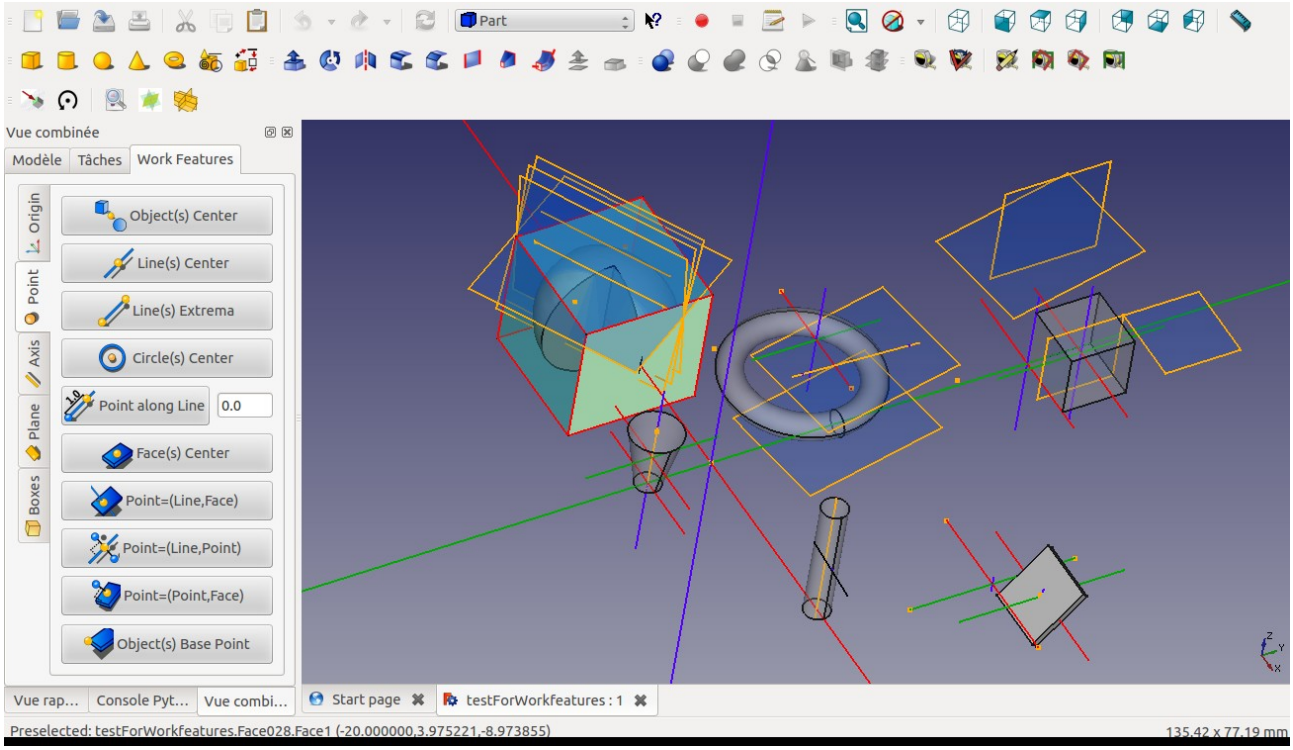
Work features for FreeCAD :

releases Documentation



Version 2015-11 **by Rentlau_64**

Table des matières

Release of 2015-02-26 :

Addition :

into Point TAB :

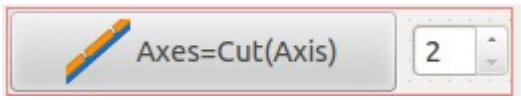
Transform Point(s) in Sketch's Point(s) by projection onto the Sketch's Plane:
- First select an existing Skecth;
- Select as much as Points needed;

Then click on this button.

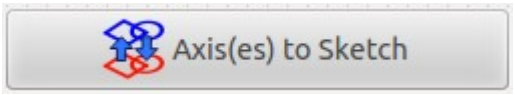
into Axis TAB :

Create an Axis parallel to an Axis and crossing a Point.

The Point will be at :
- Start of the line;
- Mid of the line;
- End of the line.



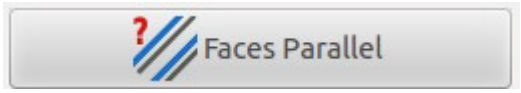
Create Axes:
Cut the selected Line in 2(n) parts and create 2(n) Axes.
The number indicates in how many parts to cut.



Transform Line(s) in Sketch's Line(s) by projection onto the Sketch's Plane:
- First select an existing Skecth;
- Select as much as Lines needed;

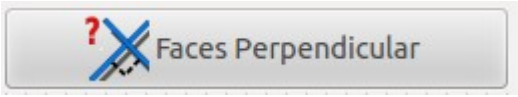
Then click on this button.

a new Check TAB:

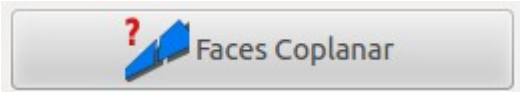


Check if two faces are Parallel:
- Select the 2 faces/planes and

Click this button



Check if two faces are Perpendicular:
- Select the 2 faces/planes and Click this button

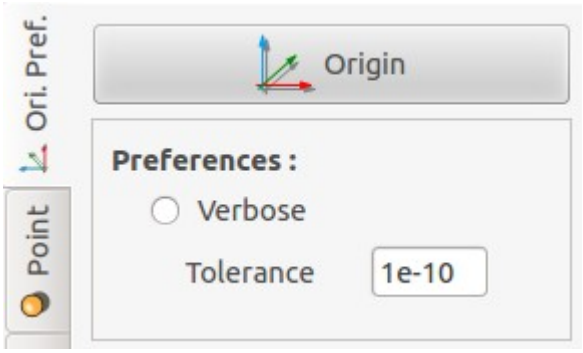


Check if two faces are Coplanar:
- Select the 2 faces/planes and Click this button

Release of 2015-03-08 :

New icon for the macro WORK FEATURE : 

Addition :
into Ori. Pref. TAB :



Possibility to set the tolerance (ie for comparison with Zero)

into Axis TAB :

Create an Axis Perpendicular to an Axis, crossing a Point and Parallel to a Plane.
-Select one Plane, one Axis and one Point ON the previous Axis.

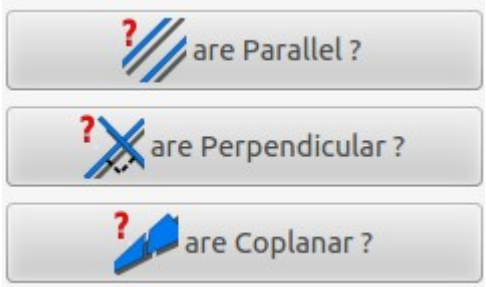
into Circle TAB :

Create Arcs:
Cut the selected Circle(s) or Arc(s) in 2(n) parts and create 2(n) Arcs.
The number indicates in how many parts to cut.

- First select as many Circles and Arcs you want
- Second set the number of parts
- Third push this button

The function is not yet developped for Cylinders.

Correction :
into Check TAB:



Functions available for two faces or two Edges

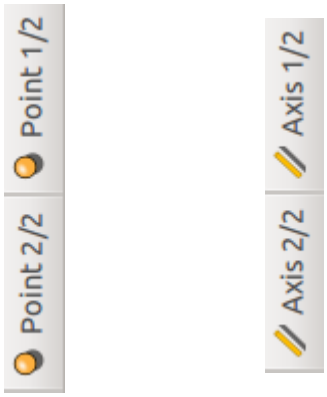
Release of 2015-03-09 :

add of missing icons and link with buttons

Release of 2015-03-15 :

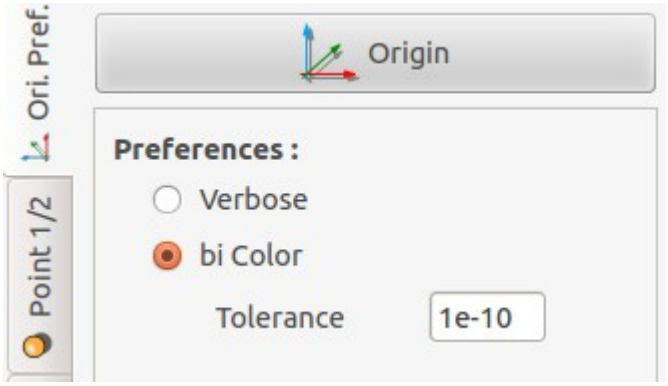
Modification :

Cut in two parts TAB "Points" and TAB "Axis"



Addition :

into "Ori. Pref." TAB :

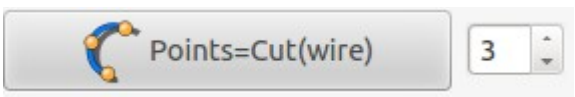


bi Color flag

Change the successive lines to be bicolor (red and white) for the following functions:

- in "Axis 1/2" TAB:
  - Axes=Cut(Wire)
- in "Circle" TAB:
  - Arcs=Cut(Circle)

into "Points 1/2" TAB :



Points = Cut (wire)

Create Points by Partition:
Cut the selected wire(s) in 2(n) parts and create 2(n) Points with function discretize.

The number indicates in how many parts to cut.
Wires can be:
Line
Circle
Arc
Ellipse
An object must also be seleted but before any Wire to cut all his edges!

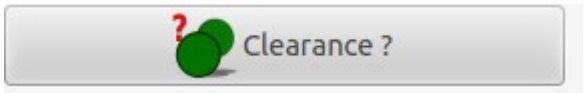
into "Axis 1/2" TAB :

Axes=Cut(Wire)

Create Axes by Partition:

Cut the selected wire(s) in 2(n) parts and create 2(n) Axes with function discretize.
The number indicates in how many parts to cut.
Wires can be:
Line
Circle
Arc
Ellipse
An object must also be seleted but before any Wire to cut all his Edges!
NB: You can change the successive lines to be bicolor (red and white) in "Ori. Pref." TAB

into "Check" TAB :



Clearance ?

Check for two Objects Clearance distance:

Quick measurements between parallel faces and similarly placed objects
- Select the 2 Objects and
Click this button

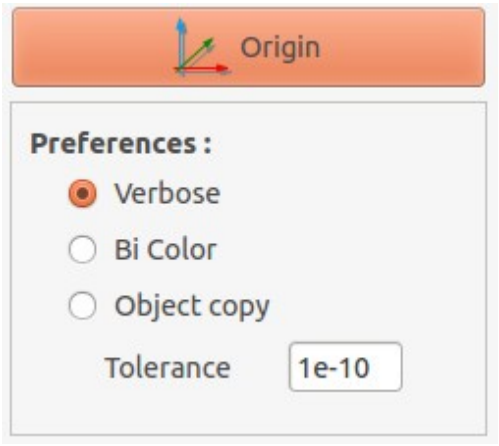
Release of 2015-03-31 :

Modification :

Cut in two parts TAB "Plane"

Addition :

into "Ori. Pref." TAB :



Object copy flag

Force the duplication of the Parent Object for the following functions:

- in "Axis 2/2" TAB:
  Axes=(Axis,Pt,dist)
  If an Edge of a Cube is selected the Cube is duplicate
  with the corresponding
  Edge at the defined distance from the original.
- in "Plane" TAB:
  Plane=(Plane,dist)

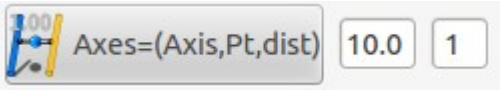
into "Points 2/2" TAB :

Point=(Point,Ax,dist):

Create a Point along the given Axis, at a given distance of the slected Point.
The Axis indicate the direction along where the Point is duplicate.
(you can also select several axes to define different directions)
- First select a Point (you can select several points) and one or several Axis
- Second push this button

NB:
- The distance between points can be defined first.
Positive number in one direction and negative in the other one.
The second number indicates the number of Points to create.

into "Axis 2/2" TAB :



Axes=(Axis,Pt,dist):

Create an Axis parallel to a given Axis, Point at a given distance.
The Axis is created along the Plane defined by the given Axis and Point.
- First select an Axis (or several Axes) and a Point
(you can also select several points to define different Planes)
- Second push this button

NB:
- The distance to the Axis created can be defined first.
Positive number in one direction and negative in the other one.
The second number indicates the number of Axes to create.
With option "Object copy" in "Ori. Pref." TAB
- If an Edge of a Cube is selected the Cube is duplicate with the corresponding
Edge at the defined distance from the original.
Several Edges of the cube can be selected.

Enlarge(Axis):

Extend an Axis at two extrema.
- First select an Axis (or several Axes)
- Second push this button

NB:
- The percentage of the extension can be defined first.

into "Plane 1/2" TAB :

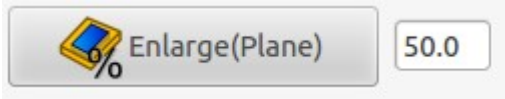
Plane=(Plane,dist):

Create a Plane parallel to a Plane at a given distance.
- First select a plane or several Planes
- Second push this button

NB:
- The distance to the plane created can be defined first.
Positive number in one direction and negative in the other one.
The second number indicates the number of planes to create.
With option "Object copy" in "Ori. Pref." TAB
- If a Face of a Cube is selected the Cube is duplicate with the corresponding Face at the defined distance from the original.

Several Faces of the cube can be selected.

into "Plane 2/2" TAB :



Enlarge(Plane):

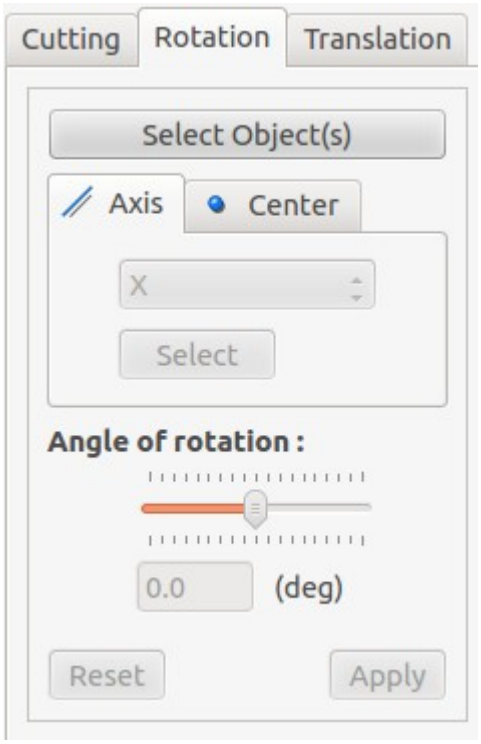
Extend a Plane in each dimension.
- First select a Plane (or several Planes)
- Second push this button

NB:

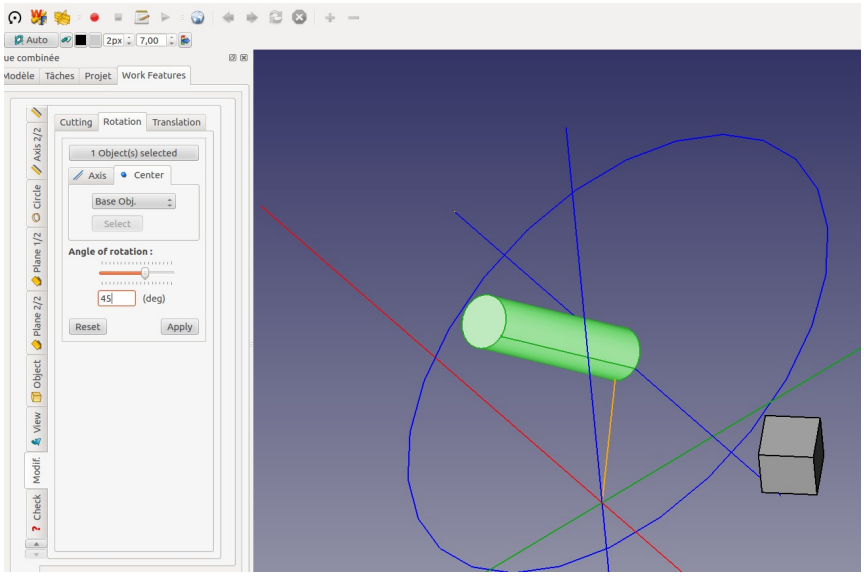
– The percentage of the extension can be defined first.

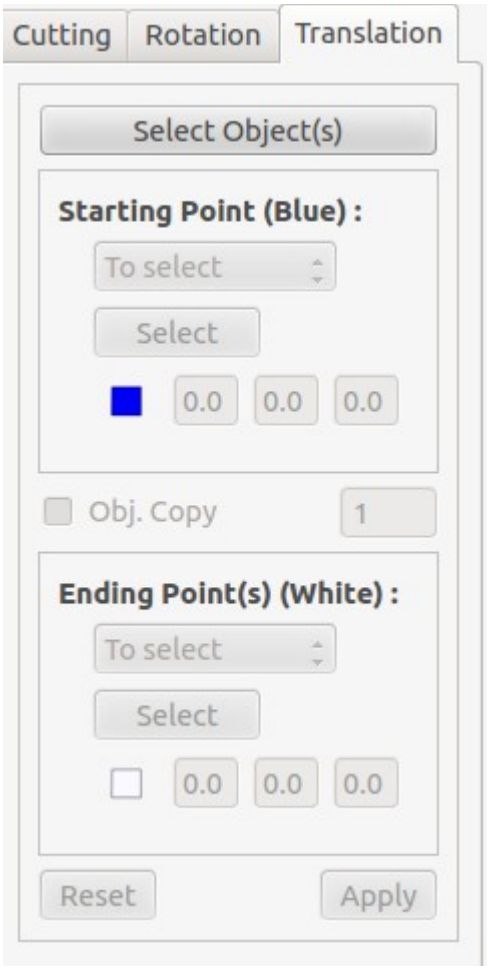
–

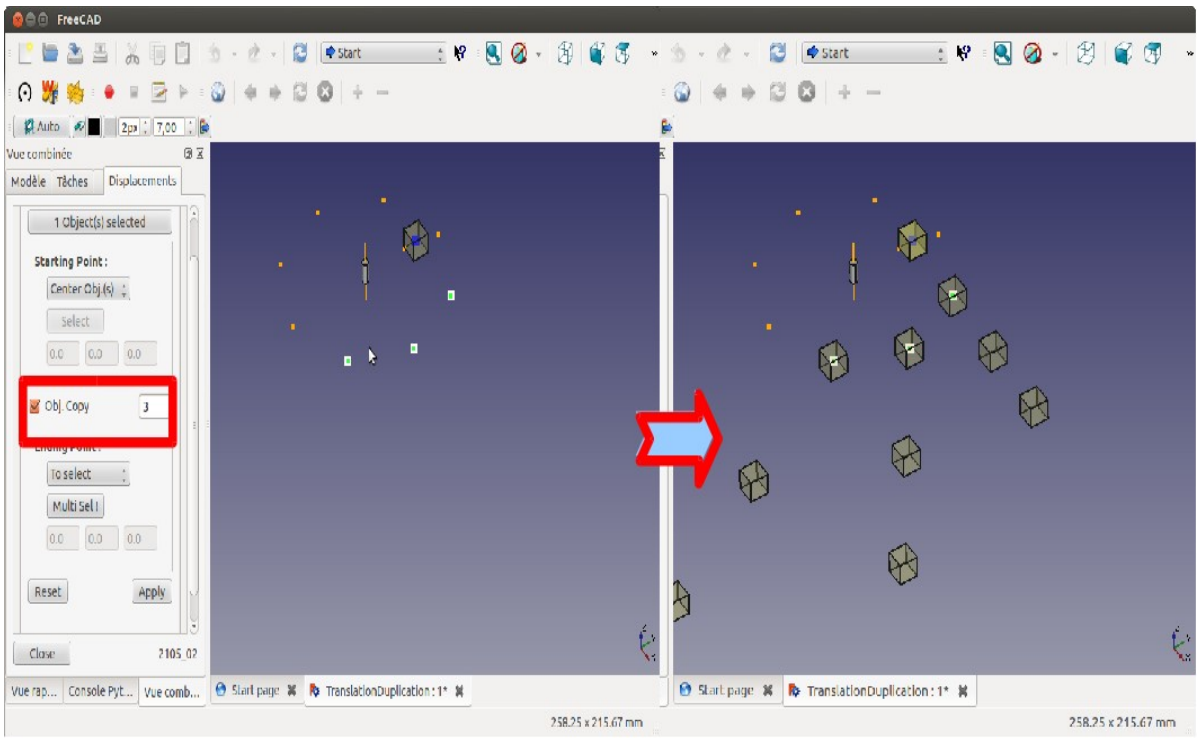
into "Modif." TAB :



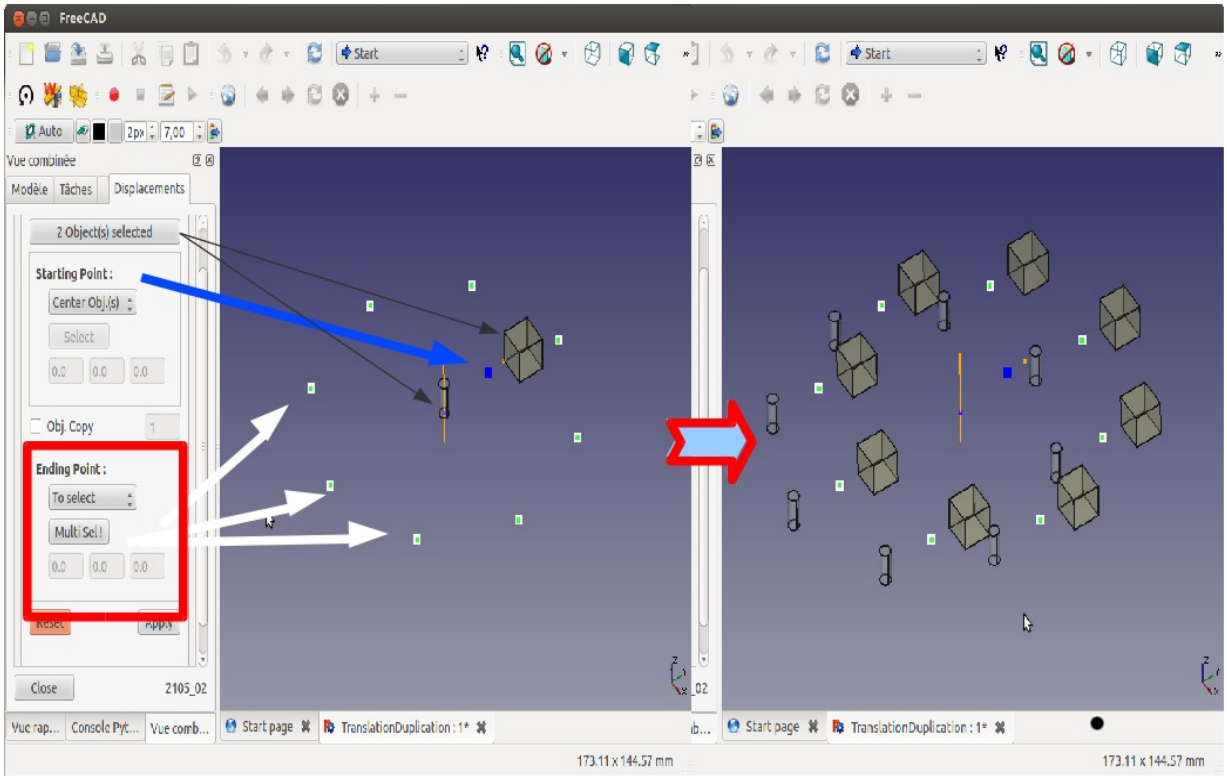
A new rotation tool with selection of Axis and center of rotation


Auto
2px
7,00
ue combinée
Modèle
Tâches
Projet
Work Features
Cutting
Rotation
Translation
1 Object(s) selected
Axis
Center
Base Obj.
Select
Angle of rotation :
45
(deg)
Reset
Apply
Axis 2/2
Circle
Plane 1/2
Plane 2/2
Object
View
Modif.
Check


Cutting
Rotation
Translation
Select Object(s)
Starting Point (Blue) :
To select
Select
0.0
0.0
0.0
Obj. Copy
1
Ending Point(s) (White) :
To select
Select
0.0
0.0
0.0
Reset
Apply



A new translation tool with duplication



Release of 2015-04-03 :

```
Traceback (most recent call last):
  File "/home/jo/.FreeCAD/Macros/start_WF.FCMacro", line 16, in <module>
    import WorkFeature
  File "/home/jo/.FreeCAD/Macros/WorkFeature/__init__.py", line 53, in <module>
    from   WF_ObjRot_2015 import *
  File "/home/jo/.FreeCAD/Macros/WorkFeature/WF_ObjRot_2015.py", line 8, in
<module>
    import ObjRotGui_2015 as ObjRotGui
<type 'exceptions.ImportError'>: No module named ObjRotGui_2015
```

Modification :

WF_ObjRot_2015.py

line 8 import ObjRotGui_2015 as ObjRotGui (deleted)

line 9 import __init__ as func (changed into)

Release of 2015-05-23 :

Bug correction :

QT Icons path for FreeCAD was replaced inducing a "not found icon" error message.

Release of 2015-05-31 :

Modification :

Add sub ***Tab Align*** into ***Modif. Tab***

Modification of Tool Rotate

new angle definition by selection of Edges

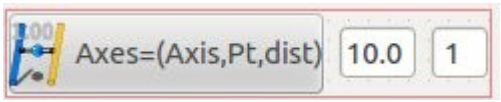
Modification of Tool Translate

Posibility to make a "deep" copy of objects

Add definition relative of ending point by user input

Addition :

into "Axis 2/2" TAB :



Axes=(Axis,Pt,Pl,a)

Create an Axis with an Angle to a origin Axis.

- First select an Axis to rotate, then a Plane and a rotation Point
- Second push this button

or

- First select an Axis to rotate, then a rotation Axis and a rotation Point
- Second push this button

NB:

The Axis is created by rotation using :

The Normal of the selected Plane as rotation Axis

and selected Point as rotation Point.

or

The second selected Axis as rotation Axis

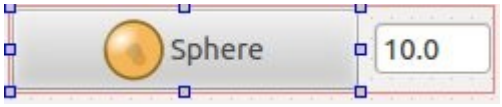
and selected Point as rotation Point.

- The angle (in degrees) of rotation can be defined first.

Positive number in one direction and negative in the other one.

- The second number indicates the number of Axes to create.

into "Object" TAB :



Sphere

Create a Sphere shell:

- First select one or several Center Point(s).
- Define Diameter if needed.

Then Click the button...

It will create Sphere shell(s) centered

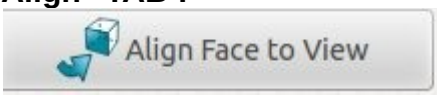
at the selected point(s).

Dome
Create a full geodesic dome shell:
- First select one or several Center Point(s).
- Define Diameter and Frequency Parameter (Integer between 1 to 10) if needed.
Then Click the button...
It will create full geodesic dome shell(s) with a X-Y-symmetry plane
for even frequencies and centered
at the selected point(s).
If Frequency Parameter = 1, the code create an icosahedron.
An icosahedron is a polyhedron with 20 faces.
Original code from : Ulrich Brammer

into "Modif." TAB :

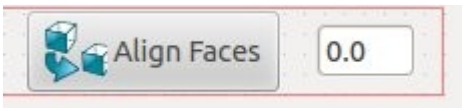
Add of Align Tab

into "Align" TAB :



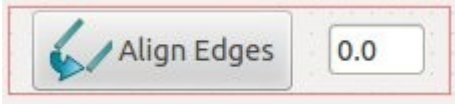
Align Face to View
Align the face of selected object(s) to the actual view Plane.

- Click first to select a Face of one or several objects.
Then Click the button.
NB:
The center of rotation is the center of the bounbing box if possible or
the center of the Face.
if the Face of the object selected is already aligned to the view Plane,
a rotation of 180 deg is applied to the object.
In this case the Axis of rotation is Z vector : Base.Vector(0, 0, 1)



Align Faces
Align the Face(s) from selected object(s) to the last Face selected.

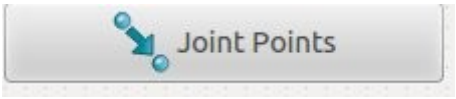
- Click first to select a Face of an object or several Faces from several objects.
- Click second to select a Face to align to.
Then Click the button.
NB:
The center of rotation is the center of the bounbing box if possible or
the center of the Face.
if the Face of the object selected is already aligned to the last one,
a rotation of 180 deg is applied to the object.
In this case the Axis of rotation is Z vector : Base.Vector(0, 0, 1)



Align Edges
Align the Edge(s) from selected object(s) to the last Edge selected.

- Click first to select an Edge of an object or several Edges from several objects.

- Click second to select an Edge to align to.
Then Click the button.
NB:
The center of rotation is the center of the bounbing box if possible or
the center of the Edge.
if the Edge of the object selected is already aligned to the last one,
a rotation of 180 deg is applied to the object.
In this case the Axis of rotation is Z vector : Base.Vector(0, 0, 1)



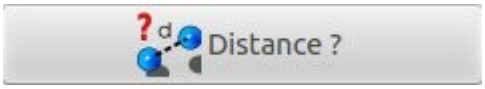
Joint Points
Joint Point(s) from selected object(s) to the last Point selected.

- Click first to select a Point of an object or several Points from several objects.
- Click second to select an Point to joint to.
Then Click the button.

into "Check" TAB :

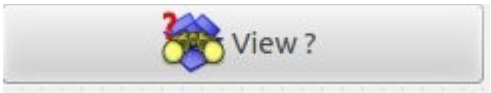
Angle
Check for two Edges/Planes angle:

Angle measurement between two Edges or two Planes
- Select the 2 Edges and
- Click this button
or
- Select the 2 Planes and
- Click this button
or
- Select one Edge and one Plane and
- Click this button
NB:
Normals of Planes will be used.



Distance
Check for two Points distance:

Distances measurement and Delta values (on main Axes) between two Points
- Select the 2 Points and
Click this button

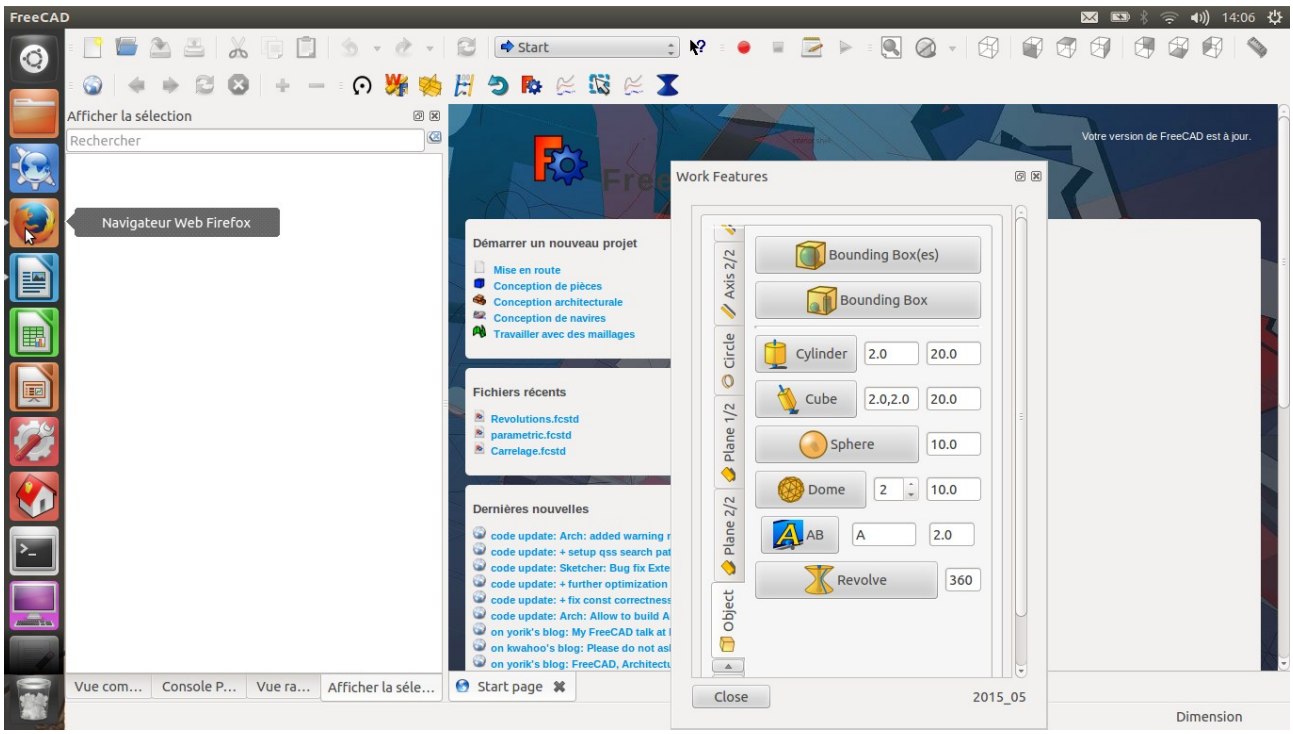


View
Detect the position of the camera.

The returned value is the value provided
by the function getCameraOrientation().

Release of 2015-06-22 :

Modification :



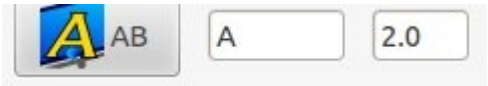
A new dock window version for Work Feature widget

Addition :

into "Circle" TAB :

Transform Circle(s) and Arc(s) in Sketch's object(s) by projection onto the Sketch's Plane:
- First select an existing Skecth;
- Select as much as Circles and arcs needed;
Then click on this button.

into "Object" TAB :



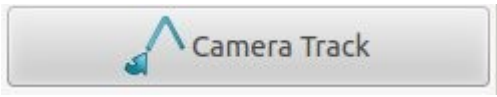
AB:
Create 3D Text attached to a Point.
- First select a Plane
- Then push this button
in this case the center of the text is attached to center of the Plane;
or

- First select a Plane and a Point on the Plane
- Then push this button
NB:
Change the text and his size if needed

Revolve:
Make the revolution of Edge(s) or Wire(s) around an Axis:
- Select one or several wire(s)
- Then push this button
or
- Select FIRST one Point as center of rotation and one Axis as rotation axis !
- Select one or several wire(s)
- Then push this button

NB:
You can also define the angle of revolution if needed
If no Axis is selected the Z axis is considered as Axis of rotation !
If no Point is selected the Origin is considered as Center of rotation !

into "Check" TAB :



Select a Wire a the camera will follow the track.
Originalcode : Tour camera by Javier Martinez Garcia November 2014

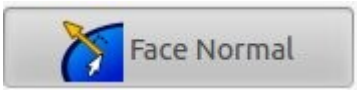
into "Check" TAB :

Check for surface Area:
Area measurement for a Plane or a set of Planes.
- Select One or several Planes and
Click this button

Release of 2015-09-02 :

Modification and addition:

into "Circle" TAB :



Create a normal Axis of a Face.

New handle of mesh objects.

To create a Normal at click location on a Face:

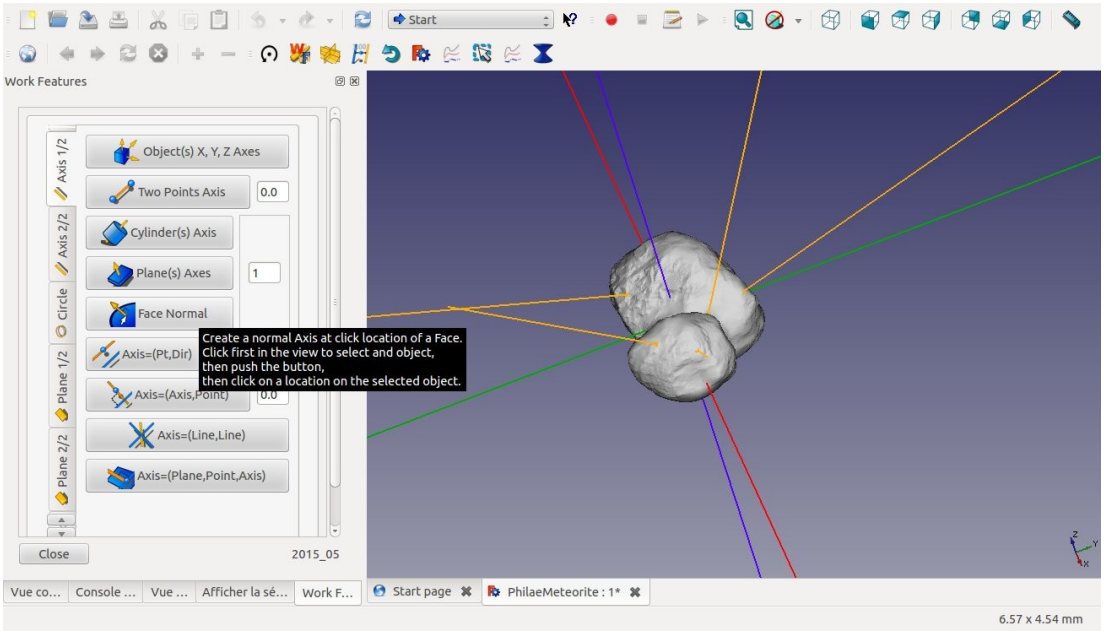
- Click first in the view to select and object,
- then push the button,
- then click on a location on the selected Face.

or

To create several Normal of the face:

- Click first in the view to select and object,
- then select one or several points of the face
- then push the button.

(These selections can also be done into the Combined View)



into "Object" TAB :

Section Sweep:

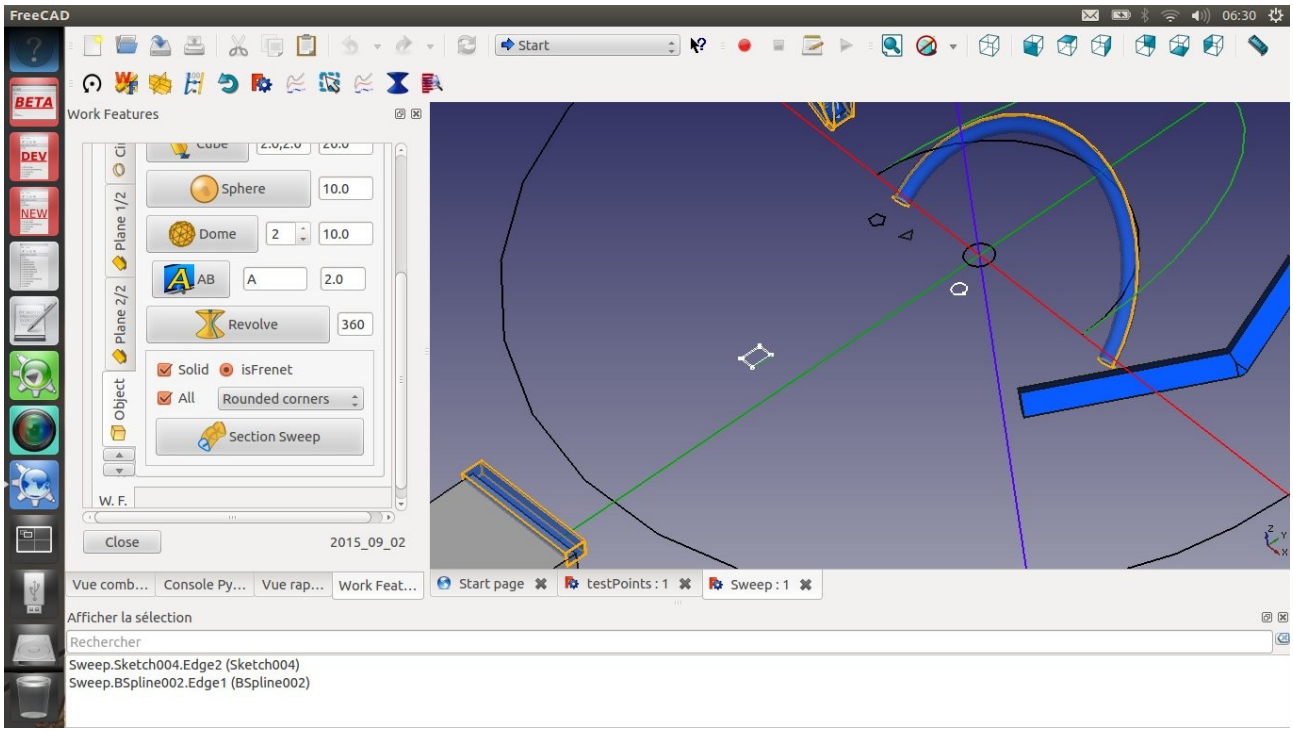
Make a loft defined by a list of profiles along a wire.

Will extrude/sweep a Section along a Trajectory like sweep from Part Workbench

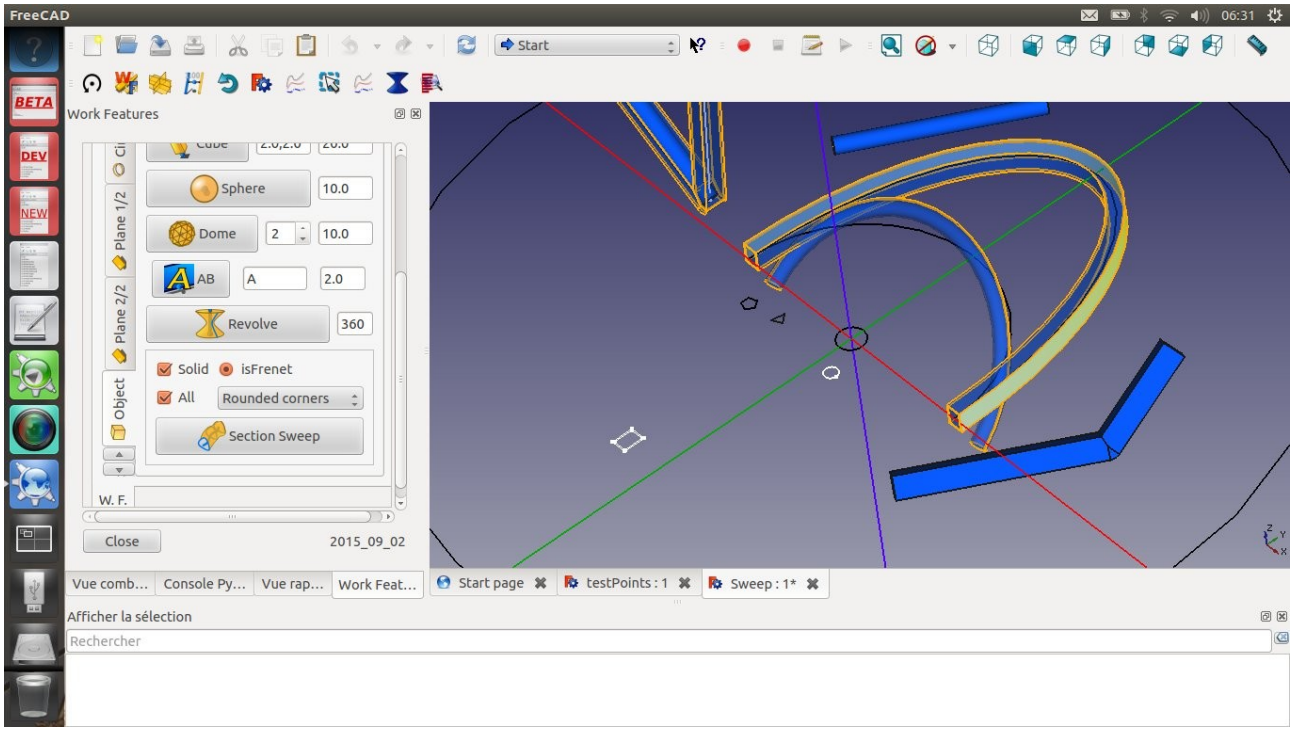
but:
- the Section center (of Mass) is move at the first point of the Trajectory and;
- the "plane" of the Section is rotate to be perpendicular to the Trajectory.

- Select first one Section wire (Closed wire will generate volumes by default)
(This Section can be a compound from sketch to realize "tube")
- Select one or several wire(s) as Trajectory(ies)
- Then push this button

NB: You can change first:
- Solid option (if toggled will generate a solid for Closed wire Section only)
- isFrenet option
- All option (means if the trajectory selected is a compound, all sub wires will be used for the sweep)
- Transition Option (Select a Transition option in case of trajectory with several wires; Transition can be:
0 (default), 1 (right corners) or 2 (rounded corners).)



Hereafter the results

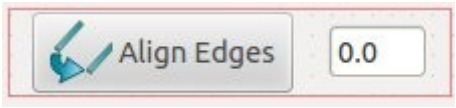


into "Align" TAB :

Align the Face(s) from selected object(s) to the last Face selected.
- Click first to select a Face of an object or several Faces from several objects. These objects will be moved.
- Click second to select a Face to align to (the last object is fixed and will never move).
Then Click the button.

NB:
The center of rotation is the center of the bounbing box if possible or the center of the Face.
if the Face of the object selected is already aligned to the last one, a rotation of 180 deg is applied to the object.
In this case the Axis of rotation is Z vector : Base.Vector(0, 0, 1)

Two clicks will rotate by 180 deg the moving objects.



Align the Edge(s) from selected object(s) to the last Edge selected.

- Click first to select an Edge of an object or several Edges from several objects. These objects will be moved.
- Click second to select an Edge to align to (the last object is fixed and will never move).
Then Click the button.

NB:
The center of rotation is the center of the bounbing box if possible or
the center of the Edge.
if the Edge of the object selected is already aligned to the last one,
a rotation of 180 deg is applied to the object.
In this case the Axis of rotation is Z vector : Base.Vector(0, 0, 1)

Two clicks will rotate by 180 deg the moving objects.

Joint Face(s) from selected object(s) to the last Face selected.
- Click first to select a Face of an object or several Faces from several objects. These objects will be moved.
- Click second to select a Face to joint to (the last object is fixed and will never move).
Then Click the button.

Two clicks will rotate by 180 deg the moving objects.

Release of 2015-10-05 :

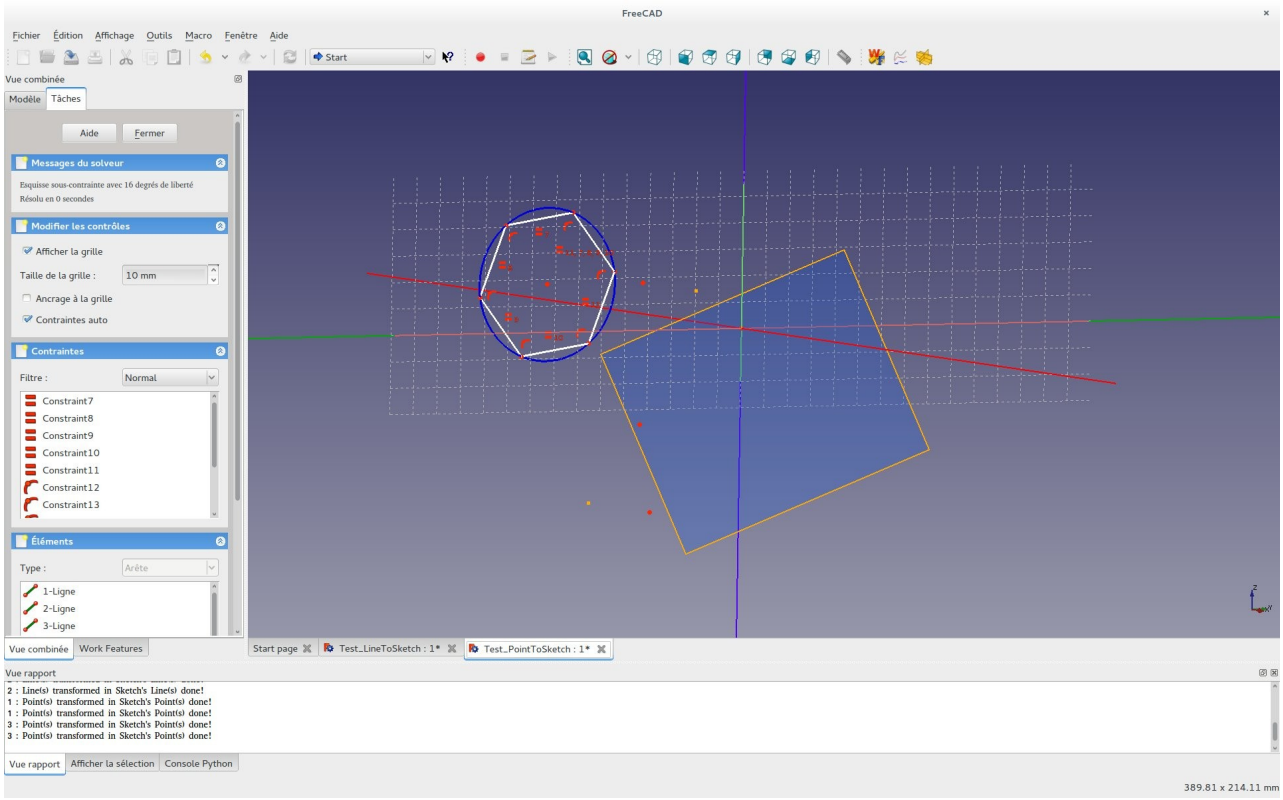
Bug correction: Projection of points and Edges onto an Sketch. It worked correctly when the Sketch was aligned on one of the 3 main planes (ie XY, XZ or YZ)
but failed when the Sketch was aligned in an other direction.

Correction done for Point(s) to Sketch and Edge(s) to Sketch (not yet for circle To Sketch, correction on going)

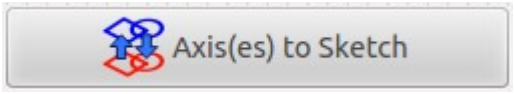
into Point TAB :

Transform Point(s) in Sketch's Point(s) by projection onto the Sketch's Plane:
- First select an existing Skecth;
- Select as much as Points needed;

Then click on this button.



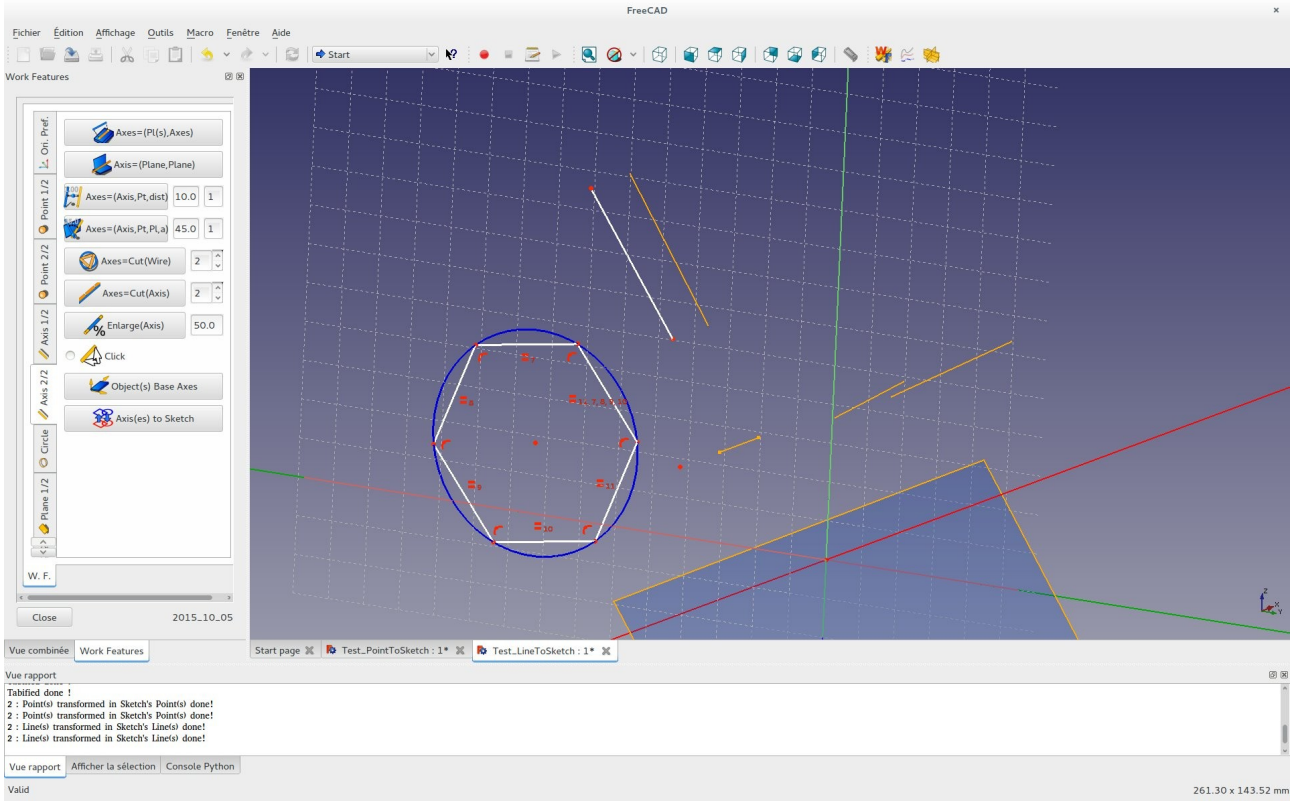
into Axis TAB :



Transform Line(s) in Sketch's Line(s) by projection onto the Sketch's Plane:

- First select an existing Skecth;
- Select as much as Lines needed;

Then click on this button.



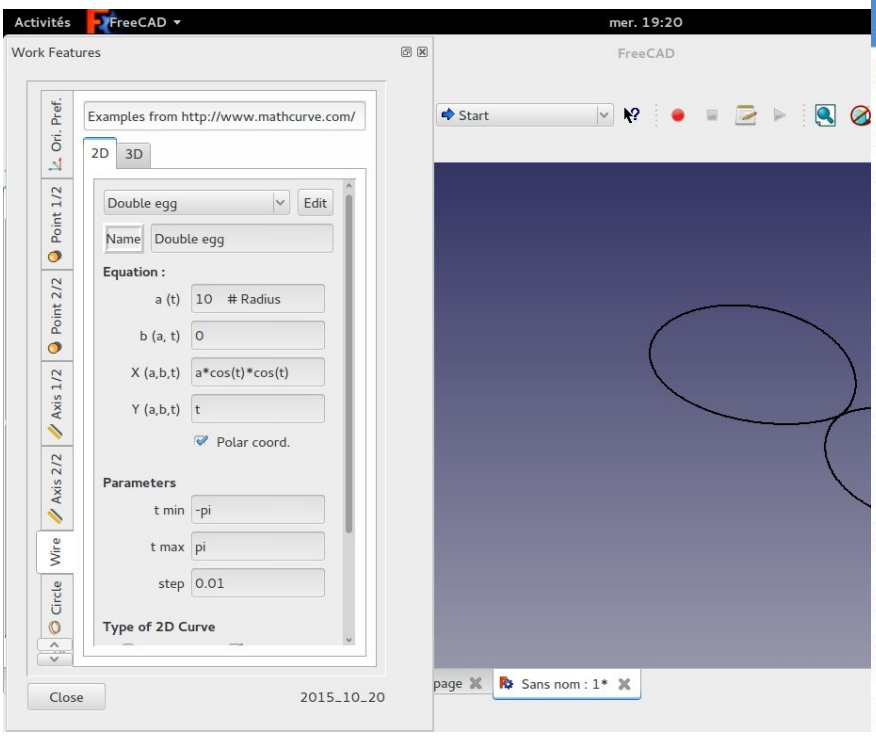
Release of 2015-11-04 :

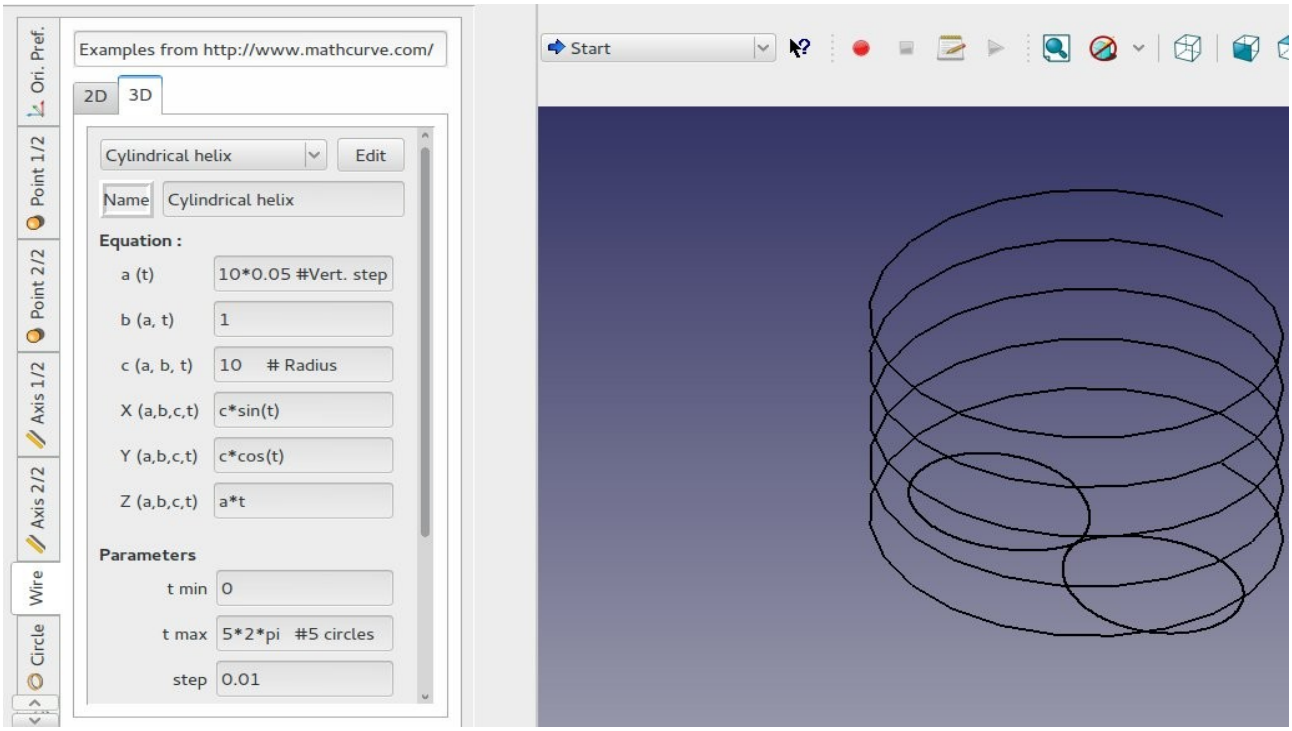
Modification and addition:

Creation of TAB : Wire

2D and 3D parametric functions panel.
A set of predefined functions is available via a combo box.

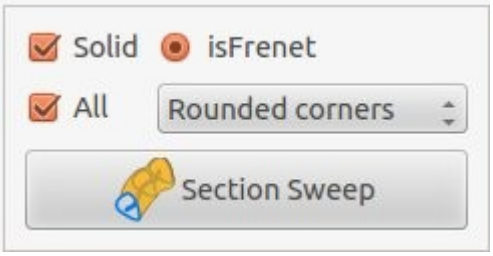
With possible saving of your parametric functions into a file in the home directory.

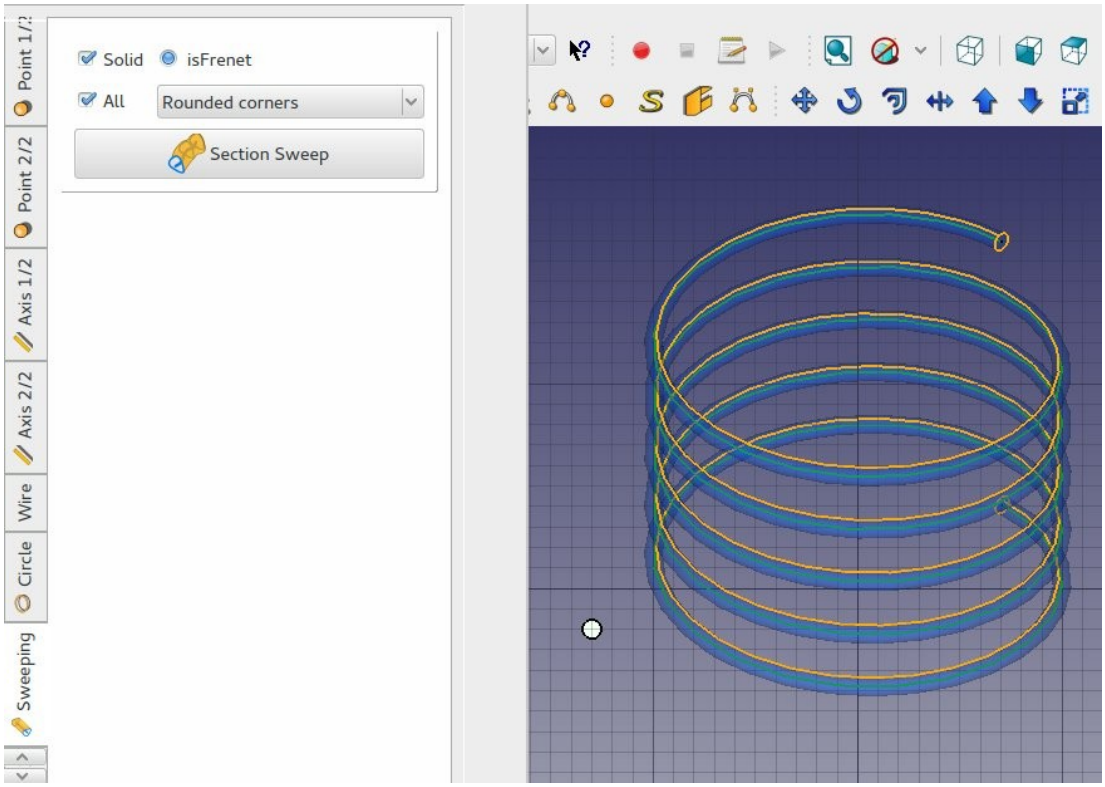




Creation of TAB : Sweeping

Move of Section Sweep from "Object" TAB into "Sweeping" TAB:



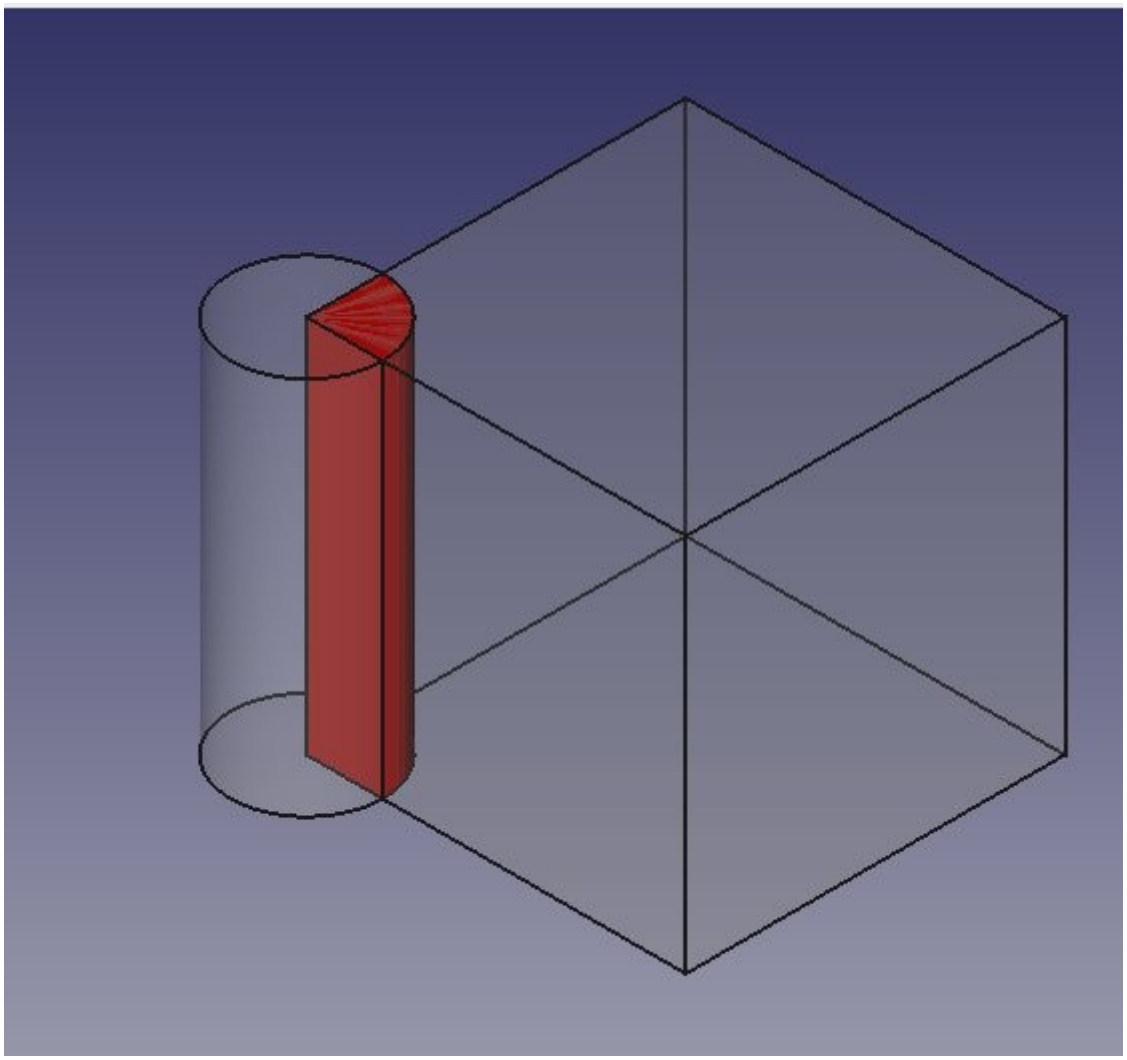


into "Object" TAB :

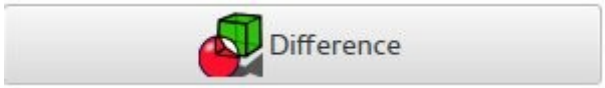
Compute the common parts between selected shapes.

- Select at least two objects and click.

Highlight common parts by showing the common shape in red and setting half-t
ransparency on original parts (the original objects are not modified).
Original code from HighlightCommon.FCMacro
https://github.com/FreeCAD/FreeCAD-
macros/blob/master/Utility/HighlightCommon.FCMacro
Authors = 2015 Javier Martinez Garcia



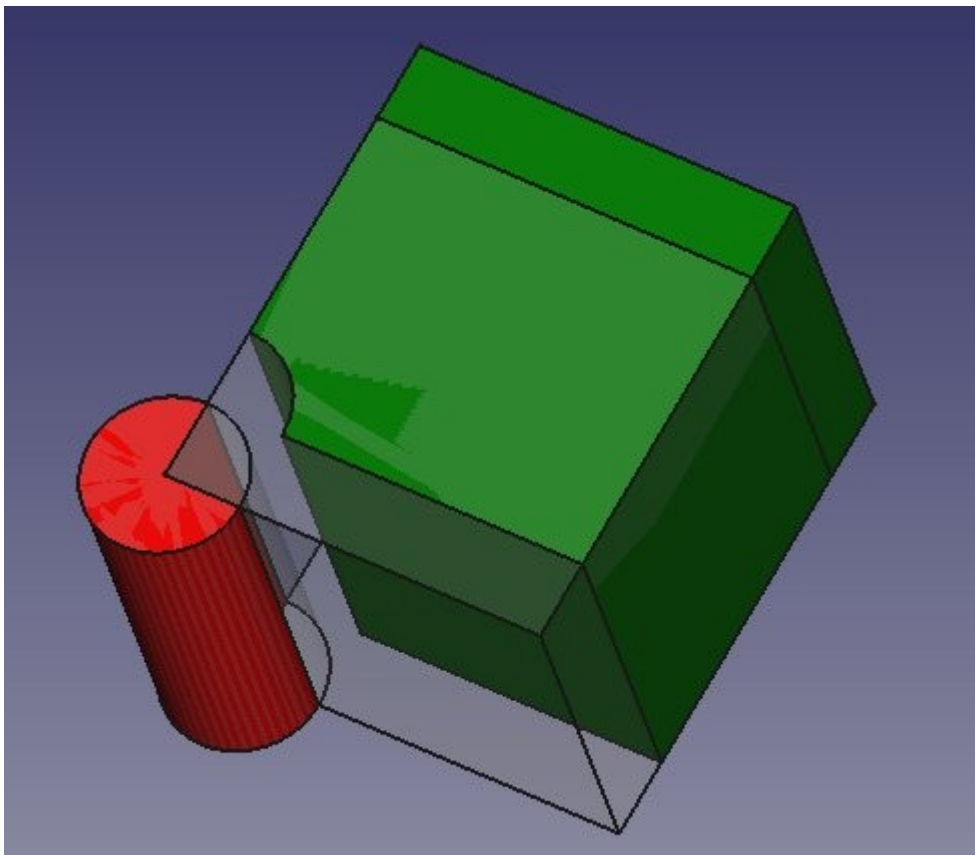
into "Object" TAB :



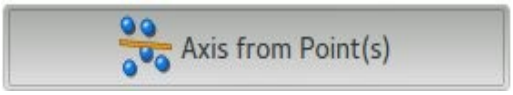
Compute the difference parts between selected shapes.

- Select two objects and click.

Compute the difference between two shapes. Additions are marked red, removals are marked green. Both original parts will be half transparent. The volume of the additions and removals are printed in the console.
Original code from HighlightDifference.FCMacro
https://github.com/FreeCAD/FreeCAD-macros/blob/master/Utility/HighlightDifference.FCMacro
Authors = 2015 Gaël Ecorchard (Galou)



into "Axis 1/2" TAB :



Axis=(Points):
Create a "best fit" line from a set of points using Singular Value Decomposition..
- First select several Points
- Then push this button

The 3 eigenvectors are generated.
Orange one is the best fit line.