FINAL PROJECT –

41615 Mechanical CAD Spring 24

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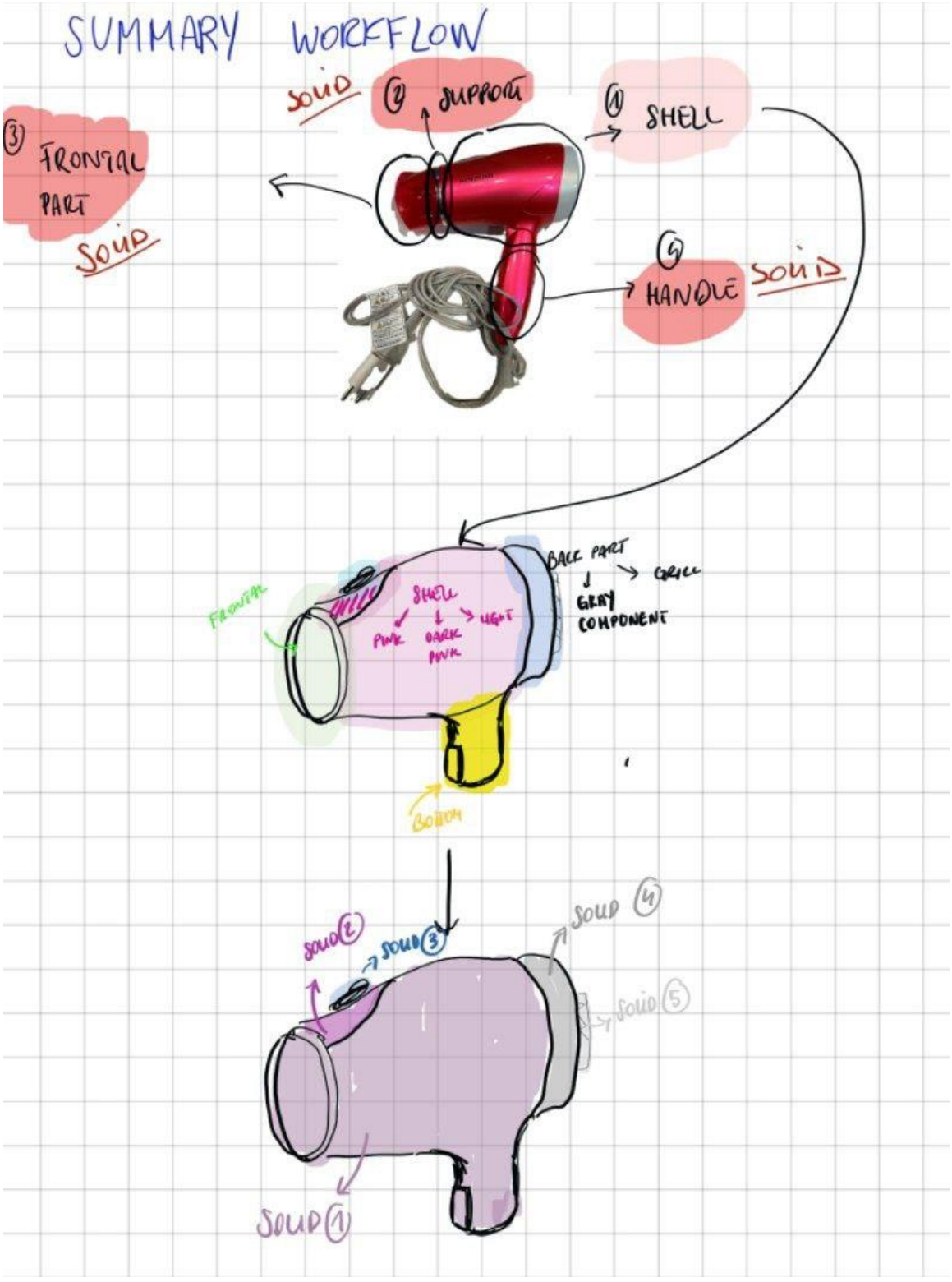

SUMMARY WORKFLOW
SOLID
2 SUPPORT
1 SHELL
3 FRONTAL PART
SOLID
4 HANDLE
SOLID
BACK PART
GRAY COMPONENT
FRONTAL
SHELL
PINK
DARK PINK
LIGHT
BOTTOM
SOLID 2
SOLID 3
SOLID 4
SOLID 5
SOLID 1

Table 1

The goal of the project is to create a CAD model that replicates the **hair dryer** shown in the figure below.

Figure 1

We started by creating the central body, which is the central shell; since the object is symmetrical, we decided to create one half and then mirror it.

Figure 2

To achieve this, we used real images of the object as reference points, so we could have a guideline. To set the images to the correct dimensions, the front image was first loaded onto the front plane, and then a reference line was drawn on the right plane to indicate the length, allowing us to adjust the image size.

Once the first image was positioned, the others were adapted accordingly, using the first one as a reference.

Below are the steps followed to create the central shell.

CENTRAL SHELL

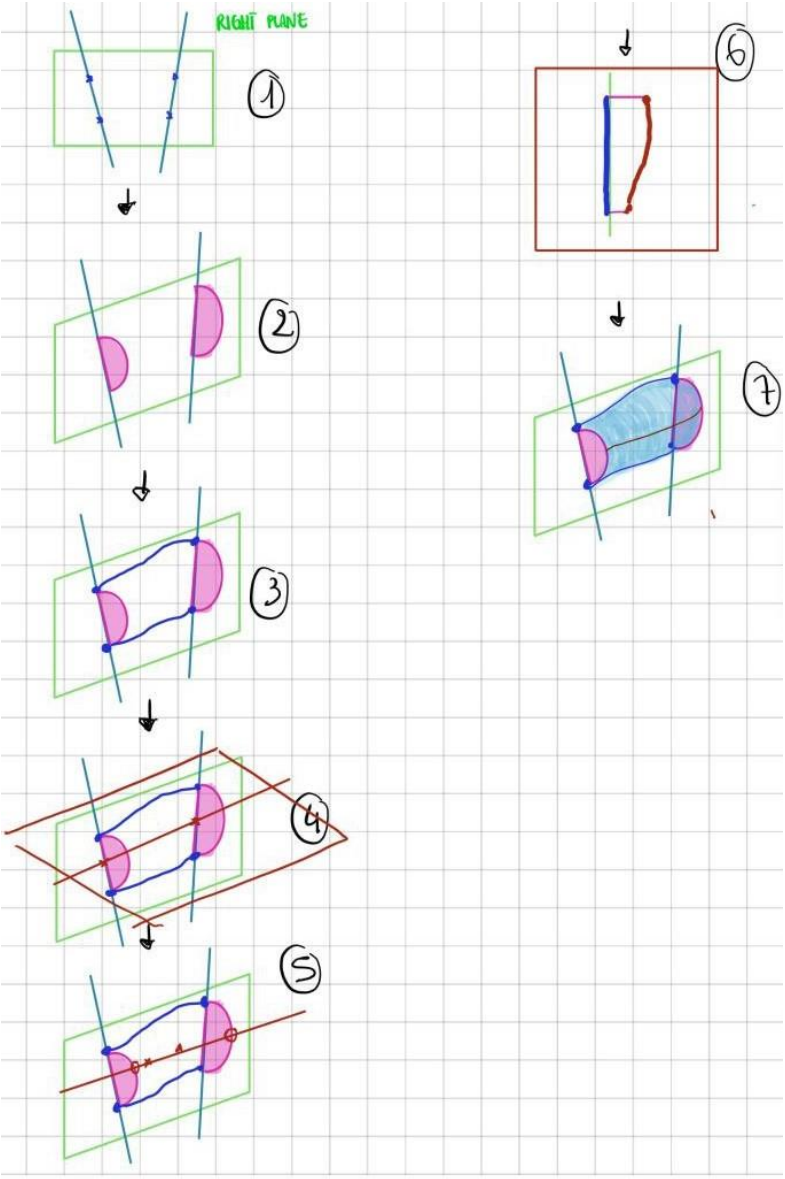
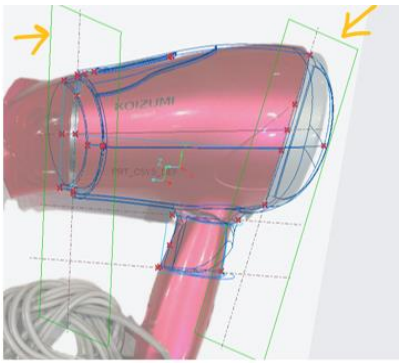


Table 2

- Using reference points, two planes were constructed that delineates the boundary between the central body and the other components (table2.1 – fig.3).



- *Figure 3 - (table2.1)*

- On these planes, two semicircles were defined to serve as the base for constructing the shell (table2.2).
- These two semicircles were then connected by two freeform curves at first (table2.3 -fig. 4).

Figure 4 - (table2.3)

To successfully define the third lateral curve connecting the semicircles, it was necessary to establish two points on the semicircles, and the following steps were taken to define them:

- Defined the center of the semicircles (table2.4).

- Defined the axis passing between the two points (which is used later) (table2.4).

- Defined the plane passing through the two points and perpendicular to the right plane, and thus perpendicular to the surface of the two circles. (table2.4)-(table2.5)

By doing so, the two points are given by the intersection between the plane formed and the semicircles (table2.5).

On that same plane, the top view image was then loaded to aid in the drawing of the lateral curve(table2.6).

At this point, the surface of the shell was created (table2.7).

Figure 5 - (6)

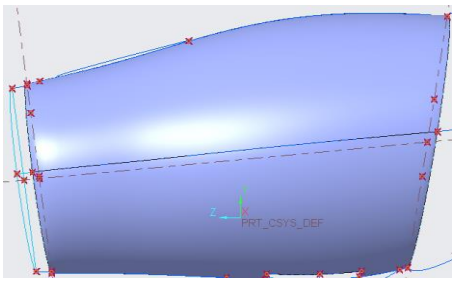


Figure 6 - (table2.7)

Figure 7

UPPER PART

At this point, we focused on the upper part, fig.7)

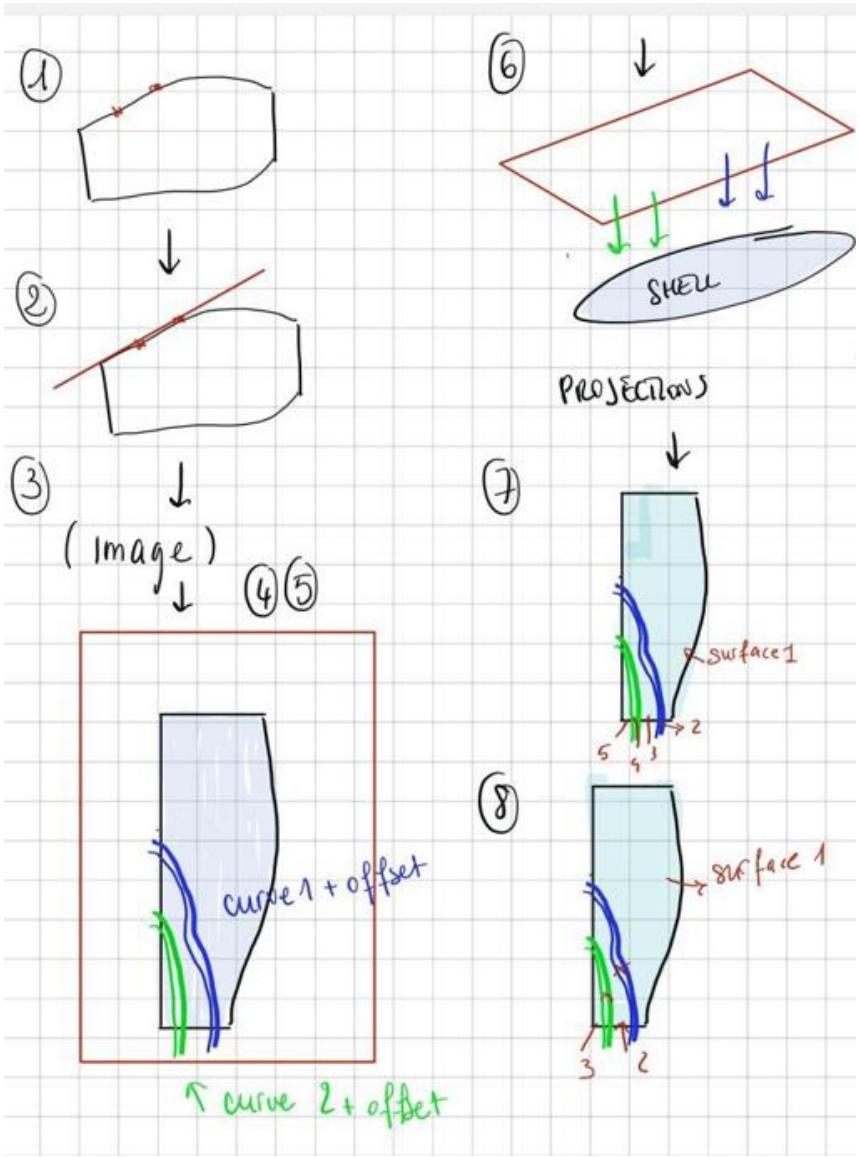


Table 3

The purpose is to divide the shell into three parts, so as to define the sections shown in the figure 7, the following steps were followed (table 3 is defined for the sake of simplicity).

1. Define two reference points that followed the profile of the upper/frontal part (figure 8).

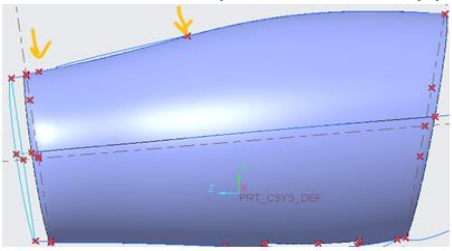


Figure 8

2. Define the plane passing through these points and perpendicular to the right plane.

3. Loaded an image of the top part of the hair dryer (to get a more realistic view).

Figure 9

4. On the previously defined plane, outline the profile curves of the two components.

5. Set an offset for both curves of 0.2mm.

6. Projected all the curves (4 in total: 1 for the light + offset, 1 for the pink part + offset).

7. Divided the shell into several sub-surfaces referring to the curves.

8. Removed the gap surfaces.

→ Applied thickness to the central part.

For the creation of the shell, we followed various methods that would allow us to achieve a volume more similar to that created with various curvatures, but we encountered several problems.

1. *The initial idea was to: Create the total surface → project the curve to be able to trim it more externally (as done before) and then remove the upper surface → define a new curve on the right plane that follows the lateral profile of the object and then define the new surface → project the second curve and trim → create a new curve on the right plane and close the surface.This subsequently caused problems with thickening.*

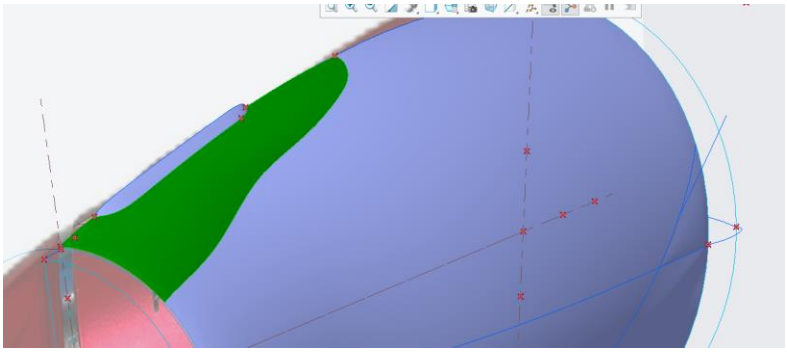


Figure 10

1. *As well, even in our method it didn't work as intended. We have tried to divide the surfaces in order to thicken them with different thicknesses, and afterward use the round feature to model the components, but it didn't work. After several attempts, we opted to leave the thickness of the shell uniform to avoid problems with the creation of other parts.*

ATTACHMENT FOR THE SPOUT

We then proceeded with the creation of the attachment for the spout.

Figure 11

1. We started by using the front semicircle and made an inward offset.
2. An offset plane was created relative to the previously inclined plane, on which we projected the offset front semicircle (red arrow fig. 12).
3. The offset semicircles were then connected with two curves on the right plane (yellow arrows fig. 12).
4. The surface was created

➔ Thicken (fig. 13).

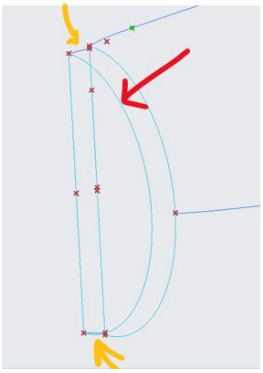


Figure 12

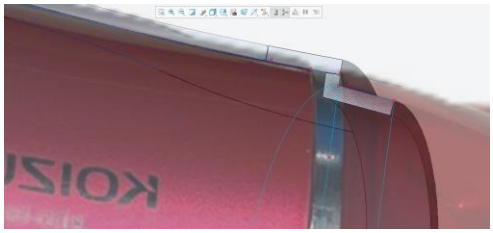


Figure 13

CONNECTION WITH THE HANDLE

We then moved on to the creation of the lower part, which is the connection with the handle.

Figure 14

This was perhaps the most difficult part to create, and in the end, we decided to follow these steps:

1. Create boundary surfaces.

- First, we created reference points → a surface passing through the two points and perpendicular to the right plane.
- Define a semicircle in that plane(yellow arrow, fig.15).
- Project the semicircle onto the surface (red arrow, fig.15).

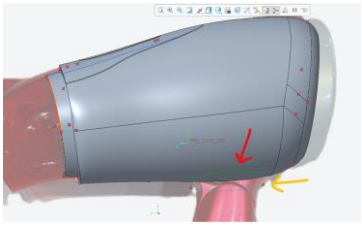


Figure 15

- Trim the surface fig.16.

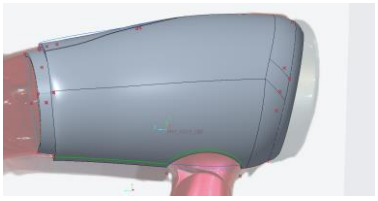


Figure 16

- Created another offset plane relative to the previous one where the spline curve was defined (yellow arrow fig. 17).
- Connected the ends of the spline to the main body (blue arrows fig. 17).

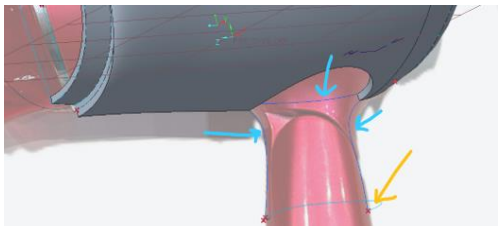


Figure 17

2. Solid creation

- Surfaces were defined using the previously created curves and also utilizing the edges of the object's body.
- Solidification.

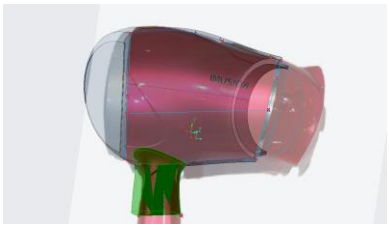


Figure 18

3. Solid modeling

- Creation of the lateral profile was achieved by creating a solid that followed the profile and then removing material.

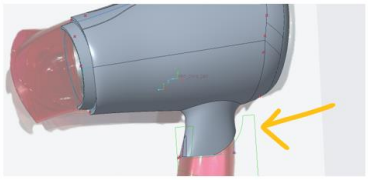


Figure 19

- Creation of the frontal profile was similarly done using the extrusion of a sketch.

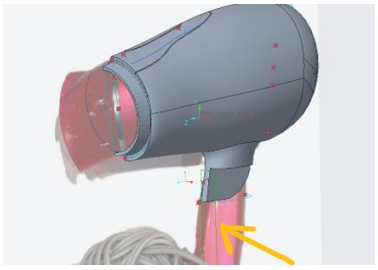


Figure 20

- Another extrusion was performed to create the frontal profile.

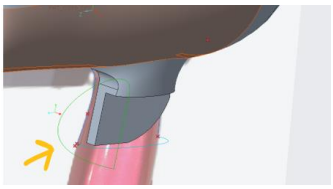


Figure 21

- A rounding operation was applied to create a smoother solid.

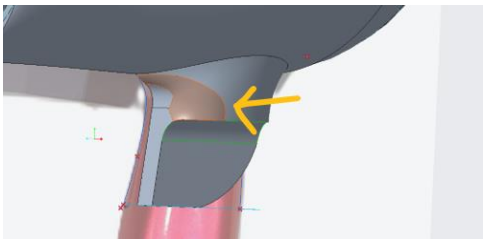


Figure 22

BACK GRAY PART

Before proceeding with the construction of the rear part, what we did was mirror the hair dryer that had just been created. This was because we noticed retrospectively that, we had built half of the rear part and mirrored it along with everything else, we didn't understand the exact reason for the error, since the surface that was to be extruded was perpendicular to the right plane (the one on which the mirroring was then performed). However, to overcome the problem, we decided to take this approach. Thus, once mirrored on the rear plane created in the initial stages, the process was as follows:

- Sketched a design that followed the rear circle → extrude → round.

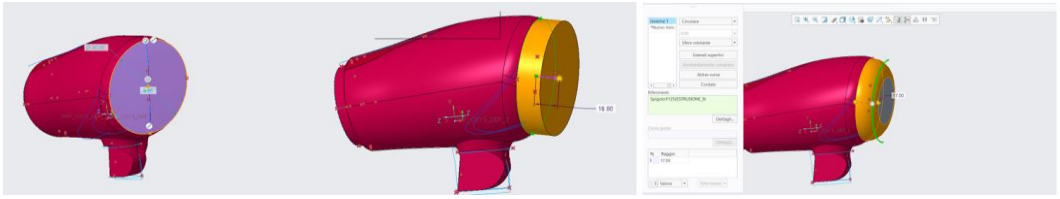


Figure 23

- Then, a sketch was created to remove part of the rear, providing a flat base for the next step.

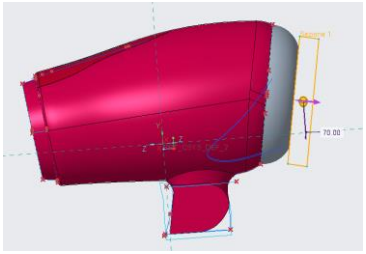


Figure 24

GRILL

Once the rear support base was created, we proceeded with the creation of the grill. The difficult part, in this case, was not creating the pattern but managing to create a curved sheet; we tried with a surface, but this did not work because it caused problems with the thickness, so we opted, although not identical to the real object, to create a flat sheet. The steps followed were:

- Created a recess on which the sheet rests, using the size of the circle formed by the previous cut. *The size of the surface seemed to be the right compromise between the actual size of the grill and the actual size of the grey part.*
  (We would have had to increase the section to make it more similar to the real object, but this would have meant reducing the grey part / we could also have increased the thickness of the grey part but we then found no suitable round value to obtain a smooth profile).

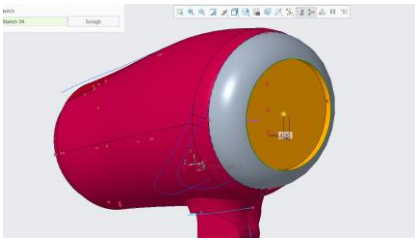


Figure 25

- Created a hole (extrusion removal).

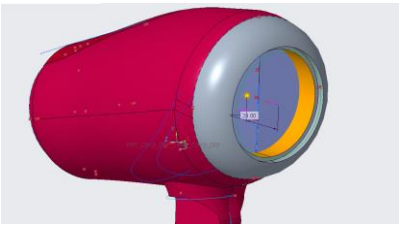


Figure 26

- After creating a reference plane on the support base, the sheet was extruded using a circle sized between that of the hole and that of the outer circle.

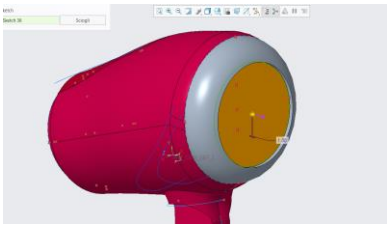
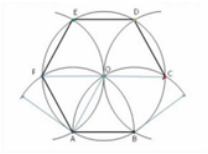


Figure 27

Subsequently, we continued with the creation of the pattern:

- Creation of the hexagonal sketch which we constructed following this pattern.



- Executed two patterns:
  - For the sketch

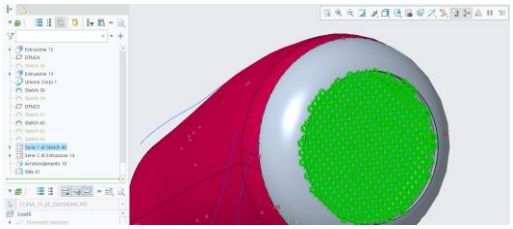


Figure 28

  - For the extrusion

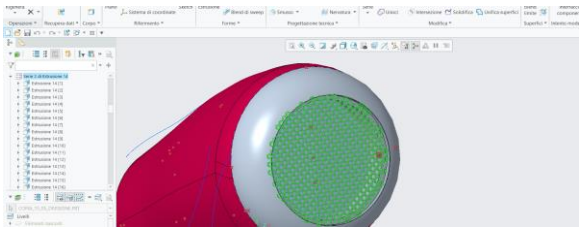


Figure 29

- Small round on the inner edge

GROOVE

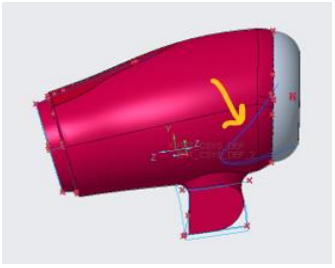


Figure 30

We were unable to create the groove, and we did not understand the problem.

We sketched the curve on a plane parallel to the right plane and projected it onto the usual surface → defined a surface between the curve and the rear edge → thickened but with the material removal mode.

We also tried to trim/divide body-surfaces. Perhaps we should have performed this operation before thickening and separating the surfaces and then thickening again (but we would have encountered the same problem we had with the shell part).

<u>As final we divided all the corpus (pink shell – dark pink shell – gray light – back grill – back gray part so that we were able to change color/materials on 3dmax).</u>

HANDLE

As previously we used a figure as support to define the correct shape and we did half object.

To create the handle, this time, instead of using surfaces we tried to solve the problem by using sketch+extrusion, *since we faced various problems by using surfaces*.

We proceed with the following steps:

- Skech + extrusion.

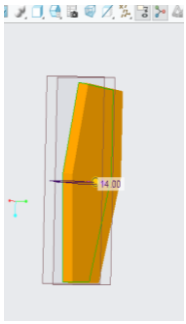


Figure 31

- Rounding.

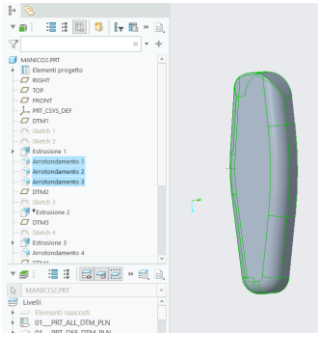


Figure 32

- Using a datum plane, we remove the internal part and create the upper hole.
- Using the previous datum plane to define the bottom hole → round the corners (fig.33).

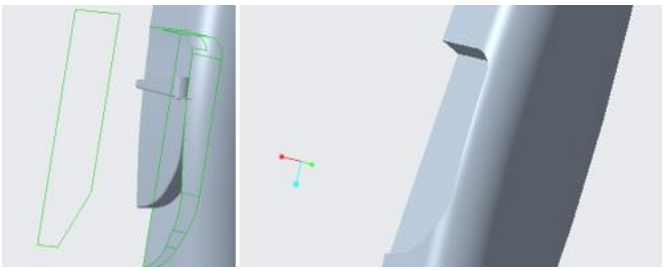


Figure 33

- Using a datum plane create the bottom.

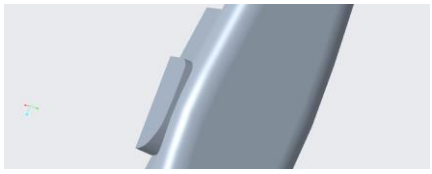


Figure 34

- Project the handle profile on the datum plane and create a sketch to remove the exceeded material.

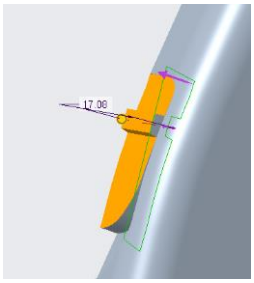


Figure 35

- Project the handle profile on a datum plane and create a sketch to remove the exceeded material to obtain the frontal shape → round.

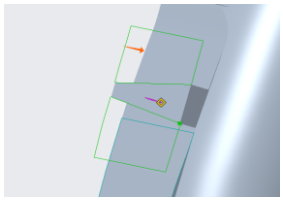


Figure 36

FRONTAL CIRCLE

To obtain the frontal circle we created a circle surface and thickened it afterwards.

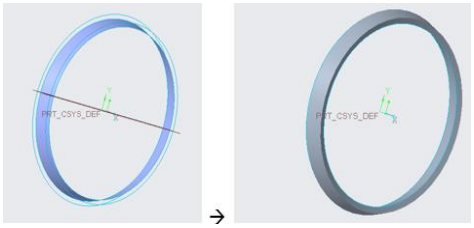
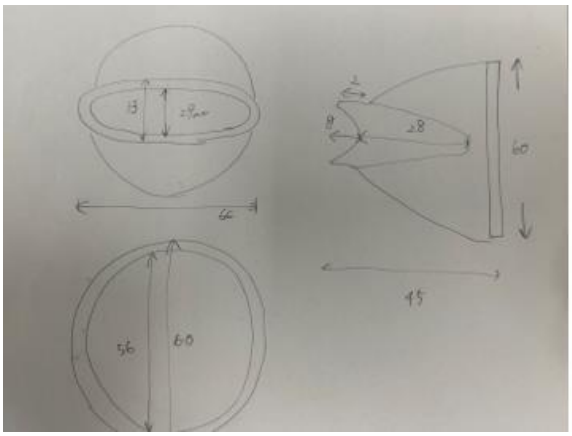


Figure 37

FRONTAL PART – SPOUT

Figure 38

To start modeling the front part, we sketched all of the dimensions. We made the datum plane to make the inclined line.



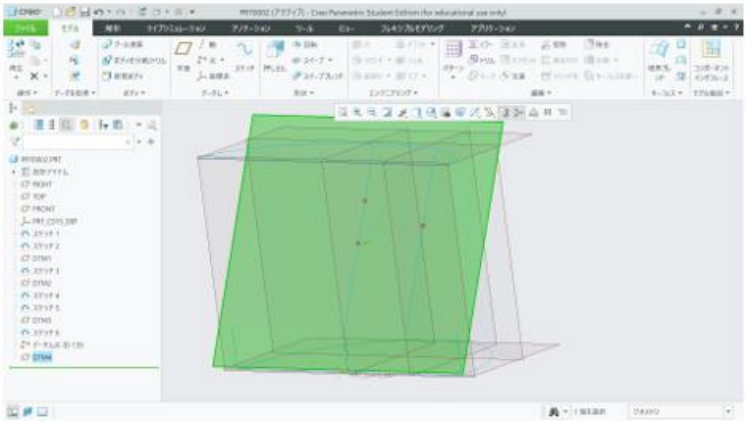


Figure 39

After creating the inclined lines, we used the surface tool to connect all the lines. We ran into difficulties making the proper lines.

- We wanted to make the corner of the shape round (circled in red below), so we tried to make the lines by using a lot of Datum planes but ended up not being able to connect the

surface. Finally, we decided to use the one we made, although it is sharper than the real model.

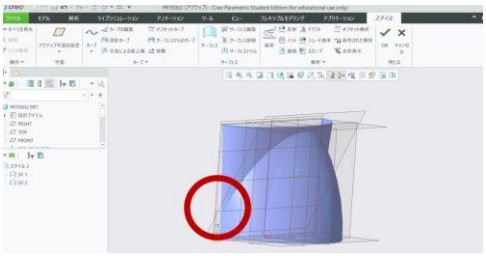


Figure 40

- We used the reflecting sketch + trimming functions to make the upper parts round.

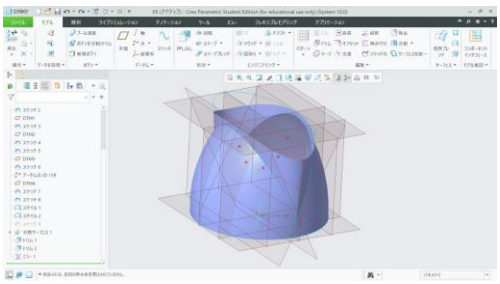


Figure 41

ASSEMBLY

As final we assembled everything, it was a bit tricky since not all the surfaces were perfectly matching but I managed to assemble it as precisely as possible.

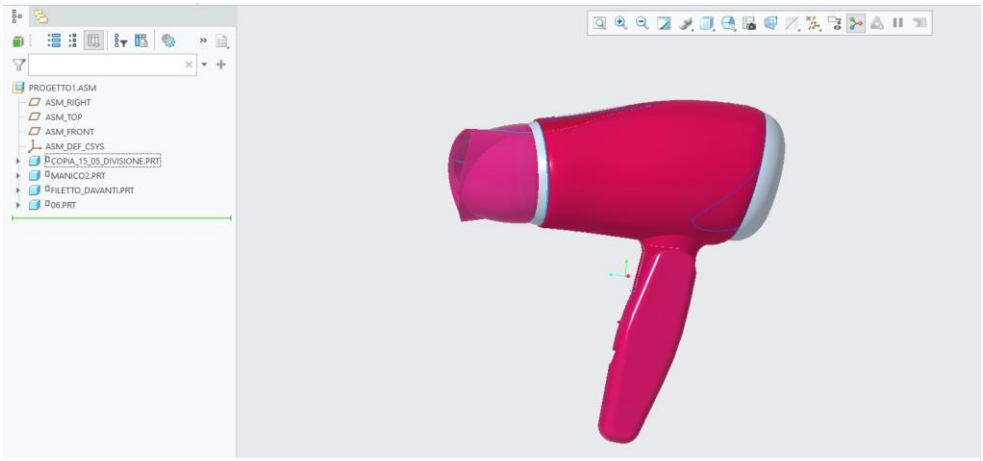


Figure 42

3dsMAX

After importing the creo file, we changed the materials and colours by using the slate material editor. To make the poster, we used the HDRI functions for rendering environments, to polish it up. We struggled to find a proper HDRI which matches to the hair dryer, so we placed it next to the sink to create a realistic environment.

When we tried to change the body's colour, it was too shiny, so we had to decrease the roughness. Also, we struggled with finding a proper colour for the front part since it is transparent and a little bit red than the body.

Figure 43

Creo issues:

- Surface management issues: Frequently, surfaces do not connect well, fail to close, or cannot form solids. The system is also unstable, sometimes working and sometimes not, leading to significant chaos with surfaces. Frequently was the time the software stopped working.
- Lack of error information: The software does not provide explanations when something goes wrong, which severely limits troubleshooting and doesn't assist the user in understanding the problem.
- Difficulty in problem decomposition: The most challenging part was breaking down the problem and representing it. Initially, surfaces were used for creating components, but for simpler handling and intuitiveness, the approach was shifted to using sketches and extrusions, like with the handle design.
- Complex construction processes: It is very easy for the overall construction process to become confusing and cluttered.
- Inability to separate surfaces for independent work: There is a limitation in starting with a surface and then dividing it for separate, to be able to for on it separately.

3dMAX issues:

- Difficulty in finding colours and materials: It is very hard for us to find proper colours and materials among a lot of options.
- User interface: Compared to creo, the user interface is difficult to understand. The shortcut key command is different from other software so it was confusing to use.

Lessons learned in Creo:

- Ability to break down complex problems: gained skills in dividing a large problem into manageable parts and addressing them step by step.
- Utilizing creo knowledge to mask unknown errors: developed the ability to use creo to work around minor unknown issues, thus continuing with the project effectively (for example, dealing with the back part of a design).
- Importance of image uploads: found the feature for uploading images to be very beneficial in aiding design processes.
- Managing multiple planes and views: learned how to efficiently manage various planes and views, and the ability to toggle different layers on and off.

Future functionalities to explore in Creo:

- Projections: delve deeper into how to use projections effectively in design.
- Solid division: explore the functionalities related to dividing solids into parts, which could enhance the modeling of complex assemblies.
- Advanced surface handling: start with a surface, divide it, and then use each subdivided surface to create distinct solids. this technique could be particularly useful for more sophisticated designs.
- Focus on tricky surfaces: since surfaces can often be tricky to manage, gaining a deeper understanding of how to manipulate them more effectively in creo could be beneficial.

Lessons learned in 3DMAX

- Importance of detail settings of colours: Main colour is, of course, important, but also other detail settings such as roughness, IOR, reflection, and transparency are very important in making it look exactly like the model.
- Beneficial use of HDRI: It is super easy and effective to use HDRI environment for rendering since it contains perfect light settings and background.

Future functionalities to explore in 3DMAX:

- Vray for 3dsMAX: We wanted to use vray for 3dsMAX since Vray has a lot of features and is able to settle detail rendering settings. However, for our computer, the Vray didn’t work well. It might be because we didn’t have much knowledge about Vray. If we had more time, we might have been able to use Vray for 3dsMAX.