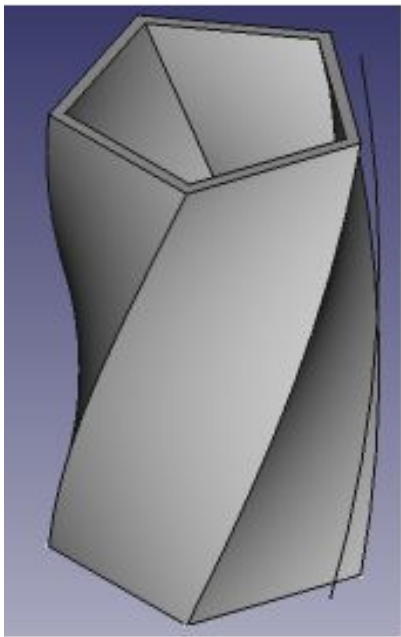# FREECAD FOR ALL

BY: Aurora and Arturo

Freecad

Today, the objective of this important activity is to show you how to use Freecad to make a simple but useful object. What figure is it?

Objectives:

- Have a good time working hard
- Participate in this international activity
- Learn how to make figures with Freecad
- Learn how to use the Part Design Mode
- Have a very useful object with only 10 steps, which only takes 5 minutes
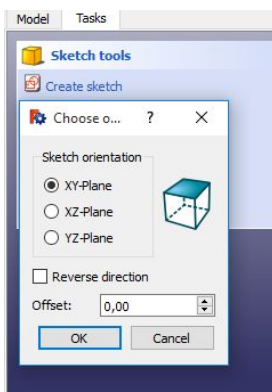
STEP 1

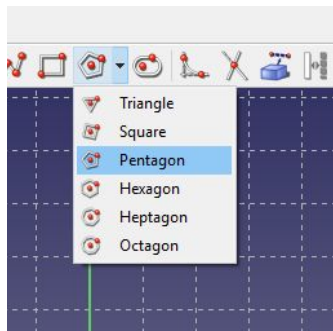First of all, we have to create a new document and select PART DESIGN mode

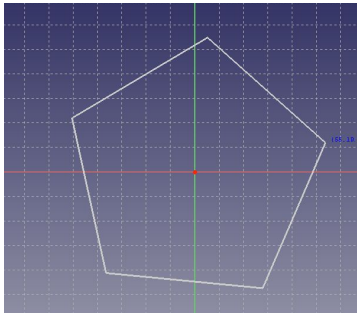## STEP 2

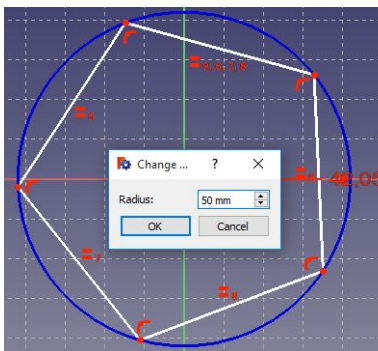Then, we create a sketch and we choose the XY level

STEP 3

In the upper part, we have to select the PENTAGON figure

STEP 4

For making the figure, the first click is to locate the centre of the figure, and the second is to determine how big it is going to be.
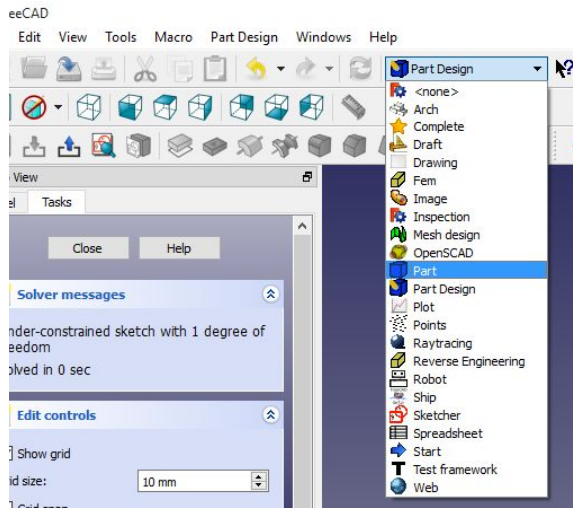
STEP 5

Once we have done all that process, we have to press into the upper point of the pentagon and the line, and after press the FIX ON AN OBJECT button

STEP 6

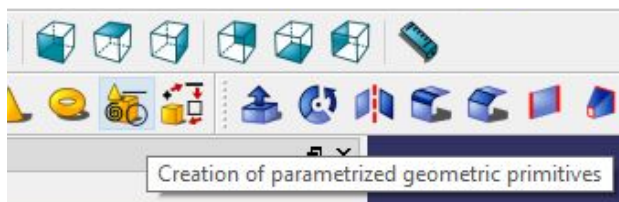Create a coincident constraint on the selected item (C)

Click in the middle point of the axis and in the center of the figure, and then click on the red big button
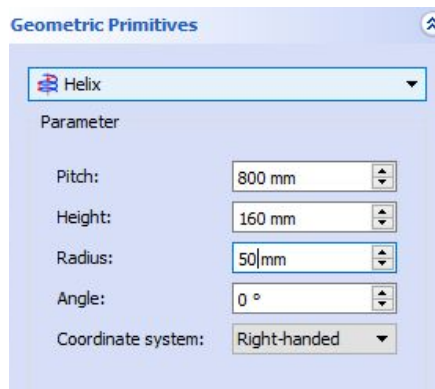
STEP 7

Now, click on "close", and later, choose the PART mode

STEP 8

Creation of parametrized geometric primitives

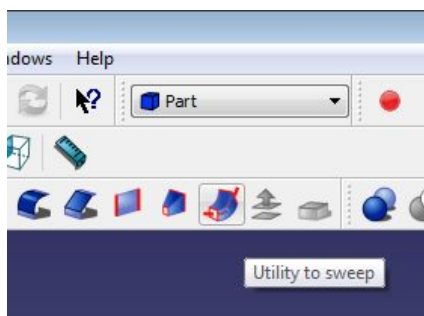For create the helix, click on creation of parametrizes geometric primitives

STEP 9

Select helix, put the measures, then click on CREATE and, after, close.
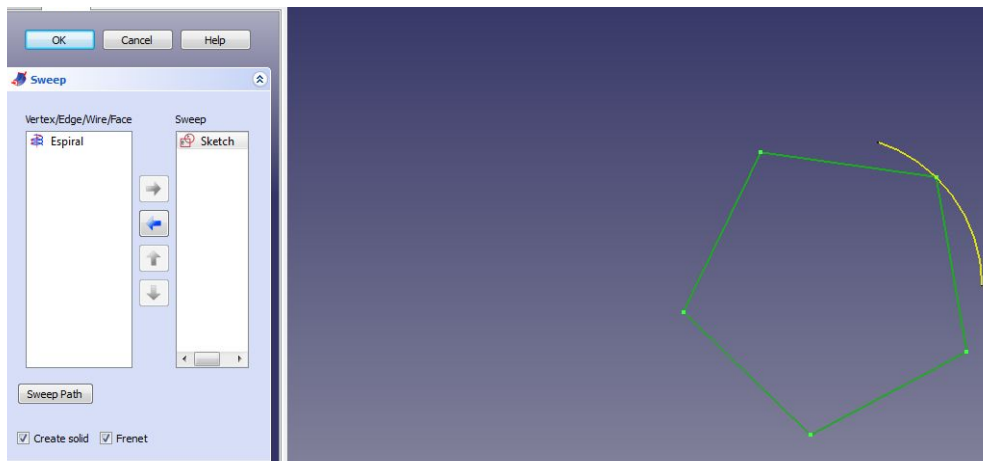
STEP 10

For the next step, click on "utility to sweep"

STEP 11

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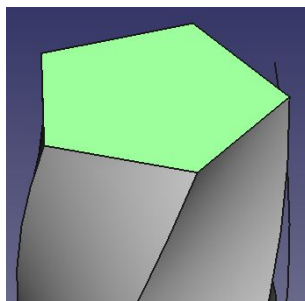First, click (in the left) on sketch, then click the blue arrow, later, click on the helix (in the blue part).

Do not for get to click in the two options below ''Create a solid'' and ''Frenet''
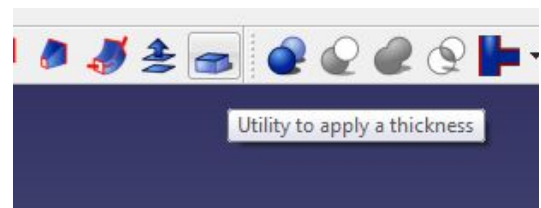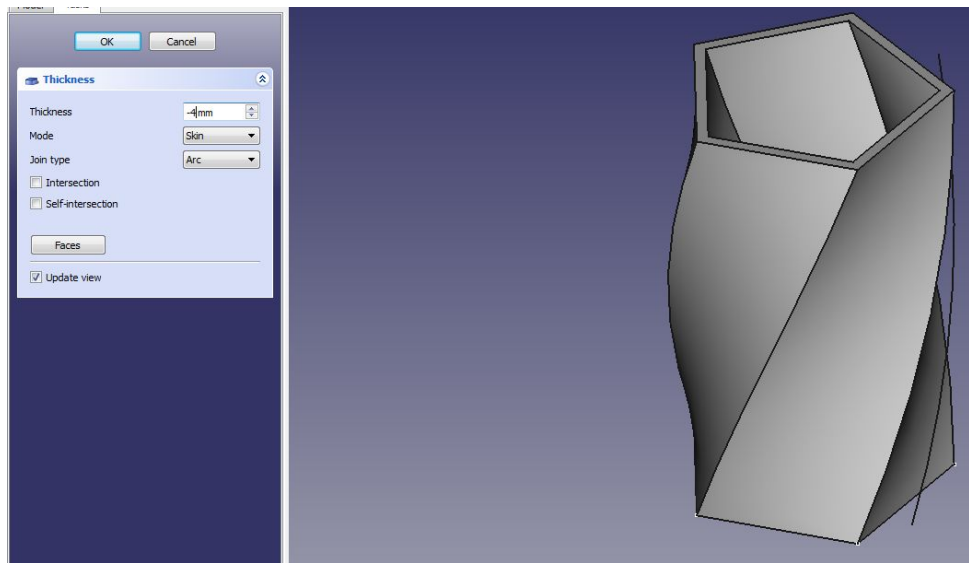
STEP 12

Click on **axonometric view**

STEP 13

Select the top of the figure, clicking on it, and then click on **utility to apply a thickness**

Put -4 mm in the thickness