A Comparison of Code Leo Predictions to Experimental Data for the Radiver Centrifugal Compressor Impeller

William Humber  
AeroDynamic Solutions, Inc.

Ron-Ho Ni  
AeroDynamic Solutions, Inc.

ABSTRACT

This is the first of a series of studies comparing predictions from the ADS solver Code Leo to experimental results obtained for the Radiver centrifugal compressor test case. In this study, the impeller row was analyzed using a structured OHH mesh generated by Code Wand. A mesh independence study was first conducted. Meshes at three granularity levels were analyzed at three operating points and found to differ by <0.1% in predicted mass flow and pressure. Based on this result, a mesh consisting of approximately 326,000 elements was selected and used to construct a speedline for the impeller row at 80% design speed. Initial pressure predictions for the impeller row were found to be in excellent agreement with experimental data both in trend and value, and within the experimental uncertainty range. A slight but consistent overprediction of pressure (~1%) was noted in the results, and further investigation revealed that the sampling resolution on the pressure profile used to calculate average pressure could materially influence results. When the number of spanwise points used to from the pressure profile to calculate average pressure was reduced to a level commensurate with the calculation made on the experimental data, pressure predictions were shown to change by as much as 3% for the Radiver impeller case. This demonstrates that the resulting average of a profile is sensitive to the sampling resolution and care should be taken to keep interpretation of the CFD consistent with the experimental data. With the coarsened averaging scheme, the impeller total pressure ratio was shown to change from a 1% overprediction to a 2% underprediction, which is still within experimental uncertainty for most points. A follow on study will focus on overall stage performance.

INTRODUCTION

As part of an ongoing series to assess the predictive capability of the ADS solver Code Leo for centrifugal compressor aero design, a simulation was conducted and compared against the Radiver test case, for which experimental data is publicly available. While both steady and unsteady data is available from the experiment, only steady data is compared. This initial paper focuses on the impeller row; a subsequent paper will address overall stage performance.

The paper begins with a quick introduction to the solver Code Leo and structured mesh generator Code Wand. The Radiver test case will then be described, followed a discussion of the CFD setup and analysis of the results.

CODE LEO AND CODE WAND

Code Leo

Code Leo is a powerful generalized solver that delivers both accurate and fast flow simulations for general flow configurations. Featuring a cell-vertex finite volume procedure for efficient and accurate approximation and an advanced multi-grid residual propagation scheme for rapid convergence, Code Leo is designed to support:

- Low speed, transonic and supersonic flow regimes up to Mach 3.5
- Steady and time accurate flow simulations
- Unstructured mesh to cover complex flow configurations of structured and unstructured meshes
- Multi-block code for flexibility to patch together complex geometries
- Variable specific gas properties for air or combustion products
- Two-equation turbulence modeling using Wilcox's k-ω model with transition and surface roughness, and choice of wall function or integration
- Multi-specie tracking
- Fast and efficient large-scale flow simulation using standard MPI routines from Argonne National Labs

Code Leo provides additional advanced capabilities for turbomachinery applications:

- Non-reflective upstream and downstream boundary conditions
• Radial equilibrium exit pressure condition
• Cavity and actuator disc models
• Inlet boundary profile and exit static pressure profile
• Mixing planes for steady multi-stage flow simulation
• Sliding mesh boundaries for time accurate simulation of rotor-stator interactions
• Ability to introduce film cooling air 2D simulations with height ratio for design use

**Code Wand**

Designed for use with the ADS flow solver Leo, Code Wand is a robust mesh generator optimized for turbomachinery blading applications. Utilizing an advanced OHH mesh topology and novel tip clearance meshing techniques, Code Wand generates high quality airfoil meshes that concentrate nodes where they’re needed most for compressor and turbine analysis. Key capabilities include:

• Advanced OHH-type mesh to optimize capture of flow phenomena near leading and trailing edges
• Proprietary clearance meshing to improve efficiency and performance of tip clearance and stator end gap meshes
• Choice of wall integration or wall function options for meshing close to airfoil walls
• Mesh smoothing (Laplace or Poisson solvers)
• Square trailing edge support
• Constant radius cut trailing edge support for centrifugal compressors
• Leading and trailing edge enhancement
• Fine grained mesh controls
• Automatic flow field initialization
• Support for wide array of Cartesian & cylindrical coordinate-based input geometry formats
• Airfoils supported include compressor and turbine airfoils, multi-airfoils such as radial impellers with splitters, and multi-stage airfoil rows.

**RADIVER CASE DESCRIPTION**

As depicted in Figure 1, the Radiver experimental setup consists of a one stage radial compressor with a variable geometry diffuser downstream of the impeller and finally a volute downstream of the diffuser. The compressor was run as a closed loop with control over the inlet pressure and temperature. The diffuser section allowed independent adjustments of the vane metal angle and the radial gap between the impeller and diffuser, although for this study only one diffuser configuration was explored.

**Experimental Measurements**

Experimental data was taken at three separate locations as shown in Figure 2. Station 2M was positioned directly downstream of the impeller trailing edge and consists of pressure data taken by pitot-static probes. Care was taken to make the instruments as small as possible to limit the effect of blockage, but a reduction in flow of 1.3% [2] was seen relative to the non-instrumented rig with the introduction of the probes.

Station 7M was taken at the diffuser trailing edge and provides both pressure data and flow angles taken by a cobra probe.

Station 8M was taken at the exit of the diffuser and provides both pressure and temperature data through a combination of cylinder and temperature probes.

Probe data was estimated to be accurate within 2% [1].

![Figure 2. Measurement stations in the radial compressor](image-url)
CFD Setup

Generation of the airfoil coordinates and endwall definition was done using geometry data provided in the publicly available data package. To support this study and future studies, meshes were created using CodeWand for both the impeller and the diffuser, but not for the volute or other downstream sections.

The diffuser geometry of 16.5º and a radial gap of R4/R2 = 1.04 were used for all tests. Additionally, all tests and data were run at 80% of design speed due to concerns over the effects of the pitot probes in the 2M plane on compressor stability.

Boundary Conditions

Inlet conditions were set to Pt = 8.67 psia and 531.67 R to match experimental conditions, with flow entering axially. Rotational speed was set to 28,160 RPM. Adiabatic walls were assumed in both the impeller and diffuser sections. Additionally, tip clearance values were set to vary from 0.0266” at the leading edge of the impeller to 0.0184” at the trailing edge of the impeller.

Mesh Study

The mesh, as shown in Figure 3, consists of an OHH-type structured multiblock mesh with a special tip clearance mesh block in the impeller. A mesh study was conducted to assess effect of mesh granularity on flow solution by creating speedlines using three different mesh levels while holding aerodynamic conditions constant. As seen in Figure 4, results showed very slight differences, less than 0.1%, in mass flow and total pressure ratio between the different mesh levels indicating that mesh independence is obtained even with the coarsest mesh granularity tested. Based on the results of the mesh study, mesh level five was used to create a speedline to compare against data.

Execution

The automated speedline generation capability available in the ADS Workbench was then used to construct the compressor speedlines. Steady flow simulations using a k-ω turbulence model were used for all points. Each point was run for 8,000 iterations, even if convergence was achieved earlier. If it was determined that convergence was not achieved by 8,000 iterations, an additional 8,000 iterations were run. Mesh count and turnaround speed are presented in Table 1. Once again, note that the mesh counts and iteration speed are for both the impeller and diffuser. Though we will only focus on the impeller in the study, the same setup will be used to assess full stage performance in a follow on paper.

<table>
<thead>
<tr>
<th>Mesh Level</th>
<th>Total Nodes</th>
<th>Total Elements</th>
<th>Iterations Per Hour</th>
</tr>
</thead>
<tbody>
<tr>
<td>LVL5</td>
<td>368,251</td>
<td>326,912</td>
<td>5,162</td>
</tr>
<tr>
<td>LVL6</td>
<td>592,179</td>
<td>534,528</td>
<td>3,171</td>
</tr>
<tr>
<td>LVL7</td>
<td>937,339</td>
<td>857,344</td>
<td>2,099</td>
</tr>
</tbody>
</table>

Table 1. Mesh counts and iteration speed
RESULTS

The results of the Radiver impeller simulation are presented in Figure 5 below. As can be seen from the figure, the total pressure ratio predictions at the exit of the impeller row agree well with measured results, both in terms of trend and magnitude. The predictions also fall within the range of experimental uncertainty, although it is interesting to note that the predictions appear to be consistently overpredicted by ~1%.

Figure 5. Impeller pressure ratio vs. corrected flow

To further assess the predictive accuracy of Code Leo, pressure profile predictions were compared to measured results at the exit of the impeller. These results are presented in Figure 6. Again, the impeller pressure prediction is shown to be in very good agreement with the experimental data at the 2M location, though slightly underpredicted between 70%-90% span.

Figure 6. Experimental data and CFD predictions for rotor exit profile at P2

Given the paucity of spanwise points used to calculate the average pressure from the experimental results and the relative abundance of CFD spanwise points used to perform the same calculation, an effort was undertaken to determine whether or not the resolution of the averaging process had a material effect on the averaged value.

Figure 7. Predicted Pt profiles at the rotor-diffuser interface location

Figure 7Figure 1 shows the profile at the interface between the impeller and diffuser domains at two resolution levels—one at the normal sampling resolution level employed by Code Leo and one at a resolution level commensurate with the experimental measurements. At this location, the CFD pressure profile is fuller at both endwalls than the low resolution averaging scheme employed with the experimental results can capture. The result is a 3% difference in pressure prediction, as seen in Figure 8. This demonstrates that the resulting average of a profile is sensitive to the sampling resolution and care should be taken to keep interpretation of the CFD consistent with the experimental data. With the coarsened averaging scheme, the impeller total pressure ratio changes from overpredicting by 1% to underpredicting by 2%, which is still within experimental uncertainty for most points.

Figure 8. Effect of the high and low resolution averaging methods at the rotor-diffuser interface location
CONCLUSION

This is the first of a series of studies comparing predictions from the ADS solver *Code Leo* experimental results obtained for the Radiver centrifugal compressor test case. In this study, the impeller row was analyzed using a structured OHH mesh generated by Code Wand. A mesh independence study was first conducted. Meshes at three granularity levels were analyzed at three operating points and found to differ by <0.1% in predicted mass flow and pressure. Based on this result, a mesh consisting of approximately 326,000 elements was selected and used to construct a speedline for the impeller row at 80% design speed. Initial predictions for the impeller row were found to be in excellent agreement with experimental data both in trend and value, and within the experimental uncertainty range. A slight but consistent overprediction of pressure (~1%) was noted in the results, and further investigation revealed that the sampling resolution on the pressure profile used to calculate average pressure could materially influence results. When the number of spanwise points used to form the pressure profile to calculate average pressure was reduced to a level commensurate with the calculation made on the experimental data, pressure predictions were shown to change by as much as 3% for the Radiver impeller case. This demonstrates that the resulting average of a profile is sensitive to the sampling resolution and care should be taken to keep interpretation of the CFD consistent with the experimental data. With the coarsened averaging scheme, the impeller total pressure ratio was shown to change from a 1% overprediction to a 2% underprediction, which is still within experimental uncertainty for most points. A follow on study will focus on overall stage performance.

ACKNOWLEDGEMENTS

The measurements of the test case “Radiver” were carried out at the Institute of Jet Propulsion and Turbomachinery at RWTH Aachen, Germany. Part of the investigation was funded by the Deutsche Forschungsgemeinschaft (DFG).

REFERENCES
