

Introduction

This comprehensive tutorial will guide you through the steps of re-creating Jamie-Hubo shells. In this tutorial, we will cover a wide range of topics, including opening 3D model files using Autodesk Inventor, exporting them as step files (.stl), and importing these files into a slicer software. We will also explore the process of generating G-code, which is essential for preparing the models for 3D printing. Additionally, we'll provide you with useful tips and insights for successful 3D printing. Furthermore, if you're interested in recreating the models, we'll showcase how to accomplish that using SolidWorks.

Hardware

- 3D printer (Prusa, Creativity, Lulzbot, X-max...)
- 3D printing filament (PLA, PETG, ABS...)

Software

- Autodesk Inventor
- Slicer software (PrusaSlicer or other depending on your 3D printer)
- SolidWorks

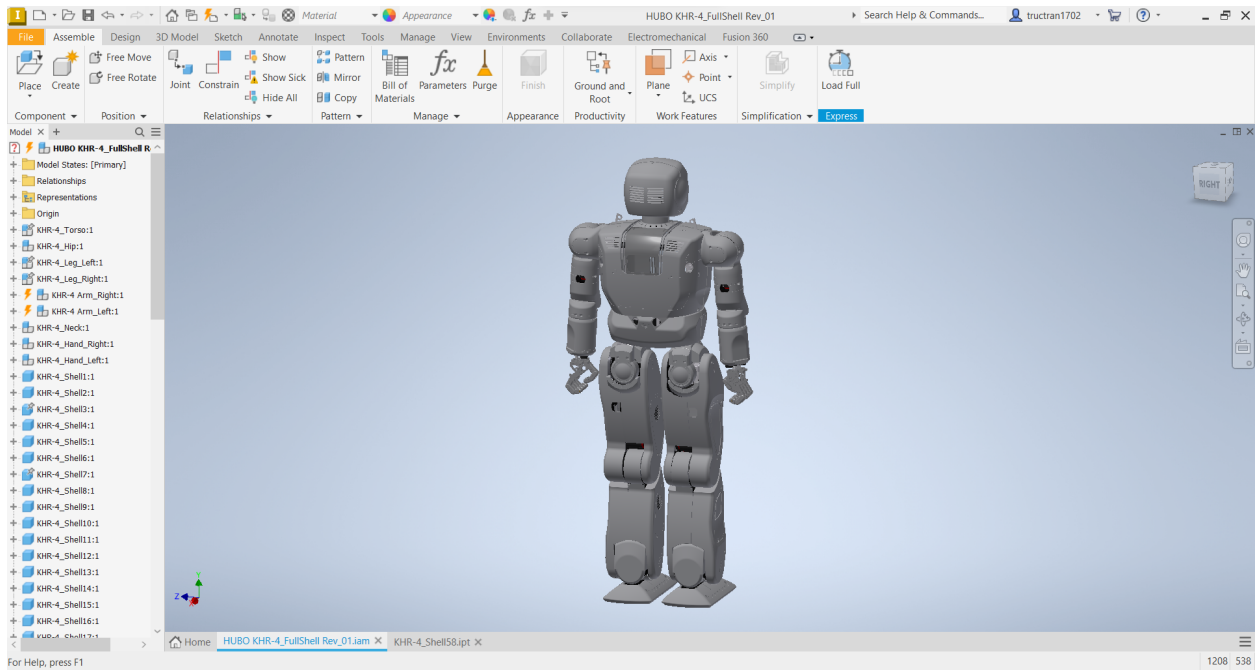
Resources

- Autodesk Inventor for student: <https://www.autodesk.com/education/edu-software/overview?sorting=featured&filters=individual>
- Autodesk Inventor base files: <https://www.dropbox.com/s/bwtxddvrvtx8anl/HUBO.zip?dl=0>

Procedures

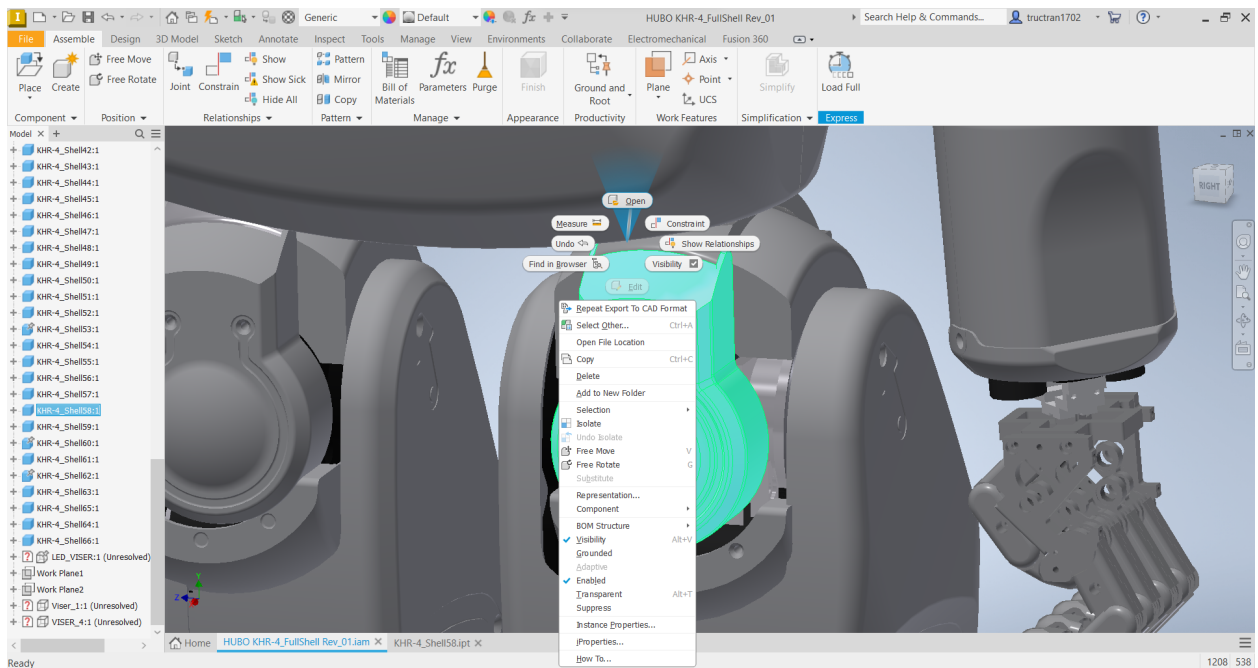
Step 1

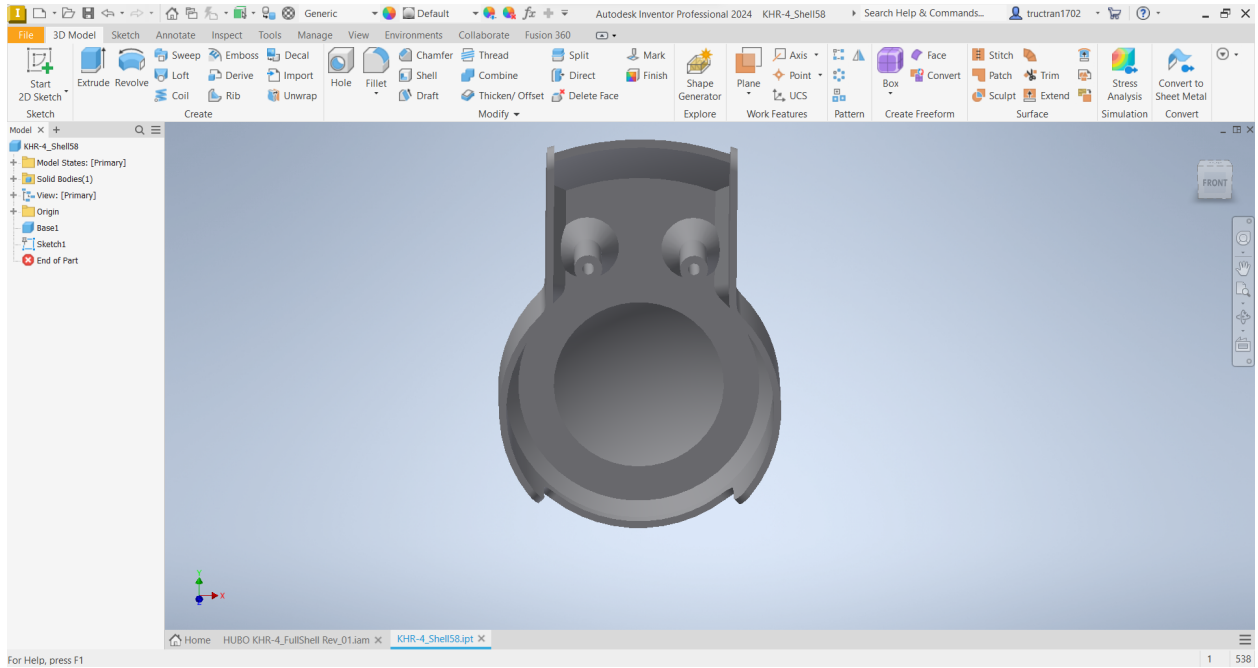
Open the assembly in Autodesk Inventor (..\HUBO\HUBO\Inventor\Finish\Shell Assembly\HUBO KHR-4_FullShell Rev_01.iam). There are 66 parts of the entire shell assembly.



Step 2

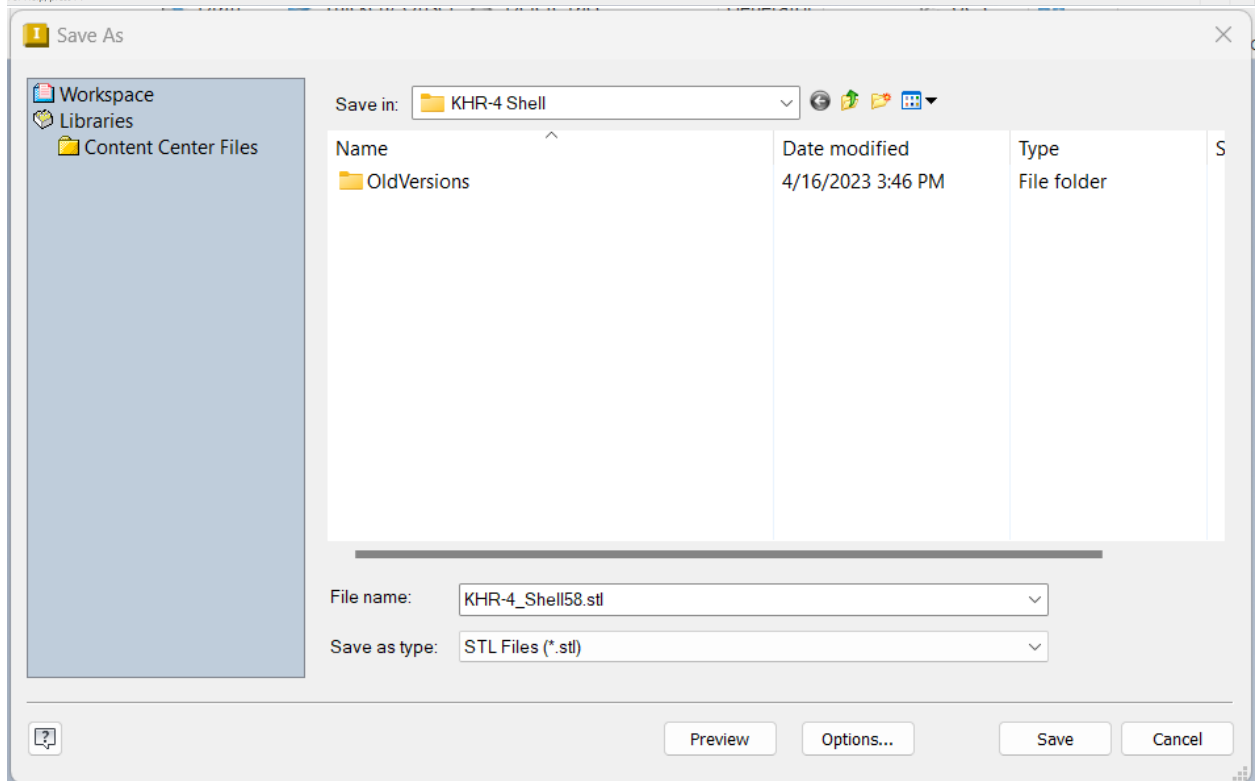
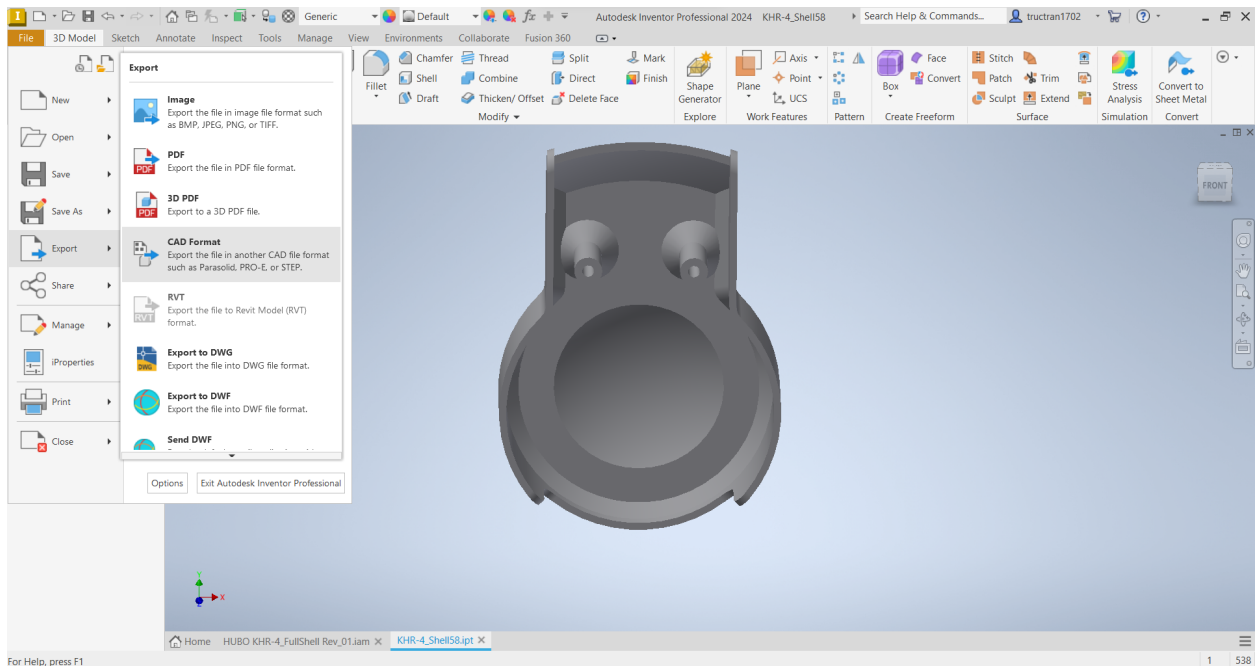
Right click and open the part you would like to 3D print. In this demonstration, we will be using part 58.





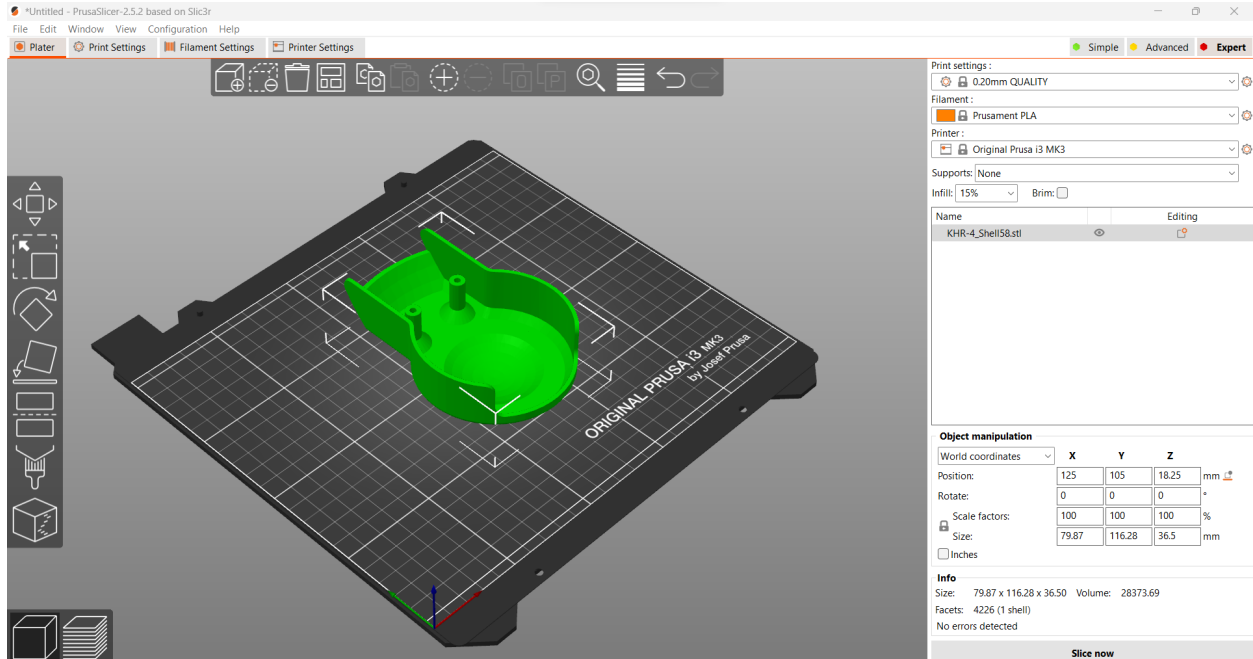
Step 3

On the top right corner, click on File > Export > CAD Format. Save the files as .stl files.



Step 4

Open PrusaSlicer, and drag the .stl files into the slicer software.



Step 5

There are three key slicing settings that are essential for 3D printing with any slicer software. Firstly, we have the filament setting. Common materials include PLA, PETG, ABS, and others. Each material possesses distinct properties such as toughness, flexibility, and melting points. For this particular demonstration, we will be using PETG as our material of choice. The next important setting is printing supports. Ideally, we would prefer to avoid using supports altogether, as they require additional material and often compromise the smoothness of our printed parts. However, in many cases, supports are necessary because 3D printing cannot create structures in thin air. In this demonstration, we will configure the supports to be "Support on build plate only". Lastly, the infill setting determines the internal structure of the printed part and ranges from 0% to 100%. A higher infill percentage results in stronger parts, but it also consumes more material and increases printing time. A commonly used infill percentage for most 3D printed parts is 15%.

Print settings :

0.20mm QUALITY (modified)

Filament :
Generic PETG

Printer :
Original Prusa i3 MK3

Supports: Support on build plate only

Infill: 15% Brim:

| Name | | Editing |
|-------------------|--|---------|
| KHR-4_Shell58.stl | | |

Object manipulation

World coordinates

| | X | Y | Z | |
|----------------|-------|--------|-------|----|
| Position: | 125 | 105 | 18.25 | mm |
| Rotate: | 0 | 0 | 0 | ° |
| Scale factors: | 100 | 100 | 100 | % |
| Size: | 79.87 | 116.28 | 36.5 | mm |

Inches

Info

Size: 79.87 x 116.28 x 36.50 Volume: 28373.69

Facets: 4226 (1 shell)

No errors detected

Slice now

Step 6

Checking the result of the print and remove the support materials.

