Maker Workbench

Release 1.0.0

David Muñoz Bernal

Aug 18, 2023

CONTENTS

On this page you can find all the information about the Maker Workbench. You have some tutorials where you can learn how to use it, as well as the documentation of all the 3D models and the functions that have been designed.

CHAPTER

ONE

TABLE OF CONTENTS

1.1 Introduction

1.1.1 Maker Workbench

Maker is a Workbench for FreeCAD that allows the modification of parametric models that includes and simplify the assembly and composition. It also provides a library of functions with which to generate new parametric models to be included in the Workbench.

1.1.2 How it works

Maker is designed for two different users:

1. Basic User: User without CAD or Programming knowledge who needs to make a design. For this user is the graphic part. The steps to follow would be:

1. Select the part
2. Enter the values you want to use
3. You already have the part you want.

Note: See the *Tutorial* section for more details

Additionally, this user may want to combine parts with each other. For this purpose, there is the *Assembly* module[1] that allows the placement of some pieces with respect to others

2. Advanced user: User with programming knowledge who wants to design parameterizable 3D models. This user has the *Functions Library*[2] to make 3D models in a simple way. You can consult the class design (*UML*) if you wish to better understand the operation

[1] The Assembly module will be upgraded
[2] The functions library is in process

1.1.3 History

Maker Workbench started with the Filter Stage project. In this project, we designed a support for the sample holder of a microscope to the URJC. In order to be able to modify the design and adapt it to the required dimensions, we chose to make a parametric design. Parametric design requires the use of a programming language to describe the model, in this case we use Python.

Based on this parameterizable design, a final degree work was conceived to create a Workbench where the Filter Stage parameters could be modified from the FreeCAD interface.

It was decided to improve this first Workbench by adding designs commonly used in mechatronic systems and functions to facilitate the placement of these designs. Also we add some optics models and ther are more models comming soon.

1.2 Install

To install the Maker Workbench you need FreeCAD.

Note: Works on FreeCAD 0.19

After installing the program, there is two ways to install the Workbench:

1. **Use the addon manager**. In FreeCAD, go to the menu *Tools/Addon manager*. In the Addon Manager choose *Configuration* from de upper rigth side. In the new window add https://github.com/URJCMakerGroup/MakerWorkbench to the Custom Repositories
2. Download Maker Workbench from the following file. Take the *Maker Workbench* folder from the *MakerWorkbench.zip* file and put it in the *Mod* folder inside the *FreeCAD* installation folder. The default path of *FreeCAD* is:

S.O.	Folder
WINDOWS	*C:/Program Files/FreeCAD 0.19/Mod*
MAC	*/Applications/FreeCAD 0.19/Mod*

After completing these steps, the Workbench will be listed in the FreeCAD Workbench

1.3 Tutorial

Note: Work in progress

1.3.1 YouTube Tutorials in Spanish

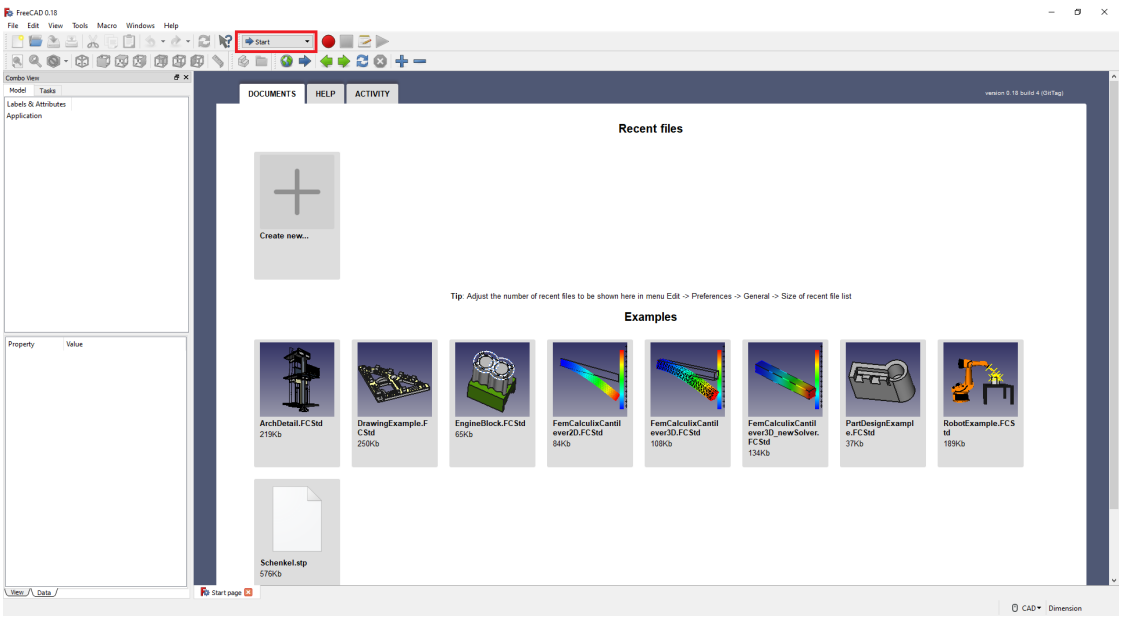
There are diferents tutorials availables in Youtube.

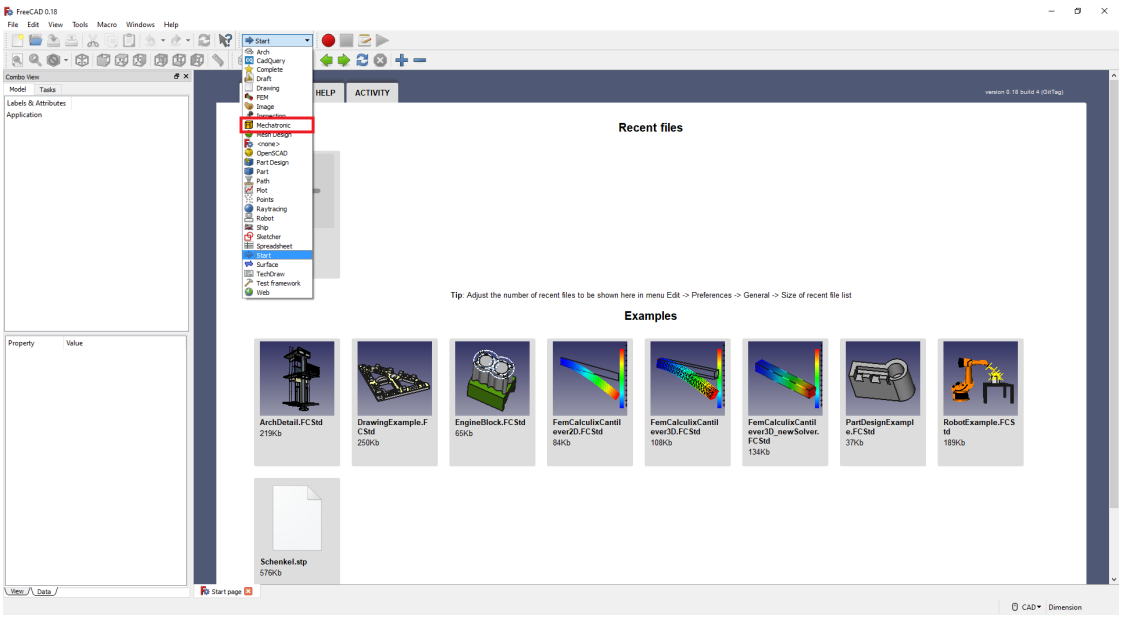
> **Warning:** We have update the workbench but we have not yet updated the followings tutorials

1.3.2 Tutorial CAD 1 - Create part

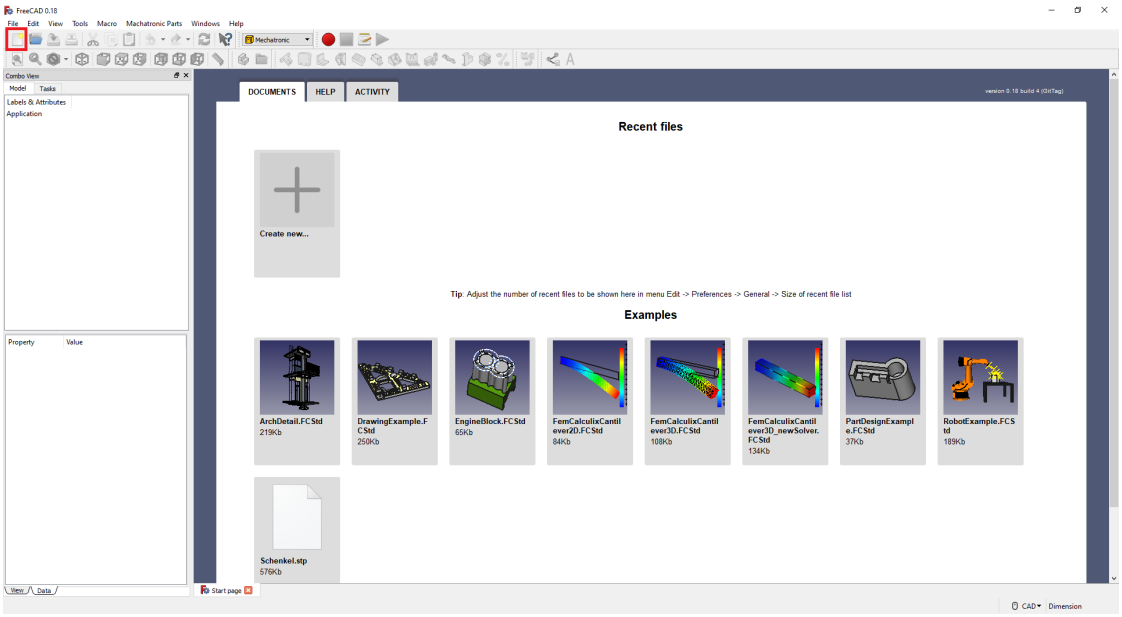
1- After opening FreeCAD, the first step is to select the Maker workbench. To do this, click from the drop-down menu on the top bar and select Maker.

> **Note:** If MakerWorkbench does not appear, check that you have followed the installation steps correctly

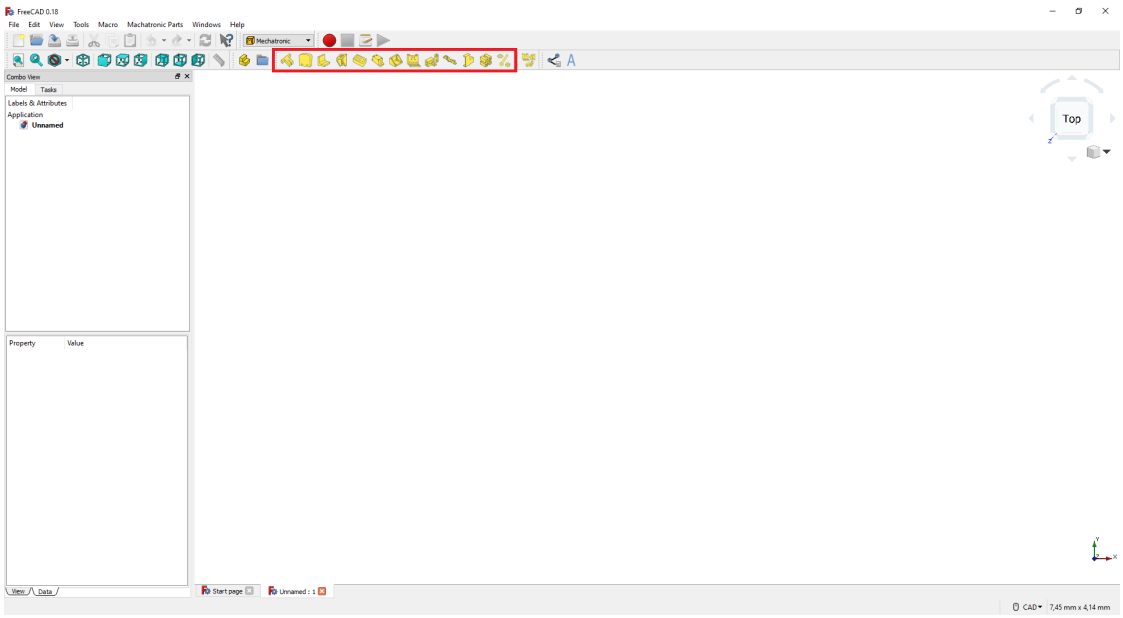




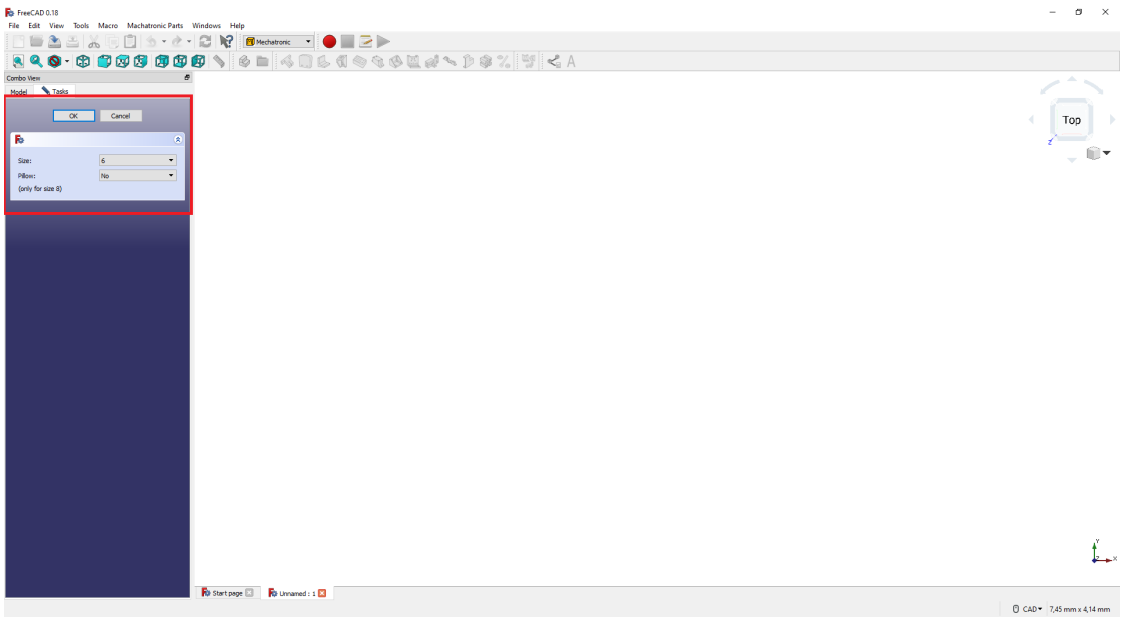
2- Once you have selected the MakerWorkbench workbench, open a new document, if you have not done so before.



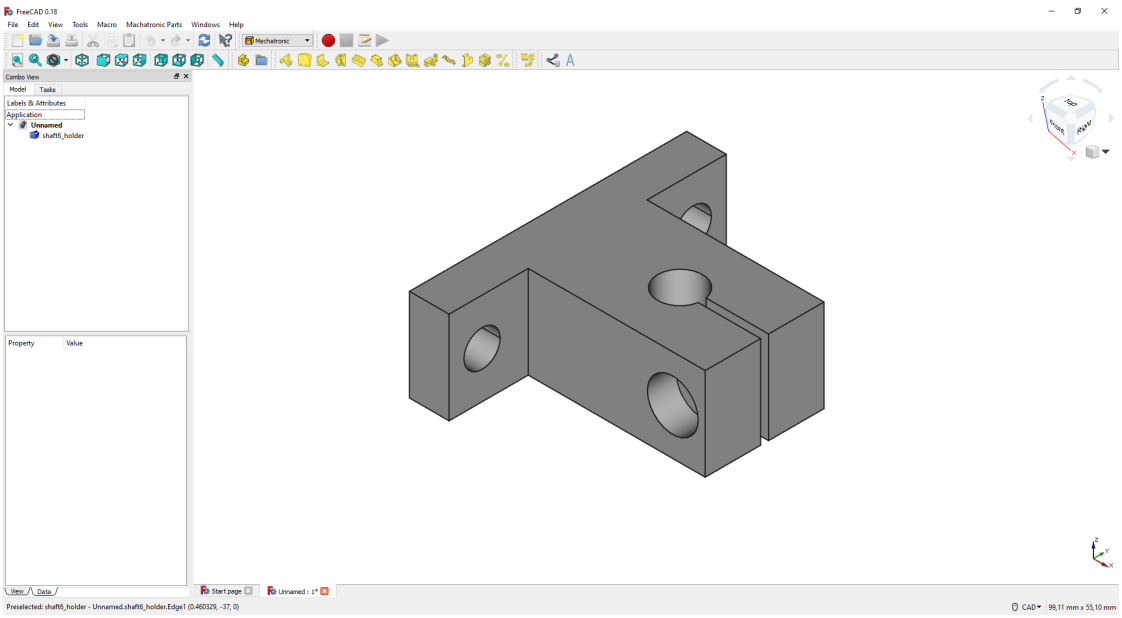
3- When you have selected the Maker tool bank, a set of icons should appear at the top. Select one of the models to make your first part. You can also select the *Maker Parts* menu to view the models available.



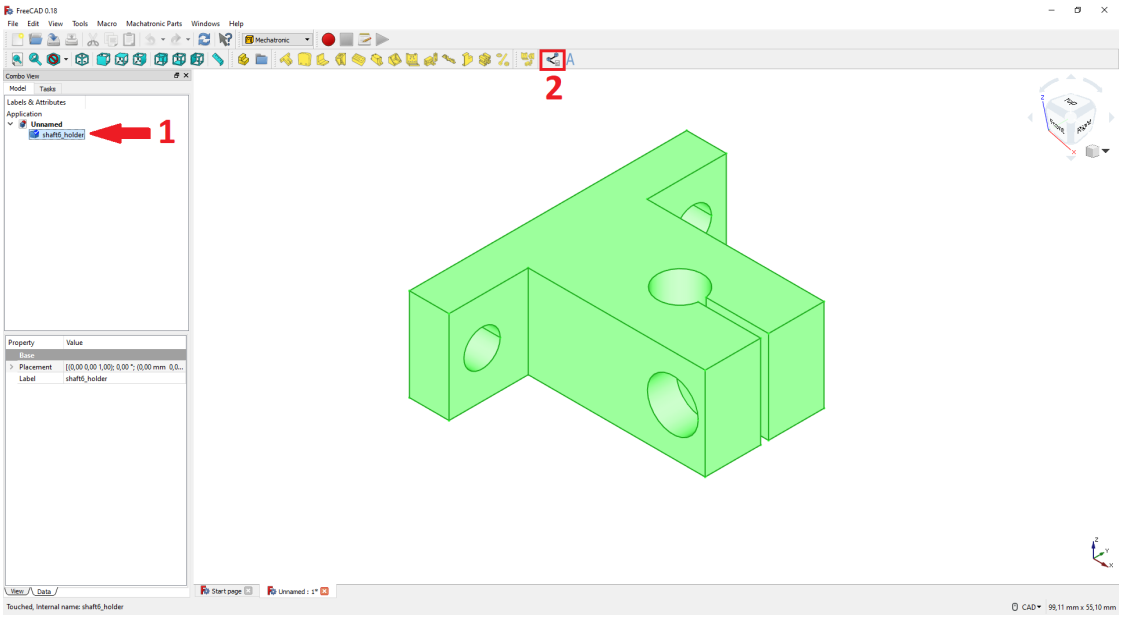
4- When one of the models is selected, the options to modify it will appear in the *Tasks* tab. Enter or select the values you want for the model. When you have finished, select *OK* to create the model.



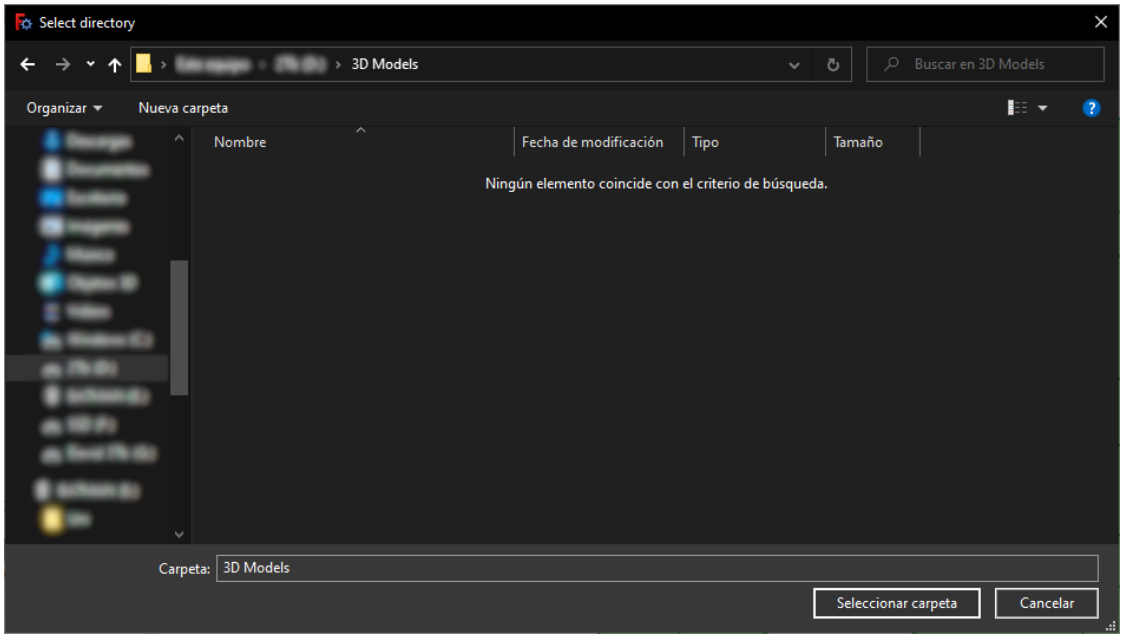
5- The model will be displayed with the options selected.



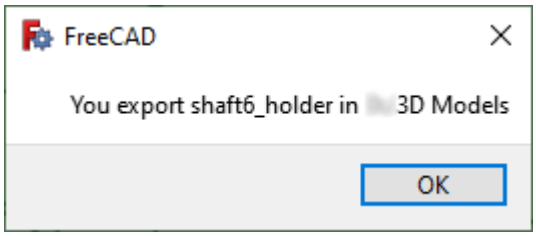
6- If you want to print the model, select it and then select the icon to export in STL format. This function also optimizes the orientation of the model for 3D printing



- A new window will be displayed where you can select the folder to save the model



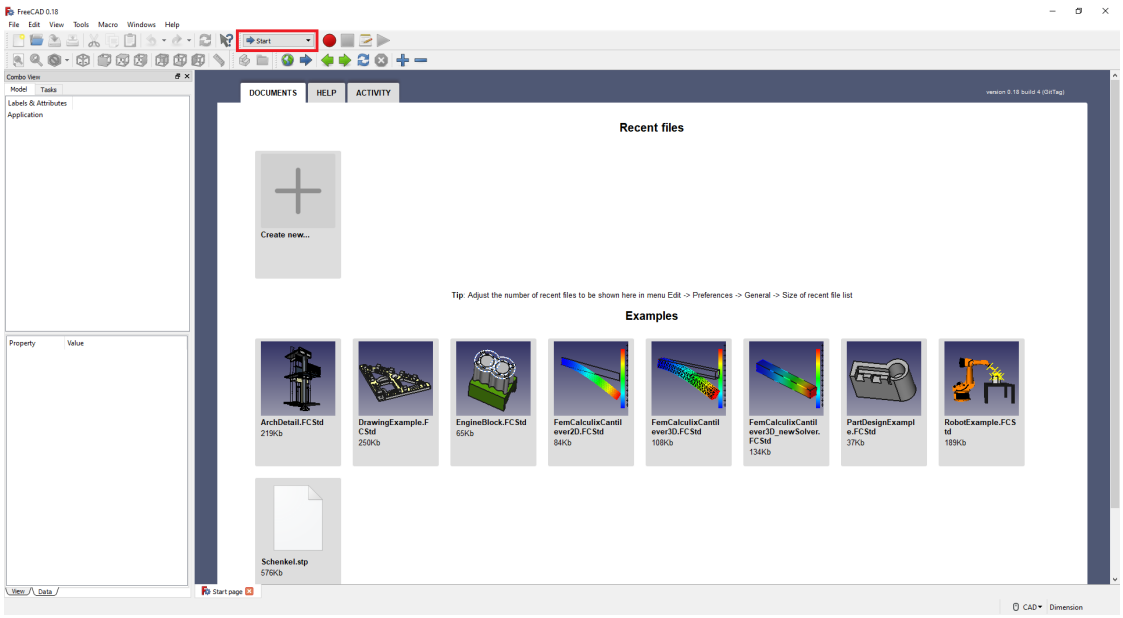
- The model will be saved with the name and in the folder shown in the

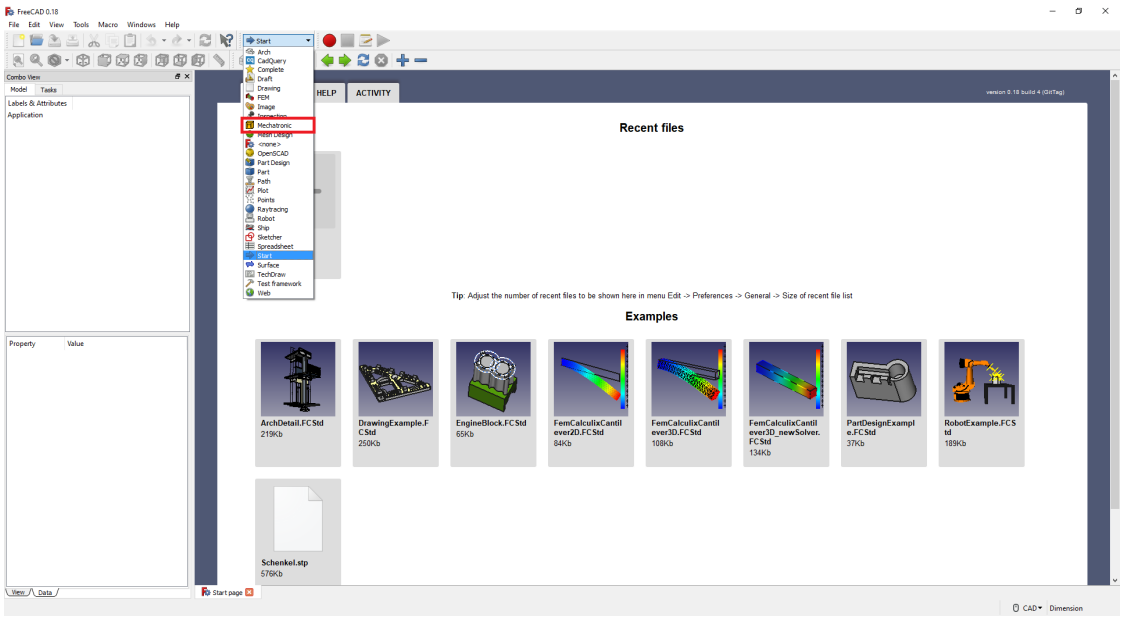


1.3.3 Tutorial CAD 2 - Crate a system

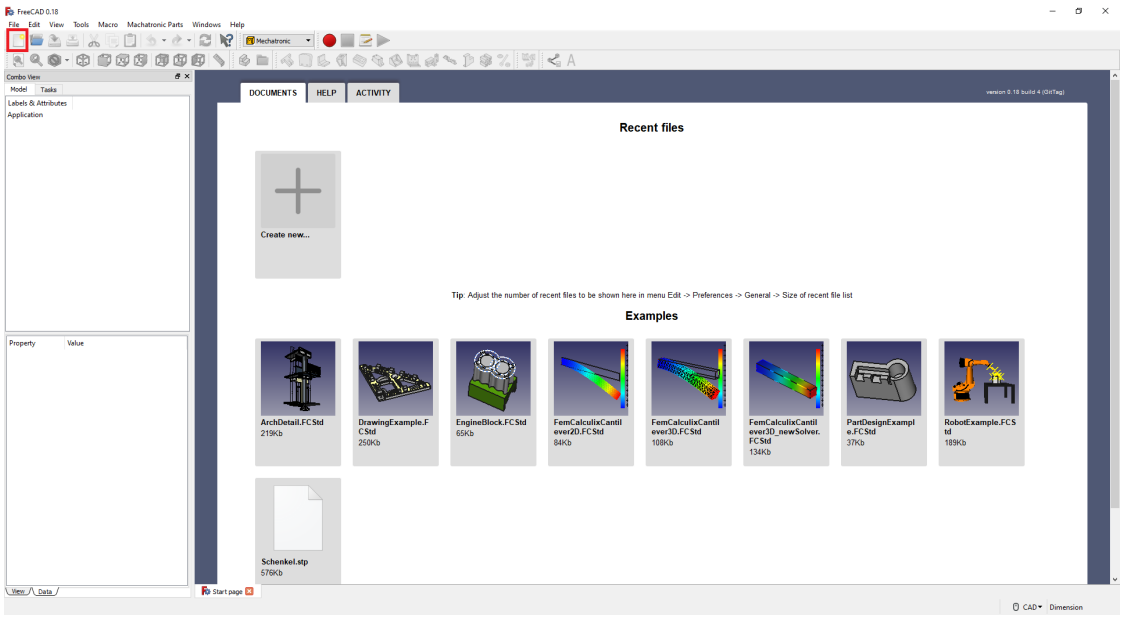
1- After opening FreeCAD, the first step is to select the Maker workbench. To do this, click from the drop-down menu on the top bar and select MakerWorkbench.

> **Note:** If MakerWorkbench does not appear, check that you have followed the installation steps correctly

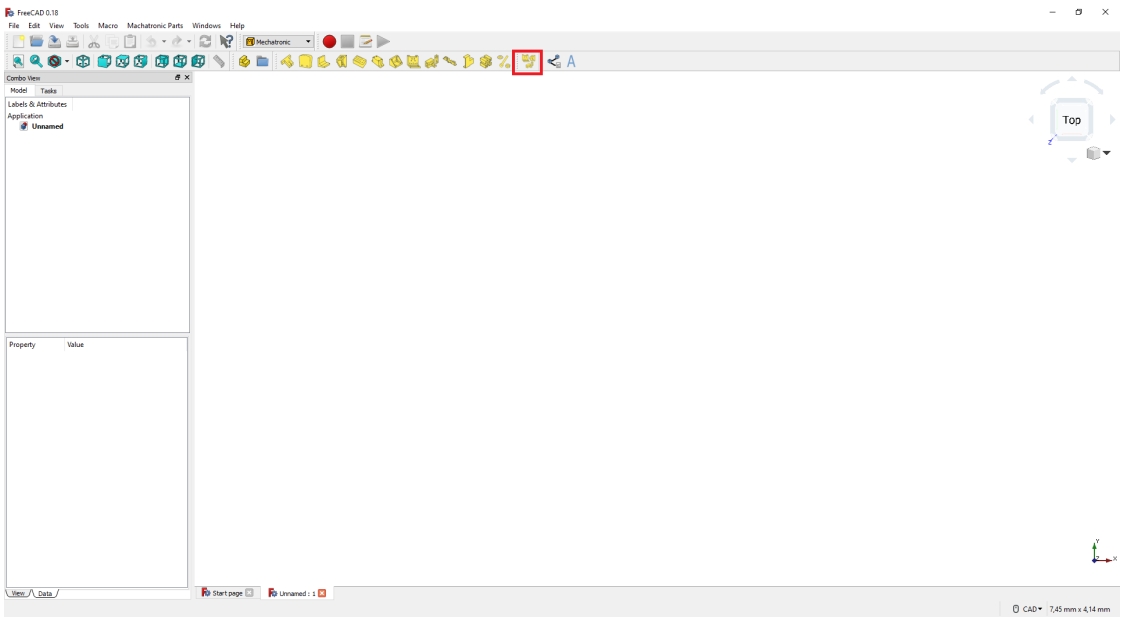




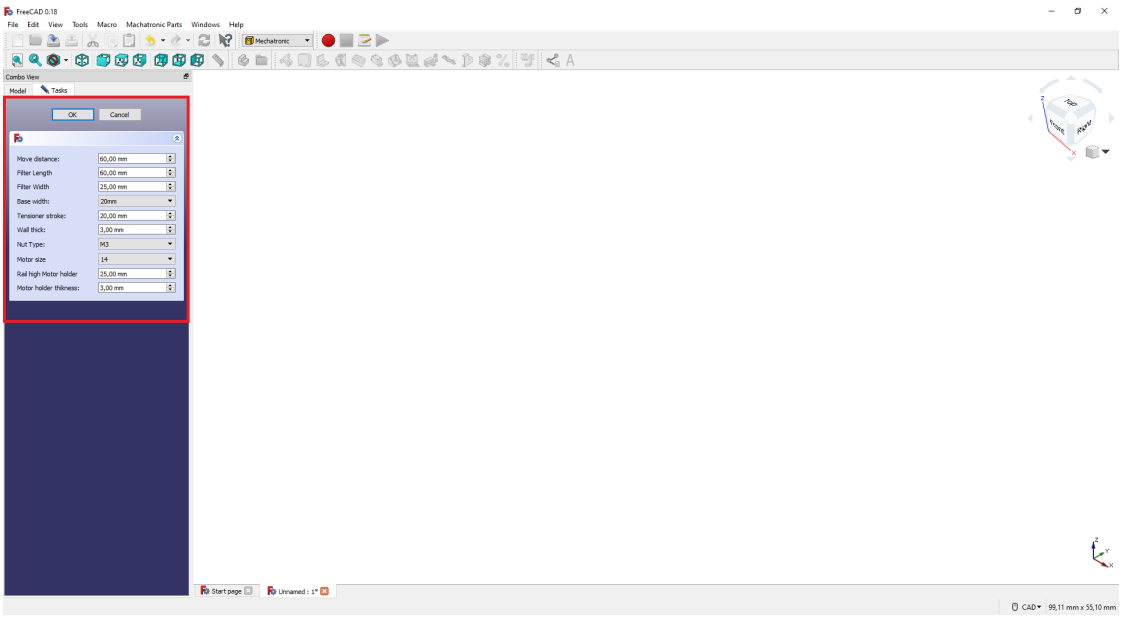
2- Once you have selected the Maker workbench, open a new document, if you have not done so before.



3- When you have selected the Maker tool bank, a set of icons should appear at the top. Select one of the systems to make your first part. You can also select the *Maker Systems* menu to view the systems available.



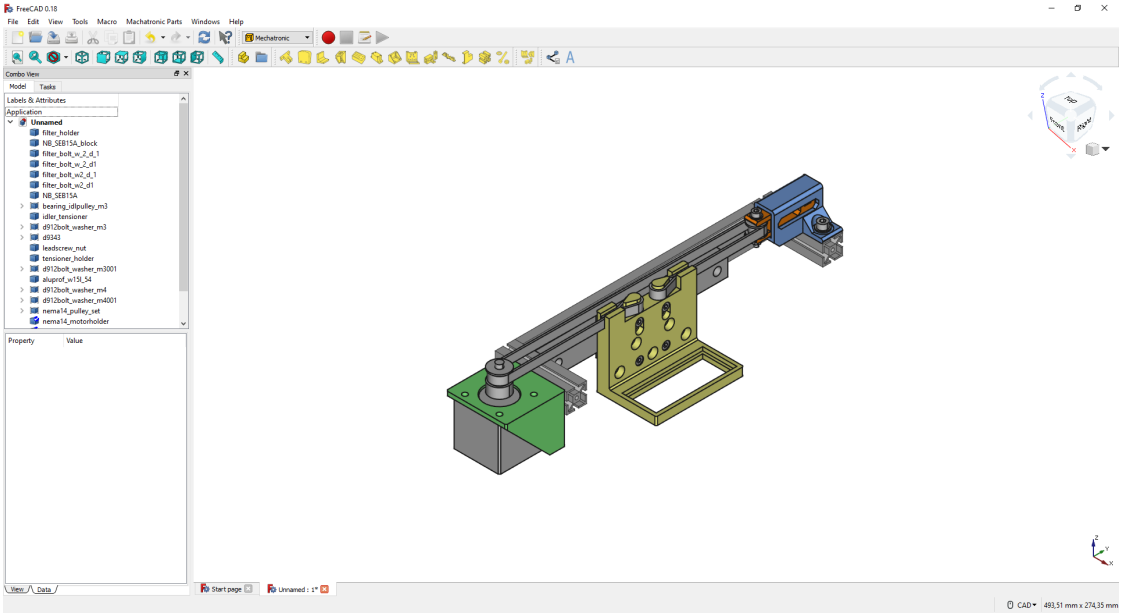
4- When one of the systems is selected, the options to modify it will appear in the *Tasks* tab. Enter or select the values you want for the system. When you have finished, select *OK* to create the system.



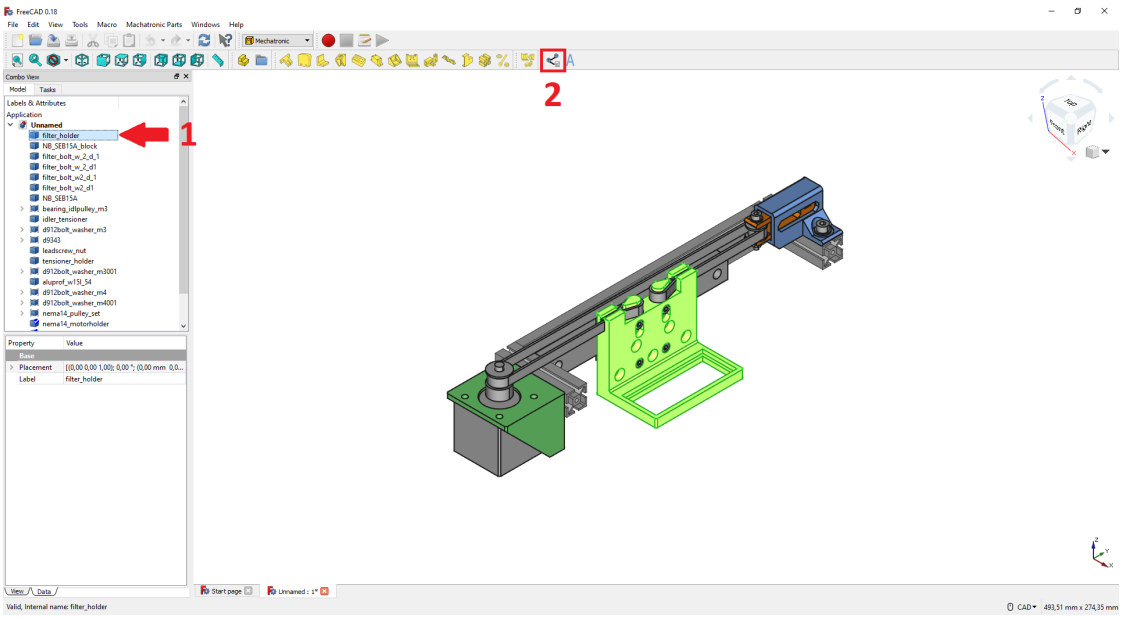
5- The system will be displayed with the options selected.

Note: This may take some time according to your computer hardware

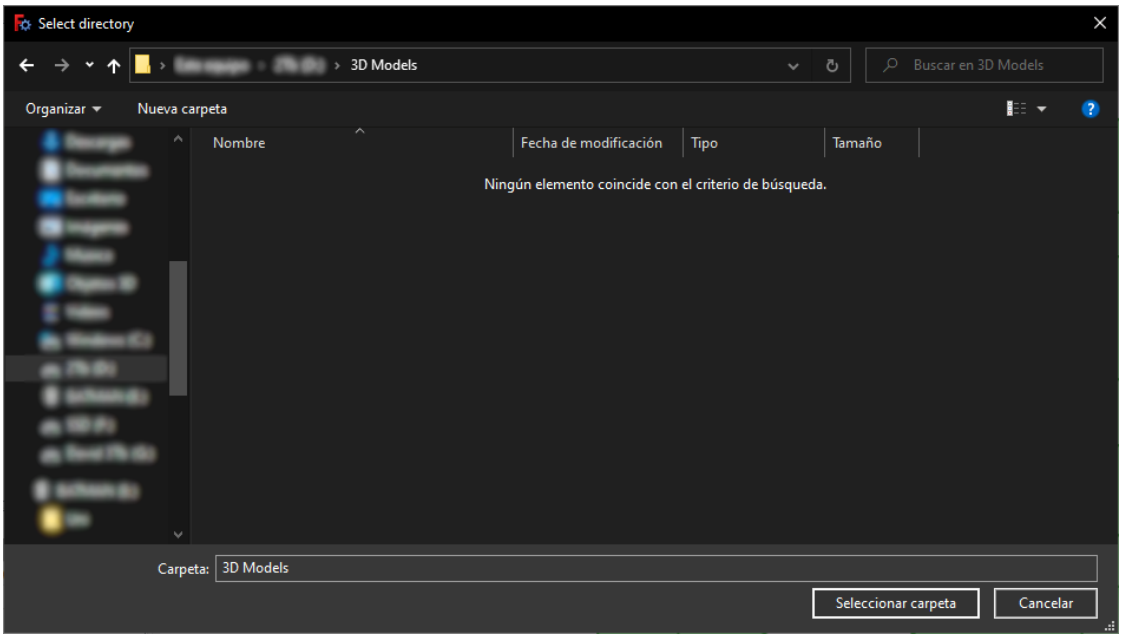
---



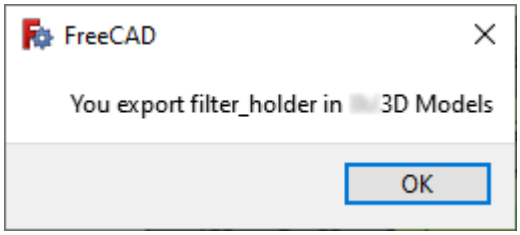
6- If you want to print one model of the system, select it and then select the icon to export in STL format. This function also optimizes the orientation of the model for 3D printing



- A new window will be displayed where you can select the folder to save the model



- The model will be saved with the name and in the folder shown in the



1.4 Wiki

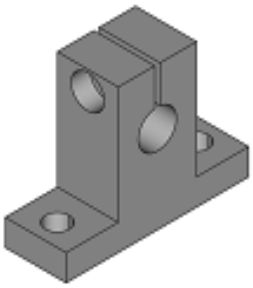
Note: This is a basic view of the Wiki

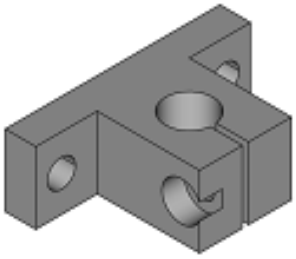
1.4.1 3D Model library

Maker Workbench

Shaft Holder

- Size
- Low profile: Only in size 8





Details

Sk_dir(size[, fc_axis_h, fc_axis_d, ...])	SK dimensions: dictionary for the dimensions .

Idler Holder

- Size of the profile on which it is mounted
- Bolt metrics
- Height
- Position of the limit switch sensor
- Height of the limit switch sensor

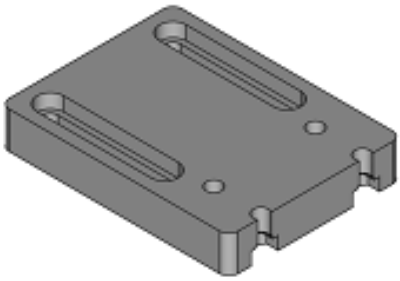
```
The model will be modified for greater efficiency
```

Details

IdlePulleyHolder(profile_size, pulleybolt_d, ...)	Creates a holder for a IdlePulley.

Limit Switches Holder

- Type
- Rail distance

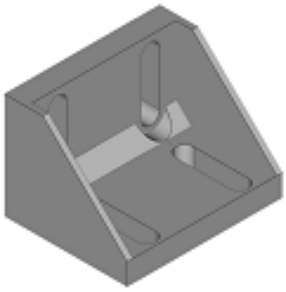


Details

SimpleEndstopHolder(d_endstop[, rail_l, ...])	Very simple endstop holder to be attached to a alu profile and that can be adjusted .

Hall stop

- Width
- Thikness
- Metric nut
- Profile size
- Reinforce

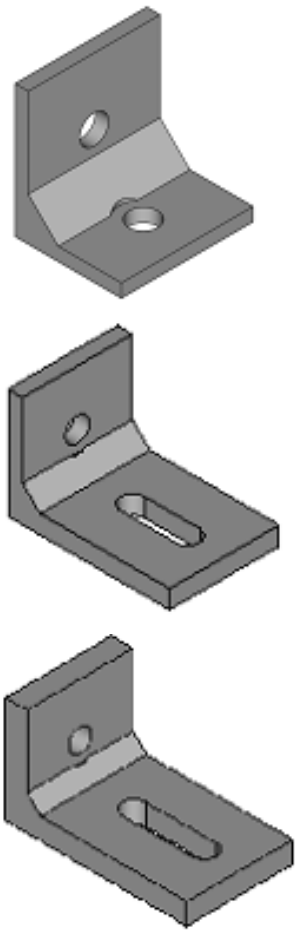


Details

`hallestop_holder`([stp_w, stp_h, base_thick, ...])

Bracket

- Type: 3 options
- Size first profile
- Size second profile
- Thickness
- Metric nut first profile
- Metric nut second profile
- Number of nuts
- Distance betwen nuts
- Type of hole
- Reinforcement: first type only
- Flap: second type only
- Distance between profiles: third type only

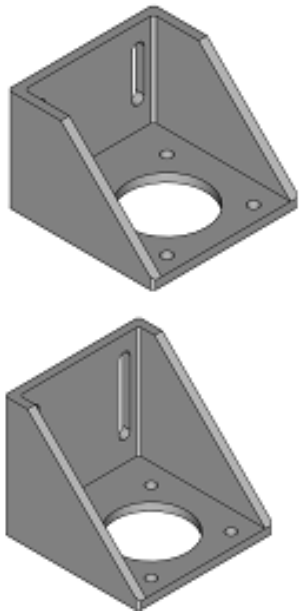


Details

AluProfBracketPerp(alusize_lin, alusize_perp)	Bracket to join 2 aluminum profiles that are perpendicular, that is, they are not on the same plane .
AluProfBracketPerpFlap(alusize_lin, alusize_perp)	Bracket to join 2 aluminum profiles that are perpendicular, that is, they are not on the same plane It is wide because it has 2 ears/flaps? on the sides, to attach to the perpendicular profile .
AluProfBracketPerpTwin(alusize_lin, . . . [, . . .])	Bracket to join 3 aluminum profiles that are perpendicular, that is, they are not on the same plane to the perpendicular profile .

Motor holder

- Size
- Height
- Thickness

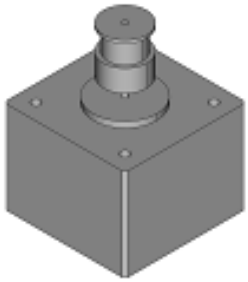


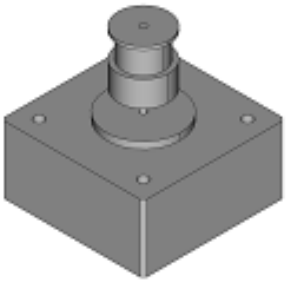
Details

NemaMotorHolder([nema_size, wall_thick, ...])	Creates a holder for a Nema motor

Motor

- Size
- Height
- Shaft height
- Shaft radius
- Shaft radius base
- Shaft height base
- Chamfer radius
- Bolt deep
- Bolt outside
- Pulley pitch
- Pulley teeth
- Pulley top flange
- Pulley bot flange
- Position in axis d
- Position in axis w
- Position in axis h
- Placement



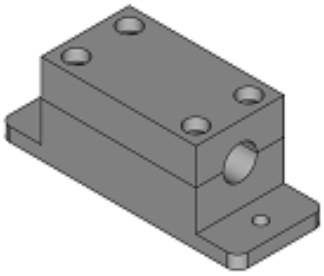


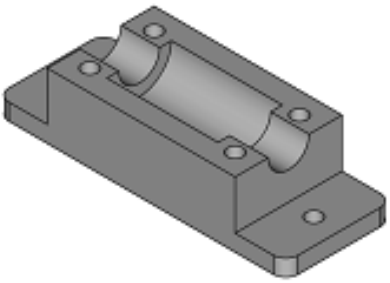
Details

NemaMotorPulleySet([nema_size, base_l, ...])	Set composed of a Nema Motor and a pulley

Lin bear house

- Type



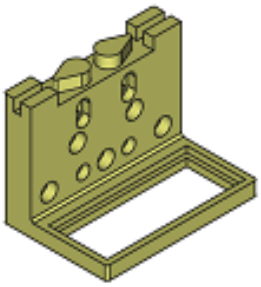


Details

ThinLinBearHouse1rail(d_lbear[, . . .])	Makes a housing for a linear bearing, but it is very thin and intended to be attached to one rail, instead of 2 it has to parts, the lower and the upper part .
ThinLinBearHouse(d_lbear[, fc_slide_axis, . . .])	Makes a housing for a linear bearing, but it is very thin and intended to be attached to 2 rail it has to parts, the lower and the upper part .
LinBearHouse(d_lbearhousing[, . . .])	Makes a housing for a linear bearing takes the dimensions from a dictionary, like the one defined in kcomp.py it has to parts, the lower and the upper part .
ThinLinBearHouseAsim(d_lbear[, fc_fro_ax, . . .])	There are

Filter holder

- Length
- Width

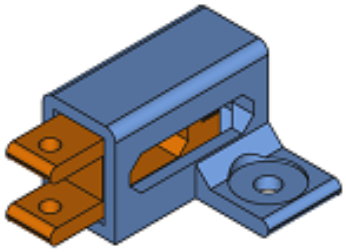


Details

PartFilterHolder([filter_l, filter_w, . . .])	Integration of a ShpFilterHolder object into a PartFilterHolder object, so it is a FreeCAD object that can be visualized in FreeCAD

Tensioner

- Belt hight
- Base width
- Thickness
- Metric nut

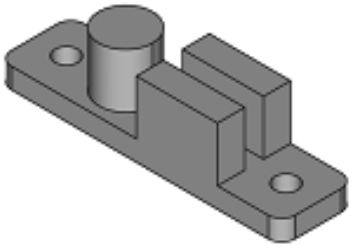


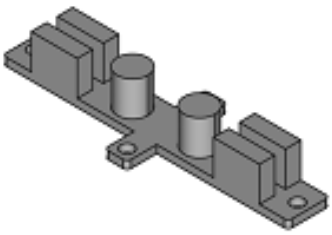
Details

TensionerSet([aluprof_w, belt_pos_h, ...])	Set composed of the idler pulley and the tensioner

Belt clamp

- Type
- Length
- Width
- Metric nut



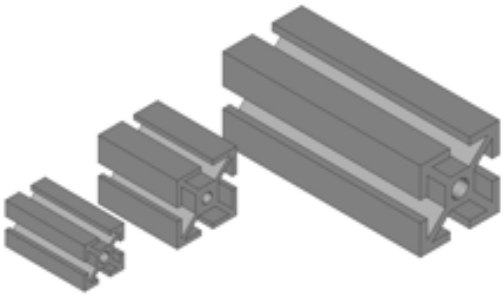


Details

BeltClamp(fc_fro_ax, fc_top_ax[, base_h, ...])	Similar to shp_topbeltclamp, but with any direction, and can have a base Creates a shape of a belt clamp.
DoubleBeltClamp([axis_h, axis_d, axis_w, ...])	Similar to BeltClamp, but in two ways Creates a shape of a double belt clamp.

Aluminium profile

- Section
- Length

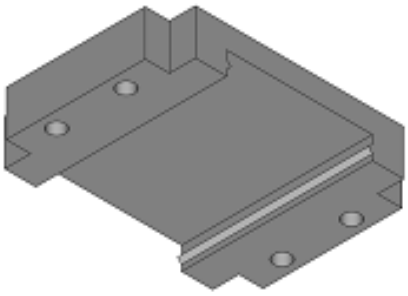


Details

PartAluProf(depth, aluprof_dict[, xtr_d, ...])	Integration of a ShpAluProf object into a PartAluProf object, so it is a FreeCAD object that can be visualized in FreeCAD Instead of using all the arguments of ShpAluProf, it will use a dictionary

Linear Guide

- Type:
  - SEBW16
  - SEB15A
  - SEB8
  - SEB10
- Position in axis d
- Position in axis w
- Position in axis h
- Placement

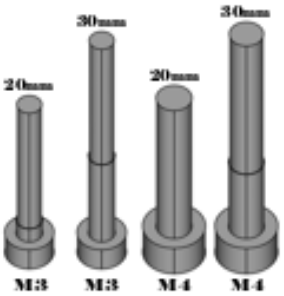


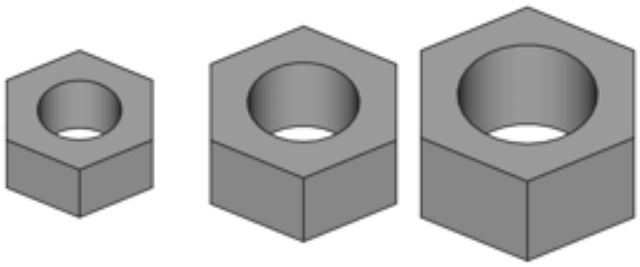
Details

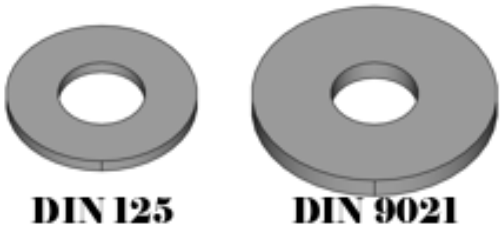
PartLinGuideBlock(block_dict, rail_dict[, ...])	Integration of a ShpLinGuideBlock object into a PartLinGuideBlock object, so it is a FreeCAD object that can be visualized in FreeCAD Instead of using all the arguments of ShpLinGuideBlock, it will use a dictionary

Bolts, Nuts & Washers

- Type
- Metric
- Bolt length







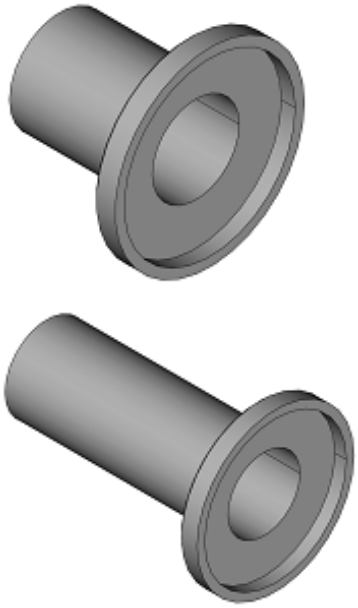
Details

Din934Nut(metric[, axis_d_apo, h_offset, ...])	Din 934 Nut
Din125Washer(metric, axis_h, pos_h[, tol, ...])	Din 125 Washer, this is the regular washer
Din9021Washer(metric, axis_h, pos_h[, tol, ...])	Din 9021 Washer, this is the larger washer
Din912Bolt(metric, shank_l[, ...])	Din 912 bolt.

Optical

TubeLense

- Length
- Placement

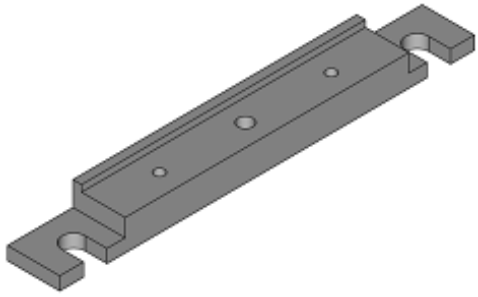


Details

SM1TubelensSm2(sm1l_size[, fc_axis, ...])	Creates a componente formed by joining: the lens tube SM1LXX + SM1A2 + SM2T2, so we have:

LCPB1M Base

- Placement

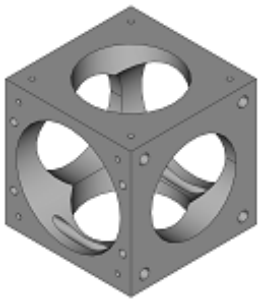


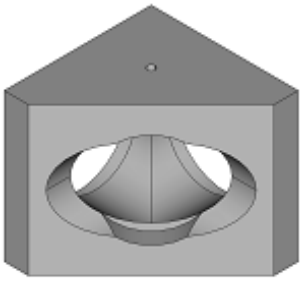
Details

lcpb1m_base([d_lcpb1m_base, fc_axis_d, ...])	Creates a lcpb1m_base for plates side, it creates from a dictionary

CageCube

- Type:
  - CageCube
  - CageCubeHalf



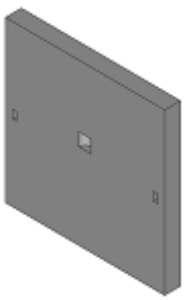


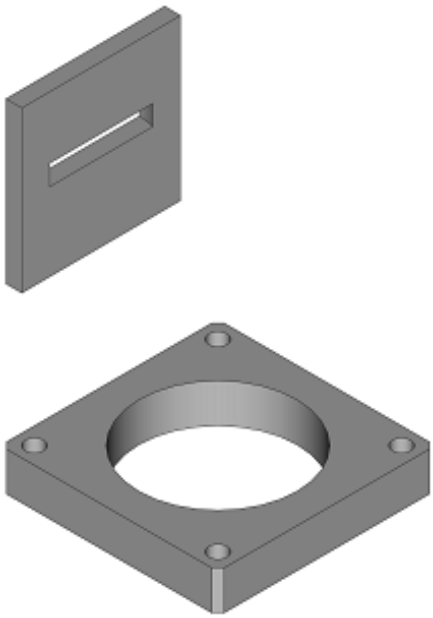
Details

f_cagecube(d_cagecube[, axis_thru_rods, ...])	Creates a cage cube, it creates from a dictionary
f_cagecubehalf(d_cagecubehalf[, axis_1, ...])	Dreates a half cage cube: 2 perpendicular sides, and a 45 degree angle side.

Plate

- Plate dictionary:
  - Lb1cm_Plate
  - Lb2c_Plate
  - Lcp01m_plate
- Placement



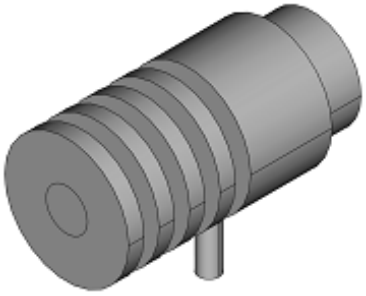


Details

Lb1cPlate(d_plate[, fc_axis_h, fc_axis_l, ...])	Creates a LB1C/M plate from thorlabs. The plate is centered
Lb2cPlate(fc_axis_h, fc_axis_l[, cl, cw, ...])	Same as plate_lb2c, but it creates an object.
lcp01m_plate([d_lcp01m_plate, fc_axis_h, ...])	Creates a lcp01m_plate side.

ThLed30

- Placement

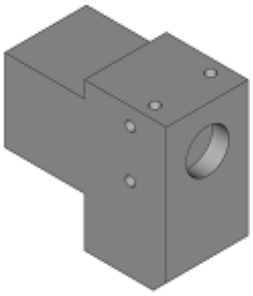


Details

ThLed30([fc_axis, fc_axis_cable, pos, name])	Creates the shape of a Thorlabs Led with 30.5 mm Heat Sink diameter The drawing is very rough .

PrizLed

- Placement

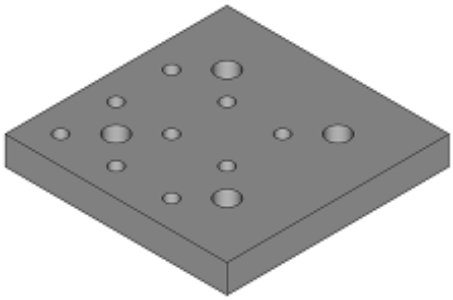


Details

PrizLed([fc_axis_led, fc_axis_clear, pos, name])	Creates the shape of a Prizmatix UHP-T-Led The drawing is very rough, and the original drawing lacks many dimensions .

BreadBoard

- Length
- Width
- Placement



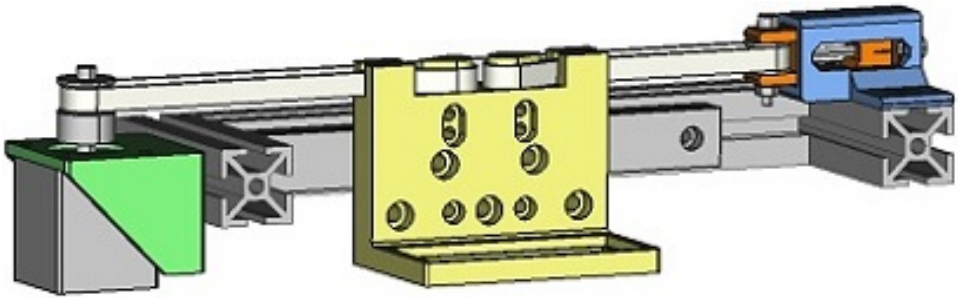
Details

f_breadboard(d_breadboard, length, width[, ...])	**param d_breadboard** Dictionary with the values

1.4.2 Systems library

Filter Stage

- Move distance
- Filter length
- Filter width
- Base width
- Tensioner stroke
- Tensioner thickness
- Metric nut
- Motor size
- Length rail motor holder
- Motor holder thickness



1.4.3 Functions Library

fcfun

NutHole(nut_r, nut_h, hole_h, name[, extra, ...])	Adding a Nut hole (hexagonal) with a prism attached to introduce the nut.
add2CylsHole(r1, h1, r2, h2, thick[, ...])	Creates a piece formed by 2 hollow cylinders
add3CylsHole(r1, h1, r2, h2, rring, hring, thick)	Creates a piece formed by 2 hollow cylinders, and a ring on the side of the larger cylinder
addBolt(r_shank, l_bolt, r_head, l_head[, ...])	Creates the hole for the bolt shank and the head or the nut Tolerances have to be included
addBoltNut_hole(r_shank, l_bolt, r_head, ...)	Creates the hole for the bolt shank, the head and the nut.
addBox(x, y, z, name[, cx, cy])	Adds a box, centered on the specified axis x and/or y, with its Placement and Rotation at zero.
addBox_cen(x, y, z, name[, cx, cy, cz])	Adds a box, centered on the specified axis, with its Placement and Rotation at zero.
addCyl(r, h, name)	Add cylinder
addCylHole(r_ext, r_int, h, name[, axis, h_disp])	Add cylinder, with inner hole:

continues on next page

Table 22 – continued from previous page

addCylHolePos(r_out, r_in, h, name[, ...])	Same as addCylHole, but avoiding the creation of many FreeCAD objects
addCylPos(r, h, name[, normal, pos])	Same as addCyl_pos, but avoiding the creation of many FreeCAD objects
addCyl_pos(r, h, name[, axis, h_disp])	Add cylinder in a position.
add_fcobj(shp, name[, doc])	Just creates a freeCAD object of the shape, just to save one line
aluprof_vec(width, thick, slot, insquare)	Creates a wire (shape), that is an approximation of a generic alum profile extrusion .
calc_desp_ncen(Length, Width, Height, vec1, vec2)	Similar to calc_rot, but calculates de displacement, when we don't want to have all of the dimensions centered First vector original direction (x,y,z) is (1,0,0) Second vector original direction (x,y,z) is (0,0,-1) The arguments vec1, vec2 are tuples (x,y,z) but they may be also FreeCAD.Vectors .
calc_rot(vec1, vec2)	Having an object with an orientation defined by 2 vectors the vectors a tuples, nor FreeCAD.Vectors use the wrapper fc_calc_rot to have FreeCAD.Vector arguments First vector original direction (x,y,z) is (1,0,0) Second vector original direction (x,y,z) is (0,0,-1) we want to rotate the object in an ortoghonal direction.
calc_rot_z(v_refz, v_refx)	Calculates de rotation like calc_rot.
edgeonaxis(edge, axis)	It tells if an edge is on an axis
equ(x, y)	Compare numbers that are the same but not exactly the same
fc_calc_desp_ncen(Length, Width, Height, ...)	Same as calc_desp_ncen but using FreeCAD.Vectors arguments
fc_calc_rot(fc_vec1, fc_vec2)	Same as calc_rot but using FreeCAD.Vectors arguments
fc_isonbase(fcv)	Just tells if a vector has 2 of the coordinates zero so it is on just a base vector
fc_isparal(fc1, fc2)	Return 1 if fc1 and fc2 are parallel (colinear), 0 if they are not
fc_isparal_nrm(fc1, fc2)	Very similar to fc_isparal, but in this case the arguments are normalized so, less operations to do.
fc_isperp(fc1, fc2)	Return 1 if fc1 and fc2 are perpendicular, 0 if they are not
fillet_len(box, e_len, radius, name)	Make a new object with fillet
filletchamfer(fco, e_len, name[, fillet, ...])	Fillet or chamfer edges of a certain length, on a certain axis and a certain coordinate
fuseshplist(shp_list)	Since multifuse methods needs to be done by a shape and a list, and usually I have a list that I want to fuse, I make this function to save the inconvenience of doing every time what I will do here Fuse multiFuse
get_bolt_bearing_sep(bolt_d, hasnut, lbearing_r)	same as get_bolt_end_sep, but when there is a bearing.
get_bolt_end_sep(bolt_d, hasnut[, sep])	Calculate Bolt separation
get_fc_perpend1(fcv)	gets a 'random' perpendicular FreeCAD.Vector
get_fclist_4perp2_fcvec(fcvec)	Gets a list of 4 FreCAD.Vector perpendicular to one base vector fcvec can only be: * (1,0,0) * (0,1,0) * (0,0,1) * (-1,0,0) * (0,-1,0) * (0,0,-1)

continues on next page

Table 22 – continued from previous page

get_fclist_4perp2_vecname(vecname)	Gets a list of 4 FreCAD.Vector perpendicular to one vecname different from get_fclist_4perp_vecname For example: .
get_fclist_4perp_fcvec(fcvec)	Gets a list of 4 FreeCAD.Vector perpendicular to one base vector fcvec can only be: * (1,0,0) * (0,1,0) * (0,0,1) * (-1,0,0) * (0,-1,0) * (0,0,-1)
get_fclist_4perp_vecname(vecname)	Gets a list of 4 FreCAD.Vector perpendicular to one vecname for example: .
get_fcvectup(tup)	Gets the FreeCAD.Vector of a tuple
get_nameofbasevec(fcvec)	From a base vector either: (1,0,0), (0,1,0), (0,0,1), (-1,0,0), (0,-1,0), (0,0,-1) Gets its name: 'x', 'y',....
get_positive_vecname(vecname)	It just get 'x' when vecname is 'x' or '-x', and the same for the others, because some functions receive only positive base vector
get_rot(v1, v2)	Calculate the rotation from v1 to v2 the difference with previous versions, such fc_calc_rot, calc_rot, calc_rot is that it is for any vector direction.
get_tangent_2circles(center1_pt, center2_pt, ...)	Returns a list of lists (matrix) with the 2 tangent points for each of the 2 tangent lines .
get_tangent_circle_pt(ext_pt, center_pt, ...)	Get the point of the tangent to the circle
get_vecname_perpend1(vecname)	Gets a perpendicular vecname
get_vecname_perpend2(vecname)	Gets the other perpendicular vecname (see get_vecname_perpend)
getfcvecofname(axis)	Returns the FreeCAD.Vecor of the vector name given
getvecofname(axis)	Get axis name renunrs the vector
regpolygon_dir_vecl(n_sides, radius, ...)	Similar to regpolygon_vecl but in any place and direction of the space calculates the vertices of a regular polygon.
regpolygon_vecl(n_sides, radius[, x_angle])	Calculates the vertices of a regular polygon.
rotateview([axisX, axisY, axisZ, angle])	Rotate the camara
shpRndRectWire([x, y, r, zpos])	Creates a wire (shape), that is a rectangle with rounded edges.
shp_2stadium_dir(length, r_s, r_l, h_tot, h_rl)	Makes to concentric stadiums, useful for making rails for bolts the length is the same for both.
shp_aluwire_dir(width, thick, slot, insquare)	Creates a wire (shape), that is an approximation of a generic alum profile extrusion.
shp_belt_dir(center_sep, rad1, rad2, height)	Makes a shape of 2 tangent circles (like a belt joining 2 circles).
shp_belt_wire_dir(center_sep, rad1, rad2[, ...])	Makes a shape of a wire with 2 circles and exterior tangent lines check here It is not easy to draw it well rad1 and rad2 can be exchanged, rad1 doesn't have to be larger.
shp_bolt(r_shank, l_bolt, r_head, l_head[, ...])	Similar to addBolt, but creates a shape instead of a FreeCAD Object Creates a shape of the bolt shank and head or the nut Tolerances have to be included if you want it for making a hole
shp_bolt_dir(r_shank, l_bolt, r_head, l_head)	Similar to shp_bolt, but it can be done in any direction Creates a shape, not a of a FreeCAD Object Creates a shape of the bolt shank and head or the nut Tolerances have to be included if you want it for making a hole

continues on next page

Table 22 – continued from previous page

shp_boltnut_dir_hole(r_shank, l_bolt, ...[, ...])	Similar to addBoltNut_hole, but in any direction and creates shapes, not FreeCAD Objects Creates the hole for the bolt shank, the head and the nut.
shp_box_dir(box_w, box_d, box_h[, ...])	Makes a shape of a box given its 3 dimensions: width, depth and height and the direction of the height and depth dimensions.
shp_box_dir_xtr(box_w, box_d, box_h[, ...])	Makes a shape of a box given its 3 dimensions: width, depth and height and the direction of the height and depth dimensions.
shp_box_rot(box_w, box_d, box_h[, axis_w, ...])	Makes a box with width, depth, height and then rotation will be referred to *axis_w = (1,0,0)* and *axis_nh = (0,0,-1)*.
shp_boxcen(x, y, z[, cx, cy, cz, pos])	Adds a shape of box, referenced on the specified axis, with its Placement and Rotation at zero.
shp_boxcenchmf(x, y, z, chmfrad[, fx, fy, ...])	Same as shp_boxcen but with a chamfered dimension
shp_boxcenfill(x, y, z, fillrad[, fx, fy, ...])	Same as shp_boxcen but with a filleted dimension
shp_boxcenxtr(x, y, z[, cx, cy, cz, xtr_nx, ...])	The same as shp_boxcen, but when it is used to cut.
shp_boxdir_fillchmfplane(box_w, box_d, box_h)	Creates a box shape (cuboid) along 3 axis.
shp_cableturn(d, w, thick_d, corner_r, ...)	Creates a shape of an electrical cable turn, in any direction But it is a shape in FreeCAD See function wire_cableturn .
shp_cir_fillchmf(shp[, circen_pos, fillet, ...])	Fillet or chamfer edges that is a circle, the shape has to be a cylinder
shp_cyl(r, h[, normal, pos])	Same as addCylPos, but just creates the shape
shp_cyl_gen(r, h[, axis_h, axis_ra, ...])	This is a generalization of shp_cylcenxtr.
shp_cylcenxtr(r, h[, normal, ch, xtr_top, ...])	Add cylinder, can be centered on the position, and also can have an extra mm on top and bottom to make cuts
shp_cylfilletchamfer(shp[, fillet, radius])	Fillet or chamfer all edges of a cylinder
shp_cylhole(r_ext, r_int, h[, axis, h_disp])	Same as addCylHole, but just a shape
shp_cylhole_arc(r_out, r_in, h[, axis_h, ...])	This is similar to make shp_cylhole_gen but not for a whole, just an arc.
shp_cylhole_bolthole(r_out, r_in, h[, ...])	This is a generalization of shp_cylholedir and shp_cylhole Makes a hollow cylinder in any position and direction, with optional extra heights, and inner and outer radius, and various locations in the cylinder
shp_cylhole_gen(r_out, r_in, h[, axis_h, ...])	This is a generalization of shp_cylholedir.
shp_cylholedir(r_out, r_in, h[, normal, pos])	Same as addCylHolePos, but just a shape Same as shp_cylhole, but this one accepts any normal
shp_extrud_face(face, length, vec_extr_axis)	Extrudes a face on any plane
shp_extrud_face_rot(face, vec_facenormal, ...)	Extrudes a face that is on plane XY, includes a rotation
shp_face_lgrail(rail_w, rail_h[, axis_l, axis_b])	Adds a shape of the profile (face) of a linear guide rail, the dent is just rough, to be able to see that it is a profile .
shp_face_rail(rail_w, rail_ws, rail_h[, ...])	Adds a shape of the profile (face) of a rail
shp_filletchamfer(shp, e_len[, fillet, ...])	Fillet or chamfer edges of a certain length, on a certain axis and a certain coordinate
shp_filletchamfer_dir(shp[, fc_axis, ...])	Fillet or chamfer edges on a certain axis
shp_filletchamfer_dirpt(shp[, fc_axis, ...])	Fillet or chamfer edges on a certain axis and a point contained in that axis

continues on next page

Table 22 – continued from previous page

shp_filletchamfer_dirpts(shp, fc_axis, fc_pts)	Fillet or chamfer edges on a certain axis and a list of point contained in that axis
shp_filletchamfer_dirs(shp, fc_axis_l[, ...])	Same as shp_filletchamfer_dir, but with a list of directions
shp_hollowbelt_dir(center_sep, rad1, rad2, ...)	Makes a shape of 2 tangent circles (like a belt joining 2 circles).
shp_nuthole(nut_r, nut_h, hole_h[, xtr_nut, ...])	Similar to NutHole, but creates a shape, in any direction.
shp_regpolygon_dir_face(n_sides, radius[, ...])	Similar to shp_regpolygon_face, but in any direction of the space makes the shape of a face of a regular polygon
shp_regpolygon_face(n_sides, radius[, ...])	Makes the shape of a face of a regular polygon
shp_regprism(n_sides, radius, length[, ...])	Makes a shape of a face of a regular polygon
shp_regprism_dirxtr(n_sides, radius, length)	Similar to shp_regprism_xtr, but in any direction makes a shape of a face of a regular polygon.
shp_regprism_xtr(n_sides, radius, length[, ...])	makes a shape of a face of a regular polygon.
shp_rndrect_face(x, y[, r, pos_z])	Same as shpRndRectWire
shp_stadium_dir(length, radius, height[, ...])	Makes a stadium shape in any direction
shp_stadium_face(l, r[, axis_rect, pos_z])	Same as shp_stadium_wire, but returns a face
shp_stadium_wire(l, r[, axis_rect, pos_z])	Creates a wire (shape), that is a rectangle with semicircles at a pair of opposite sides.
shp_stadium_wire_dir(length, radius[, ...])	Same as shp_stadium_wire but in any direction Also called discorectangle .
vecname_paral(vec1, vec2)	Given to vectors by name 'x', '-x', ...
wire_beltclamp(d, w, corner_r, conn_d, conn_sep)	Creates a wire following 2 pulleys and ending in a belt clamp But it is a wire in FreeCAD, has no volumen .
wire_cableturn(d, w, corner_r, conn_d, conn_sep)	Creates a electrical wire turn, in any direction But it is a wire in FreeCAD, has no volumen .
wire_lgrail(rail_w, rail_h[, axis_w, ...])	Creates a wire of a linear guide rail, the dent is just rough, to be able to see that it is a profile
wire_sim_xy(vecList)	Creates a wire (shape), from a list of points on the positive quadrant of XY the wire is symmetrical to both X and Y .

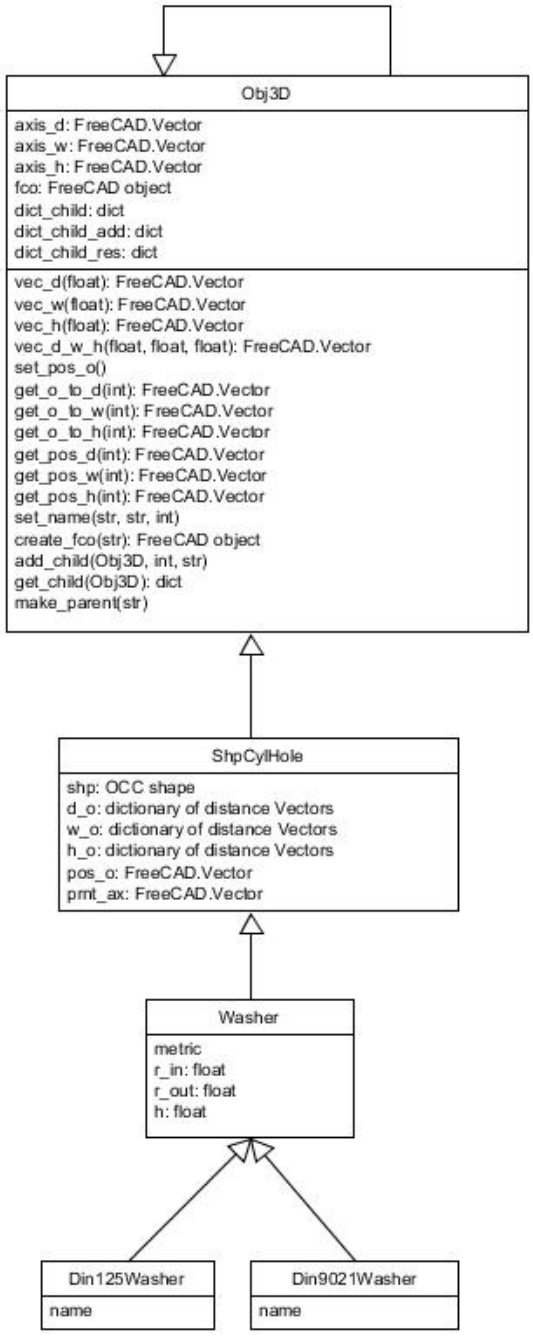
1.4.4 UML

The UML (Unified Modeling Language) is the base diagram for software development. It is a visual description of the relationships between class objects.

The main class will be "*Obj3D*" which will contain the basic information of the model:

- Internal axis:
  - axis_d
  - axis_w
  - axis_h
- Children's dictionary:
  - dict_child
  - dict_child_sum
  - dict_child_res

The rest of the classes that generate the different 3D models will be part of the *Obj3D* class


Obj3D
axis_d: FreeCAD.Vector
axis_w: FreeCAD.Vector
axis_h: FreeCAD.Vector
fco: FreeCAD object
dict_child: dict
dict_child_add: dict
dict_child_res: dict
vec_d(float): FreeCAD.Vector
vec_w(float): FreeCAD.Vector
vec_h(float): FreeCAD.Vector
vec_d_w_h(float, float, float): FreeCAD.Vector
set_pos_o()
get_o_to_d(int): FreeCAD.Vector
get_o_to_w(int): FreeCAD.Vector
get_o_to_h(int): FreeCAD.Vector
get_pos_d(int): FreeCAD.Vector
get_pos_w(int): FreeCAD.Vector
get_pos_h(int): FreeCAD.Vector
set_name(str, str, int)
create_fco(str): FreeCAD object
add_child(Obj3D, int, str)
get_child(Obj3D): dict
make_parent(str)
ShpCylHole
shp: OCC shape
d_o: dictionary of distance Vectors
w_o: dictionary of distance Vectors
h_o: dictionary of distance Vectors
pos_o: FreeCAD.Vector
prnt_ax: FreeCAD.Vector
Washer
metric
r_in: float
r_out: float
h: float
Din125Washer
name
Din9021Washer
name

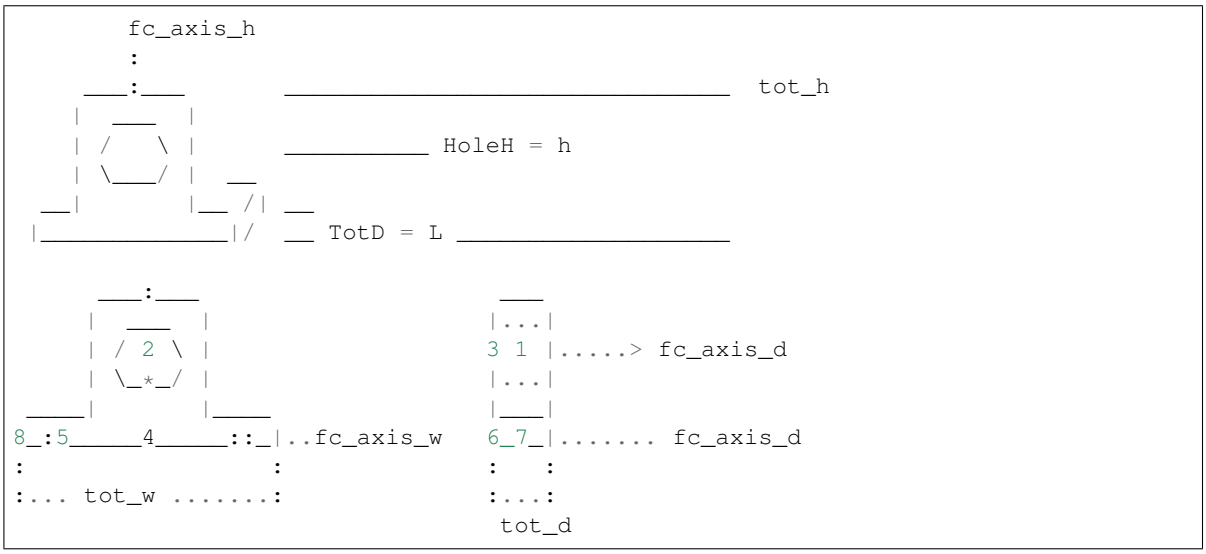
1.4.5 3D model details

Mechanical

class comps.**Sk_dir**(*size*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_d=FreeCAD.Vector*, *fc_axis_w=FreeCAD.Vector*, *ref_hr=1*, *ref_wc=1*, *ref_dc=1*, *pillow=0*, *pos=FreeCAD.Vector*, *wfco=1*, *tol=0.3*, *name='shaft_holder'*)

SK dimensions: dictionary for the dimensions

```
mbolt: is mounting bolt. it corresponds to its metric
tbolt: is the tightening bolt.
SK12 = { 'd':12.0, 'H':37.5, 'W':42.0, 'L':14.0, 'B':32.0, 'S':5.5,
         'h':23.0, 'A':21.0, 'b': 5.0, 'g':6.0,  'I':20.0,
         'mbolt': 5, 'tbolt': 4}
```



Parameters

- **fc_axis_h** (*FreeCAD.Vector*) – Axis on the height direction
- **fc_axis_d** (*FreeCAD.Vector*) – Axis on the depth (rod) direction
- **fc_axis_w** (*FreeCAD.Vector*) – Width (perpendicular) dimension, only useful if I finally include the tightening bolt, or if ref_wc != 1
- **ref_hr** (*int*) –
  - 1: reference at the Rod Height dimension (rod center): points 1, 2, 3
  - 0: reference at the base: points 4, 5
- **ref_wc** (*int*) –
  - 1: reference at the center on the width dimension (fc_axis_w) points: 2, 4,
  - 0: reference at one of the bolt holes, point 5
  - -1: reference at one end. point 8
- **ref_dc** (*int*) –

- 1: reference at the center of the depth dimension (fc_axis_d) points: 1,7
- 0: reference at one of the ends on the depth dimension points 3, 6

- **pillow** (*int*) –
  - 1 to make it the same height of a pillow block
- **pos** (*FreeCAD.Vector*) – Placement
- **wfco** (*int*) –
  - 1 to create a FreeCAD Object
- **tol** (*float*) – Tolerance of the axis
- **name** (*str*) – FreeCAD Object name

Returns FreeCAD Object of a shaft holder

Return type FreeCAD Object

class comps.**PartAluProf**(*depth*, *aluprof_dict*, *xtr_d=0*, *xtr_nd=0*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *axis_h=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos_h=0*, *pos=FreeCAD.Vector*, *model_type=1*, *name=''*)

Integration of a ShpAluProf object into a PartAluProf object, so it is a FreeCAD object that can be visualized in FreeCAD Instead of using all the arguments of ShpAluProf, it will use a dictionary

Parameters

- **depth** (*float*) – (depth) length of the bar, the extrusion
- **aluprof_dict** (*dictionary*) – Dictionary with all the information about the profile in kcomp.py there are some dictionaries that can be used, they are not exact – same as ShpAluProf
- **xtr_d** (*float*) – If >0 it will be that extra depth (length) on the direction of axis_d
- **xtr_nd** (*float*) – If >0 it will be that extra depth (length) on the opositve direction of axis_d can be V0 if pos_h = 0
- **axis_d** (*FreeCAD.Vector*) – Axis along the length (depth) direction
- **axis_w** (*FreeCAD.Vector*) – Axis along the width direction
- **axis_h** (*FreeCAD.Vector*) – Axis along the width direction
- **pos_d** (*int*) – Location of pos along axis_d (see drawing)
  - 0: start point, counting xtr_nd, if xtr_nd == 0 -> pos_d 0 and 1 will be the same
  - 1: start point, not counting xtr_nd
  - 2: middle point not conunting xtr_nd and xtr_d
  - 3: middle point conunting xtr_nd and xtr_d
  - 4: end point, not counting xtr_d
  - 5: end point considering xtr_d
- **pos_w** (*int*) – Location of pos along axis_w (see drawing). Symmetric, negative indexes means the other side
  - 0: at the center of symmetry
  - 1: at the inner square
  - 2: at the interior side of the outer part of the rail (thickness of the4 side)

- 3: at the end of the profile along axis_w

- **pos_h** (*int*) – Same as pos_w
- **pos** (*FreeCAD.Vector*) – Position of point defined by pos_d, pos_w, pos_h
- **model_type** (*int*) – Kind of model
  - 1: dimensional model: it can be printed to assemble a model,but the part will not work as defined. For example, if printed this aluminum profile will not work as defined, and also, it is not exact
- **name** (*str*) – Name of the object

class comps.**PartLinGuideBlock**(*block_dict*, *rail_dict*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *axis_h=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos_h=0*, *pos=FreeCAD.Vector*, *model_type=1*, *name=''*)

Integration of a ShpLinGuideBlock object into a PartLinGuideBlock object, so it is a FreeCAD object that can be visualized in FreeCAD Instead of using all the arguments of ShpLinGuideBlock, it will use a dictionary

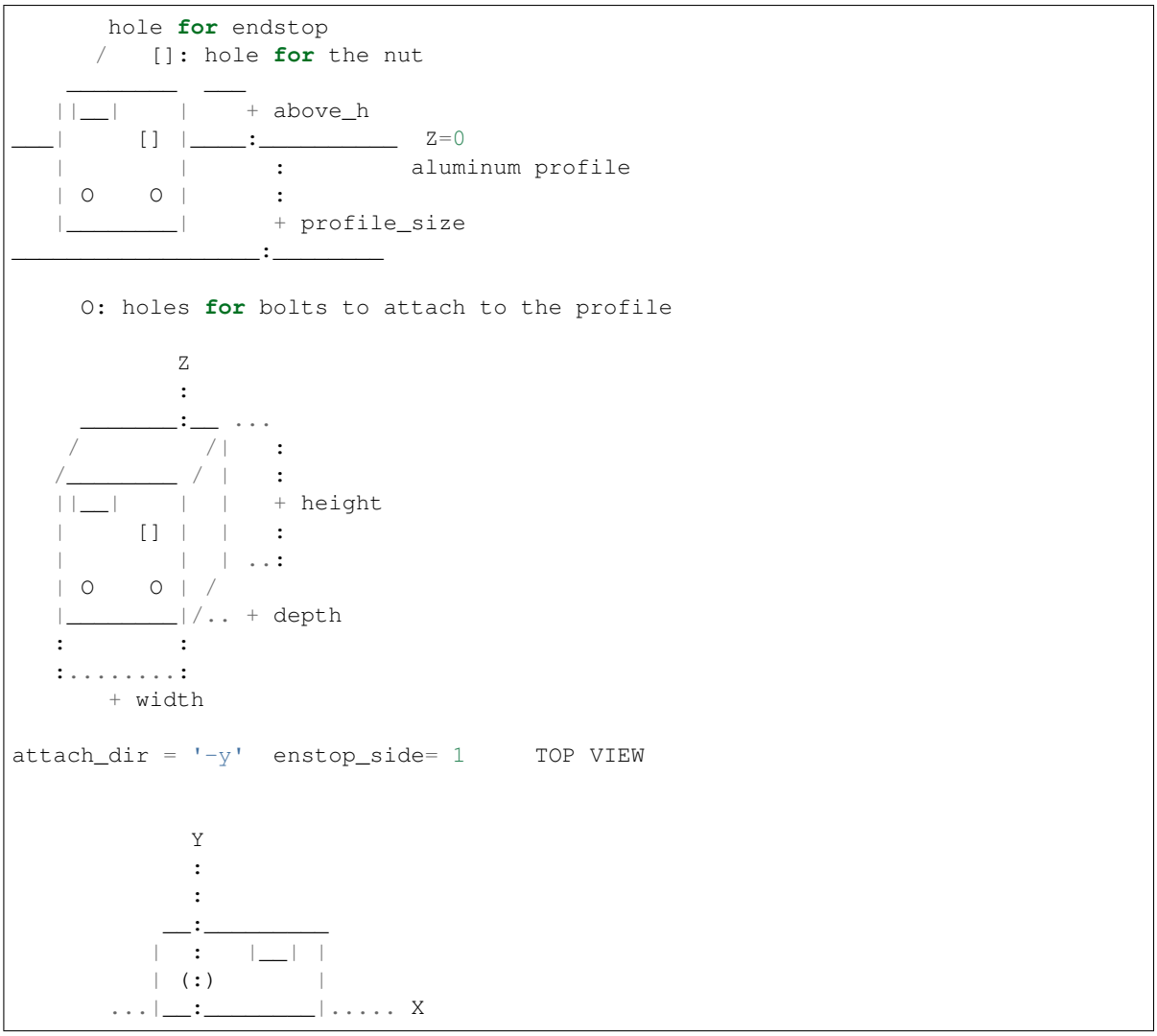
Parameters

- **block_dict** (*dictionary*) – Dictionary with the information about the block
- **rail_dict** (*dictionary*) – Dictionary with the information about the rail, it is not necessary, but if not provided, the block will not have the rail hole
- **axis_d** (*FreeCAD.Vector*) – The axis along the depth (length) of the block (and rail)
- **axis_w** (*FreeCAD.Vector*) – The axis along the width of the block
- **axis_h** (*FreeCAD.Vector*) – The axis along the height of the block, pointing up
- **pos_d** (*int*) – Location of pos along axis_d (see drawing). Symmetric, negative indexes means the other side
  - 0: at the center (symmetric)
  - 1: at the bolt hole
  - 2: at the end of the smaller part of the block
  - 3: at the end of the end of the block
- **pos_w** (*int*) – Location of pos along axis_w (see drawing). Symmetric, negative indexes means the other side
  - 0: at the center of symmetry
  - 1: at the inner hole of the rail
  - 2: at the bolt holes (it can be after the smaller part of the block)
  - 3: at the end of the smaller part of the block
  - 4: at the end of the end of the block
- **pos_h** (*int*) – Location of pos along axis_h (see drawing)
  - 0: at the bottom (could make more sense to have 0 at the top instead
  - 1: at the top of the rail hole
  - 2: at the bottom of the bolt holes, if thruholes, same as 0
  - 3: at the top end
  - 4: at the bottom of the rail (not the block), if the rail has been defined

- **pos** (*FreeCAD.Vector*) – Position at the point defined by pos_d, pos_w, pos_h

class parts.**IdlePulleyHolder**(*profile_size*, *pulleybolt_d*, *holdbolt_d*, *above_h*, *rail=0*, *mindepth=0*, *attach_dir='-y'*, *endstop_side=0*, *endstop_posh=0*, *pos=FreeCAD.Vector*, *name='idlepulleyhold'*)

Creates a holder for a IdlePulley. Usually made of bolts, washers and bearings It may include a space for a endstop It is centered at the idle pulley, but at the back, and at the profile height



Parameters

- **profile_size** (*float*) – Size of the aluminum profile. 20mm, 30mm
- **pulleybolt_d** (*float*) – Diameter of the bolt used to hold the pulley
- **holdbolt_d** (*float*) – Diameter of the bolts used to attach this part to the aluminum profile
- **above_h** (*float*) – Height of this piece above the aluminum profile
- **rail** (*float*) – Possibility of having a rail instead of holes for mounting the holder. It has been made fast, so there may be bugs
- **mindepth** (*float*) – If there is a minimum depth. Sometimes needed for the endstop to

reach its target

- **attach_dir** (*str*) – Normal vector to where the holder is attached:'x','-x','y','-y' NOW ONLY -y IS SUPPORTED. YOU CAN ROTATE IT
- **endstop_side** (*int*) – -1, 0, 1. Side where the enstop will be if attach_dir= 'x', this will be referred to the y axis if 0, there will be no endstop
- **endstop_h** (*float*) – Height of the endstop. If 0 it will be just on top of the profile
- **pos** (*FreeCAD.Vector*) – Object Placement

depth
Depth of the holder

Type float

width
Width of the holder

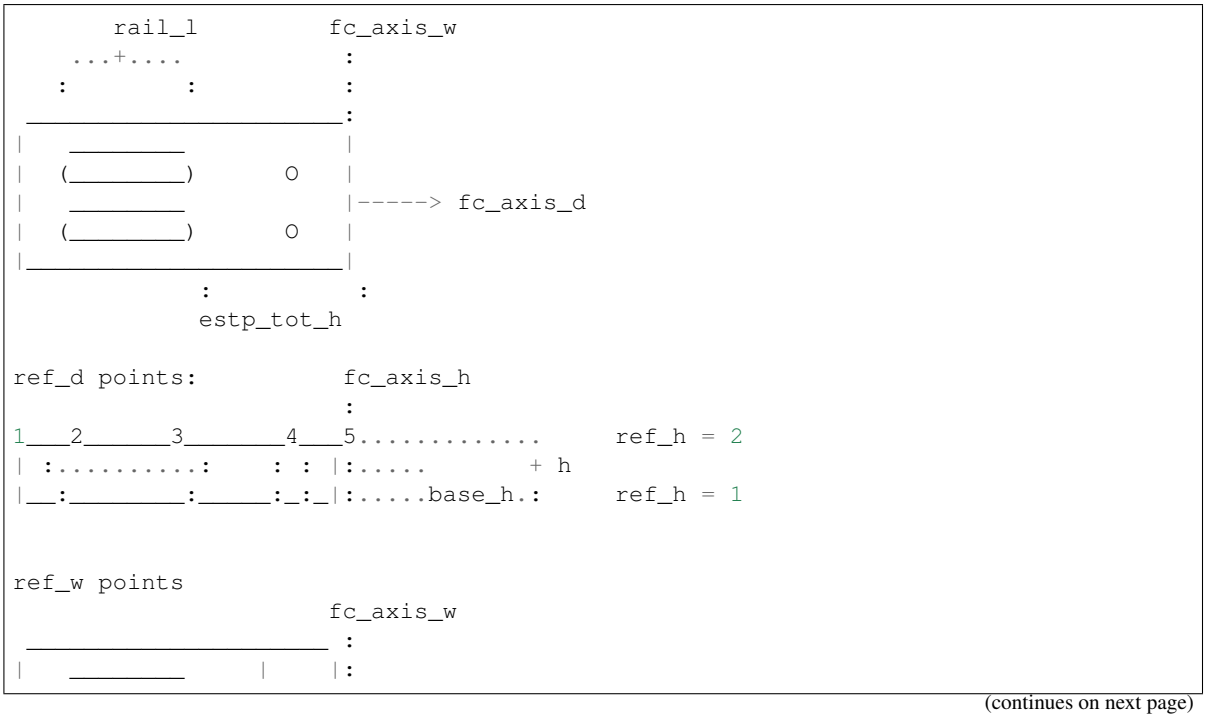
Type float

height
Height of the holder

Type float

fcoFat
Cad object of the compound

class parts.**SimpleEndstopHolder**(*d_endstop*, *rail_l=15*, *base_h=5.0*, *h=0*, *holder_out=2.0*, *mbolt_d=3.0*, *endstop_nut_dist=0*, *min_d=0*, *fc_axis_d=FreeCAD.Vector*, *fc_axis_w=FreeCAD.Vector*, *fc_axis_h=FreeCAD.Vector*, *ref_d=1*, *ref_w=1*, *ref_h=1*, *pos=FreeCAD.Vector*, *wfco=1*, *name='simple_enstop_holder'*)
Very simple endstop holder to be attached to a alu profile and that can be adjusted



(continues on next page)

(continued from previous page)

```
|   (_________) ---| 0   |:
1    _________  ---|     |:-----> fc_axis_d.
3   (_________) ---| 2   |:
4___________________|_____|:


 _________________________ .......
| :           :     : :  |:.....: endstop_nut_dist
| :...........:    :    :|:
|__:_________:_____:___:|:.....

            if endstop_nut_dist == 0
              just take the length+TOL of the nut
 _________________________
| :           :     : :  |:
| :...........:     : :  |:.....
|__:_________:_____:___:|:.....kcomp.NUT_D934_L[estp_bolt_d]+TOL
```

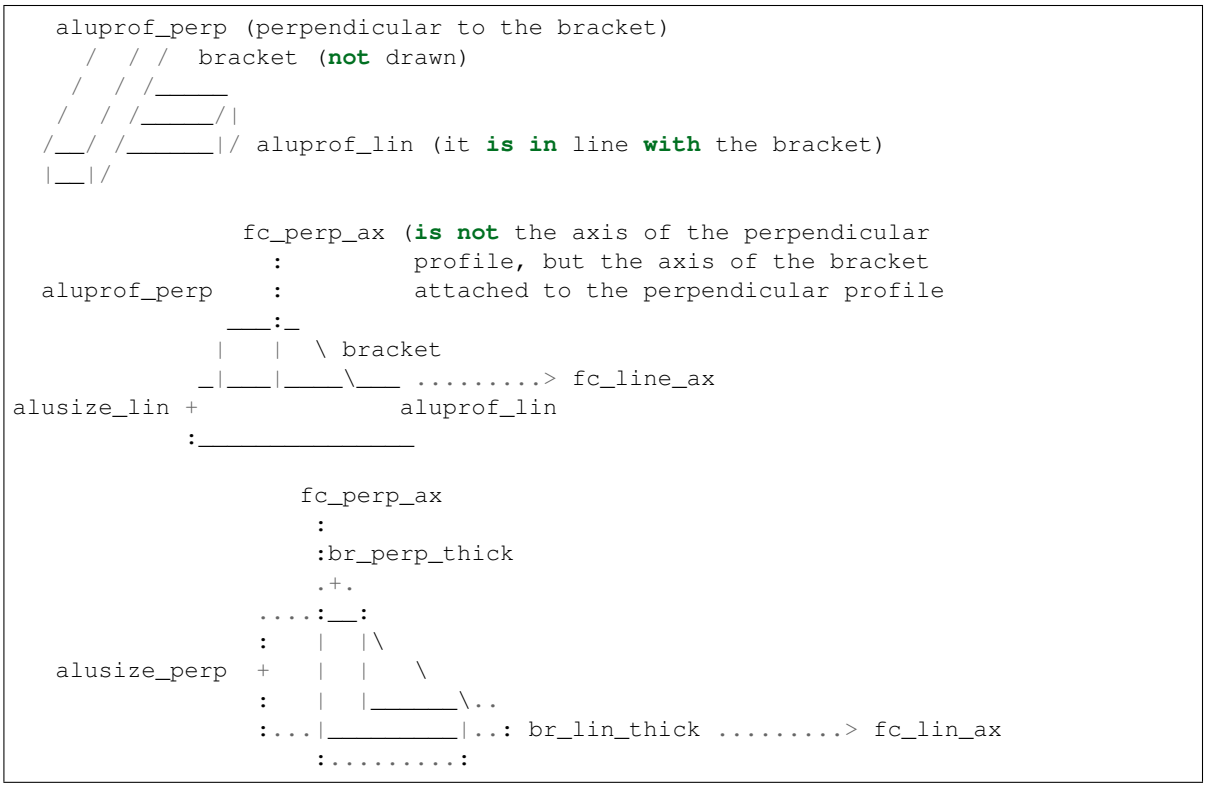
Parameters

- **d_endstop** – Dictionary of the endstop
- **rail_l** (*float*) – Length of the rail, but only the internal length, not counting the arches to make the semicircles for the bolts just from semicircle center to the other semicircle center
- **h** (*float*) – Total height, if 0 it will be the minimum height
- **base_h** (*float*) – Height for the base (for the mounting bolts)
- **holder_out** (*float*) – The endstop holder can end a little bit before to avoid it to be the endstop
- **mbolt_d** (*float*) – Diameter (metric) of the mounting bolts (for the holder not for the endstop
- **endstop_nut_dist** – Distance from the top to the endstop nut. if zero
- **min_d** (*int*) – 1: make the endstop axis_d dimension the minimum
- **fc_axis_d** (*FreeCAD Vector*) – Axis along the depth
- **fc_axis_w** (*FreeCAD Vector*) – Axis along the width
- **fc_axis_h** (*FreeCAD Vector*) – Axis along the height
- **ref_d** (*int*) – Reference (zero) of fc_axis_d
  - 1 = at the end on the side of the rails
  - 2 = at the circle center of one rail (closer to 1)
  - 3 = at the circle center of the other rail, closer to endstop
  - 4 = at the bolt of the endstop
  - 5 = at the end of the endstop (the holder ends before that)
- **ref_w** (*int*) – Reference on fc_axis_w. it is symmetrical, only the negative side
  - 1 = centered
  - 2 = at one endstop bolt the other endstop bolt will be on the direction of fc_axis_w
  - 3 = at one rail center the rail center will be on the direction of fc_axis_w

  - 4 = at the end the end will be on the direction of fc_axis_w
- **ref_h** (*int*) – Reference (zero) of fc_axis_h
  - 1: at the bottom
  - 2: on top
- **pos** (*FreeCAD.Vector*) – Object placement
- **wfco** (*int*) – 1 a freecad object will be created
- **name** (*str*) – Name of the freecad object, if created
- **rails can be countersunk to make space for the bolts** (*the*) –

class parts.**AluProfBracketPerp**(*alusize_lin*, *alusize_perp*, *br_perp_thick=3.0*, *br_lin_thick=3.0*, *bolt_lin_d=3*, *bolt_perp_d=0*, *nbolts_lin=1*, *bolts_lin_dist=0*, *bolts_lin_rail=0*, *xtr_bolt_head=3*, *xtr_bolt_head_d=0*, *reinforce=1*, *fc_perp_ax=FreeCAD.Vector*, *fc_lin_ax=FreeCAD.Vector*, *pos=FreeCAD.Vector*, *wfco=1*, *name='bracket'*)

Bracket to join 2 aluminum profiles that are perpendicular, that is, they are not on the same plane



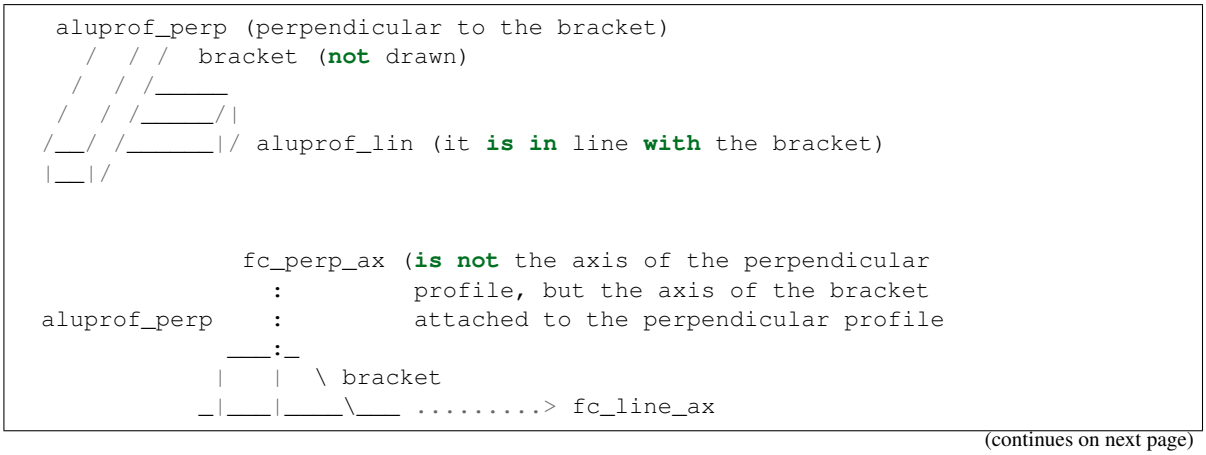
Parameters

- **alusize_lin** (*float*) – Width of the aluminum profile on the line
- **alusize_perp** (*float*) – Width of the perpendicular aluminum profile
- **br_lin_thick** (*float*) – Thickness of the line bracket
- **br_perp_thick** (*float*) – Thickness of the perpendicular bracket
- **bolt_lin_d** (*int*) – Metric of the bolt 3, 4, ... (integer)

- **bolt_perp_d** (*int*) – Metric of the bolt 3, 4, ... (integer) on the profile line if 0, the same as bolt_lin_d
- **nbolts_lin** (*int*) – Number of bolts one bolt on the fc_lin_ax, number of bolts: two bolts on the fc_lin_ax
- **bolts_lin_dist** (*float*) – If more than one bolt on fc_lin_ax, defines the distance between them. if zero, takes min distance
- **bolts_lin_rail** (*int*) – Instead of bolt holes, it will be a rail it doesn't make sense to have number of bolts with this option it will work on 2 bolts or more. If nbolts_lin == 3, it will make a rail between them. so it will be the same to have nbolts_lin = 2 and bolts_lin_dist = 20 nbolts_lin = 3 and bolts_lin_dist = 10 The rail will be 20, and it will look the same, it will be more clear to have the first option: 2 bolts
- **xtr_bolt_head** (*float*) – Extra space for the bolt head length, and making a space for it
- **xtr_bolt_head_d** (*float*) – Extra space for the bolt head diameter, and making a space for it. For the wall bolt
- **reinforce** (*int*) – 1, if it is reinforced on the sides of lin profile
- **fc_perp_ax** (*FreeCAD.Vector*) – Axis of the bracket on the perpendicular prof, see picture
- **fc_lin_ax** (*FreeCAD.Vector*) – Axis of the bracket on the aligned profile, see picture
- **pos** (*FreeCAD.Vector*) – Position of the center of the bracket on the intersection
- **wfco** (*int*) –
  - if 1: With FreeCad Object: a freecad object is created
  - if 0: only the shape
- **name** (*str*) – Name of the freecad object, if created

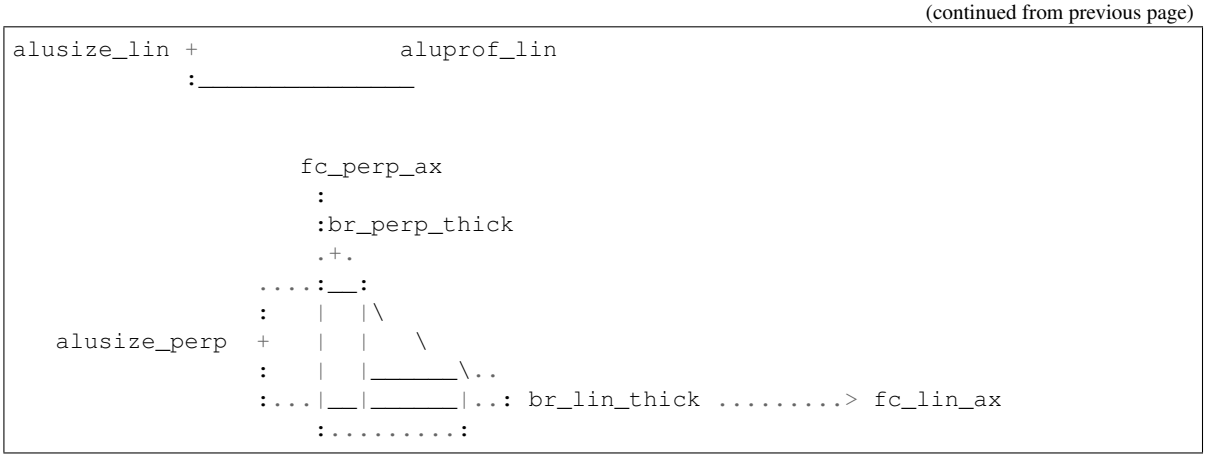
class parts.**AluProfBracketPerpFlap**(*alusize_lin*, *alusize_perp*, *br_perp_thick=3.0*, *br_lin_thick=3.0*, *bolt_lin_d=3*, *bolt_perp_d=0*, *nbolts_lin=1*, *bolts_lin_dist=0*, *bolts_lin_rail=0*, *xtr_bolt_head=1*, *sunk=1*, *flap=1*, *fc_perp_ax=FreeCAD.Vector*, *fc_lin_ax=FreeCAD.Vector*, *pos=FreeCAD.Vector*, *wfco=1*, *name='bracket_flap'*)

Bracket to join 2 aluminum profiles that are perpendicular, that is, they are not on the same plane It is wide because it has 2 ears/flaps? on the sides, to attach to the perpendicular profile



(continues on next page)

(continued from previous page)



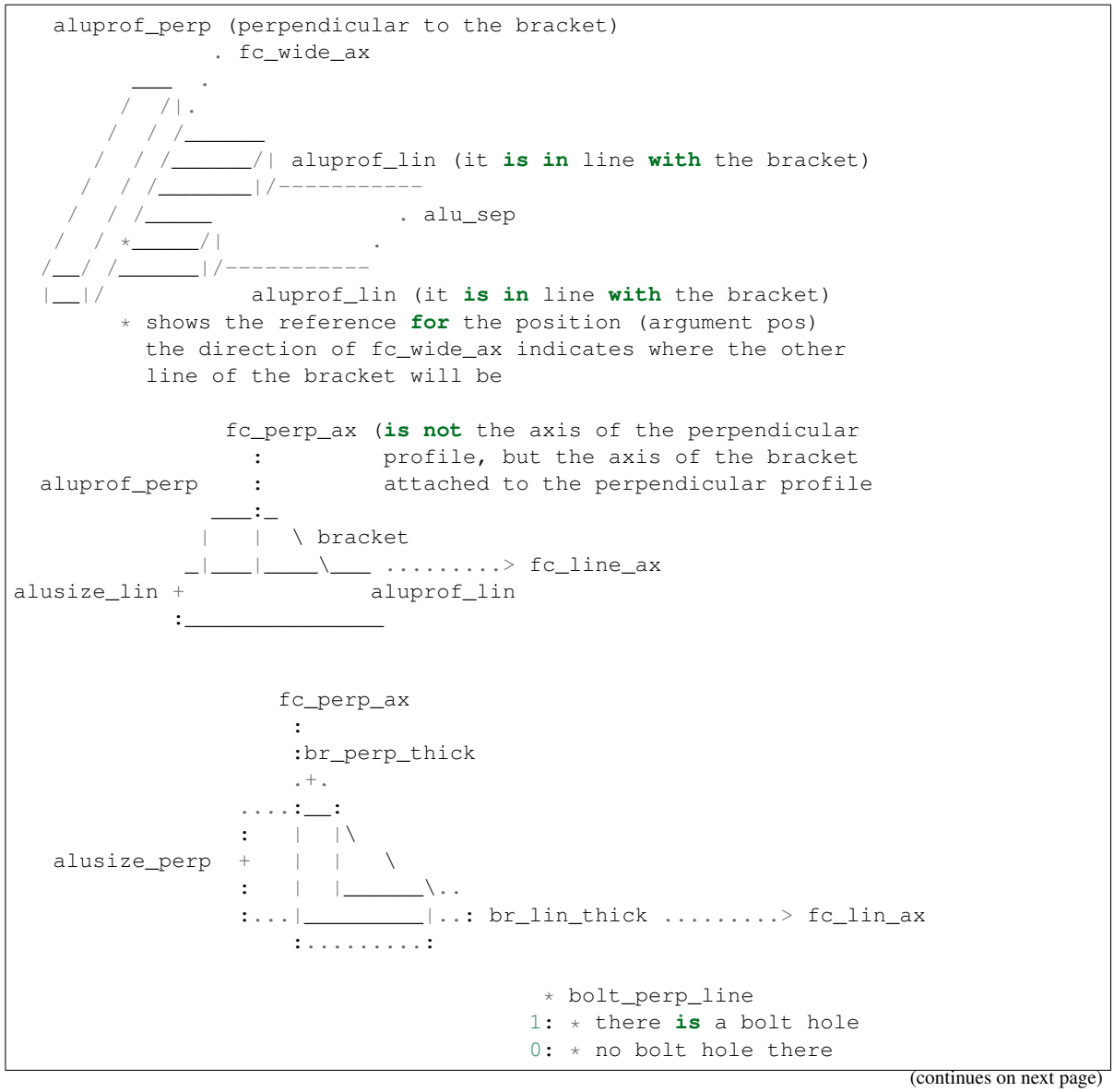
Parameters

- **alusize_lin** (*float*) – Width of the aluminum profile on the line
- **alusize_perp** (*float*) – Width of the perpendicular aluminum profile
- **br_lin_thick** (*float*) – Thickness of the line bracket
- **br_perp_thick** (*float*) – Thickness of the perpendicular bracket
- **bolt_lin_d** (*int*) – Metric of the bolt 3, 4, ... (integer)
- **bolt_perp_d** (*int*) – Metric of the bolt 3, 4, ... (integer) on the profile line if 0, the same as bolt_lin_d
- **nbolts_lin** (*int*) –
  - 1: just one bolt on the fc_lin_ax, or two bolts
  - 2: two bolts on the fc_lin_ax, or two bolts
- **bolts_lin_dist** (*float*) – If more than one bolt on fc_lin_ax, defines the distance between them. if zero, takes min distance
- **bolts_lin_rail** (*int*) – Instead of bolt holes, it will be a rail it doesn't make sense to have number of bolts with this option it will work on 2 bolts or more. If nbolts_lin == 3, it will make a rail between them. so it will be the same to have nbolts_lin = 2 and bolts_lin_dist = 20 nbolts_lin = 3 and bolts_lin_dist = 10 The rail will be 20, and it will look the same, it will be more clear to have the first option: 2 bolts
- **xtr_bolt_head** (*float*) – Extra space for the bolt head on the line to the wall (perpendicular)
- **sunk** (*int*) –
  - 0: just drilled
  - 1: if the top part is removed,
  - 2: No reinforcement at all
- **flap** (*int*) – If it has flaps or if it doesn't have flaps, it is kind of useless because it is just the middle part without bolts on the wall, but it can be used to make a union with other parts
- **fc_perp_ax** (*FreeCAD.Vector*) – Axis of the bracket on the perpendicular profile, see picture
- **fc_lin_ax** (*FreeCAD.Vector*) – Axis of the bracket on the aligned profile, see picture

- **pos** (*FreeCAD.Vector*) – Position of the center of the bracket on the intersection
- **wfco** –
  - 1: With FreeCad Object: a freecad object is created
  - 0: only the shape
- **name** (*str*) – Name of the freecad object, if created

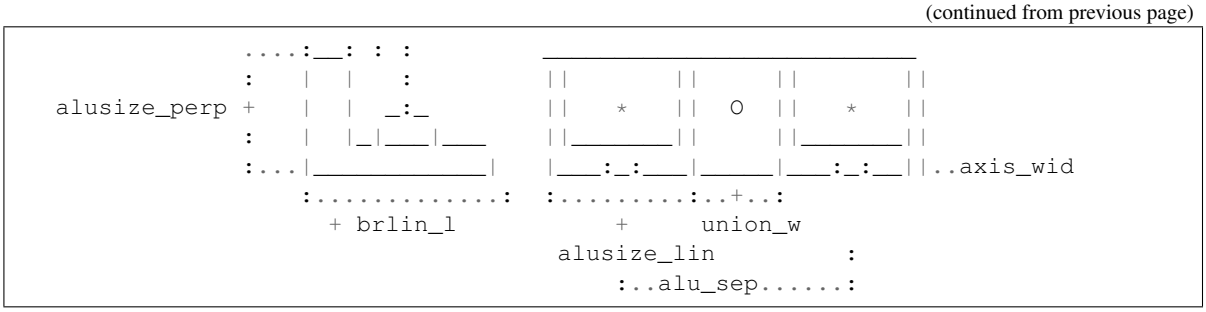
class parts.**AluProfBracketPerpTwin**(*alusize_lin*, *alusize_perp*, *alu_sep*, *br_perp_thick=3.0*, *br_lin_thick=3.0*, *bolt_lin_d=3*, *bolt_perp_d=0*, *nbolts_lin=1*, *bolts_lin_dist=1*, *bolts_lin_rail=0*, *bolt_perp_line=0*, *xtr_bolt_head=3*, *sunk=0*, *fc_perp_ax=FreeCAD.Vector*, *fc_lin_ax=FreeCAD.Vector*, *fc_wide_ax=FreeCAD.Vector*, *pos=FreeCAD.Vector*, *wfco=1*, *name='bracket_twin'*)

Bracket to join 3 aluminum profiles that are perpendicular, that is, they are not on the same plane to the perpendicular profile



(continues on next page)

(continued from previous page)



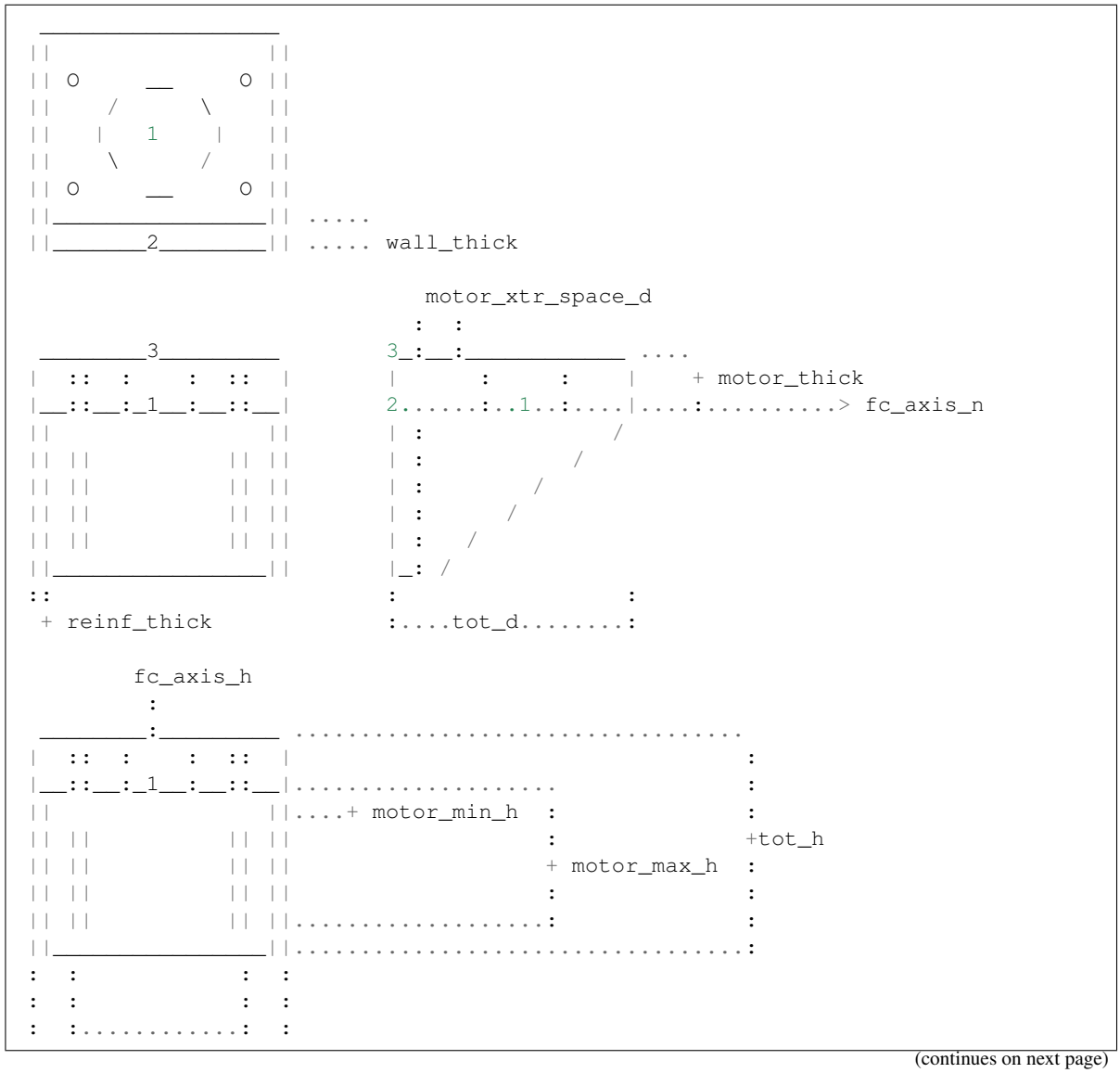
Parameters

- **alusize_lin** (*float*) – Width of the aluminum profile on the line
- **alusize_perp** (*float*) – Width of the perpendicular aluminum profile
- **alu_sep** (*float*) – Separation of the 2 parallel profiles, from their centers
- **br_lin_thick** (*float*) – Thickness of the line bracket
- **br_perp_thick** (*float*) – Thickness of the perpendicular bracket
- **bolt_lin_d** (*int*) – Metric of the bolt 3, 4, ... (integer)
- **bolt_perp_d** (*int*) – Metric of the bolt 3, 4, ... (integer) on the profile line if 0, the same as bolt_lin_d
- **nbolts_lin** (*int*) –
  - 1: just one bolt on the fc_lin_ax, or two bolts
  - 2: two bolts on the fc_lin_ax, or two bolts
- **bolts_lin_dist** (*float*) – If more than one bolt on fc_lin_ax, defines the distance between them. if zero, takes min distance
- **bolts_lin_rail** (*int*) – Instead of bolt holes, it will be a rail it doesn't make sense to have number of bolts with this option it will work on 2 bolts or more. If nbolts_lin == 3, it will make a rail between them. so it will be the same to have nbolts_lin = 2 and bolts_lin_dist = 20 nbolts_lin = 3 and bolts_lin_dist = 10 The rail will be 20, and it will look the same, it will be more clear to have the first option: 2 bolts
- **bolt_perp_line** (*int*) –
  - 1: if it has a bolt on the wall (perp) but in line with the line aluminum profiles
  - 0: no bolt
- **xtr_bolt_head** (*float*) – Extra space for the bolt head, and making a space for it only makes sense if bolt_perp_line == 1
- **sunk** (*int*) –
  - 0: No sunk, just drill holes: bolt_perp_line should be 0
  - 1: sunk, but with reinforcement if possible
  - 2: no reinforcement
- **fc_perp_ax** (*FreeCAD.Vector*) – Axis of the bracket on the perpendicular prof, see picture
- **fc_lin_ax** (*FreeCAD.Vector*) – Axis of the bracket on the aligned profile, see picture

- **fc_wide_ax** (*FreeCAD.Vector*) – Axis of the bracket on wide direction, see picture its direction shows where the other aligned profile is
- **pos** (*FreeCAD.Vector*) – Position of the center of the bracket on the intersection
- **wfco** (*int*) –
  - 1: With FreeCad Object: a freecad object is created
  - 0: only the shape
- **name** (*str*) – Name of the freecad object, if created

class parts.**NemaMotorHolder**(*nema_size=17*, *wall_thick=4.0*, *motor_thick=4.0*, *reinf_thick=4.0*, *motor_min_h=10.0*, *motor_max_h=20.0*, *rail=1*, *motor_xtr_space=2.0*, *motor_xtr_space_d=- 1*, *bolt_wall_d=4.0*, *bolt_wall_sep=30.0*, *chmf_r=1.0*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_n=FreeCAD.Vector*, *ref_axis=1*, *pos=FreeCAD.Vector*, *wfco=1*, *name='nema_holder'*)

Creates a holder for a Nema motor



(continues on next page)

(continued from previous page)

```
 :   bolt_wall_sep  :
 :                  :
 :                  :
 :.....tot_w........:
                   ::
                    motor_xtr_space

1: ref_axis = 1 & ref_bolt = 0
2: ref_axis = 0 & ref_bolt = 0
--3: ref_axis = 0 & ref_bolt = 0
```

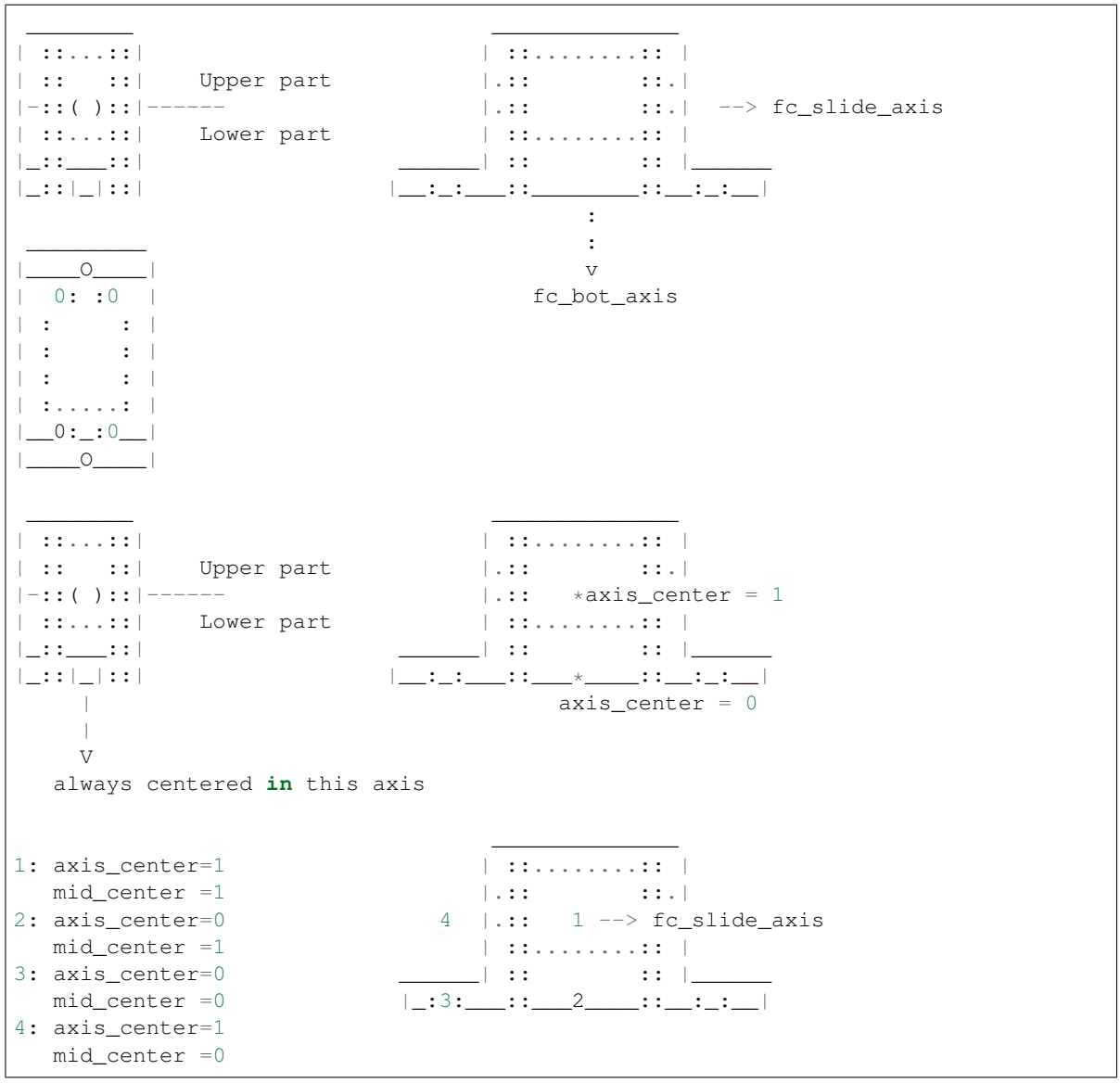
Parameters

- **nema_size** (*int*) – Size of the motor (NEMA)
- **wall_thick** (*float*) – Thickness of the side where the holder will be screwed to
- **motor_thick** (*float*) – Thickness of the top side where the motor will be screwed to
- **reinf_thick** (*float*) – Thickness of the reinforcement walls
- **motor_min_h** (*float*) – Distance of from the inner top side to the top hole of the bolts to attach the holder (see drawing)
- **motor_max_h** (*float*) – Distance of from the inner top side to the bottom hole of the bolts to attach the holder
- **rail** (*int*) –
  - 2: the rail goes all the way to the end, not closed
  - 1: the holes for the bolts are not holes, there are 2 rails, from motor_min_h to motor_max_h
  - 0: just 2 pairs of holes. One pair at defined by motor_min_h and the other defined by motor_max_h
- **motor_xtr_space** (*float*) – Extra separation between the motor and the sides
- **motor_xtr_space_d** (*float*) – Extra separation between the motor and the wall side (where the bolts) it didn't exist before, so for compatibility
  - -1: same has motor_xtr_space (compatibility), considering bolt head length
  - 0: no separation
  - >0: exact separation
- **bolt_wall_d** (*int/float*) – Metric of the bolts to attach the holder
- **bolt_wall_sep** (*float*) – Separation between the 2 bolt holes (or rails). Optional.
- **chmf_r** (*float*) – Radius of the chamfer, whenever chamfer is done
- **fc_axis_h** (*FreeCAD Vector*) – Axis along the axis of the motor
- **fc_axis_n** (*FreeCAD Vector*) – Axis normal to surface where the holder will be attached to
- **ref_axis** (*int*) –
  - 1: the zero of the vertical axis (axis_h) is on the motor axis
  - 0: the zero of the vertical axis (axis_h) is at the wall

- **pos** (*FreeCAD.Vector*) – Position of the holder (considering ref_axis)
- **wfco** (*int*) –
  - 1: creates a FreeCAD object
  - 0: only creates a shape
- **name** (*string*) – Name of the FreeCAD object

class parts.**ThinLinBearHouse1rail**(*d_lbear*, *fc_slide_axis=FreeCAD.Vector*, *fc_bot_axis=FreeCAD.Vector*, *axis_center=1*, *mid_center=1*, *pos=FreeCAD.Vector*, *name='thinlinbearhouse1rail'*)

Makes a housing for a linear bearing, but it is very thin and intended to be attached to one rail, instead of 2 it has to parts, the lower and the upper part



Parameters

- **d_lbear** (*dictionary*) – Dictionary with the dimensions of the linear bearing

- **fc_slide_axis** (*FreeCAD.Vector*) – Direction of the slide
- **fc_bot_axis** (*FreeCAD.Vector*) – Direction of the bottom
- **axis_center** (*int*) – See picture, indicates the reference point
- **mid_center** (*int*) – See picture, indicates the reference point
- **pos** (*FreeCAD.Vector*) – Position of the reference point,

n1_slide_axis

Type FreeCAD.Vector

n1_bot_axis

Type FreeCAD.Vector

n1_perp

Type FreeCAD.Vector

axis_h

Type float

boltcen_axis_dist

Type float

boltcen_perp_dist

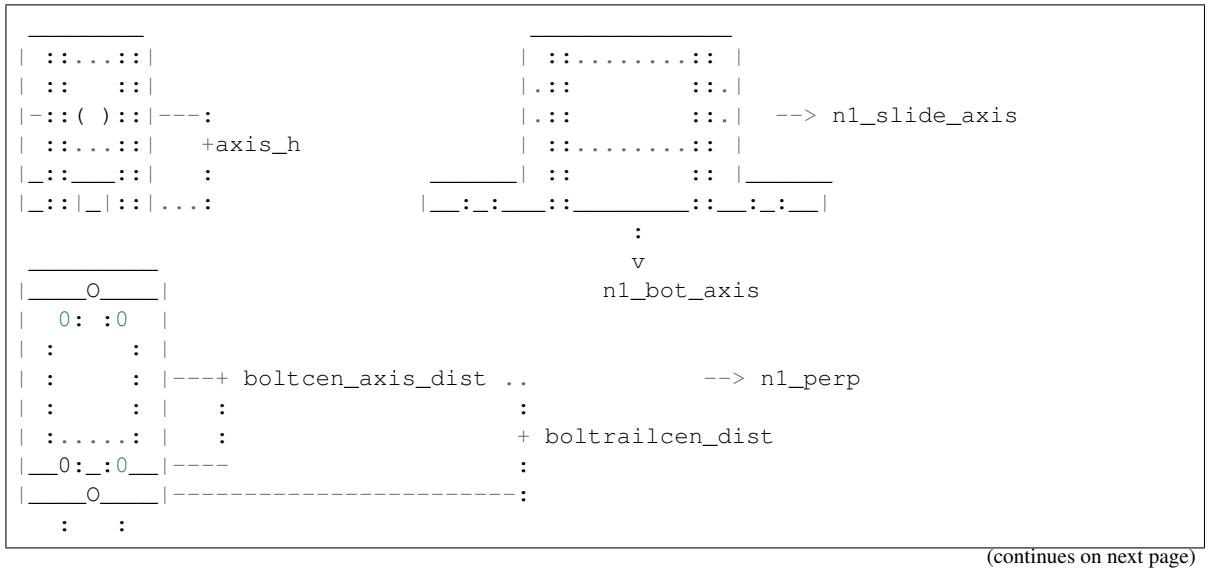
Type float

Dimensions:

- tot_h, tot_w, tot_l
- housing_l, base_h

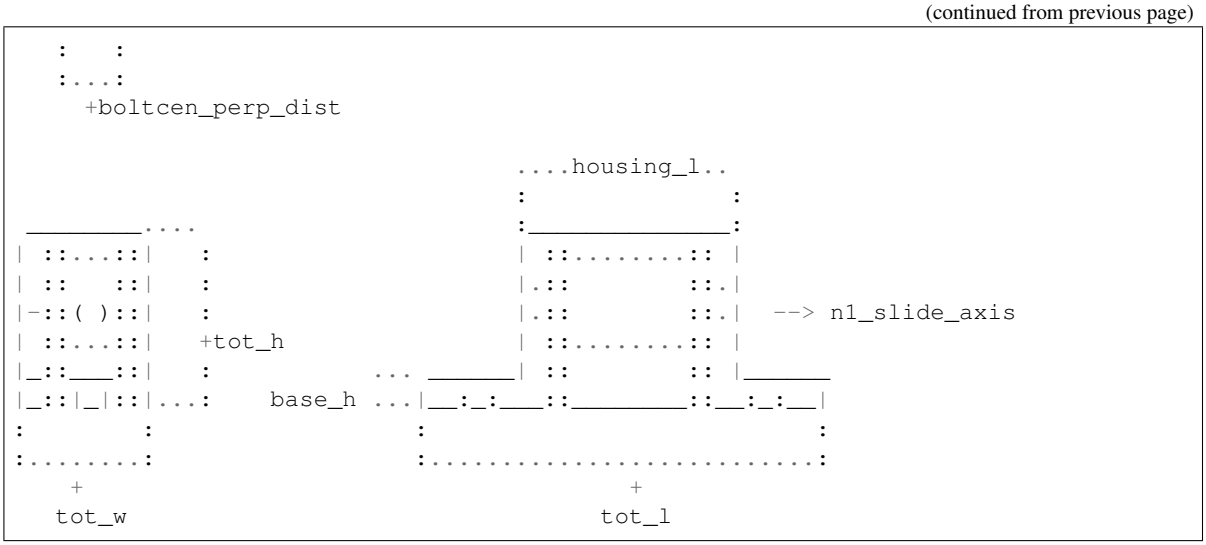
FreeCAD objects:

- fco_top : Top part of the linear bearing housing
- fco_bot : Bottom part of the linear bearing housing



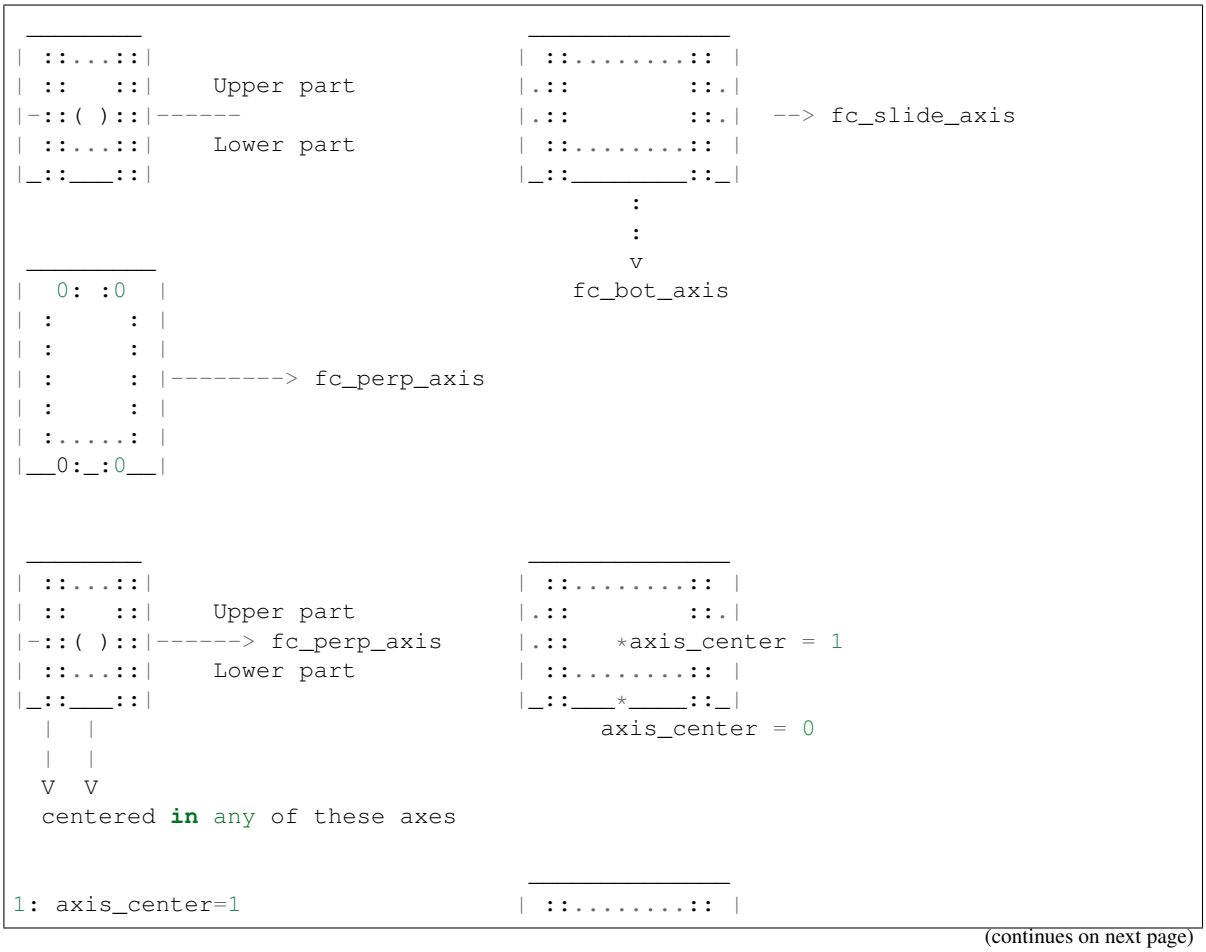
(continues on next page)

(continued from previous page)



class parts.**ThinLinBearHouse**(*d_lbear*, *fc_slide_axis=FreeCAD.Vector*, *fc_bot_axis=FreeCAD.Vector*, *fc_perp_axis=FreeCAD.Vector*, *axis_h=0*, *bolts_side=1*, *axis_center=1*, *mid_center=1*, *bolt_center=0*, *pos=FreeCAD.Vector*, *name='thinlinbearhouse'*)

Makes a housing for a linear bearing, but it is very thin and intended to be attached to 2 rail it has to parts, the lower and the upper part



(continues on next page)

(continued from previous page)

```
   mid_center =1                    |.::         ::.|
2: axis_center=0                    |.:4   1 --> fc_slide_axis
   mid_center =1                    | ::........:: |
3: axis_center=0                    | ::         :: |
   mid_center =0                    |_:3___2____::_|
4: axis_center=1

And 8 more possibilities:
5: bolt_center = 1
6: bolt_center = 0


 _________
|  5:6:   |
| :     : |
| :     : |
| :     : |--------> fc_perp_axis
| :     : |
| :.....: |
|__0:_:0__|
  mid_center =0
```

Parameters

- **d_lbear** (*dictionary*) – Dictionary with the dimensions of the linear bearing
- **fc_slide_axis** (*FreeCAD.Vector*) – Direction of the slide
- **fc_bot_axis** (*FreeCAD.Vector*) – Direction of the bottom
- **fc_perp_axis** (*FreeCAD.Vector*) – Direction of the other perpendicular direction. Not useful unless bolt_center == 1 if = V0 it doesn't matter
- **axis_h** (*int*) – Distance from the bottom to the rod axis
  - 0: take the minimum distance
  - X: (any value) take that value, if it is smaller than the minimum it will raise an error and would not take that value
- **bolts_side** (*int*) – See picture, indicates the side where is bolt
- **axis_center** (*int*) – See picture, indicates the reference point
- **mid_center** (*int*) – See picture, indicates the reference point
- **bolt_center** (*int*) – See picture, indicates the reference point, if it is on the bolt or on the axis
- **pos** (*FreeCAD.Vector*) – Position of the reference point,

n1_slide_axis

Type FreeCAD.Vector

n1_bot_axis

Type FreeCAD.Vector

n1_perp

Type FreeCAD.Vector

axis_h

Type float

boltcen_axis_dist

Type float

boltcen_perp_dist

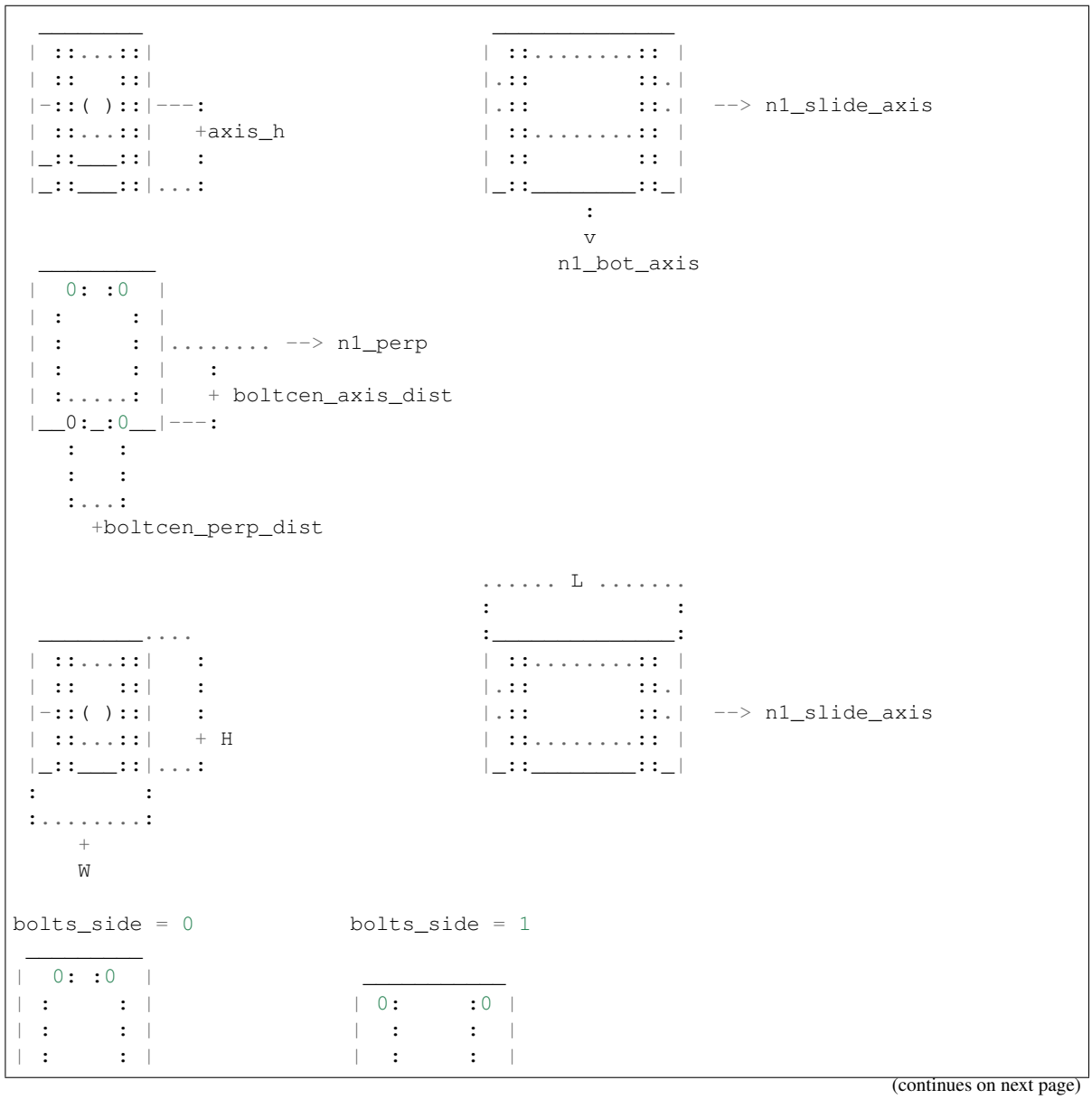
Type float

Dimensions:

- H, W, L

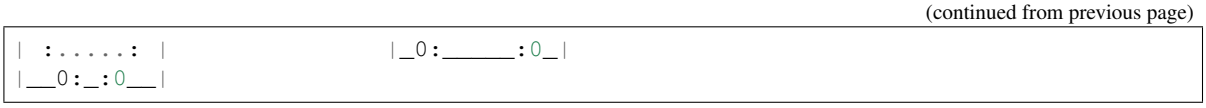
FreeCAD objects:

- fco_top : Top part of the linear bearing housing
- fco_bot : Bottom part of the linear bearing housing



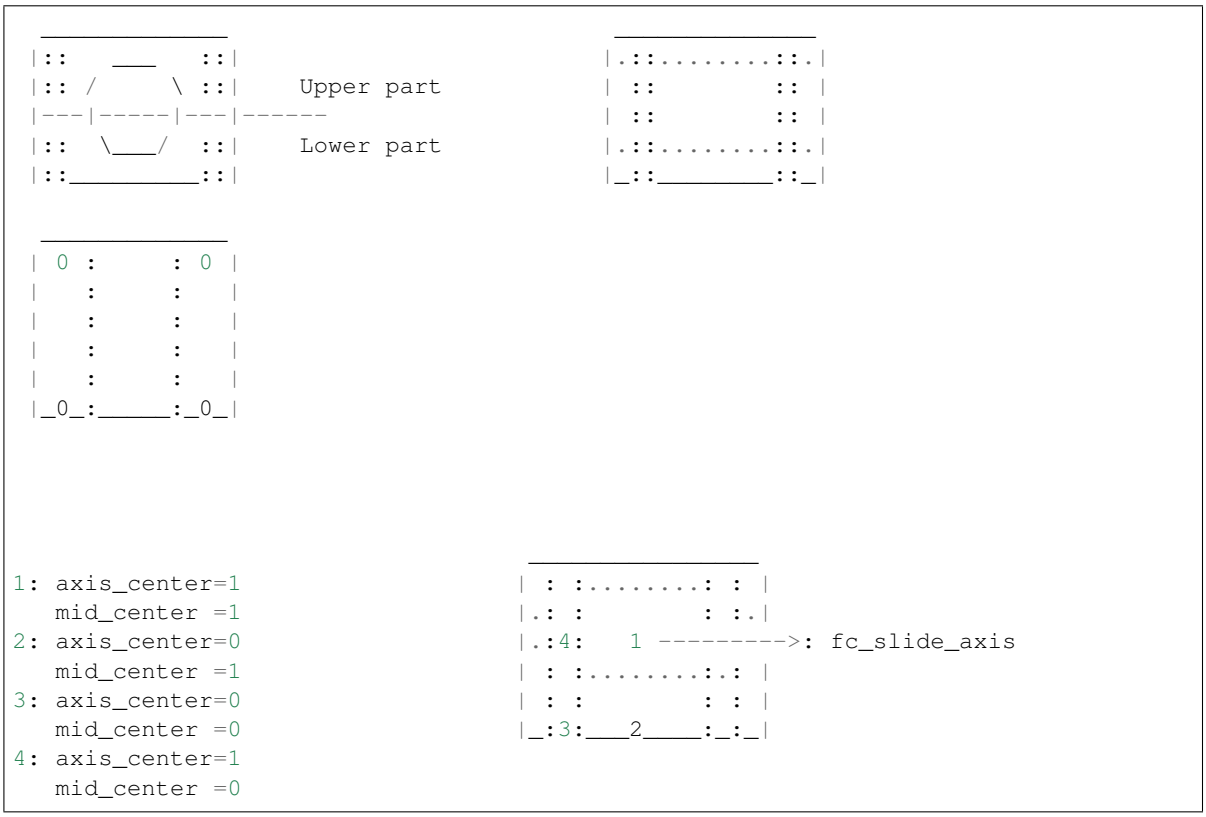
(continues on next page)

(continued from previous page)



class parts.**LinBearHouse**(*d_lbearhousing*, *fc_slide_axis=FreeCAD.Vector*, *fc_bot_axis=FreeCAD.Vector*, *axis_center=1*, *mid_center=1*, *pos=FreeCAD.Vector*, *name='linbearhouse'*)

Makes a housing for a linear bearing takes the dimensions from a dictionary, like the one defined in kcomp.py it has to parts, the lower and the upper part



class parts.**ThinLinBearHouseAsim**(*d_lbear*, *fc_fro_ax=FreeCAD.Vector*, *fc_bot_ax=FreeCAD.Vector*, *fc_sid_ax=FreeCAD.Vector*, *axis_h=0*, *bolts_side=1*, *refcen_hei=1*, *refcen_dep=1*, *refcen_wid=1*, *bolt2cen_wid_n=0*, *bolt2cen_wid_p=0*, *pos=FreeCAD.Vector*, *name='thinlinbearhouse_asim'*)

There are

3 axis:	3 planes (normal to axis)	3 distances to plane
fc_fro_ax	fro: front	D: dep: depth
fc_bot_ax	hor: horizontal)	H: hei: height
fc_sid_ax	lat: lateral (medial)	W: wid: width

The planes are on the center of the slidding rod (height and width), and on the middle of the piece (width)

The 3 axis are perpendicular, but the cross product of 2 vectors may result on the other vector or its negative.

fc_fro_ax points to the front of the figure, but it is symmetrical so it can point to the back fc_bot_ax points to the bottom of the figure (not symmetrical) fc_sid_ax points to the side of the figure. Not symmetrical if bolt2cen_wid_n or bolt2cen_wid_p are not zero

Makes a housing for a linear bearing, but it is very thin and intended to be attached to 2 rail it has to parts, the lower and the upper part

```
 ________                              _______________
| ::...::|                           | ::.........:: |
| ::   ::|    Upper part             |.::         ::.|
|-::( )::|------ Horizontal plane    |.::         ::.|  --> fc_fro_ax
| ::...::|    Lower part             | ::.........:: |
|_::___::|                           |_::_________::_|
                                             :
                                             :
 _________                                   v
|  0: :0  |                              fc_bot_axis
| :     : |
| :     : |
| :     : |--------> fc_sid_ax
| :     : |
| :.....: |
|__0:_:0__|
 ________                              _______________
| ::...::|                           | ::.........:: |
| ::   ::|    Upper part             |.::         ::.|
|-::( )::|------> fc_sid_ax          |.::   *refcen_hei = 1
| ::...::|    Lower part             | ::.........:: |
|_::___::|                           |_::___*_____::_|
  |   |                                    refcen_hei = 0
  |   |
  V   V
  centered in any of these axes
refcen_hei: reference centered on the height
          =1: the horizontal plane (height) is on the axis of the rod
          =0: the horizontal plane is at the bottom
refcen_dep: reference centered on the depth
          =1: the frontal plane (depth) is on the middle of the piece
          =0: the frontal plane is at the bolts
refcen_wid=1: reference centered on the width
              the lateral plane (width) is on the medial axis, dividing
              the piece on the right and left
          =0: the lateral plane is at the bolts


                                       _______________
1: refcen_hei=1                       | ::.........:: |
   fro_center =1                      |.::         ::.|
2: refcen_hei=0                       |.:4    1 --> fc_fro_ax
   fro_center =1                      | ::.........:: |
3: refcen_hei=0                       | ::         :: |
   fro_center =0                      |_:3___2_____::_|
4: refcen_hei=1

And 8 more possibilities:
5: refcen_wid = 0
6: refcen_wid = 1


 _________
|  5:6:   |
| :     : |
| :     : |
| :     : |--------> fc_sid_ax
```

(continues on next page)

(continued from previous page)

```
|  :      : |
|  :.....: |
|__0:_:0__|
```

Parameters

- **d_lbear** (*dictionary*) – Dictionary with the dimensions of the linear bearing
- **fc_fro_ax** (*FreeCAD.Vector*) – Direction of the slide
- **fc_bot_ax** (*FreeCAD.Vector*) – Direction of the bottom
- **fc_sid_ax** (*FreeCAD.Vector*) – Direction of the other perpendicular direction. Not useful unless refcen_wid == 0 if = V0 it doesn't matter
- **axis_h** (*float*) – Distance from the bottom to the rod axis
  - 0: take the minimum distance
  - X: (any value) take that value, if it is smaller than the minimum it will raise an error and would not take that value
- **refcen_hei** (*int*) – See picture, indicates the reference point
- **refcen_dep** (*int*) – See picture, indicates the reference point
- **refcen_wid** (*int*) – See picture, indicates the reference point, if it is on the bolt or on the axis
- **pos** (*FreeCAD.Vector*) – Position of the reference point,

nfro_ax

Type FreeCAD.Vector normalized fc_fro_ax

nbot_ax

Type FreeCAD.Vector normalized fc_bot_ax

nsid_ax

Type FreeCAD.Vector

axis_h

Type float

bolt2cen_dep

Type float

bolt2cen_wid_n

Type float

bolt2cen_wid_p

Type float

bolt2bolt_wid

Type bolt2cen_wid_n + bolt2cen_wid_p

Dimensions:

- D : float

housing_d

- W : float

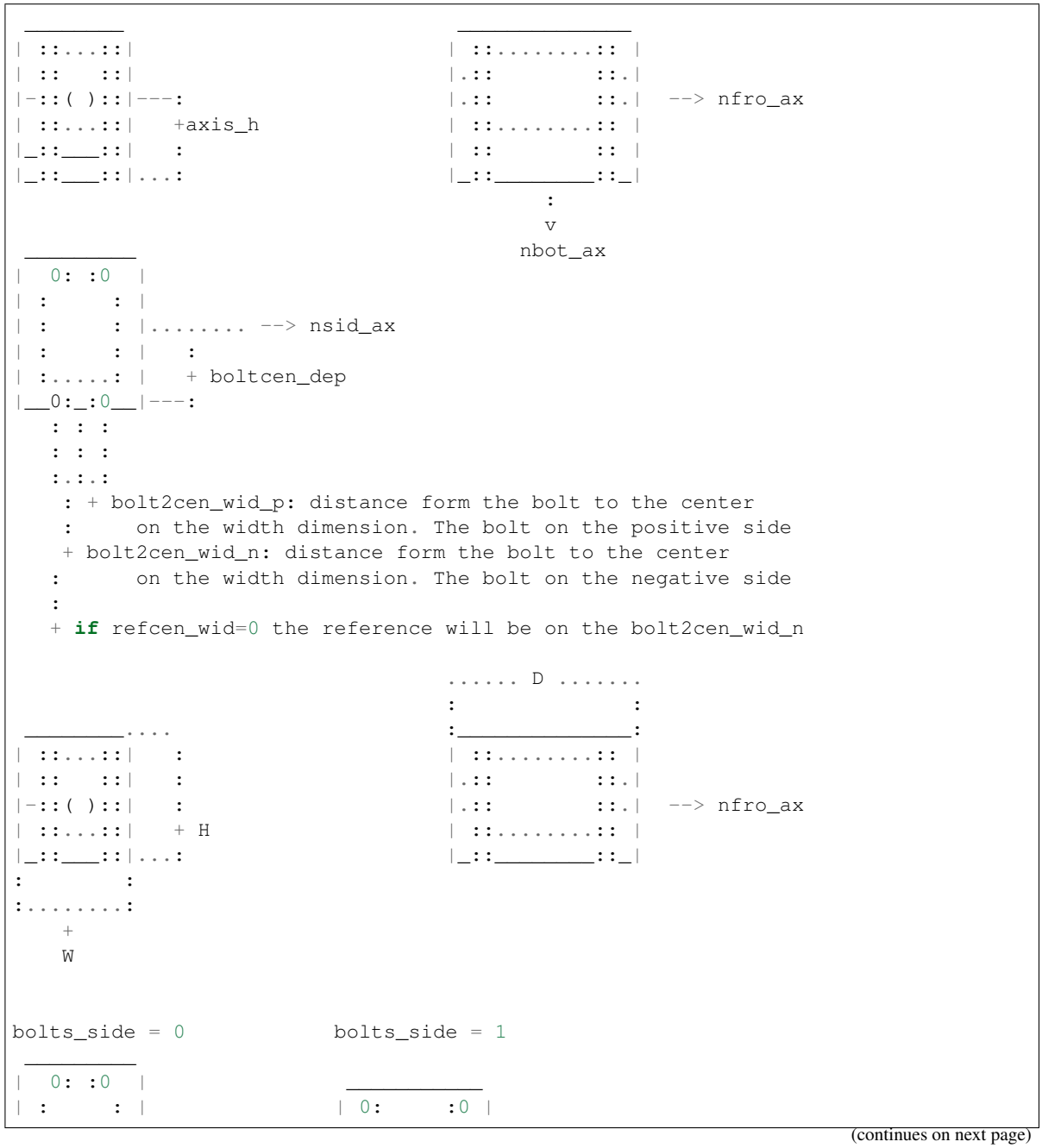
  housing_w

- H : float

  housing_h

FreeCAD objects:

- fco_top : Top part of the linear bearing housing
- fco_bot : Bottom part of the linear bearing housing



(continues on next page)

(continued from previous page)

```
| :      : |                 |  :      :  |
| :      : |                 |  :      :  |
| :......: |                 |_0:______:0_|
|__0:_:0__|
```

class filter_holder_clss.**PartFilterHolder**(*filter_l=60.0*, *filter_w=25.0*, *filter_t=2.5*, *base_h=6.0*, *hold_d=12.0*, *filt_supp_in=2.0*, *filt_rim=3.0*, *filt_cen_d=0*, *fillet_r=1.0*, *boltcol1_dist=10.0*, *boltcol2_dist=12.5*, *boltcol3_dist=25*, *boltrow1_h=0*, *boltrow1_2_dist=12.5*, *boltrow1_3_dist=20.0*, *boltrow1_4_dist=25.0*, *bolt_cen_mtr=4*, *bolt_linguide_mtr=3*, *beltclamp_t=3.0*, *beltclamp_l=12.0*, *beltclamp_h=8.0*, *clamp_post_dist=4.0*, *sm_beltpost_r=1.0*, *tol=0.4*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *axis_h=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos_h=0*, *pos=FreeCAD.Vector*, *model_type=0*, *name=''*)

Integration of a ShpFilterHolder object into a PartFilterHolder object, so it is a FreeCAD object that can be visualized in FreeCAD

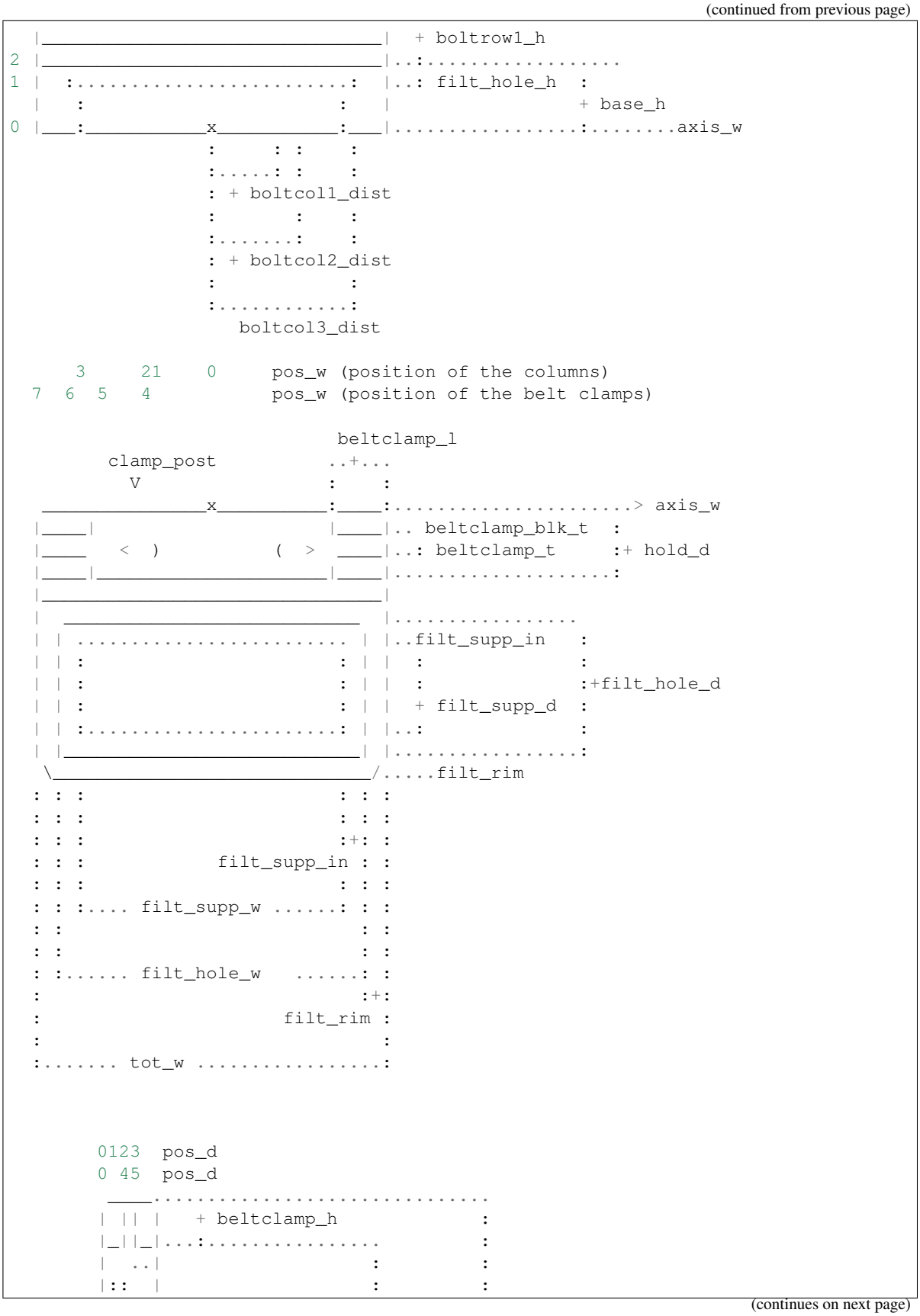
class filter_holder_clss.**ShpFilterHolder**(*filter_l=60.0*, *filter_w=25.0*, *filter_t=2.5*, *base_h=6.0*, *hold_d=12.0*, *filt_supp_in=2.0*, *filt_rim=3.0*, *filt_cen_d=0*, *fillet_r=1.0*, *boltcol1_dist=10.0*, *boltcol2_dist=12.5*, *boltcol3_dist=25*, *boltrow1_h=0*, *boltrow1_2_dist=12.5*, *boltrow1_3_dist=20.0*, *boltrow1_4_dist=25.0*, *bolt_cen_mtr=4*, *bolt_linguide_mtr=3*, *beltclamp_t=3.0*, *beltclamp_l=12.0*, *beltclamp_h=8.0*, *clamp_post_dist=4.0*, *sm_beltpost_r=1.0*, *tol=0.4*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *axis_h=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos_h=0*, *pos=FreeCAD.Vector*)

Creates the filter holder shape

```
                        beltpost_l = 3*lr_beltpost_r + sm_beltpost_r
pos_h          axis_h   :   :
|                  :    :    clamp_post_dist
v pos_w            :    :   ....
9 7___6  5___4     :    :___:  :___
8 |   |  |   |     :    |   |  |   |
7 |...|__|___|_____:____|___|__|...|...
  |         _           _          |   2 * bolt_linguide_head_r_tol
6 |        |o|         |o|         |------------------------
5 |        |o|         |o|         |--------------------  +boltrow1_4_dist
  |                                |                    :  :
  |                                |                   +boltrow1_3_dist
4 |       (O)             (O)      |--:                :  :
  |                                |  +boltrow1_2_dist :  :
  |                                |  :                :  :
3 | (O)    (o)   (O)   (o)    (O)  |--:----------------:--:
```

(continues on next page)

(continued from previous page)



(continues on next page)

(continued from previous page)

```
        |::  |                    :          :
        |  ..|                    :          :
        |  ..|                    :          :+ tot_h
        |::  |                    :          :
        |  ..|                    :+hold_h   :
        |  ..|                    :          :
        |::  |                    :          :
        |  ..|                    :          :
        |     \__________________  :          :
        |        :...........:   | :          :
        |         :          :   | :          :
        x_________:__________:___|.:.........:...>axis_d
        :               :         :
        :...............:         :
        : filt_cen_d              :
        :                         :
        :...... tot_d .........:

pos_d:  0    6  78    9    1011 12



 pos_o (origin) is at pos_d=0, pos_w=0, pos_h=0, It marked with x
```

Parameters

- **filter_l** (*float*) – Length of the filter (it will be along axis_w). Larger dimension
- **filter_w** (*float*) – Width of the filter (it will be along axis_d). Shorter dimension
- **filter_t** (*float*) – Thickness/height of the filter (it will be along axis_h). Very short
- **base_h** (*float*) – Height of the base
- **hold_d** (*float*) – Depth of the holder (just the part that holds)
- **filt_supp_in** (*float*) – How much the filter support goes inside from the filter hole
- **filt_cen_d** (*float*) – Distance from the filter center to the beginning of the filter holder along axis_d
  - 0: it will take the minimum distance or if it is smaller than the minimum distance
- **filt_rim** (*float*) – Distance from the filter to the edge of the base
- **fillet_r** (*float*) – Radius of the fillets
- **boltcol1_dist** (*float*) – Distance to the center along axis_w of the first column of bolts
- **boltcol2_dist** (*float*) – Distance to the center along axis_w of the 2nd column of bolts
- **boltcol3_dist** (*float*) – Distance to the center along axis_w of the 3rd column of bolts This column could be closer to the center than the 2nd, if distance is smaller
- **boltrow1_h** (*float*) – Distance from the top of the filter base to the first row of bolts
  - 0: the distance will be the largest head diameter in the first row in any case, it has to be larger than this

- **boltrow1_2_dist** (*float*) – Distance from the first row of bolts to the second
- **boltrow1_3_dist** (*float*) – Distance from the first row of bolts to the third
- **boltrow1_4_dist** (*float*) – Distance from the first row of bolts to the 4th
- **bolt_cen_mtr** (*integer (could be float: 2.5)*) – Diameter (metric) of the bolts at the center or at columns other than 2nd column
- **bolt_linguide_mtr** (*integer (could be float: 2.5)*) – Diameter (metric) of the bolts at the 2nd column, to attach to a linear guide
- **beltclamp_t** (*float*) – Thickness of the hole for the belt. Inside de belt clamp blocks (along axis_d)
- **beltclamp_l** (*float*) – Length of the belt clamp (along axis_w)
- **beltclamp_h** (*float*) – Height of the belt clamp: belt width + 2 (along axis_h)
- **clamp_post_dist** (*float*) – Distance from the belt clamp to the belt clamp post
- **sm_beltpost_r** (*float*) – Small radius of the belt post
- **tol** (*float*) – Tolerances to print
- **axis_d** (*FreeCAD.Vector*) – Length/depth vector of coordinate system
- **axis_w** (*FreeCAD.Vector*) – Width vector of coordinate system if V0: it will be calculated using the cross product: axis_d x axis_h
- **axis_h** (*FreeCAD.Vector*) – Height vector of coordinate system
- **pos_d** (*int*) – Location of pos along the axis_d (0,1,2,3,4,5), see drawing
  - 0: at the back of the holder
  - 1: at the end of the first clamp block
  - 2: at the center of the holder
  - 3: at the beginning of the second clamp block
  - 4: at the beginning of the bolt head hole for the central bolt
  - 5: at the beginning of the bolt head hole for the linguide bolts
  - 6: at the front side of the holder
  - 7: at the beginning of the hole for the porta
  - 8: at the inner side of the porta thruhole
  - 9: at the center of the porta
  - 10: at the outer side of the porta thruhole
  - 11: at the end of the porta
  - 12: at the end of the piece
- **pos_w** (*int*) – Location of pos along the axis_w (0-7) symmetrical
  - 0: at the center of symmetry
  - 1: at the first bolt column
  - 2: at the second bolt column
  - 3: at the third bolt column

  - 4: at the inner side of the clamp post (larger circle)
  - 5: at the outer side of the clamp post (smaller circle)
  - 6: at the inner side of the clamp rails
  - 7: at the end of the piece
- **pos_h** (*int*) – Location of pos along the axis_h (0-8)
  - 0: at the bottom (base)
  - 1: at the base for the porta
  - 2: at the top of the base
  - 3: first row of bolts
  - 4: second row of bolts
  - 5: third row of bolts
  - 6: 4th row of bolts
  - 7: at the base of the belt clamp
  - 8: at the middle of the belt clamp
  - 9: at the top of the piece
- **pos** (*FreeCAD.Vector*) – Position of the cylinder, taking into account where the center is

Note: All the parameters and attributes of parent class SinglePart

Dimensional attributes

filt_hole_d
depth of the hole for the filter (for filter_w)

Type float

filt_hole_w
width of the hole for the filter (for filter_l)

Type float

filt_hole_h
height of the hole for the filter (for filter_t)

Type float

beltclamp_blk_t
thickness (along axis_d) of each of the belt clamp blocks

Type float

beltpost_l
length of the belt post (that has a shap of 2 circles and the tangent

Type float

lr_beltpost_r
radius of the larger belt post (it has a belt shape)

Type float

clamp_lrbeltpostcen_dist
distance from the center of the larger belt post cylinder to the clamp post

Type float

prnt_ax
Best axis to print (normal direction, pointing upwards)

Type FreeCAD.Vector

d0_cen

Type int

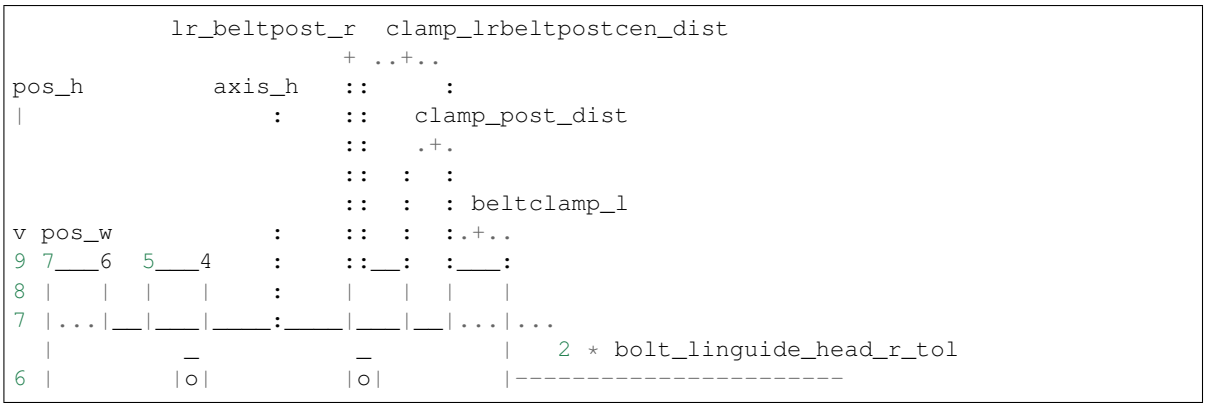
w0_cen

Type int

h0_cen
indicates if pos_h = 0 (pos_d, pos_w) is at the center along axis_h, axis_d, axis_w, or if it is at the end.

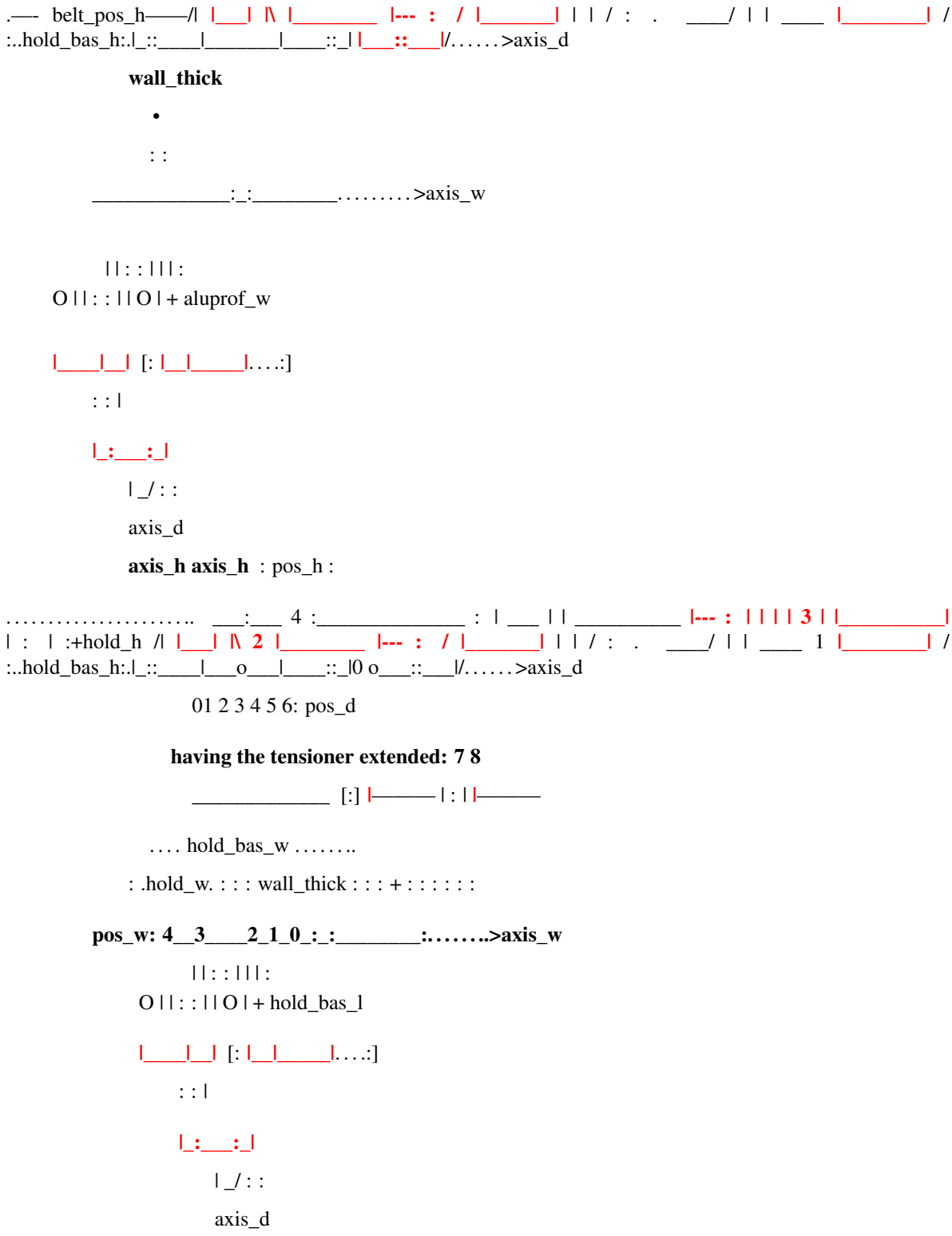
- 1 : at the center (symmetrical, or almost symmetrical)
- 0 : at the end

Type int



class tensioner_clss.**TensionerSet**(*aluprof_w=20.0*, *belt_pos_h=20.0*, *hold_bas_h=0*, *hold_hole_2sides=0*, *boltidler_mtr=3*, *bolttens_mtr=3*, *boltaluprof_mtr=3*, *tens_stroke=20.0*, *wall_thick=3.0*, *in_fillet=2.0*, *pulley_stroke_dist=0*, *nut_holder_thick=4.0*, *opt_tens_chmf=1*, *min_width=0*, *tol=0.4*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *axis_h=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos_h=0*, *pos=FreeCAD.Vector*, *group=0*, *name=''*)
Set composed of the idler pulley and the tensioner

axis_h axis_h : :



pos_o (origin) is at pos_d=0, pos_w=0, pos_h=0, It marked with o

Parameters

- **aluprof_w** (*float*) – Width of the aluminum profile

- **belt_pos_h** (*float*) – The position along axis_h where the idler pulley that conveys the belt starts. THIS POSITION IS CENTERED at the ilder pulley
- **tens_h** (*float*) – height of the ilder tensioner
- **tens_w** (*float*) – width of the ilder tensioner
- **tens_d_inside** (*float*) – Max length (depth) of the ilder tensioner that is inside the holder
- **wall_thick** (*float*) – Thickness of the walls
- **in_fillet** (*float*) – radius of the inner fillets
- **boltaluprof_mtr** (*float*) – diameter (metric) of the bolt that attaches the tensioner holder to the aluminum profile (or whatever is attached to)
- **bolttens_mtr** (*float*) – diameter (metric) of the bolt for the tensioner
- **hold_bas_h** (*float*) – height of the base of the tensioner holder if 0, it will take wall_thick
- **opt_tens_chmf** (*int*) – 1: there is a chamfer at every edge of tensioner, inside the holder 0: there is a chamfer only at the edges along axis_w, not along axis_h
- **hold_hole_2sides** (*int*) – In the tensioner holder there is a hole to see inside, it can be at each side of the holder or just on one side 0: only at one side 1: at both sides
- **min_width** (*make the rim the minimum: the diameter of the washer*) – 0: normal width: the width of the aluminum profile 1: minimum width: diameter of the washer
- **tol** (*float*) – Tolerances to print
- **axis_d** (*FreeCAD.Vector*) – depth vector of coordinate system
- **axis_w** (*FreeCAD.Vector*) – width vector of coordinate system if V0: it will be calculated using the cross product: axis_l x axis_h
- **axis_h** (*FreeCAD.Vector*) – height vector of coordinate system
- **pos_d** (*int*) – location of pos along the axis_d 0: at the back of the holder 1: at the place where the tensioner can reach all the way inside 2: at the center of the base along axis_d, where the bolts to attach

  the holder base to the aluminum profile

  3: at the end of the base 4: at the end of the holder 5: at the center of the pulley 6: at the end of the idler tensioner 7: at the center of the pulley, when idler is all the way out 8: at the end of the idler tensioner, whenit is all the way out
- **pos_w** (*int*) – location of pos along the axis_w 0: at the center of symmetry 1: at the inner walls of the holder, which is the pulley radius 2: at the end of the holder (the top part, where the base starts) 3: at the center of the bolt holes to attach the holder base to the

  aluminum profile

  4: at the end of the piece along axis_w axes have direction. So if pos_w == 3, the piece will be drawn along the positive side of axis_w
- **pos_h** (*int*) – location of pos along the axis_h (0,1,2,3,4) 0: at the bottom of the holder 1: at the top of the base of the holder (for the bolts) 2: at the bottom of the

hole where the idler tensioner goes 3: at the middle point of the hole where the idler tensioner goes 4: at the top of the holder

- **pos** (*FreeCAD.Vector*) – position of the piece
- **for the set** (*Parameters*) –
- **tens_in_ratio** (*float*) – from 0 to 1, the ratio of the stroke that the tensioner is inside. if 1: it is all the way inside if 0: it is all the way outside (all the tens_stroke)

All the parameters and attributes of parent class SinglePart
prnt_ax [FreeCAD.Vector] Best axis to print (normal direction, pointing upwards)
d0_cen : int w0_cen : int h0_cen : int
indicates if pos_h = 0 (pos_d, pos_w) is at the center along axis_h, axis_d, axis_w, or if it is at the end. 1 : at the center (symmetrical, or almost symmetrical) 0 : at the end

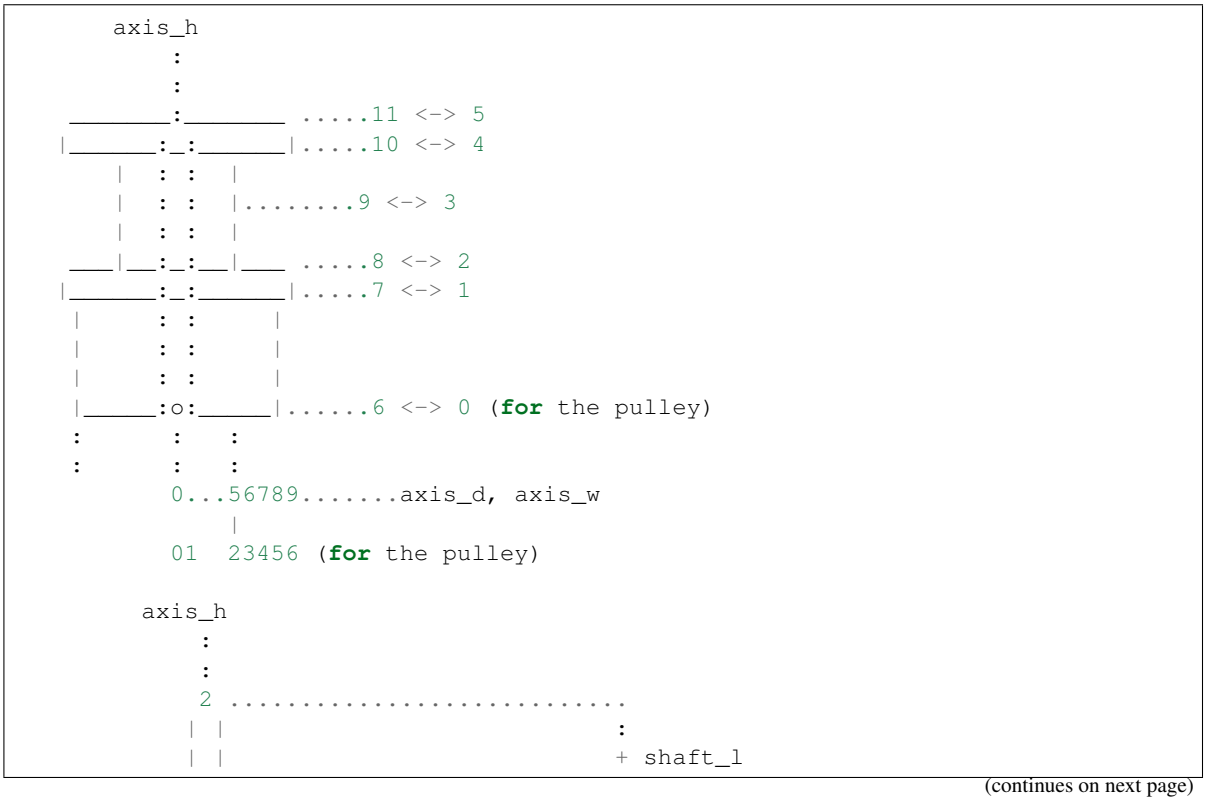
tot_d [float] total depth, including the idler tensioner
tot_d_extend [float] total depth including the idler tensioner, having it extended

Parameters:

class partset.**NemaMotorPulleySet**(*nema_size=17*, *base_l=32.0*, *shaft_l=24.0*, *shaft_r=0*, *circle_r=11.0*, *circle_h=2.0*, *chmf_r=1*, *rear_shaft_l=0*, *bolt_depth=3.0*, *pulley_pitch=2.0*, *pulley_n_teeth=20*, *pulley_toothed_h=7.5*, *pulley_top_flange_h=1.0*, *pulley_bot_flange_h=0*, *pulley_tot_h=16.0*, *pulley_flange_d=15.0*, *pulley_base_d=15.0*, *pulley_tol=0*, *pulley_pos_h=-1*, *axis_d=FreeCAD.Vector*, *axis_w=None*, *axis_h=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos_h=1*, *pos=FreeCAD.Vector*, *group=1*, *name=''*)

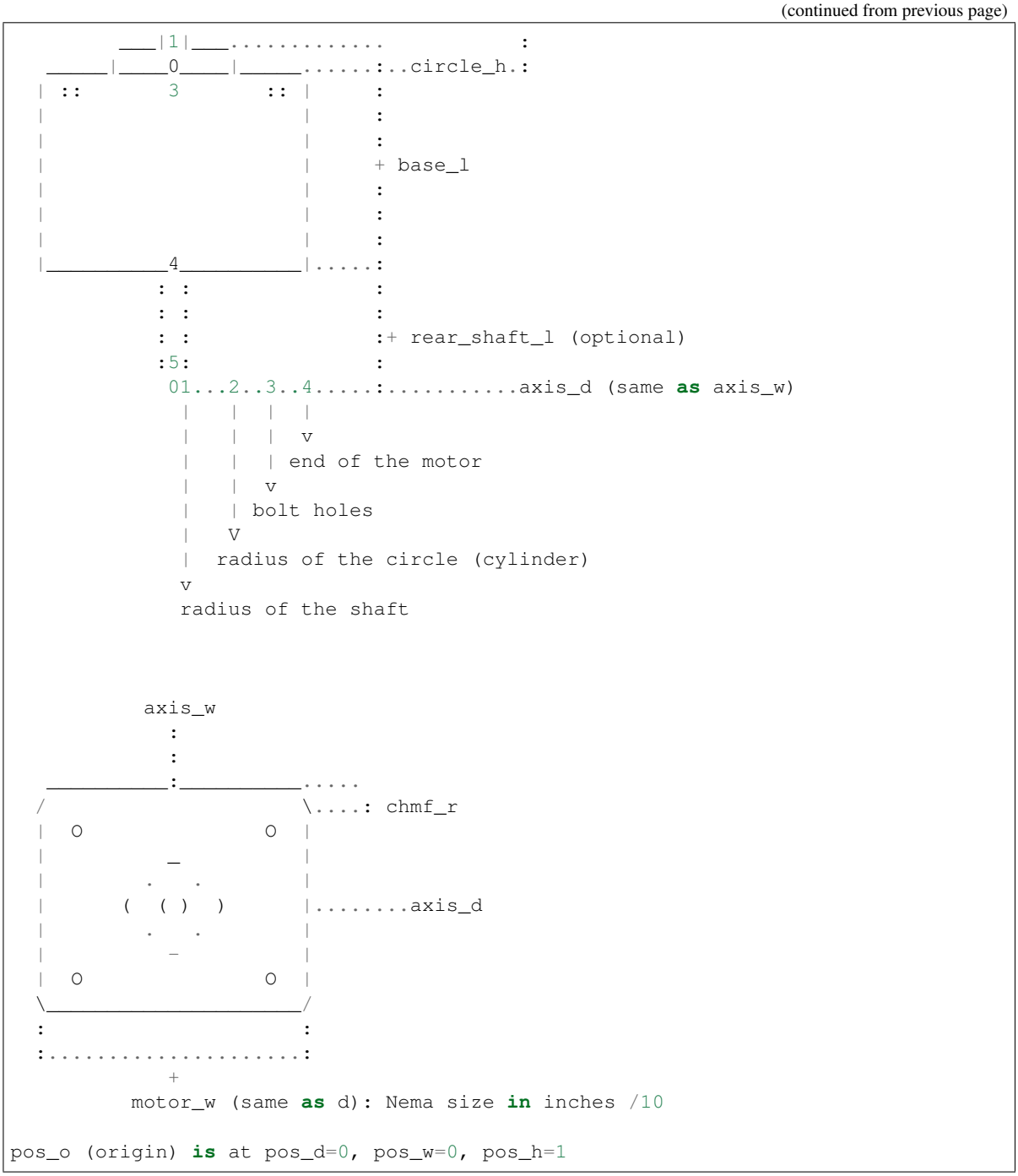
Set composed of a Nema Motor and a pulley

Number positions of the pulley will be after the positions of the motor



(continues on next page)

(continued from previous page)



Parameters

- **nema_size** (*dict*) – List of sizes defines in kcomps NEMA motor dimensions.
- **base_l** (*float,*) – Length (height) of the base
- **shaft_l** (*float,*) – Length (height) of the shaft, including the small cylinder (circle) at the base
- **shaft_r** (*float,*) – Radius of the shaft, if not defined, it will take the dimension defined in kcomp

- **circle_r** (*float,*) – Radius of the cylinder (circle) at the base of the shaft if 0 or circle_h = 0 -> no cylinder
- **circle_h** (*float,*) – Height of the cylinder at the base of the shaft if 0 or circle_r = 0 -> no cylinder
- **chmf_r** (*float,*) – Chamfer radius of the chamfer along the base length (height)
- **rear_shaft_l** (*float*) – Length of the rear shaft, 0 : no rear shaft
- **bolt_depth** (*float*) – Depth of the bolt holes of the motor
- **pulley_pitch** (*float/int*) – Distance between teeth: Typically 2mm, or 3mm
- **pulley_n_teeth** (*int*) – Number of teeth of the pulley
- **pulley_toothed_h** (*float*) – Height of the toothed part of the pulley
- **pulley_top_flange_h** (*float*) – Height (thickness) of the top flange, if 0, no top flange
- **pulley_bot_flange_h** (*float*) – Height (thickness) of the bot flange, if 0, no bottom flange
- **pulley_tot_h** (*float*) – Total height of the pulley
- **pulley_flange_d** (*float*) – Flange diameter, if 0, it will be the same as the base_d
- **pulley_base_d** (*float*) – Base diameter
- **pulley_tol** (*float*) – Tolerance for radius (it will substracted to the radius) twice for the diameter. Or added if a shape to substract
- **pulley_pos_h** (*float*) – position in mm of the pulley along the shaft
  - 0: it is at the base of the shaft
  - -1: the top of the pulley will be aligned with the end of the shaft
- **axis_d** (*FreeCAD.Vector*) – Depth vector of coordinate system (perpendicular to the height)
- **axis_w** (*FreeCAD.Vector*) – Width vector of coordinate system if V0: it will be calculated using the cross product: axis_h x axis_d
- **axis_h** (*FreeCAD.Vector*) – Height vector of coordinate system
- **pos_d** (*int*) – location of pos along the axis_d see drawing
  - Locations coinciding with the motor
    - 0: at the axis of the shaft
    - 1: at the radius of the shaft
    - 2: at the end of the circle(cylinder) at the base of the shaft
    - 3: at the bolts
    - 4: at the end of the piece
  - Locations of the pulley
    - 5: at the inner radius
    - 7: at the external radius
    - 7: at the pitch radius (outside the toothed part)

        * 8: at the end of the base (not the toothed part)
        * 9: at the end of the flange (V0 is no flange)
    - **pos_w** (*int*) – location of pos along the axis_w see drawing
        - Same locations of pos_d
    - **pos_h** (*int*) – location of pos along the axis_h, see drawing
        - 0: at the base of the shaft (not including the circle at the base of the shaft)
        - 1: at the end of the circle at the base of the shaft
        - 2: at the end of the shaft
        - 3: at the end of the bolt holes
        - 4: at the bottom base
        - 5: at the end of the rear shaft, if no rear shaft, it will be the same as pos_h = 4
        - 6: at the base of the pulley
        - 7: at the base of the bottom flange of the pulley
        - 8: at the base of the toothed part of the pulley
        - 9: at the center of the toothed part of the pulley
        - 10: at the end (top) of the toothed part of the pulley
        - 11: at the end (top) of the pulley of the pulley
    - **pos** (*FreeCAD.Vector*) – Position of the model
    - **name** (*str*) – Object name

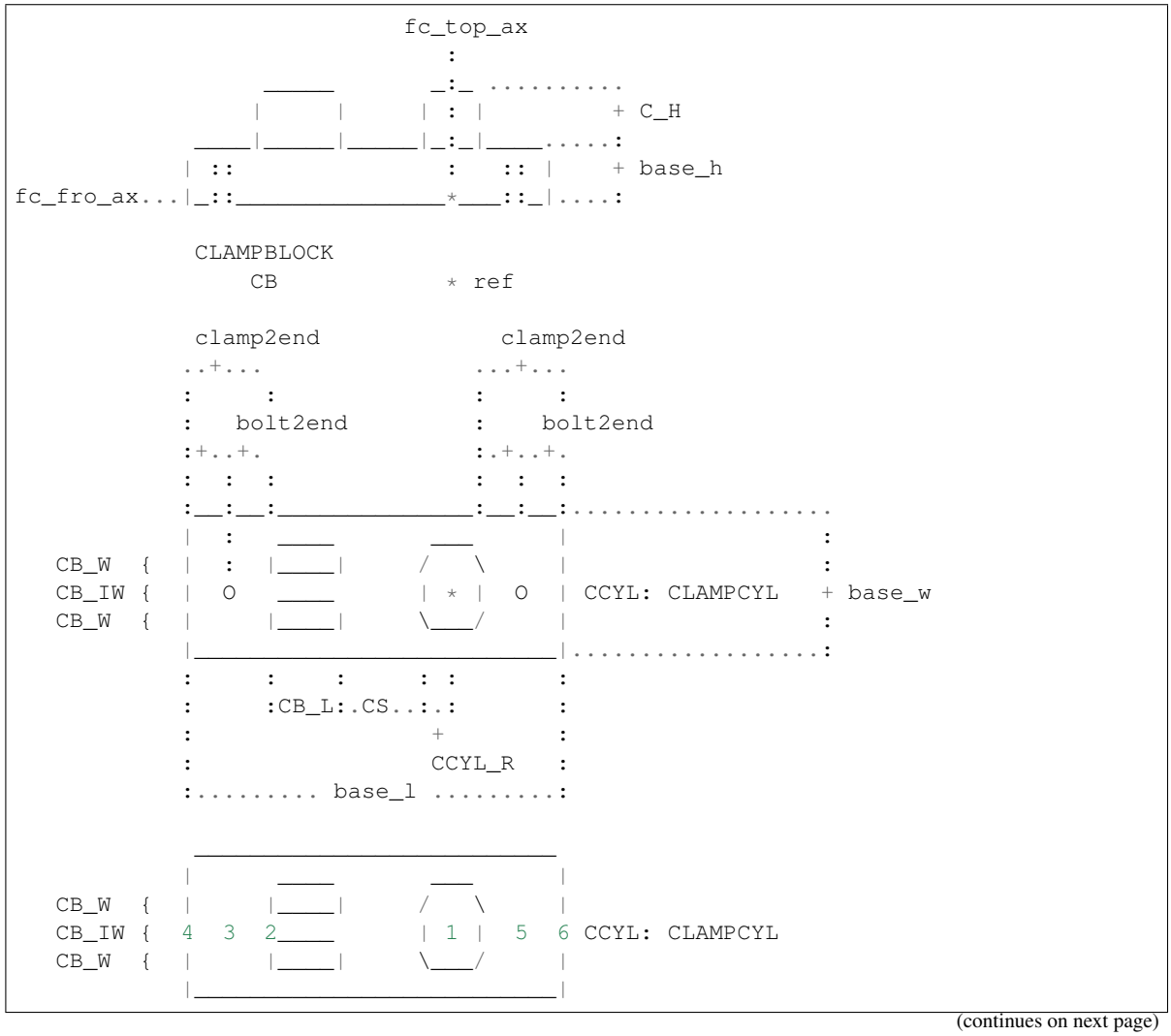
class beltcl.**BeltClamp**(*fc_fro_ax*, *fc_top_ax*, *base_h=2*, *base_l=0*, *base_w=0*, *bolt_d=3*, *bolt_csunk=0*, *ref=1*, *pos=FreeCAD.Vector*, *extra=1*, *wfco=1*, *intol=0*, *name='belt_clamp'*)

Similar to shp_topbeltclamp, but with any direction, and can have a base Creates a shape of a belt clamp. Just the rail and the cylinder and may have a rectangular base just one way: 2 clamp blocks It is referenced on the base of the clamp, but it may have 5 different positions

Parameters

- **fc_fro_ax** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the front, see pricture
- **fc_top_ax** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the top, see pricture
- **base_h** (*float*) – Height of the base,
    - if 0 and bolt_d=0: no base
    - if 0 and bolt_d!= 0: minimum base to have the bolt head and not touching with the belt (countersunk) if bolt_csunk > 0
- **base_l** (*float*) – Length of the base, if base_h not 0.
    - if 0 and bolt_d=0: will have the minimum length, defined by the clamp
    - if 0 and bolt_d!=0: will have the minimum length, defined by the clamp plus the minimum separation due to the bolt holes
- **base_w** (*float*) – Width of the base, if base_h not 0.
- **bolt_d** (*float*) – Diameter of the bolts, if zero, no bolts

- **bolt_csunk** (*float*) – If the bolt is countersunk
  - if >0: there is a whole to countersink the head of the bolt there will be an extra height if not enough bolt_d has to be > 0
  - if 0: no whole for the height, and no extra height
  - if >0, the size will determine the minimum height of the base, below the countersink hole
- **ref** (*int*) – Reference of the position (see picture below)
- **extra** (*float*) – If extra, it will have an extra height below the zero height, this is to be joined to some other piece
- **wfco** (*int*) –
  - if 1: With FreeCad Object: a freecad object is created
  - if 0: only the shape
- **intol** (*float*) – Internal extra tolerance to the dimension CB_IW, substracting to CB_W. If negative, makes CB_IW smaller.
- **name** (*str*) – Name of the freecad object, if created



(continues on next page)

(continued from previous page)

```
                  :    :
                  :CB_L:.CS.:

   References:
   1: cencyl: center of cylinder
   2: frontclamp: front of the clamps
   3: frontbolt
   4: frontbase
   5: backbolt
   6: backbase


                             fc_top_ax
                                :
                      _____      _:_
                     |     |     | : |
                  ___|_____|_____|_:_|____
                 |:..:               :  :..:|.....
fc_fro_ax...|_::_________________*___::_|....+ bolt_csunk (if not 0)
```

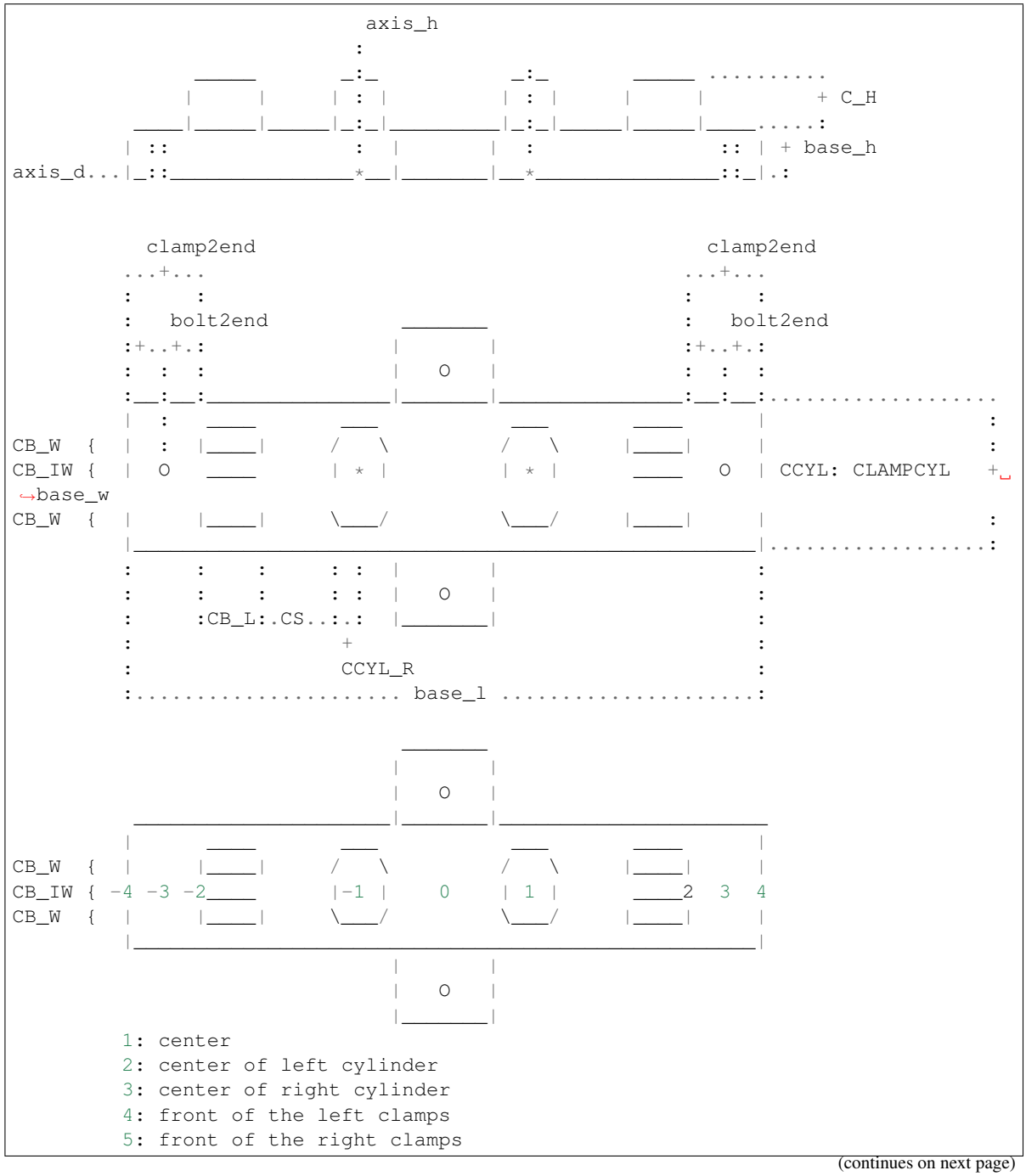
class beltcl.**DoubleBeltClamp**(*axis_h=FreeCAD.Vector*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *base_h=2*, *base_l=0*, *base_w=0*, *bolt_d=3*, *bolt_csunk=0*, *ref=1*, *pos=FreeCAD.Vector*, *extra=1*, *wfco=1*, *intol=0*, *name='double_belt_clamp'*)

Similar to BeltClamp, but in two ways Creates a shape of a double belt clamp. positions

Parameters

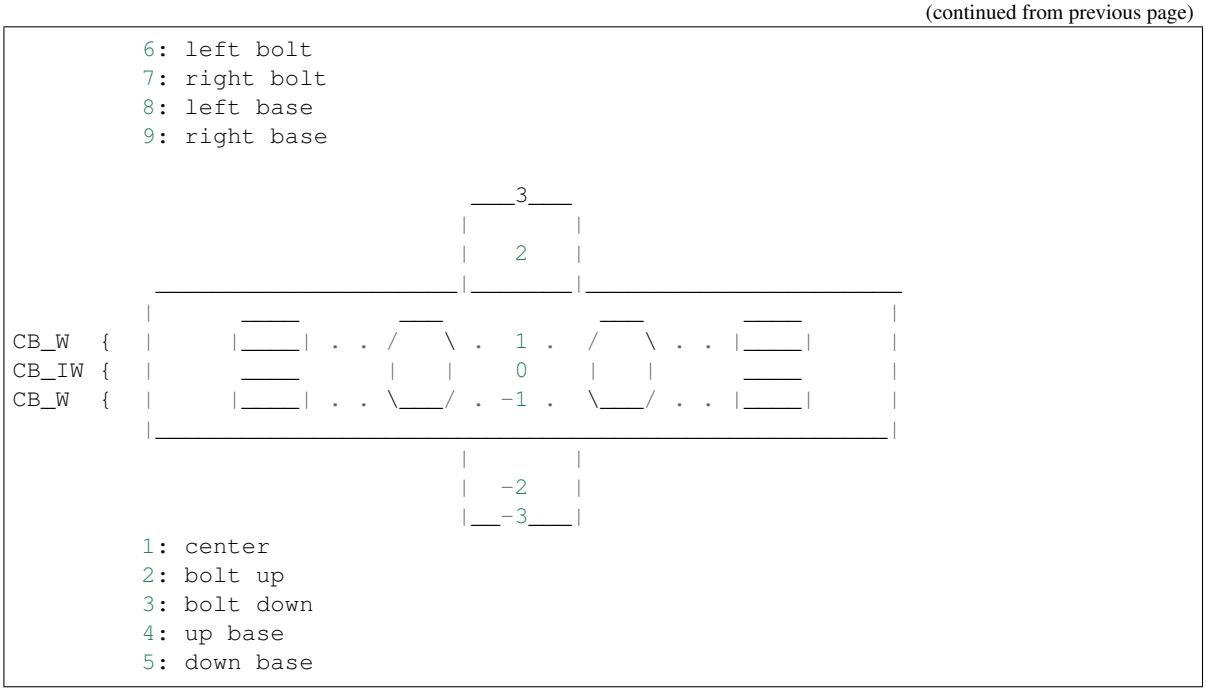
- **fc_fro_ax** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the front, see picture
- **fc_top_ax** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the top, see picture
- **base_h** (*float*) – Height of the base,
  - if 0 and bolt_d=0: no base
  - if 0 and bolt_d!= 0: minimum base to have the bolt head and not touching with the belt (countersunk) if bolt_csunk > 0
- **base_l** (*float*) – Length of the base, if base_h not 0.
  - if 0 and bolt_d=0: will have the minimum length, defined by the clamp
  - if 0 and bolt_d!=0: will have the minimum length, defined by the clamp plus the minimum separation due to the bolt holes
- **base_w** (*float*) – Width of the base, if base_h not 0.
- **bolt_d** (*float*) – Diameter of the bolts, if zero, no bolts
- **bolt_csunk** (*float*) – If the bolt is countersunk
  - if >0: there is a whole to countersink the head of the bolt there will be an extra height if not enough bolt_d has to be > 0
  - if 0: no whole for the height, and no extra height
  - if >0, the size will determine the minimum height of the base, below the countersink hole
- **ref** (*int*) – Reference of the position (see picture below)

- **extra** (*float*) – If extra, it will have an extra height below the zero height, this is to be joined to some other piece
- **wfco** (*int*) –
  - if 1: With FreeCad Object: a freecad object is created
  - if 0: only the shape
- **intol** (*float*) – Internal extra tolerance to the dimension CB_IW, substracting to CB_W. If negative, makes CB_IW smaller.
- **name** (*str*) – Name of the freecad object, if created



(continues on next page)

(continued from previous page)



class fc_clss.**Din934Nut**(*metric*, *axis_d_apo=0*, *h_offset=0*, *axis_h=FreeCAD.Vector*, *axis_d=None*, *axis_w=None*, *pos_h=0*, *pos_d=0*, *pos_w=0*, *pos=FreeCAD.Vector*, *model_type=0*, *name=''*)

Din 934 Nut

Parameters

- **metric** (*int (maybe float: 2.5)*) –
- **axis_h** (*FreeCAD.Vector*) –
- **axis_d_apo** (*int*) –
  - 0: default: axis_d points to the vertex
  - 1: axis_d points to the center of a side
- **h_offset** (*float*) – Distance from the top, just to place the Nut, see pos_h if negative, from the bottom
  - 0: default
- **axis_h** – Vector along the axis, height
- **axis_d** (*FreeCAD.Vector*) – Vector along the first vertex, a direction perpendicular to axis_h. It is not necessary if pos_d == 0. It can be None, but if None, axis_w has to be None
- **axis_w** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h and axis_d. It is not necessary if pos_w == 0. It can be None
- **pos_h** (*int*) – Location of pos along axis_h
  - 0: at the center
  - -1: at the base
  - 1: at the top
  - -2: at the base + h_offset

  - 2: at the top + h_offset
- **pos_d** (*int*) – Location of pos along axis_d (-2, -1, 0, 1, 2)
  - 0: pos is at the circunference center (axis)
  - 1: pos is at the inner circunsference, on axis_d, at r_in from the circle center
  - 2: pos is at the apothem, on axis_d
  - 3: pos is at the outer circunsference, on axis_d, at r_out from the circle center
- **pos_w** (*int*) – Location of pos along axis_w (-2, -1, 0, 1, 2)
  - 0: pos is at the circunference center
  - 1: pos is at the inner circunsference, on axis_w, at r_in from the circle center
  - 2: pos is at the apothem, on axis_w
  - 3: pos is at the outer circunsference, on axis_w, at r_out from the center
- **pos** (*FreeCAD.Vector*) – Position of the prism, taking into account where the center is
- **model_type** (*0*) – Not to print, just an outline
- **name** (*str*) – Name of the bolt

class fc_clss.**Din125Washer**(*metric*, *axis_h*, *pos_h*, *tol=0*, *pos=FreeCAD.Vector*, *model_type=0*, *name=''*)

Din 125 Washer, this is the regular washer

Parameters

- **metric** (*int (maybe float: 2.5)*) –
- **axis_h** (*FreeCAD.Vector*) – Vector along the cylinder height
- **pos_h** (*int*) – Location of pos along axis_h (0,1)
  - 0: the cylinder pos is at its base
  - 1: the cylinder pos is centered along its height
- **tol** (*float*) – Tolerance for the inner and outer radius. It is the tolerance for the diameter, so the radius will be added/subs have of this tolerance.
  - tol will be added to the inner radius (so it will be larger).
  - tol will be substracted to the outer radius (so it will be smaller).
- **model_type** (*int*) – Type of model:
  - 0: exact
  - 1: outline
- **pos** (*FreeCAD.Vector*) – Position of the cylinder, taking into account where the center is

---

Note: All the parameters and attributes of father class CylHole

---

metric

Metric of the washer

Type int or float (in case of M2.5) or even str for inches ?

model_type

Type int

class fc_clss.**Din9021Washer**(*metric*, *axis_h*, *pos_h*, *tol=0*, *pos=FreeCAD.Vector*, *model_type=0*, *name=''*)

Din 9021 Washer, this is the larger washer

Parameters

- **metric** (*int (maybe float: 2.5)*) –
- **axis_h** (*FreeCAD.Vector*) – Vector along the cylinder height
- **pos_h** (*int*) – Location of pos along axis_h (0,1)
  - 0: the cylinder pos is at its base
  - 1: the cylinder pos is centered along its height
- **tol** (*float*) – Tolerance for the inner and outer radius. It is the tolerance for the diameter, so the radius will be added/subs have of this tolerance
  - tol will be added to the inner radius (so it will be larger)
  - tol will be substracted to the outer radius (so it will be smaller)
- **model_type** (*int*) – Type of model:
  - 0: exact
  - 1: outline
- **pos** (*FreeCAD.Vector*) – Position of the cylinder, taking into account where the center is

---

Note: All the parameters and attributes of father class CylHole

---

metric

Metric of the washer

Type int or float (in case of M2.5) or even str for inches ?

model_type

Type int

class fc_clss.**Din912Bolt**(*metric*, *shank_l*, *shank_l_adjust=0*, *shank_out=0*, *head_out=0*, *axis_h=FreeCAD.Vector*, *axis_d=None*, *axis_w=None*, *pos_h=0*, *pos_d=0*, *pos_w=0*, *pos=FreeCAD.Vector*, *model_type=0*, *name=''*)

Din 912 bolt. hex socket bolt

Parameters

- **metric** (*int (may be float: 2.5)*) –
- **shank_l** (*float*) – length of the bolt, not including the head
- **shank_l_adjust** (*int*) –
  - 0: shank length will be the size of the parameter shank_l
  - -1: shank length will be the size of the closest shorter or equal to shank_l available lengths for this type of bolts
  - 1: shank length will be the size of the closest larger or equal to shank_l available lengths for this type of bolts
- **shank_out** (*float*) – Distance to the end of the shank, just for positioning, it doesnt change shank_l

  - 0: default

Note: I dont think it is necessary, but just in case

- **head_out** (*float*) – Distance to the end of the head, just for positioning, it doesnt change head_l
  - 0: default

Note: I dont think it is necessary, but just in case

- **axis_h** (*FreeCAD.Vector*) – Vector along the axis of the bolt, pointing from the head to the shank
- **axis_d** (*FreeCAD.Vector*) – Vector along the radius, a direction perpendicular to axis_h If the head is hexagonal, the direction of one vertex
- **axis_w** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h and axis_d. It is not necessary if pos_w == 0. It can be None
- **pos_h** (*int*) – Location of pos along axis_h
  - 0: top of the head, considering head_out,
  - 1: position of the head not considering head_out if head_out = 0, it will be the same as pos_h = 0
  - 2: end of the socket, if no socket, will be the same as pos_h = 0
  - 3: union of the head and the shank
  - 4: where the screw starts, if all the shank is screwed, it will be the same as pos_h = 2
  - 5: end of the shank, not considering shank_out
  - 6: end of the shank, if shank_out = 0, will be the same as pos_h = 5
  - 7: top of the head, considering xtr_head_l, if xtr_head_l = 0 will be the same as pos_h = 0
- **pos_d** (*int*) – Location of pos along axis_d (symmetric)
  - 0: pos is at the central axis
  - 1: radius of the shank
  - 2: radius of the head
- **pos_w** (*int*) – Location of pos along axis_d (symmetric)
  - 0: pos is at the central axis
  - 1: radius of the shank
  - 2: radius of the head
- **pos** (*FreeCAD.Vector*) – Position of the bolt, taking into account where the pos_h, pos_d, pos_w are
- **model_type** (*0*) – Not to print, just an outline
- **name** (*str*) – Name of the bolt

Optical

comp_optic.**f_breadboard**(*d_breadboard*, *length*, *width*, *cl=1*, *cw=1*, *ch=1*, *fc_dir_h=FreeCAD.Vector*, *fc_dir_w=FreeCAD.Vector*, *pos=FreeCAD.Vector*, *name='breadboard'*)

Parameters

- **d_breadboard** (*dict*) – Dictionary with the values
- **length** (*float*) –
- **width** (*float*) –
- **cl** (*int*) –
  - 1: the length dimension is centered
  - 0: it is not centered
- **cw** (*int*) –
  - 1: the width dimension is centered
  - 0: it is not centered
- **ch** (*int*) –
  - 1: the height dimension is centered
  - 0: it is not centered
- **fc_dir_h** (*FreeCAD.Vector*) – Vector with the direction of the height
- **fc_dir_w** (*FreeCAD.Vector*) – Vector with the direction of the width
- **pos** (*FreeCAD.Vector*) – Placement of the model
- **name** (*str*) – object name

Returns Object with the sape of a BreadBoard

Return type FreeCAD Object

comp_optic.**f_cagecube**(*d_cagecube*, *axis_thru_rods='x'*, *axis_thru_hole='y'*, *name='cagecube'*, *toprint_tol=0*)

Creates a cage cube, it creates from a dictionary

Parameters

- **d_cagecube** – Dictionary with the dimensions of the cage cube, defined in kcomp_optic.py
- **axis_thru_rods** (*str*) – Direction of rods: 'x', 'y', 'z'
- **axis_thru_hole** (*str*) – Direction big thru_hole: 'x', 'y', 'z'.

Note: Cannot be the same as axis_thru_rods There are 6 possible orientations: Thru-rods can be on X, Y or Z axis thru-hole can be on X, Y, or Z axis, but not in the same as thru-rods

---

- **toprint_tol** (*float*) –
  - 0, dimensions as they are.
  - >0 value of tolerances of the holes. multiplies the normal tolerance in kcomp.TOL

Returns

- *CageCube. The freeCAD object can be accessed by the*
- *attribute .fco*

comp_optic.**f_cagecubehalf**(*d_cagecubehalf*, *axis_1='x'*, *axis_2='y'*, *name='cagecubehalf'*)

Dreates a half cage cube: 2 perpendicular sides, and a 45 degree angle side. It creates from a dictionary

Parameters

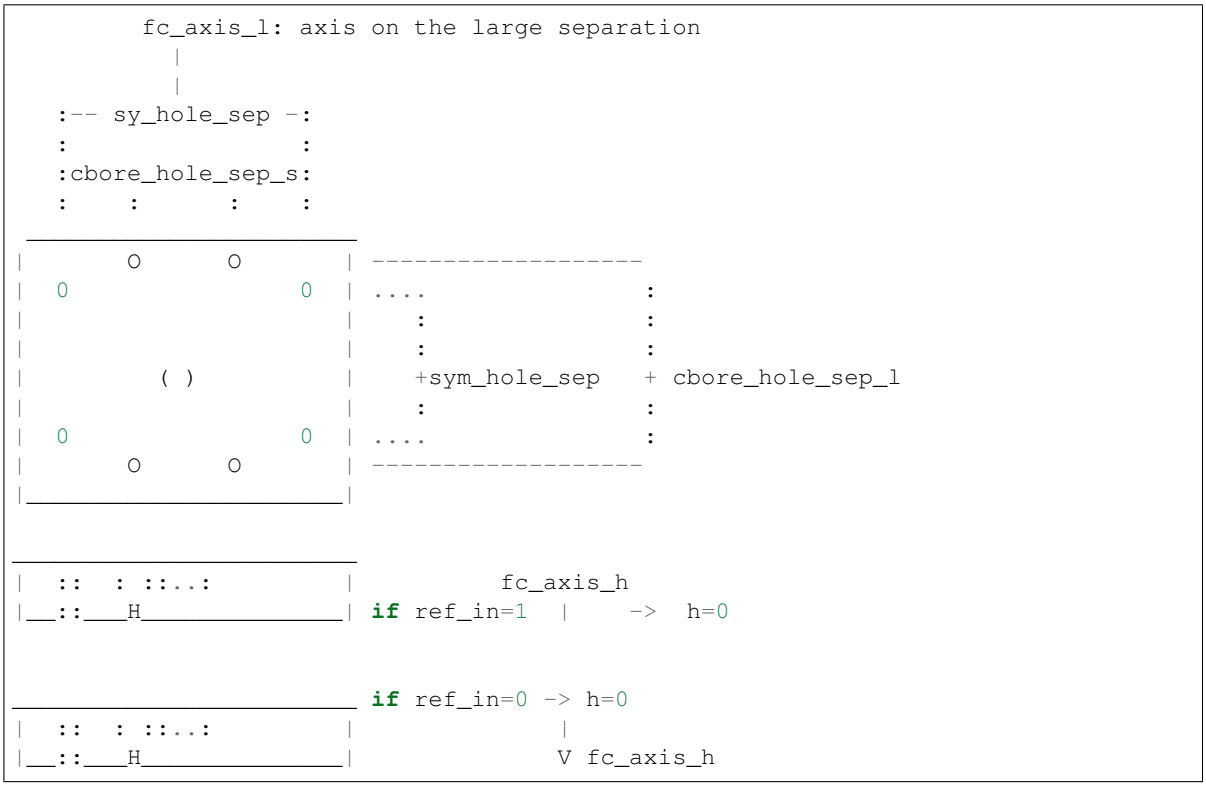
- **d_cagecubehalf** (*dict*) – Dictionary with the dimensions of the cage cube, defined in kcomp_optic.py
- **axis_1** (*str*) – Direction of the first right side: 'x', 'y', 'z', '-x', '-y', '-z'
- **axis_2** (*str*) – Direction big the other right side: 'x', 'y', 'z', '-x', '-y', '-z'

  Note: Cannot be the same as axis_1, or its negated. Has to be perpendicular There are 24 possible orientations: 6 possible axis_1 and 4 axis_2 for each axis_1

- **name** (*str*) – Name of the freecad object

class comp_optic.**Lb1cPlate**(*d_plate*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_l=FreeCAD.Vector*, *ref_in=1*, *pos=FreeCAD.Vector*, *name='lb1c_plate'*)

Creates a LB1C/M plate from thorlabs. The plate is centered



Parameters

- **d_plate** (*dict*) – Dictionary with the dimensions
- **fc_axis_h** (*FreeCAD.Vector*) – Direction of the vertical (thickness)
- **fc_axis_l** (*FreeCAD.Vector*) – Direction of the large distance of the counterbored asymmetrical holes

- **ref_in** (*int*) –
  - 1: fc_axis_h starts on the inside to outside of the plate
  - 0: fc_axis_h starts on the outside to inside of the plate
- **pos** (*FreeCAD.Vector*) – Position of the center. The center is on the center of the plate, but on the axis_h can be in either side depending on ref_in
- **name** (*str*) – Name

class comp_optic.**Lb2cPlate**(*fc_axis_h*, *fc_axis_l*, *cl=1*, *cw=1*, *ch=0*, *pos=FreeCAD.Vector*, *name='lb2c_plate'*)

Same as plate_lb2c, but it creates an object.

fc_axis_h: FreeCAD.Vector Direction of the vertical (thickness)

fc_axis_l: FreeCAD.Vector Direction of the large distance of the counterbored asymmetrical holes

cl: int

- 1: centered on the fc_axis_l direction

cw: int

- 1: centered on the axis_small direction (perpendicular to fc_axis_l and fc_axis_h)

ch: int

- 1: centered on the vertical direction (thickness)

pos: FreeCAD.Vector

Position of the center. The center is on the center of the plate, but on the axis_h can be in either side depending on ref_in

name: str Name

comp_optic.**lcp01m_plate**(*d_lcp01m_plate={'L': 71.11999999999999, 'chamfer_r': 2.0, 'mhole_d': 4.0, 'mhole_depth': 6.5, 'sym_hole_d': 6.0, 'sym_hole_sep': 60.0, 'thick': 12.7, 'thruhole_d': 51.689}*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_m=FreeCAD.Vector*, *fc_axis_p=FreeCAD.Vector*, *cm=1*, *cp=1*, *ch=1*, *pos=FreeCAD.Vector*, *wfco=1*, *name='LCP01M_PLATE'*)

Creates a lcp01m_plate side.

It creates from a dictionary

Parameters

- **d_lcp01m_plate** (*dict*) – Dictionary with the dimensions of the plate defined in kcomp_optic.py
- **fc_axis_h** (*FreeCAD.Vector*) – Direction of the vertical (thickness) from the inside of the plate
- **fc_axis_m** (*FreeCAD.Vector*) – Direction of the mounting hole goes in the mounting hole
- **fc_axis_p** (*FreeCAD.Vector*) – Perpendicular direction of axis_h and axis_m, only used if not centered on this axis
- **cm** (*int*) –
  - 1: centered on the fc_axis_m direction (point 2)
  - 0: it will be on the mounting hole (point 1)

- **cp** (*int*) –
  - 1: centered on the perpendicular direction of fc_axis_m and fc_axis_h), if 0, fc_axis_p needs to be defined
  - 0: points 5 (cm==0) or 6 (cm==1)
- **ch** (*int*) –
  - 1: centered on the vertical direction (thickness)
- **pos** (*FreeCAD.Vector*) – Position of the center.
- **wfco** (*int*) –
  - 1: a FreeCAD object is created
  - 0: only de shape is created
- **name** (*str*) – name of the freecad object (if created)

comp_optic.**lcpb1m_base**(*d_lcpb1m_base={'d_lip': 2.5, 'd_mount': 8.9, 'd_tot': 15.2, 'h_slot': 3.8, 'h_sup': 8.9, 'h_tot': 10.8, 'l_mbolt_d': 4.0, 's_mholes_d': 3.0, 's_mholes_dist': 50.0, 'slot_d': 6.0, 'slot_dist': 100.0, 'w_sup': 82.6, 'w_tot': 120.7}*, *fc_axis_d=FreeCAD.Vector*, *fc_axis_w=FreeCAD.Vector*, *fc_axis_h=FreeCAD.Vector*, *ref_d=1*, *ref_w=1*, *ref_h=1*, *pos=FreeCAD.Vector*, *wfco=1*, *toprint=0*, *name='Lcpb1mBase'*)

Creates a lcpb1m_base for plates side, it creates from a dictionary

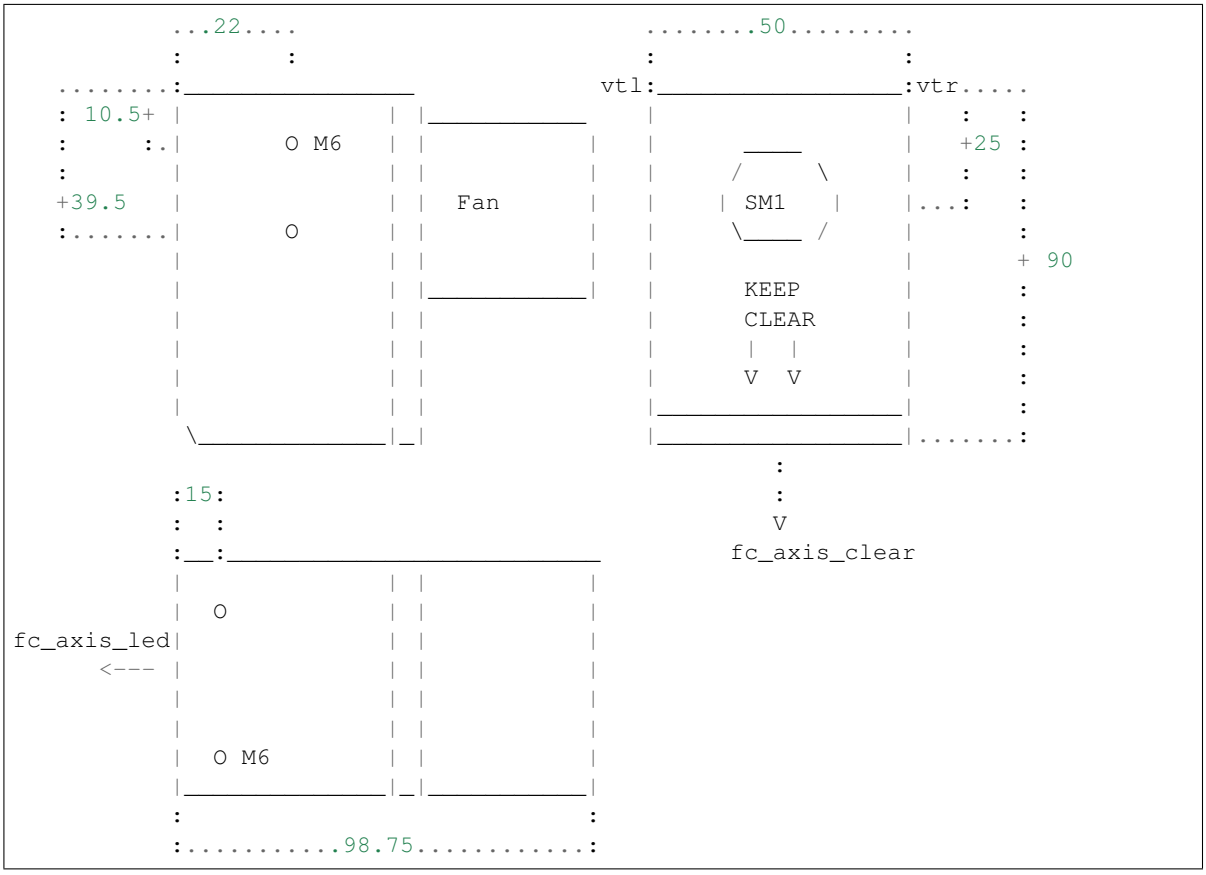
Parameters

- **d_lcpb1m_base** (*dict*) – Dictionary with the dimensions
- **fc_axis_d** (*FreeCAD.Vector*) – Direction of the deep
- **fc_axis_w** (*FreeCAD.Vector*) – Direction of the width
- **fc_axis_h** (*FreeCAD.Vector*) – Direction of the height
- **ref_d** (*int*) – Position in the fc_axis_d:
  - 1: top side
  - 2: center
  - 3: lower side
- **ref_w** (*int*) – Position in the fc_axis_w:
  - 1: center
  - 2: center in left slot
  - 3: left side
- **ref_h** (*int*) – Positionin the fc_axis_h:
  - 1: base
  - 2: top
- **pos** (*FreeCAD.Vector*) – Position of the center.
- **wfco** (*int*) –
  - 1: a FreeCAD object is created
  - 0: only de shape is created

- **toprint** (*int*) – 1 if you want to include a triangle between the shank and the head to support the shank and not building the head on the air using kcomp.LAYER3D_H
- **name** (*str*) – Name of the freecad object (if created)

comp_optic.**PrizLed**(*fc_axis_led=FreeCAD.Vector*, *fc_axis_clear=FreeCAD.Vector*, *pos=FreeCAD.Vector*, *name='prizmatix_led'*)

Creates the shape of a Prizmatix UHP-T-Led The drawing is very rough, and the original drawing lacks many dimensions



Parameters

- **fc_axis_led** (*FreeCAD.Vector*) – Direction of the led
- **fc_axis_clear** (*FreeCAD.Vector*) – Direction of the arrows indicating to keep clear
- **pos** (*FreeCAD.Vector*) – Position of the LED, on the center of the SM1 thread
- **name** (*str*) – Object name

comp_optic.**SM1TubelensSm2**(*sm1l_size*, *fc_axis=FreeCAD.Vector*, *ref_sm1=1*, *pos=FreeCAD.Vector*, *ring=1*, *name='tubelens_sm1_sm2'*)

Creates a componente formed by joining: the lens tube SM1LXX + SM1A2 + SM2T2, so we have:

- on one side a SM1 external thread
- on the other side a SM2 external thread

And inside we have a SM1 tube lens Since there are threads, they may be inserted differently, so size may vary. Therefore, size are approximate, and also, details are not drawn, such as threads, or even the part that contains the thread is not drawn:

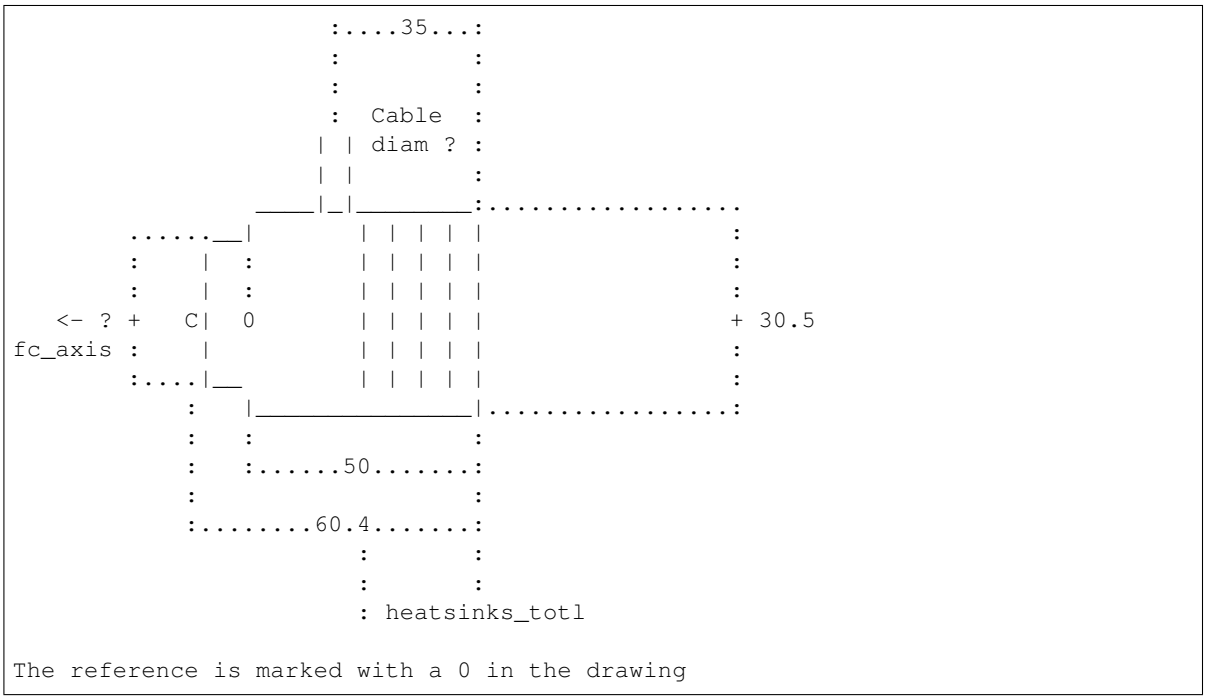
```
                                  .........................
                lens tube     _HH ...........             :
                             ||..|          :             :
LED               SM1LXX     ||   |          :             :
_____    ..... ______________||   |          :             :
   __|   :   _|..............||   |          :             :
  |      :  | :            : ||   |          :             :
  |  30.5+  | :    SM1     : ||   |          +55.9         + 57.2
  |      :  | :            : ||   |          :SM2_TLENS_D  :
  |__    :  |_:............:.||   |          :             :
_____|   :....|______________||   |          :             :
            : :              ||   |          :             :
            :3:    Lext      ||..|          :             :
                             ||__|..........:             :
 SM1_TLENS_D=30.5              HH ........................:
                           0.7: 5.6


So it will be:


  lens tube     _HH
               ||..|
    SM1LXX     ||   |
 ______________||   |
|..............||   |
|              ||   |
|    SM1       ||   |
|              ||   |
|..............||   |
|______________||   |
               ||   |
     Lext      ||..|
               ||__|
                 HH

The 3mm thread on the left is not drawn
```

Parameters

- **sm1l_size** (*float*) – Length of the side of the cube (then it will be halved)
- **fc_axis** (*FreeCAD.Vector*) – Direction of the tube lens: FreeCAD.Vector
- **axis_2** (*FreeCAD.Vector*) – Direction big the other right side:
- **ref_sm1** (*int*) –
  - 1: if the position is referred to the sm1 end
  - 0: if the position is referred to the ring end
- **pos** (*FreeCAD.Vector*) – Position of the object
- **ring** (*int*) –
  - 1: if there is ring
  - 0: there is no ring, so just the thread, at it is not drawn
- **name** (*str*) – Name of the freecad object

comp_optic.**ThLed30**(*fc_axis=FreeCAD.Vector*, *fc_axis_cable=FreeCAD.Vector*, *pos=FreeCAD.Vector*, *name='thled30'*)

Creates the shape of a Thorlabs Led with 30.5 mm Heat Sink diameter The drawing is very rough



Parameters

- **fc_axis** (*FreeCAD.Vector*) – axis on the direction of the led
- **fc_axis_cable** (*FreeCAD.Vector*) – axis on the direction of the cable
- **pos** (*FreeCAD.Vector*) – Placement of the object
- **name** (*str*) – Object name

1.4.6 Functions details

fcfun

class fcfun.**NutHole**(*nut_r*, *nut_h*, *hole_h*, *name*, *extra=1*, *nuthole_x=1*, *cx=0*, *cy=0*, *holedown=0*)

Adding a Nut hole (hexagonal) with a prism attached to introduce the nut. Tolerances are included

Parameters

- **nut_r** (*float*) – Circumradius of the hexagon
- **nut_h** (*float*) – Height of the nut, usually larger than the actual nut height, to be able to introduce it

- **hole_h** (*float*) – The hole height, from the center of the hexagon to the side it will see light
- **name** (*str*) – Name of the object (string)
- **extra** (*int*) –
  - 1 if you want 1 mm out of the hole, to cut
- **nuthole_x** (*int*) –
  - 1 : if you want that the nut height to be along the X axis and the 2*apotheme on the Y axis ie. Nut hole facing X
  - 0 : if you want that the nut height to be along the Y axis ie. Nut hole facing Y
- **cx** (*int*) –
  - 1 : if you want the coordinates referenced to the x center of the piece it can be done because it is a new shape formed from the union
- **cy** (*int*) –
  - 1 : if you want the coordinates referenced to the y center of the piece
- **holedown** (*int*) –

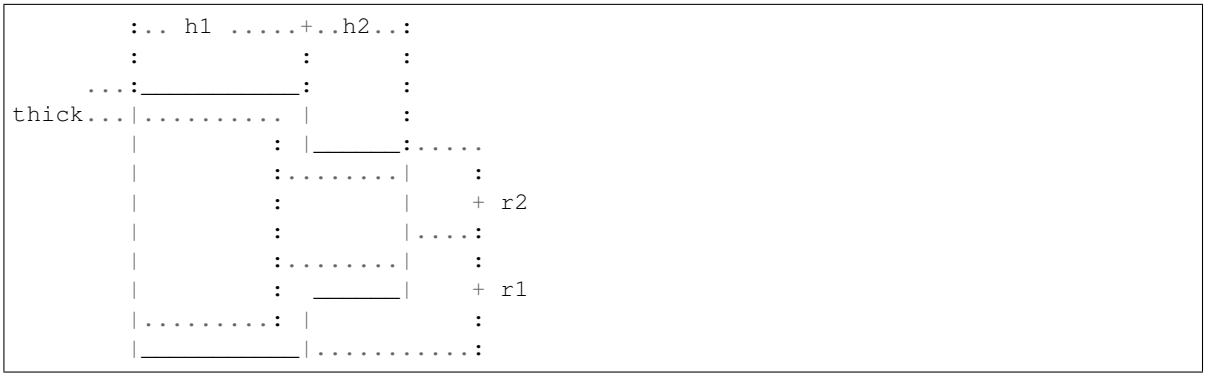
  I THINK IS THE OTHER WAY; CHECK
  - 0: the z0 is the bottom of the square (hole)
  - 1: the z0 is the center of the hexagon (nut) it can be done because it is a new shape formed from the union

Returns FreeCAD object of a nut hole

Return type FreeCAD Object

fcfun.**add2CylsHole**(*r1*, *h1*, *r2*, *h2*, *thick*, *normal=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Creates a piece formed by 2 hollow cylinders



Parameters

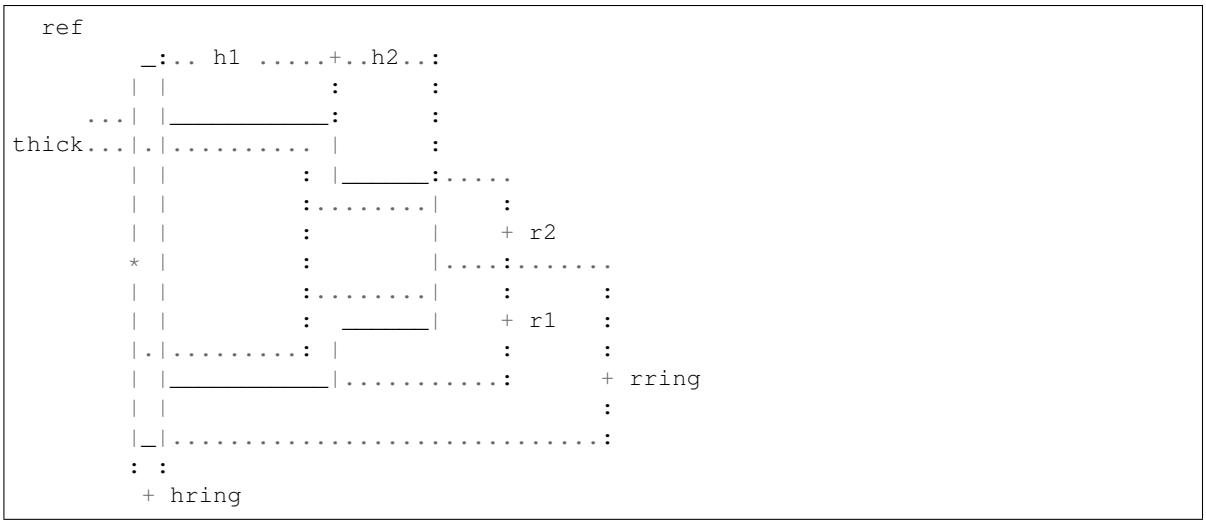
- **r1** (*float*) – Radius of the 1st cylinder. The first cylinder relative to the position pos
- **h1** (*float*) – Height of the 1st cylinder (seen from outside)
- **r2** (*float*) – Radius of the 2nd cylinder
- **h2** (*float*) – Height of the 2nd cylinder (seen from outside)
- **normal** (*FreeCAD.Vector*) – Direction of the height
- **pos** (*FreeCAD.Vector*) – Position of the center

Returns FreeCAD Shape of a two cylinders

Return type Shape

fcfun.**add3CylsHole**(*r1*, *h1*, *r2*, *h2*, *rring*, *hring*, *thick*, *normal=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Creates a piece formed by 2 hollow cylinders, and a ring on the side of the larger cylinder



Parameters

- **r1** (*float*) – Radius of the 1st cylinder. The first cylinder relative to the position pos (if this is larger than r2, the ring will go first)
- **h1** (*float*) – Height of the 1st cylinder (seen from outside)
- **r2** (*float*) – Radius of the 2nd cylinder
- **h2** (*float*) – Height of the 2nd cylinder (seen from outside)
- **rring** (*float*) – Radius of the ring, it has to be larger than r1, and r2
- **hring** (*float*) – Height of the ring, it has to be larger than r1, and r2
- **thick** (*float*) – Thickness of the walls, excluding the ring
- **normal** (*FreeCAD.Vector*) – Direction of the height
- **pos** (*FreeCAD.Vector*) – Position of the center

Returns FreeCAD Shape of a three cylinders

Return type Shape

fcfun.**addBolt**(*r_shank*, *l_bolt*, *r_head*, *l_head*, *hex_head=0*, *extra=1*, *support=1*, *headdown=1*, *name='bolt'*)

Creates the hole for the bolt shank and the head or the nut Tolerances have to be included

Parameters

- **r_shank** (*float*) – Radius of the shank (tolerance included)
- **l_bolt** (*float*) – Total length of the bolt: head & shank
- **r_head** (*float*) – Radius of the head (tolerance included)
- **l_head** (*float*) – Length of the head
- **hex_head** (*int*) –

  Indicates if the head is hexagonal or rounded

- 1: hexagonal
- 0: rounded

- **h_layer3d** (*float*) – Height of the layer for printing, if 0, means that the support is not needed
- **extra** (*int*) – 1 if you want 1 mm on top and bottom to avoid cutting on the same plane pieces after making cuts (boolean difference)
- **support** (*int*) – 1 if you want to include a triangle between the shank and the head to support the shank and not building the head on the air using kcomp.LAYER3D_H
- **headdown** (*int*) –
  - 1 if the head is down.
  - 0 if it is up

Returns FreeCAD Object of a bolt

Return type FreeCAD Object

fcfun.**addBoltNut_hole**(*r_shank*, *l_bolt*, *r_head*, *l_head*, *r_nut*, *l_nut*, *hex_head=0*, *extra=1*, *supp_head=1*, *supp_nut=1*, *headdown=1*, *name='bolt'*)

Creates the hole for the bolt shank, the head and the nut. The bolt head will be at the bottom, and the nut will be on top Tolerances have to be already included in the arguments values

Parameters

- **r_shank** (*float*) – Radius of the shank (tolerance included)
- **l_bolt** (*float*) – Total length of the bolt: head & shank
- **r_head** (*float*) – Radius of the head (tolerance included)
- **l_head** (*float*) – Length of the head
- **r_nut** (*float*) – Radius of the nut (tolerance included)
- **l_nut** (*float*) – Length of the nut. It doesn't have to be the length of the nut but how long you want the nut to be inserted
- **hex_head** (*int*) –

  Indicates if the head is hexagonal or rounded
  - 1: hexagonal
  - 0: rounded
- **zpos_nut** (*float*) – Indicates the height position of the nut, the lower part
- **h_layer3d** (*float*) – Height of the layer for printing, if 0, means that the support is not needed
- **extra** (*int*) – 1 if you want 1 mm on top and bottom to avoid cutting on the same plane pieces after making differences
- **support** (*int*) – 1 if you want to include a triangle between the shank and the head to support the shank and not building the head on the air using kcomp.LAYER3D_H

Returns FreeCAD Object of a Nut Hole

Return type FreeCAD Object

fcfun.**addBox**(*x*, *y*, *z*, *name*, *cx=False*, *cy=False*)

Adds a box, centered on the specified axis x and/or y, with its Placement and Rotation at zero. So it can be referenced absolutely from its given position

Parameters

- **x** (*float*) – Length
- **y** (*float*) – Width
- **z** (*float*) – Height
- **name** (*str*) – Object Name
- **cx** (*Boolean*) – Centered in axis x
- **cy** (*Boolean*) – Centered in axis y

Returns FreeCAD.Object with the shape of a box

Return type FreeCAD.Object

fcfun.**addBox_cen**(*x*, *y*, *z*, *name*, *cx=False*, *cy=False*, *cz=False*)

Adds a box, centered on the specified axis, with its Placement and Rotation at zero. So it can be referenced absolutely from its given position

Parameters

- **x** (*float*) – Length
- **y** (*float*) – Width
- **z** (*float*) – Height
- **name** (*str*) – Object Name
- **cx** (*Boolean*) – Centered in the X axis
- **cy** (*Boolean*) – Centered in the Y axis
- **cz** (*Boolean*) – Centered in the Z axis

Returns FreeCAD.Object with the shape of a box

Return type FreeCAD.Object

fcfun.**addCyl**(*r*, *h*, *name*)

Add cylinder

Parameters

- **r** (*float*) – Radius
- **h** (*float*) – Height

Returns Cylinder

Return type FreeCAD Object

fcfun.**addCylHole**(*r_ext*, *r_int*, *h*, *name*, *axis='z'*, *h_disp=0*)

Add cylinder, with inner hole:

Parameters

- **r_ext** (*float*) – External radius,
- **r_int** (*float*) – Internal radius,
- **h** (*float*) – Height
- **name** (*str*) – Object name
- **axis** (*str*) –

  'x', 'y' or 'z'

- 'x' will along the x axis
- 'y' will along the y axis
- 'z' will be vertical

- **h_disp** (*int*) –

  Displacement on the height.

  - if 0, the base of the cylinder will be on the plane
  - if -h/2: the plane will be cutting h/2

Returns Cylinder with hole

Return type FreeCAD.Object

fcfun.**addCylHolePos**(*r_out*, *r_in*, *h*, *name*, *normal=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Same as addCylHole, but avoiding the creation of many FreeCAD objects

Add cylinder, with inner hole

Parameters

- **r_out** (*float*) – Outside radius
- **r_in** (*float*) – Inside radius
- **h** (*float*) – Height
- **name** (*str*) – Object name
- **normal** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the normal (if its module is not one, the height will be larger than h
- **pos** (*FreeCAD.Vector*) – Position of the cylinder

Returns FreeCAD Shape of a cylinder with hole

Return type Shape

fcfun.**addCylPos**(*r*, *h*, *name*, *normal=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Same as addCyl_pos, but avoiding the creation of many FreeCAD objects

Parameters

- **r** (*float*) – Radius,
- **h** (*float*) – Height
- **name** (*str*) – Object name
- **normal** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the normal (if its module is not one, the height will be larger than h
- **pos** (*FreeCAD.Vector*) – Position of the cylinder

Returns Cylinder

Return type FreeCAD Object

fcfun.**addCyl_pos**(*r*, *h*, *name*, *axis='z'*, *h_disp=0*)

Add cylinder in a position. So it is in a certain position, with its Placement and Rotation at zero. So it can be referenced absolutely from its given position

Parameters

- **r** (*float*) – Radius
- **h** (*float*) – Height

- **name** (*str*) – Name
- **axis** (*str*) –

  'x', 'y' or 'z'

  - 'x' will along the x axis
  - 'y' will along the y axis
  - 'z' will be vertical
- **h_disp** (*int*) –

  Displacement on the height.

  - if 0, the base of the cylinder will be on the plane
  - if -h/2: the plane will be cutting h/2

Returns Cylinder

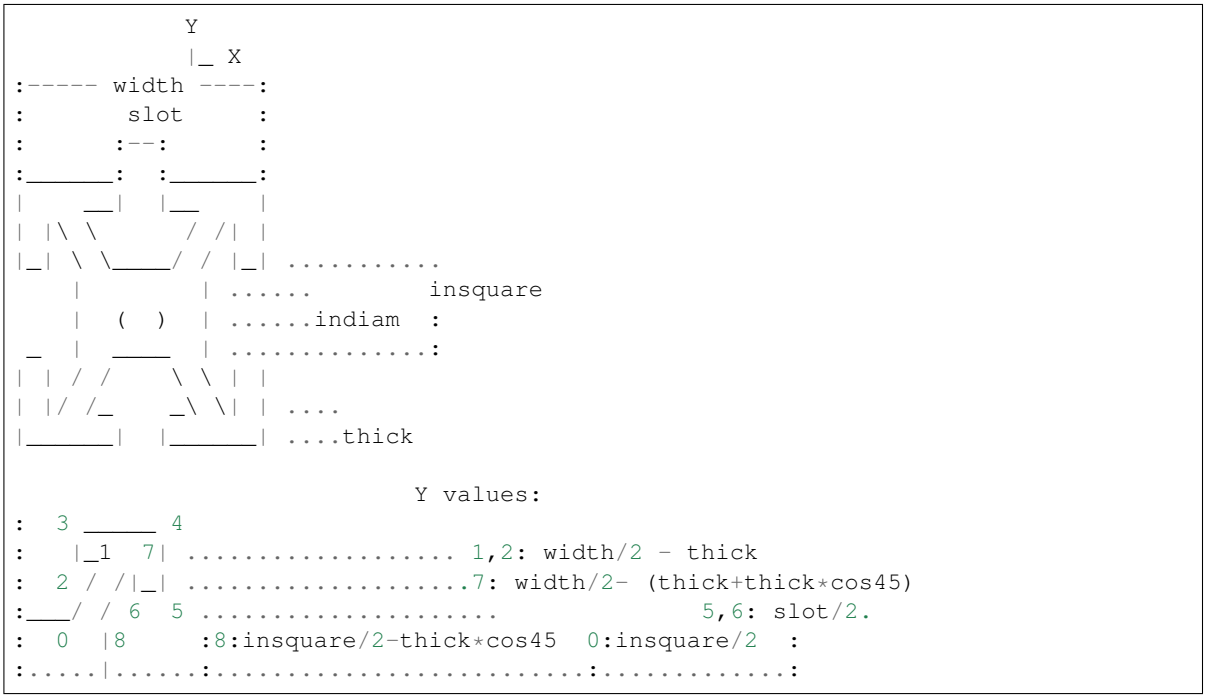
Return type FreeCAD Object

fcfun.**add_fcobj**(*shp*, *name*, *doc=None*)

Just creates a freeCAD object of the shape, just to save one line

fcfun.**aluprof_vec**(*width*, *thick*, *slot*, *insquare*)

Creates a wire (shape), that is an approximation of a generic alum profile extrusion



Parameters

- **width** (*float*) – The total width of the profile, it is a square
- **thick** (*float*) – The thickness of the side
- **slot** (*float*) – The width of the rail
- **insquare** (*float*) – The width of the inner square
- **indiam** (*float*) – The diameter of the inner hole

Returns The points of the aluminum profile positive quadrant

Return type Vector

fcfun.**calc_desp_ncen**(*Length*, *Width*, *Height*, *vec1*, *vec2*, *cx=False*, *cy=False*, *cz=False*, *H_extr=False*)

Similar to calc_rot, but calculates de displacement, when we don't want to have all of the dimensions centered First vector original direction (x,y,z) is (1,0,0) Second vector original direction (x,y,z) is (0,0,-1) The arguments vec1, vec2 are tuples (x,y,z) but they may be also FreeCAD.Vectors

```
    Z           . Y       length (x) = 1
:   _     .           width  (y) = 2
: /   /|              height (z) = 3
:/_  /  |
|   |   |             vec1 original (before rotation) = VX
|   | /               vec2 original (before rotation) = -VZ
|__|/..............X


Example after rotation and change position

Z          . Y       length (x) = 3
:   _____.           width  (y) = 2
: /     /|           height (z) = 1
:/___  //                  vec1 = VZ
|_____|/.............X    vec2 = VX

So we have to move X its original heith (3), otherwise it would
 be on the negative side, like this

     Z
     :     . Y       length (x) = 3
    _:__.            width  (y) = 2
  /   : /|           height (z) = 1
 /___://                  vec1 = VZ
|_____|/.............X    vec2 = VX

the picture is wrong, because originally it is centered, that's
why the position is moved only half of the dimension. But the concept
is valid
```

Parameters

- **vec1** (*tuples*) – Have to be on the axis: x, -x, y, -y, z, -z

```
vec1 can be (0,0,0): it means that it doesn't matter how it is␣
↪rotated
```

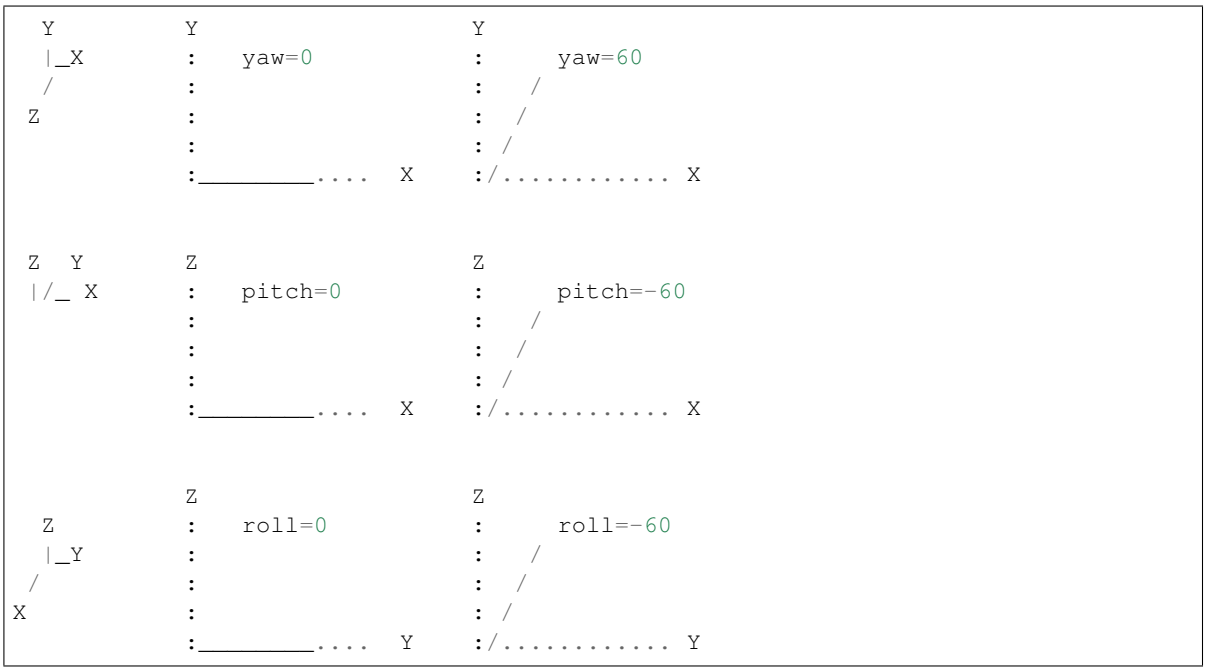
- **vec2** (*tuples*) – Have to be on the axis: x, -x, y, -y, z, -z
- **Length** (*float*) – Original dimension on X
- **Width** (*float*) – Original dimension on Y
- **Height** (*float*) – Original dimension on Z
- **cx** (*boolean*) – Position centered or not
- **cy** (*boolean*) – Position centered or not
- **cz** (*boolean*) – Position centered or not

Returns Vector of the displacement

Return type FreeCAD.Vector

fcfun.**calc_rot**(*vec1*, *vec2*)

Having an object with an orientation defined by 2 vectors the vectors a tuples, nor FreeCAD.Vectors use the wrapper fc_calc_rot to have FreeCAD.Vector arguments First vector original direction (x,y,z) is (1,0,0) Second vector original direction (x,y,z) is (0,0,-1) we want to rotate the object in an ortoghonal direction. The vectors will be in -90, 180, or 90 degrees. this function returns the Rotation given by yaw, pitch and roll In case vec1 is (0,0,0), means that it doesn't matter that vector. Yaw is the rotation of Z axis. Positive Yaw is like screwing up



Parameters

- **vec1** (*tuples*) – Direction
- **vec2** (*tuples*) – Direction

Returns

Return type FreeCAD.Rotation

fcfun.**calc_rot_z**(*v_refz*, *v_refx*)

Calculates de rotation like calc_rot. However uses a different origin axis. calc_rot uses: vec1 original direction (x,y,z) is (0,0,1) vec2 original direction (x,y,z) is (1,0,0) So it makes a change of axis before calling calc_rot

Parameters

- **v_refz** (*tuple or FreeCAD.Vector*) – Vector indicating the rotation from (0,0,1) to v_refz
- **v_refx** (*tuple or FreeCAD.Vector*) – Vector indicating the rotation from (1,0,0) to v_refx

Returns

Return type FreeCAD.Rotation

fcfun.**edgeonaxis**(*edge*, *axis*)

It tells if an edge is on an axis

Parameters

- **edge** (*Edge*) – A FreeCAD edge, with its vertices
- **axis** (*str*) – 'x', '-x', 'y', '-y', 'z', '-z'

Returns True: edge on an axis False: edge not on an axis

Return type boolean

fcfun.**equ**(*x*, *y*)
Compare numbers that are the same but not exactly the same

fcfun.**fc_calc_desp_ncen**(*Length*, *Width*, *Height*, *fc_vec1*, *fc_vec2*, *cx=False*, *cy=False*, *cz=False*, *H_extr=False*)
Same as calc_desp_ncen but using FreeCAD.Vectors arguments

fcfun.**fc_calc_rot**(*fc_vec1*, *fc_vec2*)
Same as calc_rot but using FreeCAD.Vectors arguments

fcfun.**fc_isonbase**(*fcv*)
Just tells if a vector has 2 of the coordinates zero so it is on just a base vector

fcfun.**fc_isparal**(*fc1*, *fc2*)
Return 1 if fc1 and fc2 are parallel (colinear), 0 if they are not

Parameters

- **fc1** (*FreeCAD.Vector*) – Firs vector
- **fc2** (*FreeCAD.Vector*) – Second vector

Returns

- ** 1 if fc1 and fc2 are parallel*
- ** 0 if they are not*

fcfun.**fc_isparal_nrm**(*fc1*, *fc2*)
Very similar to fc_isparal, but in this case the arguments are normalized so, less operations to do. return 1 if fc1 and fc2 are parallel (colinear), 0 if they are not

Parameters

- **fc1** (*FreeCAD.Vector*) – Firs vector
- **fc2** (*FreeCAD.Vector*) – Second vector

Returns

- ** 1 if fc1 and fc2 are parallel*
- ** 0 if they are not*

fcfun.**fc_isperp**(*fc1*, *fc2*)
Return 1 if fc1 and fc2 are perpendicular, 0 if they are not

Parameters

- **fc1** (*FreeCAD.Vector*) – Firs vector
- **fc2** (*FreeCAD.Vector*) – Second vector

Returns

- ** 1 if fc1 and fc2 are perpendicular*
- ** 0 if they are not*

fcfun.**fillet_len**(*box*, *e_len*, *radius*, *name*)
Make a new object with fillet

Parameters

- **box** (*TopoShape*) – Original shape we want to fillet
- **e_len** (*float*) – Length of the edges that we want to fillet
- **radius** (*float*) – Radius of the fillet
- **name** (*str*) – Name of the shape we want to create

Returns FreeCAD Object with fillet made

Return type FreeCAD Object

fcfun.**filletchamfer**(*fco*, *e_len*, *name*, *fillet=1*, *radius=1*, *axis='x'*, *xpos_chk=0*, *ypos_chk=0*, *zpos_chk=0*, *xpos=0*, *ypos=0*, *zpos=0*)

Fillet or chamfer edges of a certain length, on a certain axis and a certain coordinate

Parameters

- **fco** (*FreeCAD Object*) – Original FreeCAD object we want to fillet or chamfer
- **fillet** (*int*) –
  - 1 if we are doing a fillet
  - 0 if it is a chamfer
- **e_len** (*float*) – Length of the edges that we want to fillet or chamfer if e_len == 0, chamfer/fillet any length
- **radius** (*float*) – Radius of the fillet or chamfer
- **axis** (*str*) – Axis where the fillet will be
- **xpos_chk** (*int*) – If the X position will be checked
- **ypos_chk** (*int*) – If the Y position will be checked
- **zpos_chk** (*int*) – If the Z position will be checked
- **xpos** (*float*) – The X position
- **ypos** (*float*) – The Y position
- **zpos** (*float*) – The Z position
- **name** (*str*) – Name of the fco we want to create

Notes

If axis = 'x', x_pos_check will not make sense

Returns FreeCAD Object with fillet/chamfer made

Return type FreeCAD Object

fcfun.**fuseshplist**(*shp_list*)

Since multifuse methods needs to be done by a shape and a list, and usually I have a list that I want to fuse, I make this function to save the inconvenience of doing every time what I will do here Fuse multiFuse

fcfun.**get_bolt_bearing_sep**(*bolt_d*, *hasnut*, *lbearing_r*, *bsep=0*)

same as get_bolt_end_sep, but when there is a bearing. If there is a bearing, there will be more space because the nut is at the bottom or top, and the widest side is on the middle

```
                    lbearing_r
           rad     ..+...
          ..+..   :      :
 _______     :   :__:______:_
```

(continues on next page)

(continued from previous page)

```
|       |_  :  _|       .* :
|         | : |     .*    : this is the bearing section (circunference)
|         | : |    (      :
|        _| : |_   : *.    :
|_______|   :   |__:____*_:
            :   :  :       :
            :   :.bsep     :
            :              :
            :.bolt_b_sep..:
```

Parameters

- **bolt_d** (*int*) – Diameter of the bolt: 3, 4, ... for M3, M4,...
- **hasnut** (*int*) –
  - 1: if there is a nut
  - 0: if there is not a nut, so just the bolt head (smaller)
- **lbearing_r** (*float*) – Radius of the linear bearing
- **bsep** (*float*) – Separation from the outside of the nut to the end of bearing default value 0mm

Returns Minimum separation between the center of the bolt and the bearing

Return type float

fcfun.**get_bolt_end_sep**(*bolt_d*, *hasnut*, *sep=2.0*)

Calculate Bolt separation

Calculates know how much separation is needed for a bolt The bolt (din912) head diameter is usually smaller than the nut (din934) The nut max value is given by its 2*apotheme (S) (wrench size) so its max diameter is 2A x cos(30)

Example of nut and bolt head sizes:

	din912	din938	
	D	S(max)	D(max)
M3	5.5	5.5	6,35
M4	7.0	7.0	8,08
M5	8.5	8.0	9,24
M6	10.0	10.0	11,55

Therefore, if there is a nut, the nut will be used to calculate the separation

```
 _____      :     _______
|      |_   :   _|
|        |  : |
|        |  : |
|       _|  : |_
|_____|     :   |______
:      :    :
:..,..:.,.:
: sep  rad:
:          :
:....,....:
  bolt_sep
```

Parameters

- **bolt_d** (*int*) – Diameter of the bolt: 3, 4, ... for M3, M4,...
- **hasnut** (*int*) –
  - 1: if there is a nut
  - 0: if there is not a nut, so just the bolt head (smaller)
- **sep** (*float*) –

  Separation from the outside of the nut to the end, if empty, default value 2mm

Returns Minimum separation between the center of the bolt and the end

Return type float

fcfun.**get_fc_perpend1**(*fcv*)

gets a 'random' perpendicular FreeCAD.Vector

Parameters **fcv** (*FreeCAD.Vector*) – Vector from which to get perpendicular vector

Returns Random perpendicular vector

Return type FreeCAD.Vector

fcfun.**get_fclist_4perp2_fcvec**(*fcvec*)

Gets a list of 4 FreeCAD.Vector perpendicular to one base vector fcvec can only be: * (1,0,0) * (0,1,0) * (0,0,1) * (-1,0,0) * (0,-1,0) * (0,0,-1)

For example:

```
from (1,0,0) -> (0,1,0), (0,0,1), (0,-1,0), (0,0,-1)
```

Parameters **fcvec** (*vector*) – (1,0,0), (0,1,0), (0,0,1), (-1,0,0), (0,-1,0), (0,0,-1)

Returns List of FreeCAD.Vector

Return type list

fcfun.**get_fclist_4perp2_vecname**(*vecname*)

Gets a list of 4 FreeCAD.Vector perpendicular to one vecname different from get_fclist_4perp_vecname For example:

```
from 'x' -> (0,1,1), (0,-1,1), (0,-1,-1), (0,1,-1)
```

Parameters **vecname** (*str*) – 'x', '-x', 'y', '-y', 'z', '-z'

Returns List of FreeCAD.Vector

Return type list

fcfun.**get_fclist_4perp_fcvec**(*fcvec*)

Gets a list of 4 FreeCAD.Vector perpendicular to one base vector fcvec can only be: * (1,0,0) * (0,1,0) * (0,0,1) * (-1,0,0) * (0,-1,0) * (0,0,-1)

For example:

```
from (1,0,0) -> (0,1,0), (0,0,1), (0,-1,0), (0,0,-1)
```

Parameters **fcvec** (*vector*) – (1,0,0), (0,1,0), (0,0,1), (-1,0,0), (0,-1,0), (0,0,-1)

Returns List of FreeCAD.Vector

Return type list

fcfun.**get_fclist_4perp_vecname**(*vecname*)
Gets a list of 4 FreCAD.Vector perpendicular to one vecname for example:

```
from 'x' -> (0,1,0), (0,0,1), (0,-1,0), (0,0,-1)
```

Parameters **vecname** (*str*) – 'x', '-x', 'y', '-y', 'z', '-z'

Returns List of FreeCAD.Vector

Return type list

fcfun.**get_fcvectup**(*tup*)
Gets the FreeCAD.Vector of a tuple
Parameters **tup** (*tuple*) – Tuple of 3 elements

Returns FreeCAD.Vector of a tuple

Return type FreeCAD.Vector

fcfun.**get_nameofbasevec**(*fcvec*)
From a base vector either: (1,0,0), (0,1,0), (0,0,1), (-1,0,0), (0,-1,0), (0,0,-1) Gets its name: 'x', 'y',....
Returns Vector name

Return type str

fcfun.**get_positive_vecname**(*vecname*)
It just get 'x' when vecname is 'x' or '-x', and the same for the others, because some functions receive only positive base vector
Parameters **vecname** (*str*) – 'x', '-x', 'y', '-y', 'z', '-z'

Returns Vector name

Return type str

fcfun.**get_rot**(*v1*, *v2*)
Calculate the rotation from v1 to v2 the difference with previous versions, such fc_calc_rot, calc_rot, calc_rot is that it is for any vector direction. The difference with DraftVecUtils.getRotation is that getRotation doesn't work for vectors with 180 degrees.

Notes

MAYBE IT IS NOT NECESSARY, just use FreeCAD.Rotation rotation.Axis, math.degrees(rotation.Angle)
Parameters

- **v1** (*FreeCAD.Vector*) – Vector to calculate the rotation
- **v2** (*FreeCAD.Vector*) – Vector to calculate the rotation

Returns Tuple representing a quaternion rotation between v2 and v1

Return type FreeCAD.Rotation

fcfun.**get_tangent_2circles**(*center1_pt*, *center2_pt*, *rad1*, *rad2*, *axis_n*, *axis_side=None*)
Returns a list of lists (matrix) with the 2 tangent points for each of the 2 tangent lines

```
(difficult to draw in using ASCII text)


                axis_p
                   :
                T2 :          axis_side
```

(continues on next page)

(continued from previous page)

```
                  .  * r1         /  ------------
              .         .         /               :
         .         .     .       /                :
T1   .         .            . r2-r1 (r_diff)      + r2*sin(beta)
   *      .                .                      :
 r1   . alpha           beta                      :
    *----------------------*   ---- axis_c (axis going thru centers)
  C1                  :      C2
   ::                 :     :
   ::                 :.....:
   ::                   +   :
   ::           r2*cos(beta):
   ::                       :
   ::.......................:
   ::           +
   ::    C1_C2_d (hypotenuse)
   ::
   ::
   r1*cos(beta)

    alpha = atan(r_diff/C1_C2_d)
    beta = 90 - alpha

    tangent points along axis_c and axis_p

    T2_c = C2_c - r2 * cos(beta)
    T2_p = C2_p - r2 * sin(beta)

    T1_c = C1_c - r1 * cos(beta)
    T1_p = C1_p - r1 * sin(beta)
```

Parameters

- **center1_pt** (*FreeCAD.Vector*) – Center of the circle 1
- **center2_pt** (*FreeCAD.Vector*) – Center of the circle 2
- **rad1** (*float*) – Radius of the circle 1
- **rad2** (*float*) – Radius of the circle 2
- **axis_n** (*FreeCAD.Vector*) – Direction of the normal of the circle
- **axis_side** (*FreeCAD.Vector*) – Direction to the side of the tangent line, if not given, it will return the 2 points of both lines The 2 tangent lines will be at each side of axis_c. The smaller than 90 degree angle between axis_side and the 2 possible axis_p

Returns

- ** If axis_side is given –*
  - Returns a list of lists (matrix)
    - Element [0][0] is the point tangent to circle 1 at side axis_side
    - Element [0][1] is the point tangent to circle 2 at side axis_side
    - Element [1][0] is the point tangent to circle 1 at opposite side of direction of axis_side
    - Element [1][1] is the point tangent to circle 2 at opposite side of direction of axis_side

- * *If axis_side is not given, the order of the list of the lines is* – arbitrary
- * *If there is an error it will return 0*

Notes

Interesting variables

axis_p (FreeCAD.Vecrtor)
: Vector of the circle plane, perpendicular to axis_d. It can have to possible directions. If parameter axis_side is defined, it will have the direction that has less than 90 degrees related to axis_side

`fcfun.`**`get_tangent_circle_pt`**(*ext_pt*, *center_pt*, *rad*, *axis_n*, *axis_side=None*)
Get the point of the tangent to the circle

```
    (difficult to draw in using ASCII text)

       external point
        :     tangent point 1
        *---  _
         \  /   \
          \(     ) circle
    tangent \ _ /
     point 2

The 3 points: center(C), ext_pt(E) and tangent_pt(T) form a
rectangle triangle

                axis_p
                   :    axis_side
                   :      /
                   : <90 /
                   T    /
                .  *.  /
            .      90 .  rad
        .                .
    . alpha          beta .
 *-----------------------*   ---- axis_c (axis going thru centers)
E                   :      C
 :                  :     :
 :..................:     :
 :       +                 :
 :   axis_c_ET_d           :
 :                         :
 :.........................:
             +
      EC_d (hypotenuse)
```

Parameters

- **`ext_pt`** (*`FreeCAD.Vector`*) – External point
- **`center_pt`** (*`FreeCAD.Vector`*) – Center of the circle
- **`rad`** (*`float`*) – Radius of the circle
- **`axis_n`** (*`FreeCAD.Vector`*) – Direction of the normal of the circle
- **`axis_side`** (*`FreeCAD.Vector`*) – Direction to the side of the tangent point, if not given, it will return both points The 2 tangent points will be at each side of axis_c. The

smaller than 90 degree angle between axis_side and the 2 possible axis_p

Returns

- *If axis_side is not given* – returns a list with the 2 points that each point forms a line tangent to the circle. The 2 lines are defined by one of each point and the external point.
- *If axis_side is given* – Only returns a point (FreeCAD.Vector) with the tangent point defined by the direction of axis_side
- *If there is an error it will return 0*

Notes

Interesting Parameters

axis_p (FreeCAD.Vector)
: Vector of the circle plane, perpendicular to axis_d. It can have to possible directions. If parameter axis_side is defined, it will have the direction that has less than 90 degrees related to axis_side

`fcfun.`**`get_vecname_perpend1`**(*vecname*)
Gets a perpendicular vecname

Parameters **`vec`** (*`str`*) – 'x', '-x', 'y', '-y', 'z', '-z'

Returns Perpendicular vector name

Return type str

`fcfun.`**`get_vecname_perpend2`**(*vecname*)
Gets the other perpendicular vecname (see get_vecname_perpend)

Parameters **`vec`** (*`str`*) – 'x', '-x', 'y', '-y', 'z', '-z'

Returns Perpendicular vector name

Return type str

`fcfun.`**`getfcvecofname`**(*axis*)
Returns the FreeCAD.Vecor of the vector name given

`fcfun.`**`getvecofname`**(*axis*)
Get axis name renunrs the vector

`fcfun.`**`regpolygon_dir_vecl`**(*n_sides*, *radius*, *fc_normal*, *fc_verx1*, *pos*)
Similar to regpolygon_vecl but in any place and direction of the space calculates the vertices of a regular polygon. Returns a list of FreeCAD vectors with the vertices. The first vertex will be repeated at the end, this is needed to close the wire to make the shape The polygon will have the center in pos. The normal on fc_normal The direction of the first vertex on fc_verx_1

Parameters

- **`n_sides`** (*`int`*) – Number of sides of the polygon
- **`radius`** (*`float`*) – Circumradius of the polygon
- **`fc_normal`** (*`FreeCAD.Vector`*) – Direction of the normal
- **`fc_verx1`** (*`FreeCAD.Vector`*) – Direction of the first vertex
- **`pos`** (*`FreeCAD.Vector`*) – Position of the center

Returns List of FreeCAD.Vector of the vertices

Return type List

fcfun.**regpolygon_vecl**(*n_sides*, *radius*, *x_angle=0*)

Calculates the vertices of a regular polygon. Returns a list of FreeCAD vectors with the vertices. The first vertex will be repeated at the end, this is needed to close the wire to make the shape The polygon will be on axis XY (z=0).

Parameters

- **n_sides** (*int*) – Number of sides of the polygon
- **radius** (*float*) – Circumradius of the polygon
- **x_angle** (*float*) – If zero, the first vertex will be on axis x (y=0) if x_angle != 0, it will rotated some angle

Returns List of FreeCAD.Vector of the vertices

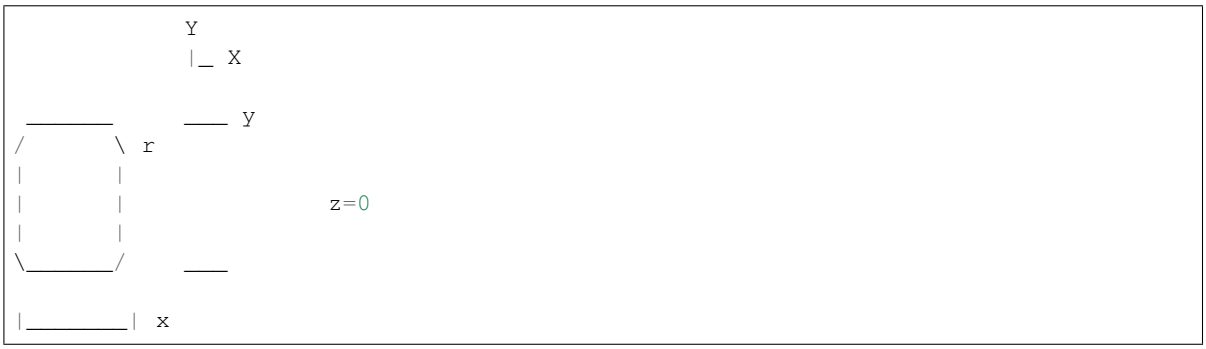
Return type List

fcfun.**rotateview**(*axisX=1.0*, *axisY=0.0*, *axisZ=0.0*, *angle=45.0*)

Rotate the camara

fcfun.**shpRndRectWire**(*x=1*, *y=1*, *r=0.5*, *zpos=0*)

Creates a wire (shape), that is a rectangle with rounded edges. if r== 0, it will be a rectangle The wire will be centered



Parameters

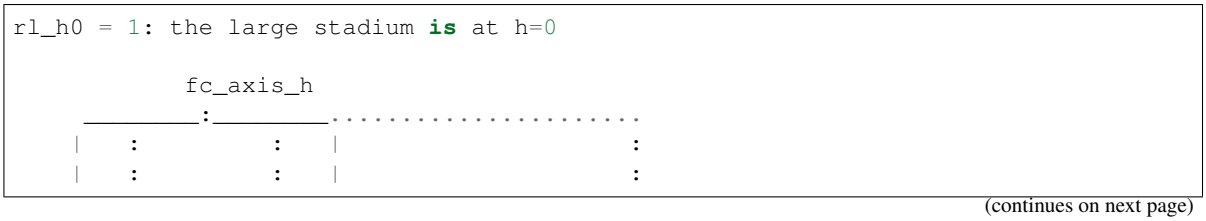
- **x** (*float*) – Dimension of the base, on the X axis
- **y** (*float*) – Dimension of the height, on the Y axis
- **r** (*float*) – Radius of the rouned edge.
- **zpos** (*float*) – Position on the Z axis

Returns FreeCAD Wire of a rounded edges rectangle

Return type Shape Wire

fcfun.**shp_2stadium_dir**(*length*, *r_s*, *r_l*, *h_tot*, *h_rl*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_l=FreeCAD.Vector*, *ref_l=1*, *rl_h0=1*, *xtr_h=0*, *xtr_nh=0*, *pos=FreeCAD.Vector*)

Makes to concentric stadiums, useful for making rails for bolts the length is the same for both. Changes the radius and the height The smaller radius will have the largest length



(continues on next page)

(continued from previous page)

```
 ___|    :          :    |___ ........         + h_tot
|        :          :        |         : h_rl  :
|________:____*_____:________|.........:........: ......> fc_axis_l
:    :   :          :
:...:.+.:..........:
:     r_s:    length
:        :
:        :
:.r_l...:


rl_h0 = 0 : the large stadium is at the end of h

             fc_axis_h
 _____________:_____________.............
|        :          :        |     : h_rl  :
|___     :          :     ___|....:        :
    |    :          :    |                 :+ h_tot
:   |    :          :    |                 :
:   |____:____*_____:____|.................: ......> fc_axis_l
:   :    :          :
:...:.+.:..........:
:     r_s:    length
:        :
:        :
:.r_l...:
```

on the axis_h, the h_rl stadium can be at the reference, or at the end of the reference (rl_h0 =0):

```
ref_l points:

    fc_axis_s
      :
      :__________
     /           \
    ( 2     1     ) -------> fc_axis_l
     \___________/
```

Parameters

- **length** (*float*) – Length of the parallels, from one semicircle center to the other
- **r_s** (*float*) – Smaller radius of the semicircles
- **r_l** (*float*) – Larger radius of the semicircles
- **h_tot** (*float*) – Total height
- **h_rl** (*float*) – Height of the larger radius stadium
- **fc_axis_h** (*FreeCAD.Vector*) – Vector on the direction of the height
- **fc_axis_l** (*FreeCAD.Vector*) – Vector on the direction of the parallels,
- **ref_l** (*int*) – Reference (zero) of the fc_axis_l
  - 1: reference on the center (makes axis_s symmetrical)
  - 2: reference at one of the semicircle centers (point 2) the other circle center will be on the direction of fc_axis_l

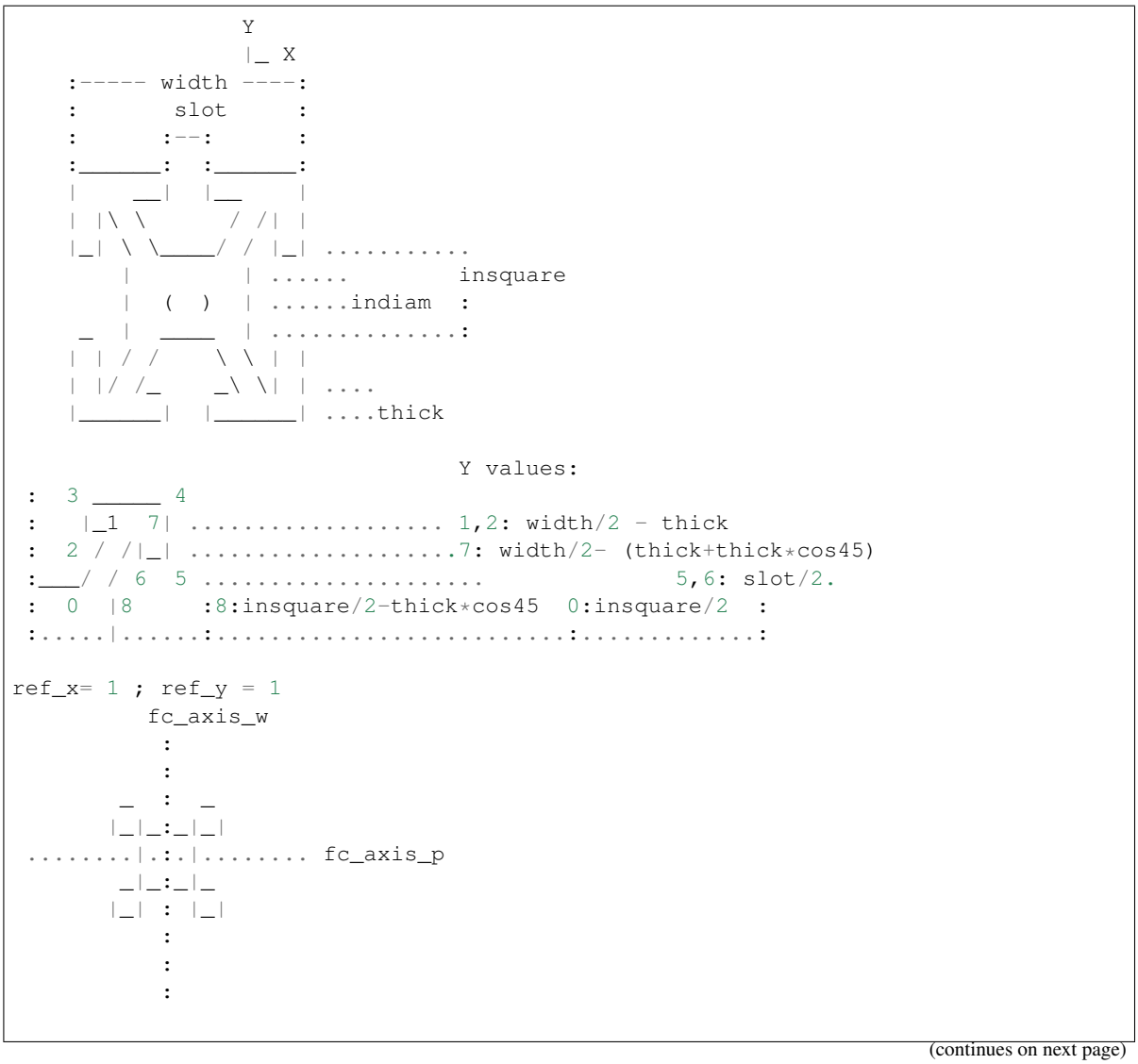
- **rl_h0** (*int*) –
  - 1: if the larger radius stadium is at the beginning of the axis_h
  - 0: at the end of axis_h
- **xtr_h** (*float*) – If >0 it will be that extra height on the direction of fc_axis_h
- **xtr_nh** (*float*) – If >0 it will be that extra height on the opositve direction of fc_axis_h
- **xtr_nh** –
- **pos** (*FreeCAD.Vector*) – Position of the reference

Returns FreeCAD Shape of a two stadiums

Return type Shape

fcfun.**shp_aluwire_dir**(*width*, *thick*, *slot*, *insquare*, *fc_axis_x=FreeCAD.Vector*, *fc_axis_y=FreeCAD.Vector*, *ref_x=1*, *ref_y=1*, *pos=FreeCAD.Vector*)

Creates a wire (shape), that is an approximation of a generic alum profile extrusion. Creates it in any position an any direction



(continues on next page)

(continued from previous page)

```
ref_x= 2 ; ref_y = 1 (the zero of axis_y is at the center)
                     (the zero of axis_x is at one side)

           fc_axis_y
           :
           :
           :
           :_     _
           |_|___|_|
 ..........:.|...|........ fc_axis_x
           :_|___|_
           |_|   |_|
           :
           :
           :
```

Parameters

- **width** (*float*) – Total width of the profile, it is a square
- **thick** (*float*) – Thickness of the side
- **slot** (*float*) – Width of the rail
- **insquare** (*float*) – Width of the inner square
- **indiam** (*float*) – Diameter of the inner hole
- **fc_axis_x** (*int*) – Is a generic X axis, can be any
  - 1: reference (zero) at the center
  - 2: reference (zero) at the side, the other end side will be on the direction of fc_axis_x
- **fc_axis_y** (*int*) – Is a generic Y axis, can be any perpendicular to fc_axis_y
  - 1: reference (zero) at the center
  - 2: reference (zero) at the side, the other end side will be on the direction of fc_axis_y
- **ref_x** (*float*) – Reference (zero) on the fc_axis_x
- **ref_y** (*float*) – Reference (zero) on the fc_axis_1
- **pos** (*FreeCAD.Vector*) – Position of the center

Returns FreeCAD Shape Wire of a aluminium profile

Return type Shape Wire

fcfun.**shp_belt_dir**(*center_sep*, *rad1*, *rad2*, *height*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_l=FreeCAD.Vector*, *ref_l=1*, *ref_h=1*, *xtr_h=0*, *xtr_nh=0*, *pos=FreeCAD.Vector*)

Makes a shape of 2 tangent circles (like a belt joining 2 circles). check shp_belt_wire_dir

Parameters

- **center_sep** (*float*) – Separation of the circle centers
- **rad1** (*float*) – Radius of the first circle, on the opposite direction of fc_axis_l
- **rad2** (*float*) – Radius of the second circle, on the direction of fc_axis_l
- **height** (*float*) – Height of the shape

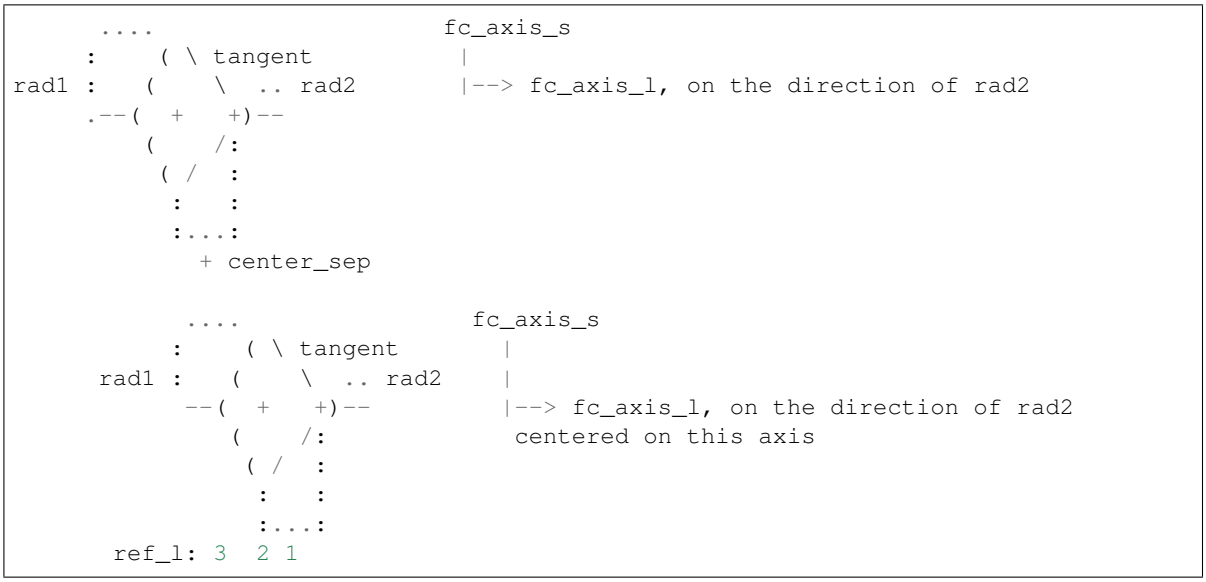
- **fc_axis_l** (*FreeCAD.Vector*) – Vector on the direction circle centers, pointing to rad2
- **fc_axis_h** (*FreeCAD.Vector*) – Vector on the height direction
- **ref_l** (*int*) – Reference (zero) of the fc_axis_l
  - 1: reference on the center
  - 2: reference at rad1 semicircle centers (point 2) the other circle center will be on the direction of fc_axis_l
  - 3: reference at the end of rad1 circle the other end will be on the direction of fc_axis_l
- **ref_h** (*int*) –
  - 1: reference is at the center of the height
  - 2: reference is at the bottom
- **xtr_h** (*float*) – If >0 it will be that extra height on the direction of fc_axis_h
- **xtr_nh** (*float*) – If >0 it will be that extra height on the opositve direction of fc_axis_h
- **pos** (*FreeCAD.Vector*) – Position of the reference

Returns FreeCAD Shape of a belt

Return type Shape

fcfun.**shp_belt_wire_dir**(*center_sep*, *rad1*, *rad2*, *fc_axis_l=FreeCAD.Vector*, *fc_axis_s=FreeCAD.Vector*, *ref_l=1*, *ref_s=1*, *pos=FreeCAD.Vector*)

Makes a shape of a wire with 2 circles and exterior tangent lines check here It is not easy to draw it well rad1 and rad2 can be exchanged, rad1 doesn't have to be larger:



Parameters

- **center_sep** (*float*) – Separation of the circle centers
- **rad1** (*float*) – Radius of the firs circle, on the opposite direction of fc_axis_l

- **fc_axis_l** (*FreeCAD.Vector*) – Vector on the direction circle centers, pointing to rad2
- **fc_axis_s** (*FreeCAD.Vector*) – Vector on the direction perpendicular to fc_axis_l, on the plane of the wire
- **ref_l** (*int*) – Reference (zero) of the fc_axis_l
  - 1: reference on the center
  - 2: reference at one of the semicircle centers (point 2) the other circle center will be on the direction of fc_axis_l
  - 3: reference at the end of rad1 circle the other end will be on the direction of fc_axis_l
- **pos** (*FreeCAD.Vector*) – Position of the reference

Returns FreeCAD Wire of a belt

Return type Shape Wire

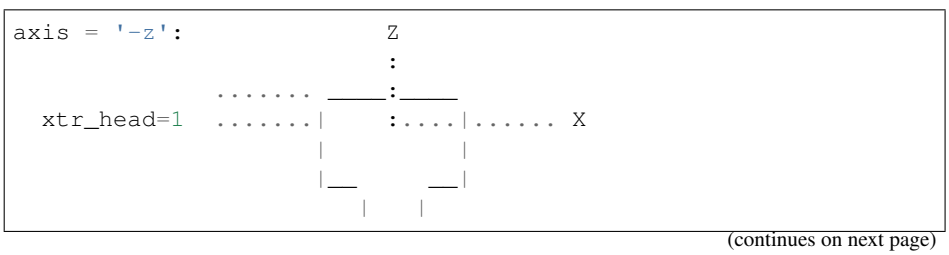
fcfun.**shp_bolt**(*r_shank*, *l_bolt*, *r_head*, *l_head*, *hex_head=0*, *xtr_head=1*, *xtr_shank=1*, *support=1*, *axis='z'*, *hex_ref='x'*, *hex_rot_angle=0*, *pos=FreeCAD.Vector*)

Similar to addBolt, but creates a shape instead of a FreeCAD Object Creates a shape of the bolt shank and head or the nut Tolerances have to be included if you want it for making a hole

It is referenced at the end of the head

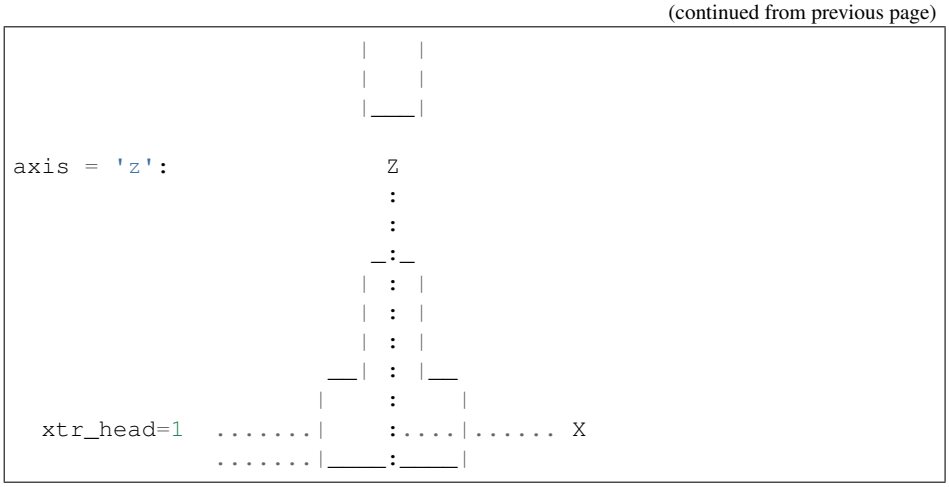
Parameters

- **r_shank** (*float*) – Radius of the shank (tolerance included)
- **l_bolt** (*float*) – Total length of the bolt: head & shank
- **r_head** (*float*) – Radius of the head (tolerance included)
- **l_head** (*float*) – Length of the head
- **hex_head** (*int*) – Indicates if the head is hexagonal or rounded
  - 1: hexagonal
  - 0: rounded
- **h_layer3d** (*float*) – Height of the layer for printing, if 0, means that the support is not needed
- **xtr_head** (*int*) – 1 if you want 1 mm on the head to avoid cutting on the same plane pieces after making cuts (boolean difference)
- **xtr_shank** (*int*) – 1 if you want 1 mm at the opposite side of the head to avoid cutting on the same plane pieces after making cuts (boolean difference)
- **support** (*int*) – 1 if you want to include a triangle between the shank and the head to support the shank and not building the head on the air using kcomp.LAYER3D_H
- **axis** (*str*) – 'x', '-x', 'y', '-y', 'z', '-z': Defines the orientation. For example:



(continues on next page)

(continued from previous page)



- **hex_ref** (*str*) – In case of a hexagonal head, this will indicate the axis that the first vertex of the nut will point hex_ref has to be perpendicular to axis, if not, it will be changed
- **hex_rot_angle** (*float*) – Angle in degrees. In case of a hexagonal head, it will indicate the angle of rotation of the hexagon referenced to hex_ref.
- **pos** (*FreeCAD.Vector*) – Position of the center of the head of the bolt

Returns FreeCAD Shape of a bolt

Return type Shape

fcfun.**shp_bolt_dir**(*r_shank*, *l_bolt*, *r_head*, *l_head*, *hex_head=0*, *xtr_head=1*, *xtr_shank=1*, *support=1*, *fc_normal=FreeCAD.Vector*, *fc_verx1=FreeCAD.Vector*, *pos_n=0*, *pos=FreeCAD.Vector*)

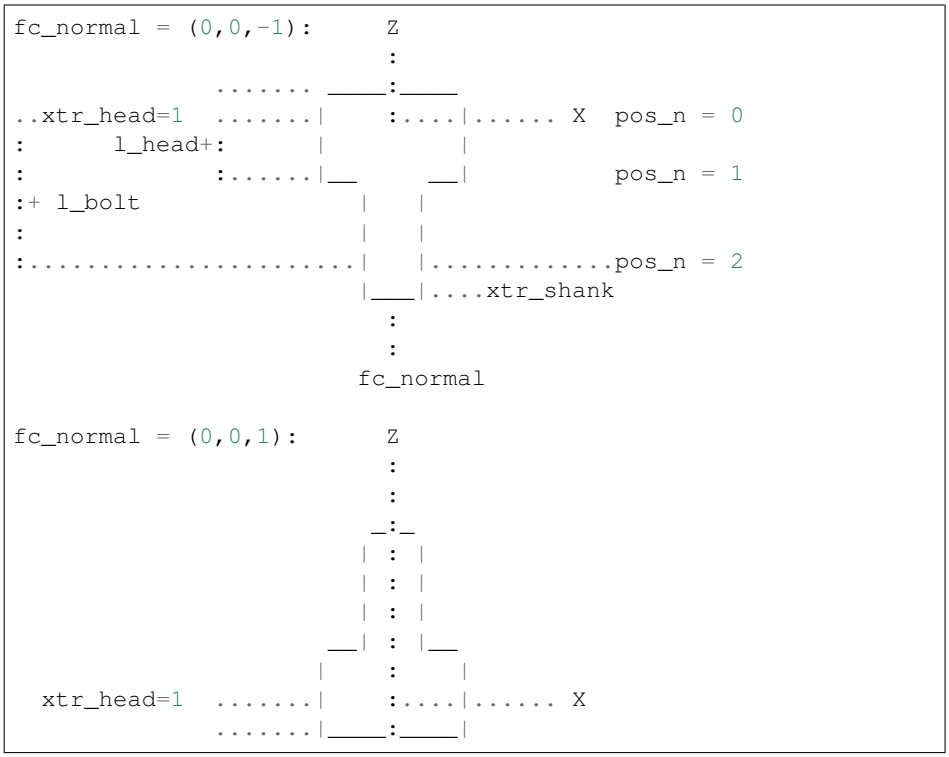
Similar to shp_bolt, but it can be done in any direction Creates a shape, not a of a FreeCAD Object Creates a shape of the bolt shank and head or the nut Tolerances have to be included if you want it for making a hole

It is referenced at the end of the head

Parameters

- **r_shank** (*float*) – Radius of the shank (tolerance included)
- **l_bolt** (*float*) – Total length of the bolt: head & shank
- **r_head** (*float*) – Radius of the head (tolerance included)
- **l_head** (*float*) – Length of the head
- **hex_head** (*int*) – Indicates if the head is hexagonal or rounded
  - 1: hexagonal
  - 0: rounded
- **h_layer3d** (*float*) – Height of the layer for printing, if 0, means that the support is not needed
- **xtr_head** (*int*) – 1 if you want 1 mm on the head to avoid cutting on the same plane pieces after making cuts (boolean difference)
- **xtr_shank** (*int*) – 1 if you want 1 mm at the opposite side of the head to avoid cutting on the same plane pieces after making cuts (boolean difference)
- **support** (*int*) – 1 if you want to include a triangle between the shank and the head to support the shank and not building the head on the air using kcomp.LAYER3D_H

- **fc_normal** (*FreeCAD.Vector*) – Defines the orientation. For example:



- **fc_verx1** (*FreeCAD.Vector*) – In case of a hexagonal head, this will indicate the axis that the first vertex of the nut will point it has to be perpendicular to fc_normal,
- **pos_n** (*int*) – Location of pos along the normal, at the cylinder center
  - 0: at the top of the head (excluding xtr_head)
  - 1: at the union of the head and the shank
  - 2: at the end of the shank (excluding xtr_shank)
- **pos** (*FreeCAD.Vector*) – Position of the center of the head of the bolt

Returns FreeCAD Shape of a bolt

Return type Shape

fcfun.**shp_boltnut_dir_hole**(*r_shank*, *l_bolt*, *r_head*, *l_head*, *r_nut*, *l_nut*, *hex_head=0*, *xtr_head=1*, *xtr_nut=1*, *supp_head=1*, *supp_nut=1*, *head-start=1*, *fc_normal=FreeCAD.Vector*, *fc_verx1=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Similar to addBoltNut_hole, but in any direction and creates shapes, not FreeCAD Objects Creates the hole for the bolt shank, the head and the nut. The bolt head will be at the bottom, and the nut will be on top Tolerances have to be already included in the arguments values

Parameters

- **r_shank** (*float*) – Radius of the shank (tolerance included)
- **l_bolt** (*float*) – Total length of the bolt: head & shank
- **r_head** (*float*) – Radius of the head (tolerance included)
- **l_head** (*float*) – Length of the head
- **r_nut** (*float*) – Radius of the nut (tolerance included)

- **l_nut** (*float*) – Length of the nut. It doesn't have to be the length of the nut but how long you want the nut to be inserted
- **hex_head** (*int*) – Indicates if the head is hexagonal or rounded
  - 1: hexagonal
  - 0: rounded
- **xtr_head** (*int*) – 1 if you want an extra size on the side of the head to avoid cutting on the same plane pieces after making differences
- **xtr_nut** (*int*) – 1 if you want an extra size on the side of the nut to avoid cutting on the same plane pieces after making differences
- **supp_head** (*int*) – 1 if you want to include a triangle between the shank and the head to support the shank and not building the head on the air using kcomp.LAYER3D_H
- **supp_nut** (*int*) – 1 if you want to include a triangle between the shank and the nut to support the shank and not building the nut on the air using kcomp.LAYER3D_H
- **headstart** (*int*) – If on pos you have the head, or if you have it on the other end
- **fc_normal** (*FreeCAD.Vector*) – Direction of the bolt
- **fc_verx1** (*FreeCAD.Vector*) – Direction of the first vertex of the hexagonal nut. Perpendicular to fc_normal. If not perpendicular or zero, means that it doesn't matter which direction and the function will obtain one perpendicular direction
- **pos** (*FreeCAD.Vector*) – Position of the head (if headstart) or of the nut

Returns FreeCAD Object of a Nut Hole

Return type FreeCAD Object

fcfun.**shp_box_dir**(*box_w*, *box_d*, *box_h*, *fc_axis_w=FreeCAD.Vector*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_d=FreeCAD.Vector*, *cw=1*, *cd=1*, *ch=1*, *pos=FreeCAD.Vector*)

Makes a shape of a box given its 3 dimensions: width, depth and height and the direction of the height and depth dimensions. The position of the box is given and also if the position is given by a corner or its center

```
     _________
    |\         \
    | \         \
    |  \_________\
     \ |         |
      \|_________|


Example of not centered on origin

  Z=fc_axis_h       . Y = fc_axis_d
        :           .
        :   ____________
        :  /:    .    / |
        : / :   .    /   | h
        :/_________/    |
        |    :.....|...|3
        |   / 4    |   /
        |  /       |  /  d
        |/_________|/.....................X
        1          2
              w
```

(continues on next page)

(continued from previous page)

```
Example of centered on origin

  Z=fc_axis_h                Y  = fc_axis_d
                :            .
              ___________    .
            /:  :     / |.
           / :  :    /  .| h
         /__:_____/.   |
        |    :.....|...|3
        |   / 4 :..|../.......................X
        |  /       | /  d
        |/_________|/
        1          2
              w
```

Parameters

- **box_w** (*float*) – Width of the box
- **box_d** (*float*) – Depth of the box
- **box_h** (*float*) – Height of the box
- **fc_axis_w** (*FreeCAD.Vector*) – Direction of the width
- **fc_axis_d** (*FreeCAD.Vector*) – Direction of the depth
- **fc_axis_h** (*FreeCAD.Vector*) – Direction of the height
- **cw** (*int*) –
  - 1: the width dimension is centered
  - 0: it is not centered
- **cd** (*int*) –
  - 1: the depth dimension is centered
  - 0: it is not centered
- **ch** (*int*) –
  - 1: the height dimension is centered
  - 0: it is not centered
- **pos** (*FreeCAD.Vector*) – Position of the box, it can be the center one corner, or a point centered in the dimensions given by cw, cd, ch

Returns Shape of a box

Return type TopoShape

fcfun.**shp_box_dir_xtr**(*box_w*, *box_d*, *box_h*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_d=FreeCAD.Vector*, *fc_axis_w=FreeCAD.Vector*, *cw=1*, *cd=1*, *ch=1*, *xtr_h=0*, *xtr_nh=0*, *xtr_d=0*, *xtr_nd=0*, *xtr_w=0*, *xtr_nw=0*, *pos=FreeCAD.Vector*)

Makes a shape of a box given its 3 dimensions: width, depth and height and the direction of the height and depth dimensions. The position of the box is given and also if the position is given by a corner or its center. Extra mm to make cuts

```
      _________
     |\         \
     | \         \
     |  \_________\
      \ |         |
       \|_________|

Example of not centered on origin

 Z=fc_axis_h       . Y = fc_axis_d
       :           .
       :    ____________
       :   /:    .    / |
       :  / :   .    /  | h
       : /__:_____/    |
       |    :.....|...|3
       |   / 4    |   /
       |  /       |  /  d
       | /________|/.....................X
       1          2
             w

Example of centered on origin

 Z=fc_axis_h                 Y   = fc_axis_d
                :             .
              ____________    .
            /:   :     /  |.
           / :   :    /  .| h
          /__:_____/.    |
         |    :.....|...|3
         |   / 4 :..|../.......................X
         |  /       |  /  d
         | /________|/
         1          2
               w
```

Parameters

- **box_w** (*float*) – Width of the box
- **box_d** (*float*) – Depth of the box
- **box_h** (*float*) – Heiht of the box
- **fc_axis_h** (*FreeCAD.Vector*) – Direction of the height
- **fc_axis_d** (*FreeCAD.Vector*) – Direction of the depth
- **fc_axis_w** (*FreeCAD.Vector*) – Direction of the width
- **cw** (*int*) –
  - 1 the width dimension is centered
  - 0 it is not centered
- **cd** (*int*) –
  - 1 the depth dimension is centered
  - 0 it is not centered

- **ch** (*int*) –
  - 1 the height dimension is centered
  - 0 it is not centered
- **xtr_w** (*float*) – If an extra mm will be added, the number will determine the size useful to make cuts
- **xtr_nw** (*float*) – If an extra mm will be added, the number will determine the size useful to make cuts
- **xtr_d** (*float*) – If an extra mm will be added, the number will determine the size useful to make cuts
- **xtr_nd** (*float*) – If an extra mm will be added, the number will determine the size useful to make cuts
- **xtr_h** (*float*) – If an extra mm will be added, the number will determine the size useful to make cuts
- **xtr_nh** (*float*) – If an extra mm will be added, the number will determine the size useful to make cuts
- **pos** (*FreeCAD.Vector*) – Position of the box, it can be the center one corner, or a point centered in the dimensions given by cw, cd, ch

Returns FreeCAD.Object with a shape of a box

Return type TopoShape

Notes

fc_axis_w not necessary, unless cw=0, then it indicates the direction of w, it has to be perpendicular to the previous

fcfun.**shp_box_rot**(*box_w*, *box_d*, *box_h*, *axis_w='x'*, *axis_nh='-z'*, *cw=1*, *cd=1*, *ch=1*)

Makes a box with width, depth, height and then rotation will be referred to *axis_w = (1,0,0)* and *axis_nh = (0,0,-1)*. Can be centered on any of the dimensions.

Parameters

- **box_w** (*float*) – The width is X
- **box_d** (*float*) – The depth is Y
- **box_h** (*float*) – The height is Z
- **cw** (*int*) – If *1* is centered
- **cd** (*int*) – If *1* is centered
- **ch** (*int*) – If *1* is centered
- **axis_w** (*str*) – Can be: *x, -x, y, -y, z, -z*
- **axis_nh** (*str*) – Can be: *x, -x, y, -y, z, -z*

Notes

Check if it makes sense to have this small function

fcfun.**shp_boxcen**(*x*, *y*, *z*, *cx=False*, *cy=False*, *cz=False*, *pos=FreeCAD.Vector*)

Adds a shape of box, referenced on the specified axis, with its Placement and Rotation at zero. So it can be referenced absolutely from its given position

Parameters

- **x** (*float*) – Length
- **y** (*float*) – Width
- **z** (*float*) – Height
- **name** (*str*) – Object Name
- **cx** (*boolean*) – Center in the length or not
- **cy** (*boolean*) – Center in the or width not
- **cz** (*boolean*) – Center in the height or not
- **pos** (*FreeCAD.Vector*) – Placement

Returns Shape of a box

Return type TopoShape

fcfun.**shp_boxcenchmf**(*x*, *y*, *z*, *chmfrad*, *fx=False*, *fy=False*, *fz=True*, *cx=False*, *cy=False*, *cz=False*, *pos=FreeCAD.Vector*)

Same as shp_boxcen but with a chamfered dimension

Parameters

- **x** (*float*) – Length
- **y** (*float*) – Width
- **z** (*float*) – Height
- **fillrad** (*float*) – Fillet size
- **fx** (*boolean*) – Fillet in x dimension
- **fy** (*boolean*) – Fillet in y dimension
- **fz** (*boolean*) – Fillet in z dimension
- **cx** (*boolean*) – Center in the length or not
- **cy** (*boolean*) – Center in the or width not
- **cz** (*boolean*) – Center in the height or not
- **pos** (*FreeCAD.Vector*) – Placement

Returns Shape of a box

Return type TopoShape

fcfun.**shp_boxcenfill**(*x*, *y*, *z*, *fillrad*, *fx=False*, *fy=False*, *fz=True*, *cx=False*, *cy=False*, *cz=False*, *pos=FreeCAD.Vector*)

Same as shp_boxcen but with a filleted dimension

Parameters

- **x** (*float*) – Length
- **y** (*float*) – Width

- **z** (*float*) – Height
- **fillrad** (*float*) – Fillet size
- **fx** (*boolean*) – Fillet in x dimension
- **fy** (*boolean*) – Fillet in y dimension
- **fz** (*boolean*) – Fillet in z dimension
- **cx** (*boolean*) – Center in the length or not
- **cy** (*boolean*) – Center in the or width not
- **cz** (*boolean*) – Center in the height or not
- **pos** (*FreeCAD.Vector*) – Placement

Returns Shape of a box

Return type TopoShape

fcfun.**shp_boxcenxtr**(*x*, *y*, *z*, *cx=False*, *cy=False*, *cz=False*, *xtr_nx=0*, *xtr_x=0*, *xtr_ny=0*, *xtr_y=0*, *xtr_nz=0*, *xtr_z=0*, *pos=FreeCAD.Vector*)

The same as shp_boxcen, but when it is used to cut. So sometimes it is useful to leave an extra 1mm on some sides to avoid making cuts sharing faces. The extra part is added but not influences on the reference

```
       | Y     cy=1, xtr_ny=1
       |
 1     |
 __________
|  |       |
|  |       |
|  |       |
|  |       |
|_|________|
```

Parameters

- **x** (*float*) – Length
- **y** (*float*) – Width
- **z** (*float*) – Height
- **cx** (*int*) – Center in the length or not
- **cy** (*int*) – Center in the or width not
- **cz** (*int*) – Center in the height or not
- **xtr_x** (*float*) – Extra mm to add in positive axis of length
- **xtr_nx** (*float*) – Extra mm to add in negative axis of length
- **xtr_y** (*float*) – Extra mm to add in positive axis of width
- **xtr_ny** (*float*) – Extra mm to add in negative axis of width
- **xtr_z** (*float*) – Extra mm to add in positive axis of height
- **xtr_nz** (*float*) – Extra mm to add in negative axis of height
- **pos** (*FreeCAD.Vector*) – Placement

Returns Shape of a box

Return type TopoShape

fcfun.**shp_boxdir_fillchmfplane**(*box_w*, *box_d*, *box_h*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *axis_h=FreeCAD.Vector*, *cw=1*, *cd=1*, *ch=1*, *xtr_d=0*, *xtr_nd=0*, *xtr_w=0*, *xtr_nw=0*, *xtr_h=0*, *xtr_nh=0*, *fillet=1*, *radius=1.0*, *plane_fill=FreeCAD.Vector*, *both_planes=1*, *edge_dir=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Creates a box shape (cuboid) along 3 axis.

The shape will be filleted or chamfered on the edges of the plane defined by the plane_fill vector. If both_planes == 1, both faces will be filleted/chamfered if both_planes == 0, only the face that plane_fill is normal and goes outwards if edge_dir has an edge direction, only those edges in that direction will be filleted/chamfered

```
Example of not centered on origin: cd=0, cw=0, ch=0

     axis_h         . axis_d
        :          .
        :   ___________....
        :  /:    .    / |  :
        : / :   .    /  |  :h
        :/_________/    |  :
        |    :.....|...|..:..
        |  /       |  /     .
        | /        | /     . d
        |/_________|/..................> axis_w
        :          :
        :....w.....:


Example of centered on origin: cd=1, cw=1, ch=1

            axis_h           axis_d
               :           .
              ___________    .
            /:  :     / |.
           / :  :    / .| h
          /__:_____/.   |
         |    :.....|...|
         |  /    :..|../..............> axis_w
         | /        | /
         |/_________|/

               w

 Example of parameter both_planes and edge_dir

 if both_planes == 1:
     if edge_dir == V0:
         edges_to_chamfer = [1,2,3,4,5,6,7,8]
     elif edge_dir == axis_w:
         edges_to_chamfer = [2,4,6,8]
     elif edge_dir == axis_d:
         edges_to_chamfer = [1,3,5,7]
 elif both_planes == 0:
     if edge_dir == V0:
         edges_to_chamfer = [1,2,3,4]
     elif edge_dir == axis_w:
         edges_to_chamfer = [2,4]
     elif edge_dir == axis_d:
```

(continues on next page)

(continued from previous page)

```
        edges_to_chamfer = [1,3]

axis_h=plane_fill
       :
       :    ____2_____
       :  /:       / |
       : 1 :      3  |
       :/_____4__/   |
       |    :...6.|...|
       |  /       |  /
       | 5        | 7
       |/___8____|/.................> axis_w

Another example of parameter both_planes

if both_planes == 1:
    edges_to_chamfer = [1,2,3,4,5,6,7,8]
elif both_planes == 0:
    edges_to_chamfer = [5,6,7,8]

    axis_h
       :
       :    ____2_____
       :  /:       / |
       : 1 :      3  |
       :/_____4__/   |
       |    :...6.|...|
       |  /       |  /
       | 5        | 7
       |/___8____|/.................> axis_w
       :
       :
       :
      V
     plane_fill = axis_h.negative()
```

Parameters

- **box_d** (*positive float*) – Depth of the box
- **box_w** (*positive float*) – Width of the box
- **box_h** (*positive float*) – Height of the box
- **axis_d** (*FreeCAD.Vector*) – Depth vector of the coordinate system
- **axis_w** (*FreeCAD.Vector*) – Width vector of the coordinate system, can be V0 if centered and will be perpendicular to axis_d and axis_w
- **axis_h** (*FreeCAD.Vector*) – Height vector of the coordinate system
- **cw** (*int*) – 1: centered along axis_w
- **cd** (*int*) – 1: centered along axis_d
- **ch** (*int*) – 1: centered along axis_h
- **xtr_d** (*float, >= 0*) – Extra depth, if there is an extra depth along axis_d
- **xtr_nd** (*float, >= 0*) – Extra depth, if there is an extra depth along axis_d.negative

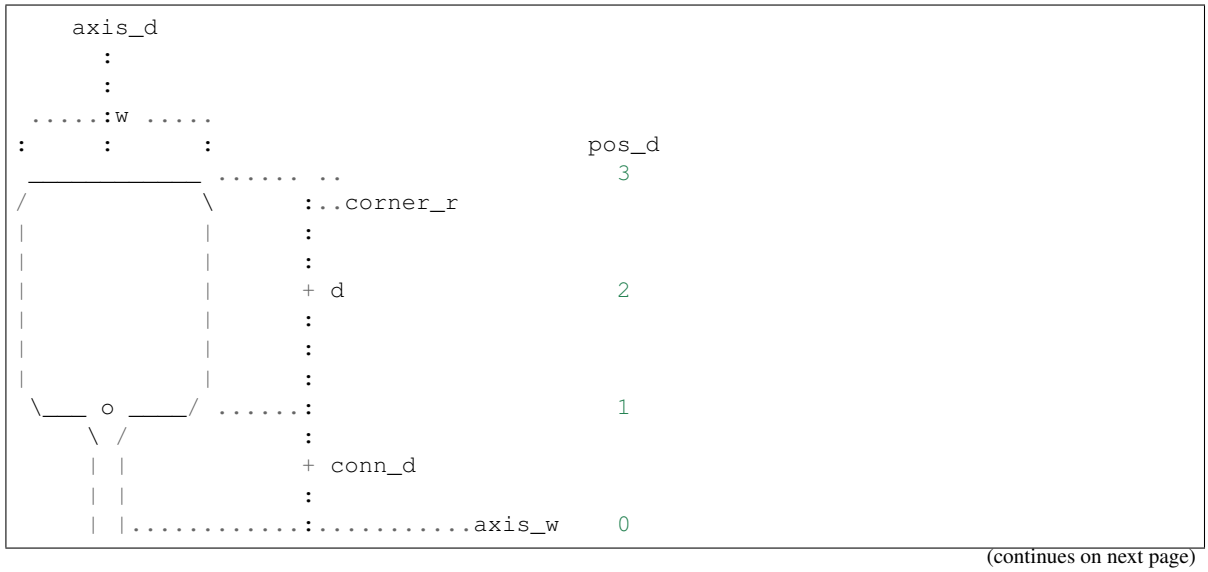
- **xtr_w** (*float, >= 0*) – Extra width, if there is an extra width along axis_w
- **xtr_nw** (*float, >= 0*) – Extra width, if there is an extra width along axis_w.negative
- **xtr_h** (*float, >= 0*) – Extra height, if there is an extra height along axis_h
- **xtr_nh** (*float, >= 0*) – Extra height, if there is an extra height along axis_h.negative
- **fillet** (*int*) –
  - 1: to fillet the edges
  - 0: to chamfer the edges
- **radius** (*float >= 0*) – radius of the fillet/chamfer
- **plane_fill** (*FreeCAD.Vector*) – Vector perpendicular to the face that is going to be filleted/chamfered
- **both_planes** (*int*) –
  - 0: fillet/chamfer only the edges on the face perpendicular to plane_fill and on the face that plane_fill goes outwards. See drawing
  - 1: fillet/chamfer the edges on both faces perpendicular to plane_fill
- **edge_dir** (*FreeCAD.Vector*) –
  - V0: fillet/chamfer all the edges of that/those faces
  - Axis: fillet/chamfer only the edges of that/those faces that are parallel to this axis
- **pos** (*FreeCAD.Vector*) – Position of the box

Returns Shape of the filleted/chamfered box

Return type TopoShape

fcfun.**shp_cableturn**(*d*, *w*, *thick_d*, *corner_r*, *conn_d*, *conn_sep*, *xtr_conn_d=0*, *closed=0*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos=FreeCAD.Vector*)

Creates a shape of an electrical cable turn, in any direction But it is a shape in FreeCAD See function wire_cableturn



(continues on next page)

(continued from previous page)

```
     : :
    conn_sep

1      0  pos_w

pos_o (orig) is at pos_d=0, pos_w=0, marked with o
```

Parameters

- **d** (*float*) – Depth/length of the turn
- **w** (*float*) – Width of the turn
- **thick_d** (*float*) – Diameter of the wire
- **corner_r** (*float*) – Radius of the corners
- **conn_d** (*float*) – Depth/length of the connector part
  - 0: there is no connecting wire
- **xtr_conn_d** (*float*) – If conn_d > 0, there can be and extra length of connector to make unions, it will not be counted as pos_d = 0 It will not work well if it is closed
- **conn_sep** (*float*) – Separation of the connectors
- **closed** (*boolean*) –
  - 0 : the ends are not closed
  - 1 : the ends are closed
- **axis_d** (*FreeCAD.Vector*) – Coordinate System Vector along the depth
- **axis_w** (*FreeCAD.Vector*) – Coordinate System Vector along the width
- **pos_d** (*int*) – Location of pos along the axis_d (0,1,2,3), see drawing
  - 0: reference at the beginning of the connector
  - 1: reference at the beginning of the turn, at the side of the connector
  - 2: reference at the middle of the turn
  - 3: reference at the end of the turn
- **pos_w** (*int*) – Location of pos along the axis_w (0,1), see drawing
  - 0: reference at the center of symmetry
  - 1: reference at the end of the turn
- **pos** (*FreeCAD.Vector*) – Position of the reference

Returns FreeCAD Shape of a electrical wire

Return type Shape

fcfun.**shp_cir_fillchmf**(*shp*, *circen_pos=FreeCAD.Vector*, *fillet=1*, *radius=1*)

Fillet or chamfer edges that is a circle, the shape has to be a cylinder

Parameters

- **shp** (*Shape*) – Original cylinder shape we want to fillet or chamfer
- **circen_pos** (*FreeCAD.Vector*) – Center of the circle
- **fillet** (*int*) –

- 1 if we are doing a fillet
- 0 if it is a chamfer

- **radius** (*float*) – Radius of the fillet or chamfer

Returns FreeCAD Shape with fillet/chamfer made

Return type Shape

fcfun.**shp_cyl**(*r*, *h*, *normal=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Same as addCylPos, but just creates the shape

Parameters

- **r** (*float*) – Radius,
- **h** (*float*) – Height
- **normal** (*FreeCAD.Vectot*) – FreeCAD.Vector pointing to the normal (if its module is not one, the height will be larger than h
- **pos** (*FreeCAD.Vector*) – Position of the cylinder

Returns FreeCAD Shape of a cylinder

Return type Shape

fcfun.**shp_cyl_gen**(*r*, *h*, *axis_h=FreeCAD.Vector*, *axis_ra=None*, *axis_rb=None*, *pos_h=0*, *pos_ra=0*, *pos_rb=0*, *xtr_top=0*, *xtr_bot=0*, *xtr_r=0*, *pos=FreeCAD.Vector*)

This is a generalization of shp_cylcenxtr. Makes a cylinder in any position and direction, with optional extra heights and radius, and various locations in the cylinder

```
pos_h = 1, pos_ra = 0, pos_rb = 0
pos at 1:
       axis_rb
         :
         :
        . .
      .     .
    (    o    ) ---- axis_ra       This o will be pos_o (origin)
      .     .
        . .

      axis_h
         :
         :
     .............
    :_____:_____:....: xtr_top
    |           |
    |           |
    |           |
    |           |
    |           |
    |           |
    |_____1_____|.............> axis_ra
    :.....o.....:.....: xtr_bot             This o will be pos_o

pos_h = 0, pos_ra = 1, pos_rb = 0
pos at x:

  axis_rb
    :
```

(continues on next page)

(continued from previous page)

```
    :
    :   . .
    : .     .
    x         ) ----> axis_ra
      .     .
        . .

      axis_h
         :
         :
     .............
    :____:____:....: xtr_top
    |         |
    |         |
    |         |
    x         |....>axis_ra
    |         |
    |         |
    |_________|.....
    :....o....:....: xtr_bot        This o will be pos_o

pos_h = 0, pos_ra = 1, pos_rb = 1
pos at x:

  axis_rb
    :
    :
    :   . .
    : .     .
    (         )
      .     .
    x   . .     ....> axis_ra

   axis_h
    :
    :
     .............
    :____:____:....: xtr_top
   ||         |
   ||         |
   ||         |
   |x         |....>axis_ra
   ||         |
   ||         |
   ||_________|.....
   ::....o....:....: xtr_bot
   :;
   xtr_r
```

Parameters

- **r** (*float*) – Radius of the cylinder
- **h** (*float*) – Height of the cylinder
- **axis_h** (*FreeCAD.Vector*) – Vector along the cylinder height
- **axis_ra** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h only make sense if pos_ra = 1. It can be None.

- **axis_rb** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h and axis_rb only make sense if pos_rb = 1 It can be None
- **pos_h** (*int*) – Location of pos along axis_h (0, 1)
  - 0: the cylinder pos is centered along its height
  - 1: the cylinder pos is at its base (not considering xtr_h)
- **pos_ra** (*int*) – Location of pos along axis_ra (0, 1)
  - 0: pos is at the circunference center
  - 1: pos is at the circunsference, on axis_ra, at r from the circle center (not at r + xtr_r)
- **pos_rb** (*int*) – Location of pos along axis_rb (0, 1)
  - 0: pos is at the circunference center
  - 1: pos is at the circunsference, on axis_rb, at r from the circle center (not at r + xtr_r)
- **xtr_top** (*float*) – Extra height on top, it is not taken under consideration when calculating the cylinder center along the height
- **xtr_bot** (*float*) – Extra height at the bottom, it is not taken under consideration when calculating the cylinder center along the height or the position of the base
- **xtr_r** (*float*) – Extra length of the radius, it is not taken under consideration when calculating pos_ra or pos_rb
- **pos** (*FreeCAD.Vector*) – Position of the cylinder, taking into account where the center is

Returns FreeCAD Shape of a cylinder

Return type Shape

fcfun.**shp_cylcenxtr**(*r*, *h*, *normal=FreeCAD.Vector*, *ch=1*, *xtr_top=0*, *xtr_bot=0*, *pos=FreeCAD.Vector*)

Add cylinder, can be centered on the position, and also can have an extra mm on top and bottom to make cuts

Parameters

- **r** (*float*) – Radius
- **h** (*float*) – Height
- **normal** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the normal
- **ch** (*int*) – Centered on the middle, of the height
- **xtr_top** (*float*) – Extra on top (but does not influence the centering)
- **xtr_bot** (*float*) – Extra on bottom (but does not influence the centering)
- **pos** (*FreeCAD.Vector*) – Position of the cylinder

Returns FreeCAD Shape of a cylinder

Return type Shape

fcfun.**shp_cylfilletchamfer**(*shp*, *fillet=1*, *radius=1*)

Fillet or chamfer all edges of a cylinder

Parameters

- **shp** (*Shape*) – Original cylinder shape we want to fillet or chamfer

- **fillet** (*int*) –
  - 1 if we are doing a fillet
  - 0 if it is a chamfer
- **radius** (*float*) – Radius of the fillet or chamfer

Returns FreeCAD Shape with fillet/chamfer made

Return type Shape

fcfun.**shp_cylhole**(*r_ext*, *r_int*, *h*, *axis='z'*, *h_disp=0.0*)

Same as addCylHole, but just a shape

Add cylinder, with inner hole:

Parameters

- **r_ext** (*float*) – External radius,
- **r_int** (*float*) – Internal radius,
- **h** (*float*) – Height
- **axis** (*str*) – 'x', 'y' or 'z'
  - 'x' will along the x axis
  - 'y' will along the y axis
  - 'z' will be vertical
- **h_disp** (*int*) – Displacement on the height.
  - if 0, the base of the cylinder will be on the plane
  - if -h/2: the plane will be cutting h/2

Returns FreeCAD Shape of a cylinder with hole

Return type Shape

fcfun.**shp_cylhole_arc**(*r_out*, *r_in*, *h*, *axis_h=FreeCAD.Vector*, *axis_ra=None*, *axis_rb=None*, *end_angle=360*, *pos_h=0*, *pos_ra=0*, *pos_rb=0*, *xtr_top=0*, *xtr_bot=0*, *xtr_r_out=0*, *xtr_r_in=0*, *pos=FreeCAD.Vector*)

This is similar to make shp_cylhole_gen but not for a whole, just an arc. I don't know how where makeCircle starts its startangle and end angle That is why I use this way

Makes a hollow cylinder in any position and direction, with optional extra heights, and inner and outer radius, and various locations in the cylinder

```
pos_h = 1, pos_ra = 0, pos_rb = 0
pos at 1:
       axis_rb
         :
         :
        . .
      . . . .
    ( (  0  ) ) ---- axis_ra
      . . . .
        . .

      axis_h
         :
         :
```

(continues on next page)

(continued from previous page)

```
      ..............
    :____:____:....: xtr_top
    | :     : |
    | :     : |
    | :     : |
    | :  0  : |     0: pos would be at 0, if pos_h == 0
    | :     : |
    | :     : |
    |_:__1__:_|....>axis_ra
    :.:..o..:.:....: xtr_bot        This o will be pos_o (orig)
    : :  :
    : :..:
    :  + :
    :r_in:
    :    :
    :....:
      +
     r_out


Values for pos_ra  (similar to pos_rb along it axis)


      axis_h
         :
         :
      ..............
    :____:____:....: xtr_top
    | :     : |
    | :     : |
    | :     : |
    2 1  0  : |....>axis_ra    (if pos_h == 0)
    | :     : |
    | :     : |
    |_:_____:_|.....
    :.:..o..:.:....: xtr_bot        This o will be pos_o (orig)
    : :  :
    : :..:
    :  + :
    :r_in:
    :    :
    :....:
      +
     r_out
```

Parameters

- **r_out** (*float*) – Radius of the outside cylinder
- **r_in** (*float*) – Radius of the inner hole of the cylinder
- **h** (*float*) – Height of the cylinder
- **axis_h** (*FreeCAD.Vector*) – Vector along the cylinder height
- **axis_ra** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h it is not necessary if pos_ra == 0 It can be None, but if None, axis_rb has to be None Defines the starting angle

- **axis_rb** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h and axis_ra it is not necessary if pos_ra == 0 It can be None
- **end_angle** (*float (in degrees)*) – Rotating from axis_ra in the direction determined by axis_h
- **pos_h** (*int*) – Location of pos along axis_h (0, 1)
  - 0: the cylinder pos is centered along its height, not considering xtr_top, xtr_bot
  - 1: the cylinder pos is at its base (not considering xtr_h)
- **pos_ra** (*int*) – Location of pos along axis_ra (0, 1)
  - 0: pos is at the circunference center
  - 1: pos is at the inner circunsference, on axis_ra, at r_in from the circle center (not at r_in + xtr_r_in)
  - 2: pos is at the outer circunsference, on axis_ra, at r_out from the circle center (not at r_out + xtr_r_out)
- **pos_rb** (*int*) – Location of pos along axis_ra (0, 1)
  - 0: pos is at the circunference center
  - 1: pos is at the inner circunsference, on axis_rb, at r_in from the circle center (not at r_in + xtr_r_in)
  - 2: pos is at the outer circunsference, on axis_rb, at r_out from the circle center (not at r_out + xtr_r_out)
- **xtr_top** (*float*) – Extra height on top, it is not taken under consideration when calculating the cylinder center along the height
- **xtr_bot** (*float*) – Extra height at the bottom, it is not taken under consideration when calculating the cylinder center along the height or the position of the base
- **xtr_r_in** (*float*) – Extra length of the inner radius (hollow cylinder), it is not taken under consideration when calculating pos_ra or pos_rb. It can be negative, so this inner radius would be smaller
- **xtr_r_out** (*float*) – Extra length of the outer radius it is not taken under consideration when calculating pos_ra or pos_rb. It can be negative, so this outer radius would be smaller
- **pos** (*FreeCAD.Vector*) – Position of the cylinder, taking into account where the center is

Returns FreeCAD Shape of a arc of the cylinder

Return type Shape

fcfun.**shp_cylhole_bolthole**(*r_out*, *r_in*, *h*, *n_bolt=4*, *d_bolt=0*, *r_bolt2cen=0*, *axis_h=FreeCAD.Vector*, *axis_ra=FreeCAD.Vector*, *axis_rb=None*, *bolt_axis_ra=1*, *pos_h=0*, *pos_ra=0*, *pos_rb=0*, *xtr_top=0*, *xtr_bot=0*, *xtr_r_out=0*, *xtr_r_in=0*, *pos=FreeCAD.Vector*)

This is a generalization of shp_cylholedir and shp_cylhole Makes a hollow cylinder in any position and direction, with optional extra heights, and inner and outer radius, and various locations in the cylinder

Also has a number of nbolt holes along a radius r_bolt2cen the bolts a equi spaced depending on the number

```
pos_h = 1, pos_ra = 0, pos_rb = 0
pos at 1:
       axis_rb
         :
         :
        . .     o: are n_bolt(4) holes
      .o. .o.
    ( (  0  ) ) ---- axis_ra
      .o. .o.
        . .

      axis_h
         :
         :
     ..............
    :____:____:....: xtr_top
    | :     : |
    | :     : |
    | :     : |
    | :  0  : |     0: pos would be at 0, if pos_h == 0
    | :     : |
    | :     : |
    |_:__1__:_|....>axis_ra
    :.:..o..:.:....: xtr_bot        This o will be pos_o (orig)
    : :  :
    : :..:
    :  + :
    :r_in:
    :    :
    :....:
      +
     r_out

Values for pos_ra  (similar to pos_rb along it axis)


        axis_h
           :
   d_bolt  :
     :.:............
    :_:_:__:__:_:....: xtr_top
    | : :     : : |
    | : :     : : |
    | : :     : : |
    3 2 1  0  : : |....>axis_ra    (if pos_h == 0)
    | : :     : : |
    | : :     : : |
    |_:_:_____:_:_|.....
    :.: :..o..:.:....: xtr_bot        This o will be pos_o (orig)
    : : :  :
    : : :..:
    : :  + :
    : :  r_in
    : :....:
    :   +
    :  r_bolt2cen:
    :    :
```

(continues on next page)

(continued from previous page)

```
:....:
  +
 r_out
```

Parameters

- **r_out** (*float*) – Radius of the outside cylinder
- **r_in** (*float*) – Radius of the inner hole of the cylinder
- **h** (*float*) – Height of the cylinder
- **n_bolt** (*int*) – Number of bolt holes, if zero no bolt holes
- **d_bolt** (*float*) – Diameter of the bolt holes
- **r_bolt2cen** (*float*) – Distance (radius) from the cylinder center to the bolt hole centers
- **bolt_axis_ra** (*int*) –
  - 1: the first bolt will be on axis ra
  - 0: the first bolt will be rotated half of the angle between to bolt holes -> centered on the side
- **axis_h** (*FreeCAD.Vector*) – Vector along the cylinder height
- **axis_ra** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h it is not necessary if pos_ra == 0 It can be None, but if None, axis_rb has to be None
- **axis_rb** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h and axis_ra it is not necessary if pos_ra == 0 It can be None
- **pos_h** (*int*) – Location of pos along axis_h (0, 1)
  - 0: the cylinder pos is centered along its height, not considering xtr_top, xtr_bot
  - 1: the cylinder pos is at its base (not considering xtr_h)
- **pos_ra** (*int*) – Location of pos along axis_ra (0, 1)
  - 0: pos is at the circunference center
  - 1: pos is at the inner circunsference, on axis_ra, at r_in from the circle center (not at r_in + xtr_r_in)
  - 2: pos is at the center of the bolt hole (one of them)
  - 3: pos is at the outer circunsference, on axis_ra, at r_out from the circle center (not at r_out + xtr_r_out)
- **pos_rb** (*int*) – Location of pos along axis_ra (0, 1)
  - 0: pos is at the circunference center
  - 1: pos is at the inner circunsference, on axis_rb, at r_in from the circle center (not at r_in + xtr_r_in)
  - 2: pos is at the center of the bolt hole (one of them)
  - 3: pos is at the outer circunsference, on axis_rb, at r_out from the circle center (not at r_out + xtr_r_out)

- **xtr_top** (*float*) – Extra height on top, it is not taken under consideration when calculating the cylinder center along the height
- **xtr_bot** (*float*) – Extra height at the bottom, it is not taken under consideration when calculating the cylinder center along the height or the position of the base
- **xtr_r_in** (*float*) – Extra length of the inner radius (hollow cylinder), it is not taken under consideration when calculating pos_ra or pos_rb. It can be negative, so this inner radius would be smaller
- **xtr_r_out** (*float*) – Extra length of the outer radius it is not taken under consideration when calculating pos_ra or pos_rb. It can be negative, so this outer radius would be smaller
- **pos** (*FreeCAD.Vector*) – Position of the cylinder, taking into account where the center is

Returns FreeCAD Shape of a cylinder with hole

Return type Shape

fcfun.**shp_cylhole_gen**(*r_out*, *r_in*, *h*, *axis_h=FreeCAD.Vector*, *axis_ra=None*, *axis_rb=None*, *pos_h=0*, *pos_ra=0*, *pos_rb=0*, *xtr_top=0*, *xtr_bot=0*, *xtr_r_out=0*, *xtr_r_in=0*, *pos=FreeCAD.Vector*)

This is a generalization of shp_cylholedir. Makes a hollow cylinder in any position and direction, with optional extra heights, and inner and outer radius, and various locations in the cylinder

```
pos_h = 1, pos_ra = 0, pos_rb = 0
pos at 1:
       axis_rb
         :
         :
        . .
      . . . .
    ( (  0  ) ) ---- axis_ra
      . . . .
        . .

      axis_h
         :
         :
     ..............
    :____:____:....: xtr_top
    | :      : |
    | :      : |
    | :      : |
    | :  0   : |      0: pos would be at 0, if pos_h == 0
    | :      : |
    | :      : |
    |_:__1__:_|....>axis_ra
    :.:..o..:.:....: xtr_bot        This o will be pos_o (orig)
    : :  :
    : :..:
    :  + :
    :r_in:
    :    :
    :....:
      +
     r_out
```

(continues on next page)

(continued from previous page)

```
Values for pos_ra  (similar to pos_rb along it axis)


      axis_h
         :
         :
      ..............
    :____:____:....: xtr_top
    | :     : |
    | :     : |
    | :     : |
    2 1  0  : |....>axis_ra    (if pos_h == 0)
    | :     : |
    | :     : |
    |_:_____:_|.....
    :.:..o..:.:....: xtr_bot        This o will be pos_o (orig)
    : :  :
    : :..:
    :  + :
    :r_in:
    :    :
    :....:
      +
     r_out
```

Parameters

- **r_out** (*float*) – Radius of the outside cylinder
- **r_in** (*float*) – Radius of the inner hole of the cylinder
- **h** (*float*) – Height of the cylinder
- **axis_h** (*FreeCAD.Vector*) – Vector along the cylinder height
- **axis_ra** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h it is not necessary if pos_ra == 0 It can be None, but if None, axis_rb has to be None
- **axis_rb** (*FreeCAD.Vector*) – Vector along the cylinder radius, a direction perpendicular to axis_h and axis_ra it is not necessary if pos_ra == 0 It can be None
- **pos_h** (*int*) – Location of pos along axis_h (0, 1)
  - 0: the cylinder pos is centered along its height, not considering xtr_top, xtr_bot
  - 1: the cylinder pos is at its base (not considering xtr_h)
- **pos_ra** (*int*) – Location of pos along axis_ra (0, 1)
  - 0: pos is at the circunference center
  - 1: pos is at the inner circunsference, on axis_ra, at r_in from the circle center (not at r_in + xtr_r_in)
  - 2: pos is at the outer circunsference, on axis_ra, at r_out from the circle center (not at r_out + xtr_r_out)
- **pos_rb** (*int*) – Location of pos along axis_ra (0, 1)
  - 0: pos is at the circunference center

  - 1: pos is at the inner circunsference, on axis_rb, at r_in from the circle center (not at r_in + xtr_r_in)
  - 2: pos is at the outer circunsference, on axis_rb, at r_out from the circle center (not at r_out + xtr_r_out)
- **xtr_top** (*float*) – Extra height on top, it is not taken under consideration when calculating the cylinder center along the height
- **xtr_bot** (*float*) – Extra height at the bottom, it is not taken under consideration when calculating the cylinder center along the height or the position of the base
- **xtr_r_in** (*float*) – Extra length of the inner radius (hollow cylinder), it is not taken under consideration when calculating pos_ra or pos_rb. It can be negative, so this inner radius would be smaller
- **xtr_r_out** (*float*) – Extra length of the outer radius it is not taken under consideration when calculating pos_ra or pos_rb. It can be negative, so this outer radius would be smaller
- **pos** (*FreeCAD.Vector*) – Position of the cylinder, taking into account where the center is

Returns FreeCAD Shape of a cylinder with hole

Return type Shape

fcfun.**shp_cylholedir**(*r_out*, *r_in*, *h*, *normal=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Same as addCylHolePos, but just a shape Same as shp_cylhole, but this one accepts any normal

Parameters

- **r_out** (*float*) – Outside radius
- **r_in** (*float*) – Inside radius
- **h** (*float*) – Height
- **normal** (*FreeCAD.Vector*) – FreeCAD.Vector pointing to the normal (if its module is not one, the height will be larger than h
- **pos** (*FreeCAD.Vector*) – Position of the cylinder

Returns FreeCAD Shape of a cylinder with hole

Return type Shape

fcfun.**shp_extrud_face**(*face*, *length*, *vec_extr_axis*, *centered=0*)

Extrudes a face on any plane

Parameters

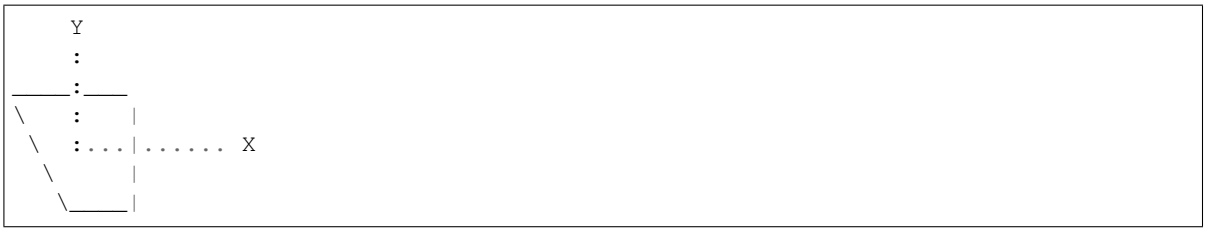
- **face** (*FreeCAD.Face*) – Face to be extruded.
- **length** (*float*) – Extrusion length
- **centered** (*int*) – 1 if the extrusion is centered (simetrical) 0 if it is not
- **vec_extr_axis** (*FreeCAD.Vector*) – Typically, it will be the same as vec_facenormal. by default, if it is 0, it will be equal to vec_facenormal It doesn't have to be on an axis, it can be diagonally

Returns FreeCAD Shape of the Face

Return type Shape

fcfun.**shp_extrud_face_rot**(*face*, *vec_facenormal*, *vec_edgx*, *length*, *centered=0*, *vec_extr_axis=0*)

Extrudes a face that is on plane XY, includes a rotation



Parameters

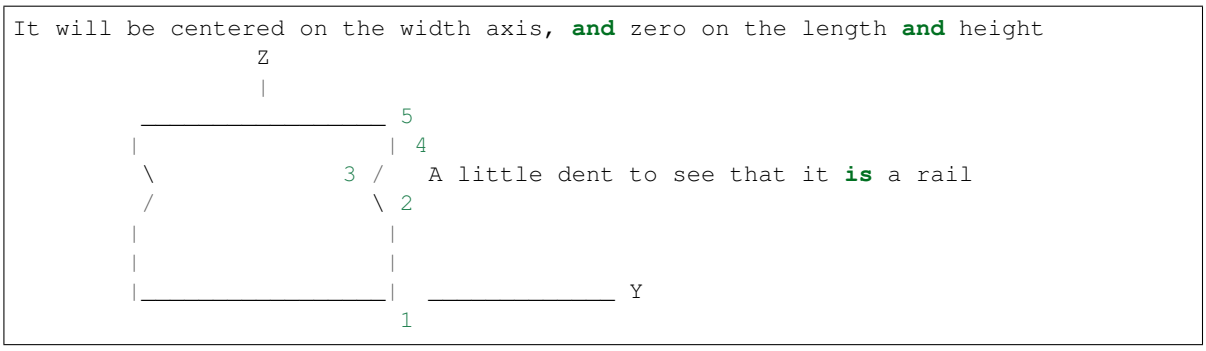
- **face** (*FreeCAD.Face*) – Face to be extruded. On plane XY
- **vec_facenormal** (*FreeCAD.Vector*) – Indicates where the normal of the face will point. The normal of the original face is VZ, but this function may rotate it depending on this argument It has to be on an axis: 'x', 'y', ..
- **vec_edgx** (*FreeCAD.Vector*) – Indicates where the edge X will be after the rotation It has to be on an axis: 'x', 'y', ..
- **length** (*float*) – Extrusion length
- **centered** (*int*) – 1 if the extrusion is centered (simetrical) 0 if it is not
- **vec_extr_axis** (*FreeCAD.Vector*) – Typically, it will be the same as vec_facenormal. by default, if it is 0, it will be equal to vec_facenormal It doesn't have to be on an axis, it can be diagonally

Returns FreeCAD Shape of a face

Return type Shape

fcfun.**shp_face_lgrail**(*rail_w*, *rail_h*, *axis_l='x'*, *axis_b='-z'*)

Adds a shape of the profile (face) of a linear guide rail, the dent is just rough, to be able to see that it is a profile



Parameters

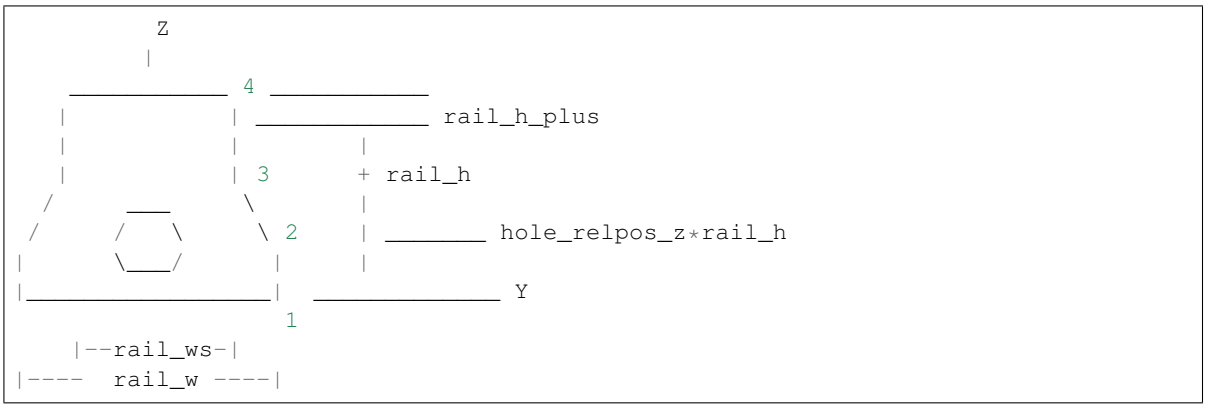
- **rail_w** (*float*) – Width of the rail
- **rail_h** (*float*) – Height of the rail
- **axis_l** (*str*) – Axis where the length of the rail is: 'x', 'y', 'z'
- **axis_b** (*str*) – Axis where the base of the rail is poingint: 'x', 'y', 'z', '-x', '-y', '-z',

Returns FreeCAD Shape Face of a rail

Return type Shape

fcfun.**shp_face_rail**(*rail_w*, *rail_ws*, *rail_h*, *rail_h_plus=0*, *offs_w=0*, *offs_h=0*, *axis_l='x'*, *axis_b='-z'*, *hole_d=0*, *hole_relpos_z=0.4*)

Adds a shape of the profile (face) of a rail



Parameters

- **rail_w** (*float*) – Width of the rail
- **rail_ws** (*float*) – Small width of the rail
- **rail_h** (*float*) – Height of the rail
- **rail_h_plus** (*float*) – Above the rail can be some height to attach, o whatever it is not included on rail_h
- **offs_w** (*float*) – Offset on the width, to make the hole
- **offs_h** (*float*) – Offset on the height, to make the hole
- **axis_l** (*str*) – The axis where the length of the rail is: 'x', 'y', 'z'
- **axis_b** (*str*) – The axis where the base of the rail is poingint: 'x', 'y', 'z', '-x', '-y', '-z', It will be centered on the width axis, and zero on the length and height
- **hole_d** (*float*) – Diameter of a hole inside the rail. To have a leadscrew
- **hole_relpos_z** (*float*) – Relative position of the center of the hole, relative to the height (the rail_h, not the total height (rail_h+rail_h_plus)

Returns FreeCAD Shape Face of a rail

Return type Shape

fcfun.**shp_filletchamfer**(*shp*, *e_len*, *fillet=1*, *radius=1*, *axis='x'*, *xpos_chk=0*, *ypos_chk=0*, *zpos_chk=0*, *xpos=0*, *ypos=0*, *zpos=0*)

Fillet or chamfer edges of a certain length, on a certain axis and a certain coordinate

Parameters

- **shp** (*Shape*) – Original shape we want to fillet or chamfer
- **fillet** (*int*) –
  - 1 if we are doing a fillet
  - 0 if it is a chamfer
- **e_len** (*float*) – Length of the edges that we want to fillet or chamfer if e_len == 0, chamfer/fillet any length
- **radius** (*float*) – Radius of the fillet or chamfer
- **axis** (*str*) – Axis where the fillet will be

- **xpos_chk** (*int*) – If the position will be checked.
- **ypos_chk** (*int*) – If the position will be checked.
- **zpos_chk** (*int*) – If the position will be checked.
- **xpos** (*float*) – The X position
- **ypos** (*float*) – The Y position
- **zpos** (*float*) – The Z position

Notes

If axis = 'x', x_pos_check will not make sense

Returns FreeCAD Shape with fillet/chamfer made

Return type Shape

fcfun.**shp_filletchamfer_dir**(*shp*, *fc_axis=FreeCAD.Vector*, *fillet=1*, *radius=1*)

Fillet or chamfer edges on a certain axis

Parameters

- **shp** (*Shape*) – Original shape we want to fillet or chamfer
- **fillet** (*int*) –
  - 1 if we are doing a fillet
  - 0 if it is a chamfer
- **radius** (*float*) – The radius of the fillet or chamfer
- **fc_axis** (*FreeCAD.Vector*) – Axis where the fillet will be

Returns FreeCAD Shape with fillet/chamfer made

Return type Shape

fcfun.**shp_filletchamfer_dirpt**(*shp*, *fc_axis=FreeCAD.Vector*, *fc_pt=FreeCAD.Vector*, *fillet=1*, *radius=1*)

Fillet or chamfer edges on a certain axis and a point contained in that axis

Parameters

- **shp** (*Shape*) – Original shape we want to fillet or chamfer
- **fc_axis** (*FreeCAD.Vector*) – Axis where the fillet will be
- **fc_pt** (*FreeCAD.Vector*) – Placement of the point
- **fillet** (*int*) –
  - 1 if we are doing a fillet
  - 0 if it is a chamfer
- **radius** (*float*) – Radius of the fillet or chamfer

Returns FreeCAD Shape with fillet/chamfer made

Return type Shape

fcfun.**shp_filletchamfer_dirpts**(*shp*, *fc_axis*, *fc_pts*, *fillet=1*, *radius=1*)

Fillet or chamfer edges on a certain axis and a list of point contained in that axis

Parameters

- **shp** (*Shape*) – Original shape we want to fillet or chamfer

- **fc_axis** (*FreeCAD.Vector*) – Axis where the fillet will be
- **fc_pts** (*FreeCAD.Vector*) – Vector list of the points
- **fillet** (*int*) –
  - 1 if we are doing a fillet
  - 0 if it is a chamfer
- **radius** (*float*) – Radius of the fillet or chamfer

Returns FreeCAD Shape with fillet/chamfer made

Return type Shape

fcfun.**shp_filletchamfer_dirs**(*shp*, *fc_axis_l*, *fillet=1*, *radius=1*)

Same as shp_filletchamfer_dir, but with a list of directions

Parameters

- **shp** (*Shape*) – Original shape we want to fillet or chamfer
- **fc_axis_l** (*list*) – List of FreeCAD.Vector. Each vector indicates the axis where the fillet/chamfer will be
- **fillet** (*int*) –
  - 1 if we are doing a fillet
  - 0 if it is a chamfer
- **radius** (*float*) – Radius of the fillet or chamfer

Returns FreeCAD Shape with fillet/chamfer made

Return type Shape

fcfun.**shp_hollowbelt_dir**(*center_sep*, *rad1*, *rad2*, *rad_thick*, *height*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_l=FreeCAD.Vector*, *ref_l=1*, *ref_h=1*, *xtr_h=0*, *xtr_nh=0*, *pos=FreeCAD.Vector*)

Makes a shape of 2 tangent circles (like a belt joining 2 circles). check shp_belt_wire_dir

Parameters

- **center_sep** (*float*) – Separation of the circle centers
- **rad1** (*float*) – Internal radius of the first circle, on the opposite direction of fc_axis_l
- **rad2** (*float*) – Internal radius of the second circle, on the direction of fc_axis_l
- **rad_thick** (*float*) – Increment to rad1 and rad2 to make the thickness.
- **height** (*float*) – Height of the shape
- **fc_axis_l** (*FreeCAD.Vector*) – Vector on the direction circle centers, pointing to rad2
- **fc_axis_h** (*FreeCAD.Vector*) – Vector on the height direction
- **ref_l** (*int*) – Reference (zero) of the fc_axis_l
  - 1: reference on the center
  - 2: reference at one of the semicircle centers (point 2) the other circle center will be on the direction of fc_axis_l
  - 3: reference at the end of rad1 circle the other end will be on the direction of fc_axis_l
- **ref_h** (*int*) –

- 1: reference is at the center of the height
- 2: reference is at the bottom

- **xtr_h** (*float*) – If >0 it will be that extra height on the direction of fc_axis_h
- **xtr_nh** (*float*) – If >0 it will be that extra height on the opositve direction of fc_axis_h
- **pos** (*FreeCAD.Vector*) – Position of the reference

Returns FreeCAD Shape

Return type Shape

fcfun.**shp_nuthole**(*nut_r*, *nut_h*, *hole_h*, *xtr_nut=1*, *xtr_hole=1*, *fc_axis_nut=FreeCAD.Vector*, *fc_axis_hole=FreeCAD.Vector*, *ref_nut_ax=1*, *ref_hole_ax=1*, *pos=FreeCAD.Vector*)

Similar to NutHole, but creates a shape, in any direction. Add a Nut hole (hexagonal) with a prism attached to introduce the nut tolerances are included

(continues on next page)

(continued from previous page)

```
    |   |
    |   |
    |_1_|----fc_axis_nut
    |   |                 ___  but pos is still referenced on the axis of
    |___|.....           |   |    the shank
              xtr_nut....|___|
```

Parameters

- **nut_r** (*float*) – Circumradius of the hexagon
- **nut_h** (*float*) – Height of the nut, usually larger than the actual nut height, to be able to introduce it
- **hole_h** (*float*) – The hole height, from the center of the hexagon to the side it will see light
- **xtr_nut** (*int*) – 1 if you want 1 mm out of the hole, to cut
- **xtr_hole** (*int*) – 1 if you want 1 mm out of the hole, to cut
- **fc_axis_nut** (*FreeCAD.Vector*) – Axis of the shank of the nut
- **fc_axis_hole** (*FreeCAD.Vector*) – Axis of the shank of the nut
- **ref_nut_ax** (*int*) – If it is referenced to the center, symmetrical point on the on the fc_axis_nut
- **ref_hole_ax** (*int*) – If it is referenced at the center of the shank, or at the end of the hole, not counting extra
- **pos** (*FreeCAD.Vector*) – Position

Returns FreeCAD Shape of a nut hole

Return type Shape

fcfun.**shp_regpolygon_dir_face**(*n_sides*, *radius*, *fc_normal=FreeCAD.Vector*, *fc_verx1=FreeCAD.Vector*, *pos=FreeCAD.Vector*)

Similar to shp_regpolygon_face, but in any direction of the space makes the shape of a face of a regular polygon

Parameters

- **n_sides** (*int*) – Number of sides of the polygon
- **radius** (*float*) – Circumradius of the polygon
- **fc_normal** (*FreeCAD.Vector*) – Direction of the normal
- **fc_verx1** (*FreeCAD.Vector*) – Direction of the first vertex
- **pos** (*FreeCAD.Vector*) – Position of the center. Default (0,0,0)

Returns FreeCAD Face of a regular polygon

Return type Shape Face

fcfun.**shp_regpolygon_face**(*n_sides*, *radius*, *n_axis='z'*, *v_axis='x'*, *edge_rot=0*, *pos=FreeCAD.Vector*)

Makes the shape of a face of a regular polygon

Parameters

- **n_sides** (*int*) – Number of sides of the polygon
- **radius** (*float*) – Circumradius of the polygon

- **n_axis** (*str*) – Axis of the normal: 'x', '-x', 'y', '-y', 'z', '-z'
- **v_axis** (*str*) – Perpendicular to n_axis, pointing to the first vertex, unless, x_angle is != 0. the vertex will be rotated x_angle degrees for v_axis
- **x_angle** (*float*) – If zero, the first vertex will be on axis v_axis if x_angle != 0, it will rotated some angle
- **pos** (*FreeCAD.Vector*) – Position of the center. Default (0,0,0)

Returns FreeCAD Face of a regular polygon

Return type Shape Face

fcfun.**shp_regprism**(*n_sides*, *radius*, *length*, *n_axis='z'*, *v_axis='x'*, *centered=0*, *edge_rot=0*, *pos=FreeCAD.Vector*)

Makes a shape of a face of a regular polygon

Parameters

- **n_sides** (*int*) – Number of sides of the polygon
- **radius** (*float*) – Circumradius of the polygon
- **length** (*float*) – Length of the polygon
- **n_axis** (*str*) – Axis of the normal: 'x', '-x', 'y', '-y', 'z', '-z'
- **v_axis** (*str*) – Perpendicular to n_axis, pointing to the first vertex, unless, x_angle is != 0. the vertex will be rotated x_angle degrees for v_axis
- **centered** (*int*) – 1 if the extrusion is centered on pos (symmetrical)
- **x_angle** (*float*) – if zero, the first vertex will be on axis v_axis if x_angle != 0, it will rotated some angle
- **pos** (*FreeCAD.Vector*) – Position of the center. Default (0,0,0)

Returns FreeCAD Shape of a regular prism

Return type Shape

fcfun.**shp_regprism_dirxtr**(*n_sides*, *radius*, *length*, *fc_normal=FreeCAD.Vector*, *fc_verx1=FreeCAD.Vector*, *centered=0*, *xtr_top=0*, *xtr_bot=0*, *pos=FreeCAD.Vector*)

Similar to shp_regprism_xtr, but in any direction makes a shape of a face of a regular polygon. Includes the possibility to add extra length on top and bottom. On top is easy, but at the bottom, the reference will be no counting that extra length added. This is useful to make boolean difference

Parameters

- **n_sides** (*int*) – Number of sides of the polygon
- **radius** (*float*) – Circumradius of the polygon
- **length** (*float*) – Length of the polygon
- **fc_normal** (*FreeCAD.Vector*) – Direction of the normal
- **fc_verx1** (*FreeCAD.Vector*) – Direction of the first vertex
- **centered** (*int*) – 1 if the extrusion is centered on pos (symmetrical)
- **xtr_top** (*float*) – Add an extra length on top. If 0, nothing added
- **xtr_bot** (*float*) – Add an extra length at the bottom. If 0, nothing added
- **pos** (*FreeCAD.Vector*) – Position of the center. Default (0,0,0)

Returns FreeCAD Shape of a regular prism

Return type Shape

fcfun.**shp_regprism_xtr**(*n_sides*, *radius*, *length*, *n_axis='z'*, *v_axis='x'*, *centered=0*, *xtr_top=0*, *xtr_bot=0*, *edge_rot=0*, *pos=FreeCAD.Vector*)

makes a shape of a face of a regular polygon. Includes the possibility to add extra length on top and bottom. On top is easy, but at the bottom, the reference will be no counting that extra length added. This is useful to make boolean difference

Parameters

- **n_sides** (*int*) – Number of sides of the polygon
- **radius** (*float*) – Circumradius of the polygon
- **length** (*float*) – Length of the polygon
- **n_axis** (*str*) – Axis of the normal: 'x', '-x', 'y', '-y', 'z', '-z'
- **v_axis** (*str*) – Perpendicular to n_axis, pointing to the first vertex, unless, x_angle is != 0. the vertex will be rotated x_angle degrees for v_axis
- **centered** (*int*) – 1 if the extrusion is centered on pos (symmetrical)
- **xtr_top** (*float*) – Add an extra length on top. If 0, nothing added
- **xtr_bot** (*float*) – Add an extra length at the bottom. If 0, nothing added
- **x_angle** (*float*) – If zero, the first vertex will be on axis v_axis if x_angle != 0, it will rotated some angle
- **pos** (*FreeCAD.Vector*) – Position of the center. Default (0,0,0)

Returns FreeCAD Shape of a regular prism

Return type Shape

fcfun.**shp_rndrect_face**(*x*, *y*, *r=0.5*, *pos_z=0*)

Same as shpRndRectWire

Parameters

- **x** (*float*) – Dimension of the base, on the X axis
- **y** (*float*) – Dimension of the height, on the Y axis
- **r** (*float*) – Radius of the rouned edge.
- **zpos** (*float*) – Position on the Z axis

Returns FreeCAD Face of a rounded edges rectangle

Return type Shape Face

fcfun.**shp_stadium_dir**(*length*, *radius*, *height*, *fc_axis_h=FreeCAD.Vector*, *fc_axis_l=FreeCAD.Vector*, *fc_axis_s=FreeCAD.Vector*, *ref_l=1*, *ref_s=1*, *ref_h=1*, *xtr_h=0*, *xtr_nh=0*, *pos=FreeCAD.Vector*)

Makes a stadium shape in any direction

```
fc_axis_s
  :
  :__________            ref_l = 2, ref_s = 1
  /          \
 3 2    1     ) -------> fc_axis_l
 5\_____4____/

fc_axis_h
 _:___________ ..........................
```

(continues on next page)

(continued from previous page)

```
|               |                                 :
|               |                                 :
|       1       |       ref_h=1                   + h
|               |                                 :
|_______2_______|.......> fc_axis_l .........:  ref_h=2
```

Parameters

- **length** (*float*) – Length of the parallels (distance between semcircle centers)
- **height** (*float*) – Height the stadium
- **fc_axis_s** (*FreeCAD.Vector*) – Direction on the short axis, not necessary if ref_s == 1 it will be the perpendicular of the other 2 vectors
- **fc_axis_h** (*FreeCAD.Vector*) – Vector on the height direction
- **ref_l** (*int*) – Reference (zero) of the fc_axis_l
  - 1: reference on the center (makes axis_s symmetrical)
  - 2: reference at one of the semicircle centers (point 2) the other circle center will be on the direction of fc_axis_l
  - 3: reference at the end (point 3) the other end will be on the direction of fc_axis_l
- **ref_s** (*int*) – Reference (zero) of the fc_axis_s
  - 1: reference at the center (makes axis_l symmetrical): p 1,2,3
  - 2: reference at the parallels lines: p: 4, 5 the other parallel will be on the direction of fc_axis_s
- **ref_h** (*int*) –
  - 1: reference is at the center of the height
  - 2: reference is at the bottom
- **xtr_h** (*float*) – If >0 it will be that extra height on the direction of fc_axis_h
- **xtr_nh** (*float*) – If >0 it will be that extra height on the opositve direction of fc_axis_h
- **pos** (*FreeCAD.Vector*) – Placement

Returns FreeCAD Shape of a stadium

Return type Shape

fcfun.**shp_stadium_face**(*l*, *r*, *axis_rect='x'*, *pos_z=0*)

Same as shp_stadium_wire, but returns a face

Parameters

- **l** (*float*) – Length of the parallels (from center to center)
- **r** (*float*) – Radius of the semicircles
- **axis_rect** (*str*) – 'x' the parallels are on axis X (as in the drawing) 'y' the parallels are on axis Y
- **pos_z** (*float*) – Position on the Z axis

Returns FreeCAD Face of a stadium

Return type Shape Face

fcfun.**shp_stadium_wire**(*l*, *r*, *axis_rect='x'*, *pos_z=0*)

Creates a wire (shape), that is a rectangle with semicircles at a pair of opposite sides. Also called discorectangle it will be centered on XY

```
                  Y
 ______ .. r      |_X
(______)--
 :    :
 :.l.:
```

Parameters

- **l** (*float*) – Length of the parallels (from center to center)
- **r** (*float*) – Radius of the semicircles
- **axis_rect** (*str*) – 'x' the parallels are on axis X (as in the drawing) 'y' the parallels are on axis Y
- **pos_z** (*float*) – Position on the Z axis

Returns FreeCAD Wire of a stadium

Return type Shape Wire

fcfun.**shp_stadium_wire_dir**(*length*, *radius*, *fc_axis_l=FreeCAD.Vector*, *fc_axis_s=FreeCAD.Vector*, *ref_l=1*, *ref_s=1*, *pos=FreeCAD.Vector*)

Same as shp_stadium_wire but in any direction Also called discorectangle

```
                              fc_axis_s
                               |
          ______ .. radius     |--> fc_axis_l
         (______)--
          :    :
          :.l.:
          length

     fc_axis_s              in this drawing,
      :  p_edge             the zero is on point 2
      :_________           ref_l = 2, ref_s = 1
     /          \
    3 2    1     ) -------> fc_axis_l
    5\_____4____/   n_edge
        n_edge
n circle        p circle
```

Parameters

- **length** (*float*) – Length of the parallels (distance between semcircle centers)
- **radius** (*float*) – Radius of the semicircles
- **fc_axis_l** (*FreeCAD.Vector*) – Vector on the direction of the paralles
- **fc_axis_s** (*FreeCAD.Vector*) – Vector on the direction perpendicular to the paralles
- **ref_l** (*int*) – Reference (zero) of the fc_axis_l
  - 1 reference on the center (makes axis_s symmetrical)
  - 2 reference at one of the semicircle centers (point 2) the other circle center will be on the direction of fc_axis_l

- 3 reference at the end (point 3) the other end will be on the direction of fc_axis_l

- **ref_s** (*int*) – Reference (zero) of the fc_axis_s
  - 1 reference at the center (makes axis_l symmetrical): p 1,2,3
  - 2 reference at the parallels lines: p: 4, 5 the other parallel will be on the direction of fc_axis_s
- **pos** (*FreeCAD.Vector*) – FreeCAD vector of the position of the reference

Returns FreeCAD Wire of a stadium

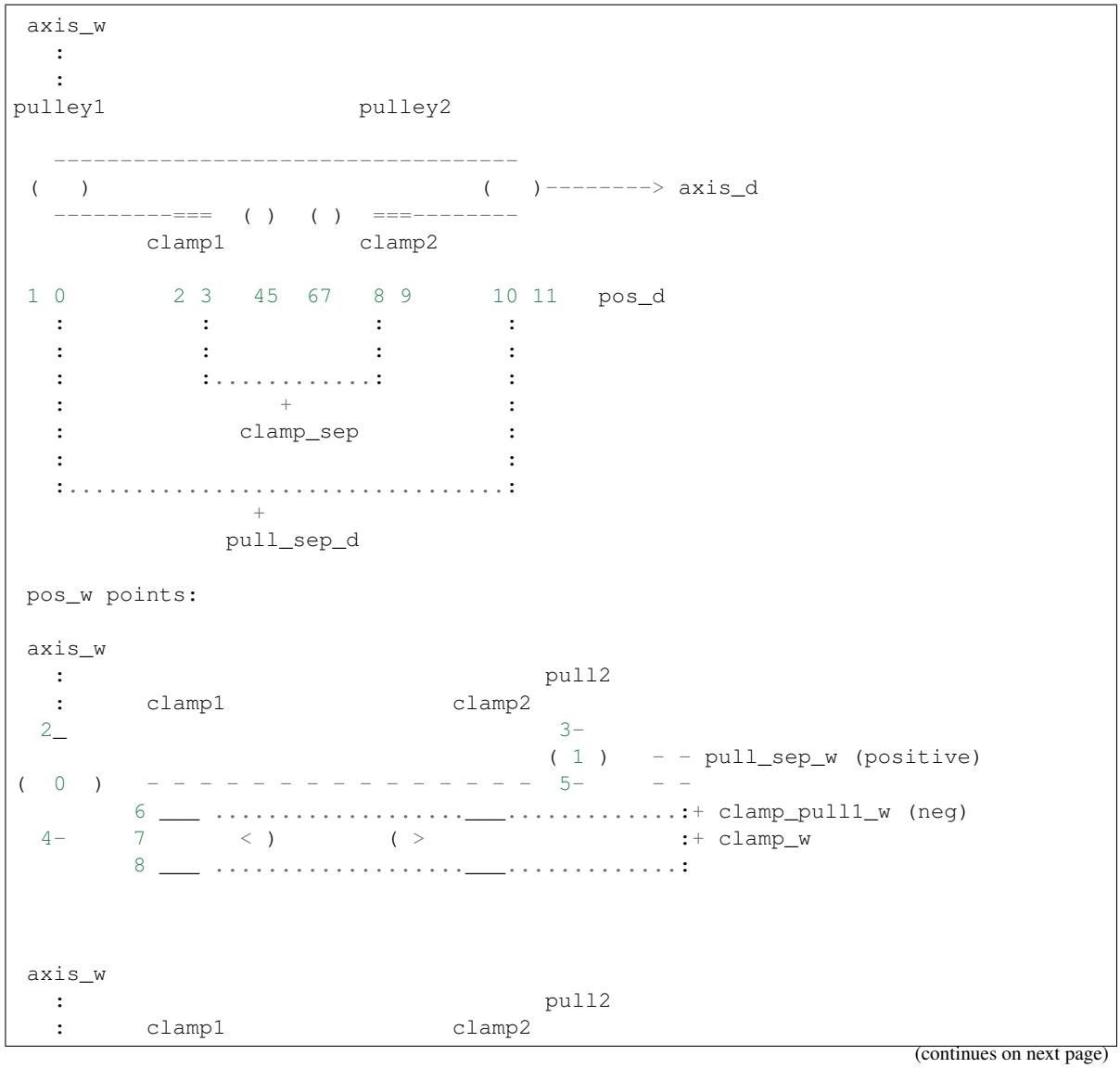
Return type Shape Wire

fcfun.**vecname_paral**(*vec1*, *vec2*)

Given to vectors by name 'x', '-x', ... indicates if they are parallel or not

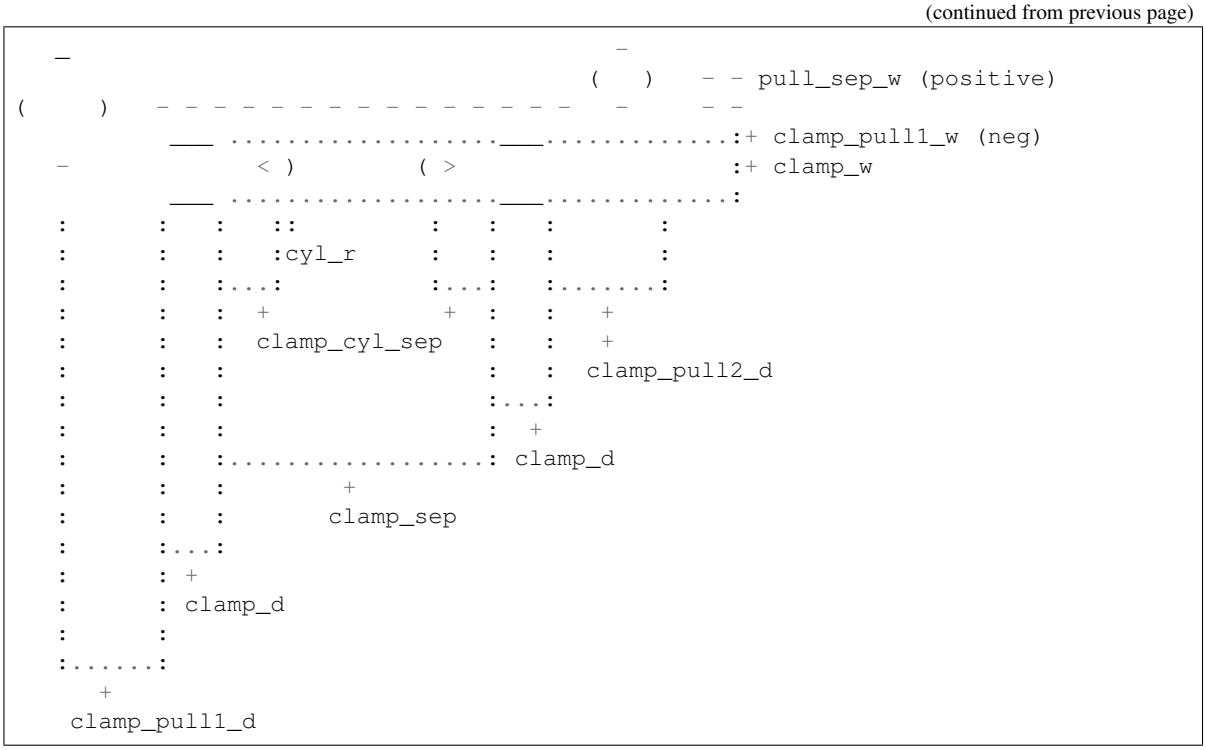
fcfun.**wire_beltclamp**(*d*, *w*, *corner_r*, *conn_d*, *conn_sep*, *xtr_conn_d=0*, *closed=0*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos=FreeCAD.Vector*)

Creates a wire following 2 pulleys and ending in a belt clamp But it is a wire in FreeCAD, has no volumen



(continues on next page)

(continued from previous page)



Parameters

- **pull1_d** (*float*) – Diameter of pulley 1
- **pull2_d** (*float*) – Diameter of pulley 2
- **pull_sep_d** (*float*) – Separation between the 2 pulleys centers along axis_d if positive, pulley 2 is further away in the direction of axis_d if negative, pulley 2 is further away opposite to the direction of axis_d
- **pull_sep_w** (*float*) – Separation between the 2 pulleys centers along axis_w if positive, pulley 2 is further away in the direction of axis_w if negative, pulley 2 is further away opposite to the direction of axis_w
- **clamp_pull1_d** (*float*) – Separation between the clamp (side closer to the center) and the center of the pulley1
- **clamp_pull1_w** (*float*) – Separation between the center of the clamp and the center of the pulley1 if positive, the clamp is further away in the direction of axis_w if negative, the clamp is further away opposite to the direction of axis_w
- **clamp_d** (*float*) – Length of the clamp (same for each clamp)
- **clamp_w** (*float*) – Width of inner space (same for each clamp)
- **clamp_sep** (*float*) – Separation between clamps, the closest ends
- **clamp_cyl_sep** (*float*) – Separation between clamp and the center of the cylinder (or the center) of the larger cylinder (when is a belt shape)
- **cyl_r** (*float*) – Radius of the cylinder for the belt, if it is not a cylinder but a shape of 2 cylinders: <) , then the raidius of the larger one
- **axis_d** (*FreeCAD.Vector*) – Coordinate System Vector along the depth
- **axis_w** (*FreeCAD.Vector*) – Coordinate System Vector along the width

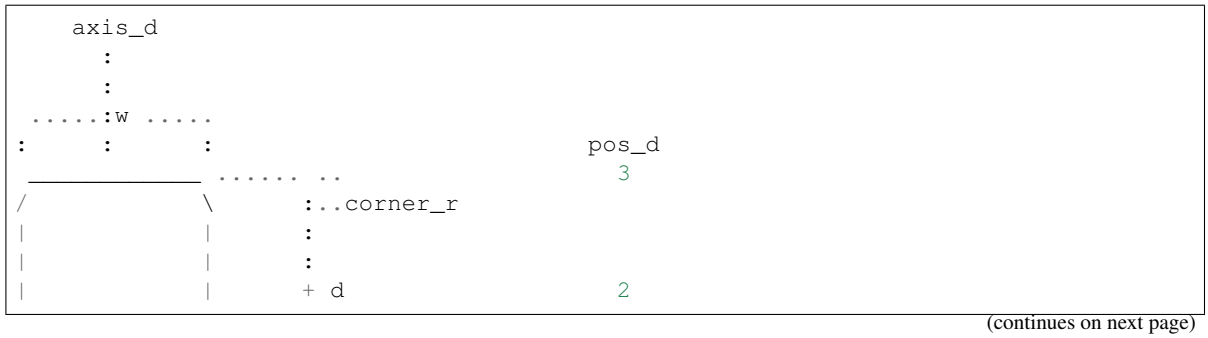
- **pos_d** (*int*) – Location of pos along the axis_d, see drawing
  - 0: center of the pulley 1
  - 1: end of pulley 1
  - 2: end of clamp 1, closest end to pulley 1
  - 3: other end of clamp 1, closest to cylinder
  - 4: center of cylinder (or shape <) 1
  - 5: external radius of cylinder 1
  - 6: external radius of cylinder 2
  - 7: center of cylinder (or shape (> 2
  - 8: end of clamp 2, closest to cylinder
  - 9: other end of clamp 2, closest end to pulley 2
  - 10: center of pulley 2
  - 11: end of pulley 2
- **pos_w** (*int*) – Location of pos along the axis_w, see drawing
  - 0: center of pulley 1
  - 1: center of pulley 2
  - 2: end (radius) of pulley 1 along axis_w
  - 3: end (radius) of pulley 2 along axis_w
  - 4: other end (radius) of pulley 1 opposite to axis_w
  - 5: other end (radius) of pulley 2 opposite to axis_w
  - 6: clamp space, closest to the pulley
  - 7: center of clamp space
  - 8: clamp space, far away from the pulley
- **pos** (*FreeCAD.Vector*) – Position of the reference

Returns FreeCAD Wire of a belt clamped

Return type Shape Wire

fcfun.**wire_cableturn**(*d*, *w*, *corner_r*, *conn_d*, *conn_sep*, *xtr_conn_d=0*, *closed=0*, *axis_d=FreeCAD.Vector*, *axis_w=FreeCAD.Vector*, *pos_d=0*, *pos_w=0*, *pos=FreeCAD.Vector*)

Creates a electrical wire turn, in any direction But it is a wire in FreeCAD, has no volumen



(continues on next page)

(continued from previous page)

```
|              |       :
|              |       :
|              |       :
 \___ o _____/ ......:                       1
     \ /              :
     | |              + conn_d
     | |              :
     | |..............:............axis_w    0
     : :
    conn_sep

1     0  pos_w

pos_o (orig) is at pos_d=0, pos_w=0, marked with o
```

Parameters

- **d** (*float*) – Depth/length of the turn
- **w** (*float*) – Width of the turn
- **corner_r** (*float*) – Radius of the corners
- **conn_d** (*float*) – Depth/length of the connector part
  - 0: there is no connecting wire
- **xtr_conn_d** (*float*) – If conn_d > 0, there can be and extra length of connector to make unions, it will not be counted as pos_d = 0 It will not work well if it is closed
- **conn_sep** (*float*) – Separation of the connectors
- **closed** (*boolean*) –
  - 0 : the ends are not closed
  - 1 : the ends are closed
- **axis_d** (*FreeCAD.Vector*) – Coordinate System Vector along the depth
- **axis_w** (*FreeCAD.Vector*) – Coordinate System Vector along the width
- **pos_d** (*int*) – Location of pos along the axis_d (0,1,2,3), see drawing
  - 0: reference at the beginning of the connector
  - 1: reference at the beginning of the turn, at the side of the connector
  - 2: reference at the middle of the turn
  - 3: reference at the end of the turn
- **pos_w** (*int*) – Location of pos along the axis_w (0,1), see drawing
  - 0: reference at the center of symmetry
  - 1: reference at the end of the turn
- **pos** (*FreeCAD.Vector*) – Position of the reference

Returns FreeCAD Wire of a electrical wire

Return type Shape Wire

fcfun.**wire_lgrail**(*rail_w*, *rail_h*, *axis_w=FreeCAD.Vector*, *axis_h=FreeCAD.Vector*, *pos_w=0*, *pos_h=0*, *pos=FreeCAD.Vector*)

Creates a wire of a linear guide rail, the dent is just rough, to be able to see that it is a profile

```
               axis_h
                 :
     ne _________2_________ e
        |                  |
     nd|                  | d
        \ nc          c /   A little dent to see that it is a rail
     nb /        1       \ b
        |                  |
        |                  |
        |                  |
     na|_________o_________|a ...... axis_w
                 0         1


                       rail_h/8
                 :       : :
     ne _________2_______:_:....
        |                  |   :+ rail_h/4
     nd|                  |..................
        \ nc          c / ....  rail_h/8   + rail_h/4
     nb /        1       \ ....  rail_h/8...:
        |                  |    :
        |                  |    + rail_h/2
        |                  |    :
     na|_________o_________|a ..:............ axis_w
                 0         1

pos_o (origin) is at pos_w = 0, pos_h = 0
```

Parameters

- **rail_w** (*float*) – Width of the rail
- **rail_h** (*float*) – Height of the rail
- **axis_w** (*FreeCAD.Vector*) – The axis where the width of the rail is
- **axis_h** (*FreeCAD.Vector*) – The axis where the height of the rail is
- **pos_w** (*int*) – Location of pos along axis_w * 0 : center of symmetry * 1 : end of the rail
- **pos_h** (*int*) – Location of pos along axis_h * 0 : bottom * 1 : middle point (this is kind of non-sense) * 2 : top point
- **pos** (*FreeCAD.Vector*) – Position, at the point defined by pos_w and pos_h

Returns Wire of a rail

Return type FreeCAD Wire

fcfun.**wire_sim_xy**(*vecList*)

Creates a wire (shape), from a list of points on the positive quadrant of XY the wire is symmetrical to both X and Y

```
          Y
          |_ X
```

(continues on next page)

(continued from previous page)

```
 __|__
/  |  \ We receive these points
|  |  |
|  |--------          z=0
|     |
\_____/
```

Parameters **vecList** (*list*) – List of FreeCAD Vectors, the have to be in order clockwise if the first or the last points are not on the axis, a new point will be created

CHAPTER

TWO

INDICES AND TABLES

- genindex
- modindex
- search

PYTHON MODULE INDEX

c

f

p

INDEX

G

H

I

L

M

N

P

R

S

T

V

W