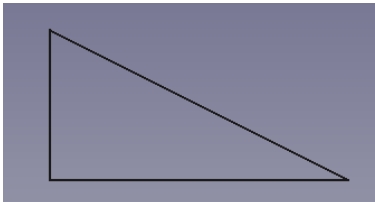# Topological scripting

One dimensional objects

Everything starts with vertices (points)

A script that draws 3 vertices in the XY plane:

```
import Part
from FreeCAD import Vector

# 3 vectors
V1=Vector(20,20,0)
V2=Vector(40,20,0)
V3=Vector(20,30,0)

# 3 points (vertices)
P1 = Part.Vertex(V1)
P2 = Part.Vertex(V2)
P3 = Part.Vertex(V3)

# make them visible:
for p in [P1, P2, P3]:
    Part.show(p)
```

Two dimensional objects

Make a triangle of the 3 vectors defined above:

```
L1 = Part.LineSegment(V1, V2)
L2 = Part.LineSegment(V2, V3)
L3 = Part.LineSegment(V3, V1)
s = Part.Shape([L1, L2, L3])
Part.show(s)
```

The line segments must be converted to a Shape to be made visible.

In some tutorials you find examples that use the Line function. Take care: in FreeCAD 0.17 this does not work, as a Line is infinite!
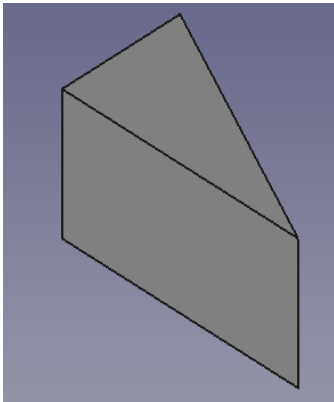
3D objects, extruded

If we extrude the triangle, it gives a hollow shell:
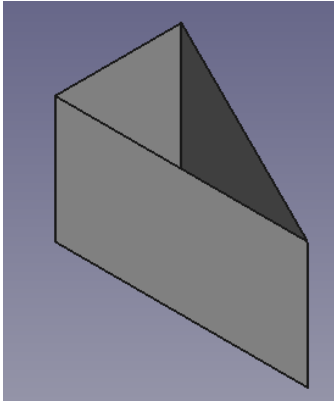

```
sh=s123.extrude(Vector(0,0,10))
Part.show(sh)
```

The extrude vector gives the direction (Z) and the length of the extrusion (10)

To create a solid body, we must create a face that can then be extruded.
To make the face, we need a wire, and this wire is made of edges:


```
E1=Part.Edge(L1)
E2=Part.Edge(L2)
E3=Part.Edge(L3)

W=Part.Wire([E1, E2, E3])
f=Part.Face(W)

solid = f.extrude(Vector(0,0,10))
Part.show(solid)
```