Development and Test Firing of a One-Piece Additively Manufactured Liquid Rocket Engine

Authors: Killian Melle*, Samuel Rappillard†
Ecole Supérieure des Techniques Aéronautiques et de la Construction Automobile (ESTACA), 12 avenue Paul Delouvrier, Montigny-le-Bretonneux 78180, France

Co-author: Vincent Demaison‡
Ecole Supérieure des Techniques Aéronautiques et de la Construction Automobile (ESTACA), 12 avenue Paul Delouvrier, Montigny-le-Bretonneux 78180, France

The goal of this study is to develop, print and evaluate a bi-liquid rocket engine, with a thrust arbitrarily defined at 100kg. This engine is designed to work with two liquid propellants, kerosene and liquid oxygen. This one-piece product is one of the first of its kind, in the state-of-the-art, using 3D metal printing. The kerosene is used to cool down the engine by running through a regenerative cycle cooling system. The liquid oxygen, a cryogenic propellant, is directly dispensed through triplet injectors into the combustion chamber. The design was validated by numeric simulations. A static fire will end the study at the end of April 2019, using a fire testing stand developed by the team.

Nomenclature

A_c	=	nozzle throat cross-sectional area
A_e	=	nozzle exit area
ALM	=	Additive Layer Manufacturing
C_d	=	characteristic velocity
C_f	=	thrust coefficient
CFD	=	Computational Fluid Dynamics
D_e	=	nozzle exit area diameter
D_t	=	nozzle throat area diameter
F	=	thrust
$g = 9.81\text{m/s}^2$	=	earth's gravitational constant
M_e	=	ratio of gas velocity to the local speed of sound
P_c	=	gas pressure in the combustion chamber
P_t	=	gas pressure at the nozzle throat
$R = 287.058\text{J/(kg K)}$	=	gas constant
S_m	=	material's allowed working stress
$T_c = 3300\text{K}$ *	=	combustion chamber flame temperature
T_e	=	temperature of the gases at the nozzle exit
T_t	=	temperature of the gases at the nozzle throat
V_c	=	chamber volume
w_t	=	mass flow
$\gamma = 1.2445$	=	ratio of gas specific heats

*Master's Degree in Aerospace Engineering, killian.melle@estaca.eu
†Master's Degree in Aerospace Engineering, samuel.rappillard@estaca.eu
‡Master's Degree in Aerospace Engineering, vincent.demaison@estaca.eu
*Table 1 in [1] and *Propellant combustion charts* in [2]

I. Introduction

ROBERT H. Goddard achieved the first successful flight with a liquid-propellant rocket on March 16, 1926. The engine used liquid oxygen and gasoline. Nowadays, 3D printing could revolutionise rocketry. ArianeGroup printed in 3D the gas generator of Vulcain 2.1, which will be integrated on Ariane 6, the European launcher. Meanwhile, the European Space Agency (ESA) developed Prometheus, a reusable and low-cost engine, almost entirely 3D-printed[†]. Production cost has been said to be ten times cheaper than a Vulcain 2. In a perspective of developing a propulsive system for a rocket probe, this final year project has been started. Its main goal is to produce and fire during a static test, a small rocket engine with a thrust of 100kg. In order to easily manufacture it, 3D printing has been chosen. ArianeGroup and the ESA proved that this technology could be used to minimise the number of pieces which must be assembly. However, let's go a bit further and imagine a rocket engine printed in one piece. This design would avoid any impermeability issues, as well as limit structural pieces such as screws and bolts. Nevertheless, this engine would encounter some other questions like its re usability or the validation of its injection face. This study is focused on those questions and how to design an additively manufactured engine. First, the document will develop the analytic sizing and architecture of the engine, followed by their validation using numeric simulations. Then, this study will talk about pros and cons of additive manufacturing and how this technology impact the engine's design. Finally, a brief presentation of the bench and the static fire test will conclude this paper. The main objective of this entire study is, at the end, to analyse its performances (thrust, pressure and temperatures) during this fire test.

II. Design of the engine

A. Bi-propellant combination

During preliminary study phase, various propellants were taken on. At first, a design review was established. Those propellants had to respect this safety specifications to limit damages and protect people and equipment in worst scenarios. Besides, they had to be compatible with the cooling system chosen. Since liquid oxygen was quickly chosen as the oxidiser due to its non-explosive nature and its efficient use, four types of fuels were considered. The latter must not be a dangerous propellant but still be performing.

In the first instance, methane and liquid hydrogen have been seen as the two best options. Indeed, they are the two most performing and used fuel in aerospace industry. But their price and complicated affordability coupled with the constraint of their cryogenic nature removed them from potential choices. The limited budget and the project's short timeline did not allow to develop the complex systems needed to use them.

Then, two other propellants have been studied: ethanol and kerosene. They have both the assets to be affordable, but they are very different on many main points.

On the one hand, ethanol has a lower burning temperature. So, it would generate less heat constraints in the combustion chamber. Ethanol combustion also allows to see perfectly Mach disks without any instruments. Cooking (II.C) which is a phenomenon induced by carbon-based fuel is also less important with ethanol. Indeed, it has a shorter carbon chain than kerosene. However, ethanol has a couple of drawbacks. During combustion, it could polymerise and so, if there are some impurities in the propellant, all the static fire test stand could suddenly blow up.

On the other hand, kerosene was considered. It has a much better heat content: 47.3 MJ/kg against 29.7 MJ/kg for the ethanol. It also has a better mixture ratio: 2.29 against 1.19. That means the engine would use less kerosene than ethanol to produce a 100kg thrust. Finally, kerosene is a better regenerative cooling propellant. There are a lot of different types of kerosene like RP-1 or Jet A1. Each has its pros and cons.

After weighing up all of the pros and cons, the kerosene has been chosen as the fuel. Because of low affordability of the RP-1, the Jet A1 kerosene will be used. Indeed, safety drawbacks of ethanol were too important to be ignored and kerosene appeared to be a much better fuel for the engine.

B. Sizing and geometry

The methodology used to calculate each geometry is inspired by[1]. The requirement specifications were written at the beginning, aiming, arbitrarily, 100kg of thrust and a specific impulse in the void of 198s. This impulse is closed to other engines with this architecture and should allow a viable geometry with some losses of performance. Pressure inside the combustion chamber is set at 20bar. The design equations will then start with the calculus of the nozzle and

[†] http://www.primante3d.com/moteur-prometheus-30062017/

end with each entrance of liquid oxygen and kerosene. Those equations are setting up the measurements of the throat area, combustion chamber, injection, and cooling system around the engine (Fig. 1).

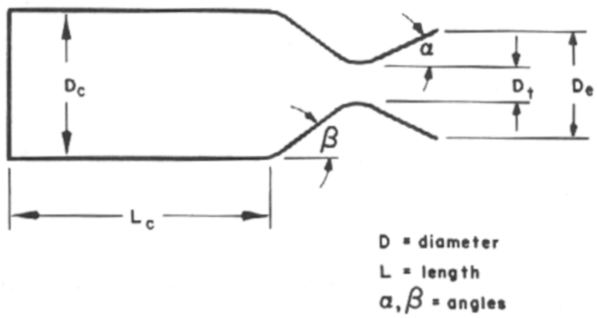

Fig. 1 Motor design configuration (Ref. [1] (Fig 4-9))

1. Design of the nozzle

Firstly, pressure of the gases at the throat is deducted with the combustion chamber pressure using the following formula:

$$P_t = P_c \left(1 + \frac{\gamma - 1}{2}\right)^{-\frac{\gamma}{\gamma-1}} \tag{1}$$

Likewise, temperature of the gases at the throat is deducted from the temperature of the gases in the combustion chamber using this formula:

$$T_t = T_c * \left(\frac{1}{1 + \frac{\gamma-1}{2}}\right) \tag{2}$$

Finally, mass flow can be determined with the specific impulse and the thrust targeted:

$$w_t = \frac{F}{ISP * g} \tag{3}$$

With those parameters, nozzle throat area can be deducted, and so, its diameter:

$$A_t = \frac{w_t}{P_t}\sqrt{\frac{R * T_t}{\gamma * g}} \tag{4}$$

Next step is to define local gas velocity using the pressure in chamber and the pressure of the atmosphere:

$$Me = \sqrt{\frac{2}{\gamma - 1}\left(\left(\frac{P_c}{P_{atm}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right)} \tag{5}$$

Nozzle exit area can then be calculated with both throat area and gas velocity:

$$A_e = \frac{A_t}{M_e}\left(\frac{1 + \frac{\gamma-1}{2}M_e^2}{\frac{\gamma+1}{2}}\right)^{\frac{\gamma+}{2(\gamma-1)}} \tag{6}$$

Results can be found in table 1[‡]. In order to size length between the throat and the exit, [1] advise using a half-angle α (Figure 1) of 15°.

[‡]Results have been calculated by taking into a corrective coefficient of 0.975 for C_d and 0.98 for C_f.

2. Design of the combustion chamber

A combustion chamber volume is determined with its characteristic chamber length, $L^\star$. Regarding the given performances, [2] advise using a value between 102cm and 127cm for a mixture LOX/RP-1. 120cm has been chosen.

Knowing throat area, chamber volume is determined as follows:

$$V_c = \frac{L^\star}{A_t} \tag{7}$$

This chamber volume includes convergent volume. [2] in figure 1.7 gives this formula to measure chamber length:

$$L_c = \exp^{0.029+\ln(D_t)^2+0.47\ln(D_t)+1.94} \tag{8}$$

In order to size diameter with the length and the volume, [1] advise using a half-angle β (Figure 1) of 35°.

3. Chamber wall thickness

The chamber combustion minimal thickness is determined by the admissible stress of the material at combustion gases' temperature. In this case, the engine is additively manufactured with Inconel 718 which has an allowable working stress S_m of 172MPa at 900°C[§]. [1] uses internal pressure in the chamber and its diameter to calculate its thickness.

$$t_c = \frac{P_c * D_c}{2 * S_m} \tag{9}$$

This thickness is able to resist to the internal pressure and hot combustion gases and should be enough to attach the cooling system on it.

Table 1 Sizing of the engine

Parameters	Variable	Value	Unit
Throat diameter	D_t	0.0004238	m^3
Exit diameter	D_e	0.073	cm
Length between throat and exit	L_e	0.034	m
Chamber volume	t_c	0.00129	cm
Chamber diameter	t_c	0.00129	cm
Chamber minimal thickness	t_c	0.00129	cm

C. Regenerative cycle cooling system

A liquid engine produces a huge amount of energy (more than 30 000 MW/m^3 for an Ariane 5). Most of it is expressed thermally through an important rise of heat. Therefore, a rocket engine must be cooled down efficiently to limit any material degradation. Many cooling techniques are frequently used in the state-of-the-art such as in [3], four were considered:

- Film cooling: a small amount of combustive or fuel runs through the walls of the engine to absorb its heat. This liquid (or gas) is not used in the propulsion process, since it is not injected into the combustion chamber. This cooling system, despite its low efficiency, can be easily combined with another technique.
- Heat sink and radioactive cooling: heat sinks are often used on small engines or satellites since its non-consumption of electrical energy. High emissivity materials are used such as molybdenum or tungsten alloys, on which is added an aluminate coating and a chromium inlay.
- Ablative cooling: an ablative material is added on sensitive areas and its abrasion during the launch will protect the engine. It is highly reliable, easy to produce and avoid any additional hydraulic circuits, pumps or tanks.
- Regenerative cooling: which has been chosen.

[§] Value given by AddUp ©

In the last cooling system, the regenerative cycle, the propellant enters at the end of the nozzle and runs into cooling canals spread inside the walls of the engine. The fuel is then injected into the combustion chamber and used for propulsion. This simulate an adiabatic environment. This last cooling system is the most effective one but is usually really complex to produce due to its numerous pieces. With an additive manufacturing, this issue does not exist. However, the regenerative cooling system has one major drawback: it cannot be used on a reusable engine due partially to cooking. Cooking is a phenomenon induced by carbon-based fuel when some cooling liquid is trapped in the canals and may plug them with its evaporation (which can not be avoided since the liquid will be static and against the hot chamber wall when the engine stops).

1. The impacts on the nozzle's throat

The nozzle throat is the most sensitive area of the engine since it has the most important heat transfer coefficient. The cooling liquid could boil, even evaporate at this temperature. The liquid can boil at three different states with those conditions:

- Normal convection;
- Nucleated ebullition (transfer of hot gas bubbles upward);
- Partial film boiling followed by film boiling (thin film of hot gas along the hot area).

Those states are inherently linked with temperature as shown in the figure 2[¶]. Film ebullition must be avoided. Indeed,

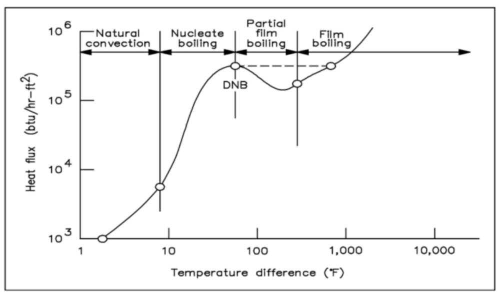

Fig. 2 States of a liquid's ebullition

the steam layer will reduce the heat transfer, and so, the cooling system's efficiency.

Besides, the rise of temperature causes a rise of gas's viscosity. Thus, thermal conductivity is impacted and is also going to rise. As shown on the Clapeyron's diagram in figure 3, the liquid can hit a critical point of pressure where liquid and gas phases are no longer separated. In the case of this study where there is a high pressure of 22bar at the

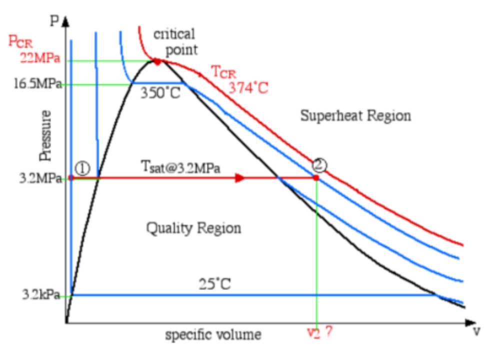

Fig. 3 Critical point of a pure liquid

throat, and high temperatures, this critical point will surely be reached. Those phenomena should then be taken into account while sizing the cooling system.

Two fuels are easily available for students for such a project: ethanol and kerosene. The first one has a boiling point at 78.2°C, the second one is at 216.3°C. To limit negative effects of previously described phenomena, kerosene has been

[¶]http://nuclearpowertraining.tpub.com/h1012v2/, Figure 13 chosen to cool down efficiently the engine.

2. Calculus of heat transfer

Conductive heat transfer For liquid propulsion engine with a small thrust, gas flow can be laminar in some areas but most of it will be turbulent in the nozzle. Using Nusselt's number Nu, the thermal transfer coefficient h can be determined with Reynold's number Re, Prandtl's number Pr, the diameter D and the liquid thermal conductivity k:

$$Nu = 0.023 * Re^{0.8} * Pr^{0.33} = \frac{h * D}{k} \tag{10}$$

With small liquid engines, Prandtl's number lies between 0.73 and 1 which can be approximated to 1 at power 0.33.

Reynold's number is related with fluid's properties (U is its velocity, ρ its density, μ its viscosity and $\dot{m}$ its mass flow) using the following formula:

$$Re = \frac{\rho * U * D}{\mu} = \frac{4 * \dot{m}}{\pi * D * U} \tag{11}$$

Finally, mass flow can be determined with this relation:

$$\dot{m} = \frac{P_c * A_t}{C^\star} \tag{12}$$

P_c is the chamber pressure, A_t the throat area and $C^\star$ the characteristic velocity.

Knowing those formulas allow to determine the thermal transfer coefficient h:

$$h = 0.023 * k \left(\frac{\pi}{4}\right)^{0.1} \dot{m}^{-0.1} \mu^{0.8} P_c^{0.9} C^{\star 0.9} \tag{13}$$

Radiation heat transfer Heavier is the aluminium amount of components, more important is the radiation of heat transfer. Besides, since exhaust gas are composed of H_2O and C_2O, radiation heat transfer increases by 10 to 30% (depending on aluminium amount in the nozzle).

The following formula allows to determine thermal flow:

$$\dot{q}_{rad}" = a_w * \sigma_T (T_c^4 - T_w^4) \tag{14}$$

With q the total heat exchanged (W), a_w coolant flow rate (m^3/s), σ_T specific heat of coolant (J/K), T_c temperature of coolant leaving jacket (K), T_w temperature of coolant entering jacket (K)

D. Computational Fluid Dynamics

In order to predict engine performance and validate its design, fluid dynamics studies are needed. These CFD studies, for Computational Fluid Dynamics, consist in studying the movements of a fluid, or their effects, by the numerical resolution of the equations regarding the fluid. All this thanks to a dedicated software such as ANSYS Fluent or SIEMENS Star CCM+. As part of the project, the Star CCM+ software was chosen for the simulations, the engine team being already familiar with this one, after using it on previous projects.

To simulate the combustion, physical parameters must be chosen. These are decisive and can drastically change the results of a simulation according to the relevance of the selected models.

The chosen models are:

- **Axisymmetric** - an axisymmetric model (Figure 4) makes it possible to simulate only a quarter of the engine and to significantly improve the calculations speed.
- **Steady** - a stationary flow is a good approximation in this study. Indeed, this simplified model, where the flow is not a function of time, is much used to save resources in the resolution of dynamic equations. It is for example widely used for studies such as the flow around an aircraft wing or fluid flowing in a tube. This simplifying solution will have very little impact on the phenomena taking place in the combustion chamber, the most interesting region.
- **Gas** - the ejected fluid is a gas from combustion.
- **Coupled Flow** - more resource-intensive, this method is more accurate and particularly suited to supersonic models.

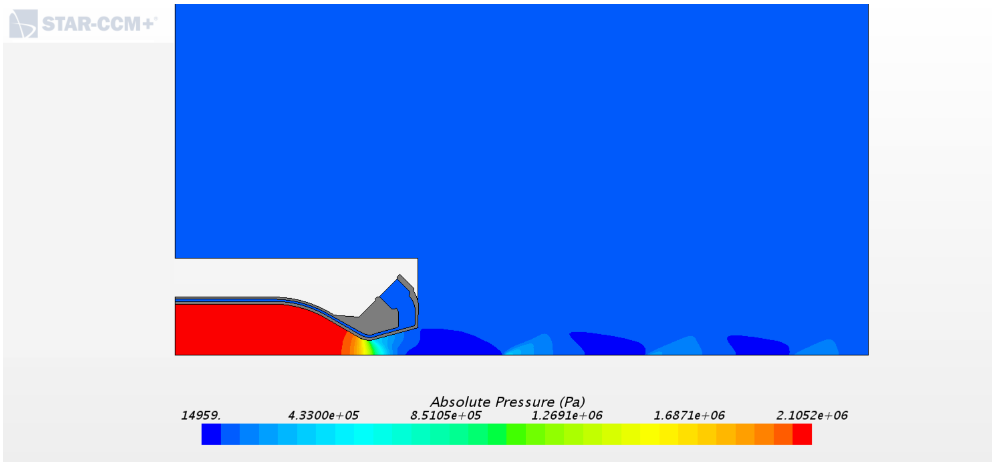

Fig. 4 Pressure of exhaust gases, *axisymmetric model*

- **Ideal Gas** - ideal gas model is generally used for nozzle simulations because it makes calculations faster by simplifying hypotheses, unlike the real gas model, which provides only a small benefit to the final result. This is supported by [4].
- **Turbulent** - the flow out of the nozzle must be turbulent, as this corresponds to a flow dominated by eddies, and apparent randomness. In the case of a rocket engine nozzle, the output stream is made of chemical transformations and shock waves that make the flow turbulent. The best methodology to choose is the Large Eddy Simulation (LES), which offers a very good resolution of the turbulent model. This solution was not chosen because of the large calculation time required. A simpler turbulent model was therefore chosen (i.e. K-Omega Turbulence).
- **K-Omega Turbulence** - more suitable than the usual K-Epsilon Turbulence, this model with two partial differential equations can predict turbulence effectively, to the detriment of more complex calculations.

The studies focus on four different topics: engine combustion simulation, cooling system efficiency, circulation in the cooling channels and thermal simulation of the LOX tank.

- **Intra-chamber combustion and flow** - This is to simulate the combustion that follows the ignition of the LOX/Jet-A1 mixture, both injected under pressure in the chamber. This study will make it possible to determine the flow velocities of the jet but also to visualise the profile of the jet at the engine outlet, in particular to highlight bad sizing choices.
  Results: The exhaust velocity calculated on the sizing Excel was *1946 m/s*. The simulation of the flow indicates *1982 m/s*, which corresponds fairly closely to what has been calculated.
- **Engine cooling efficiency and thermal analysis** - This is to ensure that the cooling system will be efficient enough not to exceed the melting temperature of Inconel. This hypothesis, already validated by the thermal sizing, will be supported by this simulation.
  Results: The temperatures found are very encouraging. Thus, a neck temperature of about *2950 K* is found, which is very close to the *2941 K* calculated in the initial Excel sizing. Finally, the Inconel wall in contact with the "hot" part of the engine indicates a maximum temperature of about *1160 K*. The melting point of the Inconel 718 being approximately 1550K, the calculated temperature leaves a certain safety coefficient.
- **Flow within the engine channels** - In order to simulate the flow of Jet-A1 and LOX in their respective channels, a CFD model was created. It will provide important data such as the pressures at input to achieve the desired injection pressure but also the desired injection speeds.
  Results (Jet-A1 channels):
  - Pressure: to get an injection pressure of 20 bar at the outlet, it is necessary to provide an injection pressure equal to 24.7 bar (Figure 5).
  - Speed: chamber injection speed will be around 30 m/s, validating what has been calculated in the previous sizing.

  Results (LOX channels):
  - Pressure: to get an injection pressure of 20 bar at the outlet, it is necessary to provide an injection pressure equal to 24.6 bar

  - Speed: chamber injection speed will be around 27 m/s, validating what has been calculated in the previous sizing.
- **Thermal analysis of the Liquid Oxygen tank** - A short thermal study of the liquid oxygen tank was carried out in order to record the temperatures that would prevail there. The tank is pressurised with nitrogen gas at ambient temperature.
  Results: The interest of this simulation is to check the temperature around the location of the joints seals. The results confirm the possible use of standard cryogenic seals.

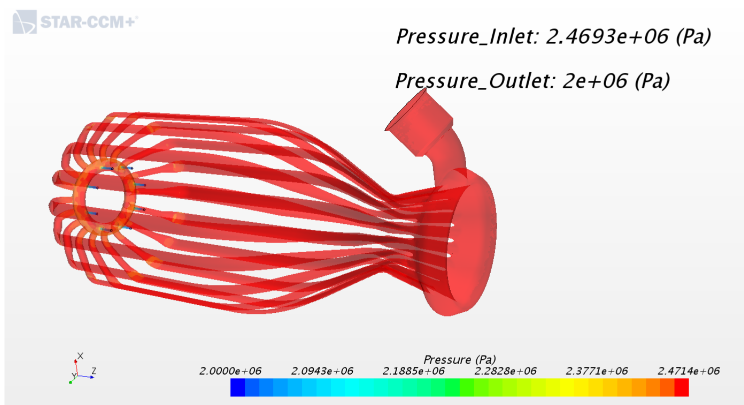

Fig. 5 Pressure results of the cooling channels

III. Optimisation for additive manufacturing and constraints

A. Optimised geometry

Additive Layer Manufacturing (ALM) allows for a wide range of shapes and designs otherwise impossible to realise with traditional manufacturing techniques. A lot of choices made in the primary design phases of the engine were easier to later implement in the final conception phase thanks to 3D printing. For instance, the decision to have a regenerative cooled engine, a cooling method generally much more complicated to implement on more conventionally built engine bells and combustion chambers, is here much less labour intensive and also cheaper to produce: two very relevant points in the commercial aerospace industry. ALM also allows for the optimisation of areas previously very hard to improve because of limitations of the assembling methods used. Still on the example of regenerative cooling, the team managed to incorporate in the final design optimised shapes with variable cross-section for the eighteen cooling channels placed around the nozzle and combustion chamber. Complex fuel, oxidiser manifolds and injection heads were only made possible to do in a single piece thanks to 3D printing. This single piece advantage allowed by ALM also simplifies the design process, as there are no more complicated interfaces between different parts subjected to very high pressure and temperature. Significant weight optimisation was not a main concern on this design since the purpose of this first iteration is not to be integrated on a rocket. However, it will certainly be in the next iteration of this engine and ALM will surely allow for complex weight-reducing structures in the engine body itself.

B. Limitations induced by ALM

Nevertheless, like every manufacturing method, there are some limitations on what can be printed using this ALM technique. The metal properties themselves need a special care as, for instance, the 3D printed parts have to be heat-treated in order to release the stresses induced by the sequential fusion of the different metal layers. There are also constraints on the shapes the printer can produce without adding more material in the form of physical supports that need to be removed after the printing process, and complicate the global manufacturing process. For instance, the maximum angle of the printed materials must not exceed 45 degrees of inclination relative to the thrust axis (constraints apparent on several areas of the final CAD model, Fig. 6). Another key problem only encountered with ALM is the issue of powder removal. The specific process chosen to print the engine, Laser Beam Melting (LBM), uses a very fine metal powder that is later fused by a powerful laser. This powder is deposited in layers (around 50µm in thickness) on the entire printing area. There is therefore leftover powder when the printing process is over. This powder will not only be all around the printed parts, but it will also fill all the negative spaces inside the parts themselves. In our case, the negative spaces are the eighteen cooling channels, the different manifolds and the injection heads. Optimal powder removal is critical in this study for a clean firing of the engine and a non-contamination of the propellants. The shapes and curves of these negative spaces were accordingly designed to allow for a complete removal of the powder by vibrating the entire engine and then flushing out the channels with compressed air.

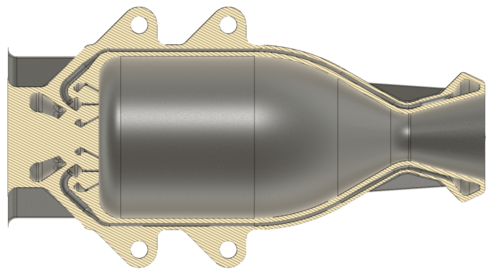

Fig. 6 Final iteration of the CAD design (sectional view)

C. Advantages and key figures

These are just some of the advantages and challenges that surround this relatively new manufacturing technique that this rocket engine design tries to take advantage of. Indeed, 3D printing allows to reduce engine cost and weight. The mass has been reduced by 30% compared to the average mass of a 1000N machined engine. Also, Additive Layer Manufacturing allows to reduce the manufacturing cost by 2.000 to 4.000 euros. That is a 30% to 40% reduction of the machined engine cost. The average price was estimated with [5]. Finally, thanks to 3D printing, the engine has been designed in a single part with a complex cooling channels system and optimised injection heads.

IV. Validation

A. Validation of the injection face

Injection face is one of the most critical parts of a rocket engine. Since the engine is a one-piece product, injectors will not be accessible on the final product. In order to validate the triplet injectors [6], a test was conducted using water instead of LOX and kerosene. This test defined the Inconel 718's roughness experimentally to calculate the diameter of each injector, so the required flow of fuel and LOX can be delivered to the combustion chamber. This parameter's value can not be published and is not relevant here since its experimental nature depends not only on the material but also on the 3D printer used, and its performances.

B. Static fire test

The study will end at the end of April 2019 with a static fire test. The engine will be integrated on a test stand developed by the team. It will collect data about pressure, temperature and thrust. This data will be analysed and compared with analytic results to evaluate numeric simulations and calculation methods with regard to additive manufacturing. The test stand [7] (Figure 7) delivers fuel and LOX which are stored in home-made tanks. They are pressurised by dedicated circuits with nitrogen. Data acquisitions are made by the LabView©software. The whole system is qualified through a study run by TILDA Conseil in partnership with ArianeGroup since the fire test location is based on their site at Vernon, France. This document introduced the study and its questions on design, printing and validation. The engine thrusts 1000N with a specific impulse of 200s. Designed with a regenerative cooling system and a triplet injectors, this product is printed in one piece. The current project studies only the engine in static fire environment, but future versions will improve each choice in order to integrate it on a student probe rocket.

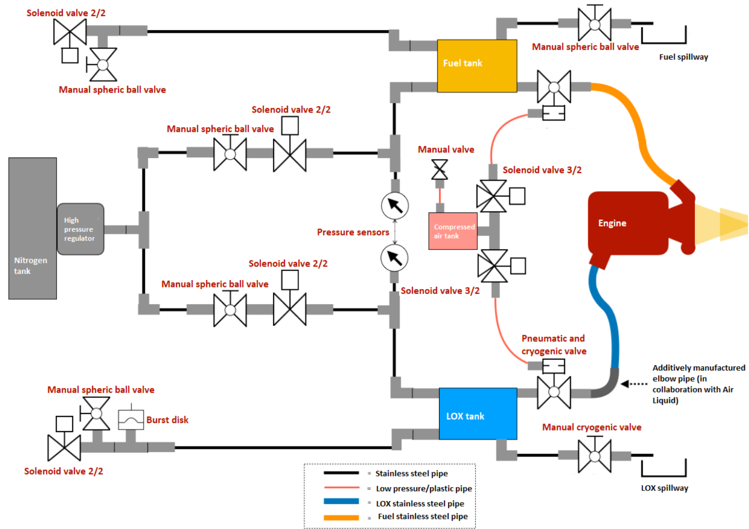

Fig. 7 Hydraulic and pneumatic diagram of the fire test stand

V. Conclusion and perspectives

Aurora Liquid Engine is a project that, beyond an academic subject, aims to test the effectiveness of 3D printing on a metal propulsion device. In the aerospace field, this technology would make great progress in terms of manufacturing costs but also in the production of ever more complex parts. Aurora's firing will pave the way for a better understanding of the use of this technology and will show the potential of additive layer manufacturing in rocket engine but also in rocket hydraulic systems. Indeed, a bend which will be use to lead liquid oxygen from tank to engine, will be printed by Addup and tested during the static fire test. Thanks to that, Air Liquide will evaluate ALM advantages with test results of the bend and enhance their acknowledges in additive layer manufacturing. From the design of the engine to its manufacture and finally its static shot, this project raises and attempts to answer the many questions related to the use of 3D metal printing in the aerospace field and helps his financial partners to explore ALM usability. It will allow new ways of design rocket engine with more complex injectors and regenerative cooling system. There will be an iteration of this engine to enhance regenerative cooling with a film cooling and optimise mass and cost to improve engine thrust. The next engine would thrust the first ESTACA sounding rocket which is in development.

Acknowledgments

The study was conducted as a final project at ESTACA by the two author and the co-author of this document, and by Robin Piebac[‖], Nicolas Deliry[**], Pierre-Louis Gautier[††], Augustin Coudray[‡‡], Loïc Valla[§§] and Fabien Massenet[¶¶]. It was evaluated on the 12th February during a viva by Dider Vuillamy[***] and Benoit Sagot[†††]. The liquid engine is printed by AddUp, who offered the team its production cost, and helped the team optimise the design. The fire test will take place inside ArianeGroup's site at Vernon, France. The operational safety study, directed by TILDA Conseil, is funded by ArianeGroup to authorise the fire testing of the engine on the testing bench developed by the team. The team thanks Air Liquid for its financial donation. Finally, the team thanks ESTACA Space Odyssey, the rocketry association of ESTACA, which helped the project financially, and by letting the team use its workshop.

[‖]Master's Degree in Aerospace Engineering, robin.piebac@estaca.eu
[**]Master's Degree in Aerospace Engineering, nicolas.deliry@estaca.eu
[††]Master's Degree in Aerospace Engineering, pierre-louis.gautier@estaca.eu
[‡‡]Master's Degree in Aerospace Engineering, augustin.coudray@estaca.eu
[§§]Master's Degree in Aerospace Engineering, loic.valla@estaca.eu
[¶¶]Master's Degree in Aerospace Engineering, fabien.massenet@estaca.eu
[***]Liquid Propulsion Engineering, ArianeGroup, didier.vuillamy@ariane.group
[†††]Teacher-researcher, ESTACA, benoit.sagot@estaca.fr

References

[1] RocketLab, *How to Design, Build and Test Small Liquid Engines*, 2nd ed., RocketLab, China Lake, California, 1971.

[2] Braeunig, R. A., *ROCKET PROPULSION*, http://www.braeunig.us, 1997, 2005, 2007, 2009, 2012.

[3] Huzel, D. K., and Huang, D. H., *Design of Liquid Propellant Rocket Engines (NASA SP-125)*, National Aeronautics and Space Administration (NASA), Washington DC., 1971.

[4] Vu, N. V., and Kracik, J., "CFD simulation of ejector: is it worth to use real gas models?" *The European Physical Journal Conferences*, 2018. doi:10.1051/epjconf/201817002075.

[5] Nieroski, J., and Friendland, E., "Liquid rocket engine cost estimating relationships," *AIAA Second Annual meeting*, 1965.

[6] *Liquid Rocket Engine Injectors (NASA SP-808)*, National Aeronautics and Space Administration (NASA), Washington DC., 1976.

[7] Santos, E. A., Alves, W. F., Prado, A. N. A., and Martins, C. A., "Development of test stand for experimental investigation of chemical and physical phenomena in Liquid Rocket engine," *Journal of Aerospace Technology and Management*, 2011. doi:10.5028/jatm.2011.03021111.