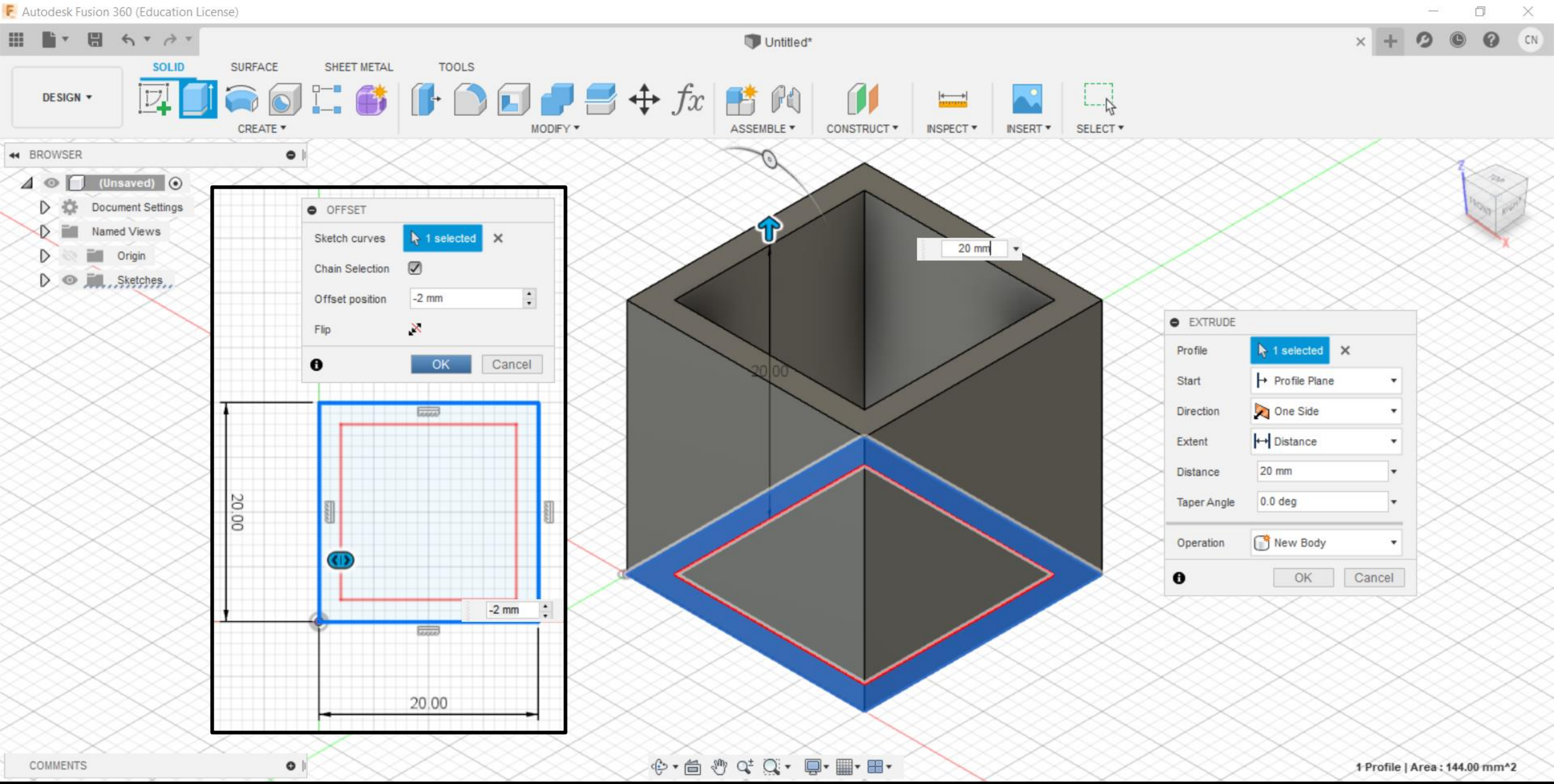


Create sketch and draw a 20mm x 20mm square. Using offset, draw another square within the first one with an offset of 2mm. Once done, finish sketch and extrude the frame by 20mm.

Autodesk Fusion 360 (Education License)

Untitled*

SOLID SURFACE SHEET METAL TOOLS

DESIGN

CREATE MODIFY ASSEMBLE CONSTRUCT INSPECT INSERT SELECT

BROWSER

(Unsaved)

Document Settings

Named Views

Origin

Bodies

Sketches

EXTRUDE

Profile 1 selected

Start Profile Plane

Direction One Side

Extent Distance

Distance -26.00 mm

Taper Angle 0.0 deg

Operation Cut

Objects To Cut

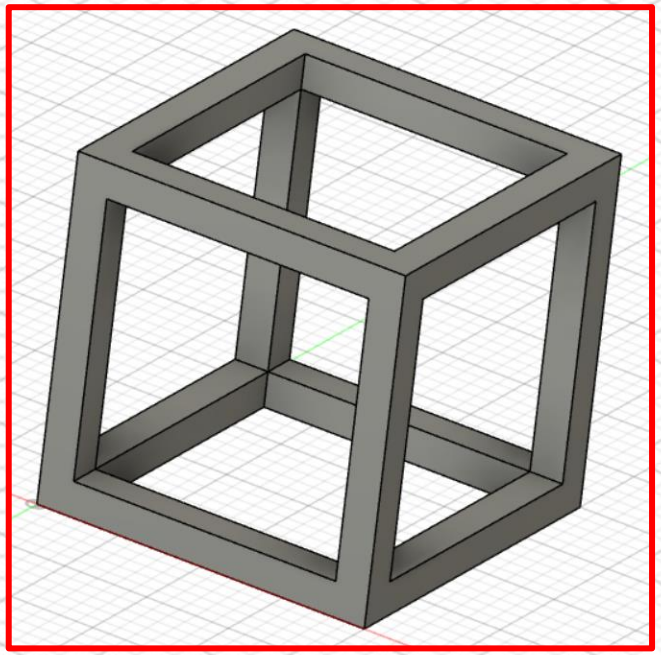
OK Cancel

16.00

16.00

26.00

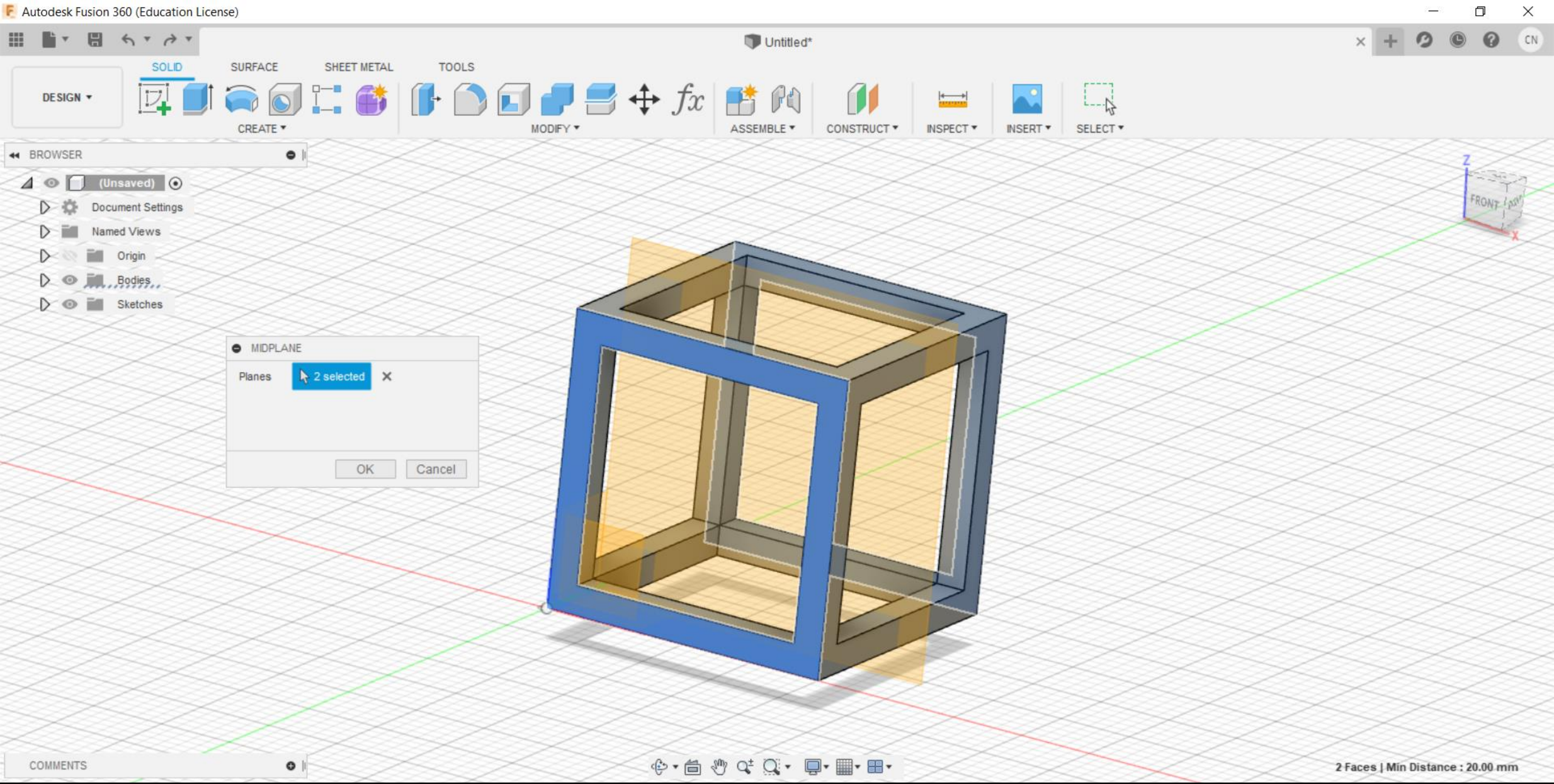
-26.00 mm



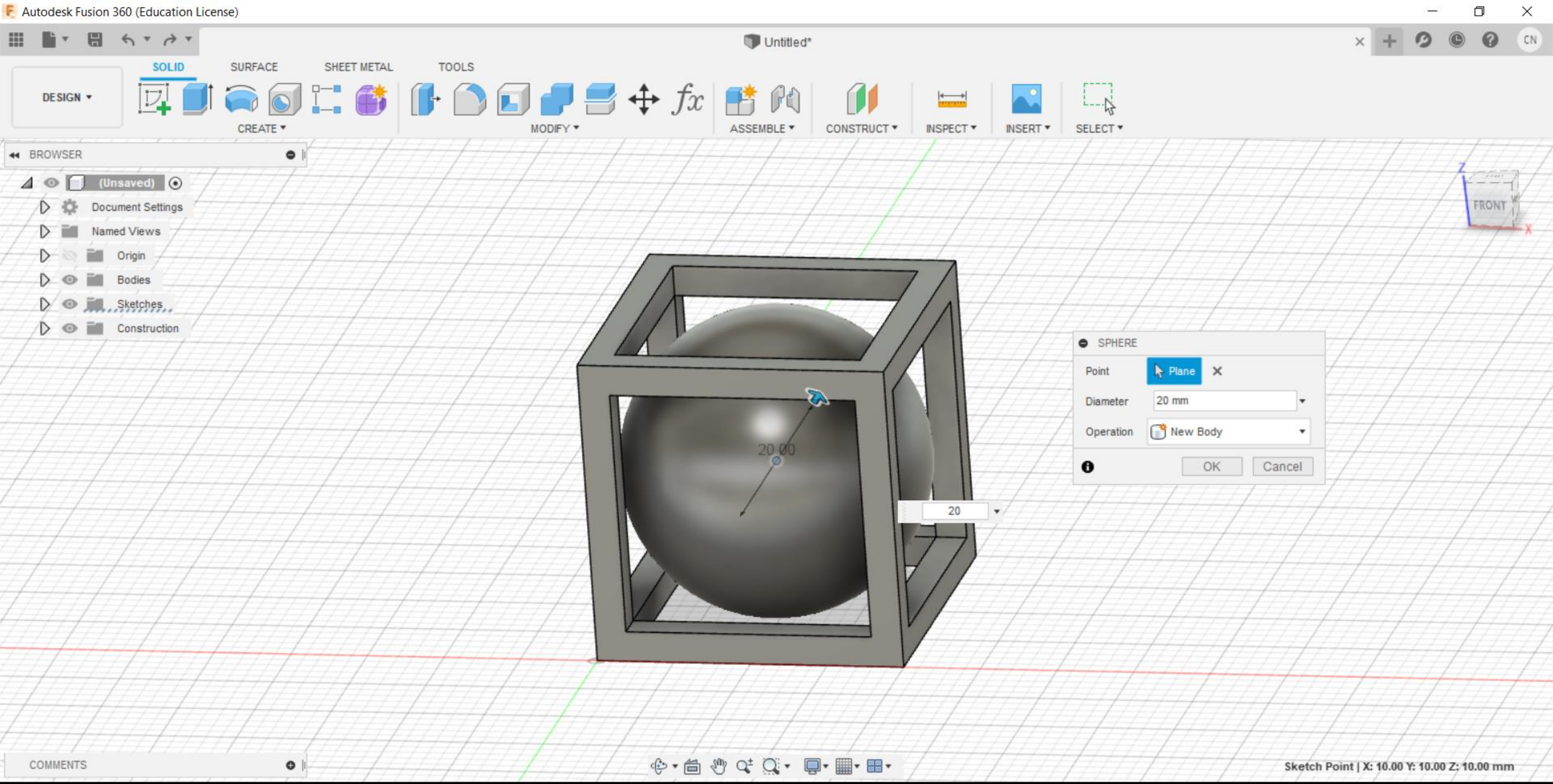
COMMENTS

1 Profile | Area : 256.00 mm^2

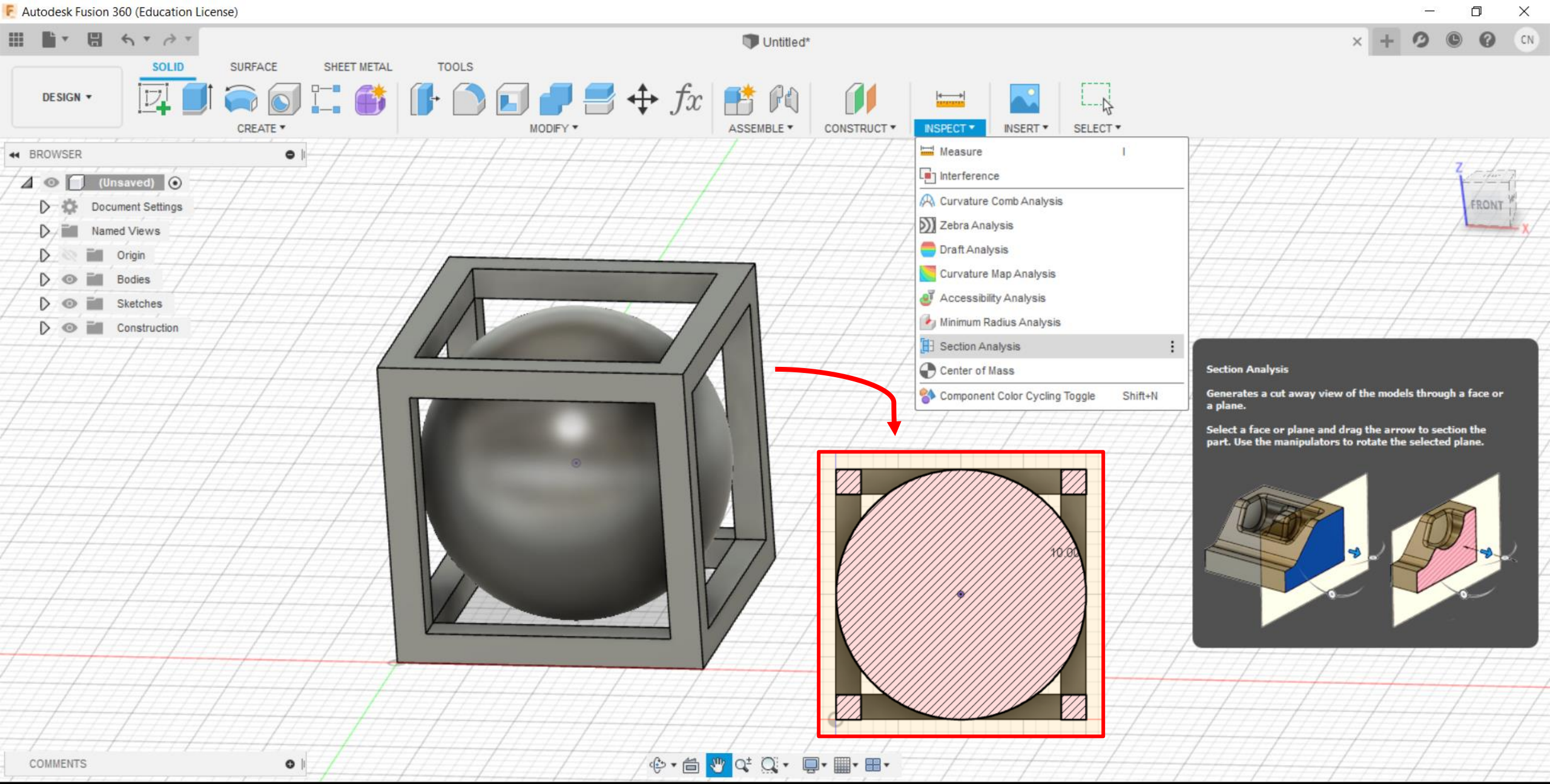
Create a new sketch on the front plane and draw a 16mm x 16mm square. Then extrude until it cuts through the opposite end. Repeat for the right/left side, the result should be a wired-frame cube.



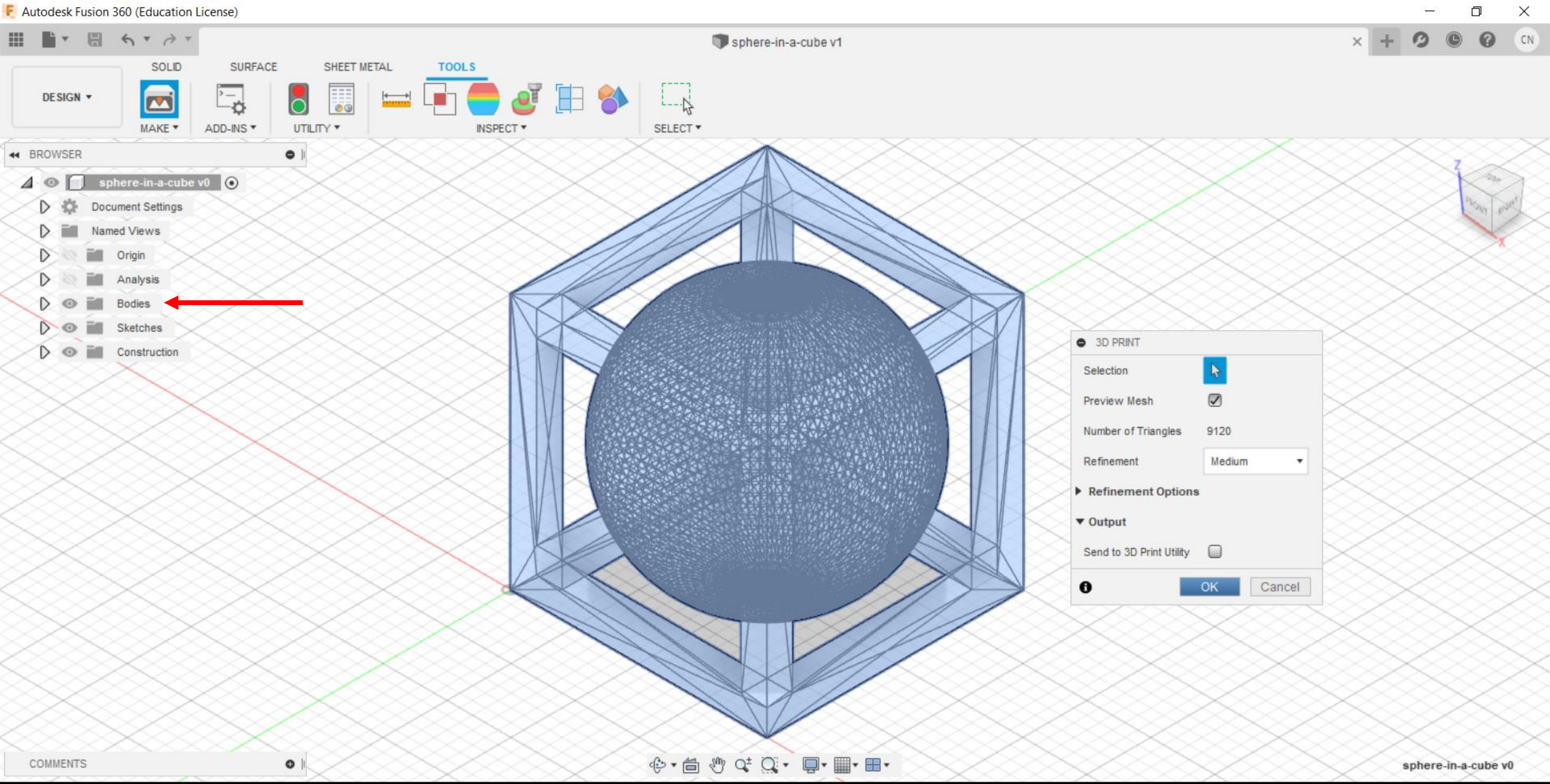
Construct a midplane between the front and back face. Then create a sketch on this plane.



Construct a sphere from the centre of the cube and set diameter as 20mm. This way, a sphere will be ‘trapped’ in the cube.



Perform a section analysis at 10mm. You can see how the sphere sits nicely within the cube.



Go to Make > 3D Print. Select the 2 bodies directly from the panel on the left. Turn on preview mesh and click OK. This will save my objects as .STL file and I will launch Cura to open it.