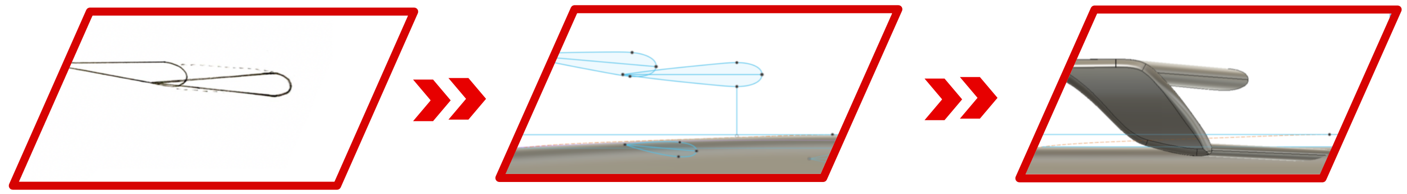
FROM PAPER TO CAD

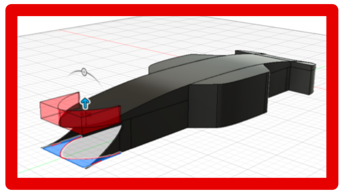
From initial design sketches and concepts, we decided to model two complete cars on Autodesk's Fusion 360 CAD Software, which would then allow us to run both virtual and physical tests on the two designs, to see which one performed better. In order to make the paper-to-CAD transition as easy as possible, rather than using the initial freehand sketches, we produced detailed engineering drawings showing several different perspectives and giving specific dimensions, according to the technical regulations. We were then able to easily reproduce these 2D drawings on Fusion 360, using the sketch function, which facilitated the entire design process.

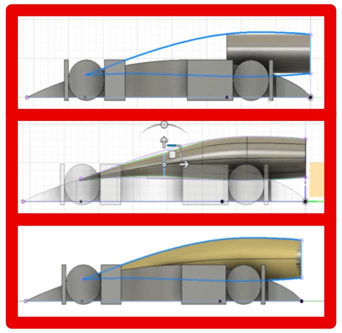
SKETCH EVOLUTION



CAD TOOLS

Much of the CAD was modelled starting from 2D sketches, which were then extruded. This allowed us to build up a rough shape of the car body through both adding and removing shapes, before filleting the edges to soften them, which would help to reduce flow separation.

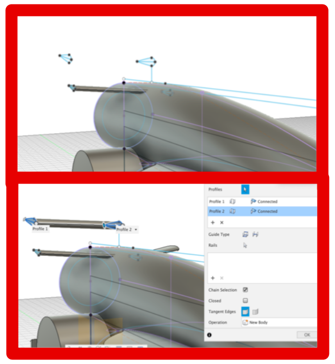


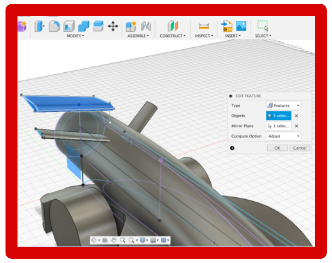


In order to create a smooth enough transition shape between the cartridge chamber and the main body, the fillet tool would not be enough. We realised we would need a more organic shape and therefore had to use the freeform surface modelling tool. We created sketches of the profile we wanted from both the top and side views of the car, which we then used as a guide to model the free-form body to the right shape. The freeform surface started as a cylinder, which was then resized and rescaled at various points, to fit the 'template' sketches.

Once the free-from surface had been created, we used the 'boundary fill' tool to turn it into a solid body and combined it (using the 'combine' tool) with the rest of the main body.

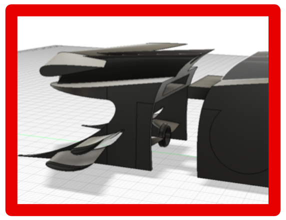
'Loft' also proved to be a very useful tool in the creation of the front and rear wings and nose of the car. Once we had sketched two aerofoil profiles, one on the central plane and the other on an offset plane, we were able to apply the 'loft' tool, which created a transition shape between the two. This produced one half of the wing, which we then mirrored along the central plane to create the complete, perfectly symmetrical aerofoil.

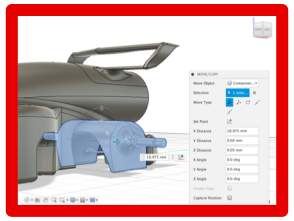




The 'mirror' function was invaluable as it allowed us to ensure that the entire car was perfectly symmetrical. From mirroring sketches and bodies in the main design, to mirroring components into the correct positions, and even in the use of mirror lines in our free-form modelling, the 'mirror' tool was used throughout the entire CAD modelling process.

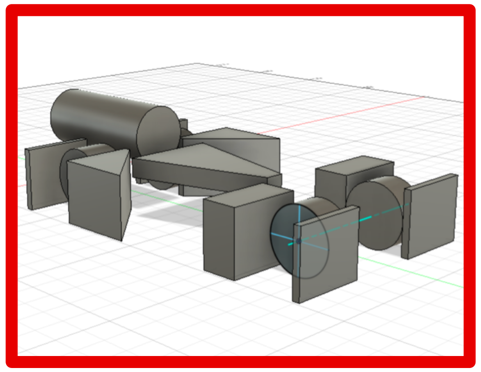
We created our car as one complete body and then separated it into its individual components. This way, we could ensure that each component would transition smoothly onto the rest of the car and also create features such as slots, which would allow for easy and accurate assembly.





We did this by sketching a profile, extending it to create a surface that ran through the car body, and then using the 'split body' tool to cut the body along that surface. We then took the separated part and added a small extrusion on one part, which corresponded to another extrusion on the other part, so that they would slot together perfectly.

ENSURING REGULATION COMPLIANCE



To help ensure our design was fully compliant with technical regulations such as Articles: P4.2, P5.5 and P7.9, we first created temporary components marking areas such as the wheel clearances, virtual cargo, and cartridge chamber safety zone and then modelled our CAD around these. This meant that we had a constant visual representation of the boundaries, whilst realising the design, reducing the risk of accidentally overlooking certain key regulations and having to modify the design later as a consequence.

RENDERINGS AND TECHNICAL DRAWINGS

Having a 3D model of our car also allowed us to very quickly and easily generate high quality renderings and extremely detailed and precise technical drawings.