

NASA/CR—2012-217438



Unsteady Aero Computation of a 1 1/2 Stage Large Scale Rotating Turbine

Wai-Ming To
University of Toledo, Toledo, Ohio

NASA STI Program . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) program plays a key part in helping NASA maintain this important role.

The NASA STI Program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI program provides access to the NASA Aeronautics and Space Database and its public interface, the NASA Technical Reports Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services also include creating custom thesauri, building customized databases, organizing and publishing research results.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA STI Help Desk at 443-757-5803
- Telephone the NASA STI Help Desk at 443-757-5802
- Write to:
NASA Center for AeroSpace Information (CASI)
7115 Standard Drive
Hanover, MD 21076-1320

NASA/CR—2012-217438



Unsteady Aero Computation of a 1 1/2 Stage Large Scale Rotating Turbine

Wai-Ming To
University of Toledo, Toledo, Ohio

Prepared under Contract NNC06BA07B, Task Number NNC10E420T

National Aeronautics and
Space Administration

Glenn Research Center
Cleveland, Ohio 44135

May 2012

This report contains preliminary findings,
subject to revision as analysis proceeds.

Trade names and trademarks are used in this report for identification
only. Their usage does not constitute an official endorsement,
either expressed or implied, by the National Aeronautics and
Space Administration.

This work was sponsored by the Fundamental Aeronautics Program
at the NASA Glenn Research Center.

Level of Review: This material has been technically reviewed by expert reviewer(s).

Available from

NASA Center for Aerospace Information
7115 Standard Drive
Hanover, MD 21076-1320

National Technical Information Service
5301 Shawnee Road
Alexandria, VA 22312

Available electronically at <http://www.sti.nasa.gov>

Unsteady Aero Computation of a 1½ Stage Large Scale Rotating Turbine

Wai-Ming To¹
University of Toledo
Toledo, Ohio 43606

Abstract

This report is the documentation of the work performed for the Subsonic Rotary Wing Project under the NASA's Fundamental Aeronautics Program. It was funded through Task Number NNC10E420T under GESS-2 Contract NNC06BA07B in the period of 10/1/2010 to 8/31/2011. The objective of the task is to provide support for the development of variable speed power turbine technology through application of computational fluid dynamics analyses. This includes work elements in mesh generation, multistage URANS simulations, and post-processing of the simulation results for comparison with the experimental data. The unsteady CFD calculations were performed with the TURBO code running in multistage single passage (phase lag) mode. Meshes for the blade rows were generated with the NASA developed TCGRID code. The CFD performance is assessed and improvements are recommended for future research in this area. For that, the United Technologies Research Center's 1½ stage Large Scale Rotating Turbine was selected to be the candidate engine configuration for this computational effort because of the completeness and availability of the data.

¹ Contact information: waiming.to@nasa.gov (216-433-5937)

Introduction

Unsteady multistage CFD code is a common tool for analyzing turbomachinery flows such as those encountered in the variable speed power turbine. Among the many choices of such flow codes, TURBO is chosen to be evaluated in this work. Since the code is largely employed in the analysis of compression systems, it is desirable to first focus on turbine aerodynamics before attempting for the more difficult heat transfer problem. Its performance in turbine application will be assessed and possible improvements are identified for future CFD simulation and TURBO code development. Also considered in this report is to provide some kind of documentation for best practices in performing turbine CFD simulations, particularly the procedure for starting from scratch with TURBO.

A relevant multistage turbine configuration for the computation is the UTRC Large Scale Rotating Rig (LSRR). Detail geometry information and aerodynamic data are available publicly from the UTRC reports [1,2]. A schematic of the rig copied from [1] is reproduced here in Figure 1. It is configured as a 1½ stage turbine with blade counts of 22-28-28. Different configurations were reported and tested in this rig, but the particular configuration for the current CFD investigation is the 1½ stage setup with the 50% nominal Stator1/Rotor gap because of the availability of the aero data from the experiment. The rotor tip has a clearance of 1.5% of the span. The flow parameters chosen for the CFD is the 0.78 flow coefficient and 0.5% inlet turbulence intensity. The inlet Mach number is approximately 0.07 at this flow coefficient with a shaft speed of 410 rpm.

Coordinates for each blade were published in the reports [1,2] for 3 spanwise positions (2%, 50%, and 98%). This information is used to generate computational meshes for the blade passages using the mesh generation code TCGRID [3]. H-mesh topology is employed throughout the work. The computational domain begins at about 1 axial cord length of Stator 1 from its leading edge and terminates at approximately 1 axial cord length of Stator 2 from its trailing edge (see Figure 2).

The computations were performed on the NASA Glenn Research Center HX Compute Cluster. Each passage is divided into 16 blocks. A total of 48 processors were requested for each run in these simulations with each block assigned to a processor.

Isolated Stator 1 calculations were also performed mostly for gauging the adequacy of grid quality and numerical input parameters. Its computational exit boundary is extended to about 1 axial cord downstream instead of ending at the Stator 1/Rotor interface.

In the next section, details on generating the meshes and preprocessing of the data for TURBO are described. The Computation Details section provides additional information about setting up the calculations, computing requirements and procedures of the simulations. Following that, the results from these calculations are compared to the test data reported in [2]. Finally, concluding remarks are given in the last section.

An accompanying DVD is also available upon request an approval by NASA. It contains all the files necessary for restarting the calculations for the isolated Stator 1 and the 1½ stage. In addition, the TURBO source code, the airfoil coordinates and their meshes are also available from the same DVD.

Mesh Generation and Preprocessing

Blade profiles for each of the blade are given in [1,2] as coordinates at 3 different radial positions. They are digitally reproduced here and plotted in Figure 3 to Figure 5. This data are provided in the DVD under the AirfoilCoordinates folder. Since the resolution around the leading and trailing edges from the raw data is insufficient for making acceptable meshes, these regions are fitted with circles to produce a smoother definition of the blade contour. Radii of these circles and coordinates of their centers are shown in the figures. The blade contours are given in inches at the 2%, 50%, and 98% span corresponding to the hub, midspan, and tip profile denoted in the figures.

This information is utilized in the input to TCGRID for generating the computational mesh for each blade passage. The radial grid lines at the blade row interface have to be matched up from both sides. It is an implicit assumption made in the TURBO code for interpolation across the interface. Input file format and input parameters for running TCGRID are well documented in [3]. The input files for generating the meshes are given in the DVD under the Mesh folder as `tcgrid.S1.input`, `tcgrid.R1.input`, and `tcgrid.S2.input` for Stator1, Rotor1, and Stator2 respectively. H-mesh topology is employed for the blade passages. The rotor mesh has 5 radial points in the tip clearance region. The mesh output from TCGRID is written in PLOT3D single grid format in Little Endian which is then converted to the multi grid Big Endian format in order to be read in by GUMBO for further mesh preprocessing. Care on the distance of the first grid point from the wall should be exercised as the turbulence model used in TURBO is of a high Reynolds number form of k - ϵ model with wall damping function. The assessment of y^+ is done with a series of preliminary runs for an isolated Stator 1 with different near wall mesh density. While the range of y^+

for the turbulence model to be valid lies between 20 and 100, a limited number of points with $y^+ < 20$ in the flow is permitted in the computation. The final revision of the meshes for the simulations are shown in Figure 2, with mesh densities of (206, 64, 61), (216, 64, 61), and (216, 64, 61) for Stator 1, Rotor, and Stator 2 respectively.

Mesh and block manipulations, block boundary condition and block connectivity settings, and blade row interface information are conveniently set up and controlled with GUMBO, a graphical user interface tool for manipulating the initial data and grid. Initial flow field, if available can also be set up with GUMBO simultaneously. However, this is only restricted to APNASA data type format. Since there is no initial flow field available, the computation will have to start from scratch with uniform flow. Files for the block connectivity, block boundary conditions, and blade row interfaces, written by GUMBO are all that is required for starting a TURBO computation. It should be remarked that these files can be generated manually without necessarily going through GUMBO. In addition, a set of input files are also needed by TURBO for setting up parameters such as inlet and exit boundary conditions, reference conditions, shaft speed, time step size and other numerical parameters. Details of the description of input parameters can be found from the link: http://www.simcenter.msstate.edu/docs/msu_turbo/. The list of files created from this pre-processing step is provided under the CASE directory in the DVD.

Initially each blade row passage is divided into 5 blocks. Although this block density resulted in a reasonable turnaround time for the isolated Stator 1 simulation, it becomes evident that larger block density is needed for stage computation in order to speed up the calculations because of the low Mach number flow. The unsteady nature of the blade row interaction has created a long numerical transient despite of a peak local Mach number of about 0.26 on the suction side of the Stator 1. Eventually, a 16 blocks/passage is adopted for the subsequent simulations. Each block is assigned to a processor and will need about 300MB of memory which is well within HX Clusters' 1GB per processor limit. For the stage computation, a total of 48 processors are required.

Computation Details

The calculations were carried out with 60 time steps per rotor passing. The solution normally converges in 10 to 15 revs which is equivalent to a wallclock time of 24 to 36 hours on the HX Clusters. For computation starting from scratch, additional time is needed for the flow field to arrive at a reasonable level. This is because of the necessity for very small time step sizes at the early stages of the calculation.

For the 22-28-28 blade count, performing a periodic sector computation of 11-14-14 blade count (half wheel) would have required 624 processors while maintaining the same computational efficiency of 16 blocks/passage. That is beyond the current HX limit. The calculations will have to be performed on a larger computer such as the NAS compute clusters which has other constraints to consider. The alternative is to reduce the number of blocks per passage by half by sacrificing the wallclock time, resulting in roughly doubling the turnaround time. Based on these considerations, it is decided to run TURBO on HX in single passage phase lag mode instead.

Adiabatic wall condition is assumed so there will be no heat transfer calculation. Uniform total condition with zero flow angles is imposed at the inflow boundary. This assumes no boundary layer present from the inlet. At the outflow boundary, radial equilibrium flow is assumed with static pressure (back pressure) imposed at the hub. Its level is to be adjusted during calculations in such a way until a desired flow coefficient is reached. It is a tedious and time consuming process involving restarting TURBO and sometimes resetting the time information of initial flow field. After running a number of trial cases, the exit static pressure is found to be somewhere between 95% and 96% of the inlet total pressure for the 1½ stage.

As mentioned earlier, the inlet turbulence intensity for this case was measured at 0.5% in the experiment. However, the turbulence kinetic energy, k , and the dissipation rate, ε , are required as the boundary conditions at the inflow boundary. While k can be reasonably approximated by 1.5 times the turbulence intensity assuming turbulence isotropy, ε is to be estimated by assuming an eddy viscosity at the inflow boundary. Trial runs with eddy viscosity values of 5, 50, and 300 show insignificant sensitivity on the midspan pressure coefficients of the blade. As a result, a value of 50 is used for the present CFD analyses.

Besides back pressure, there are instances when the time step size or the shaft speed needs to be modified in the middle of the calculation, and the same procedures for restarting the calculation will be followed. This process is not as bad as starting from scratch since the restart files or the flow solution files from some previous TURBO runs, converged or not, are usually available. Nevertheless, it will still have to undergo a numerical transient period.

Starting the computation from scratch is tedious but rather straightforward, the flow is initialized with either an uniform axial flow or throughflow. It is necessary to begin with small initial time step size or the CFL number and monitor the progress by gradually dialing up the time step size or the CFL number to the desired level. In some instances with high speed flow, it may even be necessary to lower the shaft speed in order to stabilize the numerical transient. For the current case, 500 steps per rotor passing is used initially.

A final remark on speeding up the computation by raising the inlet Mach number: it is found that the rate of convergence was not significantly improved with an inlet Mach number of about 0.15. In this case, the shaft speed has to be raised to about 900 rpm in order to maintain a similar flow coefficient. This resulted in a peak local Mach number of about 0.6 (compared to 0.26 with $M_\infty = 0.07$) on the suction side of Stator 1, which can still be considered as having negligible effect on the blade midspan pressure distribution. The exercise was carried out in hope of improving the overall wallclock time by elevating the inlet Mach number. It turns out that the combination of the numerical and computational parameters such as Newton iteration number, Jacobian matrix update frequency, Gauss-Seidel iteration number, time step size, and number of blocks per passage have a more significant impact on the computational speed, particularly the latter one.

Results

Most of the computations reported in this work converged in 10 revolutions or less based on the time averaged inlet mass flow. Once converged, time averaged flow field is obtained by averaging the time record of the unsteady field over the last 2 revs. The flow field consisting of the 5 dependent variables (q , qu , qv , qw , e) are then used to compute other state variables or flow quantities such as pressure or velocity through either the equation of state or the isentropic relations. The assumption made in this approach is essentially that the quantity derived from the time averaged dependent variables is the “same” as the time averaged of the unsteady derived quantity. The results presented here are all manipulated in this way.

Constant specific heat ratio is assumed because of the small temperature change in the flow field. Inlet total pressure and temperature are maintained at 101,325 Pascal and 294°K. Static pressure at the hub of the exit plane is adjusted until the desired flow coefficient is reached. The rotor tip clearance gap region is not gridded, but rather a clearance model is used in the calculations. In the subsequent paragraphs, results from the isolated Stator 1 calculation are presented first along with some observations when comparing to the stage Stator 1 results, and then followed by the discussion of the stage results.

A typical mass flow time history is shown in Figure 6. The time averaged inlet mass flow for both the isolated Stator 1 and the 1½ stage CFD results are plotted on the same figure for comparison. The isolated Stator 1 computation is performed with a similar mesh distribution used for Stator 1 in the stage computation, except for extending the exit plane to about 1 axial cord length downstream of the trailing edge. The exit static pressure is adjusted in a similar way as described

earlier for the stage until the resulting flow coefficient is about 0.78 which is the target flow coefficient for the stage calculation. It should be mentioned that the isolated Stator 1 adjustment is less challenging because of its steady nature and faster convergence. As seen from the figure, the stage result displays a higher level of fluctuation probably due to the unsteady nature of blade row interactions. The isolated Stator 1 result shows that the calculation converges very quickly and that the flow is steady compared to the 1½ stage result.

Figure 7 is a plot of the time averaged pressure coefficient for the isolated Stator 1. The circles represent test data from [2] for the 1½ stage experiment and are shown here for reference. The pressure coefficient, C_p , is defined as $(P_0 - p)/\frac{1}{2}\rho_0 V_{Rm}^2$, where P_0 and ρ_0 are the averaged inlet total pressure and density at midspan respectively, p is the blade surface static pressure at midspan, and V_{Rm} is the reference speed which is assumed to be the midspan rotational speed of the rotor in the stage configuration. The data has an uncertainty in measurements of $\pm 2\%$ for C_p and $\pm 1\%$ for the flow coefficient. The result shows that the pressure coefficient is not sensitive to whether Stator 1 is configured as an isolated row or in a stage (see Figure 10a for Stator 1 pressure coefficient from the stage calculation). Results from both stator only and stage calculations show that the blade surface pressure and the wake in the midspan region is two dimensional in nature.

Figure 8 shows the time averaged total pressure contour for the isolated Stator 1 and the 1½ stage at midspan. The scale of the contour for each of the blade passages is shown next to it. The grey vertical line shown behind the blade for the stage contour in Figure 8(b) is where the circumferential survey were made in the experiment along the midspan. No flow separation is observed from the calculations. It is seen from the figure that Stator 1 wake for the stage result is thicker than that for the isolated Stator 1. Upon inspection of the mesh, it is probably due to the large numerical diffusion from the poor mesh quality in the inter-blade row gap that contributes to the wake blooming. Comparing the 2 Stator 1 meshes, the isolated Stator 1 mesh changes more gradually extending to the exit plane over a longer distance, whereas the stage Stator 1 mesh must terminate at the blade row interface resulting in large stretching and skewing.

Pressure coefficients and blade exit profiles for the stage CFD results are shown in Figure 9 to Figure 12 together with the test data from [2]. The computational results are time averaged over the last 2 revs. Midspan pressure coefficient for Stator 1, Rotor 1, and Stator 2 are shown in Figure 9a, 9b, and 9c respectively. The pressure coefficient, C_p , is defined in the same way as before. The circles from the figures represent the test data. CFD results from 2 different flow coefficients are shown in the same plot. The flow coefficients are not exactly at the experimental condition of 0.78 primarily because of the mass flow sensitivity to the back pressure adjustment while converging the unsteady flow to the target time averaged flow coefficient. Nevertheless,

one can deduce from the trend in Figure 9 that higher flow coefficient results in higher pressure coefficient. For Stator 1, that would have made the CFD results closer to the test data had a higher flow coefficient been achieved. However, that would make the pressure coefficient drifting away from the data for Rotor 1 and Stator 2, making it worst on the suction side in comparison.

Another comment from Figure 9 is that the CFD result is progressively worst for the subsequent blade rows, particularly on the suction side of the blade although the shape of the pressure coefficient profile follows the test data reasonably close. It is not clear if such discrepancy is attributed to issues related to the Rotor 1 and Stator 2 blade geometry and their meshes, or the poor Stator 1 wake upon the rotor.

Circumferential variations in flow speed, flow angle, and static pressure at midspan behind Stator 1 are shown in Figure 10. The results are presented at a distance of 17% axial cord from the its trailing edge. The solid lines represent the CFD results in the figures. The test data are shown in dots which are copied from the figures reported in [2]. Similar plots for Rotor 1 and Stator 2 are illustrated in Figure 11 and Figure 12 respectively, except that the rotor data are presented at a distance of 36% axial cord from its trailing edge and the stator at a distance of 14% axial cord from its trailing edge, shown as grey lines in Figure 8 where experimental measurements were made. Moreover, the flow speed and flow angle are plotted in absolute frame of reference for the stators and in relative frame for the rotor. The static pressures are normalized in the same way as the pressure coefficients and the flow speed is normalized by V_{Rm} as before.

The comparisons show that only qualitative agreement with the test data can be drawn from these CFD results. For example, all 3 exit profiles for the flow speed in Figure 10 to Figure 12 indicate that the wakes from their respective blades are too diffused. Nonetheless, their average values are generally not too far off from the measurements except for the rotor exit flow angle which is under-turned by 6° . The following table shows the time and circumferentially averaged exit flow angles comparison with the data. Note that the angles given in the table are measured from the vertical, whereas those in the figures are measured from the horizontal, following the same convention from the UTRC report [2].

A closer examination reveals that the worst mesh quality is usually found in the inter-blade row gap regions or near the leading and trailing edges. The H-mesh topology for large turning angle turbine blade frequently results in large mesh expansion ratio and sheared cells where numerical diffusion are substantial.

AXIAL POSITION	TEST DATA	CFD
17% cord from Stator 1 TE	67.6° (Absolute)	66.33° (Absolute)
36% cord from Rotor 1 TE	58.3° (Relative)	52.30° (Relative)
14% cord from Stator 2 TE	66.3° (Absolute)	65.82° (Absolute)

Table 1. Time and Circumferentially Averaged Flow Angles Downstream of Each Blade Row.
(Axial Position Given in Percent of Axial Cord From the Trailing Edge)

Conclusion

Blade contours for the UTRC's 1½ stage LSRR in turbine configuration were generated from the coordinates given in [1,2]. The leading and trailing edges were fitted with circles to generate better definition blade profiles. This information was then used as input to TCGRID in generating meshes for the blade passages. The unsteady calculations were performed with TURBO in single passage phase lag mode. Time averaged midspan pressure coefficient for each blade are compared with the test data. Time averaged flow speed, flow angle, and static pressure at midspan blade exit plane as described in [2] are computed and compared with the data.

The comparisons show that the CFD results are only in qualitative agreement with the test data and indicate that the poor mesh quality might be the cause for the discrepancies. This is an inherent drawback for turbine blade gridded with H-mesh. Ultimately, a mesh topology such as an O-H mixed type of mesh or a multi-zone type is more appropriate and likely to be more successful for turbine blade CFD simulations.

For this to happen, supports in mesh generation are needed for sophisticated multistage mesh topology. This necessitates further TURBO code development and support in the area of general block to block MPI communication. Although built for that in mind, TURBO has not been rigorously tested in this aspect. Therefore, additional efforts in testing and validation in supporting this feature is recommended.

References

1. Dring, R.P., Fair, M.F., Joslyn, H.D., Power, G.D., and Verdon, J.M., “The Effect of Inlet Turbulence and Rotor/Stator Interactions on the Aerodynamics and Heat Transfer of a Large-Scale Rotating Rig Turbine Model, Vol. I – Final Report”, NASA Contractor Report 4079, 1987.
2. Dring, R.P., Joslyn, H.D., and Blair, M.F., “The Effect of Inlet Turbulence and Rotor/Stator Interactions on the Aerodynamics and Heat Transfer of a Large-Scale Rotating Rig Turbine Model, Vol. IV – Aerodynamic Data Tabulation”, NASA Contractor Report 179469, 1988.
3. Chima, R.V., “TCGRID 3-D Grid Generator for Turbomachinery, User's Manual and Documentation”, Version 300, July 2003.

Figures

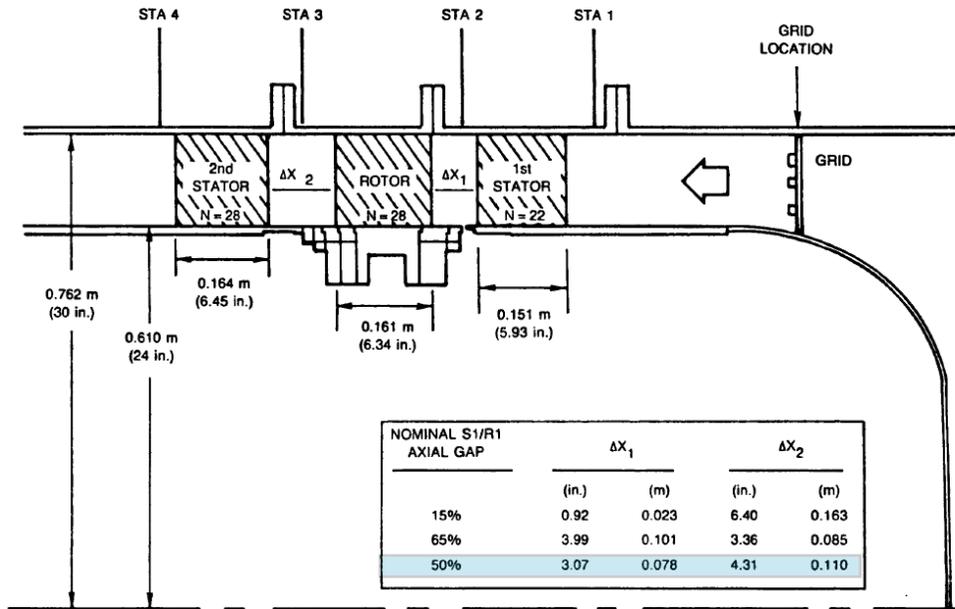
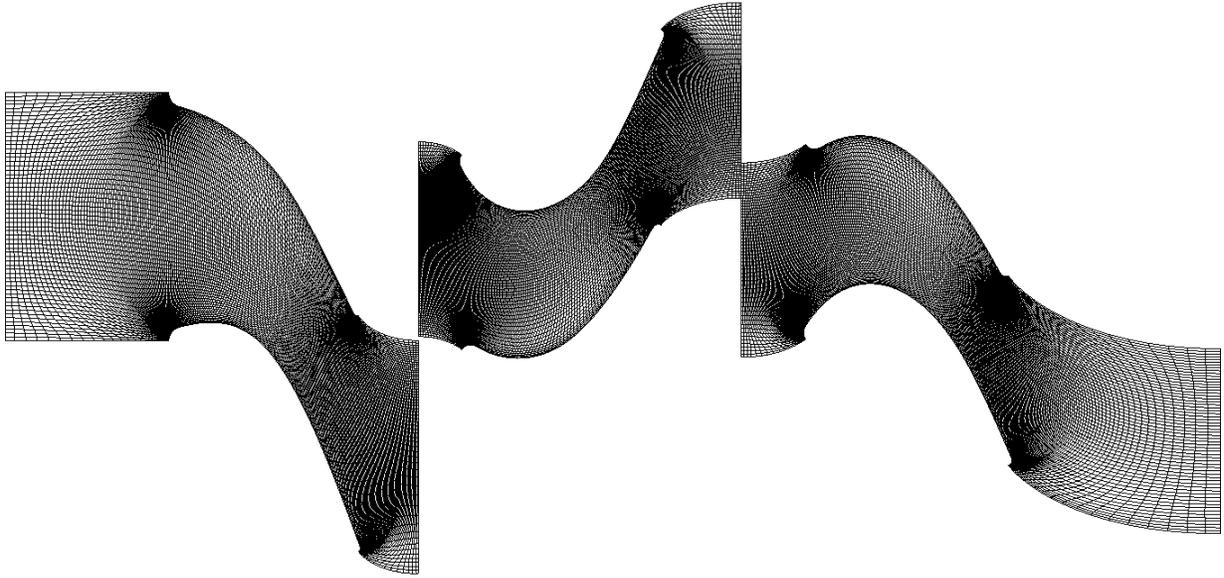
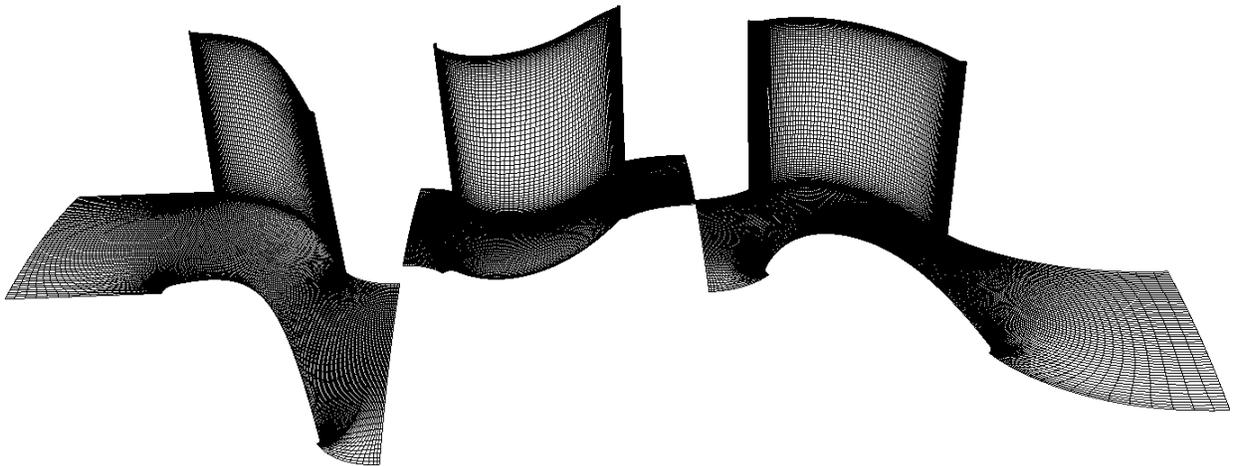


Figure 1. UTRC Large Scale Rotating Rig Schematics (diagram taken from [1] for illustration).
The Present CFD Analysis is Configured for the 50% Nominal S1/R1 Axial Gap.



(a) Midspan blade-to-blade view.



(b) 3D view.

Figure 2. H-Mesh for the 1½ Stage LSRR Computation.

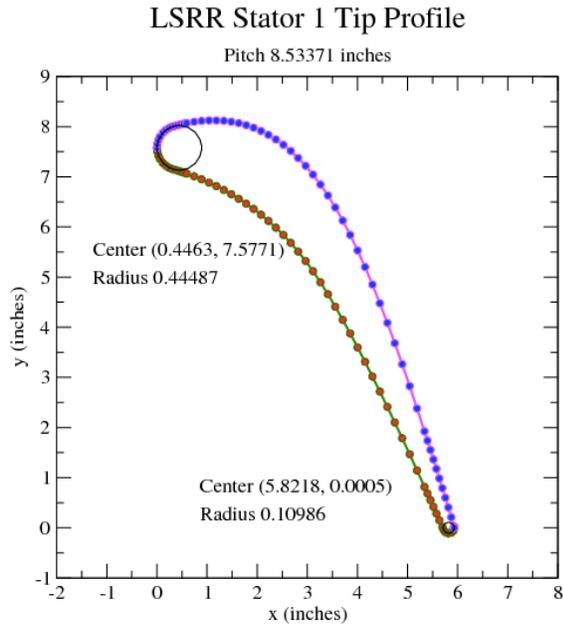
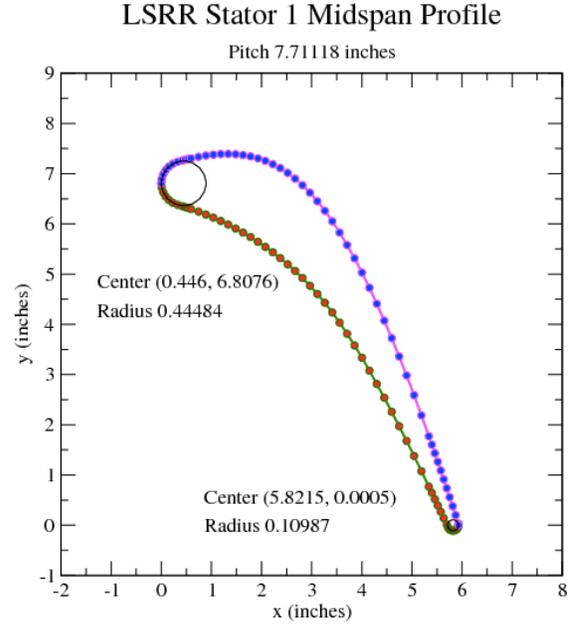
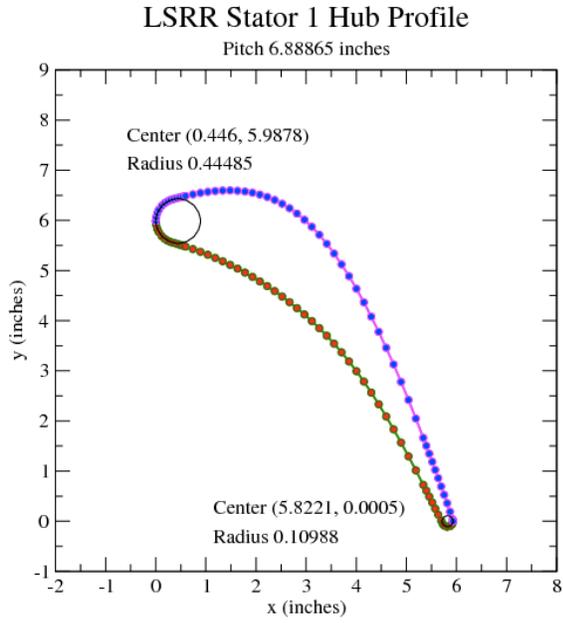


Figure 3. Stator 1 Blade Contours.

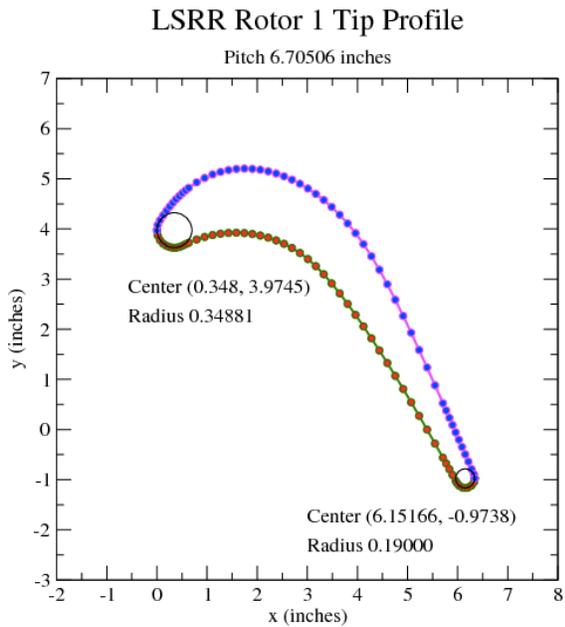
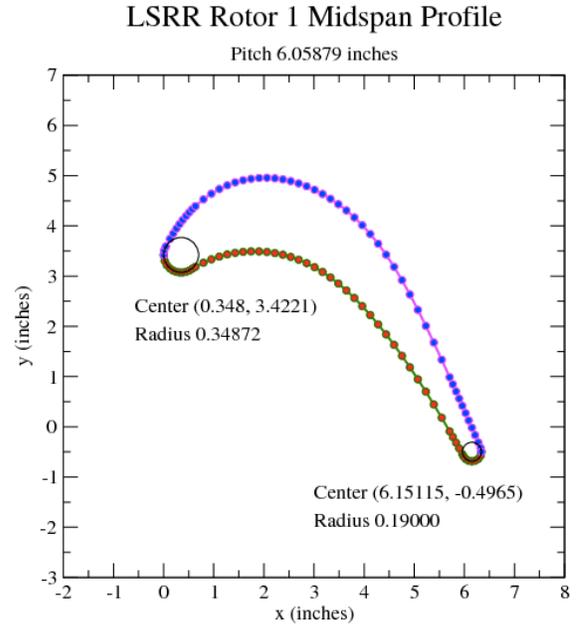
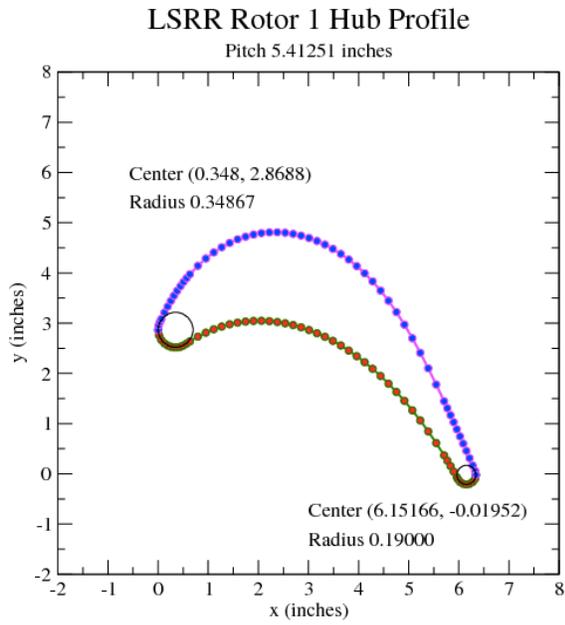


Figure 4. Rotor 1 Blade Contours.

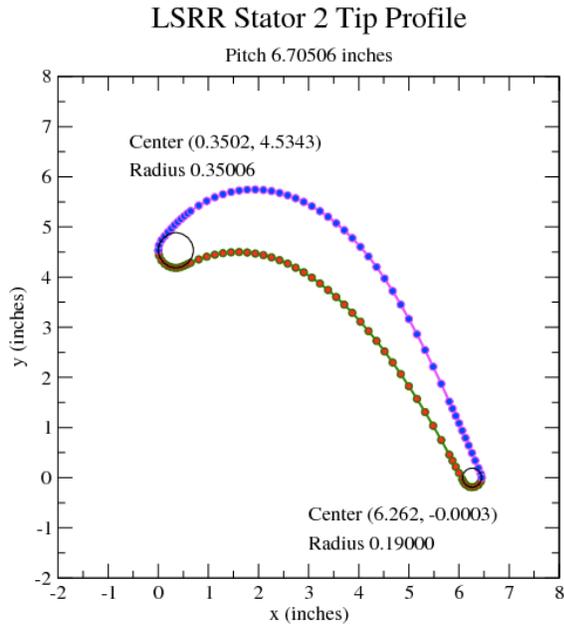
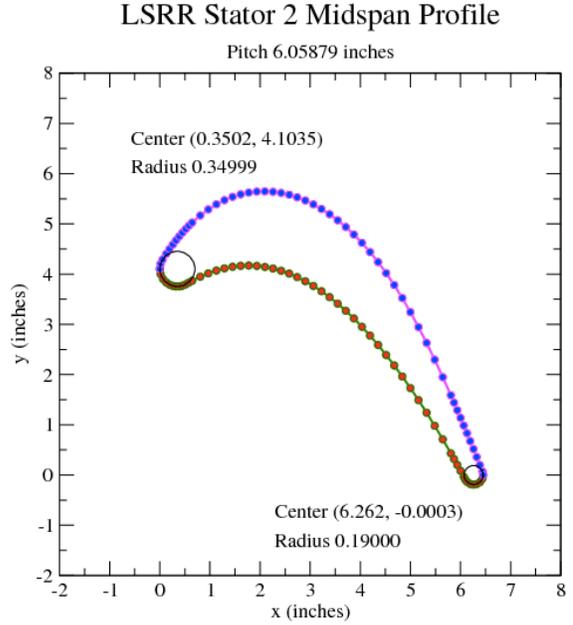
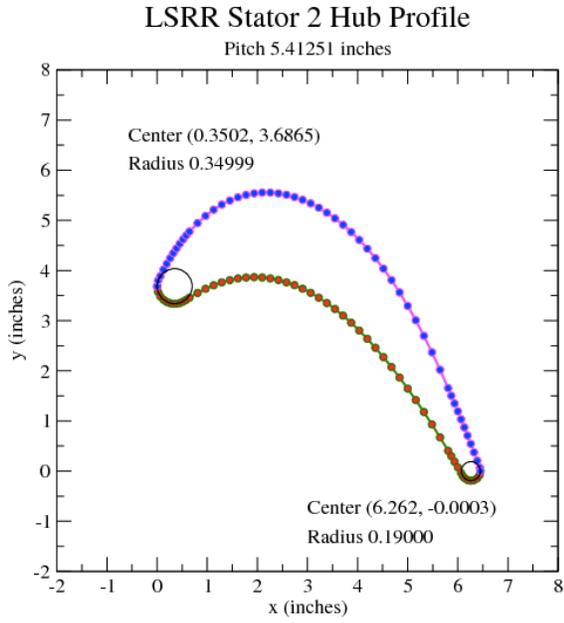


Figure 5. Stator 2 Blade Contours.

Mass Flow Convergence History

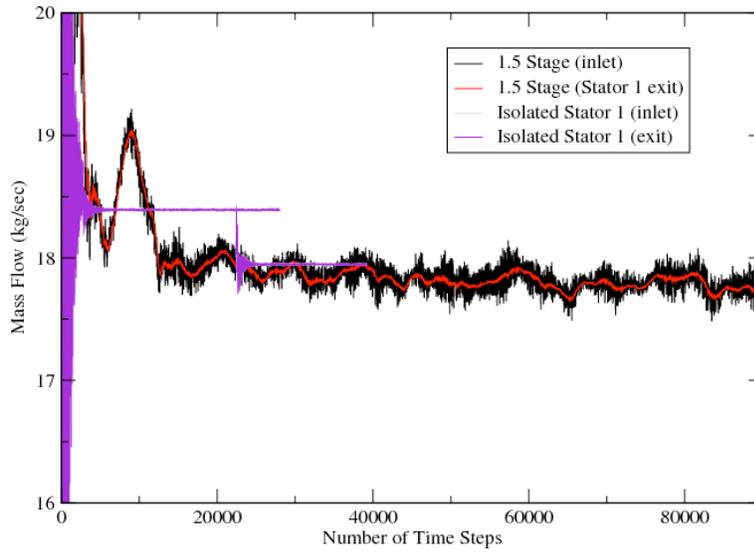


Figure 6. Mass Flow Convergence History Comparison.

LSRR Stator 1 Midspan Pressure Coefficient

Test Data at Flow Coefficient of 0.78 (+/- 1%), Isolated S1 CFD Results

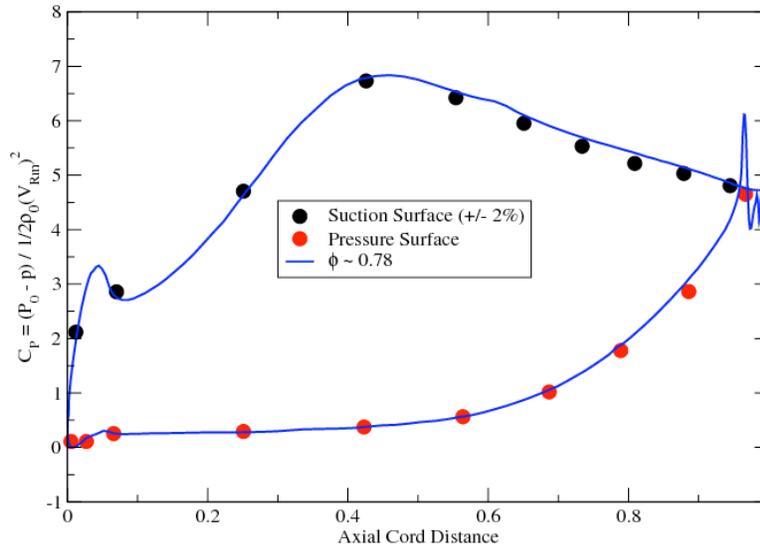
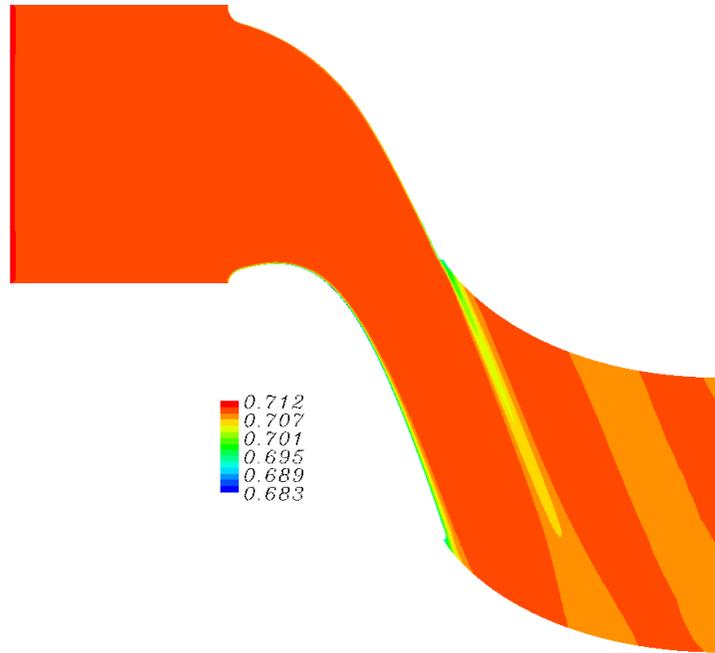
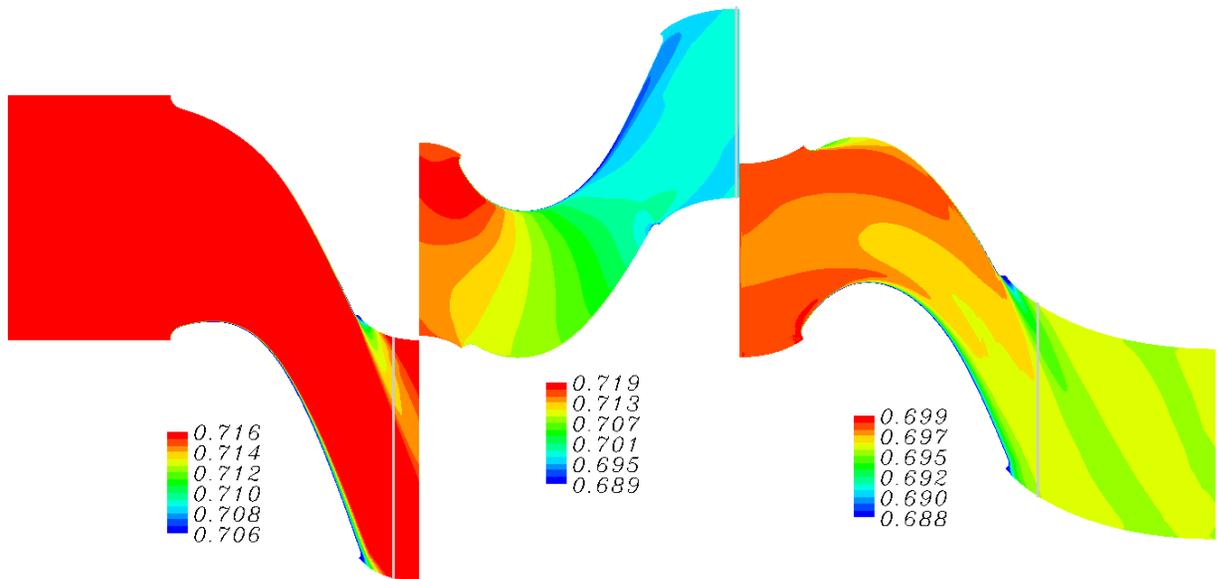


Figure 7. Isolated Stator 1 Midspan Pressure Coefficient.



(a) Isolated Stator 1 Results



(b) 1½ stage Results

Figure 8. Midspan Time Averaged Total Pressure Contour.

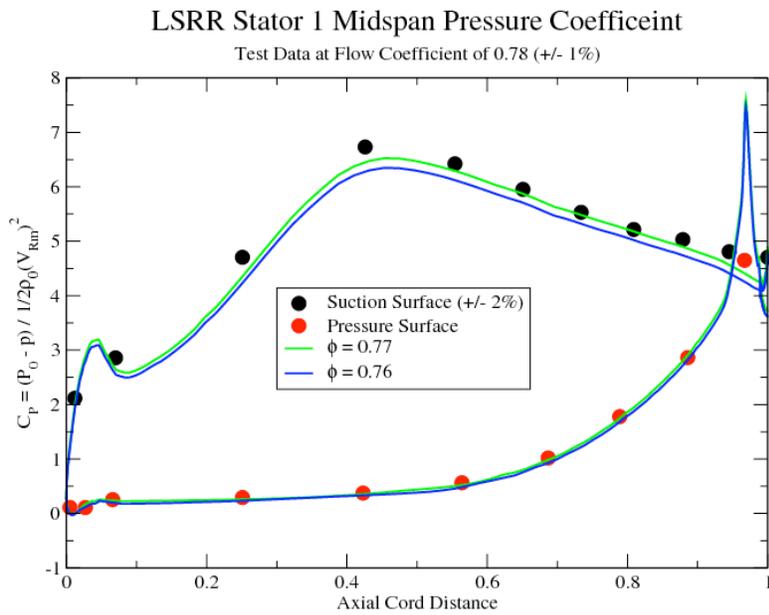


Figure 9a. Stator 1 Midspan Time Averaged Pressure Coefficient.

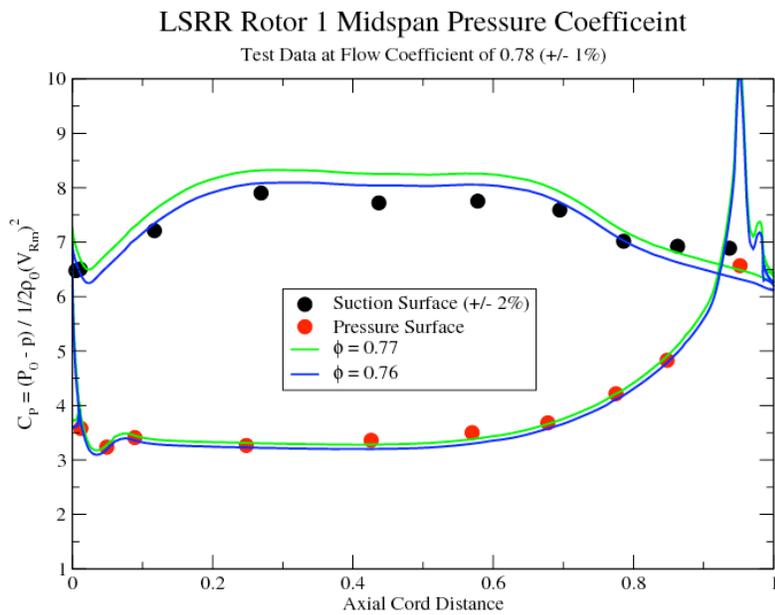


Figure 9b. Rotor 1 Midspan Time Averaged Pressure Coefficient.

LSRR Stator 2 Midspan Pressure Coefficient

Test Data at Flow Coefficient of 0.78 (+/- 1%)

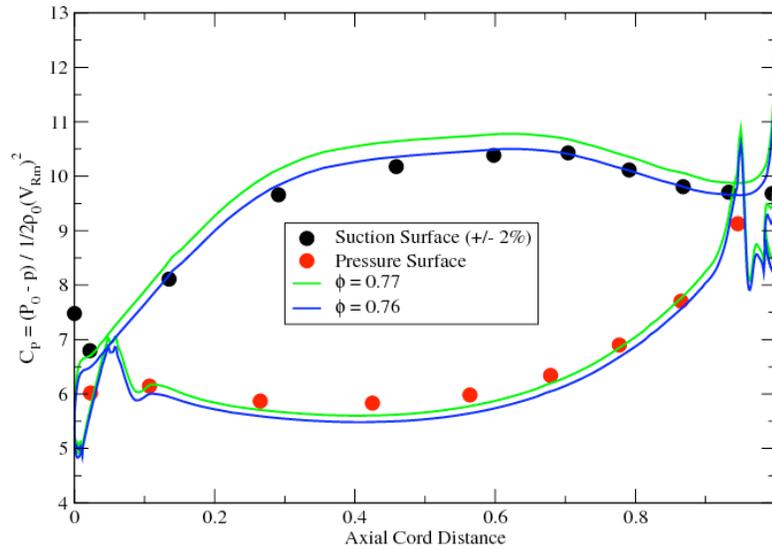


Figure 9c. Stator 2 Midspan Time Averaged Pressure Coefficient.

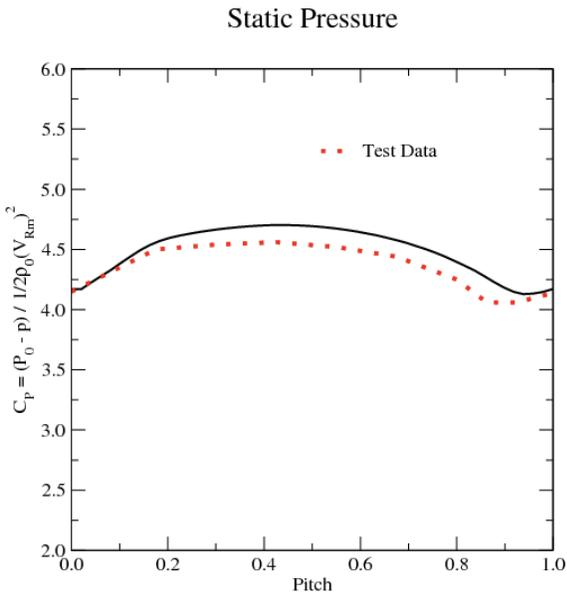
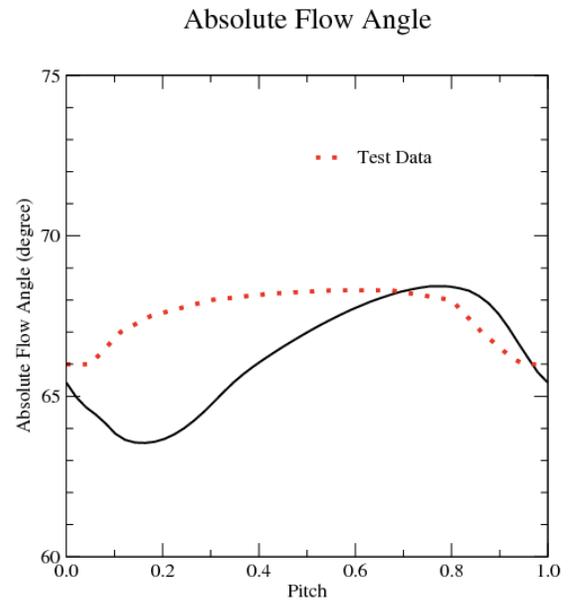
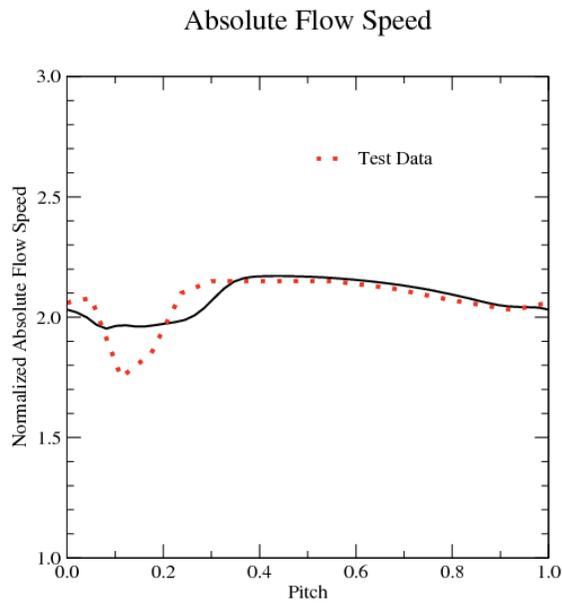


Figure 10. Midspan Stator 1 Exit Profile at 17% Axial Cord From Trailing Edge.

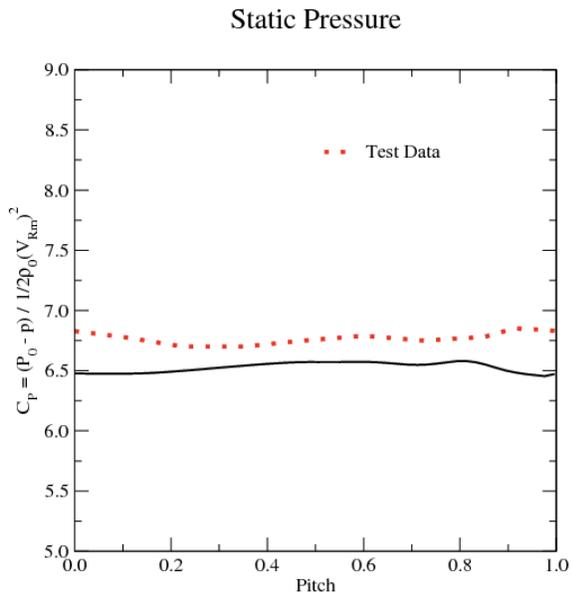
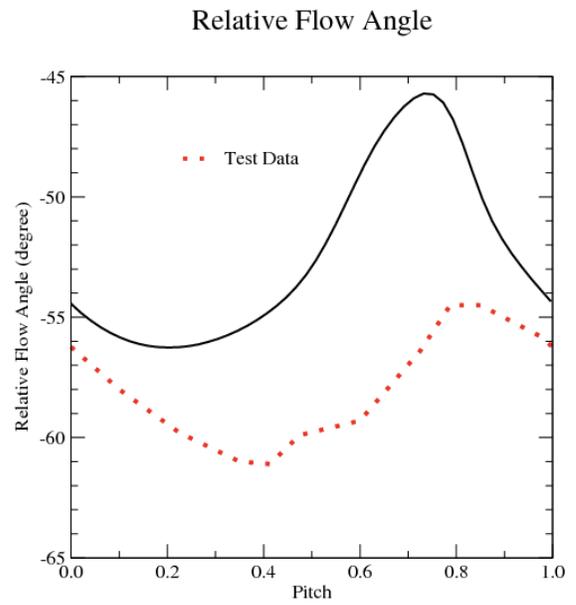
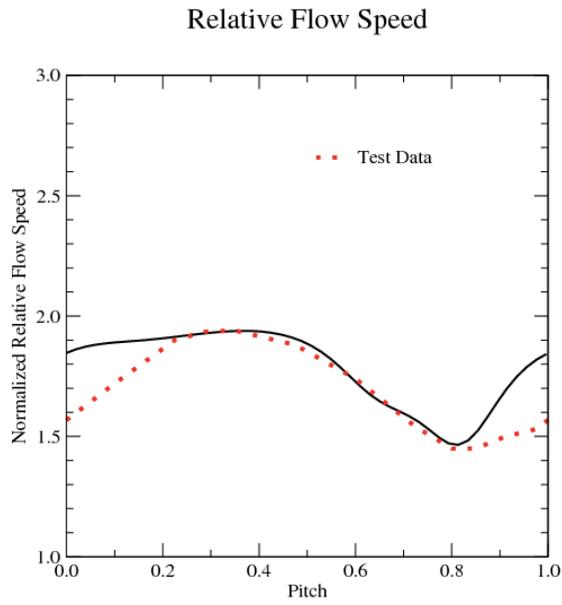


Figure 11. Midspan Rotor 1 Exit Profile at 36% Axial Cord From Trailing Edge.

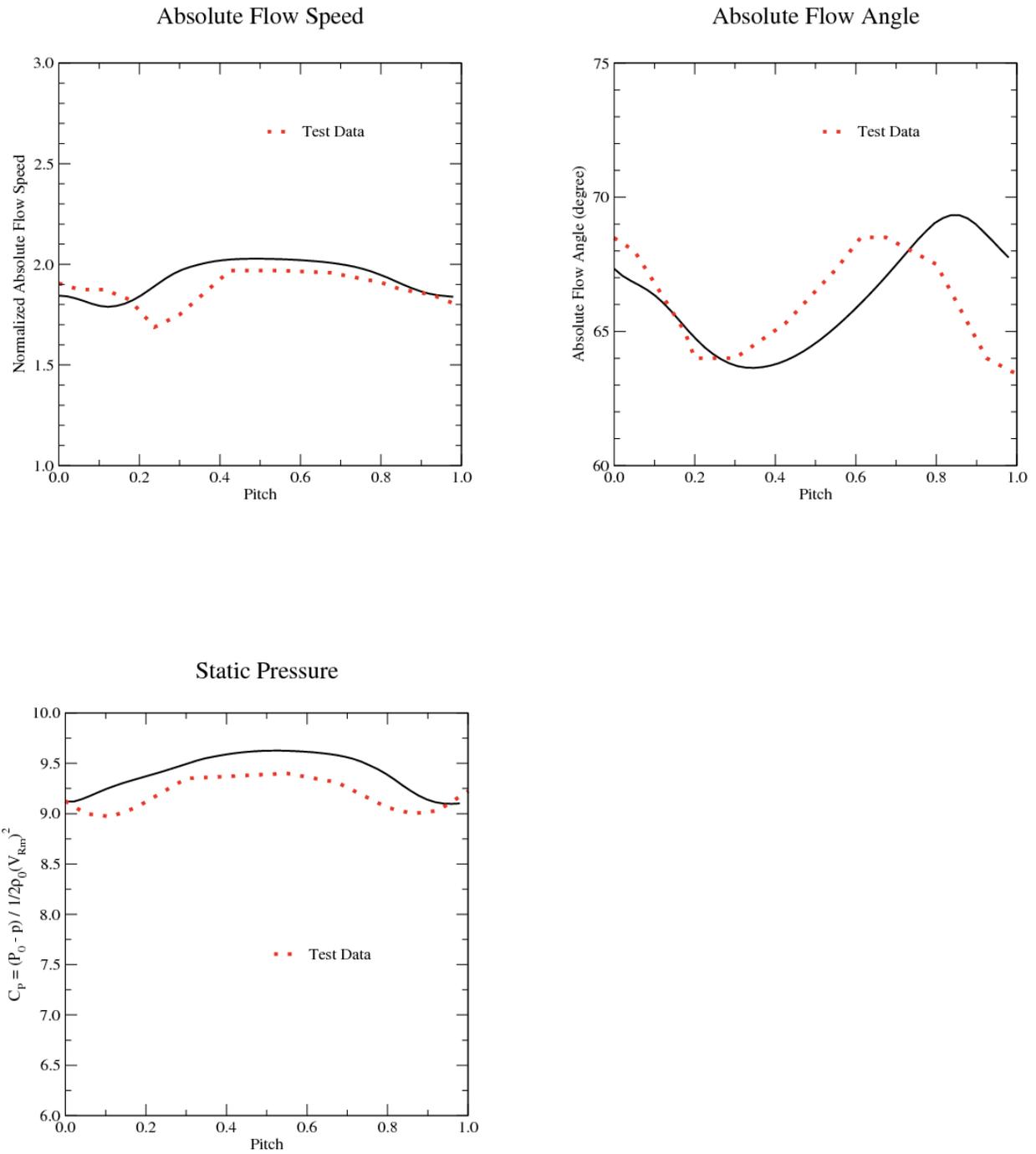


Figure 12. Midspan Stator 2 Exit Profile at 14% Axial Cord From Trailing Edge.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188		
<p>The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.</p> <p>PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.</p>					
1. REPORT DATE (DD-MM-YYYY) 01-05-2012		2. REPORT TYPE Final Contractor Report		3. DATES COVERED (From - To) 10-1-2010 - 8-31-2011	
4. TITLE AND SUBTITLE Unsteady Aero Computation of a 1 1/2 Stage Large Scale Rotating Turbine			5a. CONTRACT NUMBER NNC06BA07B		
			5b. GRANT NUMBER		
			5c. PROGRAM ELEMENT NUMBER		
6. AUTHOR(S) To, Wai-Ming			5d. PROJECT NUMBER		
			5e. TASK NUMBER NNC10E420T		
			5f. WORK UNIT NUMBER WBS 877868.02.07.03.01.02.01		
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) University of Toledo Toledo, Ohio 43606			8. PERFORMING ORGANIZATION REPORT NUMBER E-18131		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSORING/MONITOR'S ACRONYM(S) NASA		
			11. SPONSORING/MONITORING REPORT NUMBER NASA/CR-2012-217438		
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Category: 02 Available electronically at http://www.sti.nasa.gov This publication is available from the NASA Center for AeroSpace Information, 443-757-5802					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT This report is the documentation of the work performed for the Subsonic Rotary Wing Project under the NASA's Fundamental Aeronautics Program. It was funded through Task Number NNC10E420T under GESS-2 Contract NNC06BA07B in the period of 10/1/2010 to 8/31/2011. The objective of the task is to provide support for the development of variable speed power turbine technology through application of computational fluid dynamics analyses. This includes work elements in mesh generation, multistage URANS simulations, and post-processing of the simulation results for comparison with the experimental data. The unsteady CFD calculations were performed with the TURBO code running in multistage single passage (phase lag) mode. Meshes for the blade rows were generated with the NASA developed TCGRID code. The CFD performance is assessed and improvements are recommended for future research in this area. For that, the United Technologies Research Center's 1 1/2 stage Large Scale Rotating Turbine was selected to be the candidate engine configuration for this computational effort because of the completeness and availability of the data.					
15. SUBJECT TERMS Computational fluid dynamics; Aerodynamics; Axial flow turbines					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT UU	18. NUMBER OF PAGES 30	19a. NAME OF RESPONSIBLE PERSON STI Help Desk (email:help@sti.nasa.gov)
a. REPORT U	b. ABSTRACT U	c. THIS PAGE U			19b. TELEPHONE NUMBER (include area code) 443-757-5802

