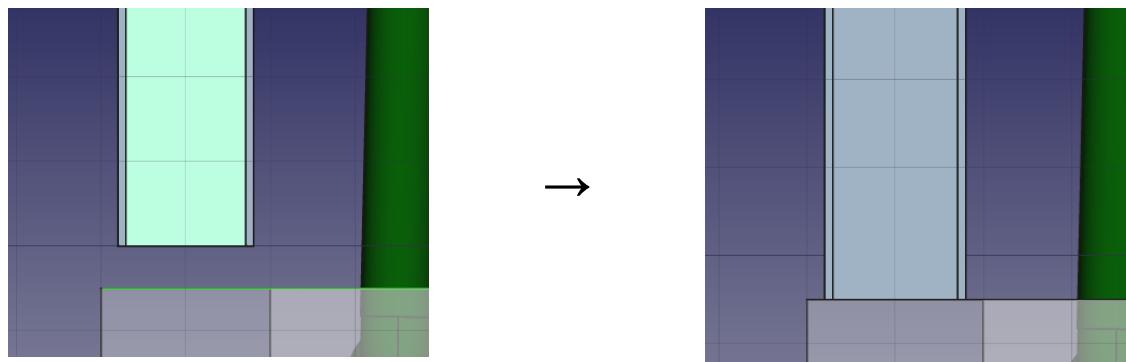
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Adjust connections at the angles...



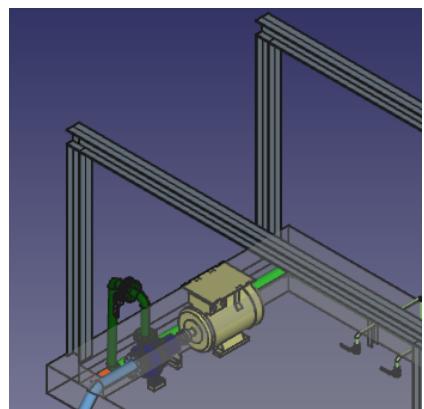
...and extend the columns to the ground.



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 pipe-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.

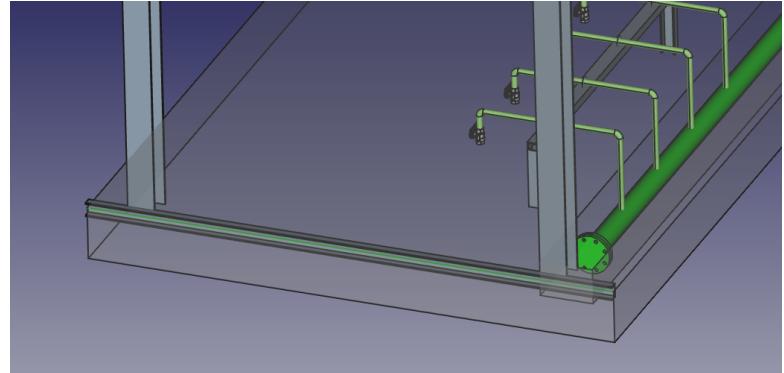


As usual open the Frame-line Manager and in the combo-box select the already created "cover" frame-line.

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

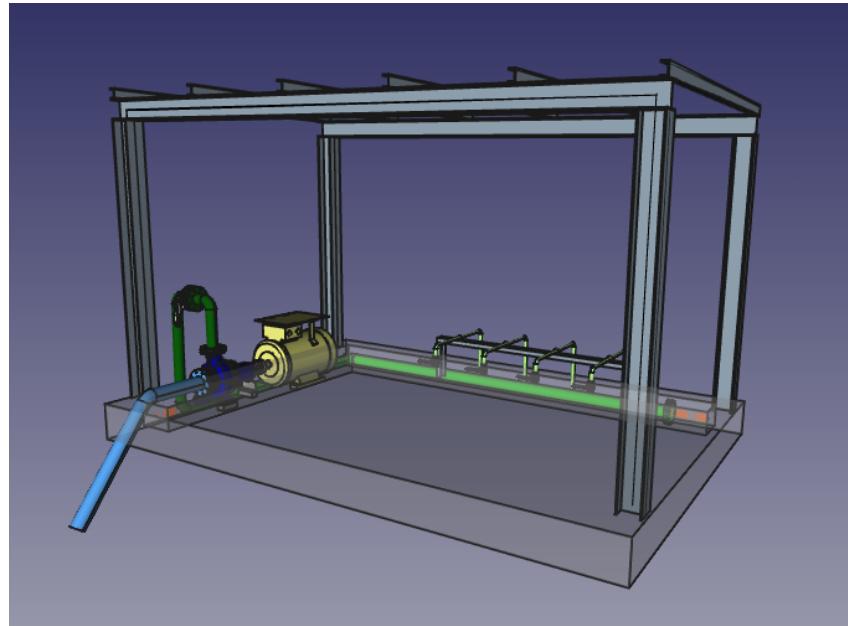
In viewport select any edge, for example the short edge of the basement.

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".

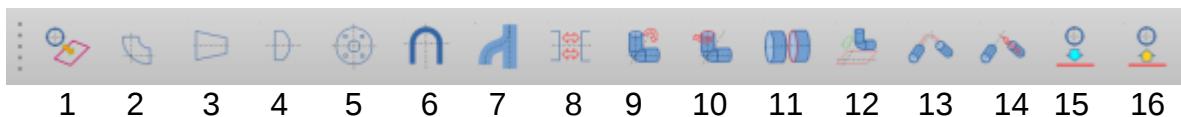
Now, to place the beam in the right place, trim it and copy it with the tools that we have learned to use should be an easy excercise... the result of which is the 3D draft below.



APPENDIX

Following explanations for each button in the toolbars.

PYPE TOOLS

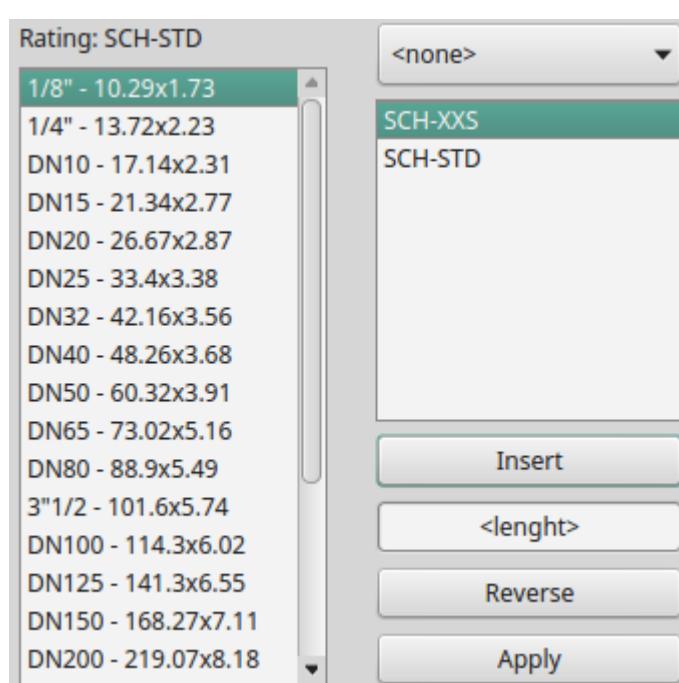


1) Insert a tube

Opens a dialog to insert tubes.

The top-right combo is a common feature for all "Insert ..." dialogs: 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 part model.

In the top-left corner is printed the currently selected pipe rating, taken from the listbox in the right column. Pipes dimensions for each piping-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 straight edges
- one or more curved edges
- 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).

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

Apply button allow to apply a different lenght or Nominal diameter to the tubes currently selected.

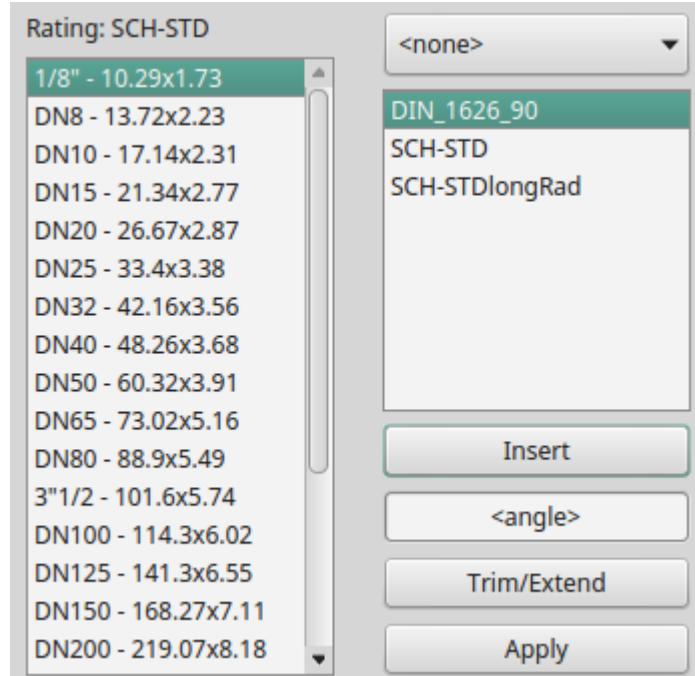
2) Insert a curve

Opens a dialog to insert one elbow.

Beside the common widgets with other "Insert..." dialogs, the **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.

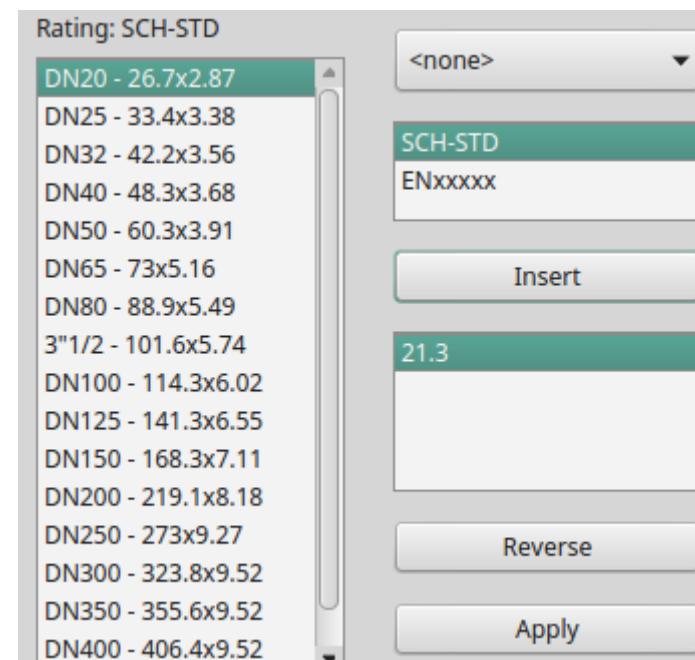
3) Insert a reduction

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.

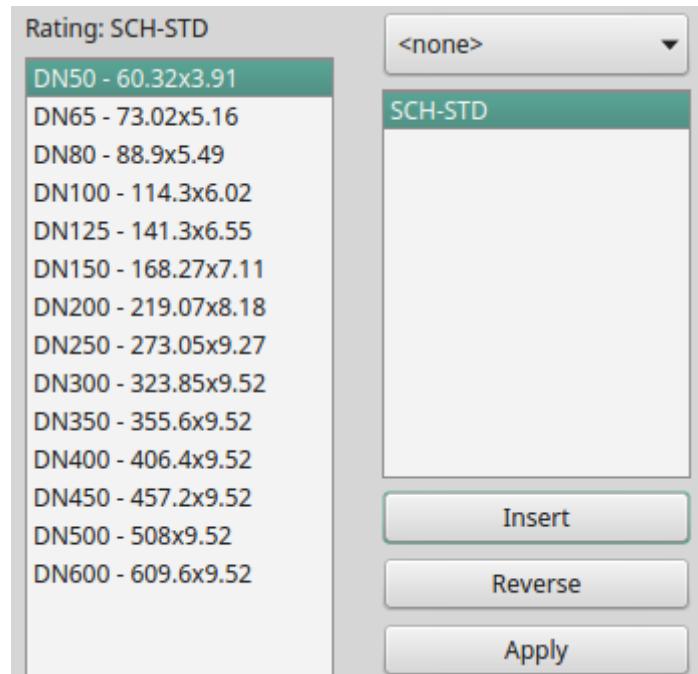
4) Insert a cap

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.



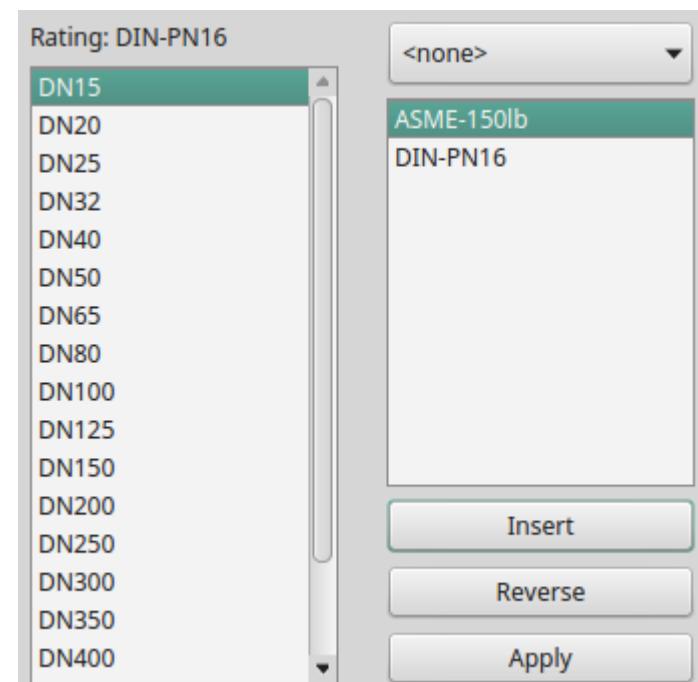
5) Insert a flange

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.



6) Insert a U-bolt

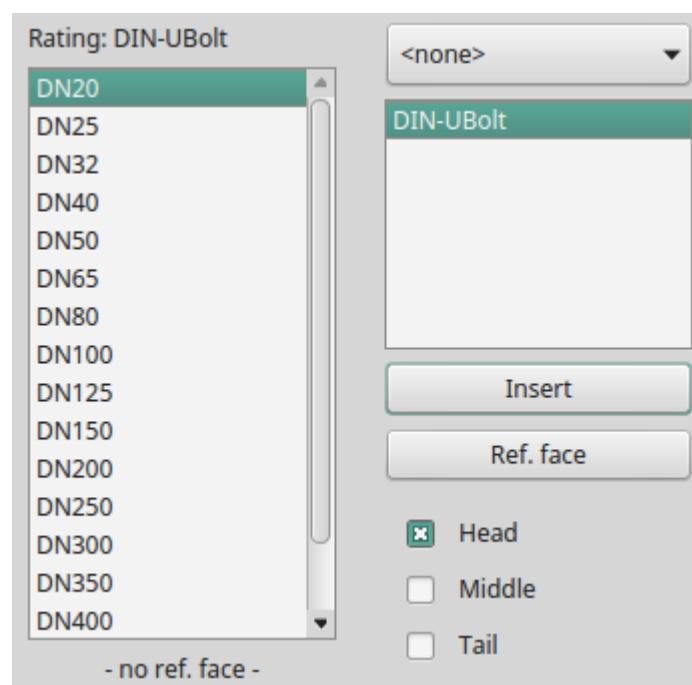
Opens dialog to insert U-bolts.

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 applied 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.



7) 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`.

It's possible to add other objects afterwards (such as Flange, Reduct...) using the relevant insertion dialogs as anticipated before. 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.

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.

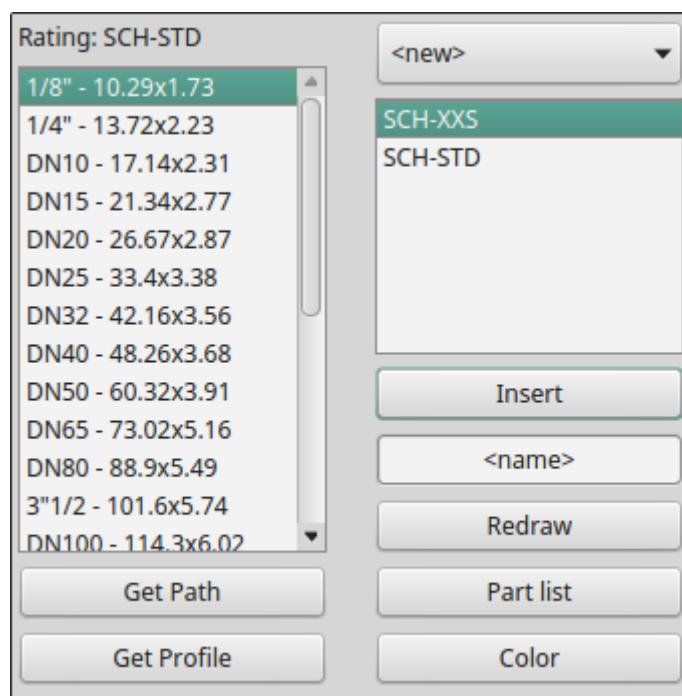
This understood, the command opens the dialog to create or modify one pipe-line.

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).

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

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



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

- *nothing is selected.* One pipe-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 intersecting, prolonging 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 pipe-line. Then pipes and curves are drawn on it as above.

If at least one pipe-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 pipe-line object, it adds them to the selected existing pipe-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.

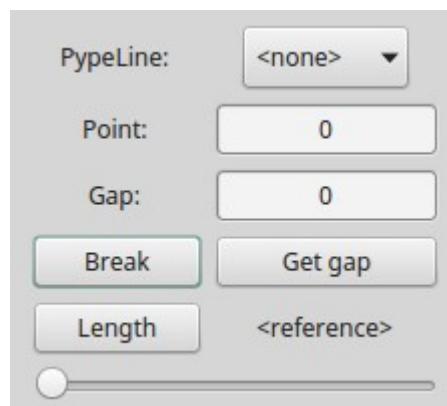
Redraw re-create tubes and curves along the .Base (if defined) after any modification to the path or the properties of the pipe-line.

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

8) Break the pipe-line

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.

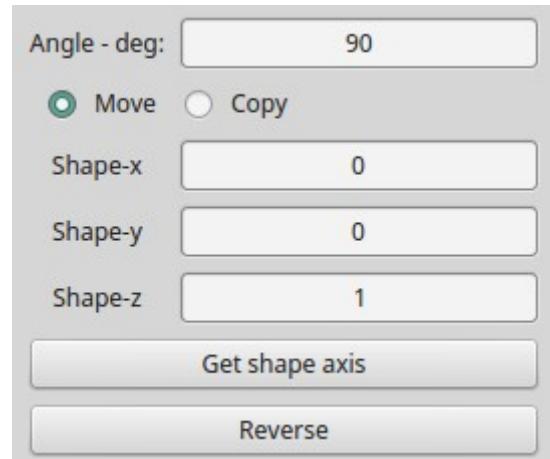
The Pipeline combo, as usual, allow to choose the group to which to assign the new objects created.

9) Rotate by axis

Opens a dialog that allow to rotate one *Part* around the axis of its *Shape*.

The axis can be defined directly by input of its coordinates or by selecting one edge + the part to be rotated and pushing **Get shape axis**.

Move/Copy radio buttons allow to choose the corresponding action. Notice that when is Copy is choosen, the original object is the one that is moved while the copy lies in the original position. That may be important in retrieving the object in the model tree.



Angle text-box defines the amount of rotation.

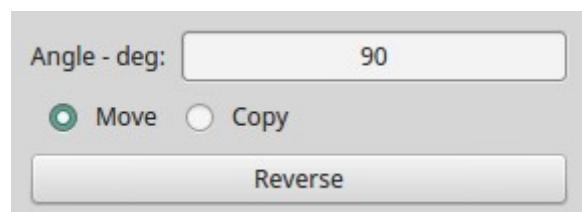
Finally press **OK** once or more time to perform the action and **Cancel** to close the dialog.

Reverse makes the rotation the opposite way round.

10) Rotate by edge

Opens a dialog to rotate any object around the axis of one of its circular edges.

The behaviour is similar to the command above but the dialog is simpler, since it's only necessary to select the circular edge of one object in the viewport and press OK to perform the action.



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) Put in the plane

(this is work-in-progress: at present time it places a curve at the intersection of two pipes)

13) Extend pipes to intersection

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

14) Extend pipe to intersection

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

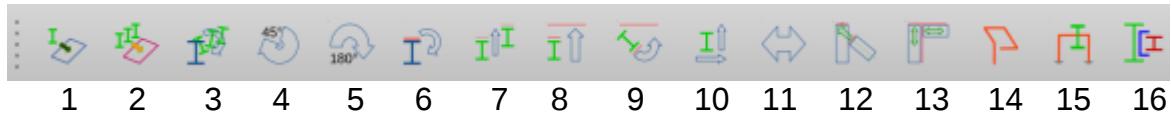
15) Lay-down the pipes

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.

16) 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.

FRAME TOOLS



- 1) *Place one-beam over one-edge*
- 2) *Fill the frame-line*
- 3) *Align flanges*
- 4) *Spin the beam*
- 5) *Reverse the beam*
- 6) *Pivot the beam*
- 7) *Flush the surfaces*
- 8) *Mate the edges*
- 9) *Rotate+join edges*
- 10) *Shift the beam*
- 11) *Stretch the beam*
- 12) *Extend the beam*
- 13) *Adjust frame angles*
- 14) *Insert a Path*
- 15) *FrameLine Manager*
- 16) *Insert Std. Sections*

UTILITIES



1 2 3 4 5 6 7

- 1) *Make a polygon*
- 2) *Polygon from file*
- 3) *Query the model*
- 4) *Align workplane*
- 5) *Offset workplane*
- 6) *Rotate workplane*
- 7) *Draw a DWire*