UNSTEADY FULL ANNULUS MULTI-STAGE COMPRESSOR CALCULATIONS – DETAILS ON CFD-EXPERIMENT COMPARISON

Oliver Reutter
DLR, German Aerospace Center
Institute of Propulsion Technology
Researcher in Dept. Fan and Compressor
Linder Höhe
51147 Köln, Germany
oliver.reutter@dlr.de

Graham Ashcroft
DLR, Institute of Propulsion Technology
Group leader in Dept. Numerical Methods

Eberhard Nicke
DLR, Institute of Propulsion Technology
Head of Dept. Fan and Compressor

Edmund Kügeler
DLR, Institute of Propulsion Technology
Head of Dept. Numerical Methods

ABSTRACT

The unsteady flow in a multi-stage compressor has been simulated using time-domain methods to gain a better insight into the complex flow physics and address the computational bottlenecks associated with large scale numerical simulations in the time domain. For these investigations the time-periodic flow in the DLR research compressor Rig250 has been simulated. Rig250 is a 4.5 stage axial flow compressor with a swan neck that includes struts for support. In the time-domain simulation the struts, IGVs and first two stages are computed time-accurately using a full annulus mesh to fully capture the blade row interactions. The remaining two stages are included in the simulation by solving, with single passages, for the time-mean flow. In total, the low Reynolds full annulus mesh comprises nearly $1x10^9$ nodes distributed over 6000 blocks.
For an operating point close to maximum efficiency, comparisons are made between the time-accurate full annulus calculation and experimental test data to see how Kiel probe stagnation pressure measurements can be resolved. It is shown, that with larger efforts in simulations also the comparison between numerical and experimental data has to be conducted with greater care. The numerical and experimental results are in good agreement.

KEYWORDS: *Full annulus, CFD, validation, Kiel probe, virtual twin*

NOMENCLATURE

CAD – **C**omputer **A**ided **D**esign
CFD – **C**omputational **F**luid **D**ynamics
DLR – German Aerospace Center, German: **D**eutsches Zentrum für **L**uft- und **R**aumfahrt
IGV – **I**nlet **G**uide **V**ane
I/O – **I**nput and **O**utput for computers
RANS – **R**eynolds-**A**veraged **N**avier **S**tokes
Rig250 – DLR research compressor

1 INTRODUCTION

Numerical simulations are integral to the design of modern multi-stage compressors. To allow engineers to continue to develop ever more efficient and compact designs, it is necessary for the design tools to provide an ever more accurate and detailed description of the underlying complex, three-dimensional, time-dependent flows. A central feature of turbomachinery flows commonly neglected in numerical simulations used for the aerodynamic design of multi-stage compressors is flow unsteadiness. Traditionally, the high computational costs associated with simulations resolving unsteady flow phenomena have prohibited their use in the design process. However, as computational resources have increased in recent years unsteady simulations have become increasingly viable. Other work on full annulus calculations has focused on stall inception [1] or transition onset [2]. Nevertheless, for large multi-stage compressors or turbines time-accurate simulations are still often prohibitively expensive.

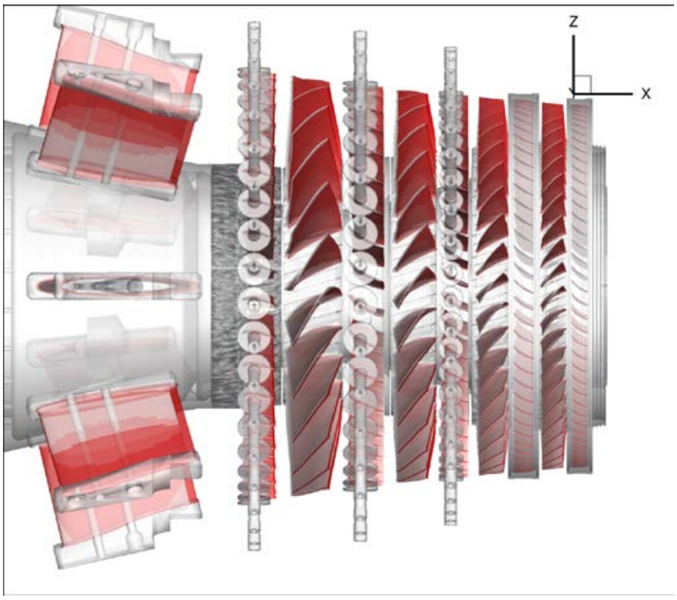

Figure 1: DLR research compressor Rig250. Numerical (CFD, only blades, red) and experimental (CAD, gray) setup put above each other to check the correct positioning.

In this work the unsteady flow in a multi-stage compressor has been simulated using time domain methods to gain a better insight into the complex flow physics and address the computational bottlenecks associated with large scale numerical simulations in the time-domain. For these investigations the time-periodic flow in the DLR research compressor Rig250 has been simulated, see Figure 1. Rig250 is a 4.5 stage axial flow compressor with a swan neck that includes struts for support. IGV as well as stator 1 and stator 2 can cover a wide range of angle settings in order to be operable at part speed, but in this paper only nominal settings were used. The compressor is aerodynamically representative of the front block of a power gas turbine. Mach number level and loading of the stages are also comparable with the front stages of an HPC in a modern aero engine. It has struts in the region of the swan neck duct used for support and also for leading cooling oil to the shaft.
The DLR Rig250 is equipped with measurement sensors for stagnation pressure, stagnation temperature, static pressure, temperature, strain gauges and dynamic pressure. It has been tested in several campaigns over the last years [3, 4]. In the time-domain simulation the strut, IGV and first two stages are computed time-accurately using a full annulus mesh to fully capture the blade row interactions. The remaining two stages are included in the simulation by solving, with single passages, only for the time-mean flow. In total, the low Reynolds full annulus mesh comprises approximately 1×10^9 nodes distributed over 6000 blocks. For this order of magnitude one needs access to a large computer cluster, but still faces the problem of long times for calculations.

Kiel probes are the standard measurement devices for stagnation pressure and have a long history of usage [5]. For an operating point close to maximum efficiency comparisons are made between the time-averaged full annulus calculation and experimental data to see how Kiel probe stagnation pressure measurements can be resolved, see Figure 2.

2 NUMERICAL SETUP

For the numerical setup the 10 rows, see Table 1, of the compressor were meshed individually with the in-house mesher PyMesh. Care was taken to have y+ values of approximately 1 on the hub, casing and on all blades and vanes, i.e. on all wetted walls, to be able to apply the low Reynolds boundary condition at all solid walls. Altogether a high resolution of the mesh was chosen to allow wakes and shocks to pass well resolved between the adjacent rows in the numerical model. At the entry a stagnation temperature and a stagnation pressure distribution taken from the experimental data was applied. The numerical setup consisted of a full annulus mesh of the strut, the IGV, Rotor1, Stator1, Rotor2 and Stator2 followed by a single passage of the Rotor3 to Stator4. The first 6 rows were calculated unsteady with a zonal connection between the rows for the unsteady calculations; the remaining rows were calculated steady state with mixing-planes between them to provide a fitting boundary condition for the first rows. The mesh consisted of nearly 1 billion nodes. Data on the blade counts is given in Table 1. All in all this makes 186 blades which were simulated.

Table 1: Blade counts for the Rig 250

Stage	Row	Blades	Simulated
	Strut & swan neck	7	Full annulus
	IGV	40	Full annulus
1	Rotor1	23	Full annulus
	Stator1	36	Full annulus
2	Rotor2	28	Full annulus
	Stator2	48	Full annulus
3	Rotor3	38	Single passage
	Stator3	68	Single passage
4	Rotor4	47	Single passage
	Stator4	80	Single passage

As the solver for the Reynolds-averaged Navier-Stokes equations the DLR-solver TRACE was used which is developed at DLR especially for turbo machinery applications [6, 7]. It has been used and validated in many different turbo machinery projects. TRACE momentarily uses an approach in which the mesh and the solution are combined in a single CGNS-format file.
This was the first project were such a large case with nearly 1 billion nodes was calculated at the Institute of Propulsion Technology. The main goal of the project was to overcome problems arising from the large size of the setup. It might be an option to split the mesh and the solution to be able to handle such large cases easier.
The placing of the vanes and blades was done according to the experimental setup in order to get the clocking right, see Figure 1. For single passage calculations the positioning within the passage is of no importance, but because of clocking for full annulus calculations it is.

3 MEASUREMENTS AND THEIR COMPARISON TO CFD

Stagnation pressure data were in part obtained by rakes and in part by stator vanes equipped with Kiel probes at the leading edge. These are fastened to the stator vanes by little metal sheets, see Figure 2. In the experiment the position of each probe is given by the position on the leading edge, which is nearly where the actual probe sensor sits.
For the measurements at the investigated operating point the incidence angle at the IGV was 0.5 °, so that the results are not dependent on the flow angle. The measurement uncertainty has been calculated to be 164 Pa. The influence of the Kiel sensors on the results has not been assessed. It is an intrinsic problem, as it is not possible to obtain experimental data without installing probes. On top

as it is rather expensive and time intensive to conduct experiments with the 4.5 stage DLR research compressor Rig250, there have not been enough resources to test the rig also without instrumentation in order to obtain global data.

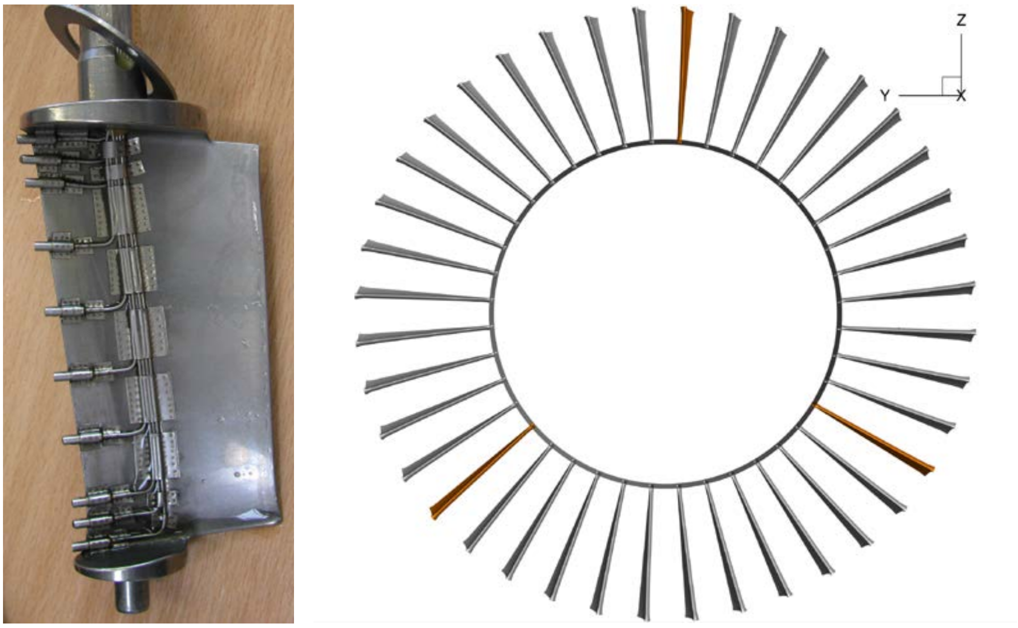

Figure 2: Instrumented IGV. There are Kiel stagnation pressure probes fastened to the leading edge of the vane. Right: The Kiel probe equipped stator vanes are marked in orange.

The probes on the leading edges of the IGVs are distributed from hub to tip according to the picture shown in Figure 2. Each vane has ten probes. At the IGV there are 3 vanes with Kiel stagnation pressure probes. The instrumented vanes are marked in orange on the right-hand side of Figure 2.

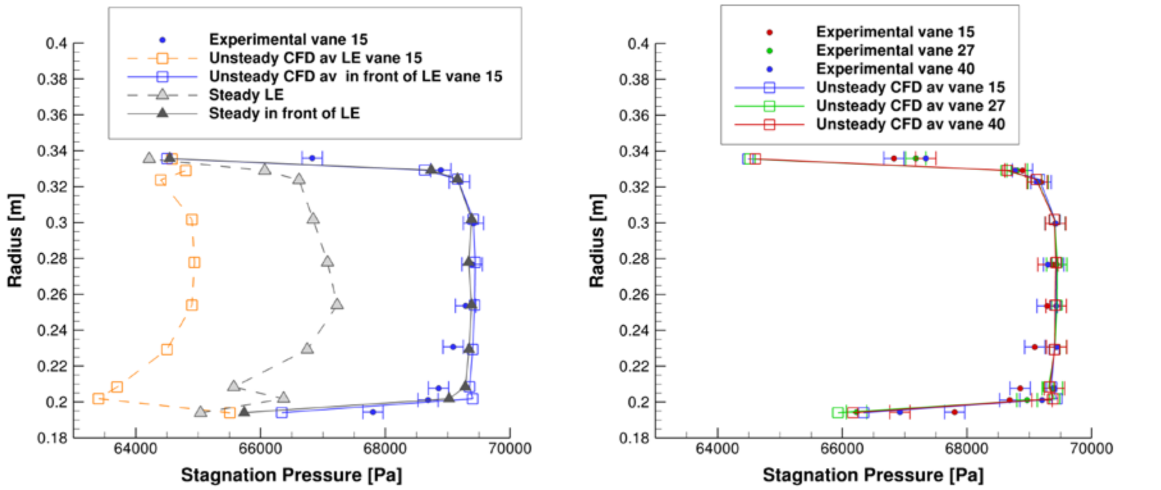

Figure 3: Time averaged stagnation pressure probes, comparison between CFD and experiment. Left: Vane 15 experimental, steady and unsteady, averaged, probes. Right: Vane 15, vane 27 and vane 40 results.

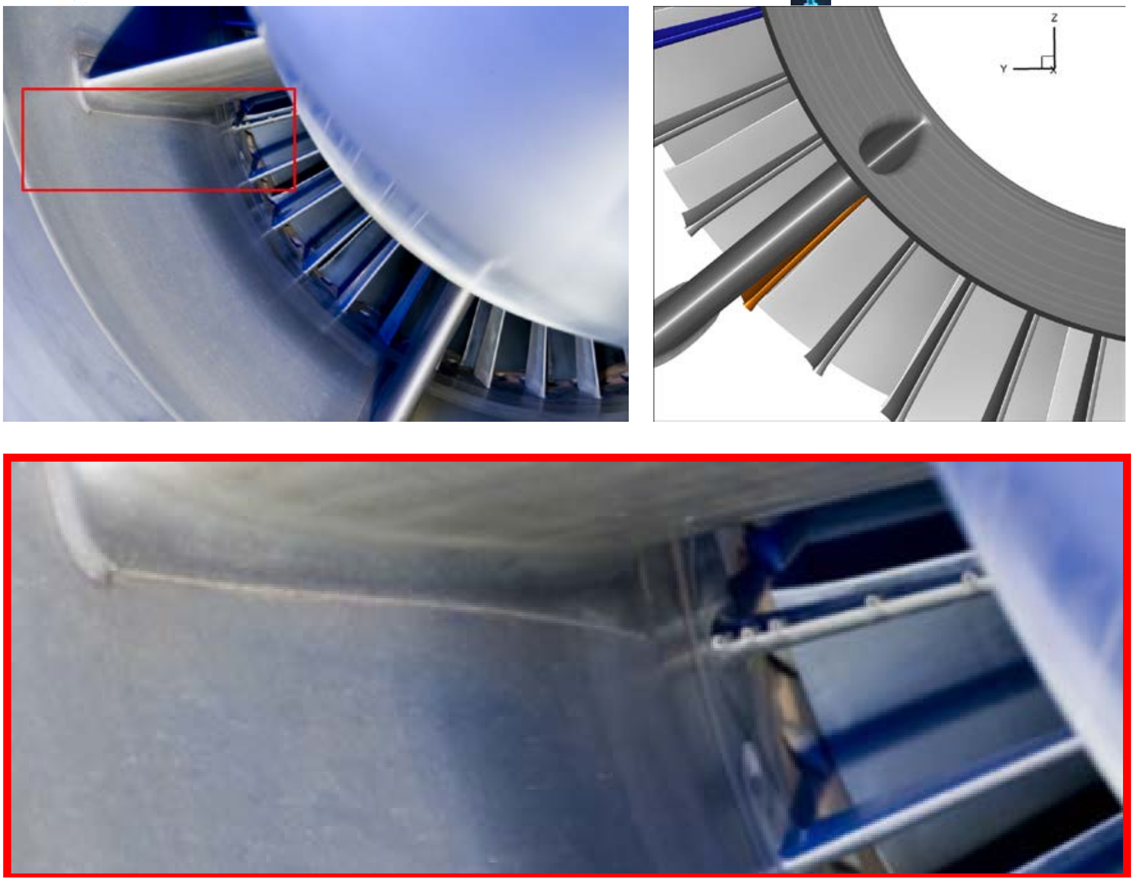

Figure 4: Location of vane 15 IGV with stagnation pressure sensors partly behind a strut, above left a photograph, above right a view of the situation in CFD with vane 15 coloured orange. Below: A detail of the photograph.

In the numerical setup it is possible to place 'numerical' probes which record the time signal at a certain mesh node during the simulation. In this output all fluid properties are recorded for each numerical time step. In the instrumentation list the position of the Kiel probes was given by the position on the leading edge, because the instrumentation is thought of as leading edge instrumentation. When setting the probes for the numerical setup two probes where placed one directly at the leading edge and one 4 mm in front of the leading edge as it was clear, that the numerical results directly at the wall and very close to the stagnation point could be extremely sensitive to the exact location and to modelling. The IGVs equipped with stagnation pressure probes are the vanes 15, 27 and 40. The comparison of the experimental stagnation pressure Kiel probe data and the numerical results for vane 15 are shown in Figure 3 on the left-hand side. One can see that directly at the leading edge the numerical probe is already seeing losses by the boundary layer. Therefore the numerical results are very sensitive to the exact position of the stagnation point on the leading edge. There is a clear difference between the steady and the unsteady simulation directly at the leading edge arising from this sensitivity. The results are very close in front of the leading edge. What is not in the numerical results is the decrease of stagnation pressure towards the hub which can be seen in the experiment. Now looking at all 3 vanes with probes vane 15, 27 and 40 in Figure 3 on the right-hand side, one can see that the numerical and experimental results fit very well for vane 27 and 40, the only larger discrepancies being in the boundary layer, where very small errors in radial position lead to large errors in stagnation pressure. But the lower stagnation pressure in the lower half of the duct is not represented in the numerical results at a position in front of the leading edge for vane 15.

From the experimental setup it was known that vane 15 was positioned partly 'behind' a strut when seen in the direction of flow. This can be seen in Figure 4, where a photograph (top left) the CAD-model (top right) and a close up of the photograph (below) are shown.

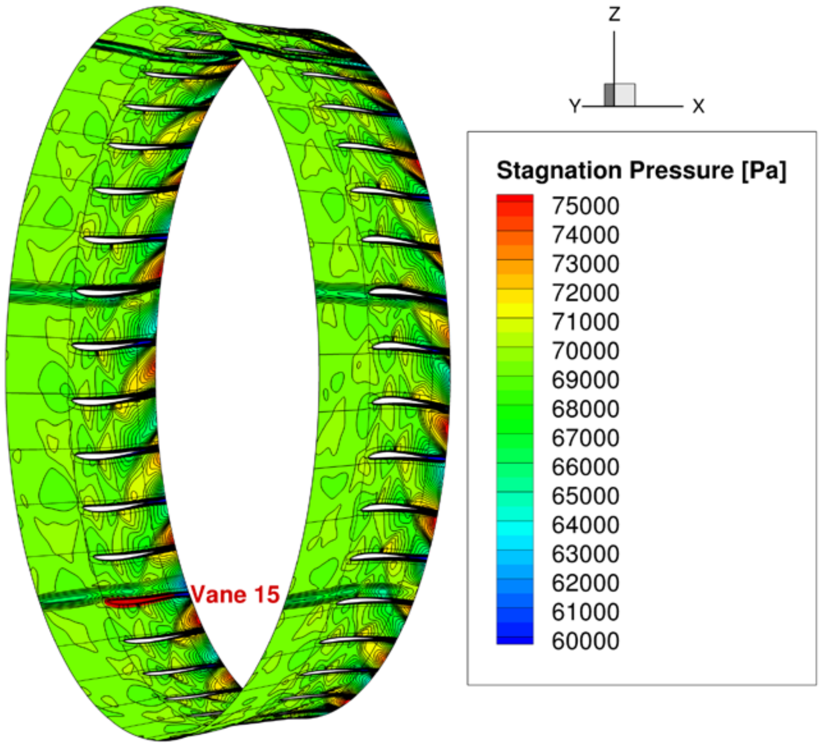

Figure 5: Full annulus of the IGV stagnation pressure distribution at the radial heigt of the second lowest experimental Kiel probe; vane 15 is marked red.

Looking at the full annulus solution of the stagnation pressure in the IGV at the height of the second Kiel probe from the hub in Figure 5, one can clearly see six of the seven wakes coming from the seven struts; one is at the bottom. One can also see, that some IGVs are fully in the wake of the strut, some IGVs are touched by the wakes and for most IGVs the wake travels through the passage or is nowhere close. Vane 15 is a vane where the wake passes very close. Now looking again at Figure 2, it can be seen on the photograph that due to the fastening of the Kiel probes, they are slightly to the pressure side and therefore a position not only in front of the leading edge, but also 2 mm to the pressure side should be chosen also for the numerical probes.

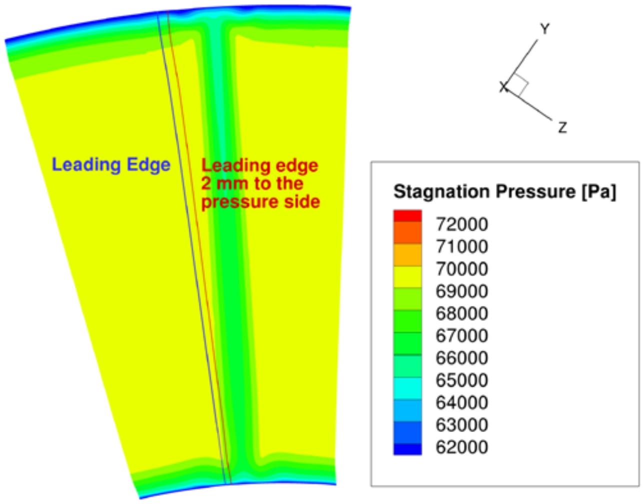

Figure 6: Position of the leading edge of IGV vane 15 in the wake of the strut.

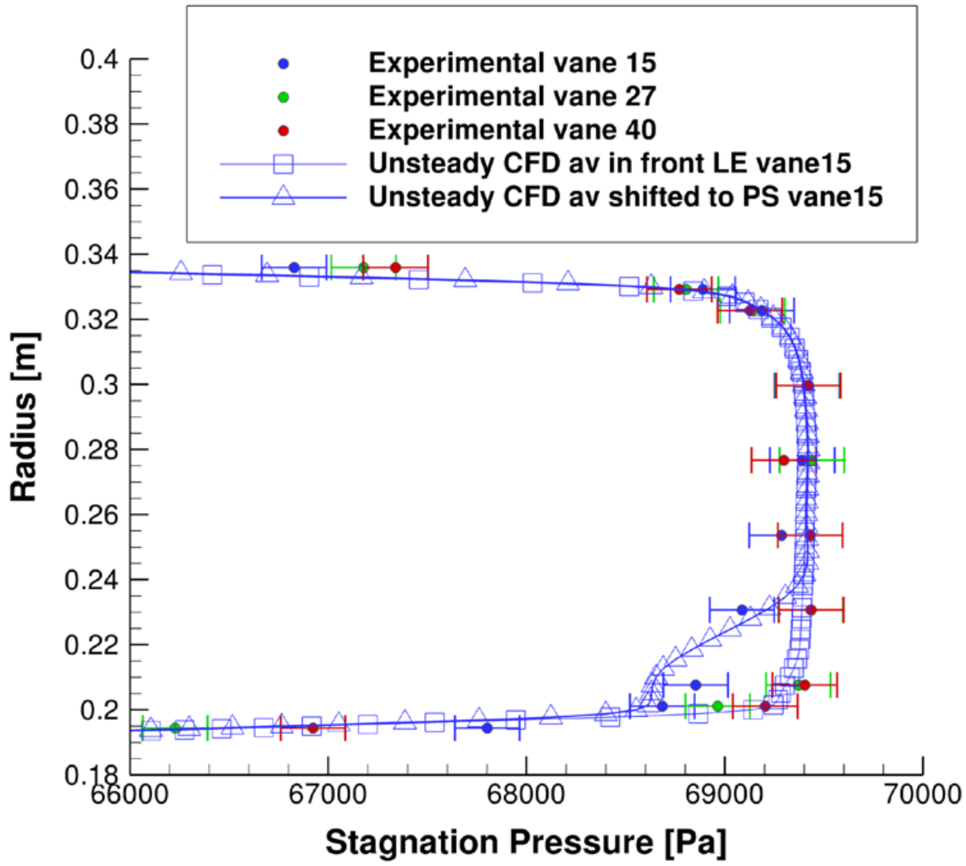

Figure 7: Comparison experiment and numerical results for vane 15 at the noted and real position.

In Figure 6 a slice of the time averaged solution in the axial position of the probe, so 4 mm in front of the IGV leading edge is shown. In blue a mesh line is shown which is directly in front of the leading edge. In red a mesh line about 2 mm to the pressure side from this line is marked. The experimental Kiel probes were fastened in this line, because they are fastened on the pressure side, see Figure 2 and Figure 4 below. It can be clearly seen, that while the strut is straight the IGV has a bow. Therefore the complete leading edge of the IGV is not in the wake of the strut but rather only the part near the hub for vane 15. The 2 mm shift was adopted based on analysis of the photograph and drawing alone, as it was not recorded and the vane is still built into the compressor and momentarily cannot be reached and measured.
In Figure 7 the effect of such a shift on the numerical results along the mesh lines from Figure 6 is given. As can be seen the averaged CFD data at the actual position of the Kiel probe in the experiment fits very well with the experimental data for vane 15. This explains the data which has been measured with the decrease of the stagnation pressure for vane 15 in the hub region very well.
In Figure 8 the stagnation pressure at the interface between the strut and the IGV bladerows is shown. It can clearly be seen, that there is a sevenfold symmetry, because there are 7 struts. Also it can be seen, that it is not astonishing, that two of three experimental Kiel probes measured the same values as there are large areas without any direct influence of the wake. Looking at the plane in Figure 8 the wakes influence about 5 % of the area. This can be compared to about 14 % of the area which is influenced by the boundary layers at the hub and the casing. Of course, for the boundary layer this effect can be measured at all circumferential positions while for the strut wake it cannot. At the same time this figure shows clearly the dilema in which everybody involved in experiments always is. Where is the best position for a limited number of probes to be placed so that the results are representative? With a limited number of probes there is no perfect place, but the needs for heightened accuracy in experiments are increasing with heightened accuracy in simulations for validation reasons. Also as the accuracy in compressor design is increasing and more and more

effects which used to be looked at as neglectible side effects are taken into account, more and more details have to be considered. In single passage steady RANS-simulations, effects like the wakes of the struts are taken into account by averaging at the mixing plane between the strut and the IGV. Then a validation with this data would use only the values of vane 27 and 40, as the partial wake at vane 15 would have to be neglected. And as the wakes do not have a huge influence the difference between the circumferentially averaged data from the calculations and the measurement data from the “clean“ IGV leading edges is small. In the experimental results there is no way of quantitatively taking the wake into account. In that point of view it used to be thought of as vane 15 was placed “wrongly“. Nowadays in the light of full annulus calculations all positions of probes are usefull for validation as long as the documentation meets the necessary accuracy levels.

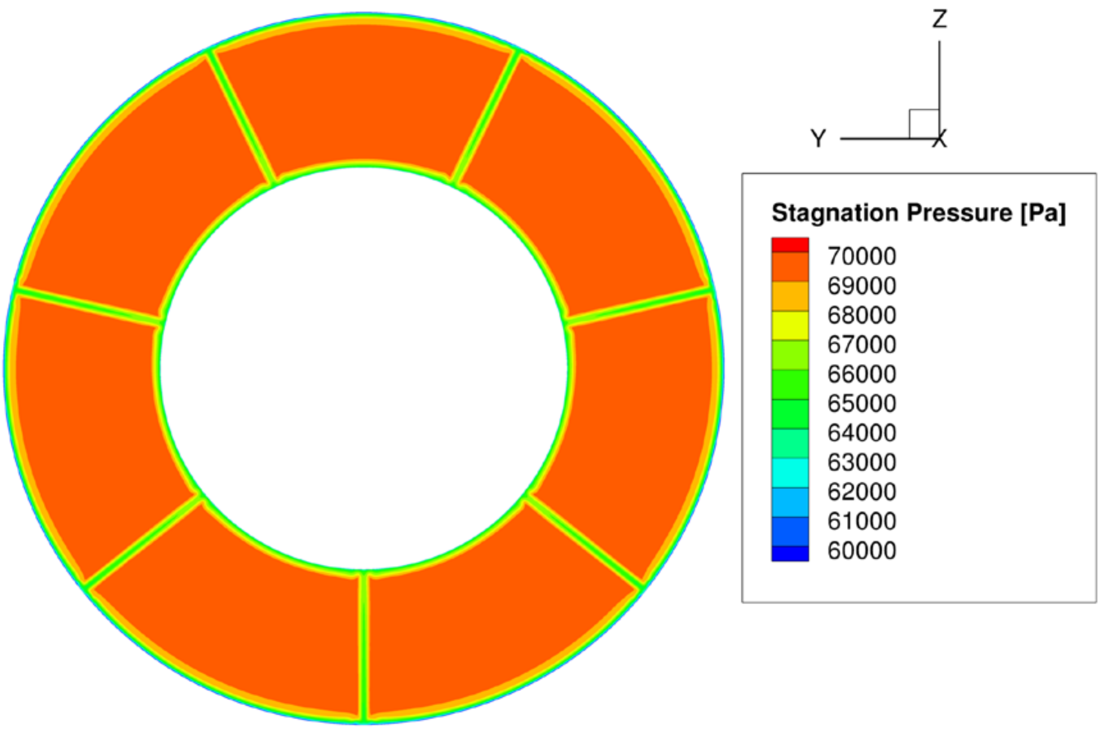

Figure 8: Stagnation pressure distribution at the interface between the strut meshes and the IGV meshes.

4 CONCLUSIONS

In this paper results from the calculation of the full annulus of the first six rows of the DLR research compressor are presented. The time resolved calculations for a mesh with nearly 1 billion nodes was successfully conducted and all numerical bottlenecks overcome. From the large amount of data which such a calculation produces a first analysis on the full annulus comparison of the IGVs instrumented with Kiel probes was conducted as they are not symmetric compared to the struts upstream. It is clear that with Kiel probe measurements the stagnation pressure at the entrance of the Kiel probe is measured.
This work shows that the ability to perform increasingly accurate full annulus, multi-stage, unsteady simulations requires accurate experimental data and equally importantly an accurate description of the test rig and its instrumentation. It is expected that the digitalization of test rigs and the development of virtual twins will provide great improvements in this respect. The numerical and experimental results are in very good agreement when heightening the accuracy of the positioning of the experimental Kiel probes. In this way also the influence of the strut on the stagnation pressure measurement on vane 15 can be used for validation purposes. It is shown that the fastening of the Kiel probe on the pressure side has an influence and has to be taken into account for good agreement. In the past typically averaged data and numerical circumferentially averaged results where compared. Not quantifiable data from experiments was omitted. Only in full annulus

calculations can data like this be used, as long as the accuracy is kept high. So with rising accuracy in simulations there is no "right" or "wrong" placement of probes anymore.

5 ACKNOWLEDGEMENTS

The work reported in this paper was conducted within the project PERFEKT, funded by the Federal Ministry of Economic Affairs and Energy (BMWi) following an Order by the German Federal Parliament under contract number 20T1309B.
Rolls-Royce Deutschland is gratefully acknowledged for the good collaboration during this project.

REFERENCES

1. Yao, J., Gorrell, S. E., Wadia, A. R.; 2008; "HIGH-FIDELITY NUMERICAL ANALYSIS OF PER-REV-TYPE INLET DISTORTION TRANSFER IN MULTISTAGE FANS - PART II: ENTIRE COMPONENT SIMULATION AND INVESTIGATION"; Proceedings of ASME Turbo Expo 2008, Berlin, Germany GT2008- -50813
2. Roeber, T., Kügeler, E., Weber, A.; 2010: "INVESTIGATION OF UNSTEADY FLOW EFFECTS IN AN AXIAL COMPRESSOR BASED ON WHOLE ANNULUS COMPUTATIONS"; , *Proceedings of ASME Turbo Expo 2010*, Glasgow, United Kingdom GT2010- -23522
3. Schönweitz, D., Goinis, G., Voges, M., Enders, G. und Johann, E.: "Experimental and Numerical Examinations of a Transonic Compressor-Stage With Casing Treatment", ASME Turbo Expo 2013, 03.-07. Juni 2013, San Antonio, USA
4. Schmitz, A., Aulich, M., Schönweitz, D. und Nicke , E.; 2012; "Novel Performance Prediction of a Transonic 4.5 Stage Compressor", ASME Turbo Expo 2012, 11.-15. Juni 2012, Kopenhagen, Denmark
5. Kiel, G. ; 1935; "TOTAL-HEAD METER WITH SMALL SENSITITVITY TO YAW"; Technical Memorandums of the National Advisory Committee for Aeronautics No. 775; Washington
6. Becker, K., Haitkamp, K., and Kügeler, E.; 2010; "Recent progress in a hybrid-grid CFD solver for turbomachinery flows", In V European Conference on Computation Fluid Dynamics ECCOMAS CFD 2010
7. Franke, M., Röber, T., Kügeler, E., and Ashcroft, G.; 2010; "Turbulence treatment in steady and unsteady turbomachinery flows", In V European Conference on Computation Fluid Dynamics ECCOMAS CFD 2010