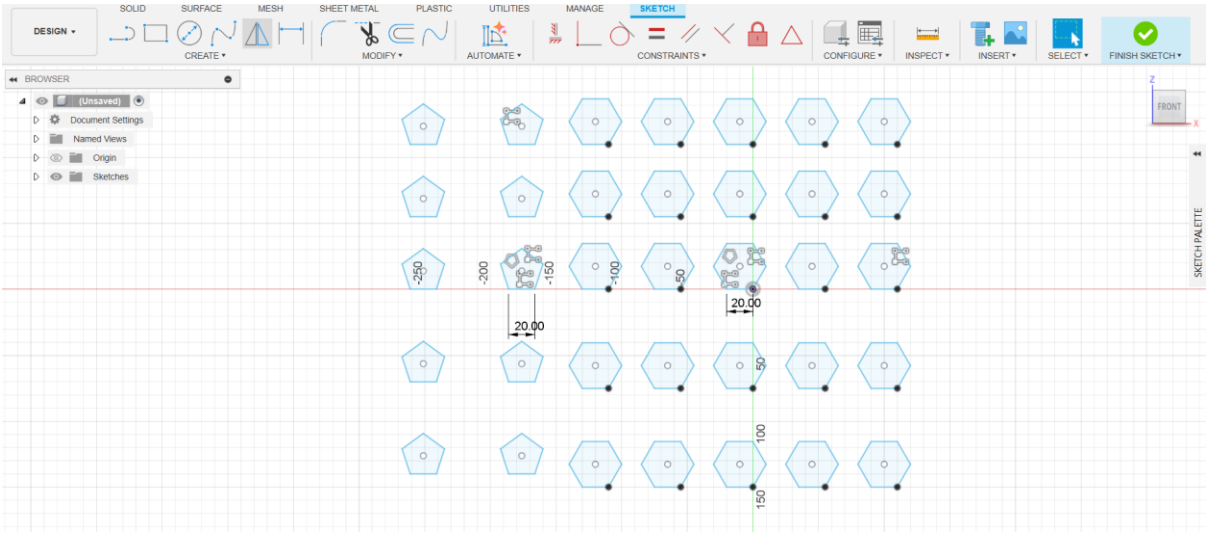
Industrial Design & Rapid Prototyping Techniques

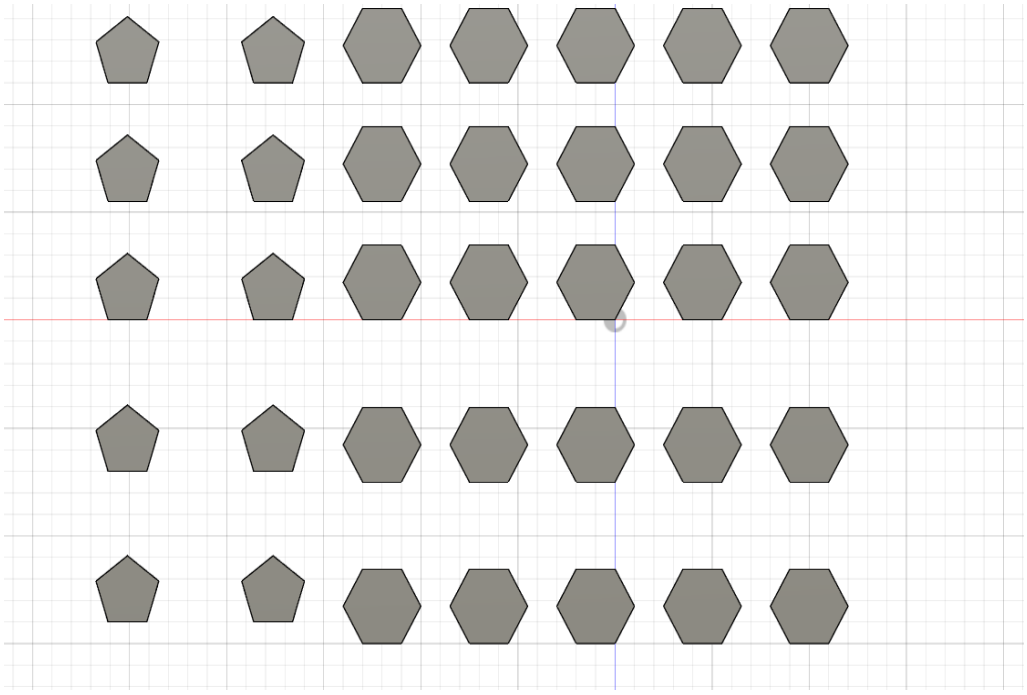
Making the Sketches on Fusion

I had made a Basic shape of a pentagon and hexagon according to the kerf of the material the material I have used here is MDF with the thickness of 2 mm.



Extrusion and making multiple pieces for kit

Here I have Extruded the single unit according to the thickness of my material and made multiple copies of it for making the sphere inside with hollow sphere.



Converting into DXF format for the Lazer cutting

After the whole unit was made I took the sketches of the unit and converted into DXF as lazer cutter reads a vector file and not a 3D file, Hence DXF.

Nesting

According to the material that was available and my size of the unit and to not waste any material I nested all the units so that nothing goes waste.

Lets Cut!!!

This is the interface of the RD works software which is connected to the lazer Cutter, Here I uploaded the nested DXF file for the cutting I set up the Power and speed of the machine in the software itself according to the material I used which was 2 mm thick play.

Power = 75.0 % (power is set in the units of percentage) speed = 8.0 mm/sec

After setting up the power and speed for the the lazer cutter I pressed the download button and the files got uploaded into the machine.

Next step I did was to go to the lazer cutter screen pannel to select the file and making the frame on the bed to see how much space on the material will the cut take. The pannel screen looks like this.

After the cutting was done I started with my assembly of the Kit .

This is how the Lazer cut pieces were looking after the assembly!

Kerf - What is it?

Kerf is the width of the material removed during cutting, which is determined by the laser beam's diameter and the material being cut. Measuring kerf when laser cutting is important for achieving accurate cuts.

How to calculate the kerf:

- Set up the laser cutter and ensure it is properly calibrated for the material you're cutting
- Cut a test piece of the material you plan to use
- Measure the width of the test piece
- Calculate the kerf -> subtract the width of the cut piece from the width of the original material. Divide the result by two to get the kerf. For example, if the original material is 3 mm wide and the cut piece is 2.8 mm wide, the kerf would be (3 mm - 2.8 mm) / 2 = 0.1 mm
- Use the calculated kerf measurement to adjust your design

It's important to note that kerf can vary based on the type of material being cut, the laser's power, and other factors. It's recommended to test the kerf for each new material or laser setting to ensure accurate cuts.

Modeling Process in Fusion 360

Step 1: Initial Sketch Setup

- Create a 3D sketch
- Draw initial polygon with 5 sides
- Establish base circle (diameter = 60 mm)
- Create construction lines for precise positioning

Step 2: Layer Development

- Create multiple polygon layers
- Ensure consistent height and dimensions
- Use construction lines to align layers

Step 3: Vertex Splitting

- Split edges into three equal segments
- Use dimension tools to ensure precision

Step 4: Face Subdivision

- Split triangular faces into:
    - 3 smaller triangles
    - 1 hexagon
- Connect vertices strategically

Step 5: Rotational Patterning

- Use circular pattern feature
- Rotate around Z-axis
- Complete 5 full rotations

Step 6: Surface Refinement

- Stitch faces together
- Patch top and bottom holes
- Convert to solid model

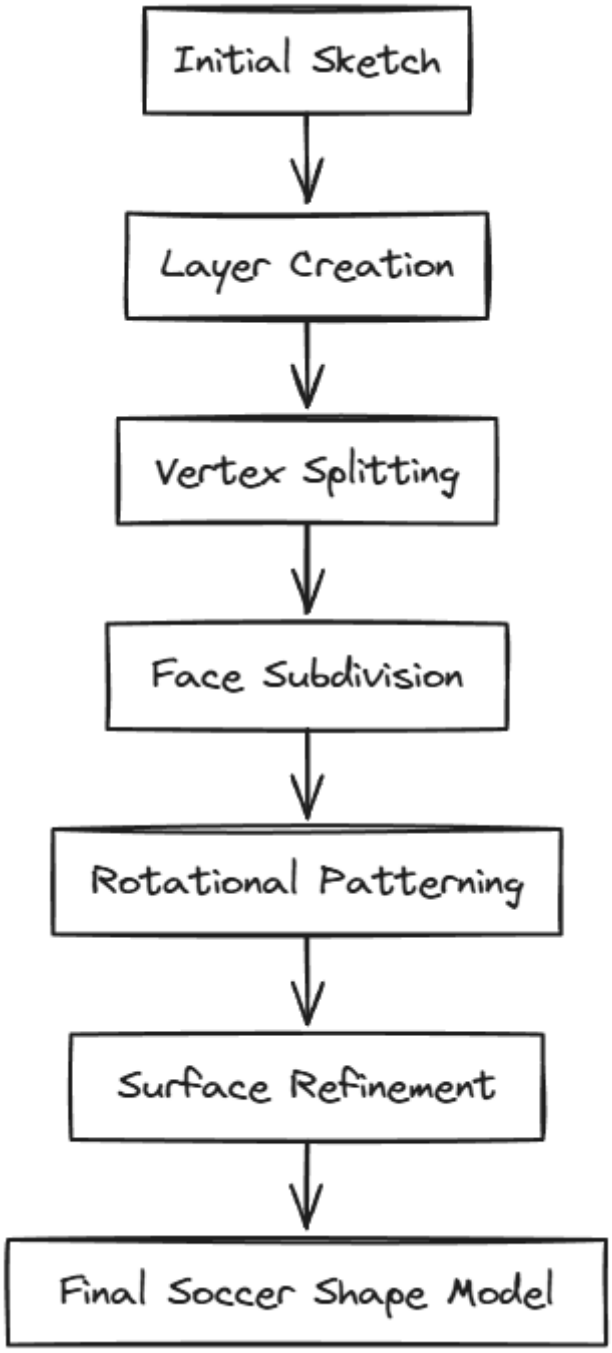
Troubleshooting Tips

- Check for dimensional accuracy
- Inspect for gaps or overlaps
- Verify sketch symmetry

Potential Challenges

- Precise measurement requirements
- Complex geometric transformations
- Maintaining consistent dimensions

Workflow Diagram



Pro Tips

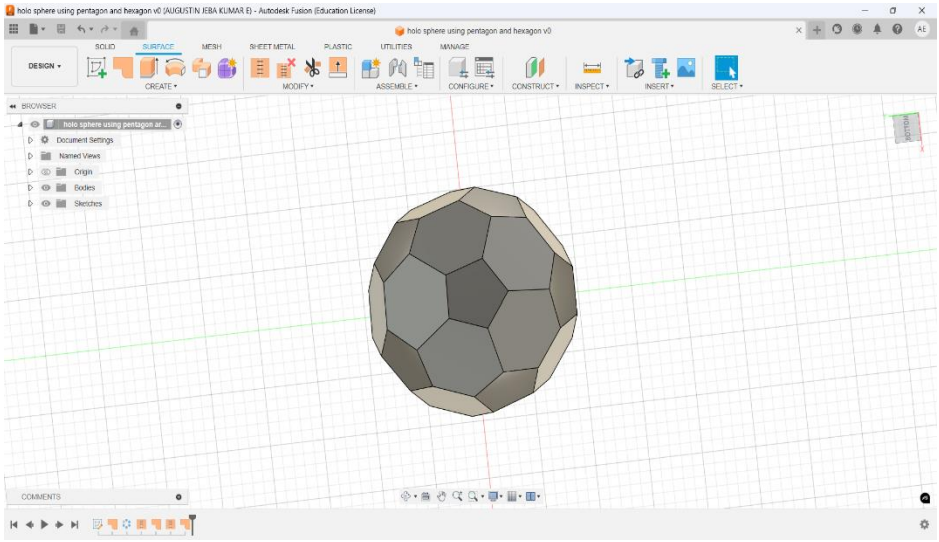
- Use construction lines for precision
- Regularly check dimensional consistency
- Leverage Fusion 360's pattern and stitch tools

Common Mistakes to Avoid

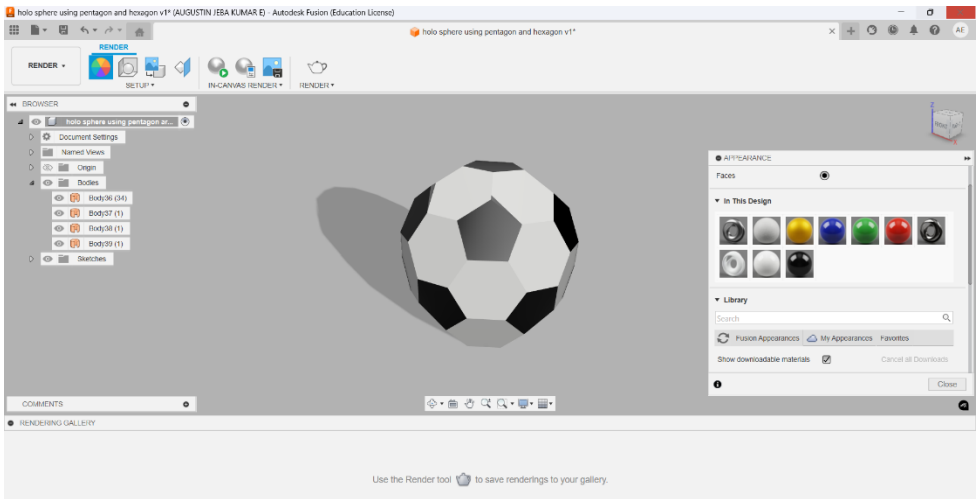
- Inconsistent edge lengths
- Improper vertex chopping
- Neglecting symmetry principles

Final 3D design using Fusion

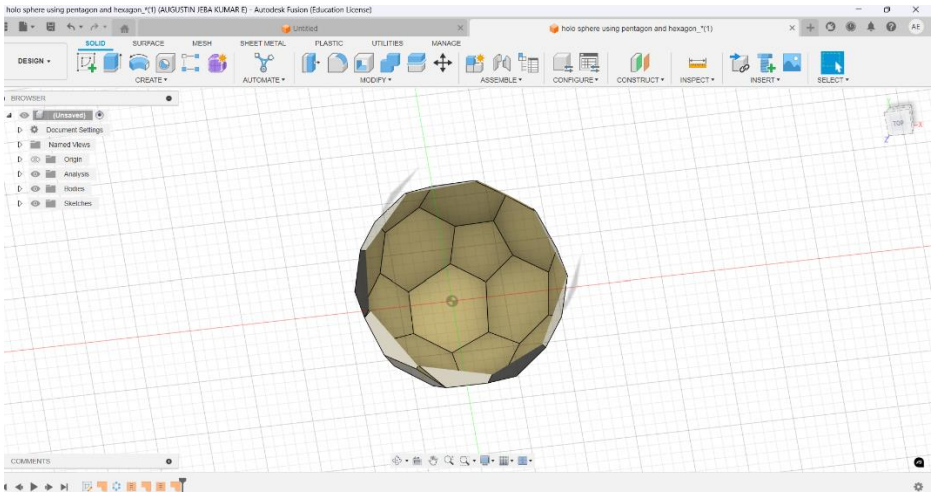
Before manually assembling all the cut pieces to make a hollow sphere . I had used Fusion 360 to make my design as a 3D model to verify whether my design parameters are suitable to make a 2D model to 3D model .



After Colouring



After cutting



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