GT ENCLOSURE DISPERSION ANALYSIS WITH DIFFERENT CFD TOOLS

De Leo, E.[1], Lucherini, G.[2], Minotti, S.[3] and Quartieri, E.[4]
[1] Baker Hughes, Via F. Matteucci 2, Firenze, 50127, Italy, elena.deleo@bakerhughes.com
[2] Baker Hughes, Via F. Matteucci 2, Firenze, 50127, Italy, gabriele.lucherini@bakerhughes.com
[3] Baker Hughes, Via F. Matteucci 2, Firenze, 50127, Italy, stefano.minotti@bakerhughes.com
[4] Baker Hughes, Via F. Matteucci 2, Firenze, 50127, Italy, eugenio.quartieri@bakerhughes.com

ABSTRACT

A gas turbine is usually installed inside an acoustic enclosure, where the fuel gas supply system is also placed. It is common practice using CFD analysis to simulate the accidental fuel gas release inside the enclosure and the consequent dispersion. These numerical studies are used to properly design the gas detection system according to specific safety criteria which are well defined when the fuel gas is a conventional natural gas. Package design is done to prevent that any sparking items and hot surfaces higher than auto-ignition temperature could be a source of ignition in case of leak. Nevertheless, it is not possible to exclude that a leakage from a theoretical point of view could be ignited and for this reason a robust design requires that the enclosure structure is able to withstand the overpressure generated by a gas cloud ignition. Moving to hydrogen as fuel gas makes this design constraint much more relevant for its known characteristics of reactiveness, large range of flammability, maximum burning velocity etc. In such context, gas leak and dispersion analysis become even more crucial, because a correct prediction of these scenarios can guide the design to a safe configuration. The present work shows a comparison of the dispersion of different leakages inside a gas turbine enclosure carried out with two different CFD tools, Ansys CFX and FLACS. This verification is considered essential since dispersion analysis results are used as initial conditions for gas cloud ignition simulations strictly necessary to predict the consequence in term of overpressure without doing experimental tests.

1.0 INTRODUCTION

Gas turbines are usually placed inside an acoustic enclosure to reduce the noise emissions. The installation of the engine in a confined space creates a series of issues to be faced during the design of the gas turbine package. Beyond the turbine itself, the enclosure contains the inlet and exhaust plenum, the fuel gas system, starting motor, fire-fighting system, other auxiliary systems and instrumentation in general. The fuel gas system is necessary to feed the turbine supplying fuel to combustors. Along this line, there are hundreds of connections both flanged and treated, which could be potential source of leak. The simultaneous presence of potential leak sources and the hot surfaces of turbine casing creates under specific circumstances, the theorical condition to have an ignition in case of an accidental leakage. The main mitigation is represented by the ventilation flow which is for this reason an essential auxiliary system [1]. Without a ventilation flow the gas turbine cannot be started, and in case it is lost during GT normal running, an emergency shut down is executed and the package is put in safe mode, emptying the fuel gas line. Even if the entire design is driven by safety concepts, it is not possible to exclude a fuel gas leakage a priori from one of the numerous connections inside the enclosure. The real purpose of a correct design is not to avoid such leaks, but to limit them as much as possible and to dilute under certain limits that are given by ISO 21789 [2]. In other words, it could be accepted that a leakage occurs and it is even followed by an ignition, but the consequences in terms of overpressure shall be contained and confined inside the GT enclosure, where it is not commonly allowed to the personnel to enter. To this aim it is fundamental the proper design of the enclosure structure which shall be able to withstand up to a certain level of impulsive overpressure in order to guarantee the safety of the surrounding environment. Several studies have been carried out in the past [3,4] to test the consequence of an ignition of natural gas, mainly composed by methane, inside a gas turbine enclosure, which is a closed and very congested volume. In this way was possible to find a relation between the accumulation of fuel gas inside the enclosure and the possible overpressure generated by an ignition.

The current interest of gas turbine manufacturers and packagers to transit from conventional natural gas to blend or full hydrogen fuels is putting under discussion the validity of the criterium given by ISO 21789. The hydrogen has different characteristics, that requires to critically evaluate the entire design approach followed up today [5]. As already known, the hydrogen is more easily ignitable than methane and also the consequences could be more severe [6]. Experimental tests would be very useful to verify dispersion, likelihood and consequence of an ignition, but it is not always easy to manage tests on real GT package configurations using hydrogen for the obvious safety concerns [7]. In this context, making recourse to numerical approach gives undeniable advantages. Baker Hughes common design requires to simulate the consequence of a fuel gas dispersion inside the GT enclosure during a possible operating scenario and the largely used tool is Ansys CFX [8]. In this way it is verified the capability of the ventilation to dilute the leakage. Starting from the flow field distribution, less ventilated zones are identified and there a leak is located or directed [9,10], in this way the dispersion study takes into account the worst case in terms of gas accumulation. The presence of the leakage entails a complication on the numerical model since the resulting leak is an under-expanded jet [11], which is due to the pressure ratio existing between the fuel gas inside the line, in the range 15÷45 barg and the pressure inside the enclosure which is close to atmospheric one. At the end of the simulation, during the post-processing, the iso-surface of flammable mixture at 100% of lower flammable limit (LFL) is identified. Then it is calculated the total amount of gas included inside this iso-surface and converted into a stochiometric cloud. When natural gas is used, the last step is to compare the volume of such cloud with the ISO 21789 limit and the design can be considered concluded even in terms of safety. Moving to hydrogen, this is no more sufficient, since currently does not exist a clear limit to be used as maximum acceptable cloud volume [12]. For this reason, an additional simulation to estimate the consequences of an accidental ignition should be introduced to complete the GT package design. In this sense, the dispersion study that for methane is the final step of the design procedure, with hydrogen this is the input for the ignition study. Starting from a certain gas distribution inside the enclosure, it is possible to simulate what effects are generated imposing an ignition in an arbitrary position. The tool used for this kind of analysis is FLACS (Flame Acceleration Simulator) by Gexcon [13]. The outcome is the prediction of the overpressure [14, 15] in the entire domain and with a dedicated variable (DPDX) it is possible to estimate the likelihood to have a deflagration or a detonation [16].

This work shows a comparison between the results of dispersion study carried out with Ansys CFX and FLACS. These computational tools require different approaches that are described trying to highlight for both pro and cons. CFX results are used as reference since they have been largely validate in the past. The aim of this study is to compare the results creating similar models without distorting the own characteristics of each tool. The gas turbine object of the study is NovaLT16 package that is suitable to run burning 100% hydrogen.

2.0 GAS TURBINE PACKAGE

The current work shows the CFD study of a NovalLT16 package designed and manufactured by Baker Hughes [17]. This turbine is used for industrial and Oil&Gas market.

The engine and its package are usually installed inside an acoustic enclosure, which is ventilated with an air flow entering upstream the inlet plenum and goes out through the outlet duct located around the GT exhaust duct (see Figure 1). The CFD model is created starting from the complete three-dimensional model generated for the mechanical and electrical design of the package and it is cleaned-up from details that do not affect the flow distribution. The criteria used for the model generation has been consolidated and validated during the last years with internal tests. Beyond the GT enclosure, a portion of the ventilation ducts are modelled, since their routing could influence the flow path.

Table 1. NovaLT16 data.

Power (MW/kHP)	17.5/23.5
Efficiency (%)	37.5
NOx emission(ppm)	15
Exhaust temperature (°C/°F)	495/923
Speed (rpm)	3900-7900

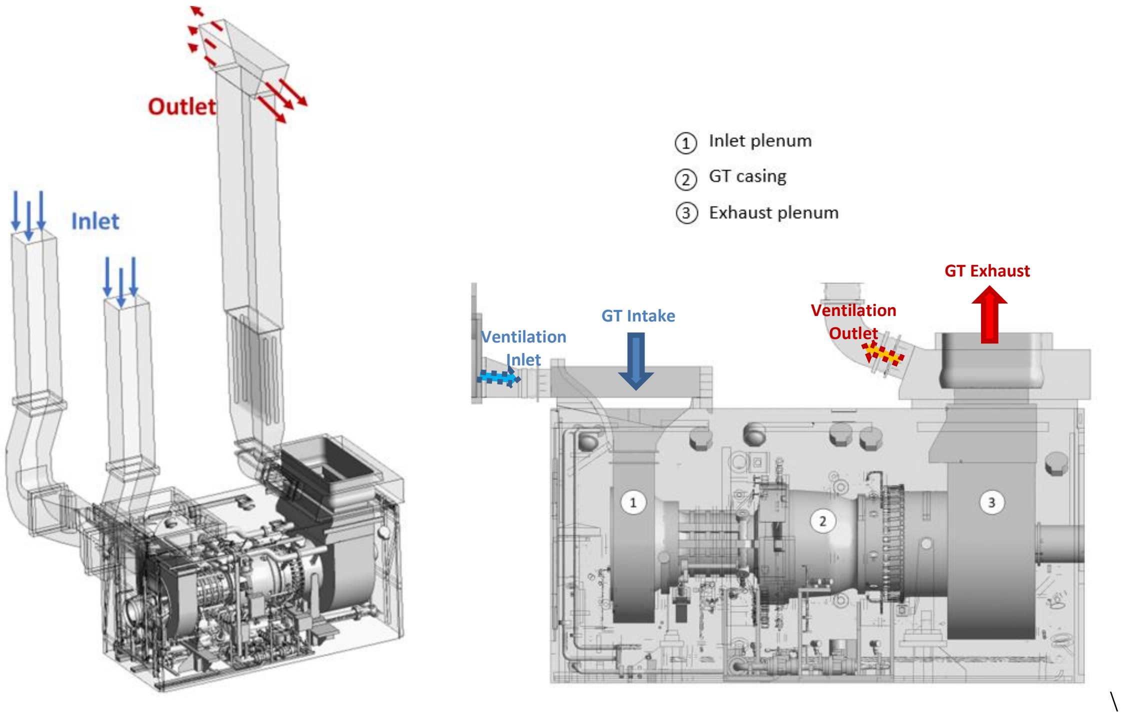

Figure 1. CFX Model

3.0 ANSYS CFX® STUDY

The computational analyses are carried out using ANSYS CFX v21, a Finite Volume Method based on Elements (EbFVM), with a Cell Vertex Formulation. The turbulence model is the Shear Stress Transport (SST) [18], which is suitable for this kind of applications [8]. The numerical grid shown in Figure 2 is characterized by both hexahedral and tetrahedral elements. The boundary layers are refined to keep y+ values around 1 on the GT casing and exhaust plenum, which are the hottest surfaces, while globally it is maintained below 3. Grid statistics are summarized in Table 2. The discretization level of this mesh is the balance between the required refinement level and the computational time constraint [8]. Additionally, the region around the leak source is refined to smaller scales to properly capture the jet profile. The leakage reaches the sonic condition in the passage area and to simplify the numerical solution, it is modelled with a pseudo-source approach derived from the experimental studies carried out by Ewan and Moodie. [19]. This allows to simulate only the subsonic jet portion, starting from Mach disk section and neglecting the flow upstream.

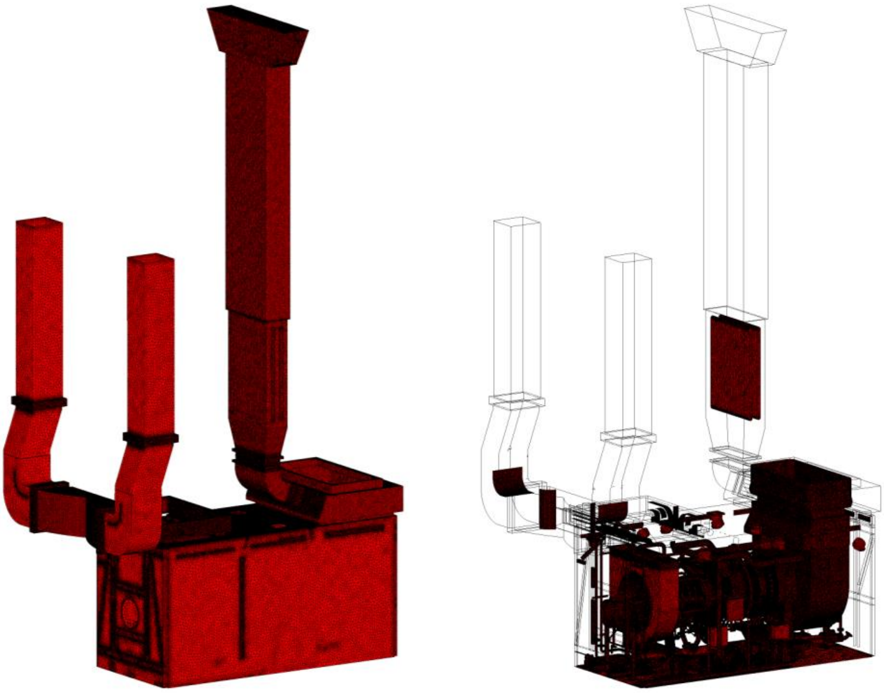

Figure 2. Volume and Surface mesh in the enclosure

Table 2. CFX Mesh values.

Number of Nodes	14.521.293
Number of Elements	65.824.535
Tetrahedra	61.293.618
Pyramids	26
Wedges	1.721.240

The thermal boundary conditions are accurately defined to simulate the engine running at full power. Temperature profiles are imposed on exhaust plenum, hot pipes and engine casing. Flow boundary conditions require to impose mass flow rate at the ventilation inlet, as well as ambient temperature and turbulence intensity, while static pressure is fixed at the outlet section. Baker Hughes usually adopts such detailed boundary conditions since these CFD models are generally used to verify not only the dispersion capability of the ventilation flow but also the proper installation of all items inside the enclosure in terms of temperature.

Regarding the fuel gas leak, mass flow rate, temperature and turbulence intensity are imposed on a specific flanged connection along fuel gas line, where the leakages could occur. The selection of leak sources location and direction (see Figure 3) is based on a screening analysis that is carried out to identify the portion of the GT enclosure more prone to accumulate gas in case of an unexpected leakage [9,10]. Table 3 lists the leak cases studied in this work.

Table 3. Leakage characteristics

	Position	Direction	Mass Flow Rate (g/s)
Leak 1	From the manifold’s top	Toward the inlet plenum	1.9
Leak 2	From the manifold’s top	Toward the top of the enclosure	1.9
Leak 3	From the skid gas	Toward the GT	1.9

Simulations are parallelized into 200 processors of a HPC cluster and the time required to get convergence, depends on case by case, the order of magnitude is around 50 hours.

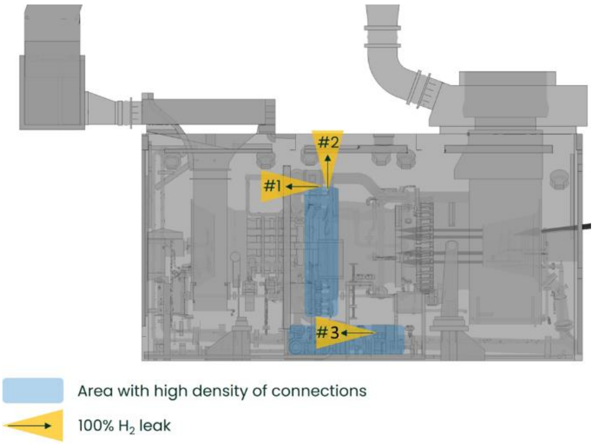

Figure 3. Position and direction of the simulated leakages

4.0 FLACS STUDY

FLACS model (Figure 4) is created starting from the 3D CAD model used in CFX. Differently from Ansys’ software, FLACS is capable only to read primitive solids like boxes, cylinder, ellipsoid, convex polyhedron, general truncated cone, torus, and rectangular torus. The model was modified creating each element as operation (addition and left difference) of primitive solids. All modifications applied on the model have been performed remaining as compliant as possible to the CFX model, even if the conversion implies a lower level of detail.

The mesh implemented in FLACS is a cartesian grid, a structured not-body fitted type of grid. The grid has been structured all around the 3D model, including not only the fluid domain but also the solid elements like the ventilation ducts. A high grid refinement has been done in the more interesting region, considering as core domain the zone around the gas turbine and around the gas supply line. A refined grid allows to mitigate the lack of accuracy due to the not-fitted body grid, in a scenario with high complex model. The mesh has a total amount of 4.6M of elements and the thinner elements can represent adequately the point of release.

As in CFX the jet is modelled with a pseudo-source model for the same reason. The analysed leaks are the same simulated with CFX and shown in Figure 3

The boundary conditions imposed are ventilation mass flow rate, ambient temperature and turbulence intensity at domain inlet, while ambient pressure at outlet. Thermal boundary conditions are not imposed since thermal exchange is neglected. This choice derives from two considerations. The first one is that the effort required to introduce sufficiently accurate boundary conditions would not be balanced by a significant improvement in results. While CFX as mentioned in paragraph 3.0 is historically used for the prediction of temperature field inside the enclosure, boundary conditions choice confirms that FLACS is aimed to mainly evaluate the gas dispersion. In the comparison of the results from these two different models, it is important to keep in mind that their own vocation is different. The second consideration regards the temperature distribution, which does not significantly affect the leak dilution since the convective contribute is much more dominant than the buoyancy effect. [20]

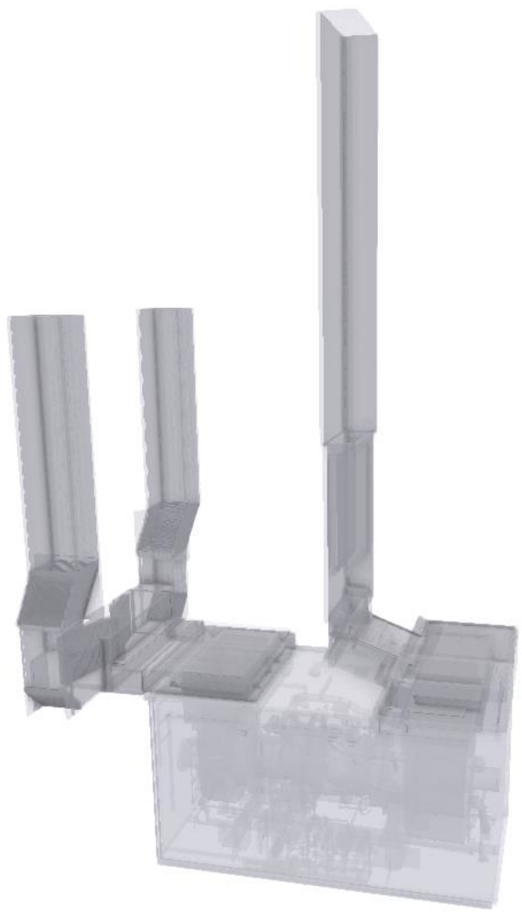

Figure 4. FLACS model

5.0 RESULTS COMPARISON

The results of both computational tools are firstly compared in terms of flow distribution inside the gas turbine enclosure. The flow entering inside the enclosure is uniformly distributed among two symmetrical ventilation ducts. Before entering inside the enclosure, the flow passes through a toroid around the gas turbine inlet duct that is used to properly split the flow upstream and downstream the inlet plenum. Several deflectors are placed inside the toroidal duct to guide the flow for avoiding stagnation regions. Differently from CFX model, in which the deflectors reproduce the real geometry, in FLACS planes with different porosities are modelled (Figure 5).

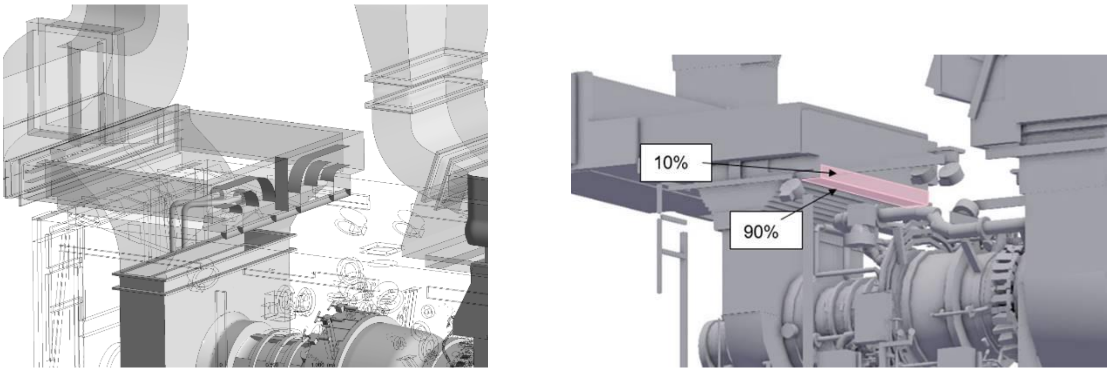

Figure 5. Deflector modelling in CFX (left) and FLACS (right)

Figures 6 and 7 show the streamline of the ventilation flow obtained with the two solvers. The air is divided between the upper and lower enclosure region to adequately ventilate the GT casing, the gas line and all the instruments installed. The flow exits from the enclosure turning around the GT exhaust duct and flowing inside the ventilation outlet duct. Both results show the same qualitative flow distribution inside the enclosure. Figures 8 shows the velocity magnitude contour at the gas turbine midplane for a more detailed comparison. The general overview of the flow field allows to highlight the commonalities between the outcome of two software. The velocity distribution between the two solutions is generally in good agreement with minor discrepancies. Dashed red circles in Figures 7 and 8 show the portion of the enclosure where the flow velocity is higher than the other case. CFX predicts an higher flow below the engine, while FLACS above the engine close to the roof and close to the ventilation outlet around the exhaust plenum, this difference could be due to the different modelling of flow deflector as shown in Figure 5. Moreover, looking the entire contour plots, the small-scale solution is more detailed for Ansys CFX, as it could be expected, since the geometric level of details is higher.

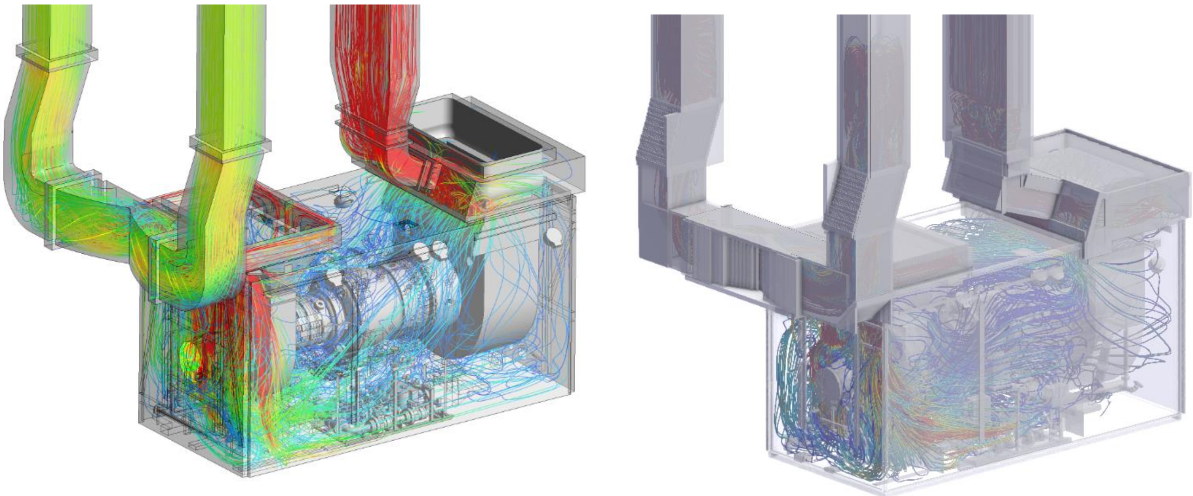

Figure 6. Streamline comparison between CFX (left) and FLACS (right)

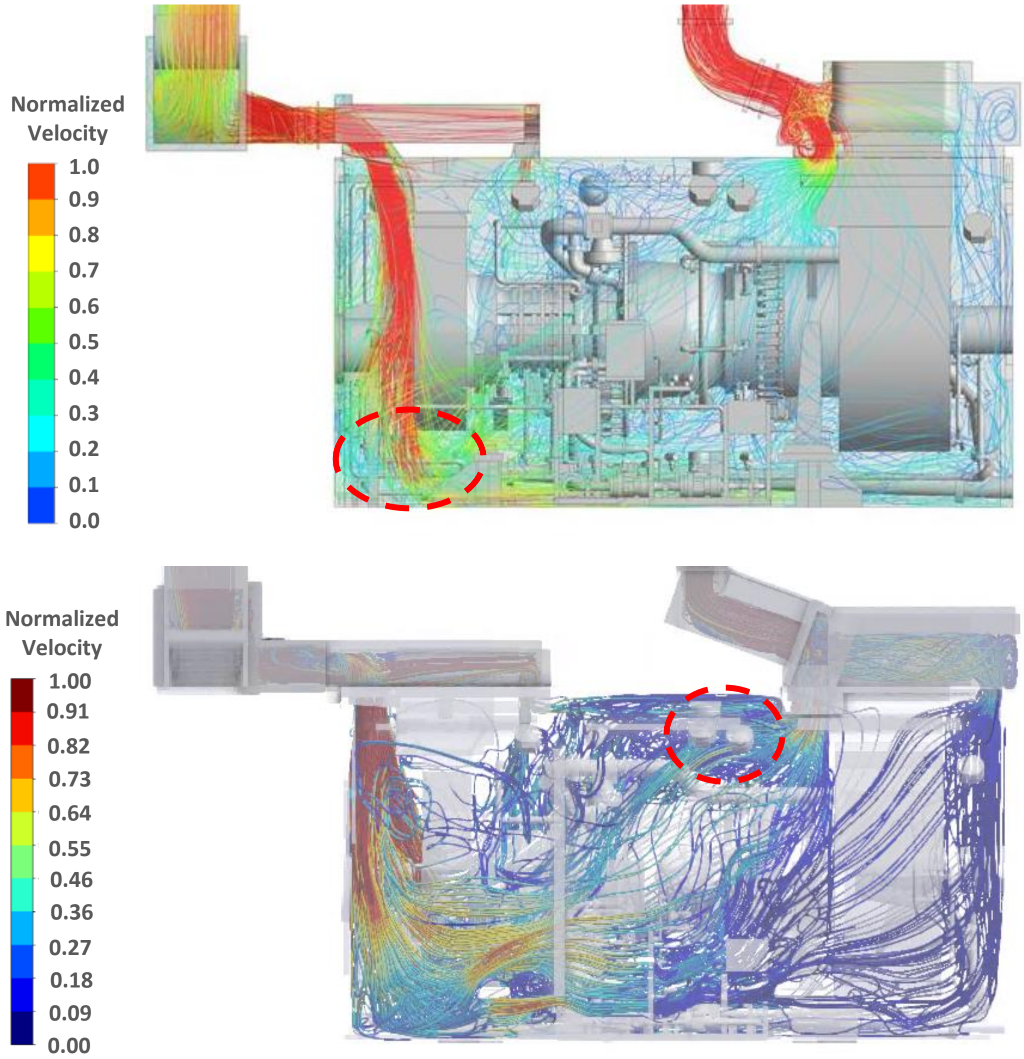

Figure 7. Side view of Streamline comparison between CFX (left) and FLACS (right)

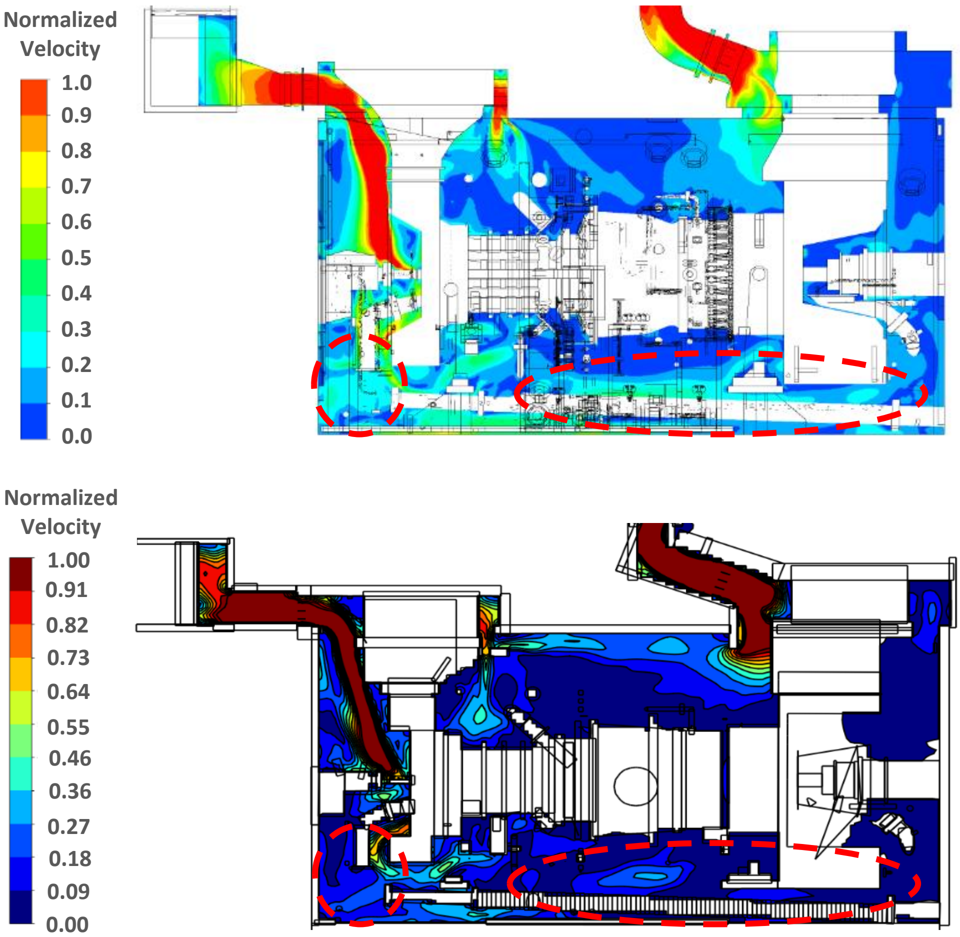

Figura 8. Flow field comparison between CFX (top) and FLACS (bottom)

The thermal field is predicted by Ansys CFX with a good accuracy that is proven by numerous experimental tests internally carried out by Baker Hughes. As shown in Figure 9 the temperature distribution is strongly nonuniform due to the thermal boundary conditions imposed on hot surfaces. For the approach described in paragraph 4.0, temperature field cannot be compared since it is not calculated in FLACS model for the choice taken by authors.

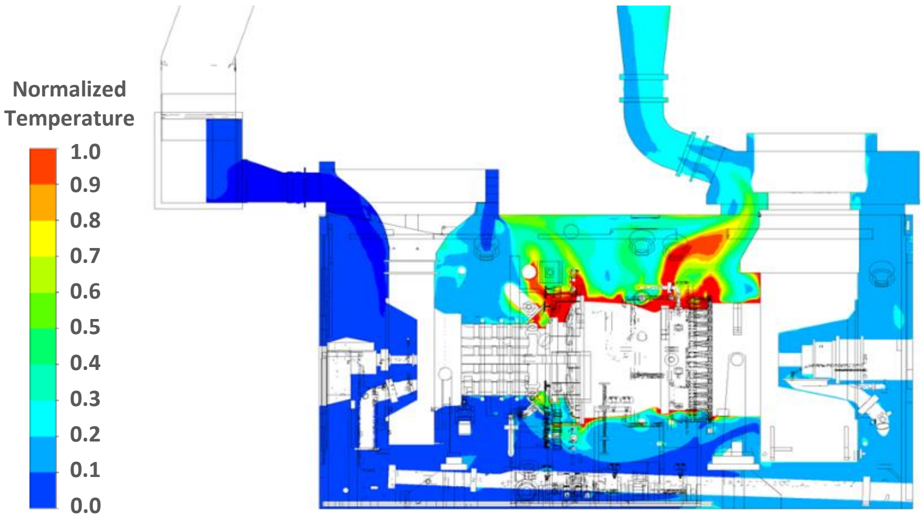

Figure 9. Temperature field in CFX normalized with ambient temperature.

Figures 10, 11 and 12 show the comparison between the flammable cloud founded with the two different tools. From a qualitative comparison point of view the clouds morphology is very similar between the solution of the two software, even if the Leak 2 is very less diluted in CFX solution. Table 4 shows the value of flammable mass and volume obtained allowing a quantitative comparison. As can be seen the resolution of Leak 1 and 3 has a good level of agreement between the two tools, although in FLACS the Leak 1 that spreads on the upper part is more diluted and the Leak 3 that spreads on the lower part of the enclosure is less diluted than in CFX. Leak 2 shows the higher discrepancy between the solutions since the FLACS cloud is far more dilute than the other one. These highlighted differences are due to the inhomogeneity inside the velocity distribution and not to discrepancies in the boundary conditions.

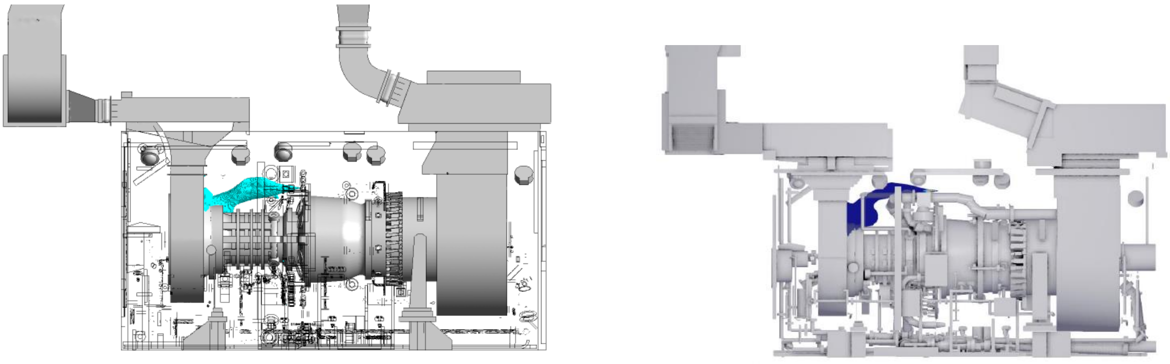

Figure 10. Flammable cloud of Leak 1 in CFX (left) and FLACS (right) © 2023 Baker Hughes Company - All rights reserved

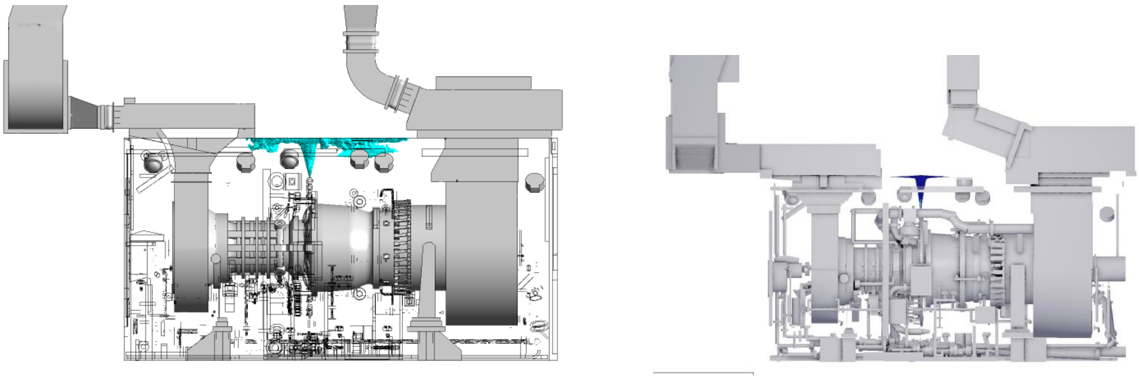

Figure 11. Flammable cloud of Leak 2 in CFX (left) and FLACS (right) © 2023 Baker Hughes Company - All rights reserved

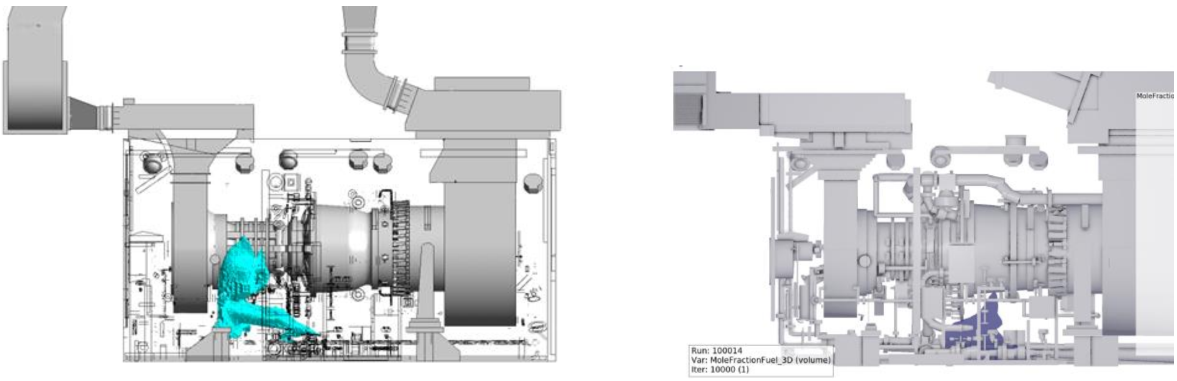

Figure 12. Flammable cloud of Leak 3 in CFX (left) and FLACS (right) © 2023 Baker Hughes Company - All rights reserved

Table 4. Results comparison

	CFX		FLACS	
	Flammable mass (kg)	**Flammable Volume (m^3)**	**Flammable mass (kg)**	**Flammable Volume (m^3)**
Leak 1	9.03 E-04	0.22	7.37 E-04	0.17
Leak 2	9.21 E-04	0.23	2.12 E-04	0.05
Leak 3	1.47 E-03	0.34	1.85 E-03	0.3

6.0 CONCLUSIONS

This works shows a comparison between two CFD solvers used to study the dispersion of a sonic hydrogen leak inside a gas turbine enclosure. The design of a turbine package is carried out to prevent such scenario, even if it could occur in rare cases. Experimental tests on real scale configurations are not easily executable for safety reasons and sufficiently accurate simulations are crucial.

The flammable clouds are in good agreements between the two solutions although the dilution grade is slightly different in some cases (Leak 1 and Leak 3) and more in the other one (Leak 2). The discrepancy between the solutions is mainly due to the differences of the flow distribution. In FLACS solutions the air-fuel flammable clouds in the upper part of the enclosure are more dilute than the CFX ones, due to the higher amount of flow distributed in this region (Figures 7 and 8). On the contrary, the cloud spread in the lower part is less dilute due to the lower velocity of the ventilation flow (Figure 7).

In this comparison, temperature distribution does not affect the leak dilution since the convective contribute is much more dominant than the buoyancy effect due to the temperature field.

Through this study comes to light the valuable FLACS capability to obtain sufficiently accurate overall flow field and cloud dispersion with saving in computational time. Baker Hughes will continue to implement FLACS simulations thanks to the high flexibility and ease in developing explosion simulations. On the other hand, also CFX will be used since it allows us to analyze in detail and proven level of accuracy the thermal flow field, required for the ventilation system design. The combination of both tools will surely provide an additional value in design process.

REFERENCES

1. Ponnuraj, B., Sultanian, B., K., Novori, A., and Pecchi, P., 3D CFD Analysis of an Industrial Gas Turbine Compartment Ventilation System, ASME paper IMECE2003-41672, 2003 International Mechanical Engineering Congress and Exposition, Washington, DC, USA, November 15–21, 2003.
2. International Standard ISO 21789:2022 Gas turbine applications – Safety, 2022
3. Santon, R.C., Ivings, M.J. and Pritchard, D.K., A New Gas Turbine Enclosure Ventilation Design Criterion, ASME paper GT2005-68725, Turbo Expo 2005, Reno-Tahoe, Nevada, USA, 2005.
4. Santon, R.C., Kidger, J. W., and Lea, C. J., Safety Developments in Gas Turbine Power Applications, ASME paper GT2002-30469, Turbo Expo 2002, Amsterdam, The Netherlands, 2002
5. Baldini, M., Cinelli, R., Minotti, S., Pampaloni, G., Quartieri, E., Rossin, S., Moving Gas Turbine package from Conventional Gas to Hydrogen Blend, International Conference on Hydrogen Safety (ICHS), Edinburgh, Scotland (Virtual), 2021.
6. Michels, H., Lindstedt, P., Moodie, K., Santon, R., Ewan, B.C.R., Literature review of CCGT/CCGE/CHP systems operating on high hydrogen content gases, Project no. PE02162.
7. Ewan, B.C.R., Moodie, K., Rattigan, W., ETI High Hydrogen Phase 2 Test Programme Report, Report Number EA/19/12.
8. Lucherini, G., Minotti, S. and Michelassi, V., The Impact of Model Assumptions on the CFD Assisted Design of Gas Turbine Packages, ASME paper GT2019-90781, Turbo Expo 2019, Phoenix, Arizona, USA, 2019.
9. Corsini, A., Delibra, G., Giovannelli, M., Lucherini, G., Minotti, S., Rossin, S. and Tieghi, L., Identification of Poorly Ventilated Zones in Gas-Turbine Enclosures with Machine Learning, GT2019-91198, Proceedings of ASME Turbo Expo 2019, Phoenix, Arizona, USA, 2019.
10. Corsini, A., Delibra, G., Giovannelli, M., Lucherini, G., Minotti, S., Rossin, S. and Tieghi, L., Prediction of Ventilation Effectiveness for LM9000 Package with Machine Learning, GT2020-14916, Proceedings of ASME Turbo Expo 2020, June 22-26, 2020, Virtual Conference.
11. Franquet, E., Perrier, V., Gibout, S., Bruel, P., Free Underexpanded Jets in a Quiescent Medium: a Review, *Prog Aerosp Sci*, 77, pp. 25-53, 2015.
12. Vye, T. S., and Miles, D., Gas Turbine Enclosures: Determining Ventilation Safety Criteria using Hydrogen Explosion Modelling, International Conference on Hydrogen Safety (ICHS), Edinburgh, Scotland (Virtual), 2021.
13. FLACS-CFD v22.1r2 User's Manual, Copyright ©Gexcon AS, 2022.
14. Lucas, M., Atanga, G., Hisken, H., Mauri, L. and Skjold, T., Simulating Vented Hydrogen Deflagrations: Improved Modelling in the CFD Tool Flacs-Hydrogen, *International Journal of Hydrogen Energy*, Vol. 46, Issue 23, 31 March 2021, pp. 12464-12473.
15. Lucas, M., Hisken, H., Skjold, T. and Arntzen, B.J., Improved modelling of hydrogen explosions – representing effects of varying concentration and reduced oxygen atmospheres, Proceedings of the 14th international Symposium on Hazards, Prevention and Mitigation of Industrial Explosions, Braunschweig, Germany, 2022.
16. Middha, P., Howell, S. and Feven, S., New Paradigms in Hydrogen Explosion Modelling using an Industrial CFD Code, International Conference on Hydrogen Safety (ICHS), Adelaide, Australia, 2019.
17. https://www.bakerhughes.com/gas-turbines/novalt-technology/novalt16
18. Menter F. R., Kuntz M., Langtry R., Ten Years of Industrial Experience with the SST Turbulence Model, Turbulence, Heat and Mass Transfer 4, Antalya, Turkey, 2003.
19. B. C. R. Ewan and K. Moodie, Structure and Velocity Measurements in Undenexpanded Jets, *Combustion Science and Technology*, pp. 275-288, 1986.
20. Lucherini, G., Minotti, S., Ragni, G. and Bologna, F., Experimental and numerical investigation on GT package scale model, ASME paper GT2018-75694, Turbo Expo 2018, Oslo, Norway, 2018.