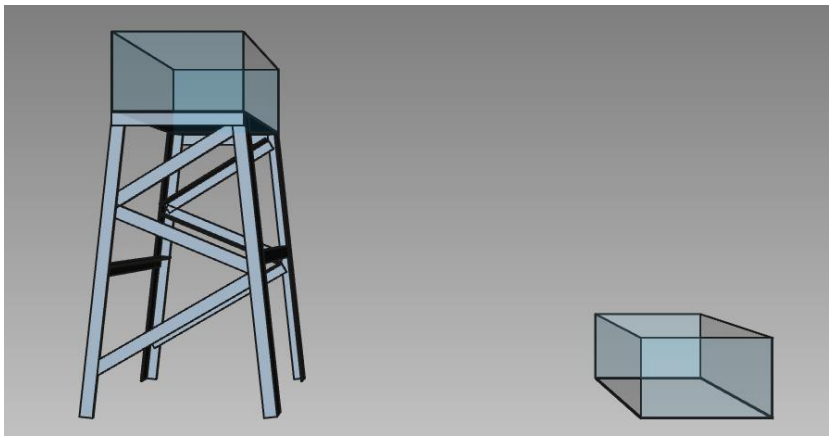
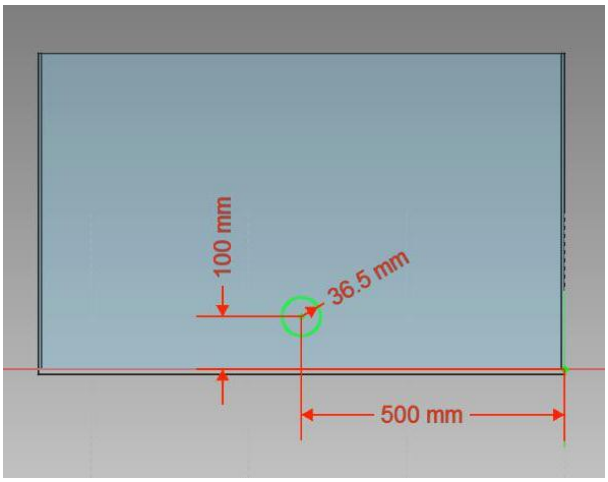
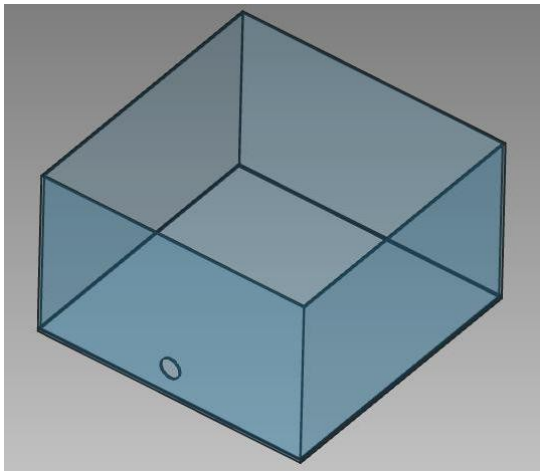
PypeTools has 14 commands now, so I think it's time for a new tutorial on that toolbar to show-up the new capabilities and a little change in workflow.

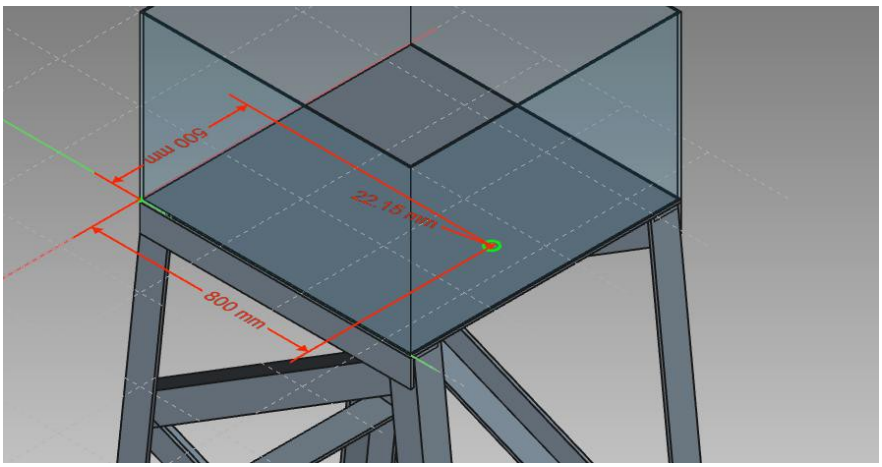


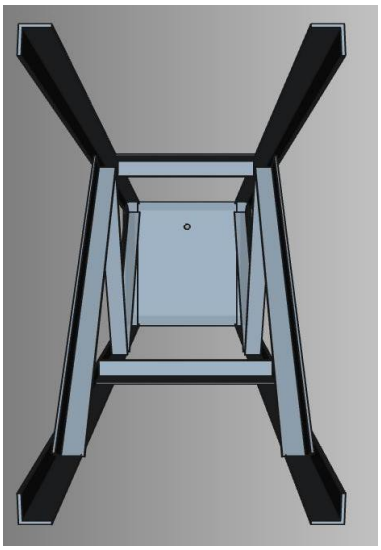
So, think that we have two separate tanks at different levels and we need to draw a pump and its pipeline to transfer the water in the lower tank up to the other tank.
To begin, it's necessary to cut the nozzles on the tanks wall. It's not necessary a special tool for this: just select the lateral face of the lower tank that stays in front of the other and create a sketch. Then make a pocket of a circle drawn as below.





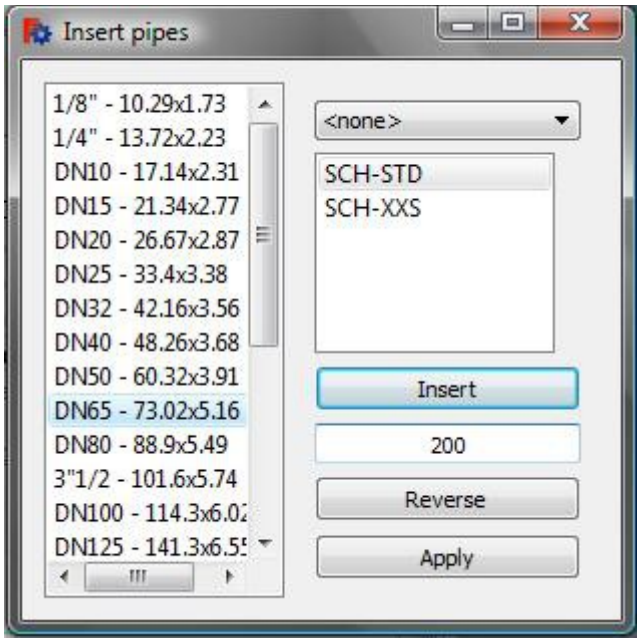
Now make the same on the bottom of the other tank:



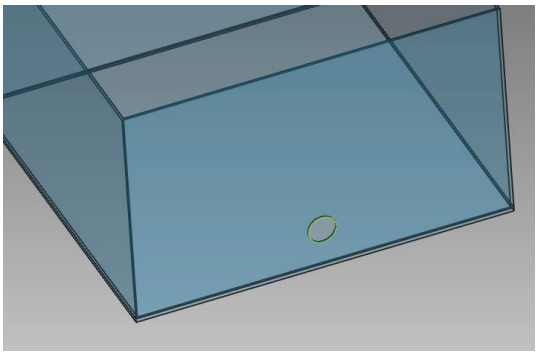


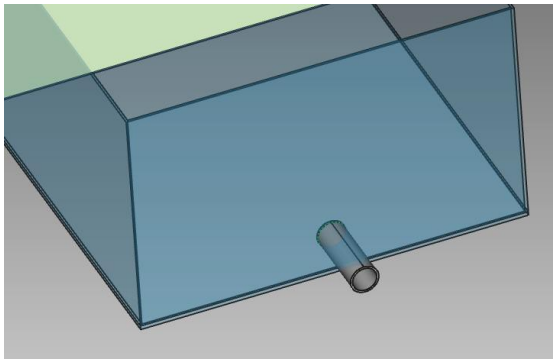
1 - SUCTION

It's possible to insert a tube at one circular edge with the command used in previous tutorial. The only thing changed is the combo-box at the top of the dialog window from which we will be able to choose the pipeline that the tube we are going to insert belongs to. At present time we still didn't create a "pypeline" object in the model so just create the pipe from scratch:

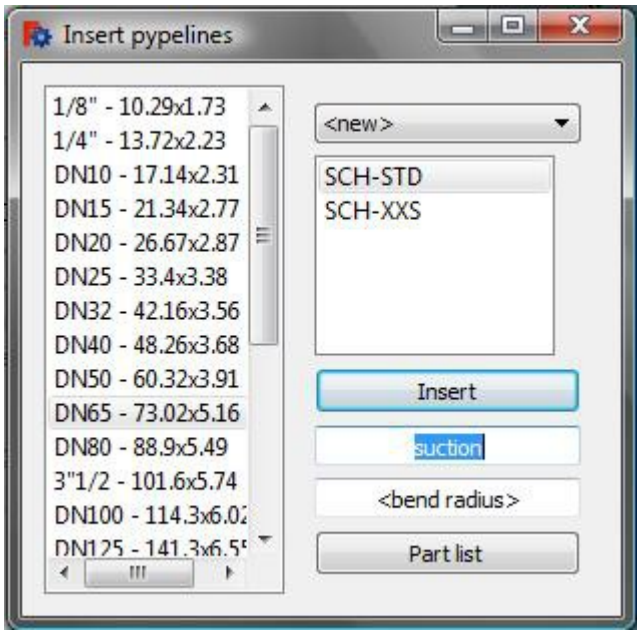


- select the edge of the hole of the lower tank;
- select the size DN65 and rating of the pipe in the dialog, without defining a length because the default 200mm will be OK for us;
- push the [Insert] button;
- if necessary, push the [Reverse] button to make the pipe extrude outside.





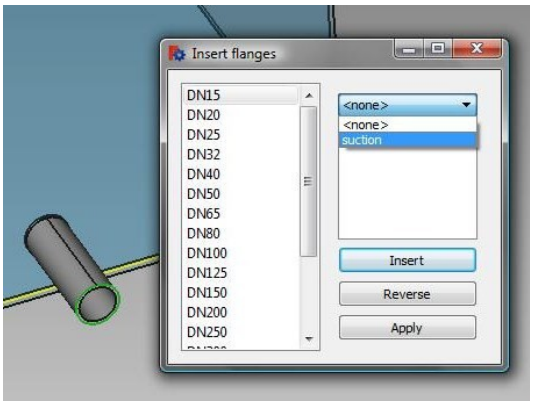
Now create the "pypeLine" object: that is practically a tag used to include all common properties of a group of piping objects; see last page for further description. It creates also a group in the model where all the components belongin to the pypeLine will be collected automatically: to do this the pypeLine name must be selected in the combo-box of the relevant insertion dialog, as told above.

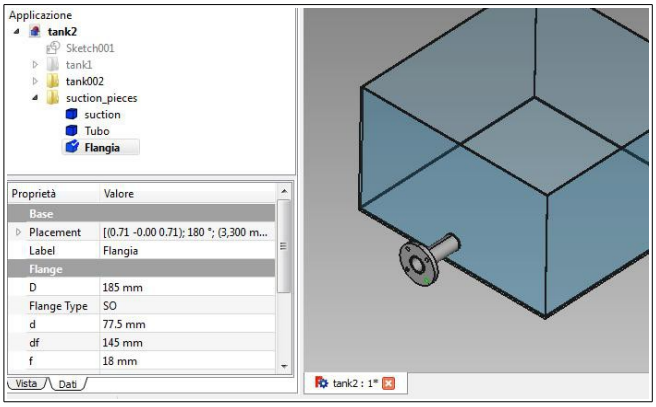


So, when the icon-button in the upper left corner of this page is pushed, a dialog similar to that used for the pipe creation appears.
The combo-box now has only the entry "<new>" because there are no pypeLines created yet. When "<new>" is selected, a new pypeLine object will be created; otherwise the pipes will be included in the existing pypeLine selected. In fact this dialog can also create the pipes and curves of the pypeline when a set of edges is selected in the viewport but now we need only to create one empty pypeLine with a significant name ("suction" - if the name is not specified, a default name will be used) and then, in the model's tree, drag the pipe that we created before into the newely created group called "suction_pieces".

Hint: [Part list] exports a kind of bill of material of the pypeline in a .csv file and you may specify a different <bend radius> for the curves rather than default "1.5xD"

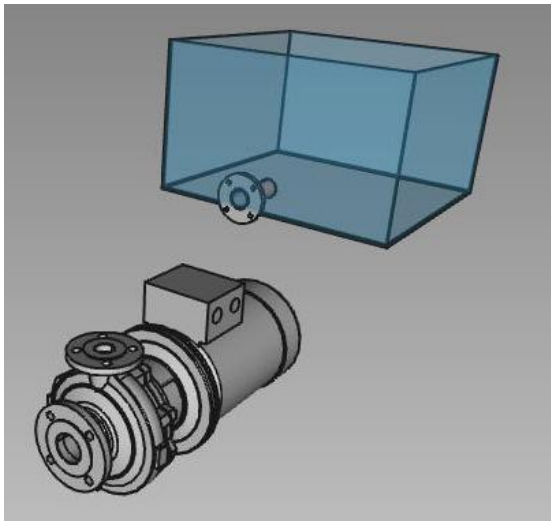
So, to complete the suction nozzle it's necessary to add one flange. This can be accomplished just selecting the edge of our pipe like in the figure below, without need of selecting the size because that will be derived from the selected pipe object. In order to make things easier, now that we created the first pypeLine object in the model, we can directly select it from the combo-box. Doing so, the new object will be created directly inside the group and we won't need to drag it in afterwards.





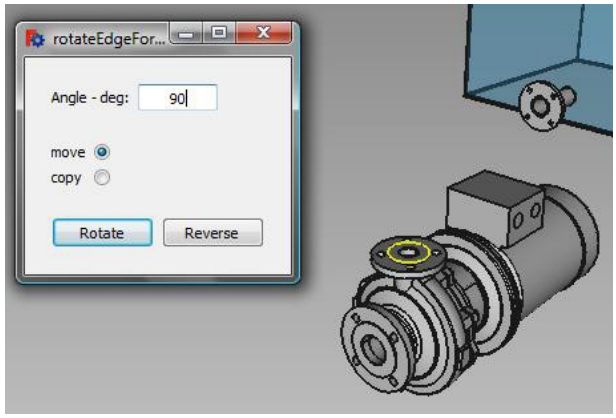
In the pictures above you see how the model tree will look like.

Hint: try to change the Label of the pypeLine object "suction" in the "Properties" list. Also the Label of the group will change accordingly.



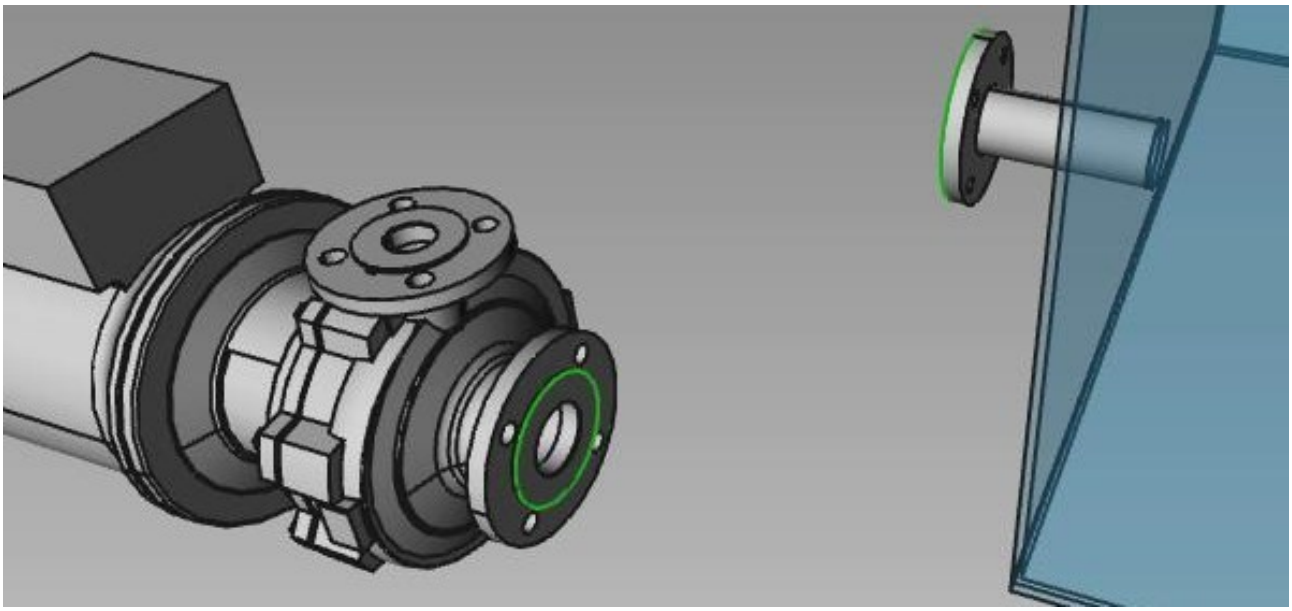
2 - IMPORTING COMPONENTS

Now to go on it's necessary to insert somewhere the model of a pump: gather that and import it from a *.step* file or merge another *.FCStd* model into this. At the end the position of the pump will likely be away from the suction nozzle that we have drawn.
For this reason the toolbar provides a few commands that will help to move and rotate the objects.

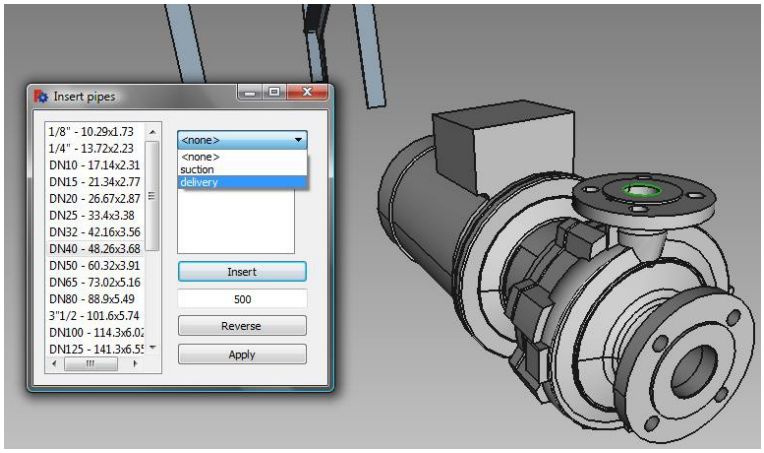


One of these tools is the "*rotate through edge*" button that rotates one object around the axis defined by one of its circular edges. In our case it's necessary to put face to face the suction flange of the pump (the one in the foreground in the above figure) with the flange of the tank. So select one of the edges of the delivery flange (the one facing upwards) and push the button with the icon like here on the left. In the dialog that appears you can input the degrees and select if the object has to be copied while it's rotated. Also, as in similar commands seen before, you have the chance to reverse the direction of rotation if it is not the one you've expected.

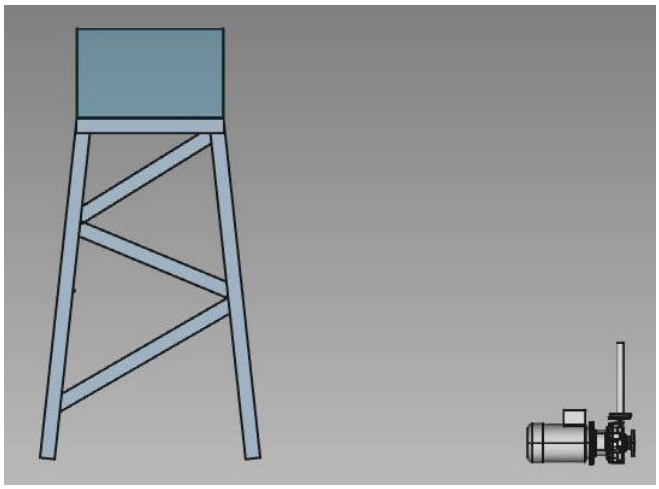
After that, we can actually mate the two flanges with "mate-pipes-edges" tool. It makes two circular edges concentric and co-planar so select first the edge of flange of the tank and then the one of the suction flange of the pump: push the button and the pump will move in the right place.



3 - DELIVERY



The suction pipe is finished and the pump is in position.
To continue the drawing its comfortable to hide the lower tank and to show the upper tank, so do it.
We may like to make the same as we did for suction pipe but this time create one empty pypeLine before, called "delivery" and then create one pipe on the edge of the delivery flange as shown to the right.
The new pipe will appear directly inside the "delivery_pieces" group.

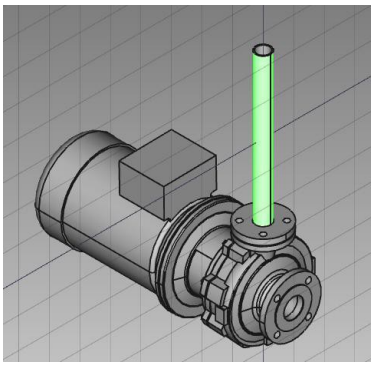


Let's see the most intresting part of the job! Now that we are familiar with the instruments of the toolbar, we are going to see how they can work in conjunctions with features of **Draft** workbench.
The drawing now, except from the suction pypeLine and tank that we hided, looks like in the figure on the right.

Activate the Draft workbench and select the cylindrical surface of the last pipe created.

Push the button 

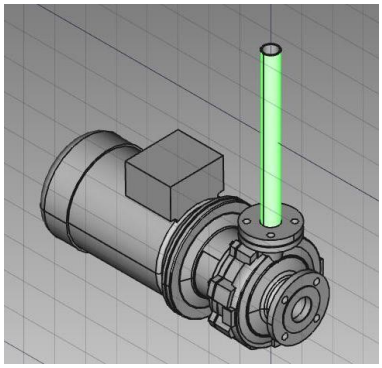
to select automatically the draft workplane and the grid will appear, defining a plane that includes the axis of the pipe like in the picture at side.



The orientation of the workplane depends on the rotation of pipe because the workplane is placed on the CenterOfMass of the cylindrical surface and is oriented according the normal vector in its middle point.
We want to draw the path of the next delivery pipes in the plane that includes pump's axis and the hole that we have cut on the upper tank.

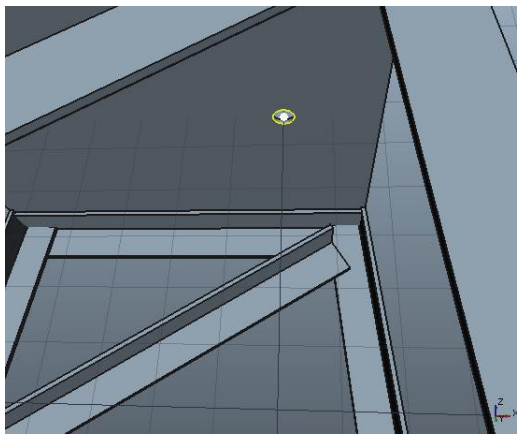
To make this rotate the vertical pipe by 90°, for example pushing on 

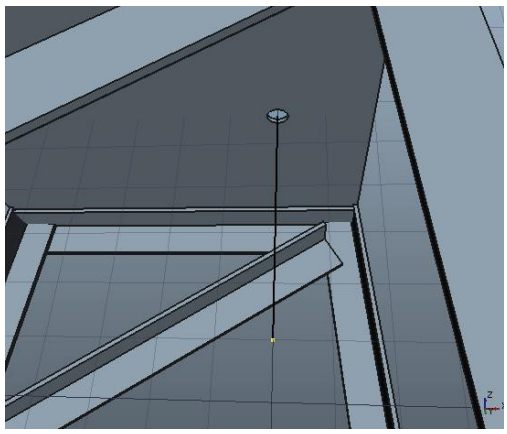
twice, and then redefine the workplane as did before.



In this way also the workplane is rotated by 90 °.

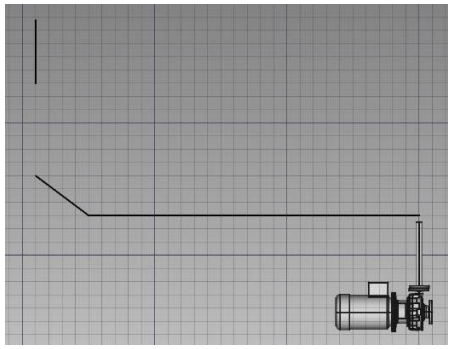
At this point it's possible to draw a vertical line descending from the hole in the bottom of the upper tank.To do this its practical to use the snapping options of Draft w.b.



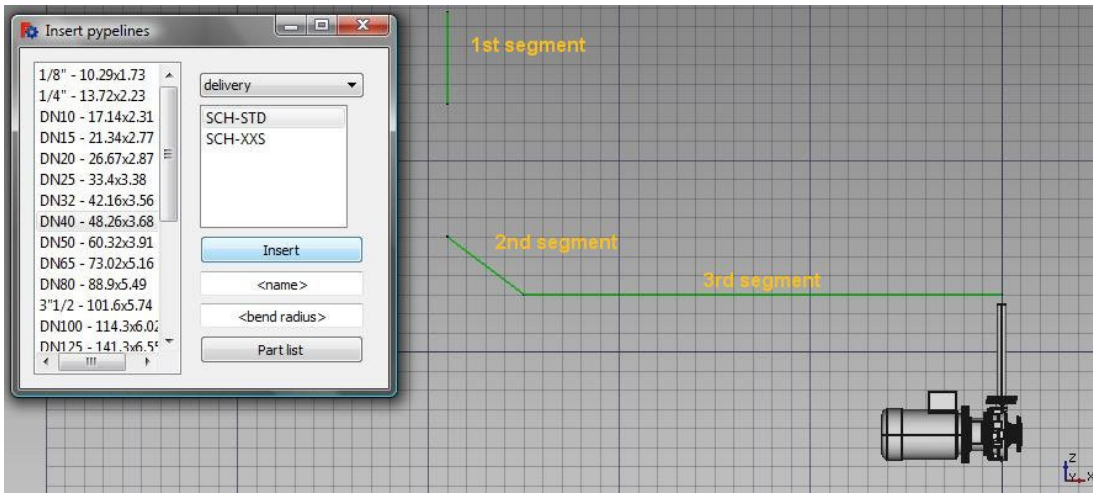


Then draw the horizontal and diagonal line, using the snap-on-grid option, as shown at side.

Hint: it's not necessary that segments ends touch because when we create the rest of pypeLine the gaps will be eliminated by the pipes.

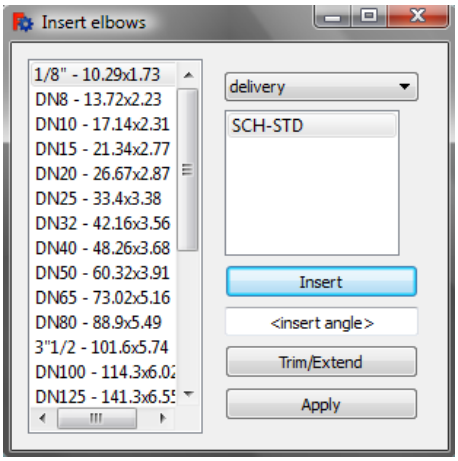


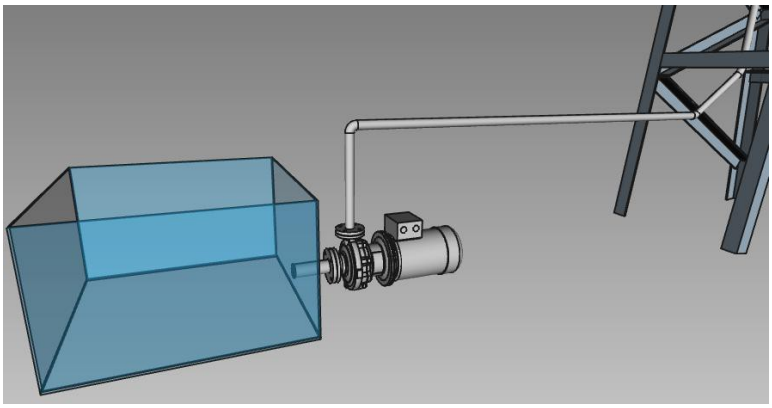
Re-use the "Insert a pype-line" tool but select in the combo-box the already existing "delivery" pype-line and in the sizes list the size DN40. After that, select in the viewport the segments that will be centerlines of the pipe-branch that we are going to create, as shown below (or in reverse order as well).



Push the [Insert] button and all pipes and curves will fit in the drawing following the drafted path.

At this point it's easy to desume that the last gap between the vertical pipe above the pump and the horizontal pipe can be filled with the "Insert a curve" command: select the two pipes in the viewport, select the "delivery" pype-line in the dialog and push [Insert].

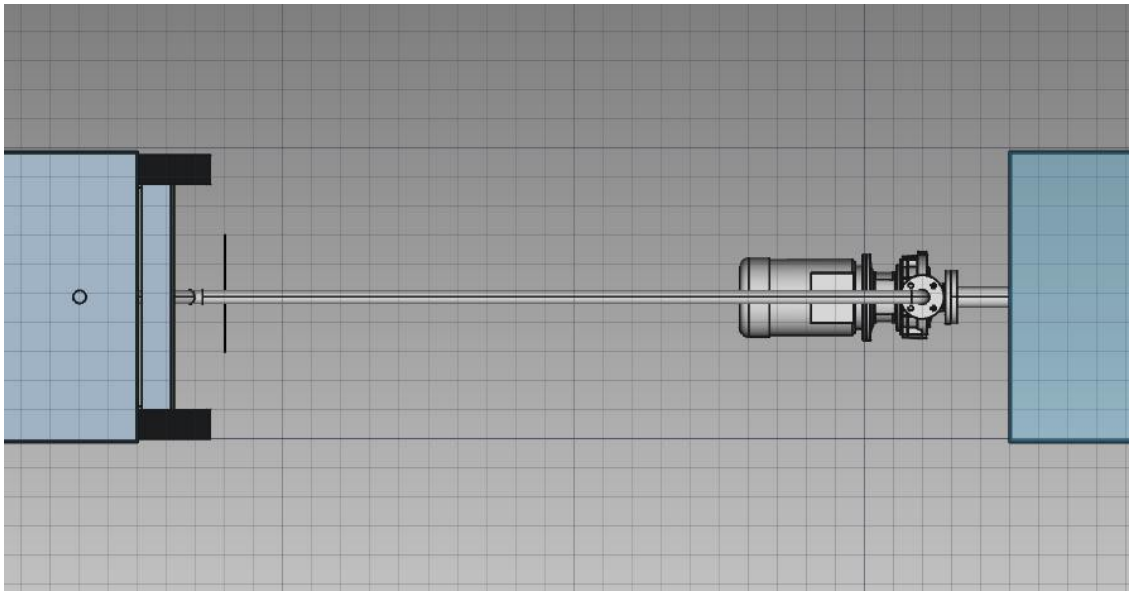




The delivery pipe is finished.

4 - SUPPORTS

A pipeline is not completed without its supports and so a few commands are included to insert them in the model.
Before to see those command it's necessary to draw the structure: for example let's start placing the draft workplane on the global XY plane and draw a line like below.

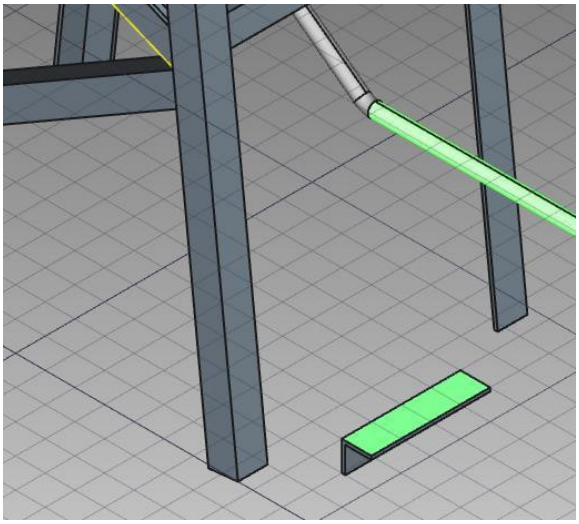


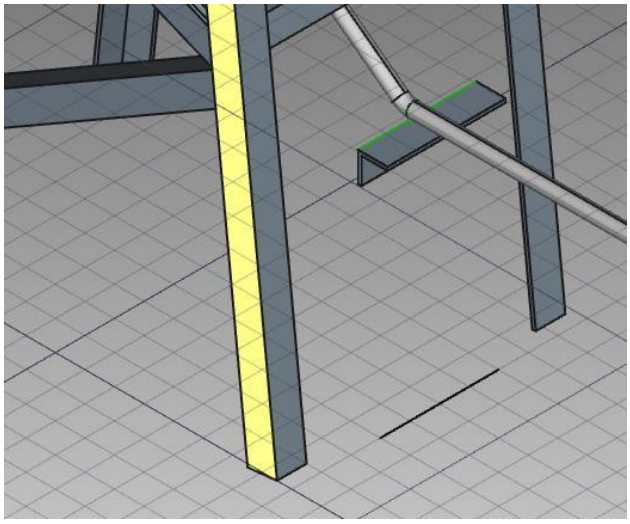
Then lay down a beam on that line with the same profile used for the supporting structure of the upper tank. To do that, use the "fillTheFrame" command as explained in the tutorial for the toolbar "Frame-tools".

Now the first specific command for piping support to use is "Raise-up the support": if you select one pipe and one plane parallel to its axis, this command will move the plane to touch the bottom of the pipe.

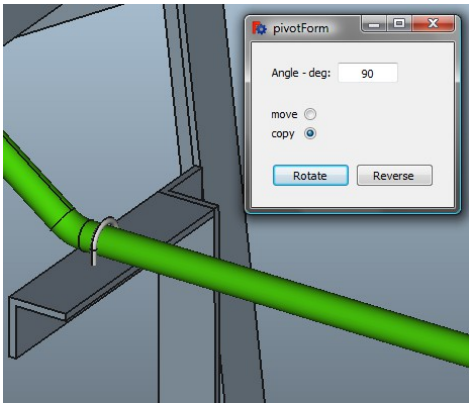
It's clear that the button next to it is the opposite command, that lays down a pipe over a plane face, if it's parallel to its axis.

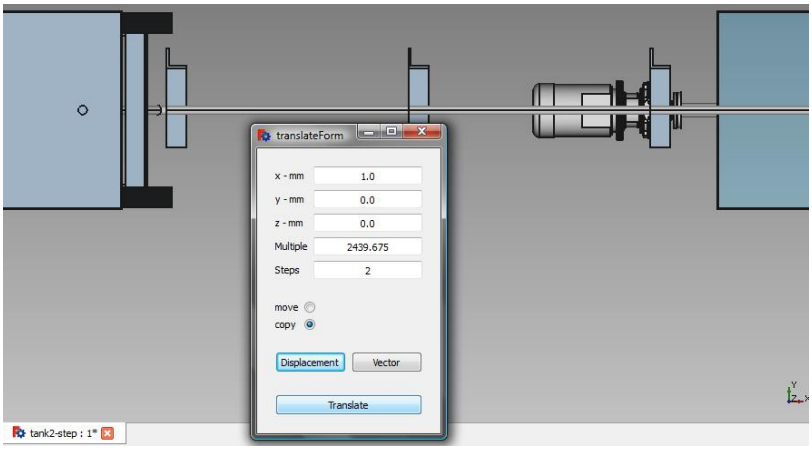
At the end, the result in our model is shown in the pictures below.





Using again some of the commands of the toolbar *Frame-tools* it's possible to complete the support structure and create two copies of it along the horizontal delivery pipe, as shown below.



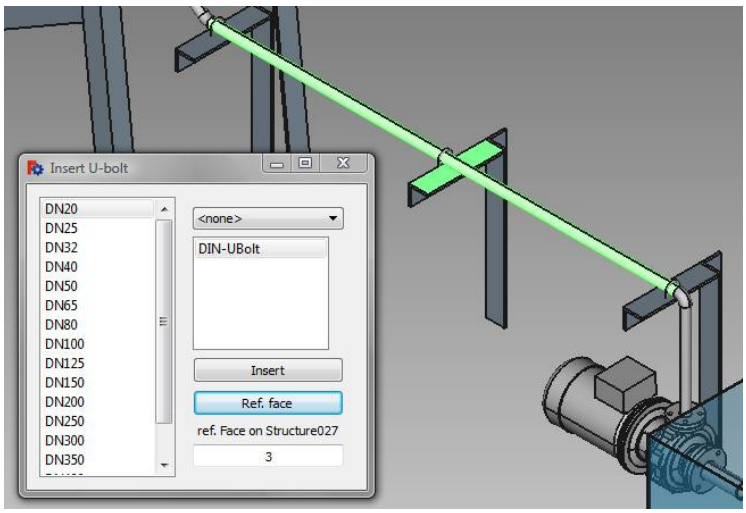


Finally with "Insert U-bolt" command we can create the fixing bolts along the pipe.

The creation dialog of this object presents as usual

- the combo-box for the pypeline selection on the top,
- below it, the list of types,
- below it the [Insert] button,
- the list of sizes on the left.

The peculiar widgets are another button, a label and a text-box.



The [Ref. face] button allow to select one plane to orient the axis of u-bolts that are going to be created. That means the (vertical) axis of symmetry of the u-bolt will be parallel to the normal of the reference face selected. If a face is selected while invoking this command it is automatically taken as the reference plane and its label will be written inside the label of the dialog. Otherwise a new face can be selected afterwards pushing on the [Ref. face] button.
The text-box allows to select how many u-bolts must be created along the selected pipes:

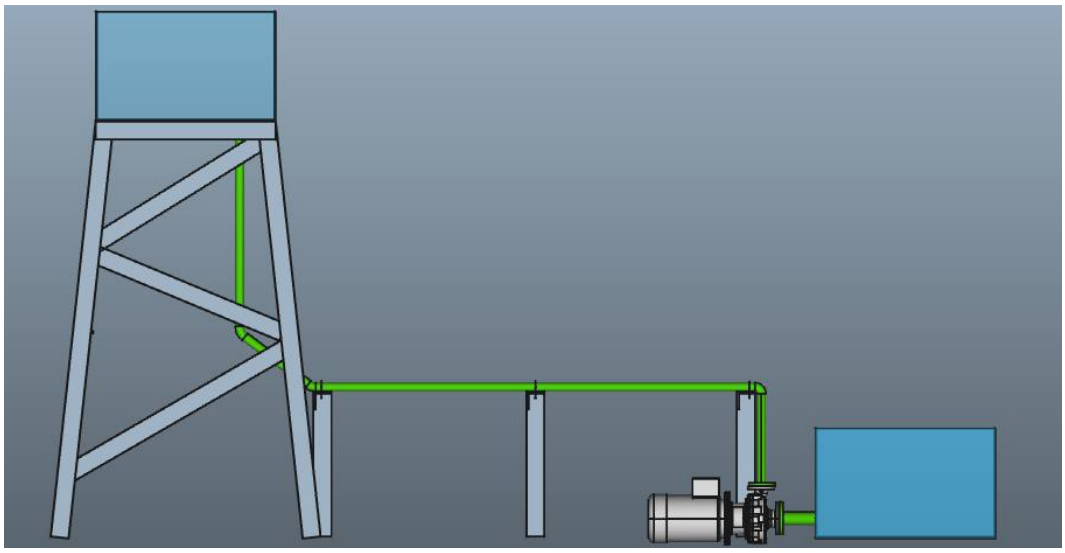
- if "1", the u-bolt is placed at the starting end of the pipe, 1 diameter from its edge;
- if "2", two u-bolts will be created at both ends of the pipes, each one at 1 diameter from the opposite edges;
- if more, u-bolts are evenly distributed along the pipes.

So just select the face and the pipe, invoke the command, write "3" in the text-box; then press [Insert]. If necessary use 

to rotate the u-bolts around the axis of pipe.

Hint: read the automatic documentation of Python's modules from the Help menu of FreeCAD to learn more about how pype-objects are created and their placement options. Browse to "pipeFeatures-py", "pipeCmd.py" and "pipeForms.py" pages.

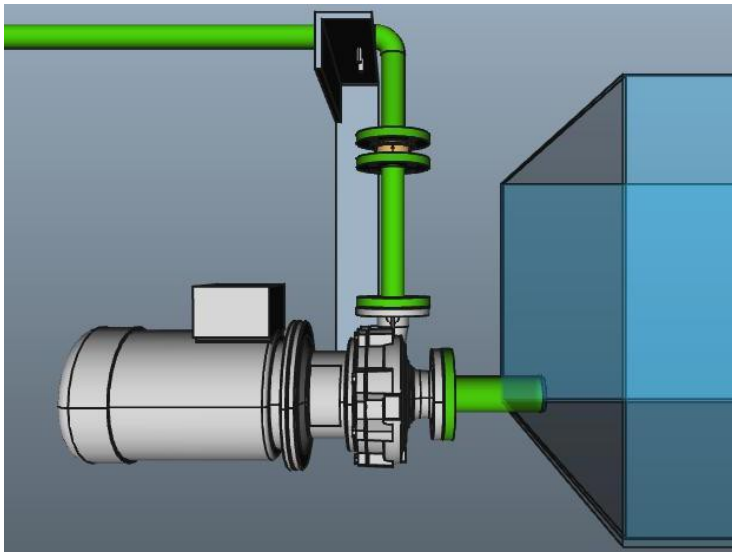
OK. Now that it's all done, the pipes should look like below.



You may also paint the pipe with your favorite colour but.... hey, that won't work!

In fact the water will flow back in the lower tank as soon as you switch off the pump.

Luckily it's easy to insert in the vertical pipe, just at the delivery of the pump, a suitable check-valve (a kind of diode for liquids) that prevents wasting energy for nothing.



Not worth to say, you have all the knowledge to do that:

- stretch the pipe;
- add the other pipe and the two flanges;
- import the model of the valve and place it between the flanges

In next page there are short descriptions of other commands and new features.

As it's clear from the tutorial, the introduction of "pypelines" is the main point that changes a bit how the toolbar could work. This object allow to gather easily the piping objects belonging to the same pipeline not only to keep the model tree clean. It is also comfortable to make some operations on them (e.g. hiding / selecting / deleting, extracting the bill of material, etc.) and define common properties that could be used in future (such as material, flow, roughness..) for calculation (such as pressure losses).

The concentric reduction object extends the set of fittings that can be inserted in the drawing as "native" objects. As for others (pipes, elbows, flanges, clamps) their properties can be defined in .csv tables included in the folder /tables with the same rules described in previous tutorial. But there is one difference: the properties "H" and "thk2" (length and thickness of the smaller diameter respectively) can be defined as "0". In that case the length of the reduction is calculated with the formula 3x(OD1-OD2) and the thickness will be the same of the major diameter.

One word is worth to say also about a little improvement of the command "Rotate through axis". Since generally it is not known the actual orientation of axis of the Shape of the object, now it's possible to define the rotation axis with an edge of the model. To make this. select the edge then select the object you need to rotate and push the [Get] button. That will calculate the axis of rotation from the mutual orientation of object and edge.

Finally, near to the button of its twin command, it's available now a push button to extend one single pipe to the intersection with the axis of another. It should be not necessary to say that this works also with beams and it's handy to produce braces in a truss structure, rather than branches from one main pipe.