

UTSR Summer Fellowship at GE Energy  
Gas Turbine Aero Group

**NEXT GENERATION GAS TURBINE  
(NGGT) AERO RIG**

**Juan Hincapie – University of Louisiana at Lafayette**

Mentor: Neil Ristau

Manager: Gunnar Siden

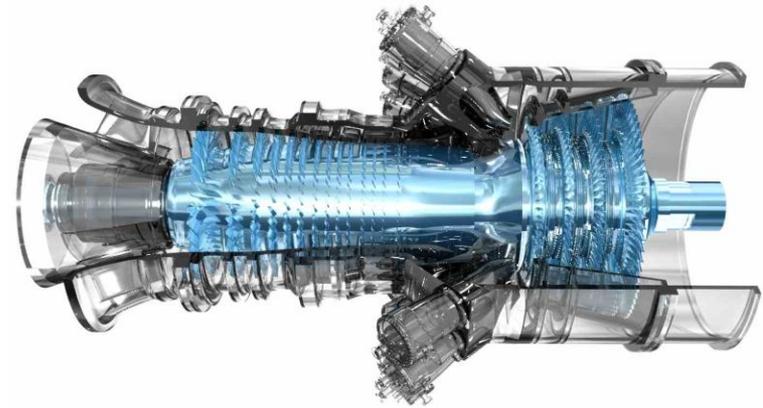
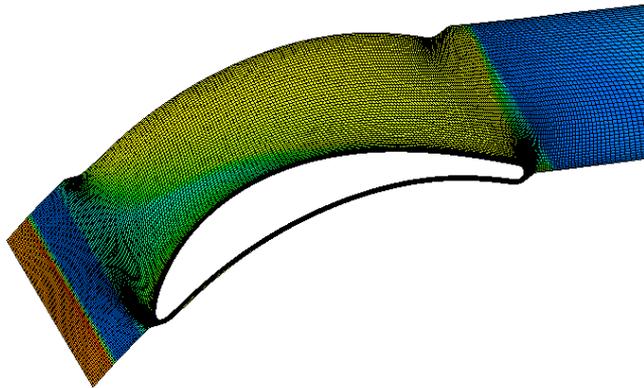


# INTRODUCTION

- In gas turbines applications, CFD plays greater part in the aerodynamic design of turbomachinery than in any other engineering application
  - Shorter design cycles – performance optimization
  - Reduced cost
- GE Energy designed, constructed, and tested a 3-stage power generation turbine rig to validate and verify CFD design tools and methodologies

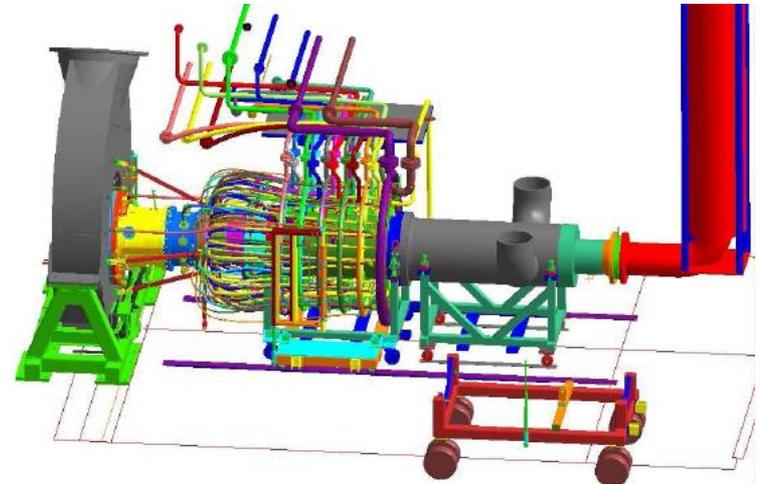
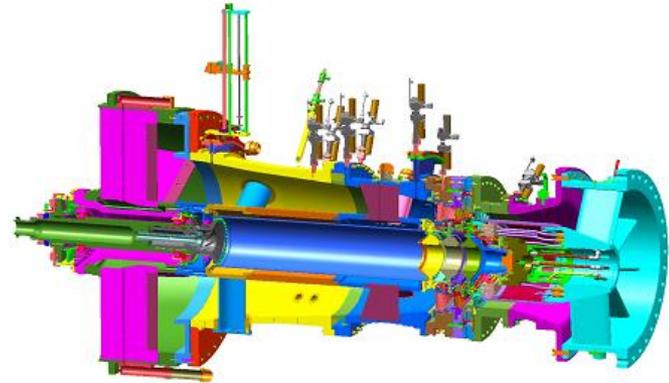
## Fellowship Description

The project involves performing data reduction, data interpretation, and CFD model analysis of a power generation turbine rig



# NGGT TURBINE AERO RIG

- 1:5 scale of the turbine from an actual power generation turbine
- Fully cooled rig with 16 independent secondary flow circuits to distribute cooling flows to airfoils, shrouds, wheelspace, etc
- Equipped with multiple sensors to measure pressure (total and static), temperature (total and static), mass flow, torque, and speed.
- Aero similarity maintained through scaling of airfoil rows, trailing edge thicknesses, throats, surface profiles, etc



# OBJECTIVES

**DATA** – Using the test data collected from the NGGT turbine rig:

- Refine a Matlab script that can be used to determine mixed-out profiles of pressure, temperature, velocity, and swirl at different locations of the flowpath
- Generate contour plots of pressure, velocity, temperature, and swirl

**CFD** – Using current CFD tools and methodologies:

- Run pre-test CFD analysis for all the operating conditions of the NGGT rig test
- Interpret the results given by the pre-test CFD model at the locations of the sensors used in the test
  - Determine mixed-out profiles of pressure, temperature, velocity, and swirl
  - Generate contour plots of pressure, velocity, temperature, and swirl
- Compare the profiles and contour plots with the data
- Update pre-test CFD model to match data to validate and verify the CFD design tools and methodologies



# DATA - Matlab

- Test data collected by data acquisition system is calibrated and reduced

Pt (psi)	Pe (psi)	Vabs(ft/s)	U(ft/s)	V(ft/s)	W(ft/s)	cirm(%)	rad(%)	Vcyl(ft/s)	Vn(ft/s)	Uz(ft/s)	Cu(ft/s)	Vr(ft/s)	PHI	alphz	Tt(F)
8.115	6.089	741.791	741.490	-15.362	14.501	35.170	19.900	741.632	741.654	741.495	-14.242	-15.362	-1.19	-1.10	121.78
8.114	6.097	740.021	739.703	-15.812	14.846	35.170	19.900	739.852	739.877	739.708	-14.587	-15.812	-1.22	-1.13	121.56
8.103	6.082	741.496	741.172	-15.078	15.909	35.170	19.900	741.343	741.331	741.177	-15.650	-15.078	-1.17	-1.21	121.64
8.103	6.079	741.937	741.635	-13.803	16.051	35.181	19.900	741.809	741.766	741.637	-15.936	-13.803	-1.07	-1.23	121.67
8.092	6.062	743.903	743.573	-12.877	18.041	35.190	19.900	743.792	743.685	743.573	-18.041	-12.877	-0.99	-1.39	121.73
8.092	6.065	743.211	742.882	-13.080	17.849	35.190	19.900	743.096	743.000	742.885	-17.704	-13.080	-1.01	-1.37	121.58
8.103	6.080	741.879	741.577	-13.814	16.039	35.190	19.900	741.750	741.711	741.583	-15.780	-13.814	-1.07	-1.22	121.64
8.119	6.106	738.960	738.667	-15.020	14.391	35.218	19.900	738.807	738.824	738.672	-14.133	-15.020	-1.16	-1.10	121.68
8.099	6.082	740.725	740.415	-13.523	16.653	35.240	19.900	740.602	740.544	740.420	-16.394	-13.523	-1.05	-1.27	121.59
8.107	6.089	740.776	740.471	-13.694	16.237	35.273	19.900	740.649	740.603	740.477	-15.979	-13.694	-1.06	-1.24	121.73
8.133	6.122	737.885	737.603	-14.978	13.883	35.300	19.900	737.733	737.760	737.608	-13.625	-14.978	-1.16	-1.06	121.90

- Script requests user input to read test data and perform radial and circumferential averaging, define constant variables, and contour limits.

- Script performs a constant area 2D mixing calculation to determine mixed-out profiles

```

Ptot_mass (Idx) = sum (RhoVzPtEA) / sum (RhoVzEA);
Ttot_mass (Idx) = sum (RhoVzTtEA) / sum (RhoVzEA);
Pstat_area (Idx) = sum (PstatEA) / sum (EA);
Cu_mix (Idx) = sum (RhoVzCuEA) / sum (RhoVzEA);
Vabs_mass (Idx) = sum (RhoVzVabsEA) / sum (RhoVzEA);
Vr_mix (Idx) = sum (RhoVzVrEA) / sum (RhoVzEA);

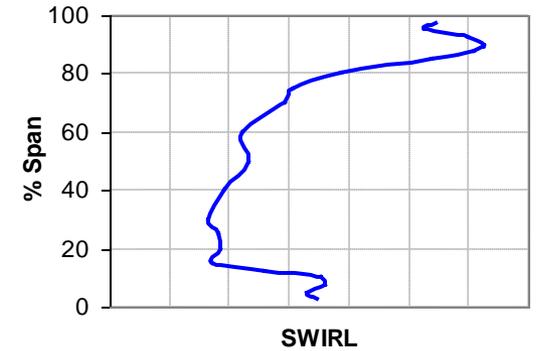
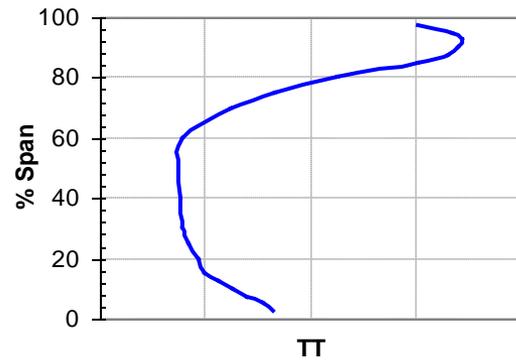
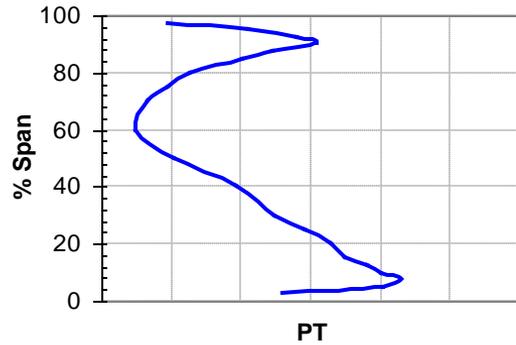
A = Cp*Ttot_mass (Idx) *Gc*J-1/2*CpCu_mix (Idx)^2; % Mixed-out average coefficient, A
B = Cp/Rg*(Pstat_area (Idx) *Gc*144+sum (RhoVzVzEA) / sum (EA)); % Mixed-out average coefficient, B
C = (Cp/Rg-0.5)*(sum (RhoVzEA) / sum (EA))^2; % Mixed-out average coefficient, C

Rho_mix (Idx) = 1 / (2*A) * (B+sqrt (B^2-4*A*C));
Vz_mix (Idx) = 1/Rho_mix (Idx) *sum (RhoVzEA) / sum (EA);
Alpha_mix (Idx) = (atan (Cu_mix (Idx) / Vz_mix (Idx)) *180/pi) +alpha_adjust;
Pstat_mix (Idx) = Pstat_area (Idx) +sum (RhoVzVzEA) / sum (EA) / (Gc*144) - (sum (RhoVzEA) / sum (EA))^2 / (Rho_mix (Idx) *Gc*144);
Tstat_mix (Idx) = Pstat_mix (Idx) / (Rg*Rho_mix (Idx)) *144/J;
Mstat_mix (Idx) = sqrt ((Cu_mix (Idx)^2+Vz_mix (Idx)^2) / (GAMSO*Rg*Tstat_mix (Idx) *J *Gc));
Ptot_mix (Idx) = Pstat_mix (Idx) * (Ttot_mass (Idx) / Tstat_mix (Idx))^ (GAMSO / (GAMSO-1));
    
```

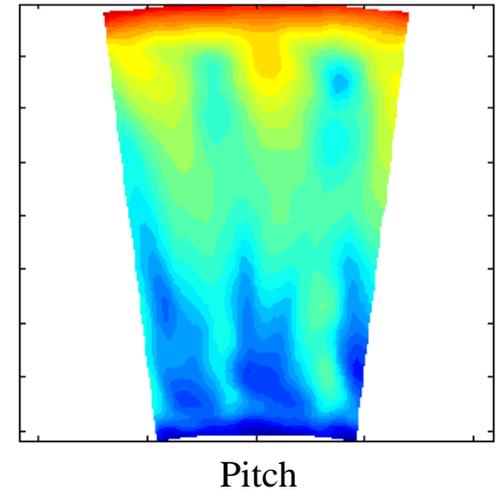
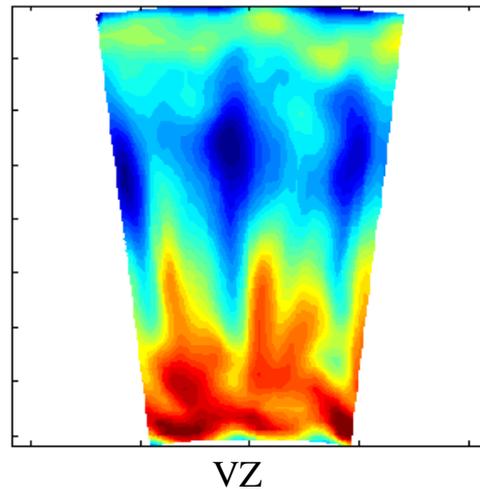
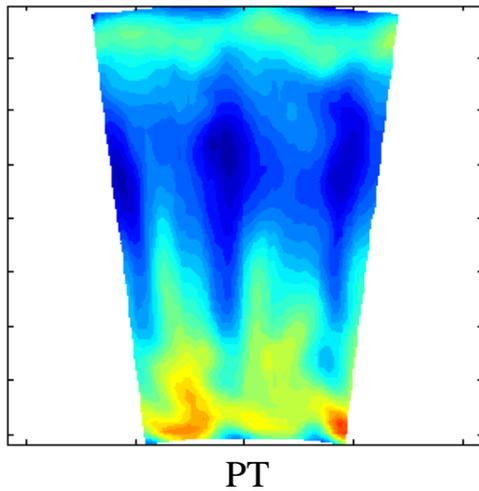


## Matlab Output

- Due to proprietary limitations, output plots are generated for illustrative purposes only
- Mixed-out profiles



- Contour Plots

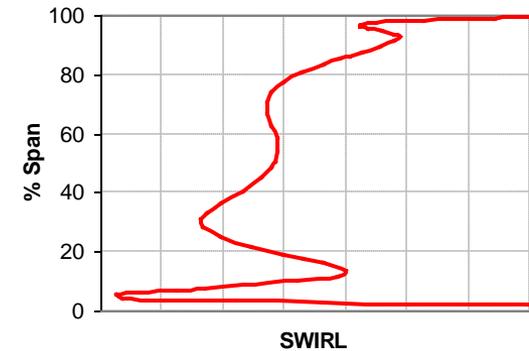
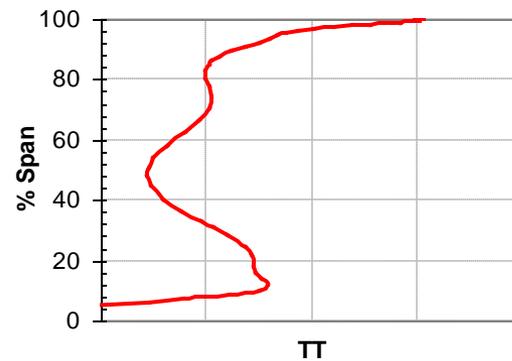
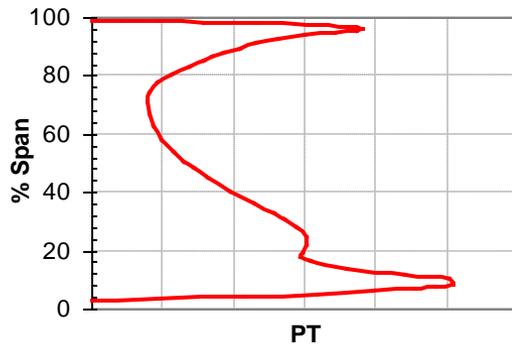


# CFD

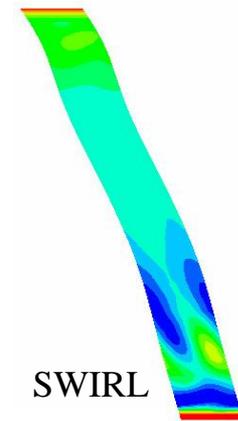
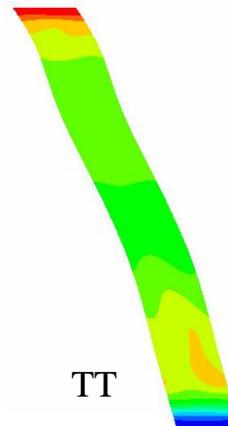
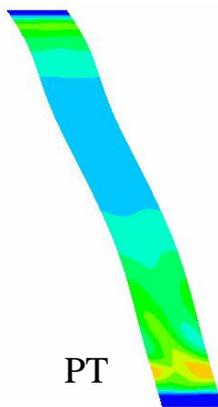
- TACOMA: In-house 3D CFD program used to solve structured or unstructured grid non-linear and linear Euler/Navier-Stokes equations for turbomachinery blade rows.
- Capable of evaluating mixed-out profiles and contour plots at any location of the flowpath

## CFD Output

- Mixed-out profiles

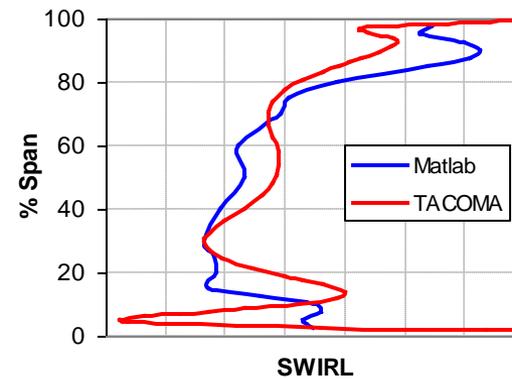
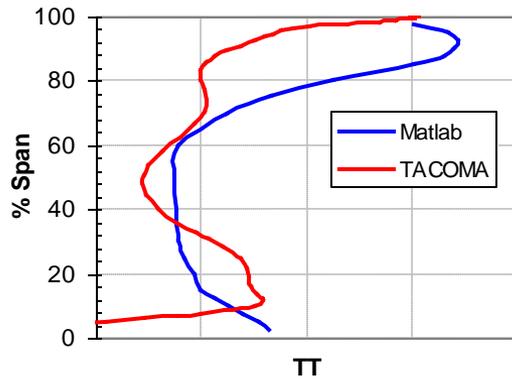
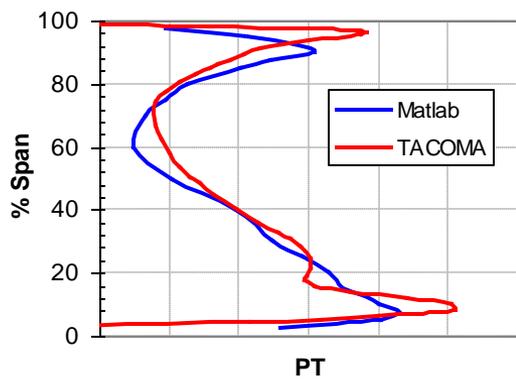


- Contour Plots

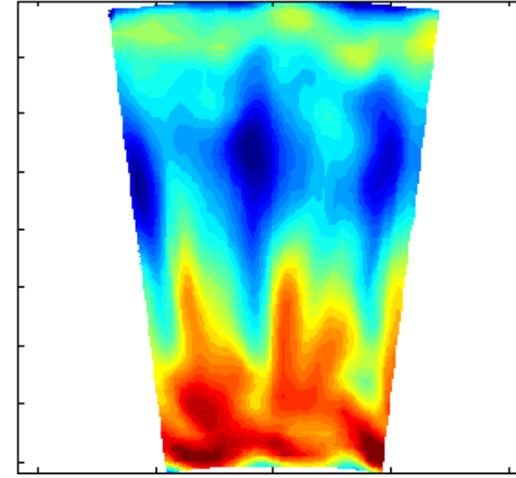
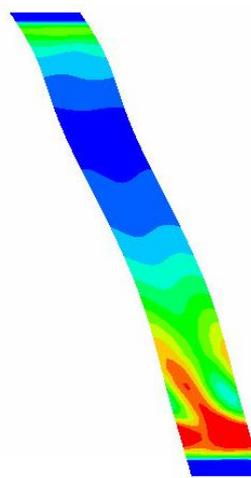
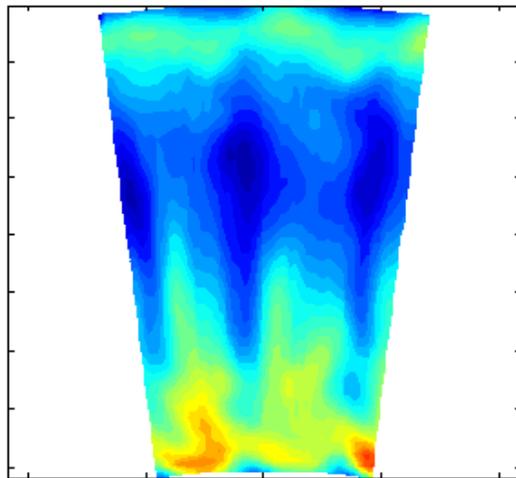
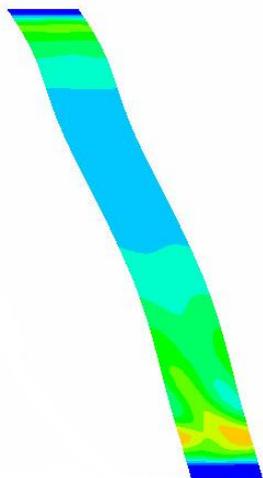


# DATA Vs. CFD

- Mixed-out profiles



- Contour Plots



PT

VZ

# RESULTS

- Mixed-out profiles
  - Pressure profile – shows a similar relationship between both cases; however, there are some discrepancies at the hub and tip of the airfoil.
  - Differences might be caused by shock losses due to secondary and/or cooling flows, which could be captured differently between the data and the CFD model.
  - Temperature profile is significantly different, which suggests that it's due to source term modeling.
  - Pre-test CFD program needs to be updated with correct flow levels, boundary conditions, source terms, and post-test configurations.
  
- Contour Plots
  - Use different frames of reference. Hence, plots do not portray the same information
  - Similar characteristics
  
- Future work - Redefine source term modeling and boundary conditions in TACOMA to match the secondary and cooling flows. The expectation is to match the CFD to the data and set a CFD model benchmark to be used for future gas turbine designs.



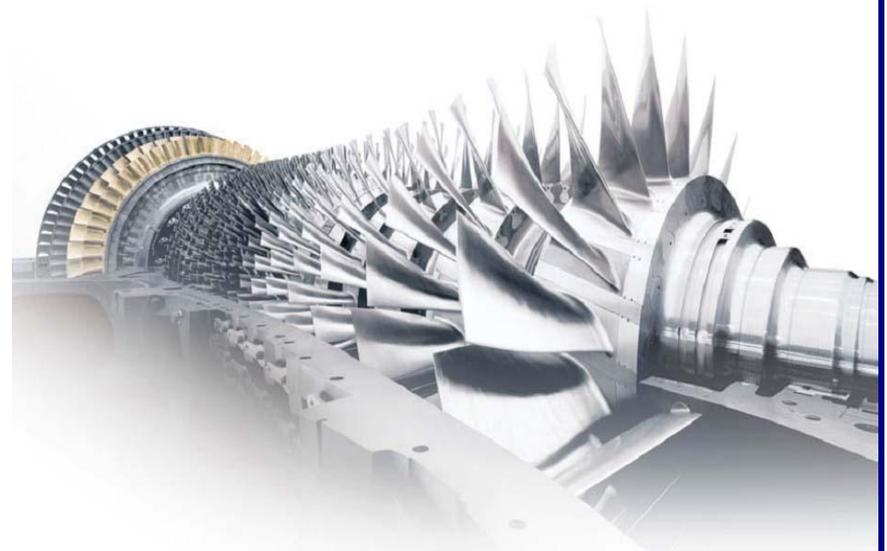
# AIRFOIL SURFACE FINISH

- CFD – Capable of modeling surface roughness but set-up to assume smooth airfoils
- SURFIN – Computer-aided tool that estimates surface roughness effects much faster than CFD

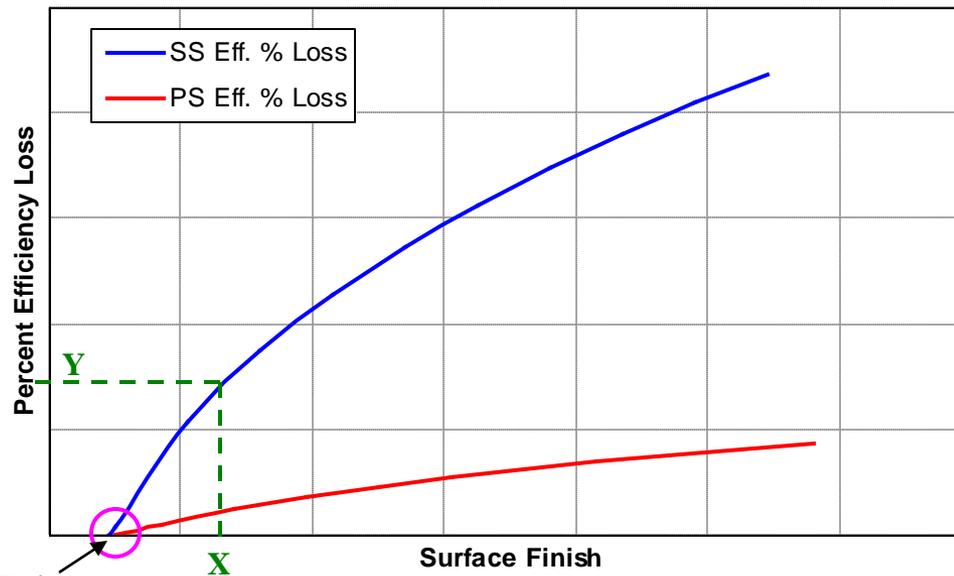
Objective: Perform a surface finish analysis of turbine airfoils for the NGGT rig using a computer-aided tool (SURFIN)

## SURFIN Program Concept:

- Design tool for estimating the effects of surface finish on airfoil performance
- Calculates surface roughness (RMS) – Causes increase in drag generating efficiency losses
- Determines Hydraulically Smooth point – Max. RMS that causes no losses



## SURFIN Results:



Hydraulically Smooth Point

- Airfoils need to be polished to an RMS equal to the Hydraulically Smooth point
- If the manufacturing process can't produce such RMS, then it is necessary to know how much efficiency is lost to adjust the CFD performance estimate
- For instance, given a measured RMS of  $x$ , the loss in efficiency corresponds to  $y$  points in the bladerow efficiency
- Typically, a small decrease in the overall turbine efficiency can increase the operating cost of an engine by thousands of dollars per hour.

# CONCLUSION

- The developed 1:5 scaled prototype of an actual power generation turbine and its instrumentation demonstrated good response to the testing operating conditions and were able to generate the data necessary to conduct research and validate and verify the advanced 3D CFD design tools and methodologies used to design gas turbines. Refine a Matlab script that can be used to determine a mixed-out profiles of pressure, temperature, velocity, and swirl at different locations of the flowpath
- The analyzed data and pre-test CFD solution depict discrepancies that can be attributed to the modeling of source terms, boundary conditions, shock losses, leaks, pre-test assumptions, and other phenomena. Hence the need to update the pre-test CFD model to validate it and provide a 3D CFD model benchmark
- Due to proprietary limitations, not all the work that was performed on the NGGT turbine aero rig or other projects could be discussed
- The UTSR Fellowship is an excellent program for all students with interest in the gas turbine industry. The experiences learned through the 10-12 week program can only be learned in an industrial work environment such as the one provided by the UTSR Industrial Sponsors

