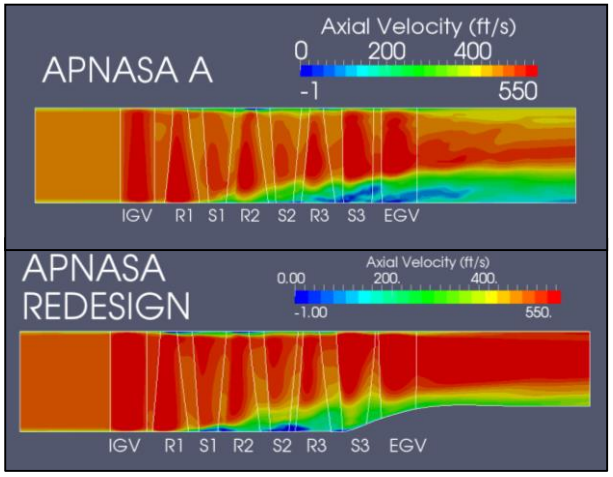
Computational Analysis and Proposed Redesign of the 3-Stage Axial Compressor in the 11-by-11-Foot Transonic Wind Tunnel at NASA Ames Research Center



Sameer Kulkarni
Joe Veres
Phil Jorgenson
Tim Beach

NASA Glenn Research Center

5/24/2017

Contents

Background on Ames 11x11 TWT

- **11- by 11-Foot Transonic Wind Tunnel (11-Foot TWT) Facility** at NASA Ames Research Center in Moffett Field, California "*has been instrumental in the development of virtually every domestically produced commercial transport and military fixed-wing airframe since the 1960s. The facility is used extensively for airframe testing and aerodynamic studies and has played a vital role in every manned space flight program.*" [1]

- Closed-loop tunnel with Mach range 0.20 to 1.40, stagnation pressure range 3.0 to 32.0 psia, Rn range 0.30 to 9.6 million per foot.

- Airflow provided by 24 foot diameter, 3-stage axial compressor powered by variable-speed induction motors.

[1] ***https://www.nasa.gov/centers/ames/orgs/aeronautics/windtunnels/11x11-wind-tunnel.html***

Unitary Plan Wind Tunnel

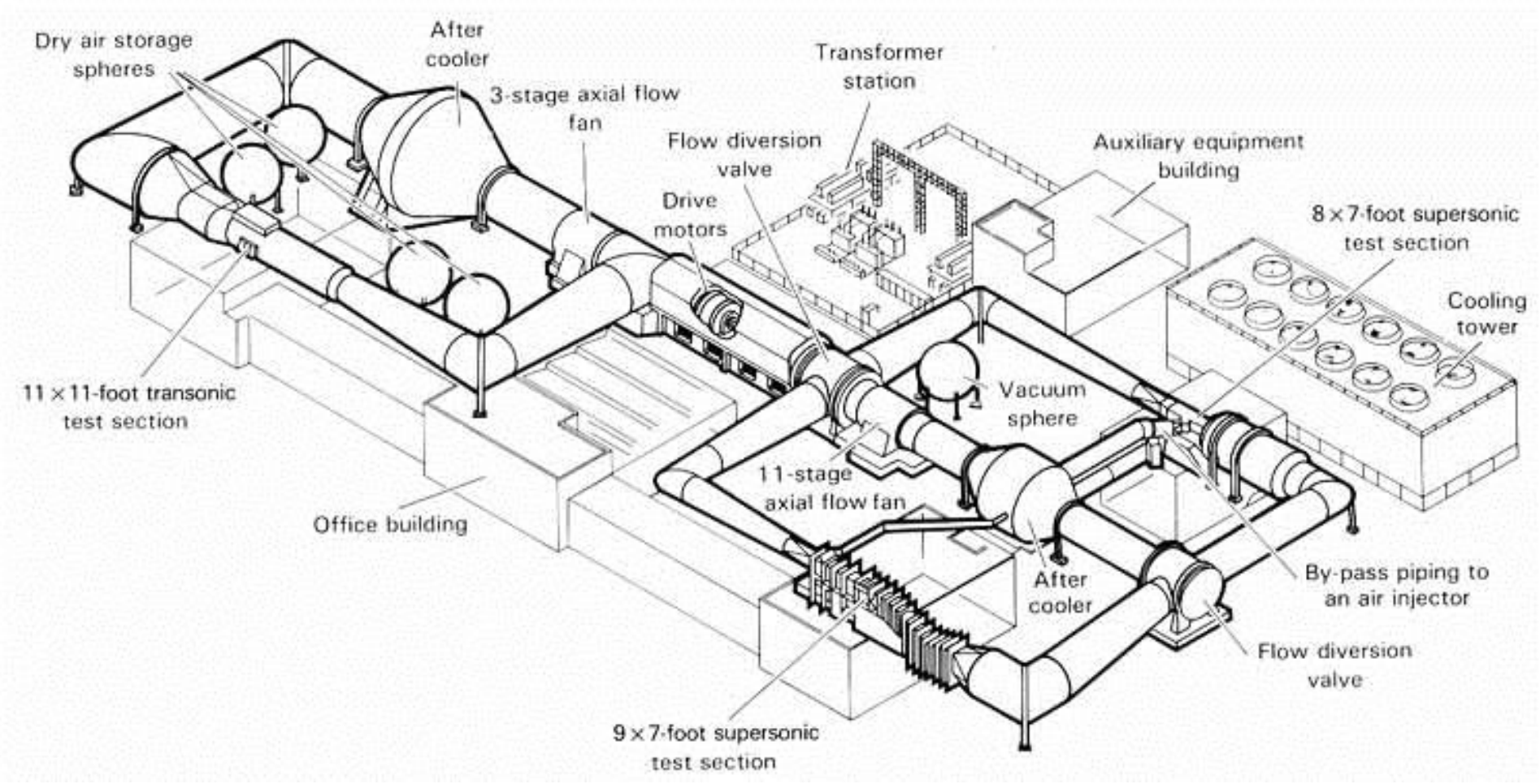


Image from Baals, Donald D., "Wind Tunnels of NASA," NASA SP-440, 1981.

Motivation

- Compressor was designed ~1950s. Aluminum blades were used to minimize weight due to rotordynamics concerns. These blades are sensitive to damage/erosion.

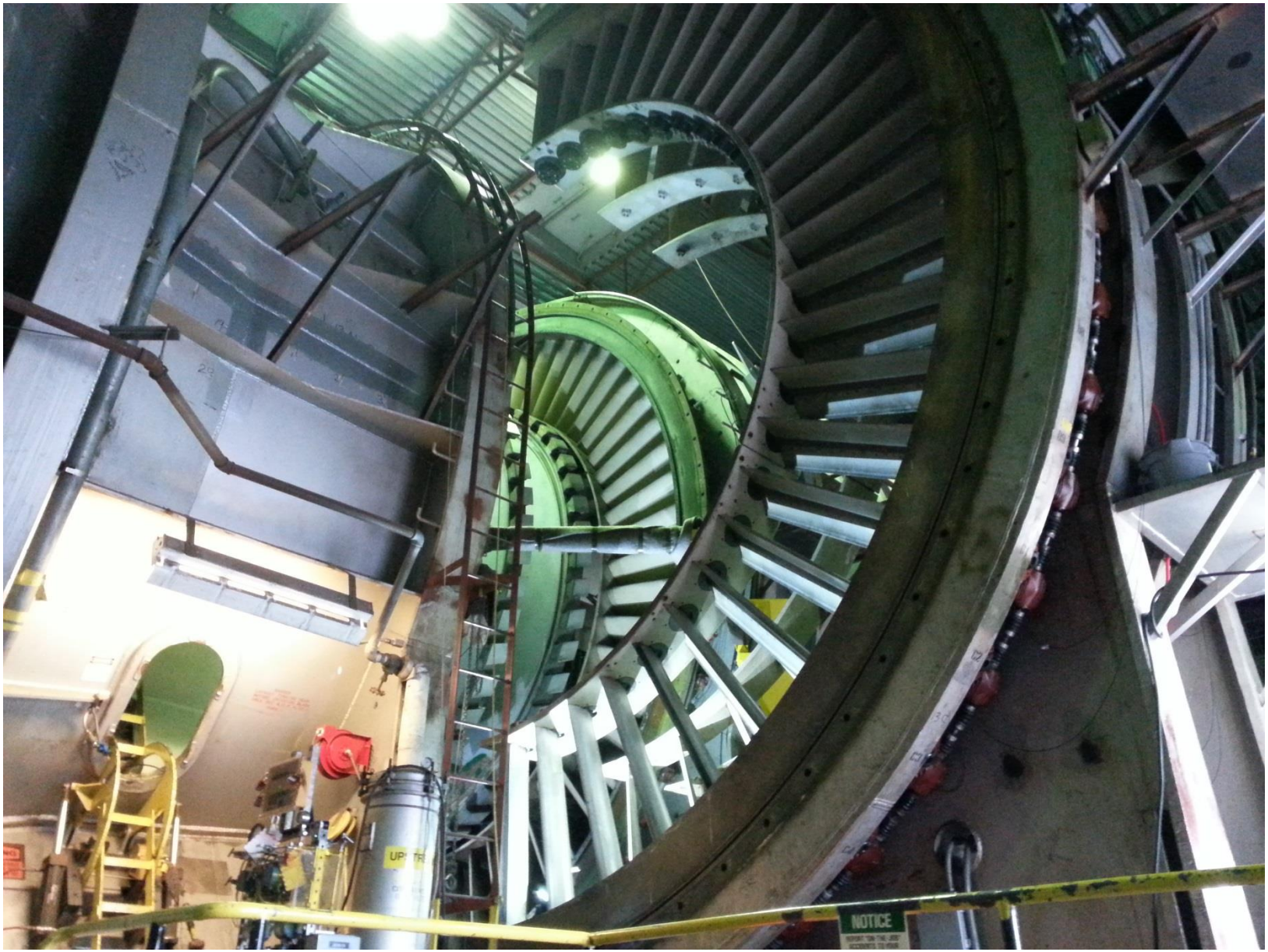
- Blades are inspected for impacts/damage every 50 hours and overhauled every 2400 hours, wherein the blades are removed for sanding and penetrant inspection.
  - Overhaul is approx. 1 man-year effort and 1 month of facility downtime.

- Re-blading with new hollow steel blades was planned to stretch inspections to 200 hours and obviate blade overhauls.

- Several facility upgrades over 60+ years (flow conditioning devices, instrumentation) have increased tunnel blockage and reduced peak test section Mach number capability.

➔Opportunity to increase test section Mach number capability to 1.45+ by increasing compressor pressure ratio via new blade design.

Compressor Case Split During Overhaul



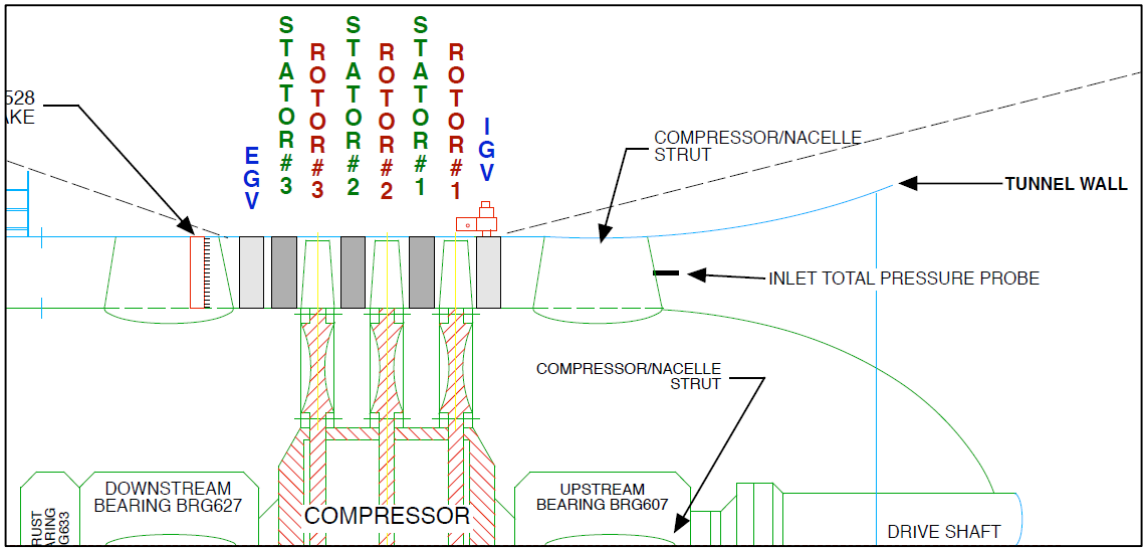
Spare Rotor Blades and Blades Awaiting Reburb

Spare Rotor Blades and Blades Awaiting Reburb

GRC Team In-between Rotor Disks

Features of the Existing Compressor

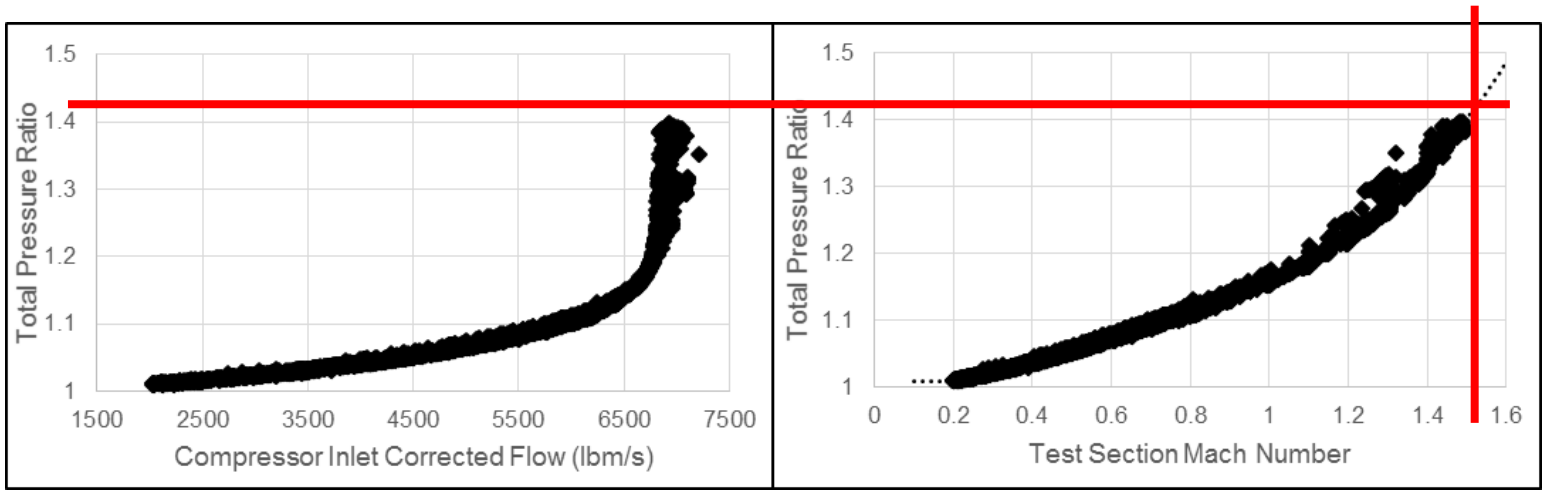
- 3-stage axial compressor with inlet guide vanes (IGVs) and exit guide vane (EGVs).
- 54 IGVs, 52 rotor blades per stage, 34 stator 1&2 vanes, 58 stator 3 vanes, 60 EGVs.
- IGVs are variable camber (hinged at mid-chord); IGV flap can vary from -7.5° (less pre-swirl) to +19.5° (more pre-swirl). Nominal 0° position gives +33° pre-swirl into rotor 1.
- Hub and casing flow paths have constant radius of 8.5 ft and 12 ft respectively.
- Approx. 0.5 inch rotor tip clearance.
- Rotational speed ranges from 150-650 RPM (rotor tip speed 190-815 ft/s).
- Test section Mach number and Reynolds number are set by varying compressor speed, IGV camber, and compressor inlet pressure.
- At peak test section Mach number of ~1.45 (empty test section), compressor inlet corrected flow rate is ~7000 lb/s and compressor overall pressure ratio is ~1.4.



Cross-sectional View of the Compressor Case

Design Constraints

- Rotor blade shapes may change, but blade count may not (re-use existing disks).
- Desire to keep new rotor shape identical across each stage to minimize tooling/fabrication costs.
- Stator vane shapes and vane counts may not change, but re-staggering existing stators is possible.
- IGVs must be remain unchanged due to complexities in removing/replacing actuators. May be possible to close IGV flap additional 2.5° for -10° total closure from nominal.
- Available margins in motor, shaft, and bearings should allow for increasing maximum rotational speed from 650 to 695 RPM.



Operating lines for the existing compressor (left) and facility (right). Red lines indicate target for a new design.

Approach

- Model the performance of the existing compressor and compare to test data to calibrate numerical tools:
  - Turbomachinery design/analysis code HT0300[2].
  - Turbomachinery RANS 3D CFD solvers APNASA[3] and ADPAC[4].
- Iterate on new compressor design (within identified design constraints) using HT0300 to achieve at least a 10% increase in total pressure rise.
- Validate predicted performance of new design with CFD using APNASA.

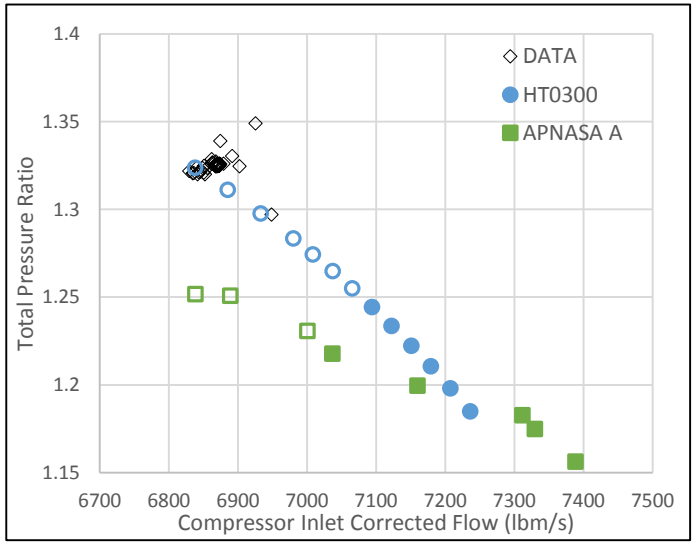
[2] Hearsey, R. M., "Program HT0300 NASA 1994 Version," Doc. No. D6-81569TN, Volumes 1 and 2, The Boeing Company, 1994.

[3] Adamczyk, J. J., "Model Equation for Simulating Flows in Multistage Turbomachinery," NASA TM-86869, 1985.

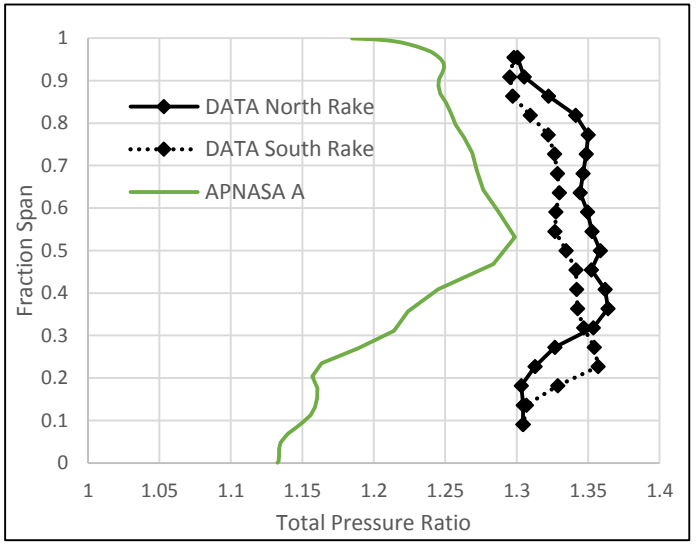
[4] Hall, E. J., Heidegger, N. J. and Delaney, R. A., "ADPAC v I .O - User's Manual," NASA CR-1999-206600, 1999.

Validating Tools Against Existing Data

- Existing compressor geometry with nominal IGV setting was used as input for HT0300 in analysis mode to generate performance predictions at 634 RPMC.
- Geometry definition was then used to generate meshes for CFD analysis using APNASA (905x51x51 grid). Initial APNASA simulation is referenced herein as “APNASA A”
- These two results were compared to data collected during a 1997 facility checkout.



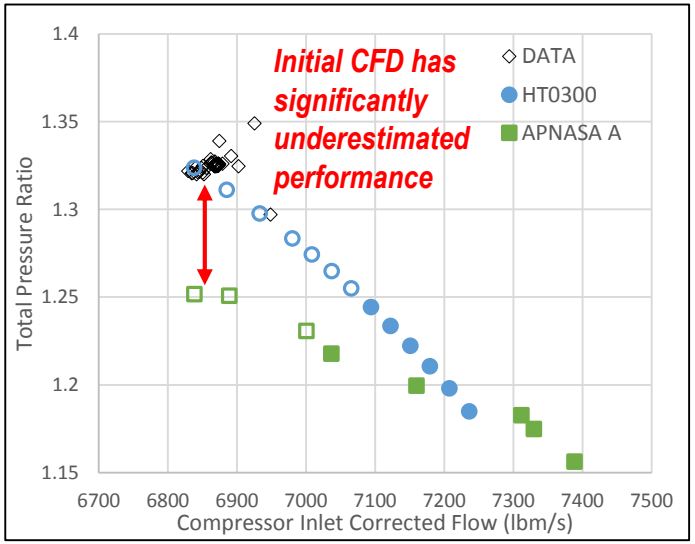
634 RPMC speedline with nominal IGV angle. Open symbols indicate diffusion factors exceeding 0.5 in the design code or unsteady/unconverged CFD results.



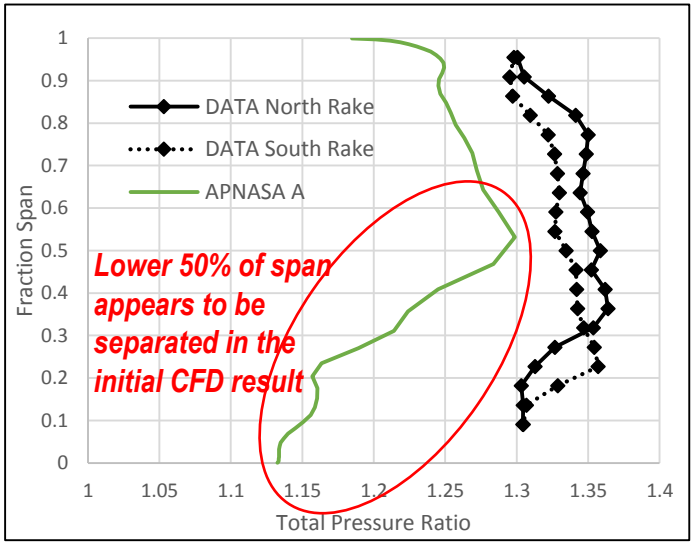
Radial profile of exit P0 normalized by inlet P0. Comparison of operating points at ~6890 lb/s.

Validating Tools Against Existing Data

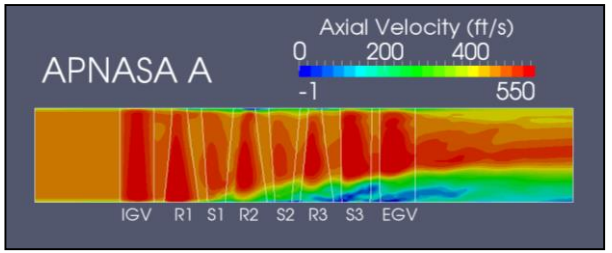
- Existing compressor geometry with nominal IGV setting was used as input for HT0300 in analysis mode to generate performance predictions at 634 RPMC.
- Geometry definition was then used to generate meshes for CFD analysis using APNASA (905x51x51 grid). Initial APNASA simulation is referenced herein as “APNASA A”
- These two results were compared to data collected during a 1997 facility checkout.



634 RPMC speedline with nominal IGV angle. Open symbols indicate diffusion factors exceeding 0.5 in the design code or unsteady/unconverged CFD results.



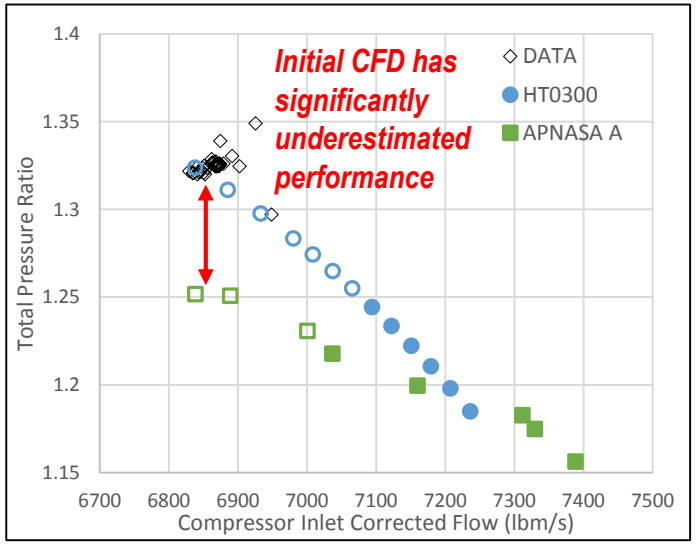
Radial profile of exit P0 normalized by inlet P0. Comparison of operating points at ~6890 lb/s.



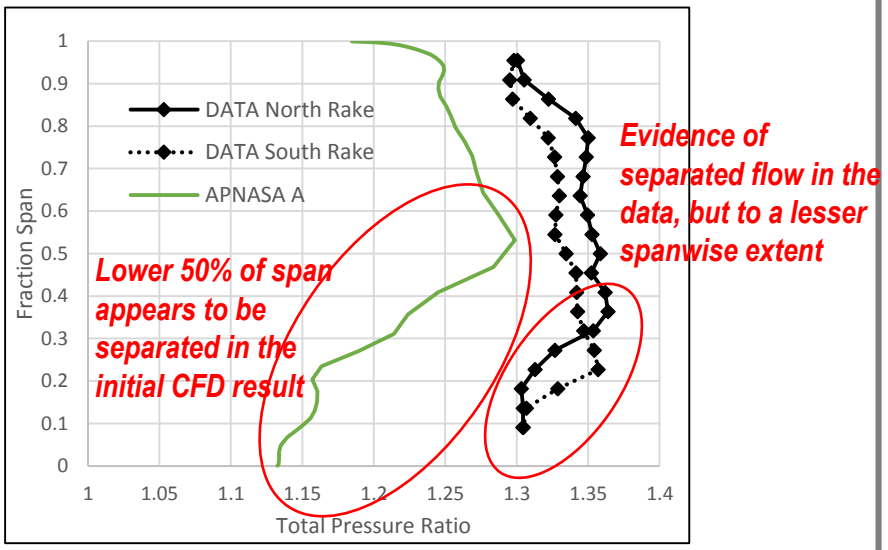
Circumferentially mass-averaged axial velocity contour of the initial CFD result showing massive hub separation.

Validating Tools Against Existing Data

- Existing compressor geometry with nominal IGV setting was used as input for HT0300 in analysis mode to generate performance predictions at 634 RPMC.
- Geometry definition was then used to generate meshes for CFD analysis using APNASA (905x51x51 grid). Initial APNASA simulation is referenced herein as “APNASA A”
- These two results were compared to data collected during a 1997 facility checkout.



634 RPMC speedline with nominal IGV angle. Open symbols indicate diffusion factors exceeding 0.5 in the design code or unsteady/unconverged CFD results.



Radial profile of exit P0 normalized by inlet P0. Comparison of operating points at ~6890 lb/s.

- Compressor hardware was examined for physical or geometrical features that were not included in the initial CFD simulations.

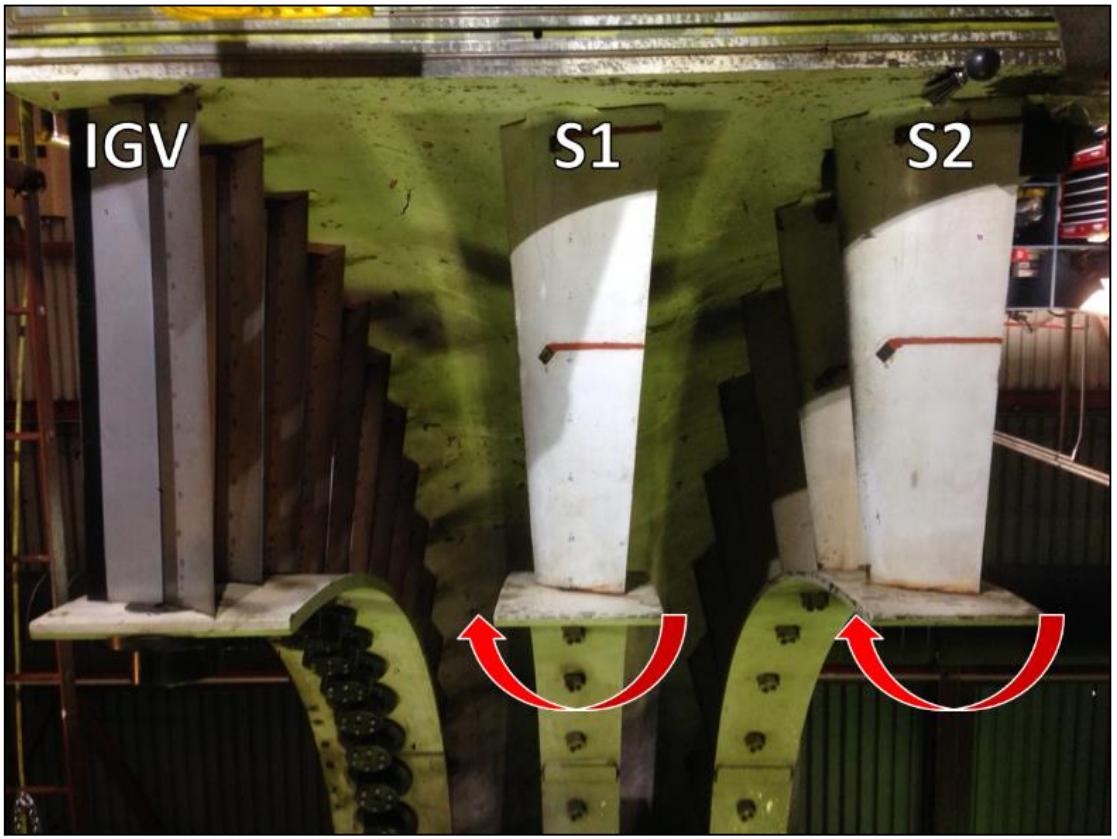
Potential Sources of Performance Mismatch

- Stator “button” endwall gap leakages: Resettable stator vanes have these gaps to avoid interference with flowpath hardware.

- Gaps can be gridded and included in CFD solution domain or modeled as periodic boundary conditions.

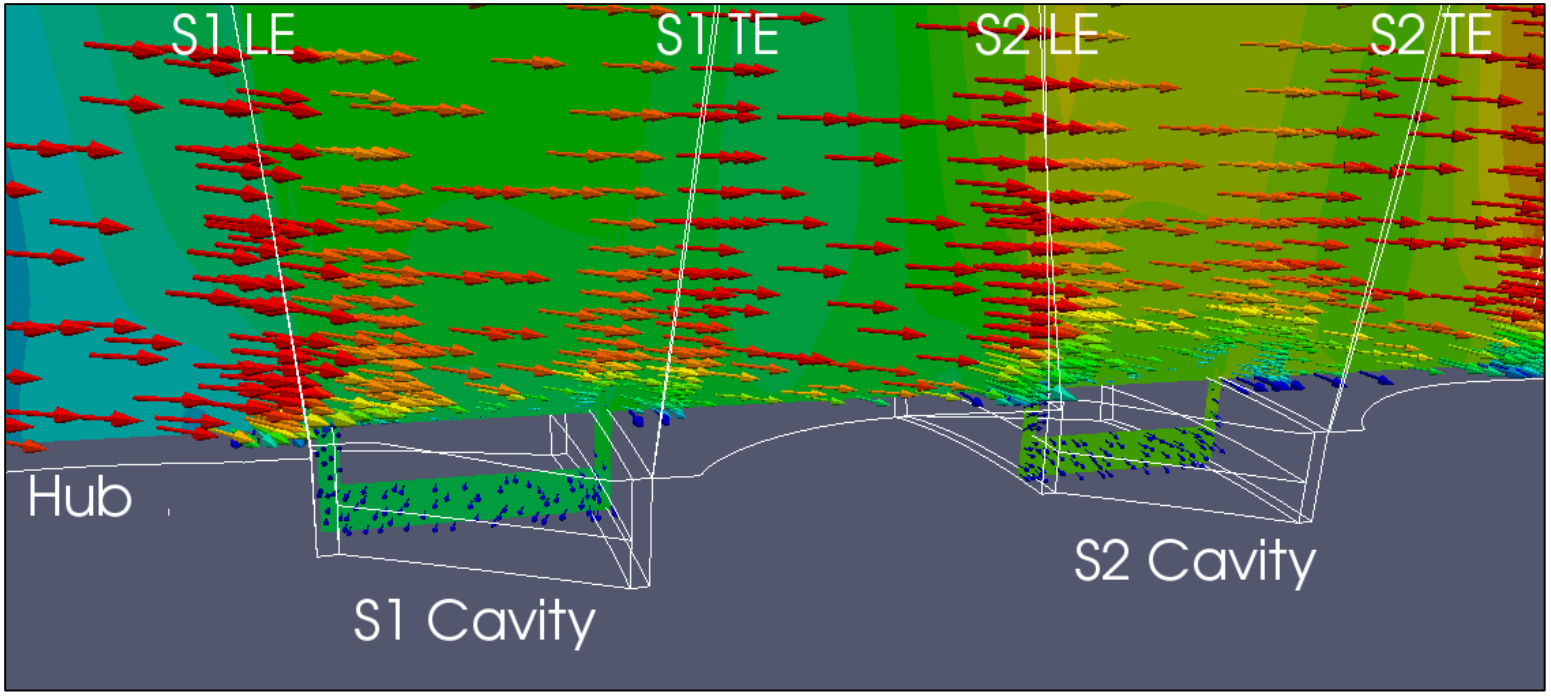
Potential Sources of Performance Mismatch

- Under-stator cavity leakage driven by static pressure gradients at the stator hubs – these are typically sealed via labyrinth seals but are unsealed in this compressor.
- Leakage path can be gridded and included in the CFD solution domain or modeled as mass flow inflow/outflow boundary conditions along the hub.



Potential Sources of Performance Mismatch

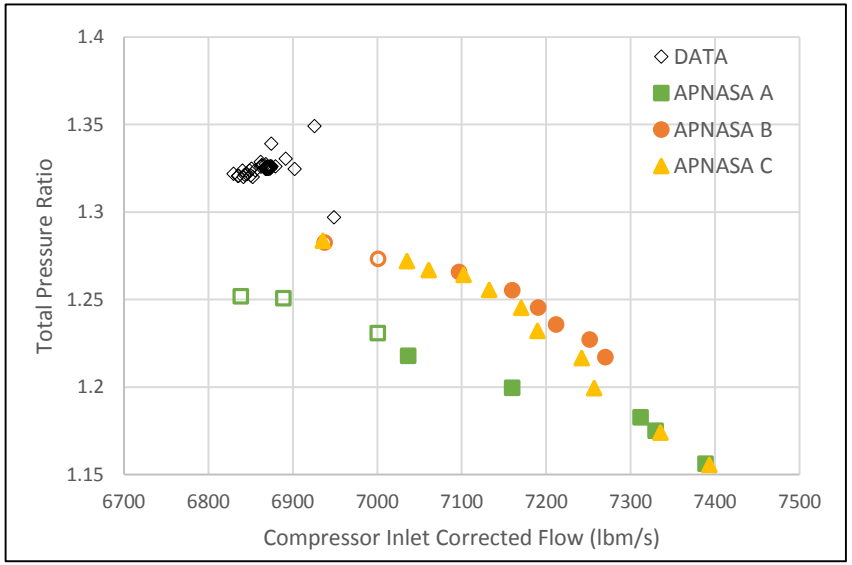
- ADPAC was used to grid the under-stator cavities and generate a CFD solution.
- Total mass flow recirculation in each cavity was found to be approx. 0.5% of compressor inlet mass flow rate, with reinjection angles of 25° and 20° upstream stator 1 and 2 respectively.



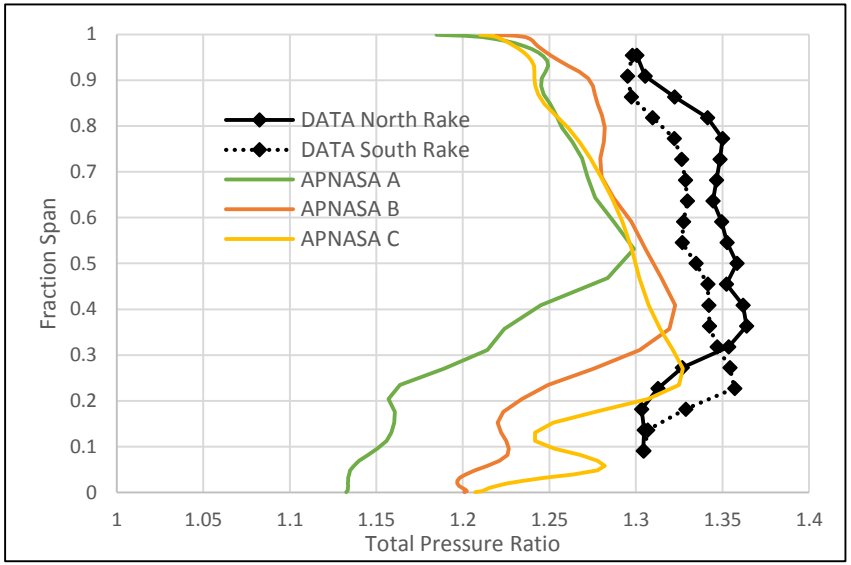
Meridional contour of static pressure showing the under-stator cavity flow recirculation, with absolute velocity vectors colored by axial velocity magnitude

Performance Impact of Modeling Stator Cavity and Button Leakages

- Additional APNASA simulations generated including stator Button gap leakages (APNASA B) and under-stator Cavity leakages (APNASA C).



634 RPMC speedline with nominal IGV angle. Open symbols indicate diffusion factors exceeding 0.5 in the design code or unsteady/unconverged CFD results.

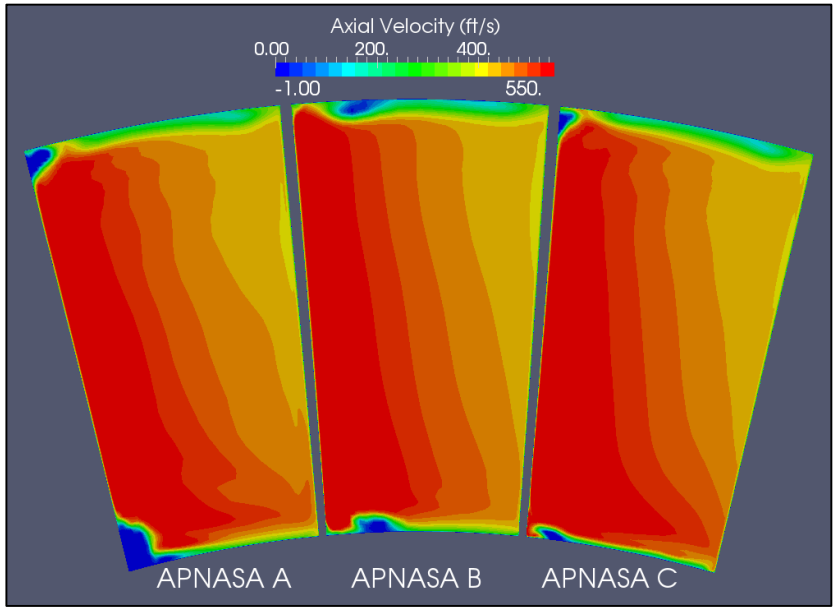


Radial profile of exit P0 normalized by inlet P0. Comparison of operating points at ~6890 lb/s.

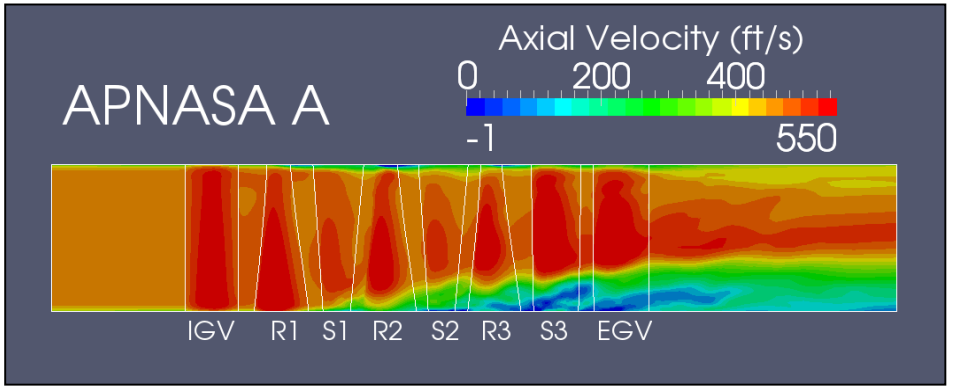
Performance Impact of Modeling Stator Cavity and Button Leakages

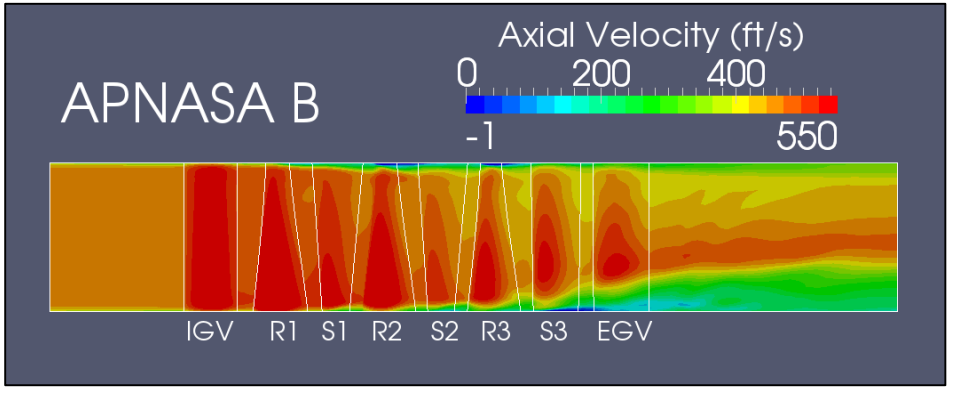
- APNASA B: Endwall gap leakage flow from pressure surface re-energizes low momentum fluid associated with corner separations on suction surface and hub/case

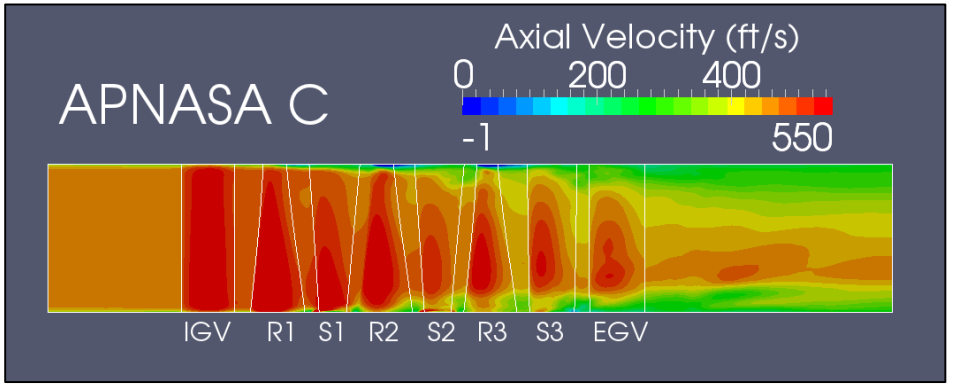
- APNASA C: Corner separation at hub is reduced due to smaller boundary layer driven by high momentum fluid leaking into the flowpath from the under-stator cavity upstream of the stator leading edge.



Cross-passage contours of axial velocity at 70% chord of stator 3.

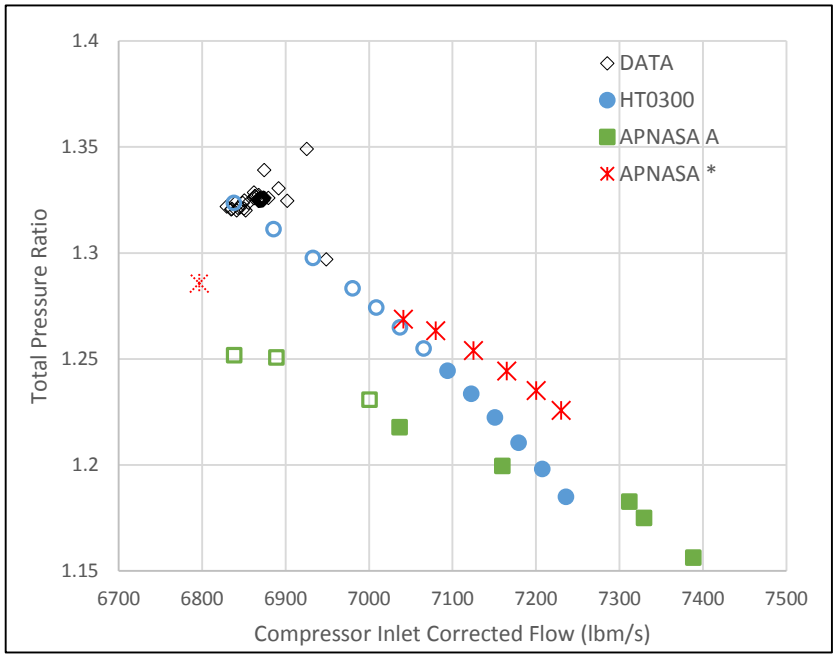




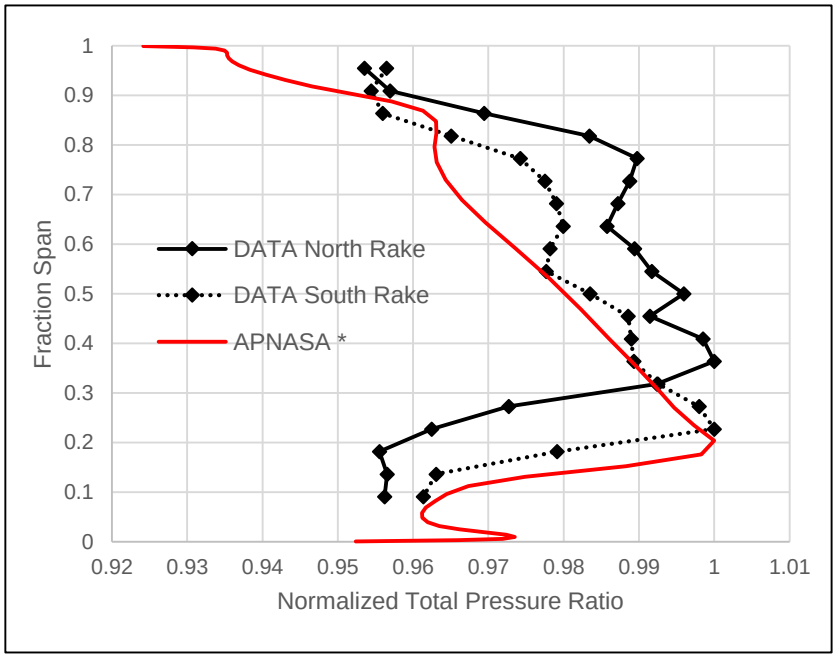


Performance Impact of Modeling Stator Cavity and Button Leakages

- Stator cavity leakage and button leakage models rolled into APNASA *
- Performance comparison has improved but it is clear that computational stability and high loadings are problematic in the existing design.
- Stator cavity leakage and button leakage models were applied to new design iterations to capture physics of the real compressor.



634 RPMC speedline with nominal IGV angle. Open symbols indicate diffusion factors exceeding 0.5 in the design code or unsteady/unconverged CFD results.



Radial profile of exit pressure ratio, normalized by the local maximum exit pressure ratio. APNASA * shows good agreement with rake data in profile shape.

Diffusion Factor Limitations

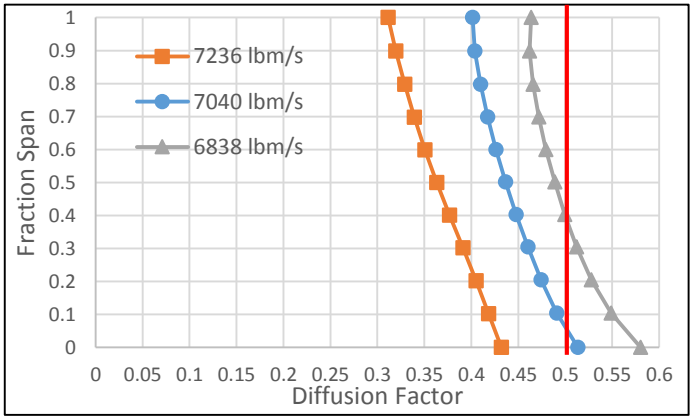
- Diffusion factor for a compressor blade element is a design parameter which is correlated with loss, separation, stability. It is defined as:

$$D = \underbrace{\left(1 - \frac{V_2}{V_1}\right)}_{\text{Diffusion due to deceleration of flow in the relative frame across the blade}} + \underbrace{\frac{\Delta V_\theta}{2\sigma V_1}}_{\text{Turning associated with the suction surface of the blade}}$$

- σ: solidity (chord/pitch)
- V_1: Axial velocity at blade row inlet
- V_2: Axial velocity at blade row exit
- ΔV_θ: Difference in inlet and exit relative tangential velocities

- Typically, losses spike for D>0.5 and flow separation from the suction surface of the blade leads to flow instability and compressor stall.

- Numerical tools suggest that the existing compressor operates at D>0.5 at high speeds, likely due to constant annular area which over-diffuses the flow and the low solidity of stators, especially at the hub.

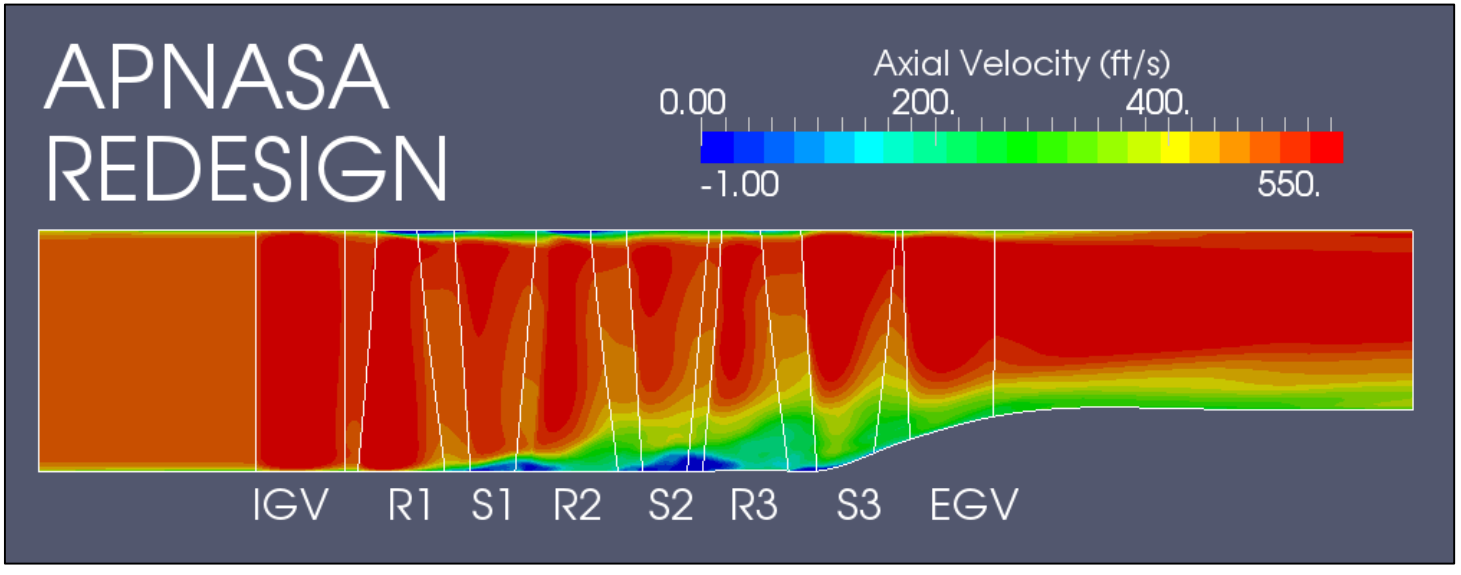


Stator 3 diffusion factor predicted by HT0300 for a range of operating points with nominal IGV angle at 634 RPMC.

Recommendations for a New Design

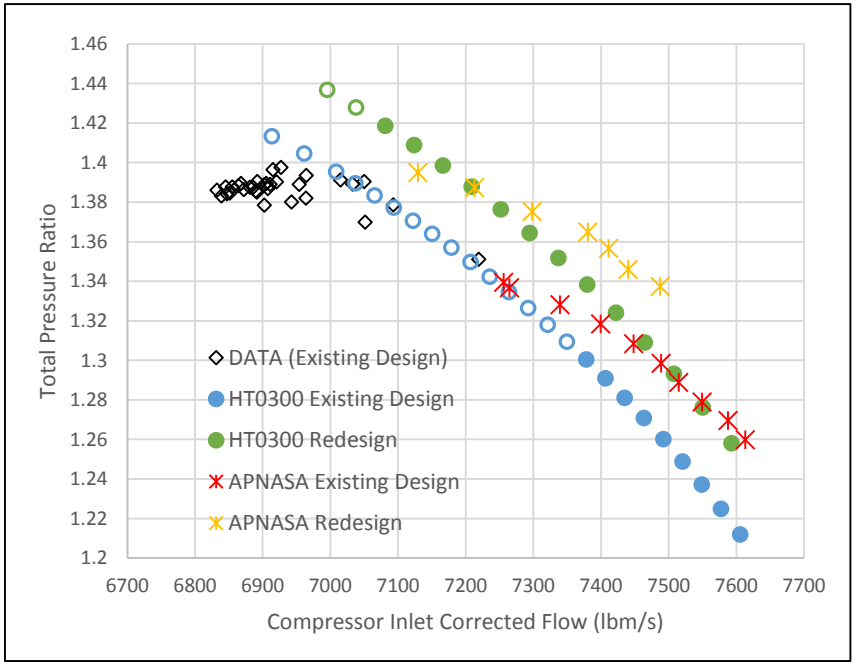
$$D = \left(1 - \frac{V_2}{V_1}\right) + \frac{\Delta V_\theta}{2\sigma V_1}$$

- Design speed was increased from 650 RPM to 690 RPM.
- Rotor blade inlet and exit angles were modified to accommodate changes in incidence at this higher speed.
- Identical rotor blade shapes were used across all three stages.
- Existing stator 1 and 2 vanes were re-staggered to reduce incidence angles.
- New airfoil shapes for stator 3 and EGV were proposed which incorporated an increase in hub radius from 8.5 ft to 9.4 ft.
- Stator 3 chord was increased by 15% at the case and 21.5% at the hub, resulting in approx. 14% increased solidity averaged across the span.



Recommendations for a New Design

- Proposed design increases overall total pressure rise by 10.5% with a corresponding decrease in stator 3 diffusion factor of 50%.
- It is estimated that this design may increase tunnel Mach number capability to 1.50.



Speedlines with -7.5° IGV angle. Open symbols indicate diffusion factors exceeding 0.5 in the design code. Existing design results are at 634 RPMC and redesign results are at 653 RPMC.