ISSUE 1 | 2019

ReSolve

Breadth of technology - Unlimited exploration

Boom Supersonic

Relaunching commercial supersonic air travel

PAGE 16

Renault

Aerodynamic optimization of Exhaust Gas Recirculation or car compressors

PAGE 42

ArianeGroup

Shaping the future of access to space

PAGE 48

NUMECA INTERNATIONAL

Breadth of technology bringing the power of unlimited design exploration

NUMECA's vision and strategy is essentially focused on innovation in all aspects of CFD technology, with a major focus on the productivity of our customer's design and analysis teams. Productivity implies a synergetic combination of product excellence, reliability of simulation methodology and overall communication between all involved in the design process.

The power of our solutions offers **Unlimited Exploration** as it drives massive productivity increases on the full cycle. Drastically reduced engineering preparation, set-up and simulation turnaround time provide the potential to explore the design envelope in more detail or go-to-market faster. This is supported by a wide **Breadth of Technology** of the NUMECA software systems as a large range of technologies is put forward in each step of the simulation process, from CAD treatment, structured and unstructured meshing to a combination of structured and unstructured solvers with technology geared to turbomachinery, combustion, multiphase, external aerodynamics, hydrodynamics, etc. all the way to the end-to-end optimization environment, including uncertainty quantification (UQ), robust design and morphing technology.

The OMNIS™ environment is built on these objectives, with all functionalities required to cover, in a highly user-friendly way, all the steps of the simulation process. A major functionality is provided by the co-processing capability, enabling to follow the convergence evolution of any quantity, or its unsteady equivalent, providing a unique insight into the "hidden" behavior of the solvers. The integration of a full design optimization framework, with advanced surrogate modeling and data mining, adds an additional dimension to the OMNIS™ unique environment. This is substantiated with morphing algorithms and smart optimization with AI and UQ. The future will hold adapted industry and application templates with best practices embedded within, allowing engineers and scientists to be efficient directly from the go and focus on the results. And as an extra dimension the OMNIS™ environment allows integrating 3rd party tools with the open API, setting new potentials for process streamlining and a breakthrough in collaborative engineering.

This second issue of our ReSolve magazine presents an overview of recent significant advances in CFD simulation technology achieved with the NUMECA software systems, covering a wide range of highly challenging applications, from aerospace to automotive to marine. These contributions illustrate the NUMECA strategy towards **Vertically Integrated Solutions**, offering the right technology for the job versus a general multipurpose approach. Our vision is to provide client solutions which put forward the best technology for the job.

In the aerospace sector, the testimony from BOOM Supersonic is confirming the impressive performance of the unstructured flow solver FINE™/Open with OpenLabs™ and the CPUBooster™, achieving results up to 14 times faster than other codes. With Masten Space Systems, simulations addressing the challenges of a reusable aircraft designed to fly to the edge of space, including hypersonic flow conditions and combustion, were performed. Another contribution describes the extended applications, over many years, of the NUMECA solvers and optimization framework to the development of the cryogenic engines of the ARIANE European launchers.

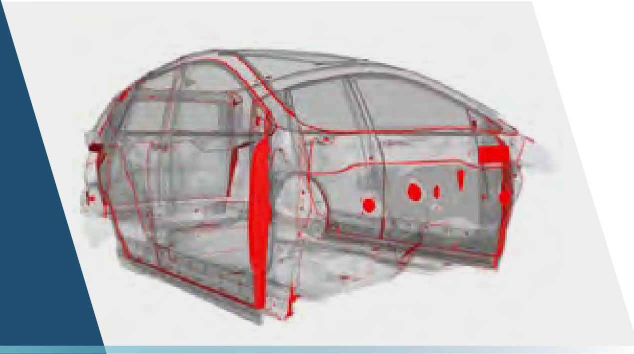
In the automotive area, significant contributions from Honda and Renault illustrate the growing penetration of our solutions in the automotive market. A significant landmark, demonstrating the innovative creativity of our developers, is a unique tool for closing undesirable holes in CAD geometry definitions, AutoSeal, which achieves its objectives in one hour instead of one week with any other tool.

With Renault, a detailed joint investigation allowed to evaluate the impact of Low Temperature Exhaust Gas Recirculation (LT-EGR) on the efficiency of the turbo-compressor, and with the Agoria Student group of the KULeuven the NUMECA software was extensively used to optimize their solar car for participation to the World Solar Challenge. In the marine area, the article in this issue illustrates some of the features behind the dominant position of FINE™/Marine in the major teams of sailing races in the world.

I would like hereby to particularly thank all the above-mentioned companies for their testimony and their confidence in the advanced NUMECA software tools.

Prof. Charles Hirsch,
President, NUMECA Int.

In this issue



Masten Space Systems

Reactive flow and heat transfer optimization for reusable spacecraft

PAGE 28

OMNIS™

Addressing today's and tomorrow's multiphysics simulation challenges - a focus on Automotive

PAGE 34

Renault

Aerodynamic optimization of Exhaust Gas Recirculation (EGR) for car compressors

PAGE 42

ArianeGroup

Shaping the future of access to space

PAGE 48

Honda demonstrates a major breakthrough in CAD preparation and meshing speed

It is common knowledge that 80% of the throughput time of a CFD simulation is spent on preparation and set-up. Typical challenges of this preparation process are poor quality of the available CAD and high complexity of the geometries. Dirty geometries containing holes and cavities and complex structures normally require a lot of manual input to result in a good quality mesh. It is a laborious process, making it difficult to create various design alternatives for iterative testing.

By **Akio Takamura,** *Chief Engineer, Honda R&D* and **Benoit Mallol**, *Head of Marine Products & Applications Group, NUMECA International.*

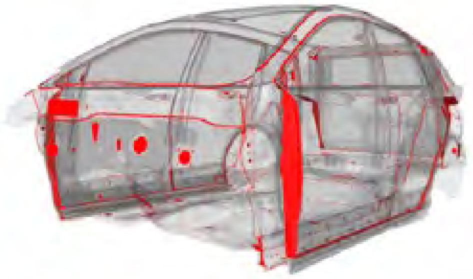
Many techniques exist to tackle these issues: from capping the holes and removing geometry features to wrapping the geometry, etc. These solutions are offered by many CFD providers, including NUMECA, and they all have their advantages and drawbacks. But none of them really respond to the full specification list that certain users are looking to fulfill: a solution that works for both CAD and STL files, that does not lose any detail of the geometry in the process and that does it as fast and as automated as possible. Honda had that exact same wishlist and found the answer in the NUMECA AutoSeal technology.

One of the typical reasons that CAD cleaning in the native CAD system can take days or even weeks depending on the complexity of the geometry, is the mere fact that the objectives of the CFD engineer differ from those of the CAD designer. Think for example about details like welds, seals, nuts, bolts, etc... All these have no added value per se for a simulation, but they are of key importance for manufacturing purposes, so the CFD engineer needs to deal with them. This is where AutoSeal combined with HEXPRESS™/ Hybrid can make a big difference, and Honda was one of its early adopters.

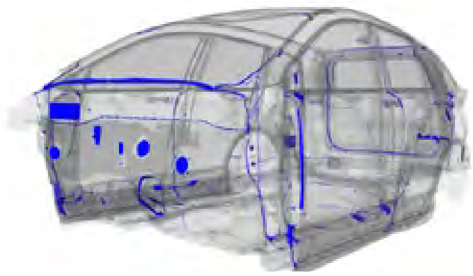
Honda was already a long-term user of HEXPRESS™/Hybrid, and the AutoSeal technology was a logical next step for making their design process more efficient. They chose to apply it for the first time for the design of cabin interiors, and they were very positive about the results. Mr. Akio Takamura, Chief Engineer at Honda Automotive, reported that where a skilled engineer typically needed one full week to close all the holes of the cabin space before, now this whole process was brought down to just about one hour with AutoSeal. And the robustness of the technology was definitely proven: They had a 100% success rate on 10 different vehicle models.

FIGURE 1 : Comparison of closure times between commercial CAD software and AutoSeal

Time to close in CAD software: 1 WEEK



Time to close in AutoSeal: 1 HOUR



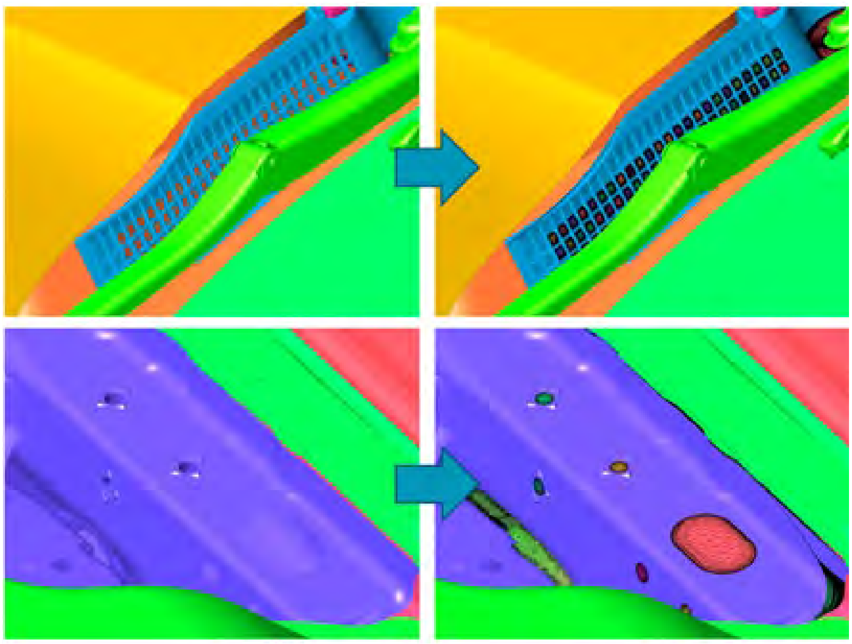
> "We have observed speed up improvements for each version of HEXPRESS™/Hybrid for the past years, but this one is a major breakthrough."

Here is how it works: Based on successive octree refinements, AutoSeal is able to detect which cells of a dirty geometry are actually inside or outside of it. This allows the system to progressively identify missing surfaces under a given user-defined threshold. AutoSeal then automatically closes the surfaces of these holes and cavities.

An additional advantage of the process is that it can all be done by one person and within the same environment: Passing to mesh set-up straight after AutoSeal has run, without the need to go back and forth with a design department for this.

In order to test whether these meshes return the exact same results as the CAD models would, Honda decided to make a comparison of the aerodynamic performance calculations. They looked at aerodynamic coefficients and passing wind speed of the heat exchanger on over 10 different models and found little to no difference between the models, stated Mr. Takamura.

FIGURE 2 : Before (left side) and after (right side) the usage of AutoSeal



To speed up the process further once the CAD is prepared, Honda used HEXPRESS™/Hybrid's unique capability to mesh in parallel on distributed memory. This technology allows for meshes to be generated in parallel on several computers or cluster modes at the same time. As a result tens of millions of cells can be created in a matter of minutes.

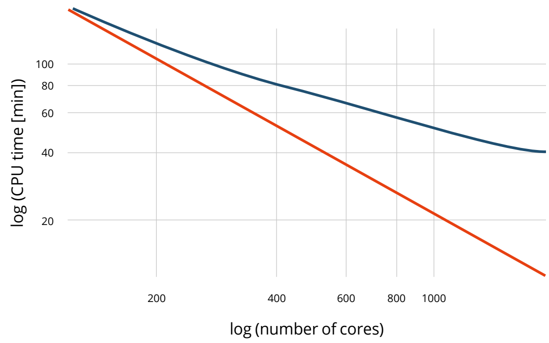
For example a mesh of 360 million cells that they made previously for the Honda CR-V model took just under 13 hours to create on 16 cores. With parallel meshing today this same mesh would take only 46 minutes on 1200 cores of a cluster, or the amount of time reflected in the table to the right depending on the amount of cores available.

FIGURE 3 : Geometry (left) and its surface mesh (right) of the CR-V model

As a next step NUMECA is working on the ability to reduce the mesh size of full cars. The concept is to mesh fluid/solid connections by imprinting the solid mesh made with a surface to volume method into the fluid made with a volume to surface method. The future of the automotive industry is bright with NUMECA's technologies for CFD.

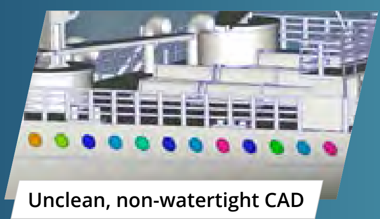
FIGURE 4 : Scalability performances of the mesh generation with HEXPRESS™/Hybrid

# cores	# cells per core	Time [min]	Speed up
120	3,000,000	176	-
240	1,500,000	107	1.65
480	750,000	74	2.37
720	500,000	60	2.92
960	375,000	52	3.38
1200	300,000	47	3.78
1440	250,000	43	4.11
1920	187,500	41	4.33



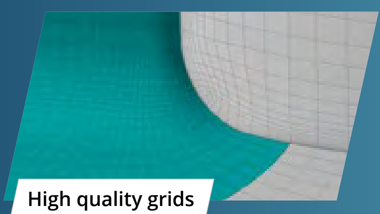
Tech Insight:
Automesh™

Solving some of your toughest meshing challenges like

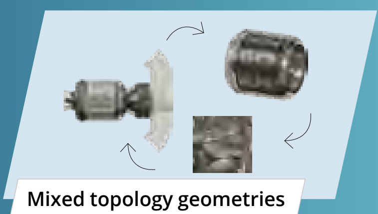


Unclean, non-watertight CAD

Complex geometries



High quality grids



Mixed topology geometries

More info and full capability chart here:
www.numeca.com/product/automesh

Ensuring your competitive advantage

Full flexibility

The meshing tools of choice - incorporated in any workflow, with any solver

Spending time where it matters

AutoSeal – from unclean CAD to watertight in a matter of minutes

High quality meets extreme speed

AutoGrid5™ structured mesher – the worldwide reference in turbomachinery meshing.

Drastic cut-down on engineering time

HEXPRESS™/Hybrid - a unique full hexahedral mesher for complex geometries – 100 M cells in less than 1 hour!


IMOCA
GLOBE SERIES
BOSS
HUGO BOSS
NOKIA
BOSS

VPLP Design: Revolutionizing hydrofoil design with advanced simulation technology

By **Xavier Guisnel**, *Naval Engineer, VPLP Design* and **Benoit Mallol**, *Head of Marine Products & Applications Group, NUMECA International.*

Hydrofoils have unleashed the speed of sailing boats since the last two America's Cups and are exclusively designed with CFD. The French company VPLP Design is at the cutting edge of hydrofoil concept and has worked with Alex Thomson Racing and Charal Sailing Team on the last generation of IMOCAs, to design efficient sailing boats that are literally flying over the oceans. Among all their objectives, VPLP has to identify the correct position of the hydrofoils in terms of angles and draughts for a required lift force. This objective traditionally required running 6 to 8 simulations per speed, until FINE™/Marine offered VPLP a new, innovative approach that reduced this workflow to one single simulation, while taking into account more physics.

Methodology

Being able to find the best position of a hydrofoil in a single computation implies that the target lift force should become an input of the simulation. The CFD code should automatically find a dynamic equilibrium position and reach the desired lift force at the same time. A new feature has been developed in FINE™/Marine to reach this objective and is now available as a quasi-static approach dedicated to hydrofoils: the flow solver adapts the hydrofoil rake and yaw angles at a given frequency according to the targeted lift. The subsequent predictions of the solver progressively and rapidly determine the dynamic equilibrium: stabilization is usually reached in about 1 to 2s of physical time.

This methodology requires both:

a) freedom in terms of hydrofoil motion which is made possible with the overset technique of FINE™/Marine,
b) a high-quality volumic mesh for accuracy and robustness.

A novel mesh generation approach based on HEXPRESS™

The proposed approach is to start NUMECA's full hex unstructured grid generator HEXPRESS™ from an initial curved block that follows the shape of the hydrofoil (see figure 2). This ensures a high-quality mesh at the hydrofoil surfaces but also at the domain boundaries.

FIGURE 1 : 3D representation of the wake of the hydrofoil

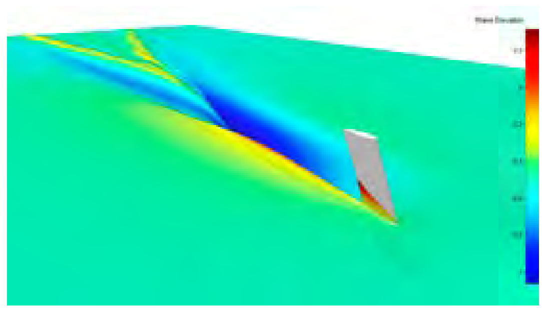
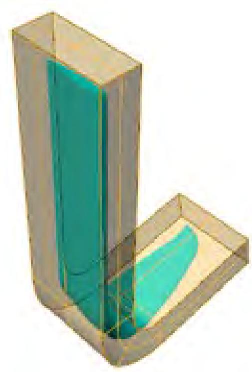


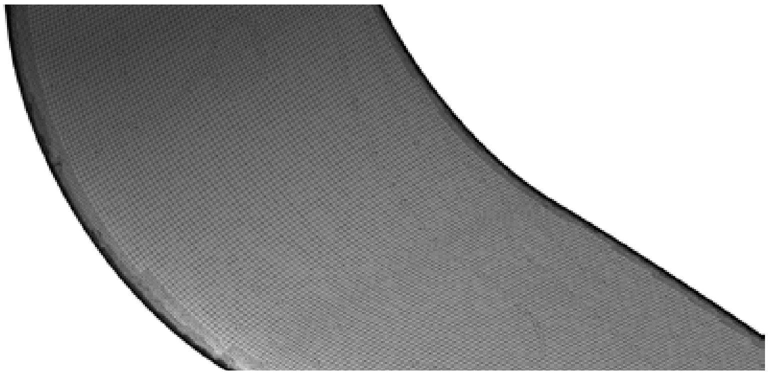
FIGURE 2 : Curved domain around the foil

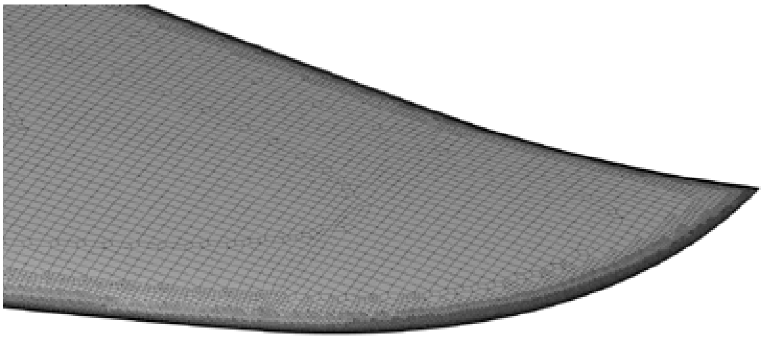


> "VPLP managed to reduce the simulation time of their design process by a factor of 8 while assessing more physics than before."

Mesh refinements and viscous layers are then performed with HEXPRESS™. Figures 3 and 4 illustrate the nice alignment of cells on the surfaces.

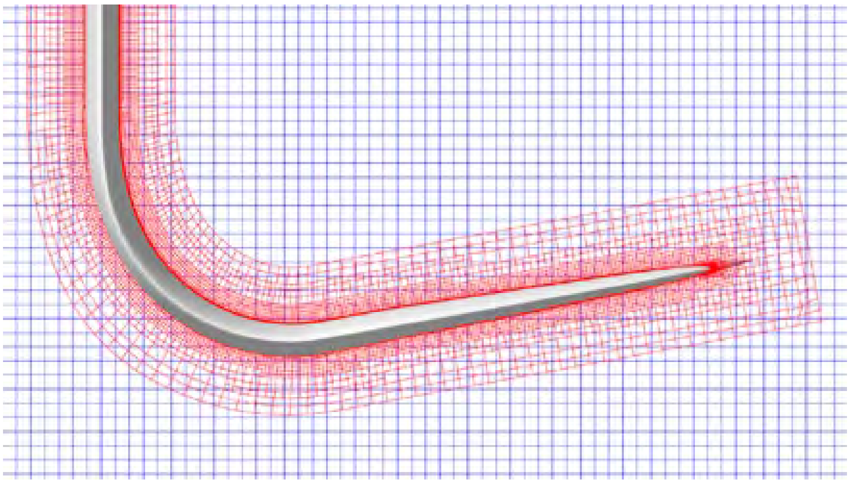
FIGURE 3 & 4 : Surface mesh on the shoulder (top) and the leading edge of the foil (bottom)





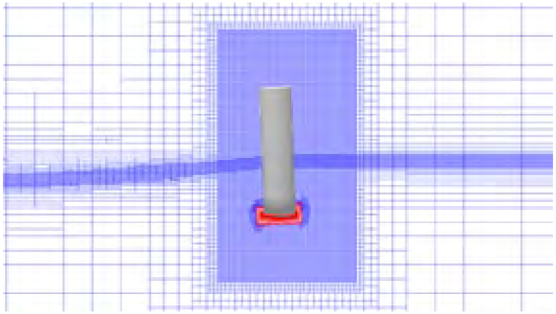
This hydrofoil mesh is hence placed inside a Cartesian background grid, allowing to travel through a virtual sea. These two meshes are connected thanks to the overset capability of FINE™/Marine, intercommunicating flow data at the boundaries of the hydrofoil domain.

FIGURE 5 : Front views of the meshes (blue: background, red: hydrofoil overlapping)



Furthermore, to ensure an ideal interpolation, the adaptive grid refinement technique dynamically refines the cells only where strictly necessary: at the free surface location during the simulation and at the overlapping grid boundaries. The total mesh size is thus reduced by 800k cells compared to an equivalent static mesh where the refinements should have been estimated.

FIGURE 6 : Side view of the dynamic free surface refinements



The importance of fluid-structure interaction

Because the hydrofoils are the only part of the sailing boat touching the water during their flight, they are exposed to high-pressure forces and their structure can deform sufficiently to influence their performance, even if this deformation will stay relatively small and linear. Hence, a modal approach can be used which only requires the mode shapes from the structure computed beforehand. The complete interaction can then be fully solved inside FINE™/Marine without interaction with an FEA code. Because the motion of the foil is relatively steady, a new and faster approach can be also used to solve the structure deformation: a quasi-static approach for deformation, as it is done for the motions. The additional CPU cost is only about 20%, yet it can bring an important impact on the design decision. In this case, for instance, the bending and torsional stiffness influence the dynamic equilibrium position, and therefore the flow field around the hydrofoil. The yaw angle changed from 2.53 to 3.40deg (see figure 7) and the drag from 8.741 to 8.935N.

FIGURE 7 : Yaw angle convergence comparing a modal “Modal_QS_UR” and non modal simulation “Motion_QS”.

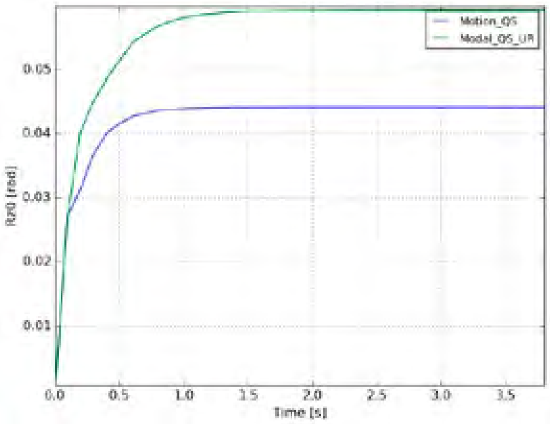
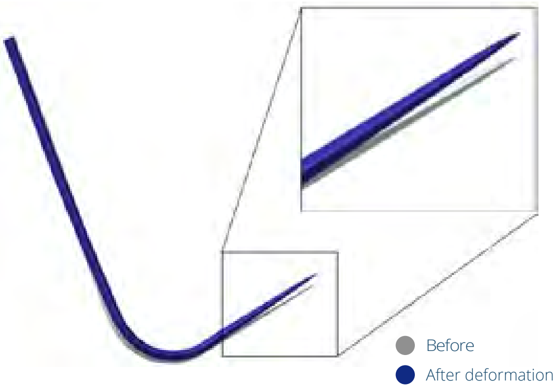


FIGURE 8 : Deformation of the structure during the FINE™/Marine simulation with the modal approach



Conclusions and perspectives

VPLP managed to reduce the simulation time of their design process by a factor of 8 while assessing more physics than before. With NUMECA's software dedicated to marine hydrodynamics they are able to study a large number of variants of their hydrofoils with a reliable, fast and robust process. As a next step, the power of NUMECA's FINE™/Design3D optimization solutions with coarse DoE, uncertainty quantification, and advanced surrogate modeling will allow optimizing the next generation of hydrofoils under real-world conditions.

Boom Supersonic: Relaunching commercial supersonic air travel

By **Michael Rybalko**, *Aeropropulsion Engineer, Boom Supersonic* and **Jean-Charles Bonaccorsi**, *Technical Director, NUMECA USA.*

> “Boom managed to achieve results up to 14 times faster than with their previous design environment.”
>
> *Tim Conners, Lead Propulsion Engineer at Boom Supersonic*

Founded in 2014 in Denver, Colorado, Boom Supersonic is redefining what it means to fly by building Overture, history’s fastest commercial airliner. Overture will travel at a speed of Mach 2.2 and a cruising altitude of up to 60,000 feet.

The fastest airliner in history

With pre-orders and options for 30 Overture airliners from Japan Airlines and Virgin Group already booked, the race is on to design the next generation supersonic plane. Besides the challenges that supersonic flight inherently imposes, Boom’s Overture designers also need to consider important environmental and social factors. The UN’s CORSIA climate agreement of carbon-neutral growth requires the offset of all international aviation emissions, whether subsonic or supersonic. To support this, Boom Supersonic plans to accommodate drop-in sustainable alternative fuels that will reduce their carbon footprint by roughly 80% and is actively looking at ways to incorporate environmentally minded innovations into Overture's design, without causing technical risk to their development timeline. One such innovation is its partnership with Prometheus Fuels, a company using clean energy to make zero-net carbon fuels out of carbon dioxide that is already in the atmosphere. Mitigating the community's exposure to the noise of sonic booms is another priority. They will do this by limiting supersonic speeds solely to trans-oceanic flight segments and implementing the latest noise-reducing technologies to ensure no increase of existing noise contours during take-off and landing.

Addressing the challenges

Due to the complexity of designing for supersonic speeds, Boom Supersonic engineers need to be able to test multiple conditions and try out many different design ideas. They also work with a very short time to concept, which means they need a solution that is fast to set up and even faster to get results. In a pilot program with NUMECA, Boom managed to achieve results up to 14 times faster than with their previous design environment, according to Tim Conners, Lead Propulsion Engineer at Boom Supersonic. NUMECA solutions have not only advanced the development of the XB-1 subscale demonstrator by providing a dramatically streamlined and highly automated workflow, but the NUMECA partnership has given Boom the opportunity for significant savings in computational resources and a reduction in design cycle time.

The unstructured hex-dominant meshing tool HEXPRESS™/Hybrid and the unstructured flow solver FINE™/Open with OpenLabs™ were selected for this task. Adding to the reduction of computational resources, Boom has been able to take advantage of CPUBooster™, NUMECA's unique convergence acceleration technique, for the majority of their runs.

> "NUMECA solutions have not only advanced the development of the XB-1 subscale demonstrator by providing a dramatically streamlined and highly automated workflow, but the NUMECA partnership has given Boom the opportunity for significant savings in computational resources and a reduction in design cycle time."

Optimization projects

Throughout the past year, several design studies used NUMECA solutions for both XB-1 and Overture, including:

- Inlet bleed plenum outflow venturi sizing and secondary flow path performance validation. These small cases were initially run on a laptop and later transitioned to the Rescale cloud computing platform as some of the first tests of the workflow.
- Isolated ejector nozzle simulations, for solver verification, which when compared with the previous in-house approach yielded a percent difference of 0.1% for nozzle gross thrust coefficient at cruise. Simulations with meshes of up to 100 million cells were run on 360 cores on Rescale.
- Cooling door sizing study. Results of 24 cases with 100 million cells were used to characterize the flow pumping characteristics of the ejector nozzle and helped match the flow schedules of the inlet and nozzle.
- Overture wing/body and wing/body/nacelle simulations. Meshes of 200-250 million cells were generated for a full-span model, and simulations were performed at cruise conditions to compare viscous results with lower fidelity preliminary design tools and inviscid simulations.
- Ejector nozzle analysis spanning the flight envelope (developed beside).

Highlight:
Ejector nozzle analysis

This analysis was part of the development of the XB-1 demonstrator. The objective was to run an analysis of the ejector nozzle spanning the flight envelope.

Meshing

To reduce engineering time and streamline the meshing process, the integration started at CAD level by naming each part/surface of the geometry. The CAD file was then transferred to HEXPRESS™/Hybrid, which applies specific mesh refinements based on the name of each part/surface. The naming is automatically performed in the CAD system, so each new iteration does not require an additional adaptation of the input geometry, saving a significant amount of engineering time.

Specific parts of the mesh can be easily swapped and connected to the rest of the geometry, automatically and in batch mode. The variable engine nozzle was set to the appropriate area for each condition, resulting in 24 different geometries.

A single ASCII input file was used to generate a high-quality mesh for each of the configurations, ensuring mesh consistency and again reducing engineering time. HEXPRESS™/Hybrid automatically captured the details of the modified geometry and repaired tiny gaps which can appear when changing the design.

The latest developments for viscous layers insertion with smooth transition were used in the near-wall region, with the wall distance of the first cell in the viscous sublayer specified for use with a low Reynolds number turbulence model.

The meshing process was done in parallel with a single in-house workstation, generating 24 meshes of 150 million cells each over the course of a weekend.

FIGURE 1 : View of the nozzle geometry

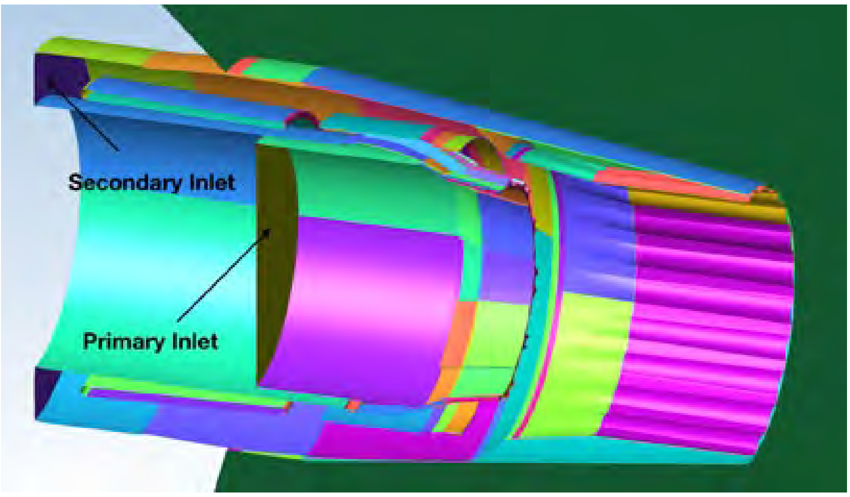
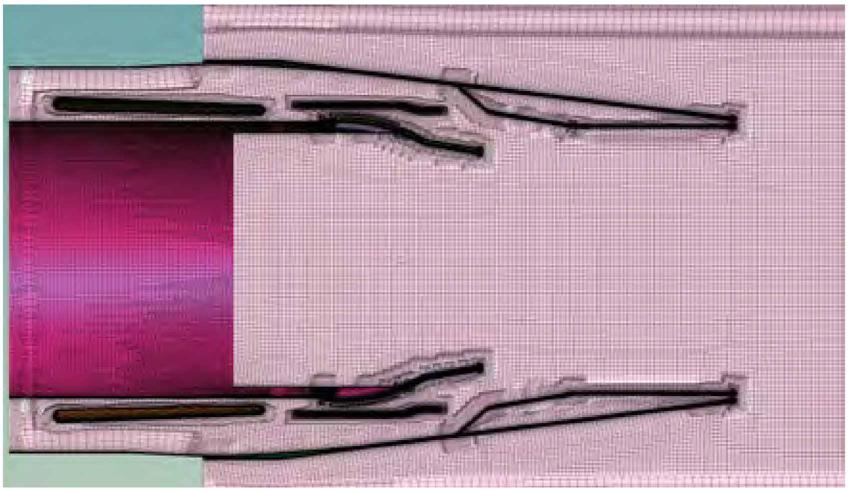


FIGURE 2 : View of the mesh



Simulations

The nozzle was simulated across the mission envelope from M0.02 at 5,000 feet to M2.2 at 40,000 feet. All cases were simulated at maximum dry thrust and with full afterburner. Five operating points for each of the 24 geometries were analyzed, resulting in 120 3D RANS simulations.

Analysis on the 150 million cell meshes was performed using 360 cores per job and was largely completed within a week and a half (3-4 hours per job). CPUBoosterTM, multi-grid acceleration, and OpenLabs™ were used in order to improve convergence and reduce CPU time.

FIGURE 3 : Centerline color contours of the Mach number

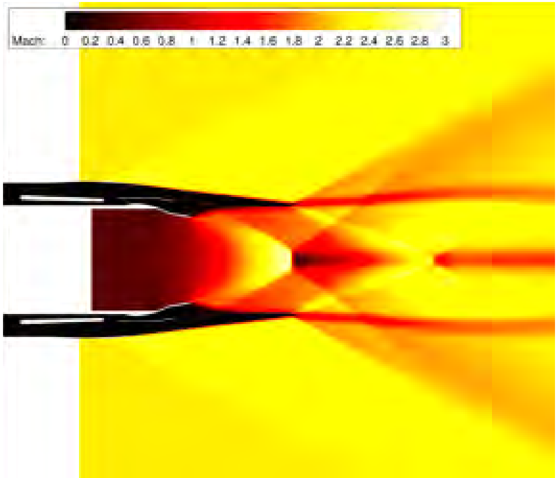
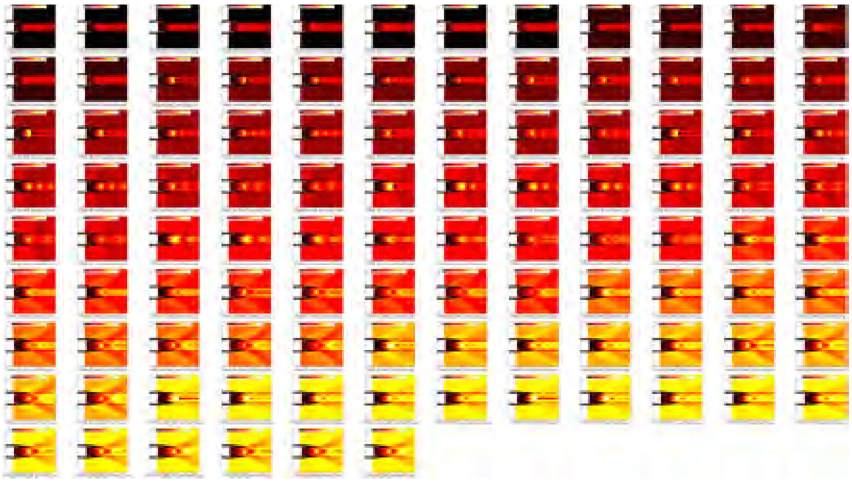


FIGURE 4 : Centerline color contours of the Mach number - mission envelope



Results

The results of these simulations provided key information applicable to a variety of multi-disciplinary design aspects, from performance to structural and hardware components allowing for evaluation and optimization of key elements such as:

- Flow pumping characteristics of the ejector nozzle
- Nozzle gross thrust coefficient used in the engine cycle and aircraft performance models
- Nozzle axial loads for nozzle attachment hardware design
- Temperature ranges and pressure deltas for material selection and structural design validation

According to Boom's engineers, this specific analysis provided the most impactful results generated to-date with NUMECA tools and highlights how well these NUMECA tools can work within a stable workflow.

Aerodynamic simulation of rotating wheels in a solar car

By **Thomas Holemans**, *student, campus Groep T Leuven, KU Leuven,* **Kristof Borgions**, *student, campus Groep T Leuven, KU Leuven,* **Maarten Vanierschot**, *Assistant professor, campus Groep T Leuven, KU Leuven* and **Colinda Goormans-Francke,** *Head of Academic Products and Applications, NUMECA International.*

> “ One of the most important objectives of solar-powered car development is to minimize power consumption by reducing drag.”

Every two years competitors from around the world set out on a grueling race across the Australian outback: the World Solar Challenge. The competing solar-powered cars are built by some of the brightest young minds on the planet. Student teams from all over the world push the limits of technological innovation by engineering and building a vehicle with their own hands, powered only by the sun. The competition is designed to promote research on solar-powered cars.

The Belgian team Agoria participates with their car called BluePoint. BluePoint is the result of years of research by different teams of thesis students from KU Leuven.

One of the most important objectives of solar-powered car development is to minimize power consumption by reducing drag. To find the optimum design within the limits of the competition, the Agoria Solar Team ran many simulations using OMNIS™, and were able to make a large variety of design changes thanks to its quick and easy, yet high-quality mesh production.

Study of the influence of the rotation of the wheels on the drag

Where in previous aerodynamic simulations of the car, the wheels were not taken into account or considered as static, the thesis of Kristof Borgions and Thomas Holemans[1] discussed in this article, focuses exactly on this part of the car. The influence of the rotation of the wheels in this research was investigated using FINE™/Open with OpenLabs™.

Simulations of the solar car were performed both with rotating wheels and with stationary wheels to be able to evaluate the impact of the rotation on the total drag of the car. A particularly interesting study as it compares simulation results taking into account real-life conditions on rotating wheels with results from wind tunnel tests where the wheels are stationary.

As a starting point, previous simulations of the car without wheels were considered, as performed by Vandervelpen and Uten[2]. To keep computational time within reasonable limits for a master's thesis work, the rim and tire were simplified. For example, the grooves in the tire were not taken into account, and the wheel arch was also simplified. The gap for the suspension was closed as the flow inside the car was excluded. In addition, only half of the car was considered in the simulation, neglecting the flow around the canopy.

FIGURE 1 : The simplified tire, rim and wheel arch

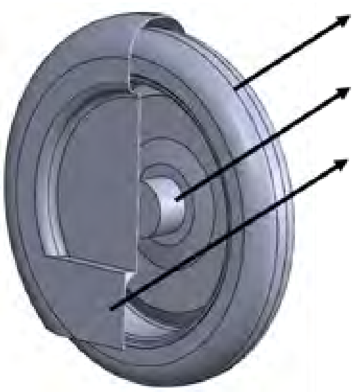


FIGURE 2 : Position of the wheel in the car

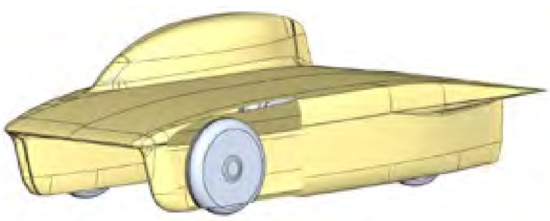
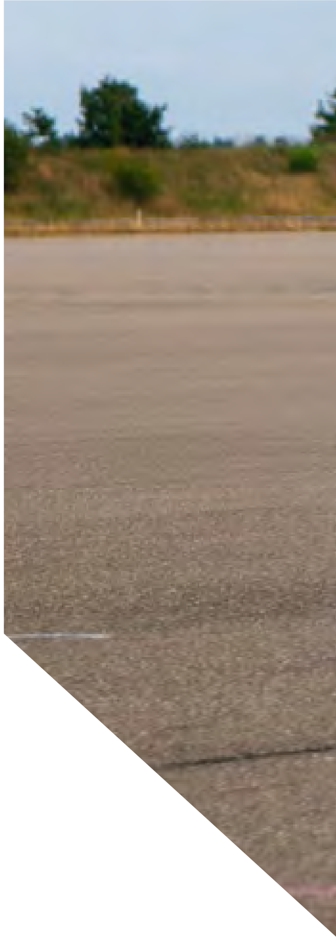


FIGURE 3 : Front view of the mesh generated by HEXPRESS™/Hybrid

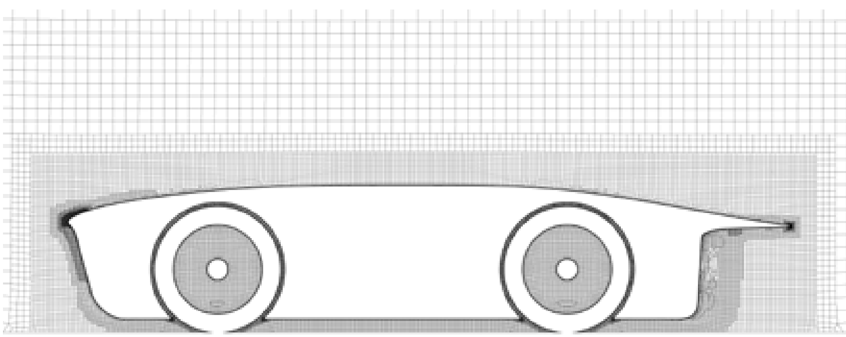


Meshing and set-up

Starting from a CAD file in Parasolid format, a full hexahedral mesh of around 11.5 million cells was generated.

OMNIS™/Hexpress was used to automatically group the many different surfaces from the original CAD. This drastically simplifies the successive steps of the simulation process. For example by having all narrow fillet surfaces in a separate group, additional refinements can easily be made to be able to accurately capture the curvature while keeping the cell count limited. The leading as well as the trailing edge of the car can be captured accurately through an appropriate refinement.

FIGURE 4 : Full hexahedral mesh at the leading edge of the car on a y-constant cut plane



Considering the significant focus on the wheels in this work, special attention was directed at the meshing. For example, the space between the wheel and the arch needed to be captured accurately. As this is variable, a proximity refinement was used: The cell size was based on the distance between The two surfaces, allowing for a limited cell count.

FIGURE 5 : Full hexahedral mesh around the wheel

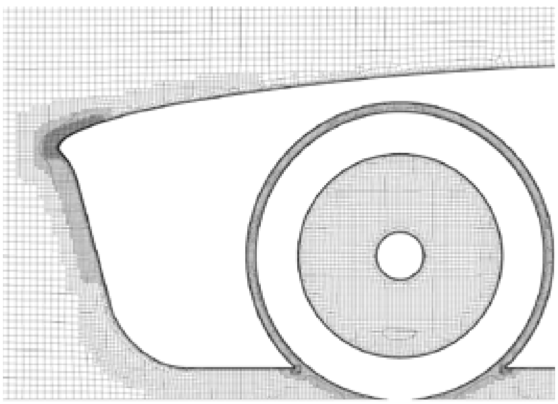
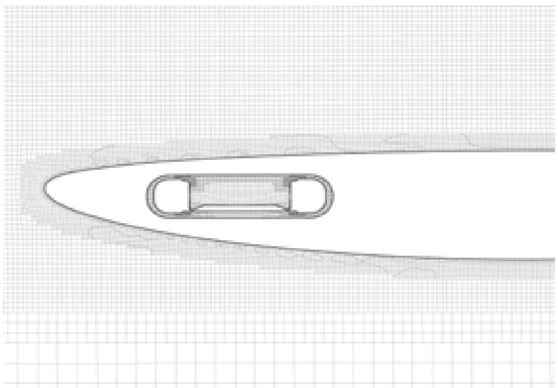


FIGURE 6 : Full hexahedral mesh on the tire surface

FIGURE 7 : Full hexahedral mesh around the front tire on horizontal cut plane



The simulations in the thesis were performed with the FINE™/Open solver for a 3D unsteady RANS simulation. Based on previous work [2] the k-ω SST turbulence model was used, as this would give the best results compared to wind tunnel measurements.

For the simulation with rotating wheels a Moving Wall boundary condition was used for the wheels, with an imposed radial velocity corresponding to the speed of the car. This was justified as the tire grooves and rim spokes weren't taken into account. In reality stationary wheels (in a wind tunnel) would correspond to a stationary ground, but in this research a moving floor was imposed to be able to compare the results of calculations made with the same conditions.

Results

The simulations were run on 26 cores with 160 Gb of RAM on a workstation of the 'Applied Fluid Mechanics and (Aero)Acoustics' research group of KU Leuven, campus Groep T Leuven. A first steady simulation was performed in 52 hours, corresponding to 4.5 CPU.h/Mpoints. For the unsteady simulations there was a remarkable difference: where the simulation with stationary wheels took 440 hours to stabilize, the simulation with rotating wheels took only 44 hours. This difference can be entirely attributed to the vortex shedding that is observed in the case of stationary wheels. Due to the rotation of the wheels, the amplitude and frequency of vortex shedding was significantly reduced.

FIGURE 8 : Definition of the cut plane

FIGURE 9 : Bottom view of the velocity for stationary (left) and rotating (right) wheels

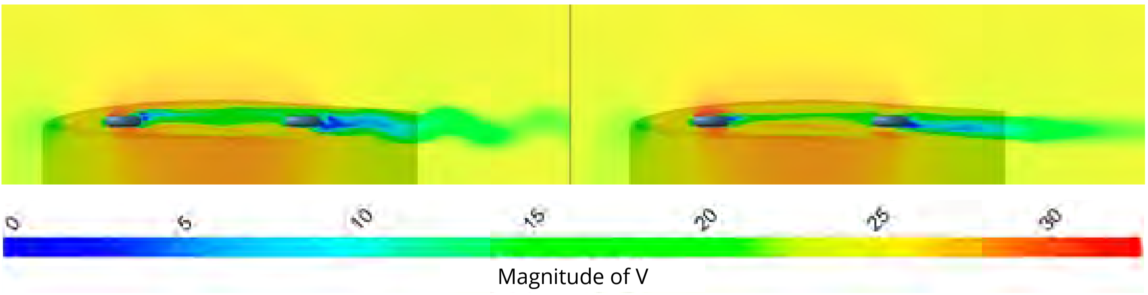
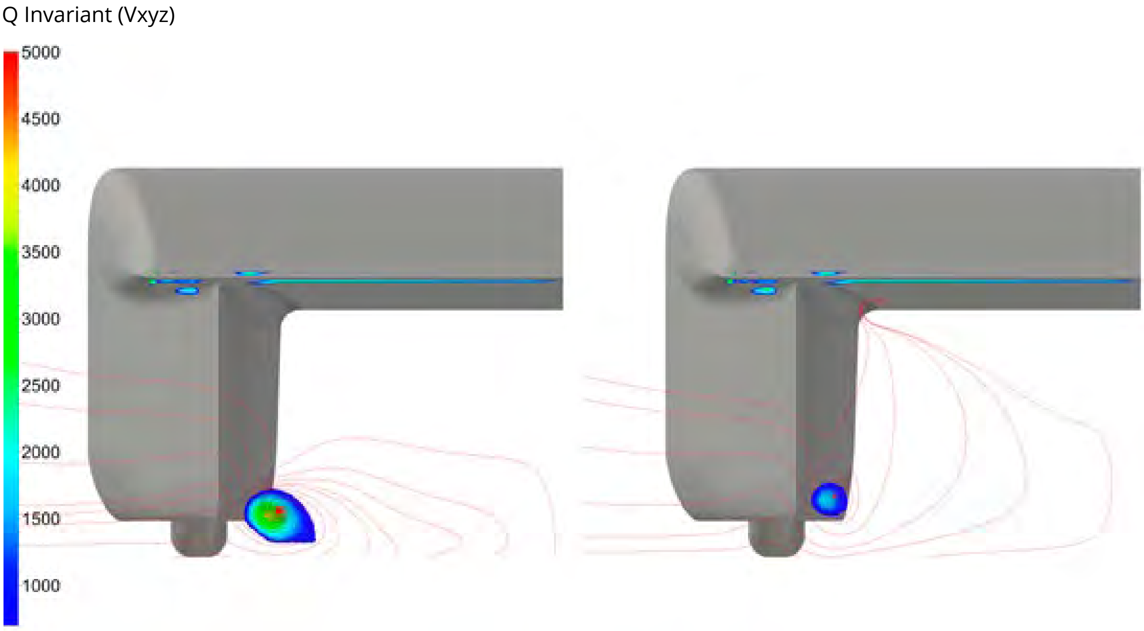


FIGURE 10 : Q-Invariant and surface streamlines (red) on a vertical cut plane downstream of the car (x=-1.8). Stationary wheels (left) show the vortex shedding compared to the rotating wheels (right).



Skin friction drag was influenced only a little by the rotation of the wheels. Pressure drag, however, showed to be largely impacted. The simulations also demonstrated that the front wheels had higher pressure drag than the back wheels. This can be explained by the lower stagnation pressure for the back wheel, as it is in the wake of the front wheel. Moreover the flow was approaching the back wheel under an angle coming from the vortex shedding caused by the front wheel. The pressure field showed that the pressure in the wake just downstream of the front wheel (on the left), was lower than the pressure in the wake of the back wheel (on the right).

Ultimately the rotation of the wheels reduces the drag on the wheels by approximately 40%. This has significant consequences for the car as a whole, as it reduces the CdA by approximately 10% - quite a considerable impact.

The simulations provided a more detailed insight in the flow field structures around the wheel caused by rotation. A small recirculation zone can be noticed at the front of the wheel and the wheel arch. Here the free stream flow from upstream and the flow between the wheel and the wheel arch come together.

The computed CdA compares within 0.95% with the one measured in a wind tunnel. As the present simulations did not include the canopy, the computed results were corrected based on previous results including the canopy.[2]

FIGURE 11 : Bottom view of static pressure on horizontal cut

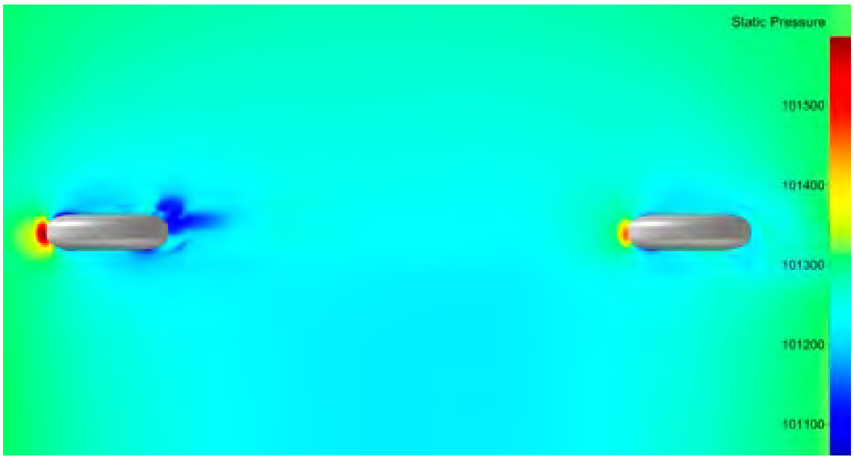
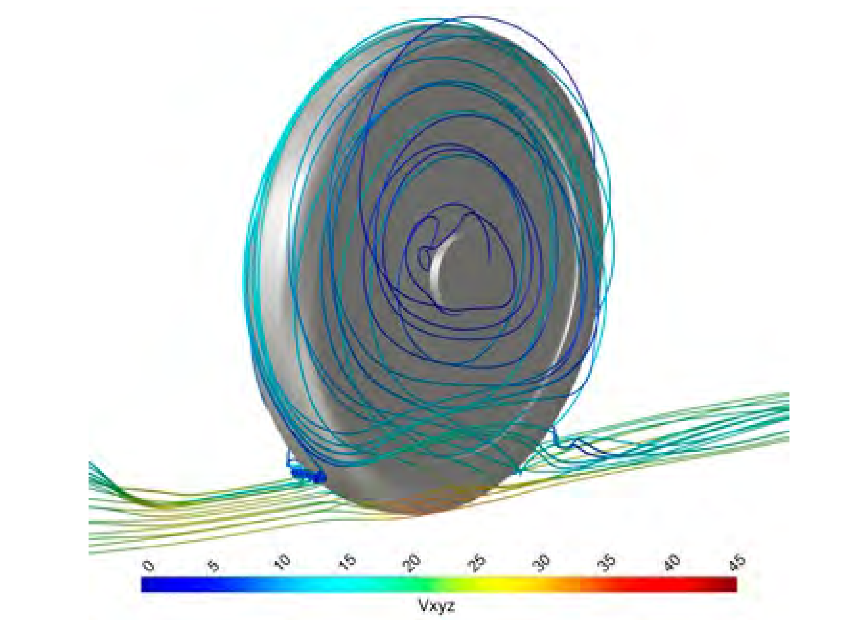


FIGURE 12 : Streamlines colored by velocity around the front tire



Conclusions

This work with OMNIS™/Hexpress and FINE™/Open with OpenLabs™ is an example of a work performed by students in the limited time available for a master's thesis. It provides more insight into the flow structure around rotating wheels and its significant impact on pressure drag. Rotation of the wheels causes as much as 40% decrease in drag on the wheels, resulting in a 10% reduction of the CdA of the complete car. This work helps in particular to increase confidence in the comparison of simulation results with experimental results in road conditions and wind tunnels.

References

1 Borgions K., Holemans T., Aerodynamic simulation of rotating wheels in a solar car. KU Leuven, Faculty of Engineering Technology. Master thesis, 2019.

2 Vandervelpen E., Uten J., Testing of turbulence models for the aerodynamic simulations of a solar car, KU Leuven, Faculty of Engineering Technology. Master thesis, 2018.

Masten Space Systems: Reactive flow and heat transfer optimization for reusable spacecraft

Masten Space Systems is a private company founded in 2004 headquartered in Mojave, California, USA by David Masten. Masten's focus on reusable rocket technology is driven by the goal of enabling space transportation and reliable planetary landers for the Earth, Moon, Mars, and beyond.

By **Allan Grosvenor**, *Aerodynamics Lead, Masten Space Systems, USA,* **Jean-Charles Bonaccorsi**, *Technical Director, NUMECA USA* and **Jan Anker**, *Head of Combustion Modeling Group, NUMECA International.*

One of Masten's recent pioneering space flight vehicle projects was to design a reusable aircraft designed to fly to the edge of space, the XS-1. The XS-1 aircraft will be capable of launching a 3,000-pound spacecraft to the earth's orbit at a cost of $5M, which is ten times less than today's launch systems. A key objective of the program is to fly the XS-1 10 times in 10 days in order to demonstrate its "aircraft-like" operability, cost efficiency and reliability. Key anticipated characteristics of the aircraft include a physical size and dry weight typical of today's business jets.

One of the new innovations in this effort is a full-scale additively manufactured aluminum thrust chamber assembly for the 25,000 lbf LOX/LCH4 dual expander propulsion system. The dual expander cycle offers an ideal cycle in this size range, enabling reusability, optimal closed-loop performance, and cost effective design and production price points.

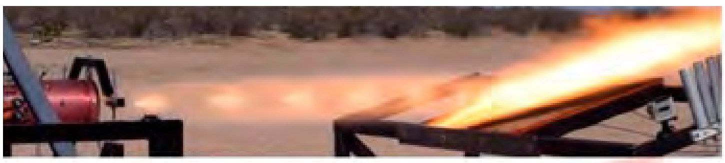
CFD for the simulation, analysis and optimization of critical spacecraft components

Masten Space Systems is using FINE™/Open with OpenLabs™, NUMECA's unstructured multi-purpose CFD solver package, for the design and analysis of various elements of reusable spacecraft and lunar vehicles for the testing of thrust chambers. In this article we talk about the numerical studies of three of their aeronautical configurations.

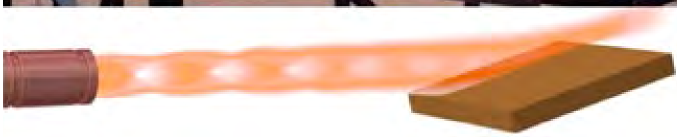
Simulation of plume impingement and thermal load

For the testing of thrust chambers, Masten Space Systems had to determine the temperature on the impingement plate in order to avoid reaching critically high temperatures. To do this, they simulated the impact of a hot plume at multiple times the speed of sound and a temperature hotter than the melting temperature of steel on a plate by using an inert multi-species approach. The results are shown in Figure 1 below, comparing the live fire test of the plume impingement in the photograph with the predicted temperature simulated by CFD. There is a clear qualitative agreement between the two.

FIGURE 1 : Plume impingement test. Comparison of a photograph and the simulated static temperature



Live fire test



CFD static temp.

"Masten Space Systems is using FINE™/Open with OpenLabs™ - NUMECA's unstructured multi-purpose CFD solver package - for the design and analysis of various elements of reusable spacecraft and lunar vehicles."

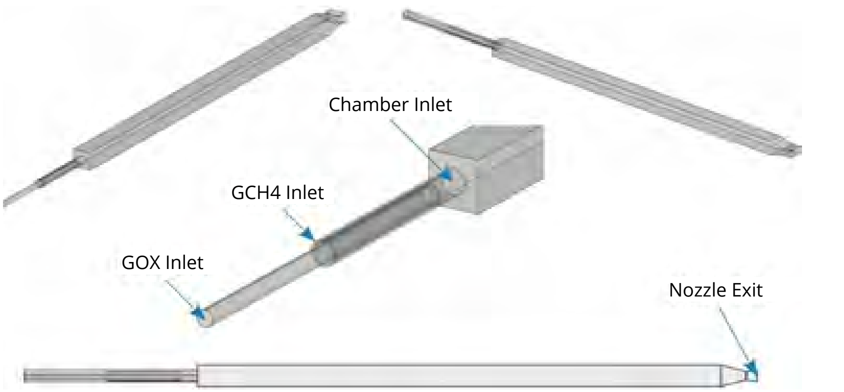
Injector study

For the optimization of spacecraft it is important to clearly understand the conditions of the injectors, combustion chamber and nozzle of the rocket engine. One of the critical points for example is that the walls cannot be subject to excessive thermal stress.

To analyze the combustion process and heat transfer in a generic injector configuration, a CFD study was carried out using a reactive multi-species simulation approach with a reduced chemical mechanism.

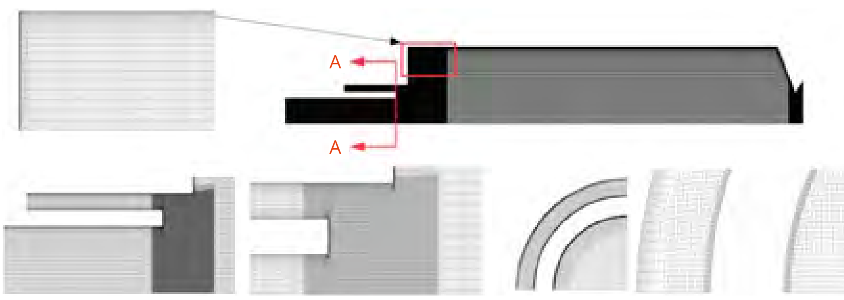
Figure 2 beside shows the injector geometry, consisting of two concentric pipes connected to a rectangular chamber. Via the inner tube Gaseous Oxygen (GOX) is injected, whereas Gaseous Methane (GCH4) is injected around it via the outer pipe. The purpose of the chamber is to create a recirculation zone, preventing the flame from blowing out. The recirculation of hot combustion products ensures a stable continuous combustion process after the engine is fired.

FIGURE 2 : Injector geometry



Two grids were generated: one coarse mesh at 1.8 million cells and one refined mesh containing 2.4 million cells with cylindrical viscous layers as shown in Figure 3.

FIGURE 3 : Injector computational grid — quarter domain, 2.4 million cells

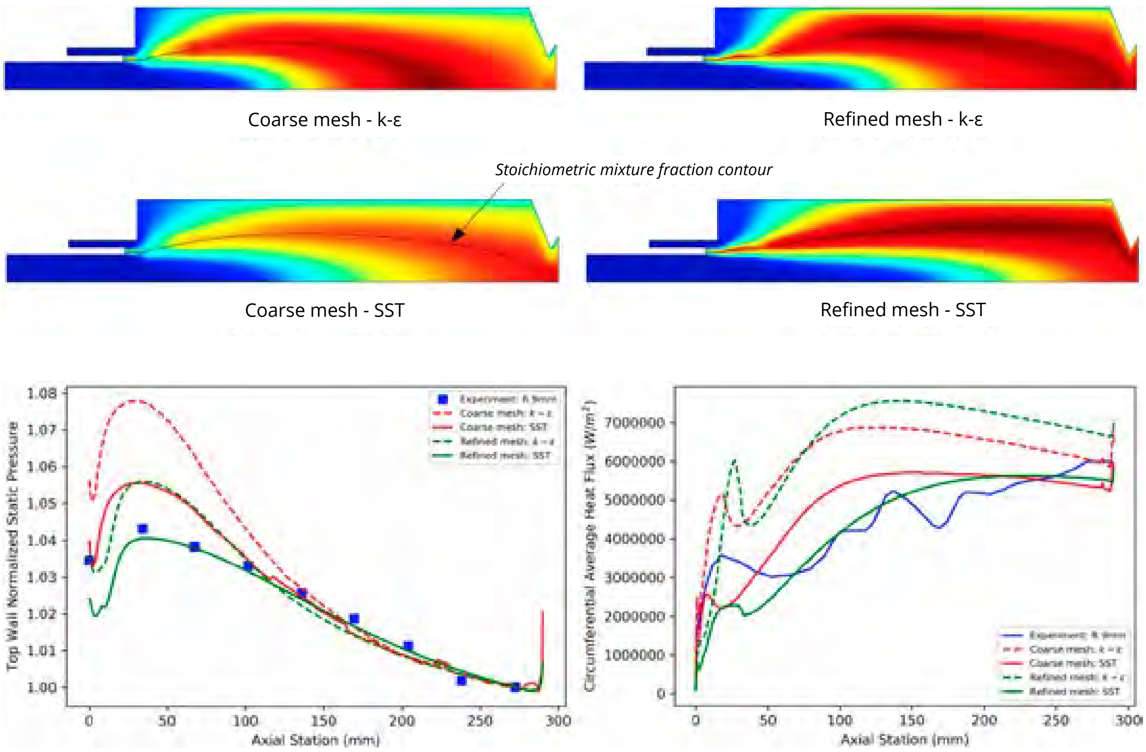


Side view vertical dimension scaled 10x

Section A-A

The simulation results were in good agreement with the measurement data. Figure 4 below shows the simulated temperature field and a comparison between the simulated and experimental values of the static pressure and heat flux.

FIGURE 4 : Predicted Injector flame temperatures, comparison of predicted static pressure and heat flux with experiment



Optimization of the cooling channel of the combustion chamber and nozzle of a rocket engine

To evaluate the potential of optimizing the cooling channel, a generic test model was constructed with 180 cooling channels. In the baseline design, the cooling channel has an Aspect Ratio (AT) of 4.

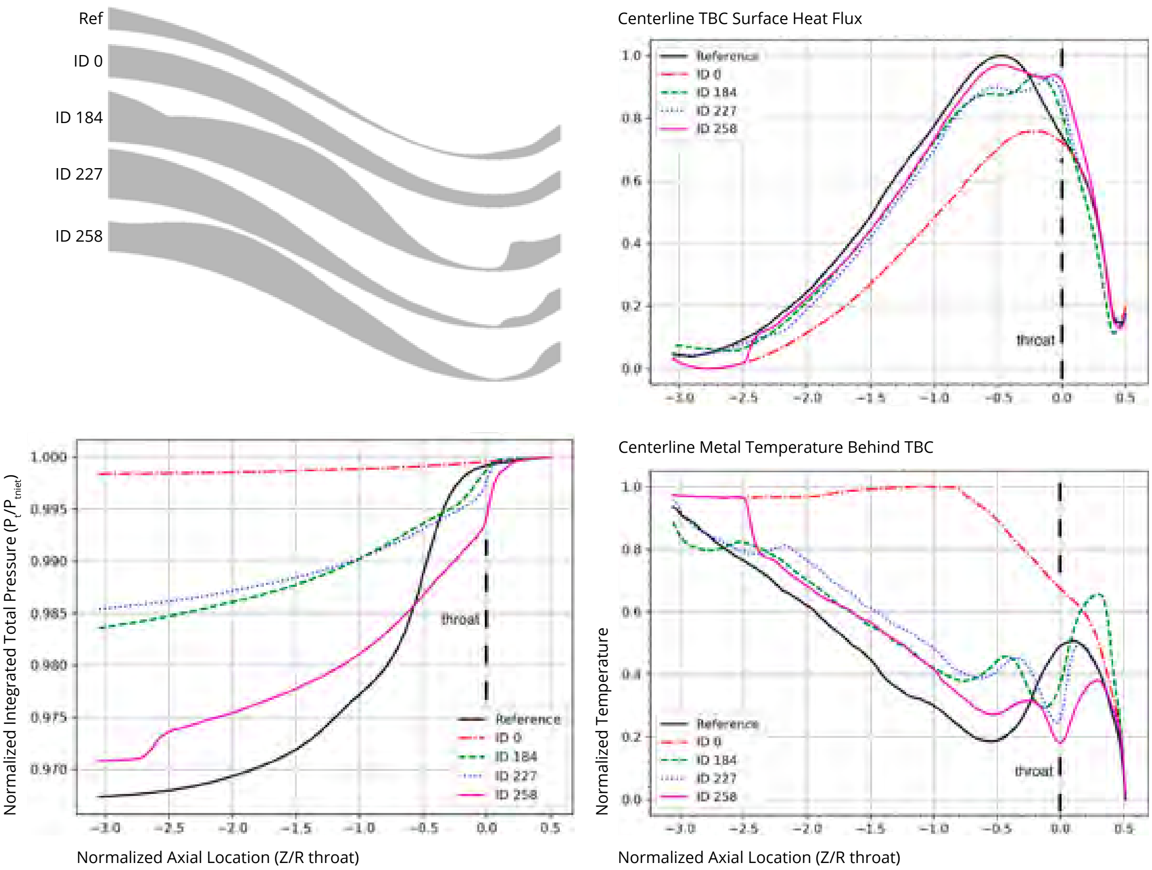
> "Conjugate heat transfer computations were run systematically in an automatic optimization loop. After analyzing the results, three designs demonstrated higher performance than the baseline design, as well as a manual design reference."

Figure 5 below displays their total pressure, heat flux and Thermal Barrier Coating (TBC) temperature predictions.

These three designs all included contractions of the CH4 channel in the vicinity of the nozzle throat and then an opening up of the passage down stream. On one hand this structure appeared to accelerate the channel flow in the throat region, resulting in higher heat flux, and therefore greater cooling in that area, and maintain attached flow downstream. On the other hand, the opening up of the channel downstream appeared to reduce total pressure losses compared to the reference design.

The uniform aspect ratio baseline (ID 0) produced the lowest total pressure loss by not accelerating the channel flow anywhere, but it subsequently demonstrated the lowest cooling performance. In the search for an optimized design, Self Organizing Maps (SOM) were used to cluster the different samples in a space, such that high-dimension space data can be visualized in 2D maps and identify the design that leads to the best compromise between cooling efficiency and pressure drop in the cooling channels. By using SOMs the design IDs 184, 227, and 258 were singled out as the ones having the most promising characteristics.

FIGURE 5 : Three sample channel geometries displaying higher cooling performance with lower pressure drop, ID 0 represents database baseline, and Reference represents a reference design that would arise from conventional design practice

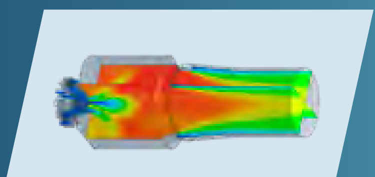


Full technical details of this case are available in the JANNAF paper - authors A. D. Grosvenor, G. S. Rixon, M. Kuhns, A. Demeulenaere, J. C. Bonaccorsi, D. P. Gutzwiller, J. E. Anker, L. Romagnosi, K. Claramunt, 2018, "Optimization of LOX/Methane Cooling Channel Geometry", Paper 5690, 65th Joint Army-Navy-NASA-Air Force (JANNAF) meeting, Long Beach, California, USA.

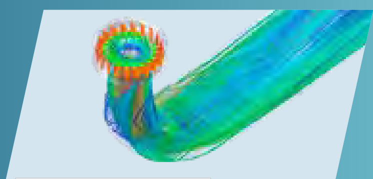
Leonard, B., Temmerman, L., 2016, "Modification of the SST k-omega model to correctly predict the shock structure in supersonic flows", Personal communication.

Tech Insight:
FINE™/Open

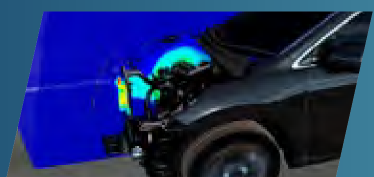
Solving some of your toughest multiphysics challenges like



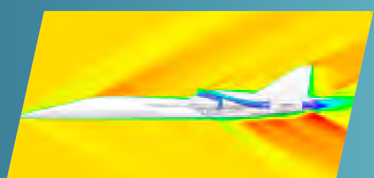
Combustion



FSI - Modal or Full Embedded Solver



Conjugate heat transfer



External Flows - incl. Supersonic flows

More info and full capability chart here:
www.numeca.com/product/fineopen

Ensuring your competitive advantage

A solver that goes with your flow

Choose between a pressure or density based approach – both tuned to performance

Speed with patented CPU Booster

Remove the boundaries to what you can analyze and save substantially on CPU spend thanks to 10x speed-up

Freedom to customize with OpenLabs™

Easily adapt the code to fit your challenges with expression based functions

Cutting-edge design and optimization including Uncertainty Quantification

Optimize your design for performance, manufacturability... and the real world

OMNIS™ - Addressing today's and tomorrow's multiphysics simulation challenges

A focus on Automotive

The automotive industry has been using CFD tools in the design and optimization processes of all parts of the vehicle, from external aerodynamics to noise reduction to thermal management to internal combustion etc. Most often within the design process, all of the above are combined together in what is called a virtual prototype to allow optimizing the vehicle as a system.
The physics behind the various applications are however quite often very different, which translates into the need of having dedicated technology for each specific application. As a consequence the designers, scientists or engineers are often using a large number of various CAE codes and software tools with different interfaces (GUIs), data set-up, structures and formats, each of them focusing on their specific discipline, with no or poor connection between them. A more global approach to the whole CAE workflow for a multidisciplinary design and optimization loop has become an absolute necessity for these users.

This is where OMNIS™ comes in: an end-to-end CAE environment with dedicated tools to solve each step of the simulation process quickly and efficiently. OMNIS™ gives global approach to the whole CAE workflow from design to results analysis, driven by the user through an ultra slick user-interface, or controlled automatically with the Python API or even by the optimization module. Its unified user-interface reduces the user's learning process and its consistent datastructure avoids slow file conversion and annoying corruption errors. The unique OMNIS™ framework enables tools to communicate with each other within one workflow, while its multiple solvers give access to a broad range of technologies for solving any fluid/acoustic flow problem, with the added flexibility of being able to plug in in-house or opensource solvers and tools via the API.

This article shows how OMNIS™, with its breadth of CFD technology behind, solves the most complex fluid flow challenges for automotive design, while at the same time reducing engineering and solving time, in a fully streamlined easy-to-use collaborative workflow.

The example of Figure 1 illustrates various applications such as: underhood thermal management, external aerodynamics, power-train, acoustics and underhood components like turbochargers. The challenge? Very different physics, different departments involved and tough constraints on throughput time.
OMNIS™ responds ...

By **Yannick Baux**,
Head of Turbomachinery Products & Applications Group, NUMECA International.

FIGURE 1 : Automotive fluid flow application overview

Simulation preparation

Setting up a successful simulation starts with preparing the geometry model for the simulation. In addition to creating the design directly in OMNIS™, an external geometry can be imported by directly opening the most popular file formats such as ACIS, IGES, STEP, STL, Parasolid, CATIA, Pro/ENGINEER or SolidEdge.

In OMNIS™, the geometry is preserved throughout the simulation process, ensuring lossless transfer between the modules and consistency of the analysis. To ensure maximal reliability of the initial design, the OMNIS™ data structure points directly to the CAD data.

All material, physical and numerical properties are linked to the CAD model entities. Identical CAD naming and hierarchies will guarantee consistency of the simulation set-up and will minimize user input within automated workflows.

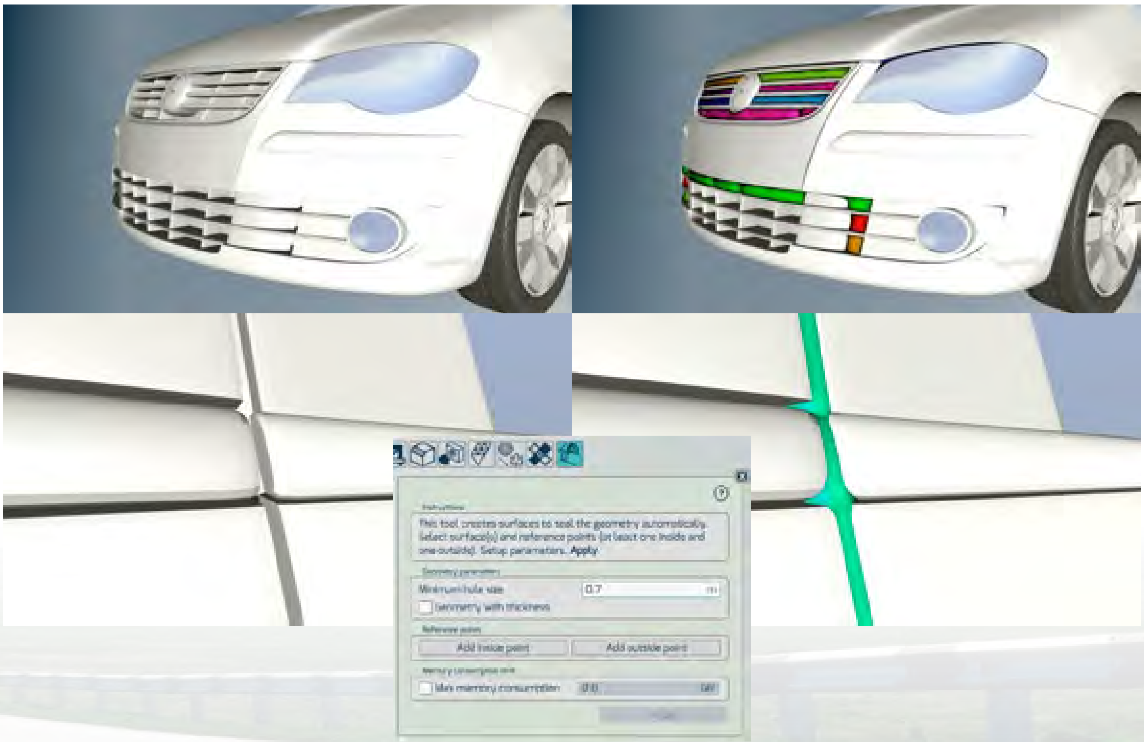
Production-level geometries can contain gaps, interferences, fasteners, and very small features. These features are often necessary for manufacturing, but add unnecessary complexity for simulation. Edit design tools become necessary to prepare the geometry for the simulation in the most automated way possible.

FIGURE 2 : Native bi-directional CAD import

In particular, one of the main time-consuming operations consists in closing the topological holes and gaps such as the gap between the car door and the passenger compartment, or the holes left after removing the screw bolts that join the two parts of the turbocharger volute. In this respect, the innovative AutoSeal outperforms all other standard edit and repair geometry tools.

Repairing a non-watertight non-conformal arbitrary geometry in one-click, cutting down the engineering time from days to a couple of hours, is now possible. Honda, for example, reported that a skilled engineer typically needed one full week to close all the holes of a cabin space before. Now this whole process has been brought down to just about one hour with AutoSeal.[1]

FIGURE 3 : AutoSeal gaps and hole closing for external aerodynamics



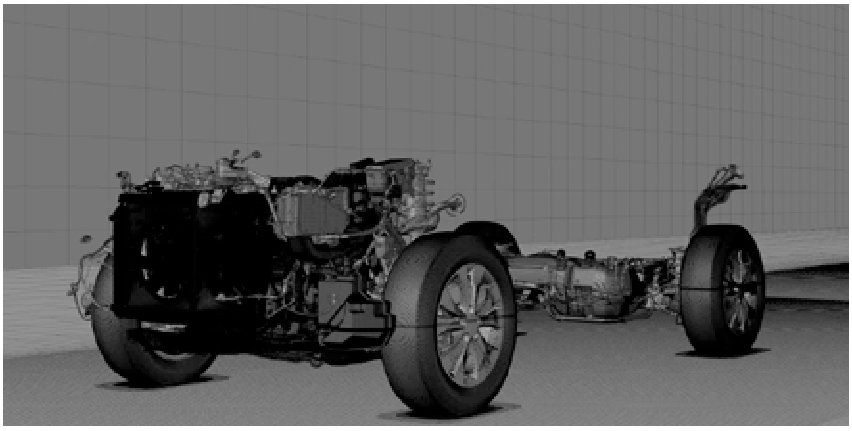
[1] For more details, check out the article "Honda demonstrates a major breakthrough in CAD preparation and meshing speed on page 6-9.

Pre-Processing

There is no single mesh generation technique available today - or seemingly in the foreseeable future - that answers all applications' requirements as they largely differ by size, shape, complexity and relevant physics. The solution necessarily comes from combining mesh generation techniques. The OMNIS™ mesh generation strategy is twofold: enhancing all mesh techniques to their best, and combining the most appropriate within the computational domain.

Unstructured mesh generation offers more flexibility to handle geometry complexity. When it comes to flexibility, OMNIS™/Hexpress users can choose between full hexahedral meshes with hanging nodes, or mixed element conformal meshes, "not-so-clean" geometry tolerant volume-to-surface approach, or surface-to-volume approach, inflation, deformation or extrusion boundary layers inflation techniques, and much more. Engineers at Honda for example, perform and analyze CFD aero-thermal computations of the underhood: radiator fans, flow around engine bay/peripherals, exhaust system, etc. Their pre-processing phase used to be time-consuming and cumbersome. When they switched to OMNIS™/Hexpress, they managed to divide their CPU time by 3 and their engineering time dropped to 30 mins/mesh instead of weeks.[2]

FIGURE 4 : Hybrid Hex-dominant mesh of the full underbody incl. engine (top) and zoom of the cooling fans of a Honda CR-V (bottom)



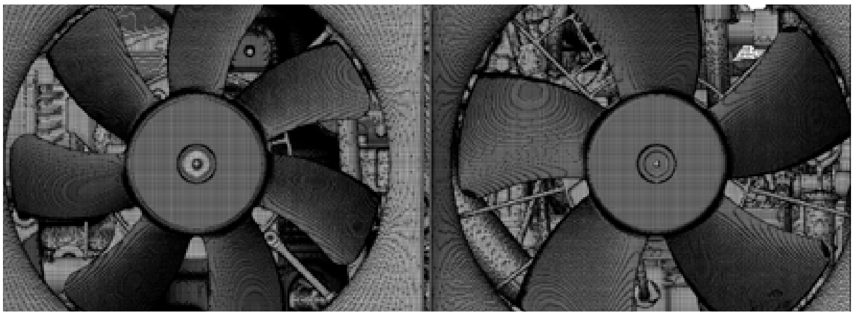


FIGURE 5 : Combined structured mesh impeller (AutoGrid5™) and unstructured mesh volute (HEXPRESS™) of a turbocharger

Fully automated multi-block structured mesh generation has been a goal for many years, because of superior mesh quality and reduced cell count. In that domain AutoGrid5™ is the undisputed worldwide reference in the Propulsion, Energy and Engine industries. Through its wizard-based application-dedicated workflow, it guides users with a few clicks to generate 100M+ high-quality cells in a few minutes for a broad range of turbomachinery applications.

[2] For more details, check out the article Honda Automobile explains how they save CPU time with HEXPRESS™/Hybrid on the NUMECA website.

OMNIS™ combines high-quality structured meshes in the bladed part with a full hexahedral unstructured mesh in the volute allowing the flow solver to converge in typically 30 minutes to 2 hours per million nodes and per core (Figure 5). Ford testifies that with a dozen cores, the aero-analysis of a new design at 3-4 operating conditions can be done in 2 hours! An impressive result compared to standard commercial solvers that need an entire day for this.[3]

A breadth of solver technology

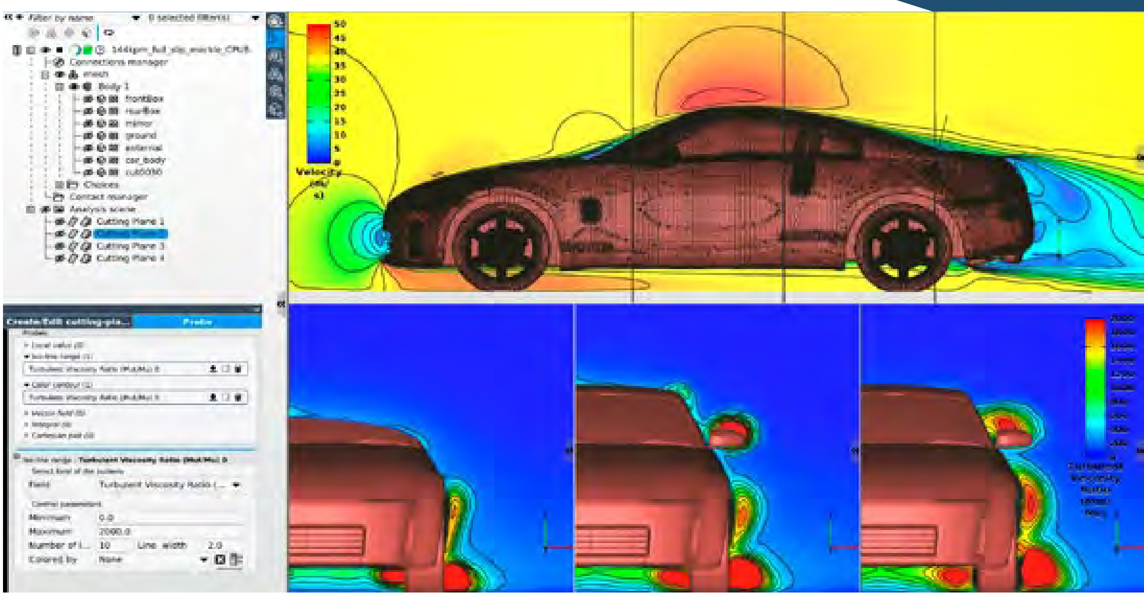
The use of multiple solvers to resolve complex engineering simulation tasks is current practice nowadays. Such an approach is extensively used for multiphysics simulations in which solvers designed for a particular physics are combined to analyze the influence and interaction of different physics phenomena on the global behavior of the geometry under analysis. From fluid-structure interaction simulations to aero-acoustics analysis and various complex flow physics such as multiphase and multispecies flows as well as the connections to an optimization framework. OMNIS™ offers a wide range of powerful solvers with FINE™/Turbo, FINE™/Open, OMNIS™/LB, FINE™/Marine, FINE™/Acoustics and FINE™/FSI-Oofelie, as well as a Python API to external and open-sources tools and solvers.

The main focus for car external aerodynamics for example is the accuracy in predicting drag and lift forces that measure the design performances. While these simulations are often heavy, the numerical algorithms must be efficient and validated, the simulation workflow must be robust and user-independent.

To achieve these objectives, OMNIS™/Open provides the automotive aerodynamics template with the best mesh, numerical and physical settings preset for maximum speed and robustness, and close-to-zero user intervention.

"Engineers at Honda perform aero-thermal computations of the underhood. Their pre-processing phase used to be time-consuming and cumbersome. When they switched to OMNIS™/Hexpress, they managed to divide their CPU time by 3 and their engineering time dropped to 30min/mesh instead of weeks."

FIGURE 6 : OMNIS™ Automotive aerodynamics template, with results shown on a Nissan 370Z



[3] For more details, check out the article "Multidisciplinary Optimization of a FORD Turbocharger Compressor Design" on the NUMECA website.

FIGURE 7 : Electric Vehicle gearbox simulation performed with OMNIS™/LB

Rotating machinery peripherals such as the turbocharger or water pump can benefit significantly from the structured approach using FINE™/Turbo, as also put forward in the previous section, boasting significant speed and precision benefits versus other technologies on the market with a speed-up of 10x-20x! When using the power of HPC combining CPU's with GPU's, as shown in Figure 7, the speed advantage is increased even more. On a centrifugal compressor a further speed up of 3 to 5 can be obtained.

On the other hand the analysis of gearbox lubrication is out of the reach of conventional flow solvers, because of moving parts and body-to-body contact. OMNIS™/LB is capable of handling complex geometries on a mesoscopic scale without the burden of having to set-up and fine-tune a mesh. The solution provides an LES-level representation allowing to capture complex phenomena such as splashing, dripping, sloshing, etc.

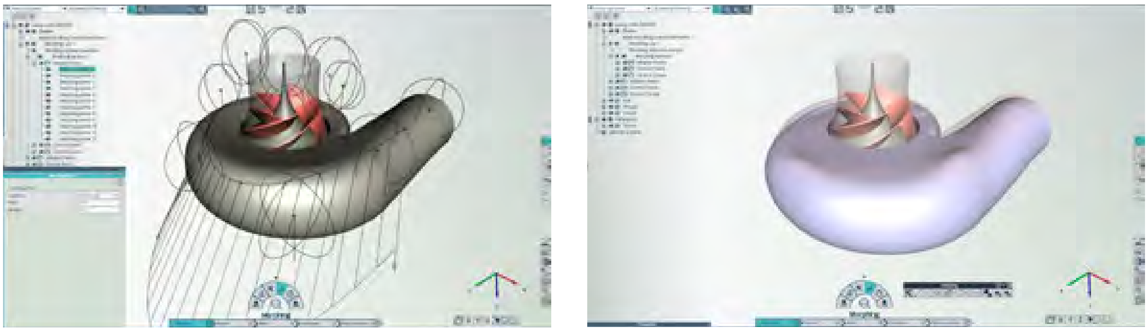
And thanks to an open architecture, OMNIS™ also allows for in-house solver integration based on the powerful Solver Plugin API. The C/C++, Python or Fortran APIs provide all the building blocks, which are then put together to couple an external solver, giving it access to all the capabilities described above.

Design Exploration

Running a single scenario is often just the first step in the simulation process. In most cases, engineers want to study design alternatives and compare their results. OMNIS™ offers multiple options to vary the designs. They can be parametrically imported from the CAD system thanks to the OMNIS™ bidirectional CAD gateway, or generated with the OMNIS™ Agile parametric blade modeler for rotating machinery, or even deformed from the original geometry with the OMNIS™ morphing tool as shown in Figure 8.

Either way, OMNIS™ defines multi-design setups to automatically apply the same mesh and simulation process for each unique design variation, providing an automatic way to create, analyze, and update your designs.

FIGURE 8 : OMNIS™/Morphing of a turbocharger volute. Inflation points or Morphing curves driven (left) and Morphed geometry (right)



"Ford declares that with a dozen cores the aero-analysis of a new design at 3-4 operating conditions can be done in 2 hours. An impressive result compared to standard commercial solvers that need an entire day for this."

Renault: Aerodynamic optimization of Exhaust Gas Recirculation (EGR) for car compressors

By **Stephane Guilain**, *Tech. Expert in PWT Aerodynamic and Engine Air Filling DEA-MA – Advanced Engineering, Renault* and **Donavan Dieu**, *Senior Consulting Engineer, Numflo.*

Pollution and fuel consumption are the most important features of thermal engines that must be drastically improved in the coming years. With soaring pollution in cities around the world, legislators are demanding car manufacturers to put systems on the market that are as clean and efficient as possible, whatever the driving style or conditions, from traffic jams to high-load mountain trips, in hot and cold conditions or even when setting new lap records on the Nurburgring and Spa Francorchamps in the Renault Megane RS Trophy R.

Furthermore to reduce CO_2 levels, thus diminishing the effects on global warming, fuel consumption must be significantly limited in those same real-world usage conditions. These environmental pressures are translated into legislation: the newest EU7 emission rules and new CAFE regulation will be implemented in Europe from 2023.

In order to achieve these goals, all consuming parts of cars are being meticulously analyzed to try and decrease losses to the maximum through design improvements, while taking into account inherent negative effects such as condensation issues for the compressor.
It is within this framework that Renault turned to Numflo - the consulting group of NUMECA International, which is renowned for its top level expertise in multiphysics design and analysis. The specific focus of the first study was to evaluate the impact of Low Temperature Exhaust Gas Recirculation (LT-EGR) on the efficiency of their turbo-compressors through CFD analysis. The power and reliability of NUMECA software tools were crucial for this project. At the end of the study, using the flow solutions obtained by Numflo, Renault performed condensation analyses with a dedicated software. Condensation can indeed occur when ambient temperatures are low and, in the long term, can damage the blades and create icing problems.

FIGURE 1 : Representation of the EGR geometry

Geometry

The study focuses on the impact of 5 geometrical parameters of the LT-EGR injection on the compressor wheel efficiency:

- Radius of the EGR injection
- Axial distance between the EGR injection and the compressor
- Three angles defining the orientation of the EGR around the inlet pipe.

The EGR geometry is generated using the IGG™ block structured mesher, with a script. This script automatically generates a new geometry for each new set of the 5 parameters. The inlet pipe and the compressor wheel are provided by Renault and remain unchanged during the simulation process.

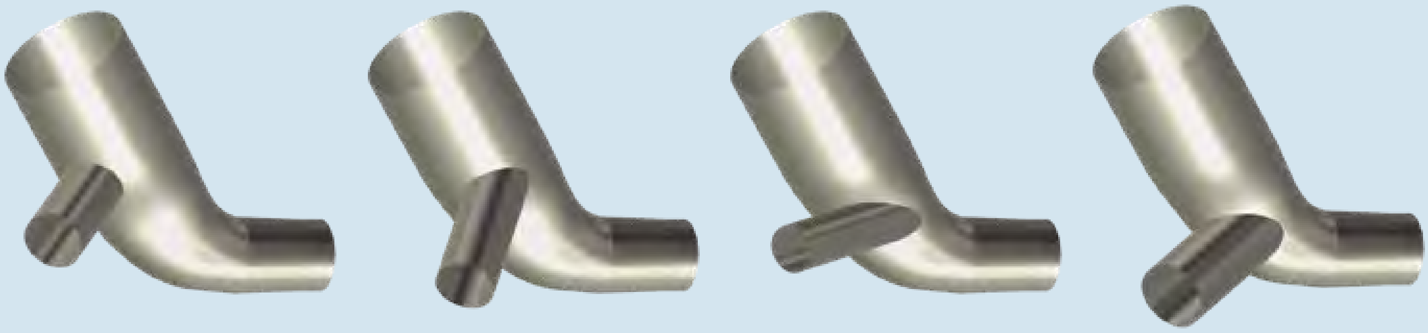
Meshing

On the numerical side, the meshing of the wheel consists of a high-fidelity block-structured grid, created with the automatic mesh generator AutoGrid5™. Only one periodic channel of the wheel is meshed. For the inlet pipe and the EGR, an automatic unstructured mesh is generated with HEXPRESS™/Hybrid software. The meshing process of the inlet pipe and EGR is automated using a dedicated script, ensuring good quality meshes regardless of the position of the EGR. Then, both meshes are reassembled, creating the final mesh used for the CFD simulations.

Non-Linear Harmonic approach

Flow distortion introduced by the inlet pipe and EGR inside the impeller is considered by means of the innovative Non-Linear Harmonic (NLH) method in FINE™/Open. This approach solves flow perturbations in the frequency domain, allowing for high accuracy of numerical results in comparison with state-of–the-art time-marching models, at a significantly reduced computation cost. This approach allows for transmitting the 360° flow distortion generated inside the inlet pipe into the wheel, having a direct impact on its aerodynamic performances.

FIGURE 2 : Examples of various EGR geometries generated by the IGG™ script



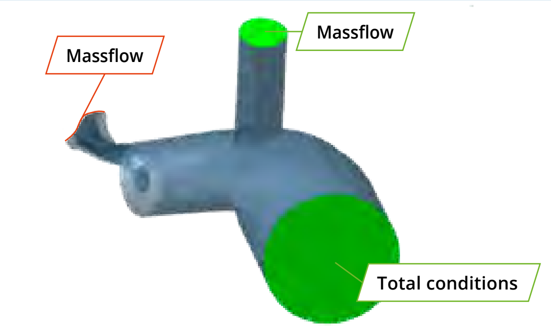
Simulations set-up

The fluid is air considered as a perfect gas. The following boundary conditions are used

- Main inlet : Total conditions and the flow direction
- Secondary inlet : Massflow
- Main outlet : Massflow

The first solution is started using constant values per domain.

FIGURE 3 : Representation of the boundary conditions



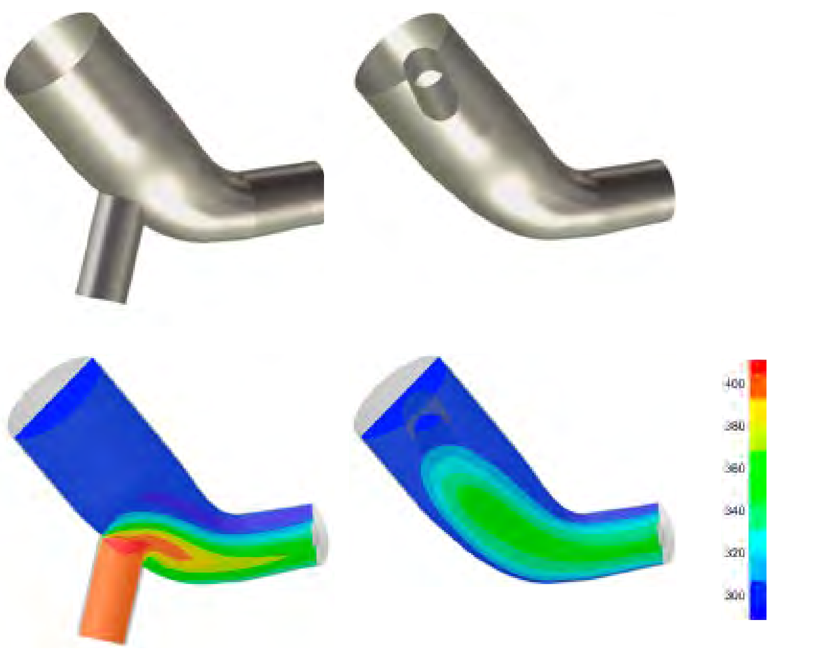
Results

The DoE (Design of Experiments) is generated using FINE™/Design3D with the Minamo module. A total of 26 elements is generated randomly using a « Latinized Centroidal Voronoi Tessellation » law. The analysis of the results of the database shows that higher wheel efficiency is obtained for an EGR located far from the wheel, and for an EGR with a small radius. Further flow analysis indicates that the best configurations demonstrate an important mixture between the main flow and the flow coming from the EGR, leading to less distortion at the inlet of the compressor wheel.

“Higher wheel efficiency is obtained for an EGR located far from the wheel, and for an EGR with a small radius.”

Based on the Minamo module, it is possible to run a deep analysis of the database to understand the influence and the relation between the free parameters and their impact on the performances. Below, an "ANOVA" graphic is providing the global sensibility of the free parameters for the considered objective.

FIGURE 4 : Comparison of the baseline and best efficiency design



A "Self organizing map" can also be used to project multidimensional data on a 2D plot. Based on an objective, it can easily allow the engineer to check if the available free parameters have (or not) the same impact on the objective. An example is provided below for the free parameters and a given objective. The highest value of the objective (green rectangle) corresponds to high values of "L" and low values of the "GAMMA".

FIGURE 5 : The "ANOVA" graph for the 6 free parameters

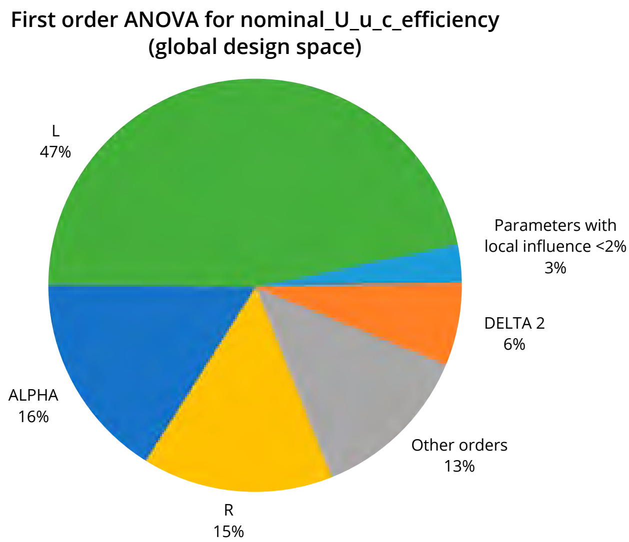
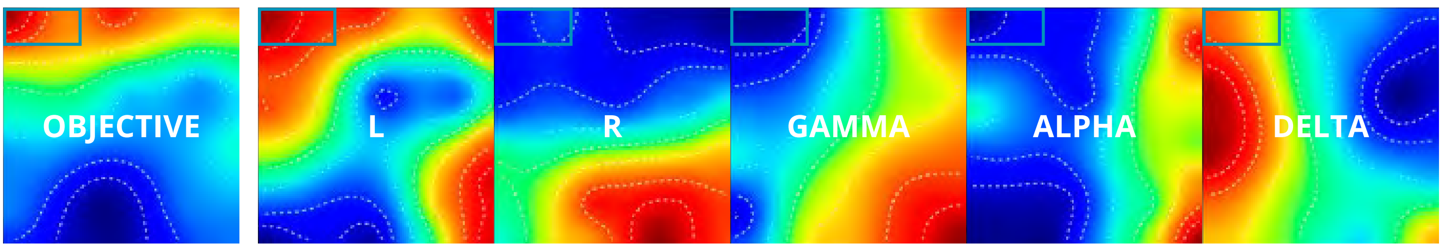


FIGURE 6 : Self organizing map showing the correlation between the free parameters and the final objective



Conclusions

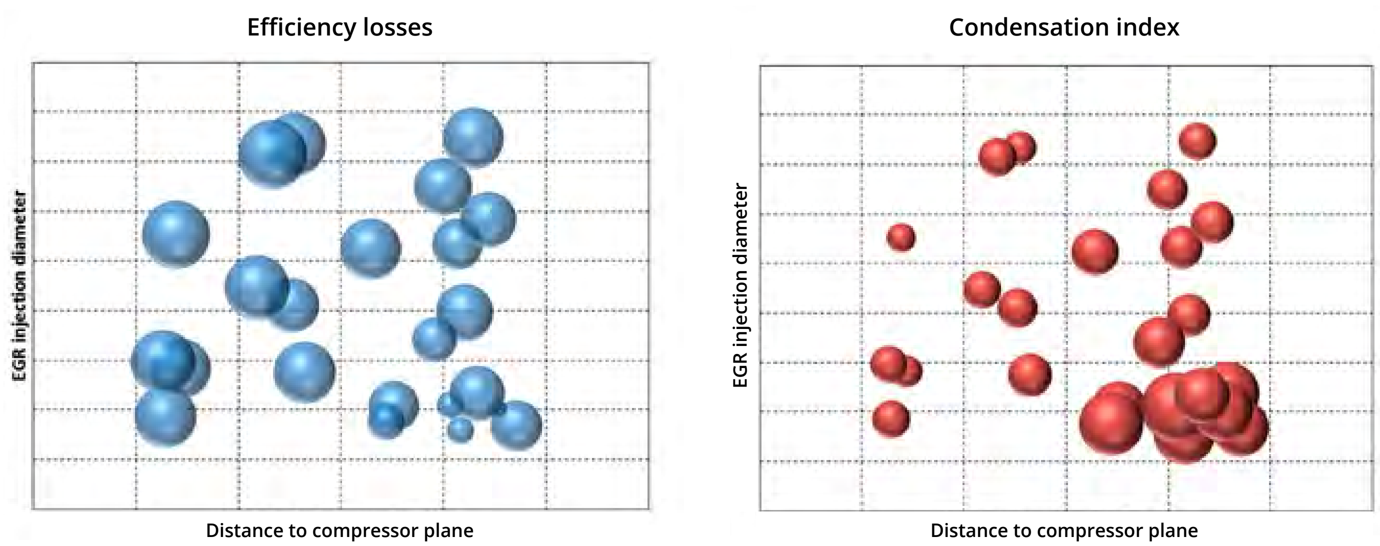
The condensation analyses performed by Renault, based on the Numflo CFD results, demonstrate that improvement of the wheel efficiency and improvement of the condensation issues lead to a selection of opposite values of the free parameters. In other words, if we want to increase the wheel efficiency, we inevitably also increase the condensation phenomenon.

The figures below show the impact of the two most important geometrical parameters of the EGR (distance to compressor plane and EGR injection diameter) on the efficiency losses and condensation index. The size of the bubbles is proportional to the losses and condensation level. From the left graph, it can be seen that losses are minimal for a high distance of the EGR to compressor plane and for a small radius of the EGR. However, this region corresponds to the worst condensation index. A compromise has to be performed between best efficiency and low condensation index as these 2 objectives are antagonistic.

The next step will be to perform an optimization taking into account the CFD (aerodynamic phenomenon) and the condensation aspects in a coupled manner.

> “ The condensation analyses performed by Renault, based on the Numflo CFD results, demonstrate that if we want to increase wheel efficiency, we inevitably also increase the condensation phenomenon.”

FIGURE 7 : Antagonist behavior between the two main objectives



ArianeGroup: Shaping the future of access to space

ArianeGroup is the world's leading designer and manufacturer of rocket launchers, a joint venture equally owned by Airbus and Safran. Their activities cover the entire life-cycle of a space launcher: design, development, production, operation and commercial service – the latter through their subsidiary Arianespace. They built and operate Ariane 5, the most reliable launcher on the commercial market today, and are developing the next-generation Ariane 6 launcher, for which they are the design authority.

By **Anne-Marie Schelkens**, *Marketing & Communications Coordinator.*

For more than 20 years now, ArianeGroup has trusted NUMECA's software tools for the design, analysis and optimization of the Liquid Hydrogen Turbopump of the Vulcain® rocket engine and its successors. Meanwhile NUMFLO, the NUMECA consultancy branch, has been providing consultancy service and expertise in fluid, thermal- & aero-mechanical simulation for the development of the Ariane launchers. Multi-physics simulation is crucial for the assessment of hydrodynamic performances at different stages of rocket engine development, before performing expensive test campaigns. Ariane Deneuve, Head of JOLE33, Turbomachinery Functional Design at ArianeGroup: "NUMECA's software solution is part of the standard at ArianeGroup in designing and validating our machines".

The Vulcain® engine

The Vulcain® is a cryogenic rocket engine, fueled by liquid hydrogen and liquid oxygen. These propellants are stored in two separate tanks at very low temperatures – hence "cryogenic". The liquid hydrogen is at -250°C and the liquid oxygen at -180°C. The propellant is fed at high pressure to the Vulcain® engine by two separate turbopumps, offering very impressive performance. The hydrogen turbopump is more powerful. It develops some 14MW, or nearly twice the power of a TGV high-speed train, but is concentrated in a space that would almost fit under the hood of your car. With a thermal output equal to that of a nuclear power plant (approximately 3,000 megawatts), the Vulcain®2 engine provides 130 tons of thrust for 7 minutes.

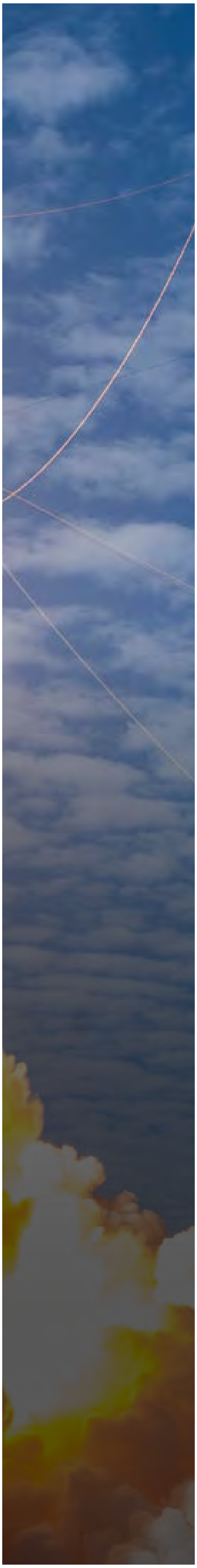
> " The FINE™ solvers are the fastest CFD solvers on the market."

Optimization through simulation

Optimizing the hydrodynamic performances of a space rocket engine like this entails detailed calculation of many physics, from thermodynamics to fluid-structure interactions, evaporation and cavitation. This is a challenging task given the complexity of geometrical details and of interaction effects between the components, which must be modeled with accuracy and within an acceptable computation time.

ArianeGroup uses a powerful combination of NUMECA's solvers FINE™/Turbo and FINE™/Open with OpenLabs™ with the AutoMesh™ suite for the mesh generation.

The FINE™ solvers are the fastest CFD solvers on the market. With the CPUBooster™ module, typical convergence time is below 1CPUh/Mpoints/core. This is a critical advantage for complex optimization cases like this, where many runs have to be performed.



FINE™/Open offers a large range of physical models that are critical for the simulation of TurboPumps:

- FINE™/Tabgen takes into consideration the real thermodynamic properties of liquid hydrogen,
- key advanced features such as the Non-Linear Harmonic (NLH) model take into account the unsteadiness of the flow due to rotor/stator interactions,
- the modeling of the cavitation is performed through a model jointly developed,
- a coupled NLH/Modal approach module is applied for Fluid-Structure Interactions.

For data preparation and mesh generation, ArianeGroup uses AutoMesh™ to provide high quality and fast block structured and unstructured meshes for the space pumps in the finest geometric details, including blades with fillets, small gaps between rotating and non-rotating walls, cavities, static fins and seals, etc....

> " Our main goal for the next version of the engine, the follow-up of the Vulcain®2.1, is to reduce the costs of the turbomachine by 50% while maintaining iso-performance."

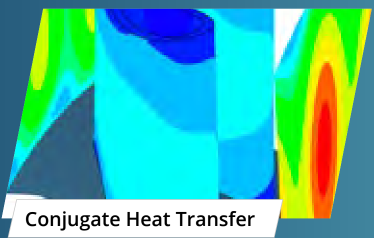
What's next?

ArianeGroup's main goal for the next version of the engine, the follow-up of the Vulcain®2.1, is to reduce the costs of the turbomachine by 50% while maintaining iso-performance. ArianeGroup plans to achieve this mainly through optimization of the hydraulic vein of the impeller with the FINE™/Turbo solver and by achieving a deeper understanding of the physics of the machine in order to limit the number of trials.

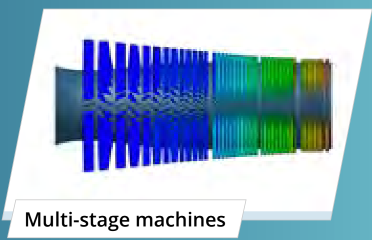
NUMECA and NUMFLO are ready for the next level, supporting ArianeGroup to get to space faster and with less energy consumption!

Tech Insight: FINE™/Turbo

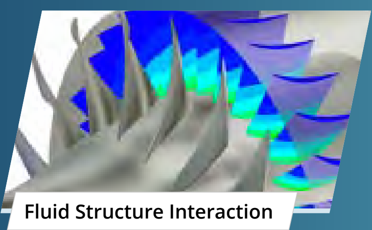
Solving some of your toughest turbomachine challenges like



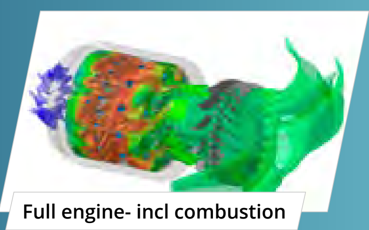
Conjugate Heat Transfer



Multi-stage machines



Fluid Structure Interaction



Full engine- incl combustion

More info and full capability chart here:
www.numeca.com/product/fineturbo

Ensuring your competitive advantage

Unprecedented speed

Cut valuable CPUh with proven 20x faster solution than any other on the market

Solutions first time right

Avoid expensive design failures early on thanks to 25 years of built-in turbomachinery expertise

Powerful design optimization including uncertainty quantification

Stretch the envelope of your design and know where the edges are

Unique non-linear harmonic (NLH) technology

Solve your unsteady flows in less than 1 hour

Want to be part of future issues?

Do you use NUMECA solutions to solve your toughest challenges? Join future editions of ReSolve and share your simulation story.

You will be eligible to win an all-inclusive access to one of the 2020 NUMECA Technology Seminars (Paris, Nuremberg, Los Angeles, Detroit, Tokyo, Beijing)!

Send us your ideas for an article at info@numeca.com

Go digital

Check out the online version of the magazine with exclusive content.

www.numeca.com/resolve