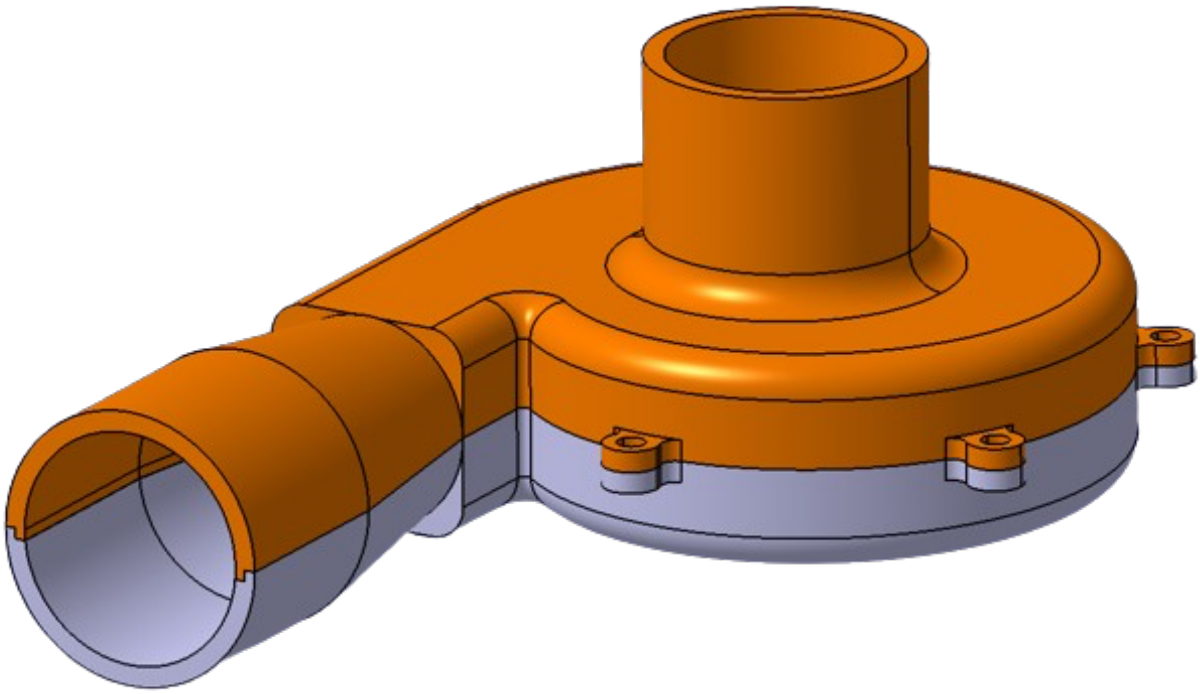
ÉTUDE DE CAS
Centrifugal Compressor



BRUGEROLLE Baptiste
BERDESH Daulet
EYMERY Guillaume
CHUTTARSING Emeryc

ENSAM PARIS
Semestre GIE

2023-2024

Arts et Métiers Sciences et Technologies

TABLE OF CONTENT

TABLE OF CONTENT

I. Introduction

In the pursuit of a sustainable and eco-friendly future, the development of Fuel Cell Electric Vehicles (FCEVs) stands as a groundbreaking solution to address the environmental challenges posed by traditional internal combustion engines. As we begin to understand the urgency of reducing carbon emissions to preserve our planet, hydrogen emerges as an attractive alternative energy source, offering never seen before advantages in its ability to power FCEVs. A part of the success of this revolution of transportation comes from an essential component of FCEVs: the centrifugal compressor, who plays a crucial role in optimizing the efficiency and performance of these vehicles.

As the transition to sustainable transportation becomes a necessity, because of the ever-growing climate crisis and the detrimental impact of conventional fossil fuel-powered vehicles on the environment, FCEVs present a compelling alternative, harnessing the potential of hydrogen as a clean and efficient source of energy. Hydrogen, when used in fuel cells, produces electricity with water vapor as the only emission, making it a zero-emission solution. The development and widespread adoption of FCEVs are thus critical for reducing the environmental impact of transportation and achieving a carbon-neutral future.

While the production of Hydrogen still is not completely carbon-neutral, heavy and steady research is being done to develop better methods for producing green hydrogen. With this philosophy in mind, developing technologies based on Hydrogen is relevant. Indeed, hydrogen as a fuel possesses many advantages over traditional energy sources. Its abundance and clean combustion contribute to its role as a key player in the pursuit of sustainable energy solutions. FCEVs capitalize on these benefits by converting hydrogen into electricity through a chemical reaction with oxygen, powering electric motors to drive the vehicle. This process not only eliminates harmful emissions but also provides a tangible pathway toward energy independence and reduced reliance on finite fossil fuel resources.

The efficiency and overall performance of FCEVs is crucially based on the effectiveness of their components, with the centrifugal compressor emerging as a backbone in this innovative technology. The design of the centrifugal compressor we developed optimizes the process of supplying high-pressure air to the fuel cell, ensuring efficient and continuous power generation. By compressing the incoming air to the required pressure, the centrifugal compressor enhances the performance of the fuel cell stack, maximizing the conversion of hydrogen into electricity. Its design minimizes energy losses and heat generation, contributing to the overall efficiency and longevity of the FCEV.

II. Sizing of the Centrifugal Compressor

In this section, we will focus our attention on how we determined the design parameters of our centrifugal compressor, which shape its geometry and estimate its operating point's rotational speed. The process we followed requires a functional analysis of a centrifugal compressor to determine analytical equations, from which we computed the values of the design parameters using various computing methods.

1. Functional analysis

Air is a compressible fluid, therefore the fluid density ρ is not constant over time, making the Incompressible Fluids theory's assumptions not applicable for our compressor. Thus, we will use thermodynamics equations to avoid discarding ρ, and write a Python program to compute all the values.

The three fundamental equations we will use are the following:

STATE EQUATION

$$P = \rho RT$$

- P: pressure (Pa)
- ρ: fluid density (kg/m^3)
- R: ideal gas constant $(J/mol/K)$
- T: temperature (K)

CONTINUITY EQUATION

$$\dot{m} = \rho A v$$

- $\dot{m}$: mass flow rate (kg/s)
- ρ: fluid density (kg/m^3)
- A: cross-sectional area of the fluid's flow (m^2)
- v: fluid velocity (m/s)

FIRST LAW OF THERMODYNAMICS: ENERGY EQUATION

$$\mathrm{h} + \frac{\mathrm{v}^2}{2} + \mathrm{gz} = \mathrm{cst}$$

- h: specific enthalpy (J)
- v: fluid velocity (m/s)
- g: gravitational acceleration (m/s^2)
- z: altitude (m)

2. Choice of design parameters

a. Design parameters: Inlet

A key element for designing the compressor is determining the inlet angles of the flow's passage. To achieve this, we used the inlet velocity triangle. The height of the inlet part of the blades is also a crucial element we need to calculate.

1. By determining the inlet temperature $T_1 = 291.2081\,K$ using the known absolute inlet velocity $c_1 = 60\,m/s$ and the air's heat capacity $c_p = 1004.5\,J/kg/K$, we can calculate the inlet pressure $P_1 = 0.9906\,bar$ from Laplace equations. Thus, using the state equation, the inlet fluid density $\rho_1 = 1.1848\,kg/m^3$ can be calculated.

2. With a mass flow rate computed from the Fuel Cell's characteristics $\boxed{\dot{m} = 0.0299\,m/s}$, we can determine the inlet cross-section area of the fluid's flow $A_1 = 4.2093 \cdot 10^4\,m^2$ using the continuity equation. With a known peripheral inlet hub radius $r_{1h} = 0.01\,m$, we can now calculate the inlet shroud radius $r_{1s} = 0.0153\,m$.

3. Through isentropic considerations, compressor parameters and using the ideal gas consideration, an approximated real enthalpy variation $\Delta h_0 = 14.3305\,kJ/kg$ is computed. Using the Cordier diagram with a known specific speed $N_s = 0.8$, we can determine the rotational speed of the compressor $\boxed{N = 62961\,rpm}$ at its operating point.

4. Knowing N, r_{1h} and r_{1s}, we can now easily calculate the peripheral inlet hub velocity $u_{1h} = 65.9328\,m/s$ and the peripheral inlet hub velocity $u_{1s} = 100.8554\,m/s$.

With all this knowledge, we can finally determine the inlet velocity triangle. The following angles can be calculated:

- $\boxed{\beta_{1h} = arctan\left(\frac{r_{1h}}{c_1}\right) = 47.6972°}$ the hub inlet flow angle
- $\boxed{\beta_{1s} = arctan\left(\frac{r_{1s}}{c_1}\right) = 59.2511°}$ the shroud inlet flow angle

Finally, the height of the inlet part of the blade can be easily calculated:

- $\boxed{b_1 = r_{1s} - r_{1h} = 5{,}3\,mm}$

b. Design parameters: Outlet

A key element for designing the compressor is determining the outlet angle of the flow's passage. To achieve this, we used the inlet velocity triangle. The height of the outlet part of the blades is also a crucial element we need to calculate.

To determine the outlet angle, we will use the first law of thermodynamics applied to the compressor. By discarding the effect of gravity (small altitude variation) and considering there is no pre-rotation of the compressor (starting position: $u_1 = 0\ m/s$), as well as the efficiency of the compressor (power factor $pF = 1.03$ / slip factor σ), with $v = c_u u$, c_u being the projection of the absolute velocity c onto the peripheral outlet velocity u, we have:

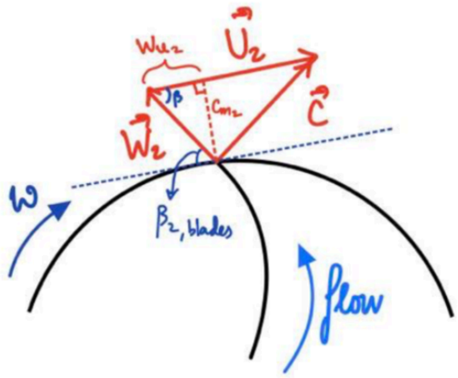
$$\Delta h_0 = pF\sigma c_{u2} u_2$$

We use the Gulich's slip factor, which follows the next equation:

$$\sigma = f_1 \left(1 - \frac{\sqrt{\sin \beta_2}}{Z_{eff}^{0.7}}\right) k_w$$

We have a radial impeller, so $f_1 = 0.98$, and we take $k_w = 1$. There are $Z = 8$ blades in the propeller, with a length of splitting blades compared to the real blades $L_s = 0.5$, we have $Z_{eff} = 12$.

We can now determine c_{u2} as a function of β_2 using the following demonstration:



$$\begin{aligned} c_{u2} &= u_2 - w_{u2} \\ &= u_2 - \frac{c_{m2}}{\tan \beta_2} \\ &= u_2 - c_{m1} \frac{c_{m2}}{c_{m1}} \frac{1}{\tan \beta_2} \\ c_{u2} &= u_2 - c_1 \frac{c_{m2}}{c_{m1}} \frac{1}{\tan \beta_2} \end{aligned}$$

Where we know the absolute inlet velocity $c_1 = 60\ m/s$, the ratio $\frac{c_{m2}}{c_{m1}} = 0.8$.

The peripheral outlet velocity u_2 can be calculated using the Cordier diagram. With a known specific radius $D_s = 3.25$, we can find the outlet diameter D_2 with $\Delta h_0 = 14.3305\ kJ/kg$, $\dot{m} = 0.0299\ m/s$ and $\rho_1 = 1.1848\ kg/m^3$. Using the rotational speed of the compressor $N = 62961\ rpm$, we can now easily calculate u_2.

Thus, we find the following implicit expression:

$$pFf_1 \left(1 - \frac{\sqrt{\sin \beta_2}}{Z_{eff}^{0.7}}\right) k_w \left(u_2 - c_1 \times \frac{c_{m2}}{c_{m1}} \times \frac{1}{\tan \beta_2}\right) - \Delta h_0 = 0$$

While the only variable of this equation is the searched angle β_2, it is not solvable by hand. Thus, we will implicitly solve this equation by running the following Python code, in which we called y the outlet angle, with a given initial value of 30°.

```
def f(y):
    return (u2 - c1 * cm2_1 / tan(np.deg2rad(y))) * u2 \
           * (1 - ((sin(np.deg2rad(y))) ** 0.5) / Zeff ** 0.7) * 0.98 * p_f - dH_e
```

The computed output is the following value:

- $\boxed{\beta_{2,bl} = 67.7373°}$

Using the outlet velocity triangle as well as the outlet thermodynamics variables, we can determine the ratio of the blade's outlet part's height over the outlet diameter $\frac{b_2}{D_2} = 0.0877$. Thus:

- $\boxed{b_2 = D_2 \frac{b_2}{D_2} = 4{,}1\ mm}$

c. Verification

There are some criteria that we need to verify to ensure that the design parameters we find are acceptable. We will focus on two criteria: the Mach number, and the power needed to run the centrifugal compressor.

MACH NUMBER

The Mach number is calculated from the following equation:

$$M = \frac{v}{c}$$

- M: Mach number $(\emptyset)$
- v: fluid velocity (m/s)
- c: sound velocity inside the fluid (m/s)

Using all the previously calculated values, we can compute the Mach number at the shroud inlet M_{1s} and at the outlet M_2:

- $M_{1s} = 0.343$
- $M_2 = 0.291$

Both Mach number are under 0.8 and around or above 0.3, where compressibility becomes significant. This criteria is verified.

POWER

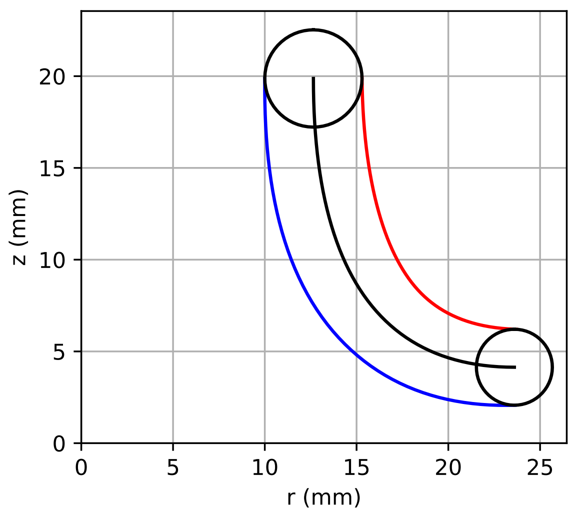
The computed power necessary for running the centrifugal compressor is:

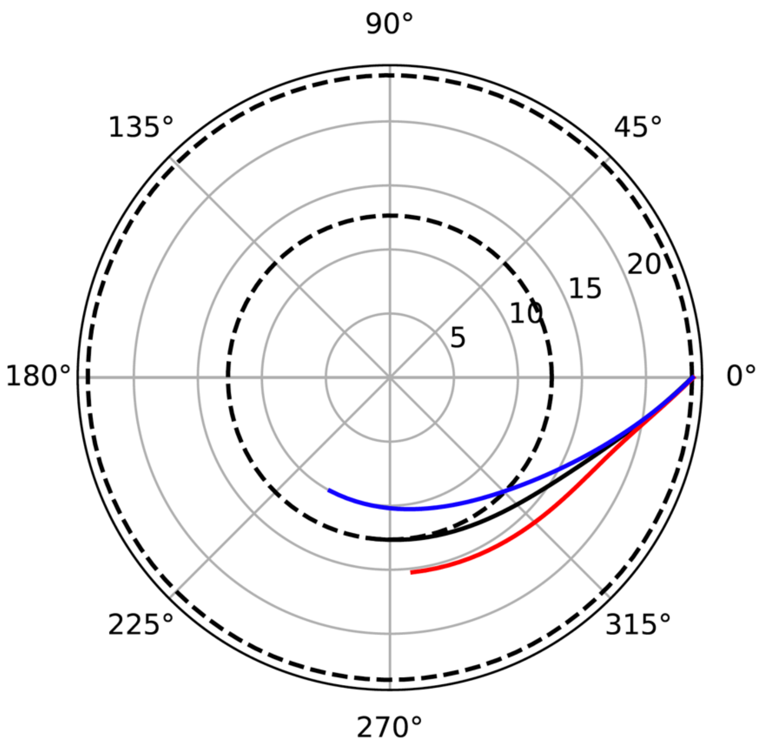
- $P = 0.4514\ kW$

Which is under 1.5 kW, the power of our motor. This criteria is verified.

3. Sizing

Using a Python program that we were given, we were able to compute a meridian (top) and frontal (bottom) view of our compressor with the design parameters we obtained. The result of this section was crucial as it allows a concrete visualization of the centrifugal compressor and will be fundamental for designing the compressor on a CAD software for future CFD simulations and 3D-printing for lab testing.





III. Tank filling

1. Problem overview

The goal of this section is to simulate the filling of a $V_{tank} = 125\text{L} = 0.125\text{m}^3$ tank using H_2 gas with the following initial parameters:

- $T_{ext} = 20°\text{C}$
- $T_0 = 20°\text{C}$
- $\text{P}_0 = 15 \text{ bars}$
- $q_m = 5kg/5min = 1kg/min = 0.0167kg/s$
- $r_{H_2} = \frac{8.314}{0.002} = 4157J/kg/K$

To study the tank's filling simulation, we will compute the temperature and the pressure inside the tank over time.

First, we will try to implement our problem into Python using two distinct methods: *ideal gas considerations* VS *real gas considerations* – achieved by using the CoolProp lib.

We will then compare these two methods by modifying the temperature of the H_2 gas sent to the tank (*inlet temperature T_{in}*) as well as the *heat transfer coefficient h_{ex}* of the tank. We will be able to conclude on the impact of these parameters on the tank's temperature and pressure.

Then, we will try to fill the 125L tank with H_2 at 700 bars under three cases – who still follow the previously mentioned initial parameters:

- $T_{in} = -40°\text{C}$, no external heat transfer
- $T_{in} = -20°\text{C}$, $A = 100W/m^2/K$
- Tank filled in 5 minutes without reaching 60°C

For all three cases, we will compute the filling time as well as the end temperature. For the last case, we will also compute the pressure drop after the temperature inside the tank goes back to the ambient temperature T_{ext}.

2. Implementation of the problem in Python

a. The equations: assumptions

Throughout this first part, we will make the following assumptions on our problem:

- Constant volume inside the tank (rigid walls)
- Homogeneous temperature of the gas inside the tank T_{gas}
- Constant wall heat transfer h_{ex}
- Constant time step used for computing Δt

b. The equations: ideal gas

First, let's use the laws of thermodynamics with ideal gas. The following steps are done:

1. Computing the tank's mass of $\mathbf{H_2}$gas $\mathbf{m}$

m follows this law over time:

$$\frac{dm}{dt} = q_m$$

Since we have a constant H_2gas mass flow rate and a constant time step, we can use the following approximation to easily compute m over time:

$$\frac{m_{t+1} - m_t}{\Delta t} \approx q_m \Rightarrow \boxed{m_{t+1} = q_m \Delta t + m_t}$$

2. Computing the tank's internal energy $\mathbf{U}$

U follows this law over time, with $c_p = \frac{r\gamma}{\gamma - 1}$ and $\gamma = 1.4$ considering H_2 is a diatomic ideal gas:

$$\frac{dU}{dt} = q_m h_{in} - h_{ex} S_{lat}\left(T_{gas} - T_{ext}\right)$$

With $h_{in} = c_p T_{in}$ – ideal gas considerations. Using the same approximation as before, we have:

$$\boxed{U_{t+1} = \left(q_m c_p T_{in} - h_{ex} S_{lat}\left(T_{gas,t} - T_{ext}\right)\right)\Delta t + U_t}$$

With $T_{gas,t}$ being the gas temperature inside the tank at the given time t.

3. Computing the tank's temperature using the Joule-Thomson effect

Since we assumed H_2 is a diatomic ideal gas, we have the following law, with $c_v = \frac{c_p}{\gamma}$

$$U = m c_v T$$

The temperature $T = T_{gas}$ becomes easily computable:

$$\boxed{T = \frac{U}{m c_v}}$$

4. Computing the tank's pressure using the $\mathbf{H_2}$gas density inside the tank $\boldsymbol{\rho}$

Since we assumed H_2 is a diatomic ideal gas, we have the following law:

$$\rho = \frac{P}{rT}$$

Considering the constant volume, we also have:

$$\rho = \frac{m}{V_{tank}}$$

The pressure P becomes easily computable:

$$\boxed{P = \frac{mrT}{V_{tank}}}$$

c. The equations: real gas

Similarly, our end goal is to determine the temperature and the pressure inside the tank over time. This time let's use the laws of thermodynamics without ideal gas consideration, and we will instead be using the CoolProp lib. The following steps are done:

1. **Computing the tank's mass of $\mathbf{H_2}$gas $\mathbf{m}$:**

Same process as the ideal gas process

2. **Computing the tank's internal energy $\mathbf{U}$ and specific internal energy $\boldsymbol{u}$**

Same process as the ideal gas process, except we compute h_{in} using CoolProp:

$$h_{in} = PropsSI(P_t; T_{in})$$

With P_t the pressure inside the tank at the given time t. We also easily compute u:

$$\boxed{u = \frac{U}{m}}$$

3. **Computing the $\mathbf{H_2}$ gas density inside the tank $\boldsymbol{\rho}$**

Using the constant volume, we easily compute ρ:

$$\boxed{\rho = \frac{m}{V_{tank}}}$$

4. **Computing the tank's temperature using CoolProp, $\boldsymbol{\rho}$ and $\boldsymbol{u}$**

We compute T using CoolProp:

$$\boxed{T = PropsSI(u; \rho)}$$

5. **Computing the tank's pressure using CoolProp, $\boldsymbol{\rho}$ and $\boldsymbol{u}$**

We compute P using CoolProp:

$$\boxed{P = PropsSI(u; \rho)}$$

d. Python implementation

Writing a python program following the previous steps, we have these two scripts found in the next page, that are both limited to run the simulations up to 5 minutes. The first one is for the ideal gas simulation, the second one is for the real gas simulation.

```
Vec_TimePF = [0]
Vec_mPF = [m0]
Vec_PPF = [P0]
Vec_TPF = [T0]
Vec_UPF = [m0 * Cv * Tin]

while Vec_TimePF[-1] <= 5 * 60:
    Vec_TimePF.append(Vec_TimePF[-1] + dt)
    Vec_mPF.append(qm * dt + Vec_mPF[-1])
    Vec_UPF.append((qm * Cp * Tin - A * Slat * (Vec_TPF[-1] - Text)) * dt + Vec_UPF[-1])
    Vec_TPF.append(Vec_UPF[-1] / Cv / Vec_m[-1])
    Vec_PPF.append(Vec_TPF[-1] * r * Vec_mPF[-1] / Vtank)
```

```
Vec_Time = [0]
Vec_m = [m0]
Vec_P = [P0]
Vec_T = [T0]
Vec_U = [m0 * Cv * Tin]
Vec_u = [Vec_U[-1] / Vec_m[-1]]

while Vec_Time[-1] <= 5 * 60:
    Vec_Time.append(Vec_Time[-1] + dt)
    Vec_m.append(qm * dt + Vec_m[-1])
    Vec_U.append((qm * CP.PropsSI("H", "P", Vec_P[-1], "T", Tin, "H2") - A * Slat * (Vec_T[-1] - Text)) * dt + Vec_U[-1])
    Vec_u.append(Vec_U[-1] / Vec_m[-1])
    rho = Vec_m[-1] / Vtank
    Vec_T.append(CP.PropsSI("T", "U", Vec_u[-1], "D", rho, "H2"))
    Vec_P.append(CP.PropsSI("P", "U", Vec_u[-1], "D", rho, "H2"))
```

3. Impact of temperature and heat transfer coefficient on both gas models

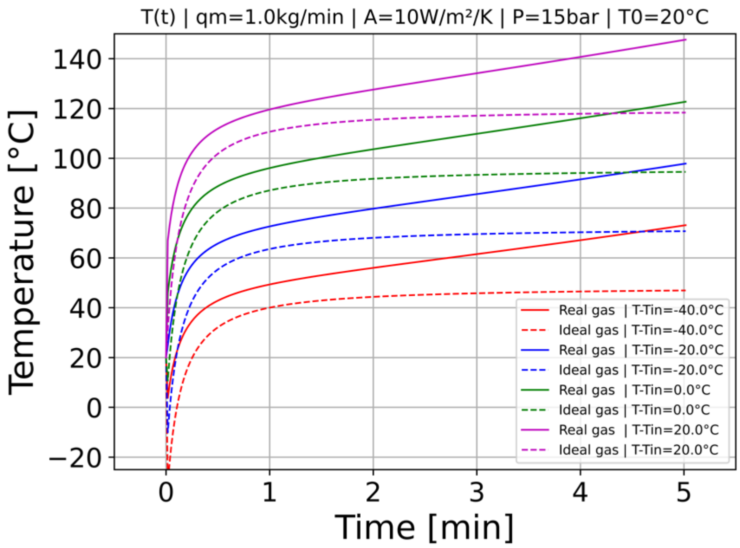
Using the two scripts up above, we were able to easily plot, for both ideal and real gas models, the temperature and pression of the tank over time. We used the following inlet temperatures:

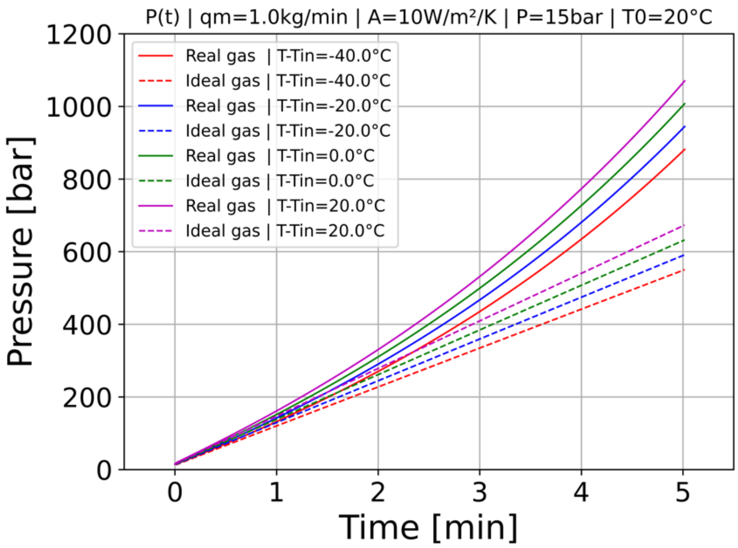
- $T_{in} = -40°C$
- $T_{in} = -20°C$
- $T_{in} = 0°C$
- $T_{in} = 20°C$

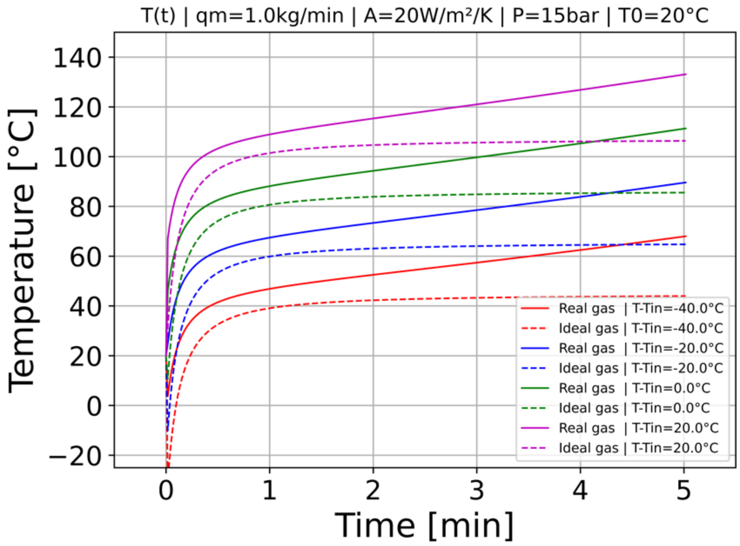
All these inlet temperatures were also computed using the following heat transfer coefficients – in the Python scripts, h_{ex} is called A

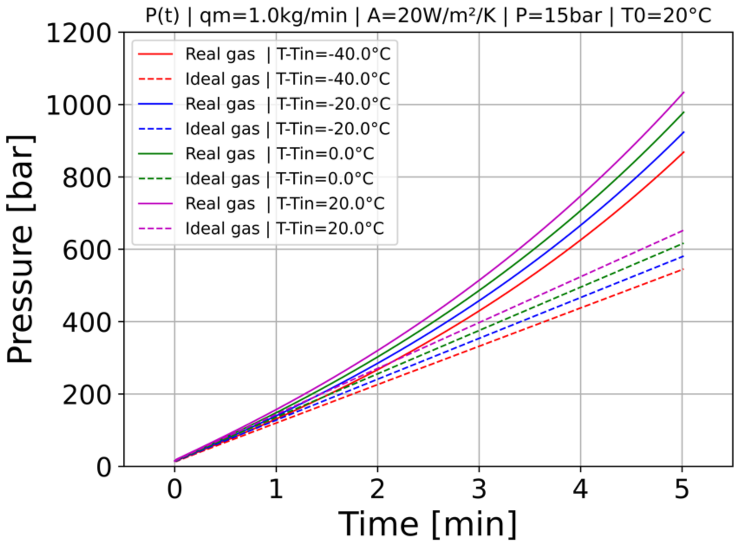
- $A = 10W/m^2/K$
- $A = 20W/m^2/K$
- $A = 50W/m^2/K$
- $A = 100W/m^2/K$

Here are the following plots:

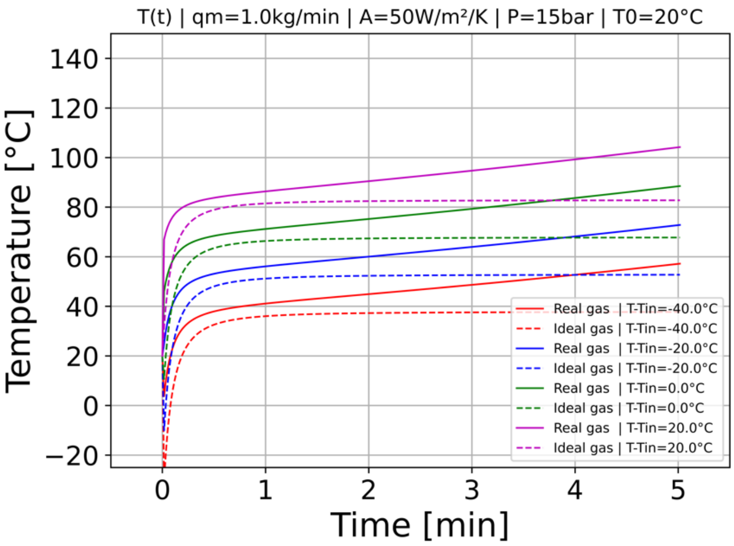

T(t) | qm=1.0kg/min | A=10W/m²/K | P=15bar | T0=20°C
Temperature [°C]
Time [min]
Real gas | T-Tin=-40.0°C
Ideal gas | T-Tin=-40.0°C
Real gas | T-Tin=-20.0°C
Ideal gas | T-Tin=-20.0°C
Real gas | T-Tin=0.0°C
Ideal gas | T-Tin=0.0°C
Real gas | T-Tin=20.0°C
Ideal gas | T-Tin=20.0°C

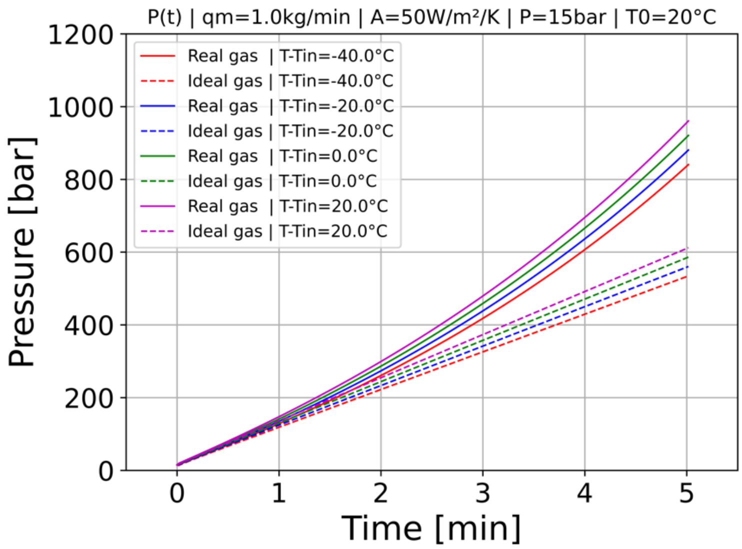

P(t) | qm=1.0kg/min | A=10W/m²/K | P=15bar | T0=20°C
Pressure [bar]
Time [min]
Real gas | T-Tin=-40.0°C
Ideal gas | T-Tin=-40.0°C
Real gas | T-Tin=-20.0°C
Ideal gas | T-Tin=-20.0°C
Real gas | T-Tin=0.0°C
Ideal gas | T-Tin=0.0°C
Real gas | T-Tin=20.0°C
Ideal gas | T-Tin=20.0°C

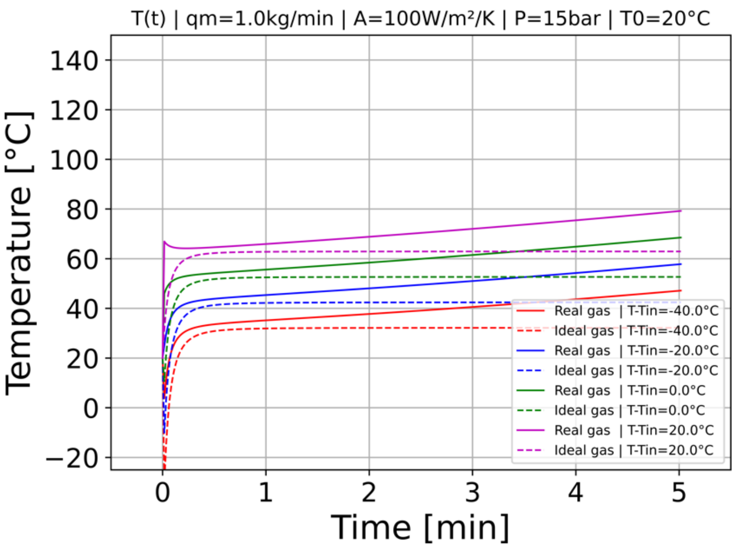

T(t) | qm=1.0kg/min | A=20W/m²/K | P=15bar | T0=20°C
Temperature [°C]
Time [min]
Real gas | T-Tin=-40.0°C
Ideal gas | T-Tin=-40.0°C
Real gas | T-Tin=-20.0°C
Ideal gas | T-Tin=-20.0°C
Real gas | T-Tin=0.0°C
Ideal gas | T-Tin=0.0°C
Real gas | T-Tin=20.0°C
Ideal gas | T-Tin=20.0°C

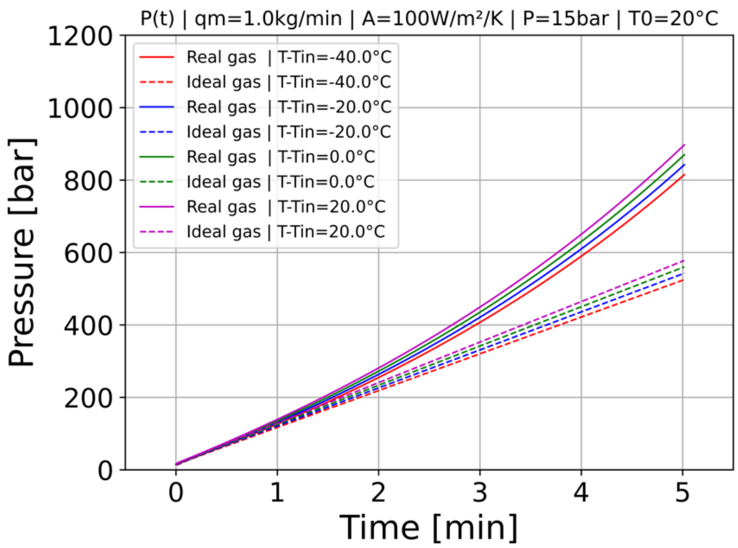

P(t) | qm=1.0kg/min | A=20W/m²/K | P=15bar | T0=20°C
Pressure [bar]
Time [min]
Real gas | T-Tin=-40.0°C
Ideal gas | T-Tin=-40.0°C
Real gas | T-Tin=-20.0°C
Ideal gas | T-Tin=-20.0°C
Real gas | T-Tin=0.0°C
Ideal gas | T-Tin=0.0°C
Real gas | T-Tin=20.0°C
Ideal gas | T-Tin=20.0°C

TANK FILLING









From these plots we can conclude that:

- With all other parameters fixed, an increase in the inlet temperature causes an increase in temperature and in pressure inside the tank.
- With all other parameters fixed, an increase in the heat transfer coefficient causes a decrease in temperature and in pressure inside the tank.

Considering the laws used in the ideal gas model, those results make perfect sense. For the temperature increase, using the perfect gas formula, we have $Pv = rT$, so if the temperature increases, the pressure increases also. For the heat transfer coefficient, an increase means a decrease in the internal energy of the system because of the convection term in the law, which causes a decrease in temperature, which using the perfect gas formula explains a decrease also in pressure.

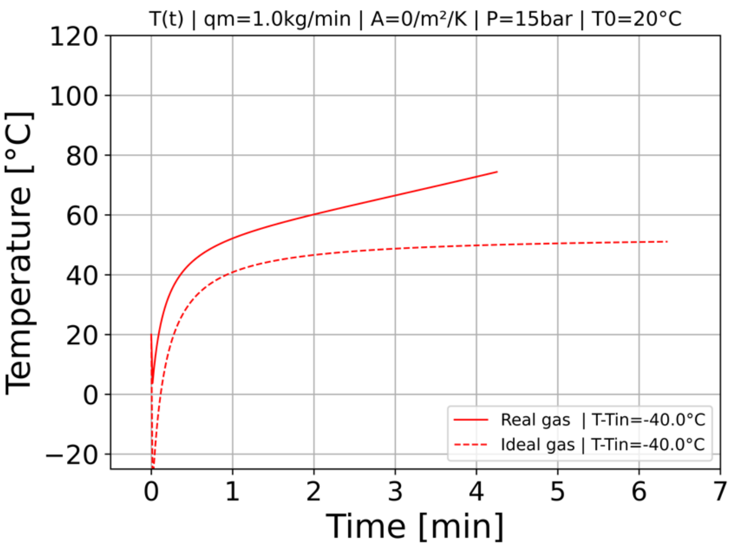
4. Filling simulation

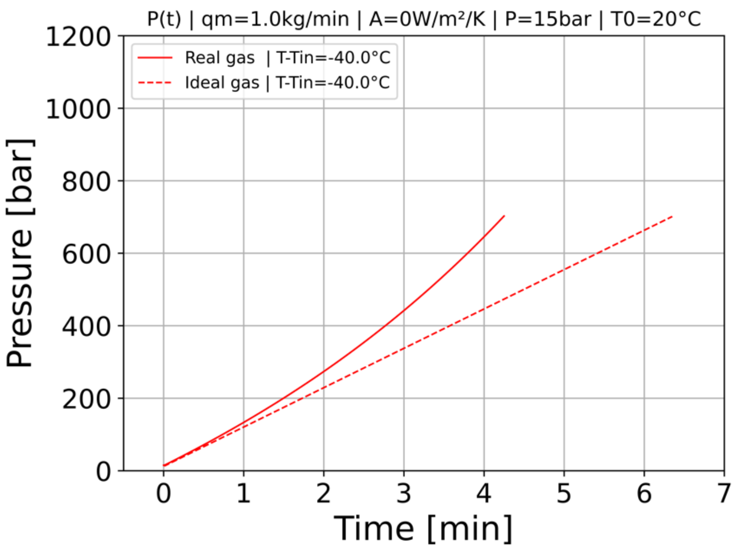
a. Case 1

The first case states that there is no external heat transfer, so the internal energy becomes:

$$U_{t+1} = q_m h_{in} \Delta t + U_t$$

Modifying our scripts accordingly, we have the following plots:



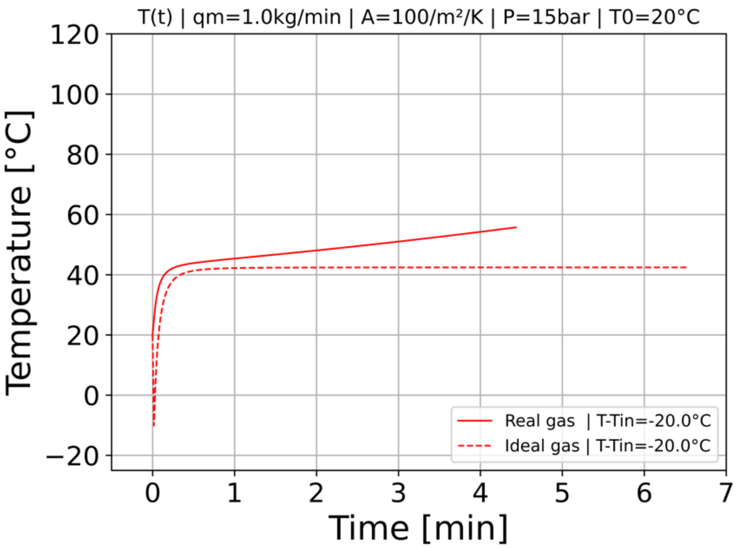


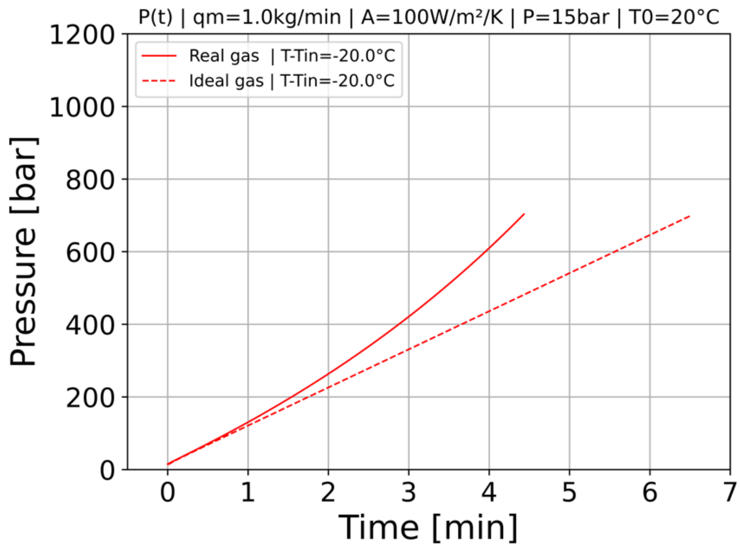
The following values were computed:

- $t_{real\ gas} = 255s$
- $T_{real\ gas} = 74℃$
- $t_{ideal\ gas} = 381s$
- $T_{ideal\ gas} = 51℃$

b. Case 2

The second case requires no changes in the equations found in the part before. Modifying our scripts accordingly, we have the following plots:





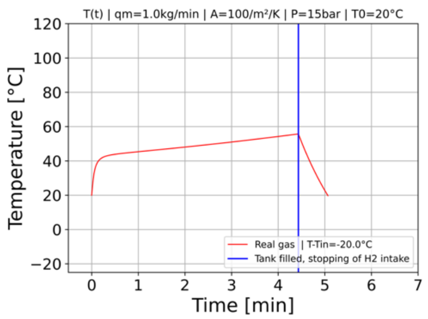
The following values were computed:

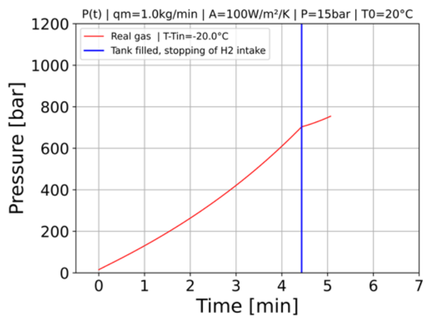
- $t_{real\ gas} = 266s$
- $T_{real\ gas} = 55℃$
- $t_{ideal\ gas} = 391s$
- $T_{ideal\ gas} = 42℃$

c. Case 3

For this case, we will only focus on the real gas simulation. The parameters of the previous case already give us a simulation that meets the requirements. This time, instead of stopping the simulation, we will stop the H_2 intake and let the simulation run until the temperature goes back to T_{ext}. The stopping of H_2 intake changes the internal energy equation to:

$$U_{t+1} = -h_{ex}S_{lat}\left(T_{gas,t} - T_{ext}\right)\Delta t + U_t$$





The following values were computed:

- $t_{filling} = 266s$
- $t_{cooling} = 38s$
- $P_{end} = 754\ bars$

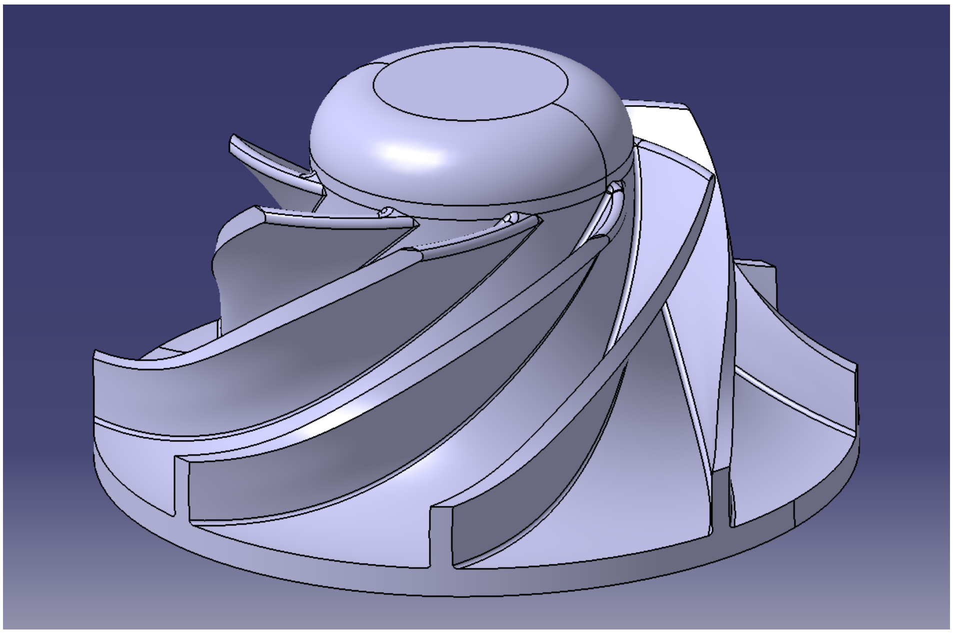
Notice that the pressure keeps increasing even after the H_2 intake has been stopped, which means we must be careful when making the specifications of the tank.

IV. 3D CAD of the Centrifugal Compressor

The 3D design of our centrifugal compressor is critical for two reasons. Firstly, we will be able to 3D-print our centrifugal compressor using the CAD file to test its efficiency in a wind tunnel. Lastly, the CAD file can be imported into a CFD software to simulate the thermodynamic variables, the velocity, and the isentropic efficiency of the compressor over time. For these reasons, great focus was made to obtain the best 3D design possible. To make this CAD, we used the CATIA V5 Software.

1. Rotor

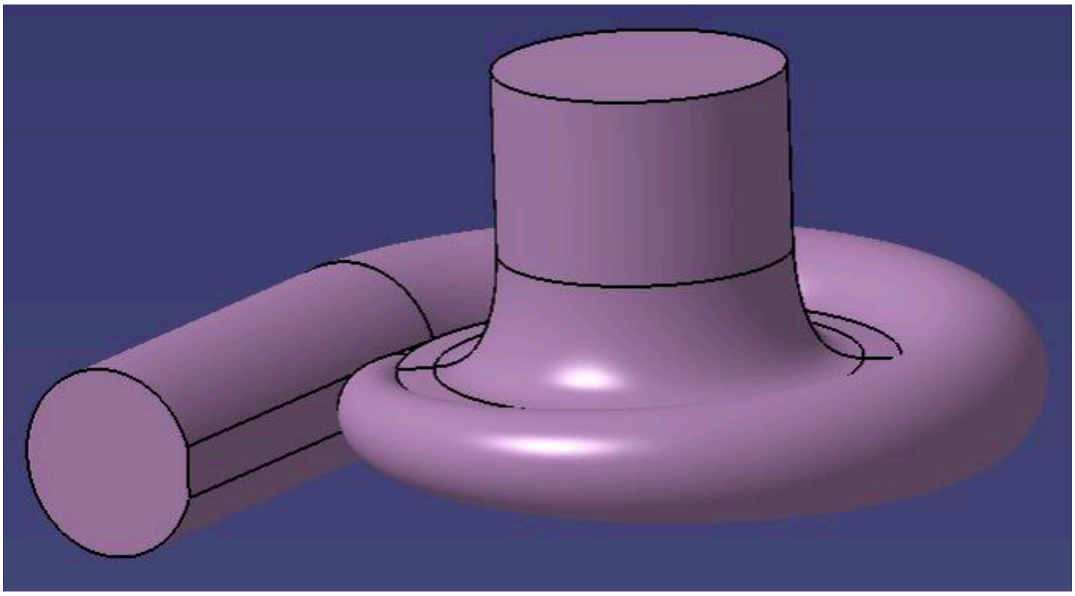
The centerpiece of a centrifugal compressor is its rotor, as it allows the fluid to flow swiftly through the compressor. The quality of the rotor's design will significantly impact the final efficiency of our compressor, which is why we spent so much time determining all its parameters in hope of having the best rotor shape possible. Thanks to the Python program we used, we generated a .CSV file containing coordinates for many points to draw the rotor's blade, which we imported inside CATIA V5. After shaping the whole rotor using the Generative Shape Design tool, we obtained a volume object using the Part Design tool, which can be used for 3D-printing or for CFD simulations.



2. Housing

a. General shape

The shroud part of the rotor's housing was done by projecting the top curve of a blade onto a vertical plane to shape the shroud. That line was also translated to prevent friction between the rotor and the shroud. Then, we used a script to plot progressively bigger circles using 2 static points – top and bottom of the passageway for the fluid – and a moving third point – center of the circle. We obtained the following volute:

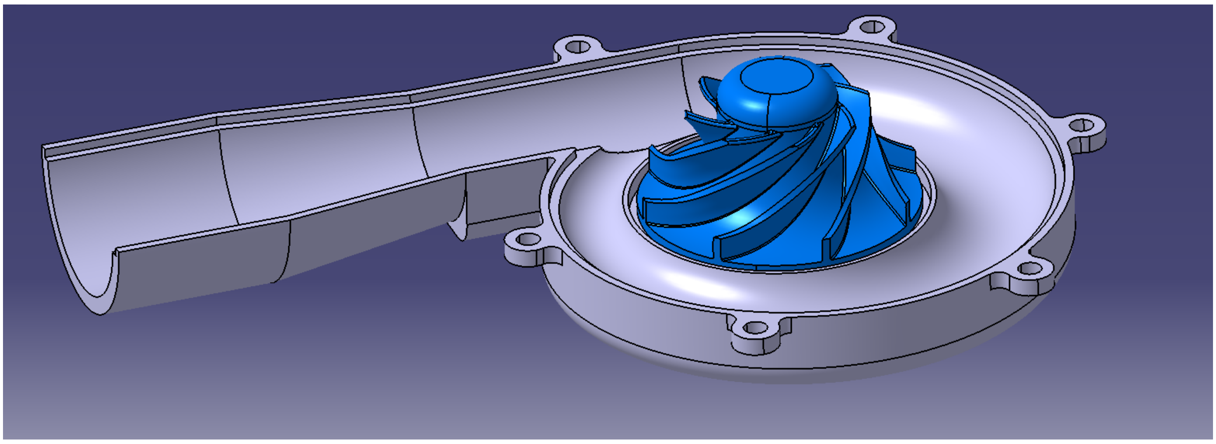


We subtracted this volute to a block we created in Part Design. After separating the new block in half, we obtained the top and bottom plates of the compressor.

b. Finishing touch

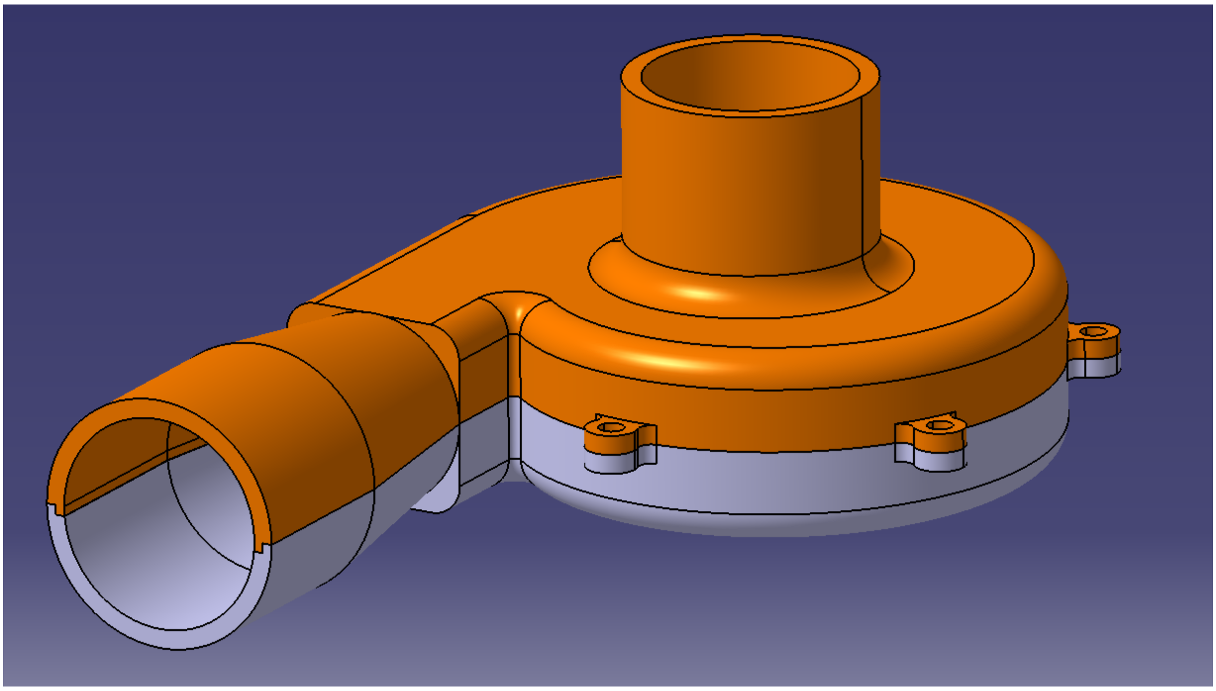
We removed a lot of the material from the top and bottom plates to lose mass. We added a groove that circles around the bottom plate, and added a ledge on the top plate so both parts can fit nicely together to help them set in place. Afterwards, we added holes on the top of the bottom plate, and on the bottom of the top plate, all of them aligned in pairs. Thus, we can put screws to maintain both parts in place.

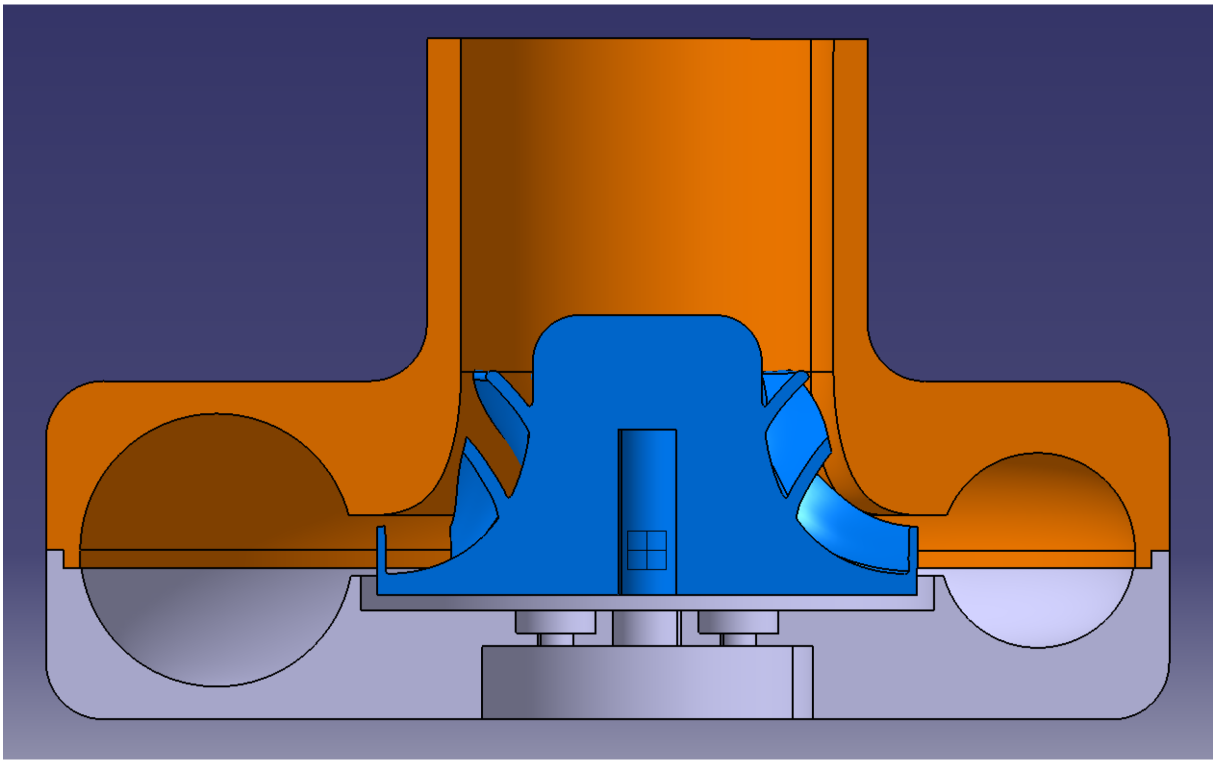
A circular pocket was also made on the bottom part to accommodate the rotor, as well as a centered hole to pass the motor's shaft through, and 4 holes in a cross to screw the motor to the bottom plate. We also added a long extension to fit the wind tunnel's diameter.



3. Assembly

With the rotor, the bottom plate, and the top plate all ready, we had to assemble them together. Using the Assembly Design tool, we were able to align the rotor axially to the bottom and top plates. We positioned the bottom and top plates using contact constraints. Here is the final assembly result, which we can import in CFD and 3D-printing software:



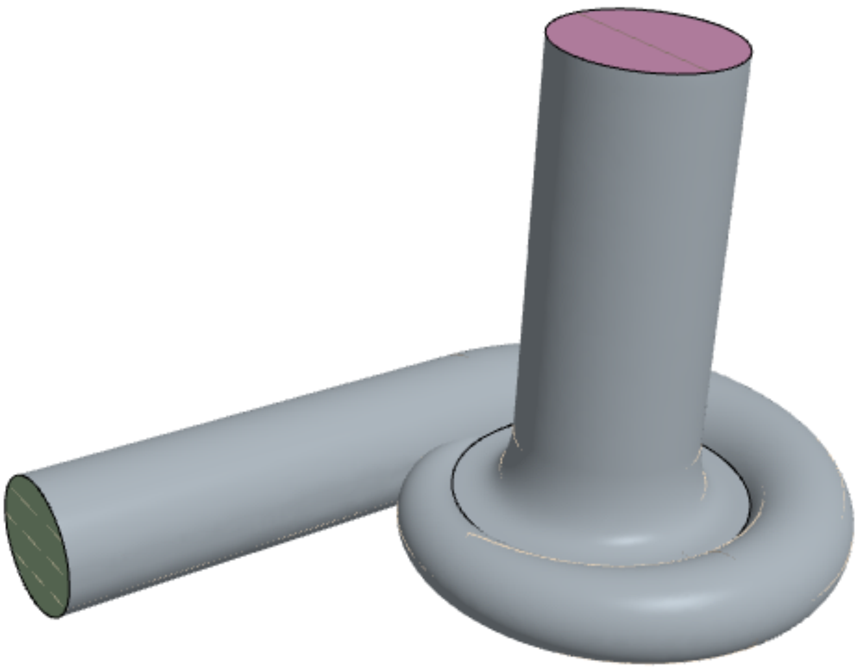


V. CFD Simulations

Now in possession of a CAD file for our compressor, we can use a CFD software to simulate the compressor to have an idea of what to expect from our design in real conditions. We used the STAR-CCM+ Software to make the CFD simulations from our CAD file.

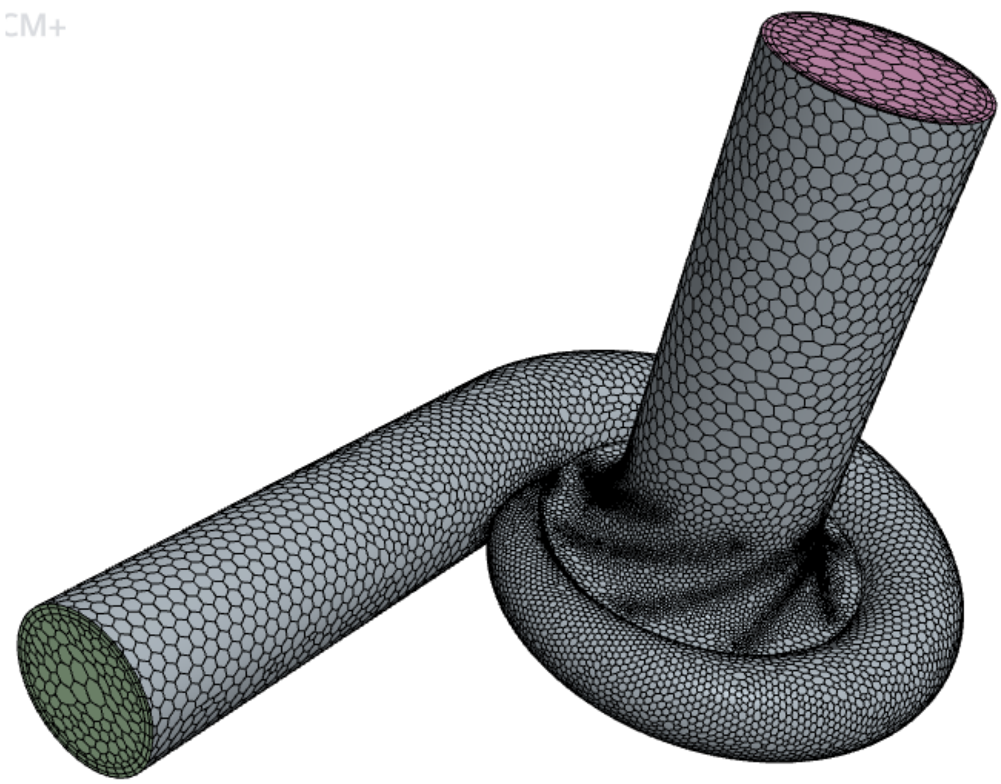
1. Volute

We imported the CAD file of our compressor into STAR-CCM+, where we used the fill-in feature to extract the volute of our compressor, which corresponds to the part where the fluid will flow. However, we need to differentiate two parts: a rotating one (the vertical cylinder in the image below), and a static one (the rest). After separating both parts by using basic shapes and Boolean operations, we obtained the following result:



2. Meshing and configuring

The second part of the process to run CFD simulations is to establish a mesh of the volute. To do this, we used Automated Mesh Operation tool (an FEM tool) to remesh the surface, as well as mesh the volume using a Polyhedral mesher with prism layers. We also set the base size to $1\ mm$, which will be the value used for all relative size controls for the meshing process. After almost 6 minutes of processing, the output mesh contained 536.366 cells, $2.740.302$ faces and $2.072.317$ vertices, as shown below. The smaller and the greater the number of elements there are, the more accurate the results of our simulations will be, at the cost of a longer computing time and possibly and longer converging time. We can notice how in more complex parts of the volute's geometry, there are way more elements than in the parts where there is simpler geometry.



```
Mesh Operation Automated Mesh complete. CPU Time: 344.99, Wall Time: 344.99, Memory: 699.68 MB
----------------------------------------
Volume Meshing Pipeline Completed: CPU Time: 345.08, Wall Time: 345.08, Memory: 699.68 MB
        Cells: 536366   Faces: 2740302  Vertices: 2072317
```

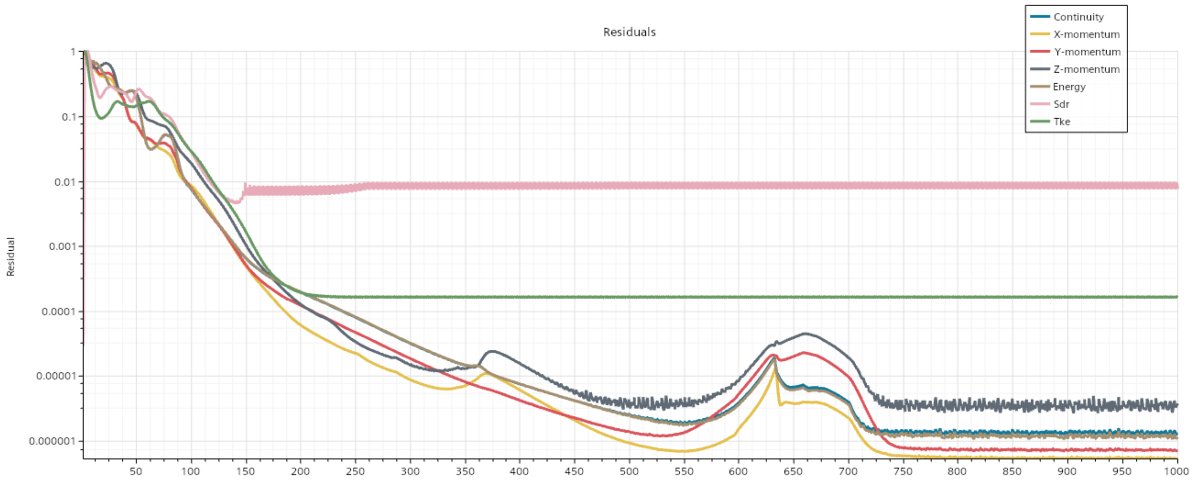
3. Simulations

a. Configuring: first case

Before running any simulations, we need to configure the initial conditions of our system. We define two surfaces: the inlet and the outlet. We also define the interface between the rotating and static parts of the volute. While we imposed Total Pressure and Total Temperature at the inlet, we imposed the mass flow rate of the outlet. Indeed, a mass flow rate condition usually yields better results and improves the stability of the simulation. For this first configuration, we used the mass flow rate computed earlier for the operating point of the centrifugal compressor: $\dot{m} = 0.0299\ m/s$. We also used the rotational speed of the operating point of the compressor $N = 62961\ rpm$

b. Converging

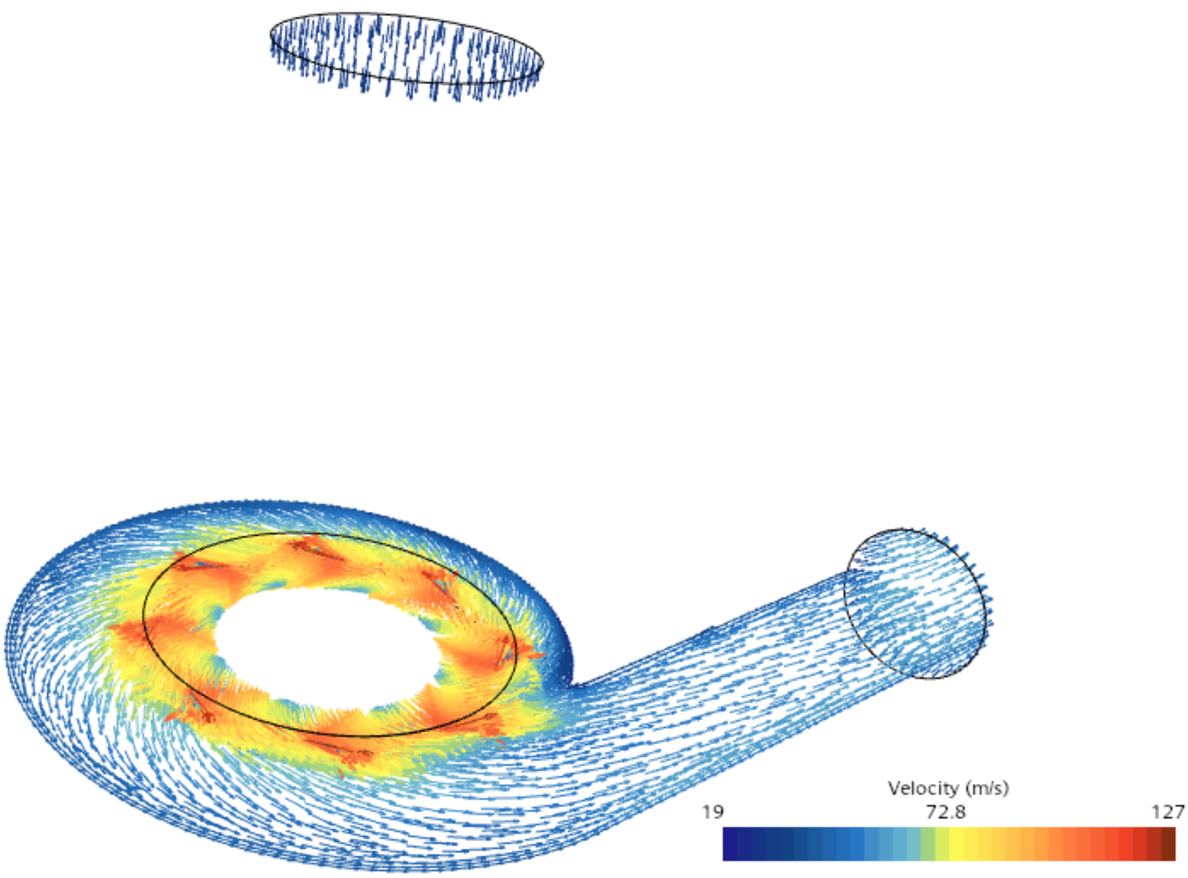
While our mesh may be refined, it is important to keep in mind that the algorithms used are iterative, thus the algorithm has a chance of diverging, or at least not converging enough. We evaluate the converging of an algorithm by plotting the residuals of many values. Residuals should be as small as possible and stagnate after several iterations. Here is the plot of these residuals after 1000 iterations:

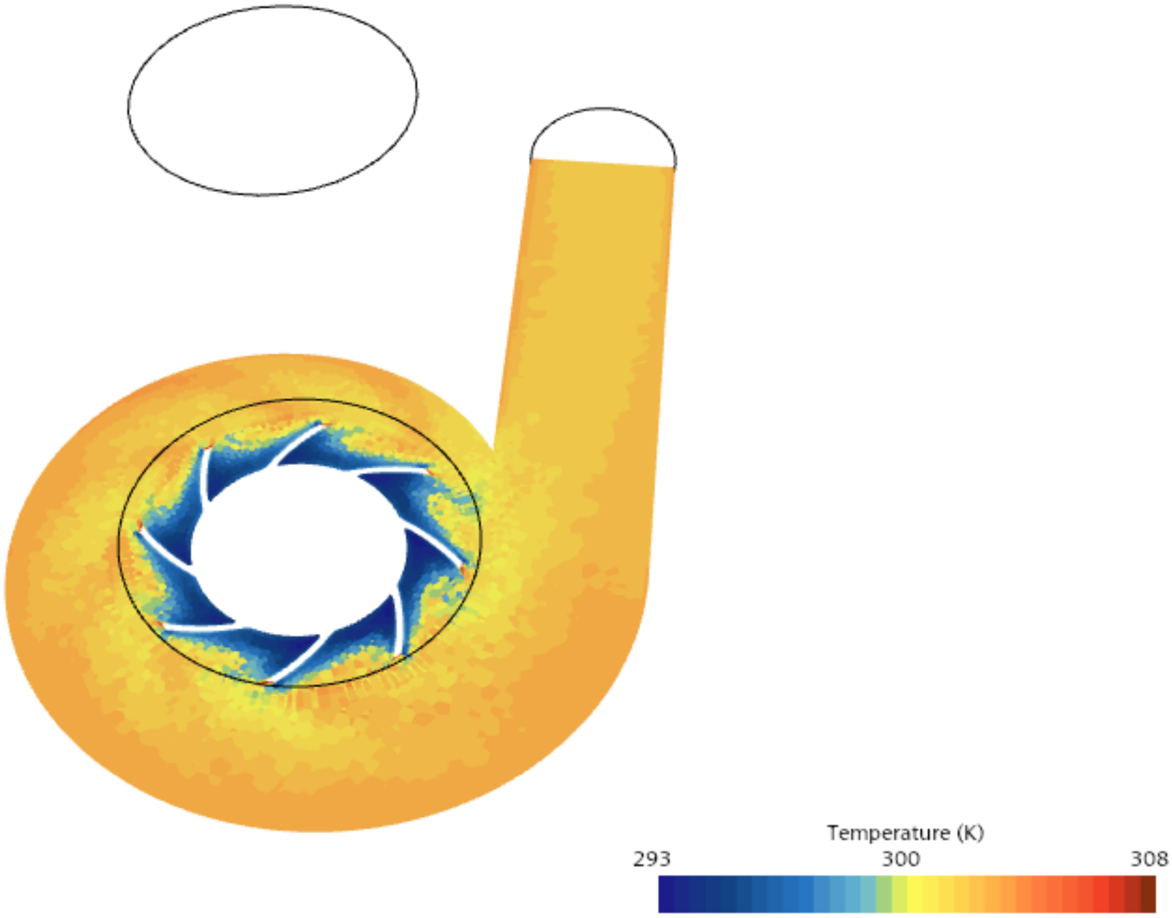


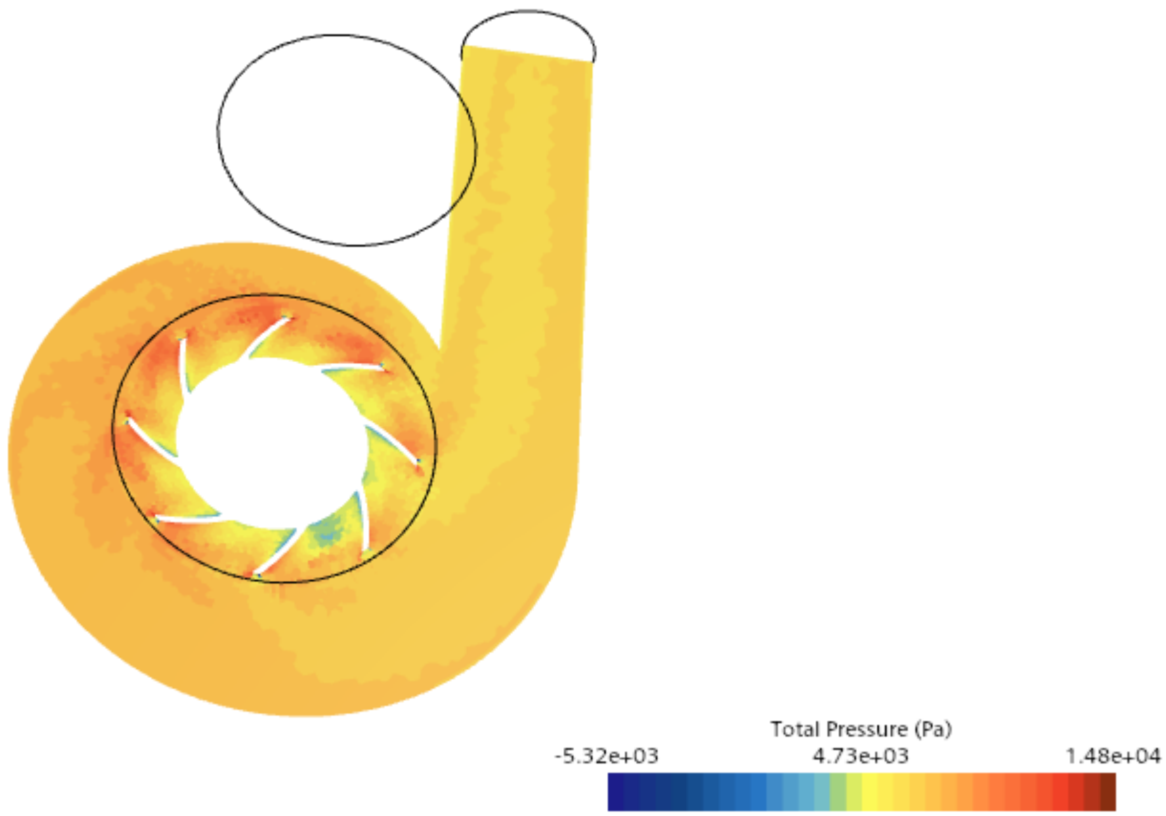
As we can see, the various residuals all converge after 750 iterations, with values ranging from 0.01 to 0.000001. This means that our simulation was successful, and as such the values we obtained from the simulation can be exploited.

c. Views

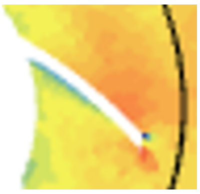
While plotting many thermodynamic variables, the simulation also yields view that allows us to see these variables' behaviors inside the compressor. Here are the velocity, temperature, and pressure views we plotted:

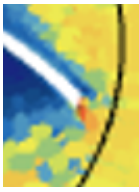



Temperature (K)
293
300
308


Total Pressure (Pa)
-5.32e+03
4.73e+03
1.48e+04

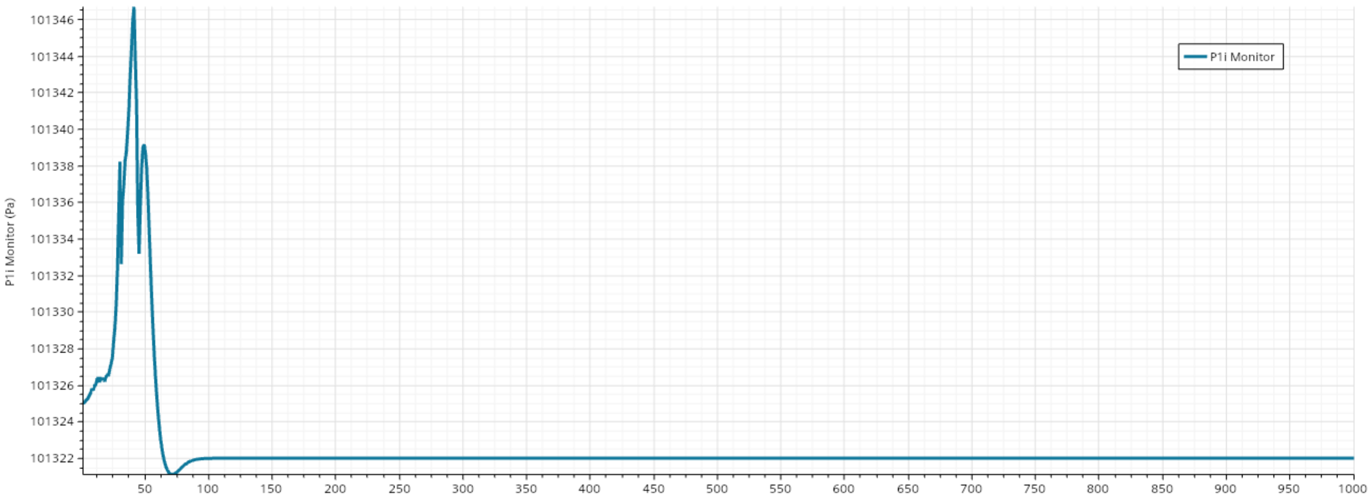
We can note that the pressure is extremely low (left), and the temperature is abnormally high (right) on all elements at the tip of the compressor's blades. Being the most complex places of the simulation, we could increase the number of elements locally to yield more accurate results in these places.

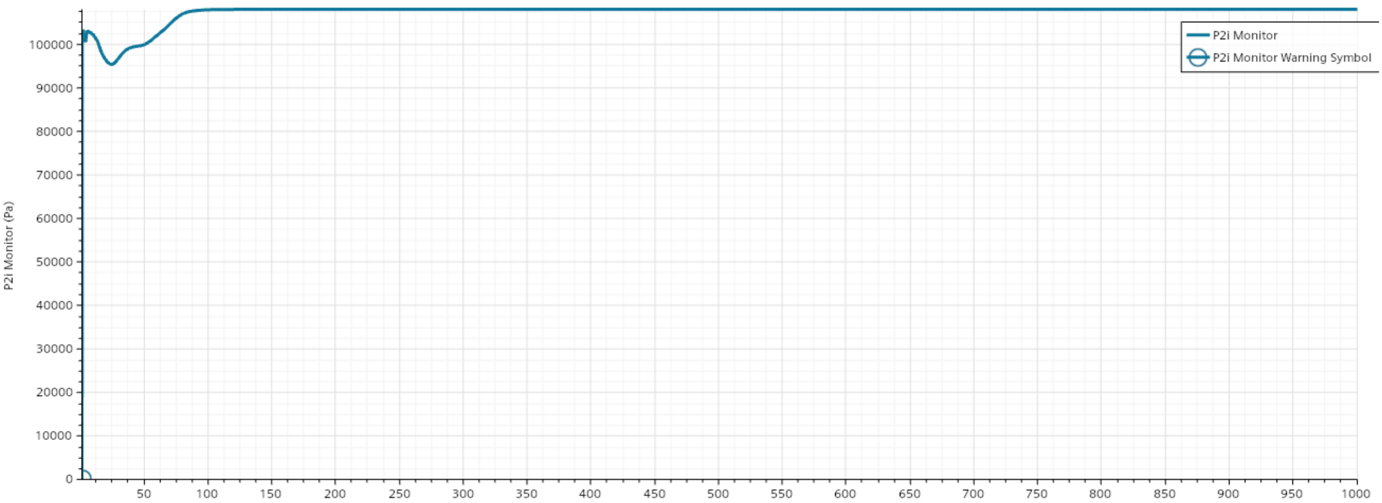


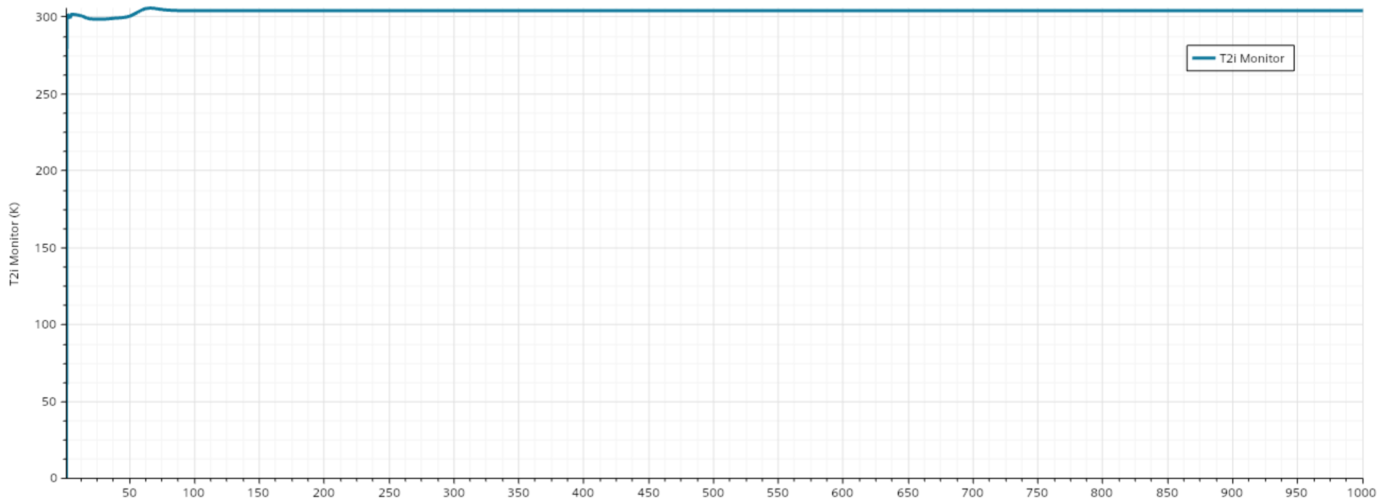


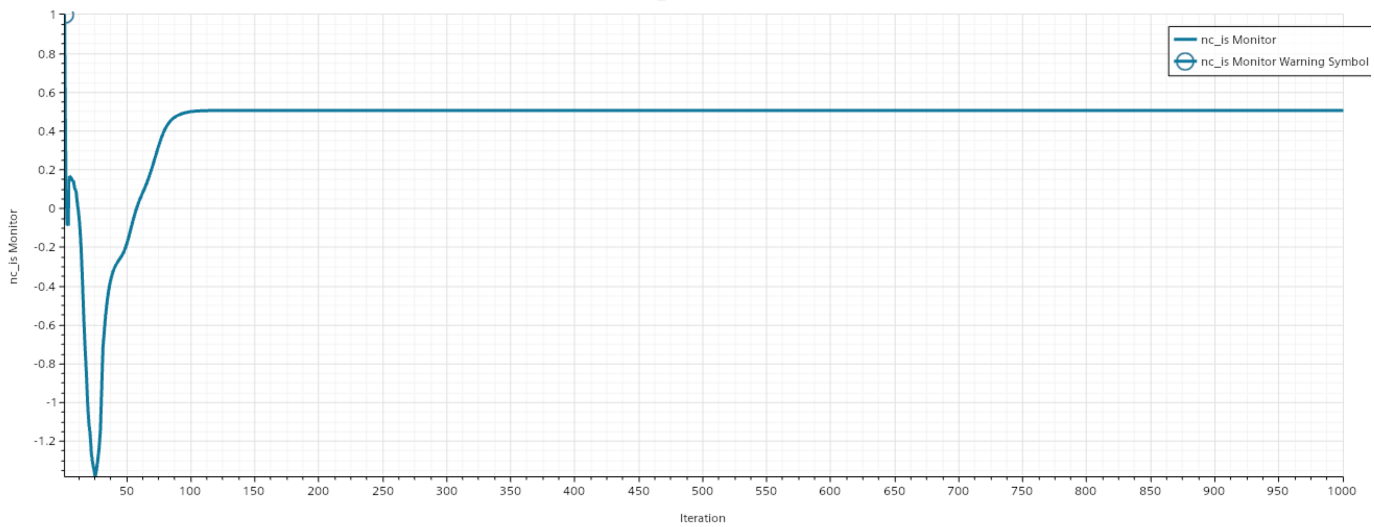
d. Plots

The thermodynamic variables that are relevant to our problem are the isentropic pressure and temperature, as well as the isentropic efficiency of the compressor. As we want to study these variables on the global scale of the compressor, we plotted the inlet isentropic pressure, the outlet isentropic pressure and temperature, and finally the isentropic efficiency. Having access to both these pressures also gives us access to the compression ratio of our centrifugal compressor, which is the tangible result of its operation. We obtain the following plots:









We can notice that the plots all stagnate at around 100 iterations, while the residuals took much longer to truly stabilize. This can be explained by the fact that we only look at the inlet and outlet, or global values of the compressor, which have much more simple geometry than inside the compressor, which residuals also consider, and thus would need more time to converge than the inlet and outlet variables.

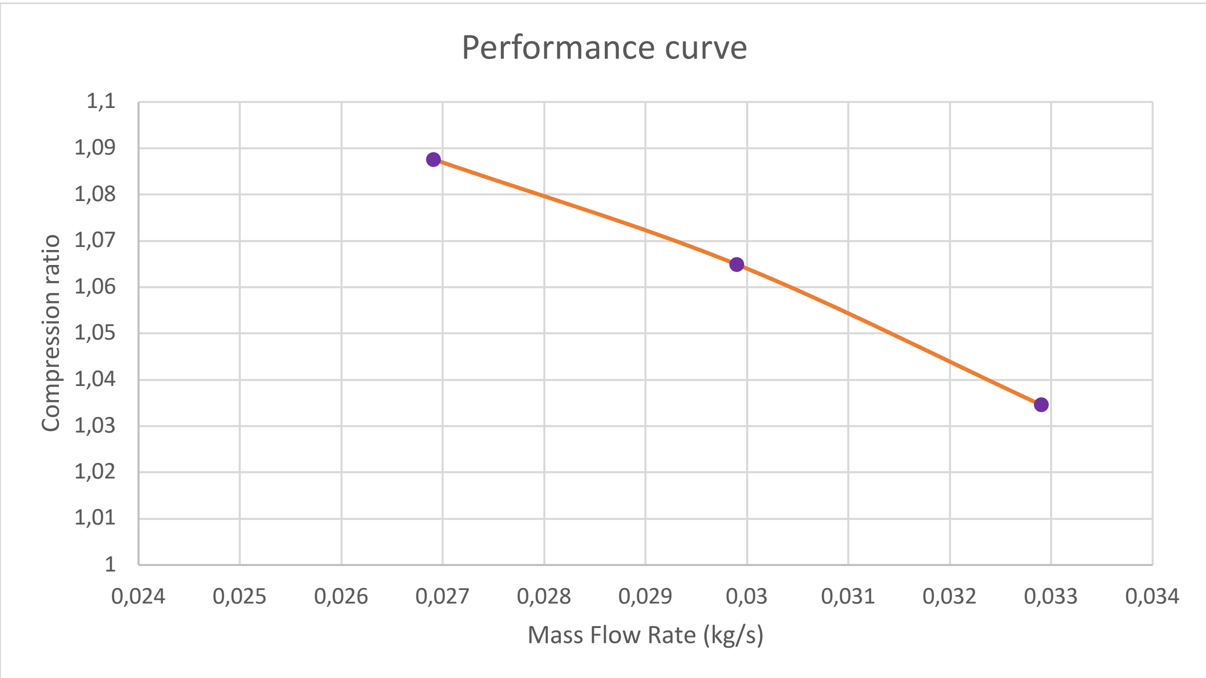
e. Configuration: second case and third case

For these last two cases, we ran two other simulations, this time with the mass flow rate and rotational speed at 10% more or 10% less than their operating point values. Thus, one simulation was run with $\dot{m} = 0.02691\ m/s$ and $N = 65381\ rpm$, and another was run with $\dot{m} = 0.0329\ m/s$ and $N = 59140\ rpm$.

4. Results

With these 3 simulations, we were able to compute the inlet isentropic pressure, outlet isentropic pressure and temperature, compression ratio, and isentropic efficiency of our centrifugal compressor at its operating point, 10% above, and 10% below. While giving us an estimation of how these thermodynamic variables evolve in various operating conditions, the key information we obtained from these simulations is the performance curve of our centrifugal compressor. The final step will be to test the performance of our 3D-printed compressor in a wind tunnel at the same operating conditions than the simulations and compare both results.

$\dot{m}\ (kg/s)$	$P_{2i}\ (Pa)$	$P_{1i}\ (Pa)$	$T_{2i}\ (K)$	$\eta_{c,is}$	Π	$N\ (rpm)$
0.02691	110199	101322	305.5	0.57578	1.087612	65381
0.0299	107898	101322	303.7	0.5032256	1.064905	62961
0.0329	104825	101328	301.7	0.3354212	1.034582	59140



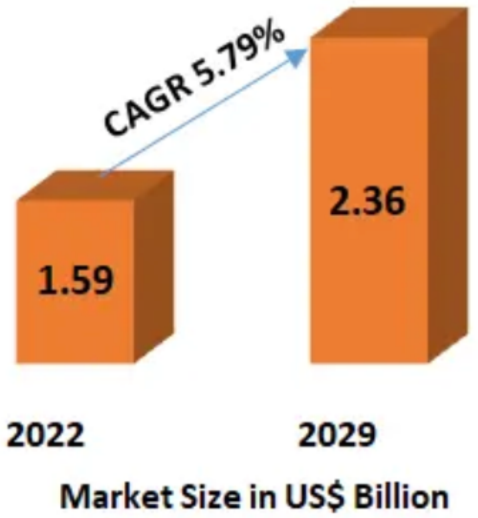
VI. Industrialization file

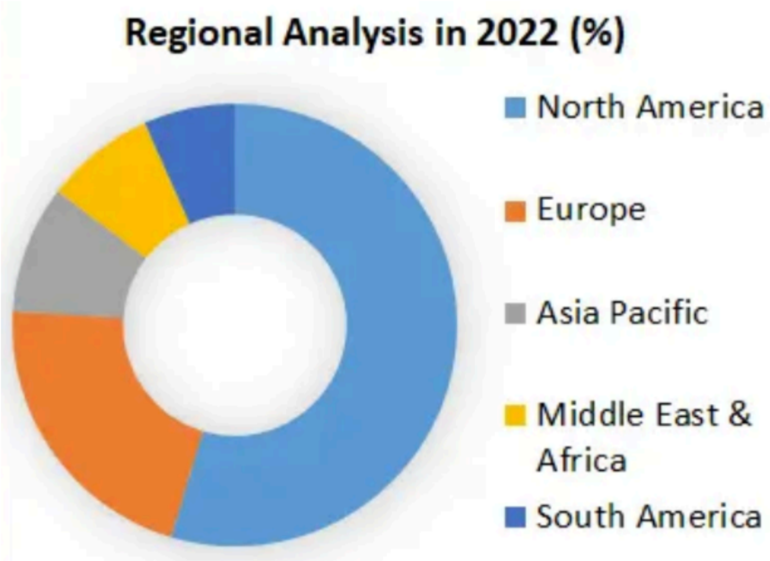
This section is dedicated to the study of the hydrogen compressor market. This information is key as it helps us understand the current and projected state of demand for hydrogen compressors. With this knowledge, we will be able to establish a rough estimate of what stocks we need, as well as the costs of making our compressors.

1. Market size and regional distribution

To determine our needed stock and the quantities we need to produce, we first need to evaluate the market size and the regional distribution of hydrogen compressors. As of 2022, the global market size of hydrogen compressors is $\$1.59B$, and a Compound Annual Growth Rate of 5.79% is projected until 2029, where the global market size should theoretically be near $\$2.36B$.

Furthermore, North America is by far the biggest user of hydrogen compressors, followed by Europe and Asia Pacific. As hydrogen compressors found their usage in various fields such as Energy Production, the Chemical Industry, Oil and Gas extraction and refinement, Transportation, and others, it is not surprising to see the leaders in these fields representing more than 80% of the market.





Source: https://www.maximizemarketresearch.com/market-report/hydrogen-compressor-market/168514/

A non-exhaustive list of the key players for hydrogen compressors is showed in the next page. From these 16 companies, excluding the global-scaled ones, 7 of them are based in Europe, 5 in the US, and only 2 in Asia. While hydrogen compressors are a new and thriving technology, the competition is fierce, especially in Europe. However, very few companies are based in France, they are mostly based in Italy, Switzerland, and Germany. Thus, focusing our targeted clients on French companies and maybe French individuals seems like the best plan of action right now. However, we do not have access on the precise market size for hydrogen compressors in France, so we cannot make a good estimate on how much quantities we should produce as well as evaluating stocks.

- Adicomp
- Ariel Corporation
- Atlas Copco Group
- Baker Hughes
- Burckhardt Compression AG
- Coltri
- Corken Inc.
- Fornovo Gas Compressors
- Haug Kompressoren AG
- Howden Group Ltd
- Hydro-Pac Inc.
- Indian Compressors Ltd
- Ingersoll Rand Inc
- Sauer Compressors
- Sundyne Corp.
- Toplong Compressors

2. Material choice analysis

Choosing the right material is crucial when making an industrialization file, as it will be a key factor in evaluating the total costs of production. In our case, we can choose up to two different materials: one for the rotor, which is in motion, and one for the housing which is static.

Several materials are used for making a hydrogen compressor, the most common ones being:

- Composite materials: lightweight, highly resistant / complex manufacturing, costly
- Stainless steel: highly resistant, easy manufacturing / heavy
- Aluminum alloys: lightweight / not very resistant
- Titanium alloys: highly resistant / costly, complex manufacturing
- Nickel-based alloys: perfectly suited for hydrogen / very costly

Since the housing does not need to be highly resistant, as it is not in motion and do not receive too much stress from the air flow, we could choose an aluminum alloy to reduce weight.

However, regarding the rotor, it needs to be resistant as it is in motion and receives high stress from the air flowing through it at high velocity. Stainless steel could be a great choice in that regards. Even though it is heavy, other materials are more costly and harder to manufacture, and the rotor has a very complex shape. This means that having an easily manufacturing material would considerably reduce the production costs.

VII. Summary and Conclusion

In this report, we looked at all the aspects that went into making our centrifugal compressor for FCEVs. We went into detail on the Sizing of the Centrifugal Compressor, the Filling of the Tank, the 3D CAD of the Centrifugal Compressor, the CFD Simulations we made, and finally the industrialization file we wrote to better understand the market for our compressors.

For sizing the centrifugal compressor, we made a functional analysis of the system, using turbomachinery theory for compressible fluids. From the established equations, we were able to compute design parameters we previously chosen using Python scripts. After criteria verification, we were able to fully size the rotor of the compressor.

We then looked into simulating the filling of the hydrogen tank, using again a Python script we wrote. After studying two approaches: one with the ideal gas assumptions, and the other with no assumptions, we ran many simulations under different conditions to evaluate the behavior of thermodynamic variables such as pressure and temperature inside the tank over time.

Afterwards, we 3D-designed the centrifugal compressor using the CAD software CATIA V5. By following the previously found design parameters, we carefully designed the compressor, as this step was crucial for 3D-printing the compressor as well as simulating it. Thanks to CATIA V5's many tools, we were able to make a final assembly composed of 3 parts: a rotor, and two plates representing the housing.

Importing the assembly CAD file we made in the STAR-CCM+ software, we were able to extract the volute of the fluid flowing through the compressor. After meshing and configuring this volute, we made various CFD simulations under different operating conditions in order to establish the performance curve of the compressor.

Finally, after analyzing the market size and the materials we could use for the compressor, we established that for now, the best course of action for us would be to focus our market on French companies and individuals, by producing our compressors using an aluminum alloy for the housing, and stainless steel for the rotor.

In conclusion, we were able to fully develop a centrifugal compressor, from its preliminary phase to its potential industrialization by sizing, 3D-designing and simulating it.