Virtual Certification of Gas Turbine Engines - Visualizing the DLR Rig250 Compressor

Arun Prabhakar[a,*], Christopher Goddard[b], Dario Amirante[c], Istvan Reguly[e], Axel Gerstenberger[d], Jan-Friedrich Suhrmann[d], Stephen A. Jarvis[f], Leigh Lapworth[b], Gihan R. Mudalige[a,*]

[a]*University of Warwick, UK*
[b]*Rolls-Royce plc, UK*
[c]*University of Surrey, UK*
[d]*Rolls-Royce Deutschland Ltd & Co KG, Germany*
[e]*Pazmany Peter Catholic University, Hungary*
[f]*University of Birmingham, UK*

Abstract

High Performance Computing (HPC) critically underpins the design of aero-engines. With global emissions targets, engine designs require a fundamental change including designs utilizing sustainable aviation fuels and electric/hybrid flight. Virtual certification of designs with HPC is recognized as a key technology to meet these challenges, but require analysis on models with higher fidelity, using ultra-large scale executions. In this explanatory SC-SciVis showcase, we present results from time-accurate simulations of a 4.6B-element full 360-degree model of a production-representative gas turbine engine compressor, the Rig250 at DLR. This represents a grand challenge problem, at the fidelity for virtual certification standards. The results are achieved through Rolls-Royce's Hydra CFD suite on ARCHER2. The compressor is visualized under off-design conditions, demonstrating flow contours of velocity, Mach number and iso-surfaces of vorticity. The level of detail and the HPC simulations leading to the visualizations demonstrate a step-change towards achieving virtual certification objectives under production settings.

Keywords:
Catalyst Visualization, Virtual Certification, Gas Turbine Engines

1. Introduction

Companies such as Rolls-Royce, crucially depend on HPC based numerical simulations. These complex multi-physics and engineering applications, predominantly based on computational fluid dynamics (CFD), even in their simplest form, provide insights into aspects of aircraft engines which could not otherwise be achieved in the absence of physical testing. Recent global Net-zero emissions targets [1, 2] have placed unprecedented challenges on aero engine designs demanding a fundamental change to engine architectures including designs that utilize sustainable aviation fuels and electric and hybrid flight. Virtual certification of designs with HPC is recognized as a key technology to meet these goals, but require analysis on models with higher fidelity, using ultra-large scale executions, while at the same time provide exceptional levels of trust in the accuracy of the simulations.

Full engine simulation, a critical segment of the virtual certification process consists of a number of components, of which computing the flow through the engine forms a key part of the analysis. In a gas turbine engine, the main thrust that drives the aircraft is provided by the fan, in the front of the engine (see Figure 1) which is connected to a shaft powered by a turbine at the rear of the engine. The turbine is in turn driven by the exhaust from combustion of compressed air which is delivered to the turbine through a compressor and subsequently sprayed with fuel and ignited in a combustion chamber. In this energy cycle, the net power available to drive the fan and accelerate the core stream is given by the difference between turbine work and compressor work. Compressor and turbine isentropic

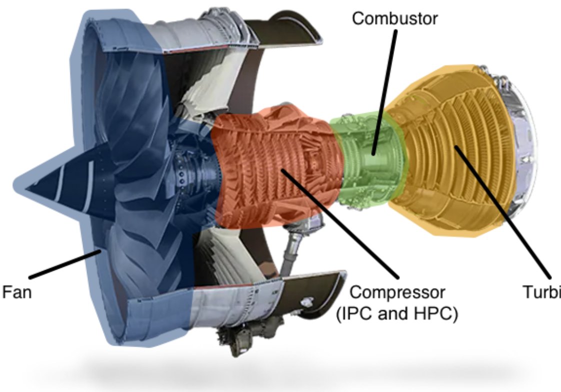
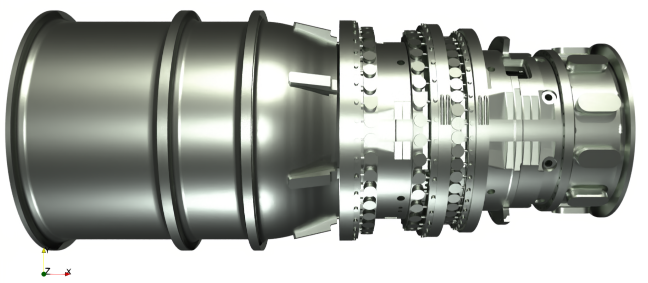
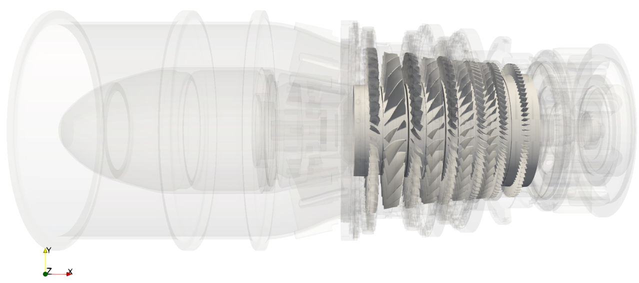


Figure 1: RR Trent XWB Engine [3, 4] (©Rolls-Royce plc. Reproduced with Permission)

*Corresponding author: Arun Prabhakar, Dept. of Computer Science, University of Warwick CV4 7AL, UK.
Email addresses: arun.prabhakar@warwick.ac.uk, Christopher.Goddard@Rolls-Royce.com, d.amirante@surrey.ac.uk, reguly.istvan@itk.ppke.hu, Axel.Gerstenberger@Rolls-Royce.com, Jan-Friedrich.Suhrmann@Rolls-Royce.com, s.a.jarvis@bham.ac.uk, Leigh.Lapworth@Rolls-Royce.com, g.mudaliger@warwick.ac.uk



(a) Ray-traced image rendered using Intel OSPRay in SS02/ParaView.



(b) Location of the CFD simulation domain (opaque)

Figure 2: Rig250 high-pressure compressor test rig.

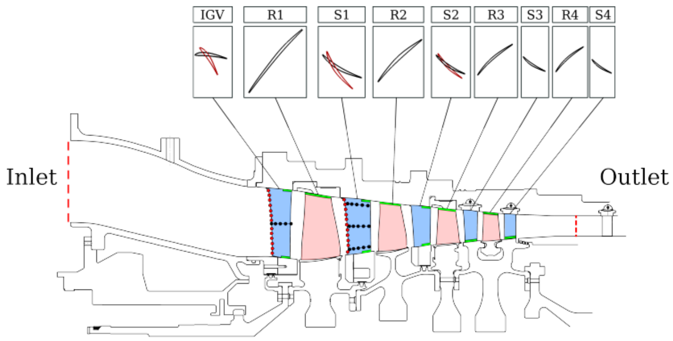


Figure 3: Rig250 Schematic. Blue – stationary zones, Pink – rotating zones (reproduced with permission [5])

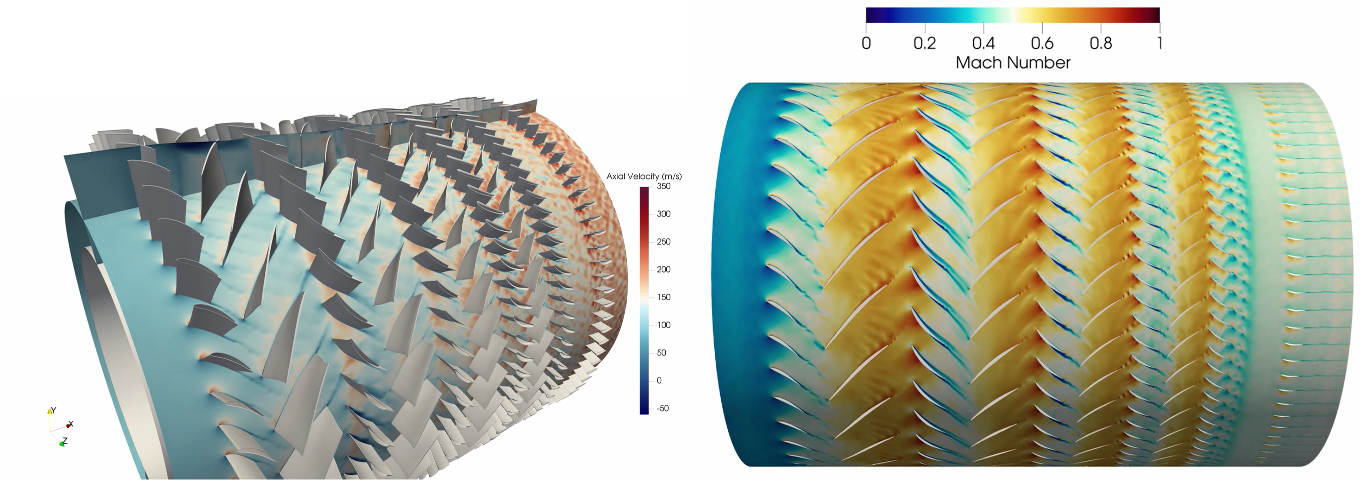
efficiencies, a measure of the degradation of energy, are therefore among the most important parameters affecting the performance of an engine. These parameters relate the effective work needed to compress or expand the flow, to that of an ideal, reversible process, in which the component is assumed to be free from frictional losses. Although isentropic efficiencies are nowadays above 90%, enormous effort continues to be made so as to improve the aerodynamic design of single components. This is particularly challenging for an axial compressor, where the secondary flows are more intense than in a turbine, frictional losses are more difficult to predict, and blades are more prone to stall in off-design conditions [6].

Simulation tools used by industry predominantly rely on the steady Reynolds-Averaged Navier-Stokes models (RANS), which are essentially PDEs providing a generally accepted mathematical model to describe the motions of turbulent flow. RANS models are limited in that, for example, they do not take into account the unsteady rotor-stator interaction that is commonplace in production compressors. Extending to Unsteady-RANS (URANS) significantly improves accuracy of the solution, but leads to increased mesh sizes to meet the required fidelity and additionally requires modeling the relative motion between rotating and stationary parts of the mesh. All these conditions render the cost of unsteady full annulus simulations prohibitively high in the past.

In this SC Scientific Visualization and Data Analytics showcase, we visualize the time-accurate URANS simulations of a full production aero-engine compressor, the Rig250 compressor from the German Aerospace Center (DLR). The results were achieved as part of the recent performance portable HPC code development [7] and its related simulation of the flow through a 4.6B element full 360 degree model of the compressor. This is a mesh and problem size representing a grand challenge problem, going beyond the fidelity requirements for virtual certification standards, realized for the first time within a tractable time-to-solution. In the accompanying video we present the results from this grand challenge simulation, through an explanatory illustration of the operation of modern gas-turbine engine compressors (Section 2), together with details of the HPC simulation application setup that led to the results and the ParaView Catalyst [8] configuration for generating the visualizations (Section 3). The high-level of detail presented (Section 4) demonstrate a step-change towards achieving virtual certification objectives under production settings using practicable HPC solutions.

2. Rig 250 Compressor Model

The Rig250 research compressor at DLR is a 4.5-stage high-pressure compressor test rig, and is representative of a modern high pressure compressor in an aero gas turbine engine [5] (see Figure 3). As air flows through the rig, it encounters several different rows of airfoils, also called *vanes* or *blades*: firstly the inlet guide vanes (IGV), then four stages of rotor and stator blades, and finally the outlet guide vanes (OGV). The action of the rotating rotor blades on the air flow causes the tangential velocity component of the air to increase, while the stationary stator blades remove this tangential velocity, causing the pressure and temperature of the air to increase. In modern aero gas turbine engines, the overall pressure rise of the air as it flow through the entire compressor system is approaching a ratio of 50:1 [9]. The IGV and first two rows of stator blades are variable, which means that the angle of the blades can be varied for different operating conditions and rotational speeds of the rotors, in order to achieve the optimum angle of incidence of the air flow onto the rotors and reducing the chance of stall occurring in the compressor. In the present work, we simulate an off-design condition with the rotors rotating at approximately 11,000 RPM, or 85% of the design speed.



(a) Axial velocity.

(b) Mach number on a cut-plane

Figure 4: Flow contours on the Rig250 compressor

Figure 2a shows an overall view of the Rig250 high-pressure compressor rig geometry. The image was rendered using the Intel OSPRay [10] pathtracer in ParaView 5.9.1, using 10 rays per pixel and with noise reduction via Intel Open Image Denoise [11]. The realism provided by the OSPRay pathtracer, in terms of material properties, shadows and reflections, helps to make the visualization of the complex rig geometry more effective. Figure 2b shows a rasterised image of the Rig250 geometry, highlighting the location of the CFD simulation domain in relation to the overall rig geometry.

3. HPC Simulation and Visualization Setup

3.1. Rolls-Royce Hydra and JM76 Coupler

To simulate the flow through the Rig250 compressor we carried out a time accurate URANS computation over the full annulus rotors and stators using the Rolls-Royce in-house production CFD solver, Hydra [12, 13] implementing a sliding planes numerical setup. Hydra has been in development for over 20 years, starting from a Fortran 77 code-base and have recently been re-engineered [14, 15] to use the OP2 embedded domain specific language [16] to realize a performance portable application, OP2-Hydra, capable of running on multi-core and many-core hardware. The sliding-planes setup was implemented using JM76 [17], Rolls-Royce's in-house coupler framework.

The relative motion between stator and rotor passages were handled by solving the flow equations in each zone in a relative frame of reference. The unsteady interaction between rotors and stators were accounted for using the industry de-facto method of sliding planes. However the setup was implemented using a coupled approach where discrete coupler software was run separately, but simultaneously to the CFD simulations of the blade rows, on a set of processors dedicated exclusively to performing the search and interpolation between interfaces. The related HPC scaling and performance results of these simulations without Catalyst visualization pipelines can be found in

Table 1: Systems specifications.

System	ARCHER2 HPE Cray EX [18]
Processor	AMD EPYC 7742 @ 2.25 GHz
(procs×cores)/node	2×64
Memory/node	256 GB
Interconnect	HPE Cray Slingshot 2×100 Gb/s bi-directional/node
OS	HPE Cray LE (based on SLES 15)
Compilers	GNU 10.2.0
Compiler Flags	`-O2 -eF -fPIC`
Power/node	660W

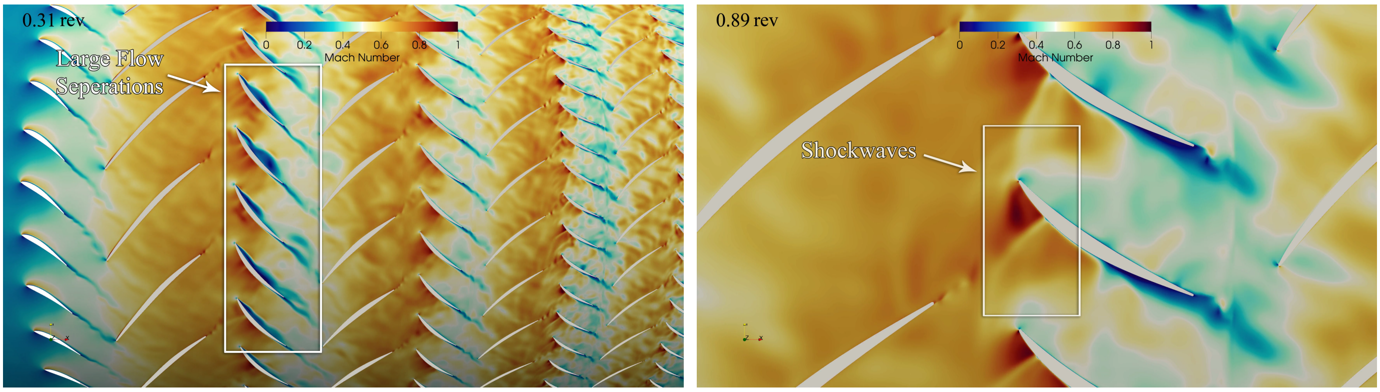
[7]. Performance and scaling indicate a speed up of the simulation by an order of a magnitude relative to current production capability.

The visualization results were obtained from the ARCHER2 [18] HPC system, an HPE-Cray EX system located at EPCC UK. Each ARCHER2 node consists of two AMD EPYC 7742 processors each with 64 cores (128 total cores) arranged in a 8 NUMA regions per node (16 cores per NUMA region) configuration. Each node is equipped with 256 GB memory. The nodes are interconnected by a HPE Cray Slingshot, 2×100 Gb/s bi-directional per node network. Table 1 briefly details the main specifications of ARCHER2.

3.2. Visualization Setup

SS02 is the Rolls-Royce in-house visualization tool, based on ParaView (currently ParaView 5.9.1), with proprietary readers, filters and writers that enable Rolls-Royce engineers to read in and perform complex analyses of data sets from the various simulation codes used within the company.

The vast increases in computer processing power of the world's largest supercomputers in recent years, such as ARCHER2 at EPCC, has enabled huge increases in the size and fidelity of engineering simulations. However, the rate of increase of computer processing power has been much greater than the rate of increase of I/O bandwidth and capacity. *In situ visualization*, or *co-processing*, where analysis and reduction



(a) Flow separation. (b) Shock waves

Figure 5: Flow features with axial velocity.

are performed on the simulation data in memory as the simulation is running, is an effective solution to avoiding the I/O bottleneck. Apart from reducing the amount of data written out during a simulation, the other main advantage of in situ visualization is that it gives the engineer access to *more* data, meaning that analysis at higher temporal or spatial resolutions can be performed in order to gain more insight.

In previous work [19], several different in situ visualization approaches have been implemented in the Rolls-Royce Hydra CFD code and explored; the present work uses the ParaView Catalyst library for in situ visualization.

3.2.1. Catalyst Pipeline

Figure 6 shows the Catalyst pipeline used to generate the required data extractors. The order of the filters in the pipeline can have a significant impact on the memory usage and time taken to process the Catalyst pipeline during the simulation. For example, it is important to reduce dimensionality of the data as early as possible in the pipeline, to prevent expensive operations from being performed on the full volumetric dataset. Hence the *Rotate for time* filter which does a computationally expensive calculation, is applied last before exporting vtm files. The resulting vtm extract files, that are written out during the simulation, are significantly smaller compared to an unsteady flow file. For the 4th stator row which comprises of the largest amount of nodes for this Rig250 model, the size of a vtm file per timestep is about 2GB, compared to an unsteady flow file which is about 30GB for that row. This represents a 15× reduction in file size.

4. Visualizations

4.1. Axial Velocity

A key parameter to analyze is the flow velocity (Figure 4a) through the compressor. Due to viscosity, the air passing over each blade develops a boundary layer next the surface. This can be seen when viewing the airfoil – the cross-sectional shape of the blade. Air passing over the airfoil develops a boundary layer next the surface. The width of the boundary layer increases along the airfoil until it reaches the trailing edge where it forms a separation in the flow, also known as a wake, that propagates downstream. The flow within the boundary layers and wakes

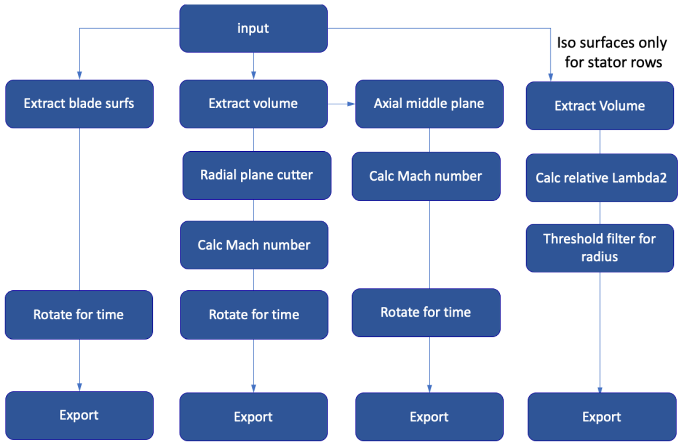


Figure 6: Catalyst Pipeline

is turbulent and unsteady. Overlaid on this is the periodic unsteadiness due to the relative motion of the rotors and stators. Focusing on the IGV and rotor 1 on the left, the wakes from the IGVs are clearly visible. We can also see the IGV wakes being "cut" by the rotors as they move around the annulus. The IGV wakes propagate through rotor 1 and can be seen as the diagonal stripes moving downstream through the rotor passages.

4.2. Mach Number

Figure 4b shows contours of the Mach number of the airflow on a cutplane through the Rig250 compressor. The Mach number, which is the ratio of the flow velocity to the speed of sound is another important parameter computed and observed in simulations in CFD analysis. A Mach number of 1 indicates a local velocity equal to the speed of sound. In this simulation of the Rig250, observing the Mach number across the compressor, it can be seen how the wake propagation and the rotor "cutting" (Figure 5a). The Mach number is higher in the rotors than the stators due to the higher tangential velocities induced by the rotor motion. Of interest is the flow in stator 1 which shows a much thicker boundary than the other stators and also frequent separations within the stator passage. The simulation here confirms the *off-design* conditions tested. The stator 1 is a variable vane and at off-design conditions, the variable vanes are set at a higher angle of attack to prevent the compressor stalling. This

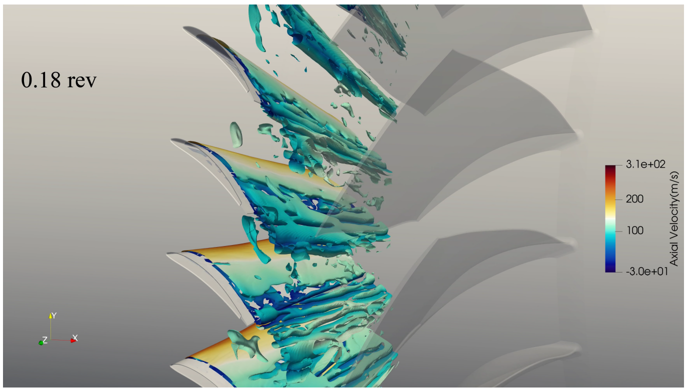


Figure 7: Axial velocity isosurfaces

can be seen by the fact that the boundary layers of the downstream stators are, relatively, more stable. Again this is consistent with our experimental understanding of compressors.

Zooming into one of the stator blades (Figure 5b), we can see the unsteady shock waves that form on the suction side of the stator. These propagate upstream and interact with the rotor wakes. The rotor-stator interaction causes the shock waves to move forward along the stator and new ones to form. The high degree of unsteadiness is again due to the off-design condition at which the simulation was run.

4.3. Axial Velocity Isosurfaces

Visualizing the axial velocity in 3-dimensions gives rise to isosurfaces (Figure 7). The view presents surfaces of the flow that has constant axial velocity and display separate regions of forward and reverse velocity. In this visualization of isosurfaces we see that the IGV boundary layers and near wake are largely 2-dimensional in nature near the mid-span. This is as expected as the IGVs are sitting in a clean inlet flow with a low angle of attack. We see evidence of the more 3-dimensional separations that occur at the junctions of the stators and the endwalls.

5. Conclusion

In this SC22 Scientific Visualization and Data Analytics showcase, we visualized the time-accurate, unsteady CFD simulation of a high-pressure compressor test rig, the Rig250 from the German Aerospace Center - DLR - to understand the factors affecting operation during a typical engine flight cycle.

The visualizations were achieved through recent work at the University of Warwick and Rolls-Royce, simulating the flow through a 4.6B element full 360 degree model of the compressor, a mesh and problem size representing a grand challenge problem, going beyond the fidelity requirements for virtual certification standards. The level of detail presented and the HPC simulation leading to the solution demonstrate a step-change towards achieving these objectives under production settings using practicable HPC systems.

Acknowledgments

This research is supported by Rolls-Royce plc., and by the UK EPSRC (EP/S005072/1 – Strategic Partnership in Computational Science for Advanced Simulation and Modelling of Engineering Systems – ASiMoV). Gihan Mudalige was supported by the Royal Society Industry Fellowship Scheme (INF/R1/1800 12). The work used the ARCHER2 UK National Supercomputing Service (`https://www.archer2.ac.uk`). Axel Gerstenberger's and Jan-Friedrich Suhrmann's work was conducted within the framework of the PRESTIGE project (20T1716A), funded by Rolls-Royce Deutschland Ltd & Co KG and the Bundesministerium für Wirtschaft und Energie; Rolls-Royce Deutschland's and DLR's permission to publish this work is greatly acknowledged. We are thankful to EPCC for the support provided in using the ARCHER2 cluster. We would also like to thank Paolo Adami and the Hydra development team at Rolls-Royce plc., for invaluable technical discussions.

References

[1] Net Zero Strategy: Build Back Greener, HM Government, Crown Copyright, 2021, uK Department for Business Energy and Industrial Strategy.

[2] Flightpath 2050 : Europe's vision for aviation : maintaining global leadership and serving society's needs, Publications Office, 2011, european Commission and Directorate-General for Mobility and Transport and Directorate-General for Research and Innovation. doi:doi/10.2777/50266.

[3] World's most efficient large aero-engine - trent xwb (Accessed May 2021).
URL `https://www.rolls-royce.com/products-and-services/civil-aerospace/airlines/trent-xwb.aspx#section-technology`

[4] A. Powell, K. Choudry, A. Prabhakar, I. Reguly, D. Amirante, S. Jarvis, G. Mudalige, Predictive analysis of large-scale coupled cfd simulations with the cpx mini-app, in: 2021 IEEE 28th International Conference on High Performance Computing, Data, and Analytics (HiPC), 2021, pp. 141–151. doi:10.1109/HiPC53243.2021.00028.

[5] V. Marciniak, A. Weber, E. Küegler, Modelling transition for the design of modern axial turbomachines, in: 6th European Conference on Computational Fluid Dynamics, Barcelona, Spain, 2014, pp. 20–25.

[6] N. Cumpsty, Jet Propulsion: A Simple Guide to the Aerodynamic and Thermodynamic Design and Performance of Jet Engines, 2nd Edition, Cambridge University Press, 2003. doi:10.1017/CBO9780511809415.

[7] G. Mudalige, I. Reguly, A. Prabhakar, D. Amirante, L. Lapworth, S. Jarvis, Towards virtual certification of gas turbine engines with performance-portable simulations, in: 2022 IEEE International Conference on Cluster Computing (CLUSTER), 2022.

[8] N. Fabian, K. Moreland, D. Thompson, A. Bauer, P. Marion, B. Geveci, M. Rasquin, K. Jansen, The ParaView coprocessing library: A scalable, general purpose in situ visualization library, in: 1st IEEE Symposium on Large-Scale Data Analysis and Visualization (LDAV), 2011, pp. 89–96. doi:10.1109/LDAV.2011.6092322.

[9] Rolls-Royce plc, The Jet Engine, Rolls-Royce Technical Publications, 2005.

[10] I. Wald, G. Johnson, J. Amstutz, C. Brownlee, A. Knoll, J. Jeffers, J. Günther, P. Navratil, OSPRay – A CPU Ray Tracing Framework for Scientific Visualization, IEEE Transactions on Visualization and Computer Graphics 23. doi:10.1109/TVCG.2016.2599041.

[11] Intel: Open Image Denoise (Accessed Jul 2022).
URL `https://www.openimagedenoise.org`

[12] P. Moinier, J.-D. Muller, M. B. Giles, Edge-based multigrid and preconditioning for hybrid grids, AIAA Journal 40 (10) (2002) 1954–1960. doi:10.2514/2.1556.
URL `https://doi.org/10.2514/2.1556`

[13] M. B. Giles, M. C. Duta, J.-D. Muller, N. A. Pierce, Algorithm developments for discrete adjoint methods, AIAA Journal 41 (2) (2003) 198–205. doi:10.2514/2.1961.
URL `https://doi.org/10.2514/2.1961`

[14] I. Z. Reguly, G. R. Mudalige, C. Bertolli, M. B. Giles, A. Betts, P. H. J. Kelly, D. Radford, Acceleration of a Full-Scale Industrial CFD Applica-

tion with OP2, IEEE Transactions on Parallel and Distributed Systems 27 (5) (2016) 1265–1278. doi:10.1109/TPDS.2015.2453972.

[15] I. Z. Reguly, G. R. Mudalige, Modernising an industrial cfd application, in: 2020 Eighth International Symposium on Computing and Networking Workshops (CANDARW), 2020, pp. 191–196. doi:10.1109/CANDARW51189.2020.00046.

[16] OP2 for Many-Core Platforms, `https://github.com/OP-DSL/OP2-Common` (2022 (Online)).

[17] D. Amirante, V. Ganine, N. J. Hills, P. Adami, A coupling framework for multi-domain modelling and multi-physics simulations, Entropy 23 (6). doi:10.3390/e23060758.

[18] ARCHER2, `https://www.archer2.ac.uk` (Accessed Jan 2022).

[19] A. Gerstenberger, M. Meyer, P. Adami, J. Suhrmann, 3D CFD in-situ co-processing for turbomachinery design, in: ISAV'21: In Situ Infrastructures for Enabling Extreme-Scale Analysis and Visualization, 2021. doi:10.1145/3490138.