VALIDATION AND VERIFICATION OF RANS SOLVERS FOR TUDA-GLR-OPENSTAGE TRANSONIC AXIAL COMPRESSOR

Mingmin Zhu\textsuperscript{1}, Xiao He\textsuperscript{2}, Fabian Klausmann\textsuperscript{3}, Jinfang Teng\textsuperscript{1} and Mehdi Vahdati\textsuperscript{2}

\textsuperscript{1} School of Aeronautics and Astronautics, Shanghai Jiao Tong University, Shanghai 200240, China
\textsuperscript{2} Department of Mechanical Engineering, Imperial College London, London SW7 2AZ, UK
\textsuperscript{3} Institute of Gas Turbines and Aerospace Propulsion, Technische Universität Darmstadt, Darmstadt 64287, Germany

ABSTRACT
This paper presents a comprehensive validation and verification study of turbomachinery Reynolds-averaged Navier-Stokes flow solvers on the transonic axial compressor TUDa-GLR-OpenStage. Two commercial solvers namely Ansys CFX and Numeca FineTurbo are adopted to provide the benchmark solutions, which can be used for verification of other RANS solvers in the future. Based on these solvers, five sets of grids, three advection schemes (i.e., central difference and second-order upwind from Numeca and high resolution from Ansys), two turbulence models (i.e., Spalart-Allmaras and Menter $k-\omega$ Shear Stress Transport) and two rotor-stator interface models (i.e., mixing plane and sliding plane) are investigated to quantify their effects on predicting the performance and the flow field of the compressor stage. Some geometric uncertainties and errors including the rotor running tip gap size, the stator hub cavity and the stage throat area change are also discussed. Results show that the choice of grid density and turbulence model is most sensitive to the prediction; the choice of rotor-stator interface model mainly affects the near-tip flow field, but the effect on the overall characteristics is relatively small; the choice of advection scheme has limited effect on the prediction when using a refined mesh. Based on a grid density and an advection scheme with a small discretization error, none of the turbulence models and the rotor-stator interface models can perfectly capture the overall characteristics at the peak-efficiency condition. Possible reasons for the difference between the simulations and the experiment are discussed. The grids and the major CFD results presented in this work are open-accessed to the community for further research. The results and discussions presented in this paper provide a useful reference for future practices of RANS simulations for compressors.

1 INTRODUCTION
Reynolds-averaged Navier-Stokes (RANS) simulation is the industrial workhorse for predicting turbomachinery flows, but users can occasionally find inconsistency of RANS solvers such as:

- Using the “same” numerical settings from different RANS solvers can predict different results.
- Each RANS solver has a “best-practice” list of numerical settings based on previous user experience, but this list may vary from one solver to another.
- When using the “best-practice” settings of a solver, the quantity of interest may be overpredicted in one case and underpredicted in another, despite the flow phenomena of the cases being similar.

This inconsistency not only undermines users’ trust in the RANS solvers, but also hinders developers from understanding and improving the numerical models used in RANS simulations.

The source of RANS solver inconsistency can be rooted back to the uncertainties and errors in experiments and simulations. In terms of experiments, uncertainties and errors in geometries and test instruments vary for different test cases and test rigs; in some cases, the experimental uncertainties and errors may be pronounced enough to overwhelm the difference in RANS results. In terms of simulations, the prediction is subjected to many user-specified numerical settings; comparisons among different RANS solvers at the same numerical settings are seldom reported due to user preference (i.e., following the “best-practice” guideline of a specific solver), coding preference (e.g., an in-house variant of a model is implemented instead of the original one) and potential coding errors.

To tackle the issue of RANS solver inconsistency, a series
of validation and verifications (V&V) studies of RANS solvers are crucial. Following the definitions from the American Institute of Aeronautics and Astronautics (AIAA) [1]:

- **Validation** is the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model.
- **Verification** is the process of determining that a model implementation accurately represents the developer’s conceptual description of the model and the solution to the model.

In other words, validation compares simulation results against measured data, which quantifies the simulation accuracy; verification compares simulation results obtained by different codes, which helps exclude potential coding errors. Verification is the first step towards a successful validation study.

V&V studies have drawn attention from the general aerospace sector for the past ten years [2]. In response, the NASA Turbulence Modeling Resource (TMR) website was established, which serves as an open repository of RANS turbulence model documentations and V&V test cases. However, V&V studies have not yet received much attention in the turbomachinery community. The last major V&V campaign for compressor flows dates back to the 1994 International Gas Turbine Institute (IGTI) blind test on the NASA Rotor 37 [3], along with many individual validation efforts on other NASA compressor geometries [4] over the years. From today’s perspective, the measurement accuracy of these data may no longer present state-of-the-art. Online and version-controlled documentation of these geometries and measured data was also not achieved due to the limitations of the times—users today need to find the most updated geometries and data from pieces of literature by themselves (e.g., characteristics and tip gap size of NASA Rotor 67 [5]), and some details of the geometry are still not known (e.g., running tip gap size of rotors, hub cavity geometry of stators). In addition, it is challenging to conclude from the early V&V campaign due to lack of solver verification [3]. Therefore, a new V&V campaign of turbomachinery RANS solvers based on recent common research models of compressors is of high interest.

In the 2020 GPPS Chania conference, the Institute of Gas Turbines and Aerospace Propulsion (GLR) of Technische Universität Darmstadt released the TUDa-GLR-OpenStage transonic axial compressor including the geometry in digitized form and an abundant set of measurement data [6]. This is an ideal compressor test case for RANS V&V purposes. In this work, established commercial turbomachinery RANS solvers namely Ansys CFX and Numeca FineTurbo are performed on this case to provide the benchmark solutions. These benchmark solutions along with the associated grids and boundary conditions will be released to the public for future verification of other RANS solvers. Based on the commercial solvers, the effects of grid density, advection scheme, turbulence model and rotor-stator interface model on predicting the TUDa-GLR-OpenStage case are investigated. These results will be presented after introducing the experimental and numerical setup of the case.

2 METHODOLOGY
2.1 Case Description

The investigated TUDa-GLR-OpenStage is a single-stage high-speed axial compressor, which represents a typical front stage of a high-pressure compressor in a commercial turbofan engine. The compressor stage consists of a blisk rotor with radially stacked CDA-airfoils, a stator with optimized 3D shapes, and an outlet guide vane (OGV) that straightens the flow. Pictures of the rotor and stator geometries are shown in Fig. 1. The rotor is originally designed by MTU and first tested in 1994. It has been investigated extensively in a series of previous experimental and numerical works [7–9]. The rotor blades are highly cambered near the hub and thin near the tip. The running tip gap size is approximately 0.75 mm at the design condition, and the hub fillet radius is 5 mm. The stator is designed conjointly between the GLR and the German Aerospace Center (DLR). It is optimized via an automated multi-objective optimization process to suppress the separation size [10]. The stator blades have a forward sweep feature near both endwalls and a bow feature towards the pressure surface near the casing. The shapes of the fillets at both endwalls are prescribed by a digitized geometry file. More details about the investigated stage are summarized in Table 1.

2.2 Test Rig

The TUDa-GLR-OpenStage was tested on the transonic compressor rig of GLR, which can measure the steady-state performance, the aerodynamic instabilities and the blade vibration levels of the compressors. The schematic of the test facility is shown in Fig. 2. The inlet flow passes through an inlet throttle, a settling chamber and a mass flow measurement section before reaching the compressor core. The compressor is driven by a DC motor with a gearbox. Shaft input torque and rotor speed is measured using a torque meter.

The measurement sections of the compressor are also marked in Fig. 2. At the stage inlet section (ME15), radial

---

Table 1: Design parameters of TUDa-GLR-OpenStage [6].

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Symbols</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor tip Mach number</td>
<td>( M_t )</td>
<td>1.17</td>
</tr>
<tr>
<td>Rotor mean aspect ratio</td>
<td>( (r_t - r_h)/c_m )</td>
<td>0.94</td>
</tr>
<tr>
<td>Rotor hub-to-tip ratio</td>
<td>( r_h/r_t )</td>
<td>0.51</td>
</tr>
<tr>
<td>Rotor tip gap-to-chord ratio</td>
<td>( \tau/c_t )</td>
<td>0.8%</td>
</tr>
<tr>
<td>Rotor tip radius (mm)</td>
<td>( r_t )</td>
<td>190</td>
</tr>
<tr>
<td>Rotor tip chord length (mm)</td>
<td>( c_t )</td>
<td>94</td>
</tr>
<tr>
<td>Rotor design speed (RPM)</td>
<td>( N )</td>
<td>20000</td>
</tr>
<tr>
<td>Stage flow coefficient</td>
<td>( \phi )</td>
<td>0.37</td>
</tr>
<tr>
<td>Stage loading coefficient</td>
<td>( \psi )</td>
<td>0.40</td>
</tr>
<tr>
<td>Stage mass flow (kg/s)</td>
<td>( m )</td>
<td>16.0</td>
</tr>
<tr>
<td>Stage total pressure ratio</td>
<td>( \pi )</td>
<td>1.50</td>
</tr>
<tr>
<td>Number of blades</td>
<td>( Z )</td>
<td>16/29/5</td>
</tr>
</tbody>
</table>

In this work, two widely used commercial RANS flow solvers are adopted: Ansys CFX 20.1 and Numeca FineTurbo discretizes the RANS equations using a cell-centered control volume approach based on structured meshes. There are two spatial discretization schemes with second-order accuracy available: the central scheme with a Jameson-Schmidt-Turkel type dissipation term [13] and the upwind scheme using flux difference splitting from Roe [14]. The temporal discretization is achieved by an explicit fourth-order accurate Runge-Kutta integration scheme. For steady-state cases, the time-marching algorithm is adopted. Local time stepping, implicit residual smoothing and multigrid techniques are applied to accelerate convergence. The modified Spalart-Allmaras (SA) turbulence model [15] (i.e., “SA-noft2” according to NASA TMR, with a modified strain rate term to improve numerical stability [16]) is set as the default for turbomachinery simulations. The SST turbulence model (i.e., “SST” according to NASA TMR) [17] is also available. More details about the solver can be found in its user guide [18].

2.4 Flow Domain, Boundary Conditions and Grids

The simulation is conducted by using a single-passage flow domain with the circumferentially periodic boundary condition, as illustrated in Fig. 3. The inlet plane is set at the stage inlet section ME15. The measured total pressure and total temperature profiles are used to prescribe the inlet boundary condition. In addition, the inlet flow direction is axial; the turbulence intensity and turbulence length scale are estimated as 4% and 0.09 mm respectively, which are equivalent to an eddy viscosity ratio of 35. At the rotor-stator and the stator-OGV interface, a mixing plane boundary condition is used. The outlet boundary is set at the location about 1.5 times the compressor core axial length downstream of the OGV. A radial equivalent backpressure is assigned at the outlet boundary, but a constant backpressure value can also be used since the outlet flow is turned to the axial direction by the OGV. When approaching the stall limit, a step increase of backpressure of 0.2 kPa is used. The last converged point is taken as the numerical “stall” point. It should be noted that the real-world stall phenomena are by nature unsteady and circumferentially non-periodic, which cannot be reproduced by the single-passage steady-state RANS simulations. Therefore, only flow conditions away from the peak pressure point are analyzed in this work.

The flow domain is discretized by hexahedron grids with...
3 RESULTS

Based on the numerical approach, the effects of grid density, advection scheme, turbulence model, and rotor-stator interface model are discussed accordingly. Stage performance characteristics and radial flow profiles at the rotor and stator exits are compared with the experimental results. Post-processes (i.e., measurement location and averaging method) of these simulation data are consistent with that used in the measured data. Differences between numerical and experimental results, as well as between different numerical settings, will be discussed.

3.1 Effect of Grid Density

In this section, five sets of grids (detailed in Table 2) are used for RANS simulations. The stage performance characteristics at the design speed are presented in Fig. 5, where simulation results are represented by solid curves with open symbols, and the experimental data are denoted by solid squares with error bars. As the grid density increases, the characteristic curves gradually translate towards the top-right corner (i.e., higher pressure ratio, efficiency and mass flow rate). No evident changes in the stall margin and the slopes of pressure ratio and efficiency curve are observed. The results from the Coarse grid match the best with the measured data, but these results are subjected to pronounced discretization error (i.e., the difference between the Coarse and UltraFine results) and thus cannot be taken as evidence of a good validation. When refining the grid further to the Fine and UltraFine grids, the variation of pressure ratio and the efficiency are within 0.01 and 0.1% respectively, indicating a grid-independent result.

To examine the effect of grid density in detail, Fig. 6 presents the circumferentially mass-averaged radial profiles downstream the rotor and the stator, where the curves represent the numerical results obtained from the five grids and the solid circles denote the experimental results.

For the rotor exit total pressure ratio and absolute flow angle profiles shown in Fig. 6(a), grid refinement leads to an increased pressure ratio and flow angle at all spans. The Fine grid and the UltraFine grid results show an evident overprediction of the pressure ratio above 80% span, which can be caused by the geometric uncertainty near the rotor tip and the deficiency of turbulence modeling. Minor differences below 80% span are due to the difference of rotor inlet incidence. Since the choke mass flow is slightly overpredicted in the Fine grid and the UltraFine grid simulation results, the incidence of the simulation will be slightly higher than the measurement when comparing at the same mass flow rate. Hence, the pressure ratio and flow angle (i.e., work input) are slightly overpredicted below 80% span.

For the rotor exit total pressure ratio and total temperature ratio shown in Fig. 6(b), it is observed that again a grid refinement increases the pressure ratio and the temperature ratio at all spans. The overprediction of pressure ratio and work input at the rotor exit generally propagates to the stage exit. In particular, for the lower 20% span, the measured total pressure ratio is evidently smaller than the simulations. This near-hub difference can be caused by the geometric uncertainty near the stator hub and the deficiency of turbulence modeling.

To check the grid independence further, Fig. 7 plots the pressure ratio and efficiency at the PE condition against the nominal grid spacing $\Delta = (1/N_g)^{1/3}$, where $N_g$ is the total number of grid points. For completeness, results obtained by using the Numeca central solver with the SA and SST turbulence models are also included. In this plot, the open symbols represent the simulation data, the cross symbol with error bars at $\Delta = 0$ denotes the experimental data, and the curves are the least-squares fits of parabolas based on the simulation data. Ideally, the discretization error scales with
Table 2: Details of TUDa-GLR-OpenStage grids.

<table>
<thead>
<tr>
<th>Grid topology</th>
<th>UltraCoarse</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
<th>UltraFine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor Total radial grid point</td>
<td>37</td>
<td>53</td>
<td>81</td>
<td>121</td>
<td>181</td>
</tr>
<tr>
<td>Tip gap radial grid point</td>
<td>9</td>
<td>13</td>
<td>21</td>
<td>33</td>
<td>49</td>
</tr>
<tr>
<td>Hub fillet radial grid point</td>
<td>9</td>
<td>13</td>
<td>17</td>
<td>29</td>
<td>45</td>
</tr>
<tr>
<td>Total tangential grid point</td>
<td>29</td>
<td>41</td>
<td>65</td>
<td>93</td>
<td>145</td>
</tr>
<tr>
<td>Total axial grid point</td>
<td>41</td>
<td>61</td>
<td>97</td>
<td>141</td>
<td>213</td>
</tr>
<tr>
<td>Boundary layer grid point</td>
<td>9</td>
<td>13</td>
<td>21</td>
<td>29</td>
<td>45</td>
</tr>
<tr>
<td>Tip gap O-grid point</td>
<td>5</td>
<td>9</td>
<td>13</td>
<td>21</td>
<td>29</td>
</tr>
<tr>
<td>Total grid point (million)</td>
<td>0.12</td>
<td>0.28</td>
<td>1.08</td>
<td>3.36</td>
<td>11.77</td>
</tr>
<tr>
<td>Stator Total radial grid point</td>
<td>41</td>
<td>65</td>
<td>93</td>
<td>137</td>
<td>201</td>
</tr>
<tr>
<td>Tip/hub fillet radial grid point</td>
<td>9</td>
<td>13</td>
<td>21</td>
<td>29</td>
<td>49</td>
</tr>
<tr>
<td>Total tangential grid point</td>
<td>17</td>
<td>29</td>
<td>41</td>
<td>69</td>
<td>101</td>
</tr>
<tr>
<td>Total axial grid point</td>
<td>37</td>
<td>53</td>
<td>85</td>
<td>129</td>
<td>189</td>
</tr>
<tr>
<td>Boundary layer grid point</td>
<td>9</td>
<td>13</td>
<td>21</td>
<td>29</td>
<td>45</td>
</tr>
<tr>
<td>Tip gap O-grid point</td>
<td>5</td>
<td>9</td>
<td>13</td>
<td>21</td>
<td>29</td>
</tr>
<tr>
<td>Total grid point (million)</td>
<td>0.04</td>
<td>0.16</td>
<td>0.53</td>
<td>1.80</td>
<td>5.81</td>
</tr>
</tbody>
</table>

Figure 5: Effect of grid density on predicting the TUDa-GLR-OpenStage performance characteristics: (a) total pressure ratio and (b) isentropic efficiency; results obtained by using the Ansys CFX solver, the SST turbulence model and the mixing plane rotor-stator interface model.

3.2 Effect of Advection Scheme

In this section, the central scheme and upwind scheme of Numeca and the high resolution scheme of CFX are compared with each other, all of which are second-order accurate. The rest of the numerical settings are kept the “same”, including the boundary conditions, the Fine grid, the SST turbulence model and the mixing plane rotor-stator interface model. Ideally, the error related to the advection scheme scales with the square of the grid spacing, which is deemed small when using the Fine grid. Thus, overlapped results are expected.

The stage performance characteristics at the design speed are presented in Fig. 8. It shows that the Numeca(central) and the Numeca(upwind) results are almost overlapping with each other as expected, but the CFX result has slightly higher values of pressure ratio, choke mass flow rate and efficiency. The difference in pressure ratio and efficiency at the PE condition is 0.02 and 0.3% respectively, which are small but not negligible.

Detailed examinations on the radial profiles at the PE condition are presented in Fig. 9. It again shows good agreement between Numeca(central) and Numeca(upwind)
Figure 6: Effect of grid density on predicting the circumferentially mass-averaged radial profiles at the PE condition: (a) rotor exit (ME21) and (b) stage exit (ME30); results obtained by using the Ansys CFX solver, the SST turbulence model and the mixing plane rotor-stator interface model.

![Figure 6](image)

Figure 7: Grid convergence of TUDa-GLR-OpenStage at the PE condition: (a) total pressure ratio and (b) isentropic efficiency.

![Figure 7](image)

results, but the CFX result shows a slightly higher rotor pressure ratio, stage pressure ratio and stage temperature ratio for all radial spans, as well as the rotor exit absolute flow angle below 80% span. The maximum differences for the rotor pressure ratio, rotor exit flow angle, stage pressure ratio and stage temperature ratio are 0.02, 2 deg, 0.03 and 0.005 respectively.

The main reason for the minor difference between the CFX result and the two Numeca results is that the rest numerical settings are not kept exactly the same—CFX uses the 2003 version of the SST turbulence model [11], but Numeca uses the original 1994 version [17]; CFX uses a scalable wall function for the viscous wall boundaries, but the Numeca results do not use a wall function.

Regarding the two versions of the SST model, the major modification of the 2003 version is that the denominator of the eddy viscosity definition uses the strain rate magnitude, rather than the vorticity magnitude in the 1994 version. In regions where the effect of rotation surpasses shear (e.g., tip leakage vortex), the 2003 version tends to predict a higher value of eddy viscosity, a smaller blockage size (due to the turbulent mixing effect) and hence a higher pressure ratio and work input. However, such effects can only explain the near-tip difference but not that of the lower spans.

Regarding the use of wall functions, the standard wall function is expected to produce the same results as that obtained without using the wall function in this case. This is because the grids used in this case are designed to have an averaged y+ value below 3; therefore, the standard wall function will calculate the wall shear stress in the viscous sublayer branch rather than the log-law branch, the latter of which only holds true for attached flows with small pressure gradients. However, the scalable wall function used by CFX limits the y+ value to be greater or equal to 11.06. This limiter ensures that consistent predictions on the boundary layer flows can be obtained when using an arbitrary y+ value below the threshold [12]. As a result, the wall shear stress is always calculated in the log-law branch, leading to smaller skin friction and thus a higher total pressure ratio at full span.

While the above explanations are conceptually plausible, further verifications from a third solver are needed to consolidate the understanding in the future.
3.3 Effect of Turbulence Model

In this section, the SA turbulence model is compared against the SST turbulence model. The rest of the numerical settings are kept the same, including the boundary conditions, the Numeca(central) solver, the Fine grid and the mixing plane rotor-stator interface model.

The stage performance characteristics at the design speed are compared with experimental data in Fig. 10. In general, the choice of turbulence model has a great influence on both the pressure ratio and the efficiency. The SA model predicts a fair agreement with the measured data in terms of the choke mass flow, the stall margin, the NS pressure ratio and the PE efficiency. Deficiencies are that the PE pressure ratio is underpredicted by 0.03, and the NS efficiency is overpredicted by 1.4%. The SST model accurately predicts the choke mass flow and the pressure ratio at the PE condition, but it overpredicts the NS pressure ratio by 0.03, the PE efficiency by 1.8%, and the NS efficiency by 2.8%.

Comparisons between the two turbulence models on predicting the radial profiles are presented in Fig. 11. At the rotor exit shown in Fig. 11 (a), the SA model predicts the flow angle profile well but underpredicts the pressure ratio below 80% span, which accounts for the underprediction of the overall pressure ratio presented in Fig. 10(a); the SST model overpredicts the pressure ratio above the 80% span and the flow angle at full span. At the stage exit shown in Fig. 11 (b), the deficiency of the SA model from the rotor passage propagates to the stage exit, leading to a generally underpredicted pressure ratio and temperature ratio profile; the SST model overpredicts the pressure ratio near both endwalls, where the near-tip difference is propagated from the rotor passage and the near-hub difference is generated in the stator passage.

Based on the above observations, there is no consistent evidence showing one turbulence model is superior to the other, and the choice of SA or SST turbulence model remains an open option for the users. The deficiency of the SA model at the PE condition mainly occurs in the rotor. From the radial
profiles at the PE condition, the SA model seems to predict the work input at a lower incidence or a higher deviation. The former is related to the casing boundary layer thickness in the inlet duct, which is sensitive to the inlet turbulence boundary condition; the latter is related to the boundary layer thickness on the blade surface, which is generally related to the turbulence model form. To help understand the behavior of the SA model, a detailed examination of the eddy viscosity fields is needed in the future. The deficiency of the SST model at the PE condition exists near both endwalls. The overprediction of pressure ratio near the casing originates from the rotor, which can be caused by the deficiency of the SST model in predicting tip leakage flow or the near-tip geometric uncertainty. The overprediction of pressure ratio near the hub originates from the stator, which can be caused by stator hub leakage flow through the secondary passage or the deficiency of the SST model in predicting the corner separation in the stator. Further analysis in the flow field is needed for future research.

### 3.4 Effect of Rotor-Stator Interface Model

In this section, the mixing plane rotor-stator interface model (typically used in steady-state simulations) is compared against the sliding plane model used in unsteady simulations. The investigation is performed using the Ansys CFX solver, the Medium grid topology and the SST turbulence model. To reduce the required computational resources, the number of stator blades has been changed from 29 to 32 so that a periodic sector of one rotor blade and two stator blades (1R2S) can be used for the sliding plane model. The time step used in the unsteady simulation corresponds to $\Delta t \cdot BPF = 50$, and the second-order backward Euler scheme is used for time stepping. The simulation has been performed over 10 rotor revolutions to reach a statistically steady-state, and the data obtained in the last rotor revolution is used for time averaging.

The predicted characteristic values from both the 1R1S domain and the 1R2S domain are compared in Table 3. Firstly, the steady-state performances before and after changing the number of stator blades are examined. It shows that steady-
Table 3: Characteristic values of experimental results and numerical results using different rotor-stator interface models; results obtained by using the CFX solver, the Medium grid and the SST turbulence model.

<table>
<thead>
<tr>
<th>Case</th>
<th>Peak efficiency</th>
<th>m (kg/s)</th>
<th>π∗ (-)</th>
<th>η∗ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td></td>
<td>16.00</td>
<td>1.49</td>
<td>86.9</td>
</tr>
<tr>
<td>1R1S, steady</td>
<td></td>
<td>16.01</td>
<td>1.51</td>
<td>88.5</td>
</tr>
<tr>
<td>1R2S, steady</td>
<td></td>
<td>16.02</td>
<td>1.51</td>
<td>88.4</td>
</tr>
<tr>
<td>1R2S, unsteady</td>
<td></td>
<td>15.98</td>
<td>1.50</td>
<td>87.7</td>
</tr>
</tbody>
</table>

Figure 12: Effect of rotor-stator interface model on predicting the circumferentially mass-averaged radial profiles at the PE condition and stage exit (ME30); results obtained by using the Ansys CFX solver, the Medium grid and the SST turbulence model.

4 DISCUSSIONS ON GEOMETRIC UNCERTAINTY AND ERROR

The above analyses are based on the assumption that the geometry used in the simulation is exactly the same as the one used in the experiment. In this section, the effects of some real geometry features are discussed, including the rotor running tip gap shape, the stator hub cavity and the stage throat area. For brevity, only radial profiles of total pressure ratio at the PE condition are presented.

4.1 Effect of Rotor Running Tip Gap Shape

In the experiment, the rotor running tip gap shape has been measured by a capacitive blade tip timing system, and the average values and chordwise distributions of the tip gap size are provided in Klausmann et al. [6]. Based on the measurement, four cases with different rotor tip gap shapes are investigated, as summarized in Table 4. For cases with a uniform tip gap, the chordwise-averaged value of the measured tip gap size is adopted; for cases with a non-uniform tip gap, the chordwise distribution of the measured tip gap size is used. The tip gap size of all the four cases is slightly higher than the datum size used in Sec. 3, which potentially explains the overprediction of the rotor exit pressure ratio above 80% span. For consistency, all the cases are meshed using the same topology as the Fine grid.

Table 4: Case setups with different rotor tip gap shape.

<table>
<thead>
<tr>
<th>Case</th>
<th>Tip gap size (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>13% cₜ</td>
</tr>
<tr>
<td>Datum</td>
<td>0.75</td>
</tr>
<tr>
<td>TC, PE, uniform</td>
<td>0.77</td>
</tr>
<tr>
<td>TC, NS, uniform</td>
<td>0.87</td>
</tr>
<tr>
<td>TC, PE, non-uniform</td>
<td>0.72</td>
</tr>
<tr>
<td>TC, NS, non-uniform</td>
<td>0.83</td>
</tr>
</tbody>
</table>

The resulting difference in the stage total pressure ratio and the stage efficiency is within 0.01 and 0.1% at both the PE and the NS conditions. Their performance maps are not shown for brevity. For the radial profiles presented in Fig. 13, the uncertainty of tip gap shape does not show a pronounced effect either. The variation of tip gap size of the four investigated cases is between 0.8% and 0.9% of blade tip chord. According to previous researches [19, 20], such a small change in tip gap size has negligible effects on the compressor performance, which is consistent with the findings of the current work.

4.2 Effect of Stator Hub Cavity

In the test rig, a secondary flow passage that connects the stator inlet hub to the stator outlet hub exists. A ring seal is installed and the leakage flow rate is deemed small, but its exact value remains uncertain. This stator state results obtained from both domains are almost the same, indicating the scaling of the stator geometry barely changes the flow physics at the PE condition. Based on the converged steady-state solution of the 1R2S, the unsteady simulation is performed. The unsteady results show closer agreement with the experiment: the difference between the 1R2S unsteady simulation and the measurement are only 0.01 and 0.8% in terms of the pressure ratio and efficiency.

The radial profiles obtained from the steady and unsteady simulations are compared in Fig. 12. Again, the steady-state results from the 1R2S domain are almost the same as those of 1R1S. No significant difference between the two interface models is observed except that the sliding plane result shows better agreement in the near-tip temperature ratio. The overprediction of the stage pressure ratio below 20% span persists. These results indicate that the stator flow field downstream of the rotor tip leakage flow is most sensitive to the use of the mixing plane model.
Figure 13: Effect of tip gap size on predicting the circumferentially mass-averaged radial profiles of total pressure ratio at the PE condition: (a) rotor exit (ME21) and (b) stage exit (ME30); results obtained by using the Numeca solver, the Fine grid, the SST turbulence model and the mixing plane rotor-stator interface model.

Figure 14: Illustration of stator hub cavity.

hub cavity flow potentially explains the overprediction of the stage exit pressure ratio below 20% span. In this section, an initial simulation is conducted considering part of the cavity geometry between the rotor and the stator, as illustrated in Fig. 14. Compared to the experiment, the cavity size is exaggerated so that its effect can be captured by CFD more easily; the leakage flow from the stator outlet to the stator inlet is not yet considered. The cases with and without the stator hub cavity use the same grid topology as the Fine grid, and the location of the rotor-stator interface of both cases are adjusted slightly according to the location of the cavity.

The resulting difference in the stage total pressure ratio and the stage efficiency is within 0.01 and 0.1% at both the PE and the NS conditions. No pronounced differences between the two cases are observed in the radial profile plots of Fig. 15 either. Such results suggest that the cavity domain with zero leakage flow has a negligible effect on the main flow; rather, a non-zero leakage flow can potentially lead to an evident difference in the main flow. According to previous researches [21,22], a leakage mass flow rate as small as 0.3% of the main flow rate can lead to a 1% to 2% reduction in total pressure ratio below the 20% span. For future research, simulations considering the full secondary flow passage are needed.

4.3 Effect of Stage Throat Area Change

Due to the manufacturing tolerance and the hot-geometry effect, the compressor geometry tested in the experiment will have a slightly different throat area compared to the simulations. When comparing the flow field at the same mass flow rate, the slight difference in the throat area can lead to an evident change in the incoming flow incidence, hence the evident difference in compressor pressure ratio and work input. To avoid such effect of the throat area change, a fair comparison between the experiment and CFD can be made at the same normalized mass flow rate \( m/m_c \). In this section, three operating points near the experimental PE condition are compared with each other, whose mass flow rates are 15.86 kg/s, 16.00 kg/s and 16.09 kg/s respectively. The point with 16.09 kg/s mass flow rate, in particular, has almost the same \( m/m_c \) value as the experiment.

The radial pressure ratio profiles are compared in Fig. 16. Results show that the pressure ratio profile translates towards the low-pressure direction when the mass flow rate is increased. When comparing the experiment and CFD at the same \( m/m_c \) value (i.e., 16.09 kg/s for CFD), reasonably good agreement is achieved between 30% and 70% span, but the overprediction of pressure ratio above the 80% span and below the 20% span persists. This suggests that the inconsistency near both endwalls comes from other uncertainties and errors, e.g., deficiency of turbulence modeling.

5 CONCLUSIONS AND FUTURE WORKS

A comprehensive validation and verification study of turbomachinery RANS flow solvers is conducted for the transonic axial compressor TUDa-GLR-OpenStage. Two commercial solvers namely Ansys CFX and Numeca FineTurbo are adopted to provide the benchmark solutions. Based on these solvers, five sets of grids, three advection schemes, two turbulence models and two rotor-stator interface
models are investigated to quantify their effects on predicting the performance and the flow field of the compressor stage. Several conclusions are drawn as follows.

When using the grid density and the advection scheme with a small discretization error, none of the investigated turbulence models (SA and SST) and the rotor-stator interface models (mixing plane and sliding plane) can perfectly capture the compressor characteristics at the PE condition of the design speed. The differences mainly come from the near-tip flow of the rotor and the near-hub flow of the stator. The former is most likely related to the turbulence modeling deficiency in predicting tip leakage flow and the unknown geometric uncertainty near the tip. The latter is most likely related to the turbulence modeling deficiency in predicting hub corner separation and the neglect of the stator hub leakage flow. To improve the prediction accuracy further, future research should focus on the evaluation of turbulence models (e.g., variants of SA and SST, more sophisticated models, etc.) and the real-geometry effects (e.g., stator hub leakage flow).

During the analysis of the grid density effect in this work, a method to estimate the discretization error induced by the grid was presented. Based on this method and the investigated compressor, it is found that a grid with about 1.8 to 3.4 million grid points per blade passage achieves grid-independent results in the compressor overall performance and the radial profiles; a grid with about 0.5 to 1.1 million grid points per blade passage also shows sufficiently small discretization error in the compressor overall performance. The former grid is recommended for in-depth flow field analysis such as evaluation studies of turbulence models; the latter grid is recommended for optimization of compressor geometry and full-annulus unsteady simulations, where the computation cost is equally important as the accuracy. The grids generated in this paper are open-accessed to the public. The recommended number of grid points can be used as a reference for other similar machines. The presented method to estimate discretization error is applicable to other cases.

When using a refined grid, the choice among different second-order accurate advection schemes should in theory have limited effects on predicting the compressor flow fields. This is validated when comparing the central scheme with the upwind scheme in Numeca, but an evident difference between the high resolution scheme of CFX and the two schemes of Numeca is observed. Such a difference indicates inconsistent numerical settings (e.g., with/without wall function, different versions of the SST model) between the two solvers. For future research, results from more RANS flow solvers are needed to help understand the effects of these numerical settings.

Comparing the SA model with the SST model, there is no consistent evidence in the investigated compressor showing one model is better than the other, and the choice between the two models remains an open option for the users. In general, the SST model predicts higher pressure ratio and efficiency characteristics but a narrower stall margin than the SA model. It should be noted that the investigation of turbulence models in this work is based only on the Numeca solver, and more RANS flow solvers using the SA and the SST models are needed to verify the results. Also, in-depth analyses in the main flow field and the eddy viscosity field are required to help understand the difference between the two turbulence models.

Comparing the mixing plane model with the sliding plane model, results from the latter model show a closer agreement with the experiment at the PE condition, especially in the near-tip flow field of the stator downstream of the rotor tip leakage flow. However, the difference in the overall characteristics is only 0.01 in pressure ratio and 0.7% in efficiency at the PE condition. Thus, the use of the mixing plane model with a single-passage flow domain is cost-effective for predicting the performance characteristics away from stall. For future research, the effect of the rotor-stator interface model at the NS condition needs to be investigated, and more in-depth flow field analyses are required to help understand the difference between the two models.

ACKNOWLEDGMENT

Mingmin Zhu and Jinfang Teng acknowledge the supports of Natural Science Foundation of China (No. 51906140) and National Science and Technology Major Project (2017-II-0004-0017). Xiao He acknowledges the Imperial College President PhD Scholarship. The TUDa-GLR-OpenStage measurement campaign is supported by the EU-H2020 funded ARiAS project (No. 769346).

DATA AVAILABILITY STATEMENT

The grids and the major CFD results presented in this paper will be uploaded to the GPPS workshop website.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>chord length (mm)</td>
</tr>
<tr>
<td>M</td>
<td>Mach number (-)</td>
</tr>
<tr>
<td>m</td>
<td>mass flow rate (kg/s)</td>
</tr>
<tr>
<td>N</td>
<td>rotor design speed (RPM)</td>
</tr>
</tbody>
</table>

LATINS AND GREEKS
\( p_t \)  
Total pressure (Pa)

\( r \)  
Radius (mm)

\( T_t \)  
Total temperature (K)

\( Z \)  
Number of blades (-)

\( \alpha \)  
Absolute flow angle (from axial) (deg)

\( \Delta \)  
Nominal grid spacing (-)

\( \eta^* \)  
Isentropic efficiency (-)

\( \pi^* \)  
Pressure ratio (-)

\( \phi \)  
Stage flow coefficient (-)

\( \psi \)  
Stage loading coefficient (-)

\( \tau \)  
tip gap size (mm)

\( \tau^* \)  
Temperature ratio (-)

Subscripts

\( h \)  
hub

\( m \)  
mid-span

\( t \)  
tip

Acronyms

1R1S  
one rotor blade and one stator blade

1R2S  
one rotor blade and two stator blades

EXP  
experiment

NS  
near stall

OGV  
outlet guide vane

RANS  
Reynolds-averaged Navier-Stokes

PE  
peak efficiency

SA  
Spalart-Allmaras

SIM  
simulation

SST  
shear stress transport

TC  
tip clearance

V&V  
validation and verification

REFERENCES


