Combustor Wall Cooling with Dirt Mitigation and a Combustor Simulator
Project 68

Lead investigator: Reid Berdanier, Pennsylvania State University
Project manager: Joshua Glottmann, FAA

October 16, 2025
Alexandria, Virginia

This research was funded by the U.S. Federal Aviation Administration Office of Environment and Energy through ASCENT, the FAA Center of Excellence for Alternative Jet Fuels and the Environment, project ASCENT 68 through FAA Award Number 13-C-AJFE-PSU-132 under the supervision of Joshua Glottmann. Any opinions, findings, conclusions or recommendations expressed in this material are those of the authors and do not necessarily reflect the views of the FAA.

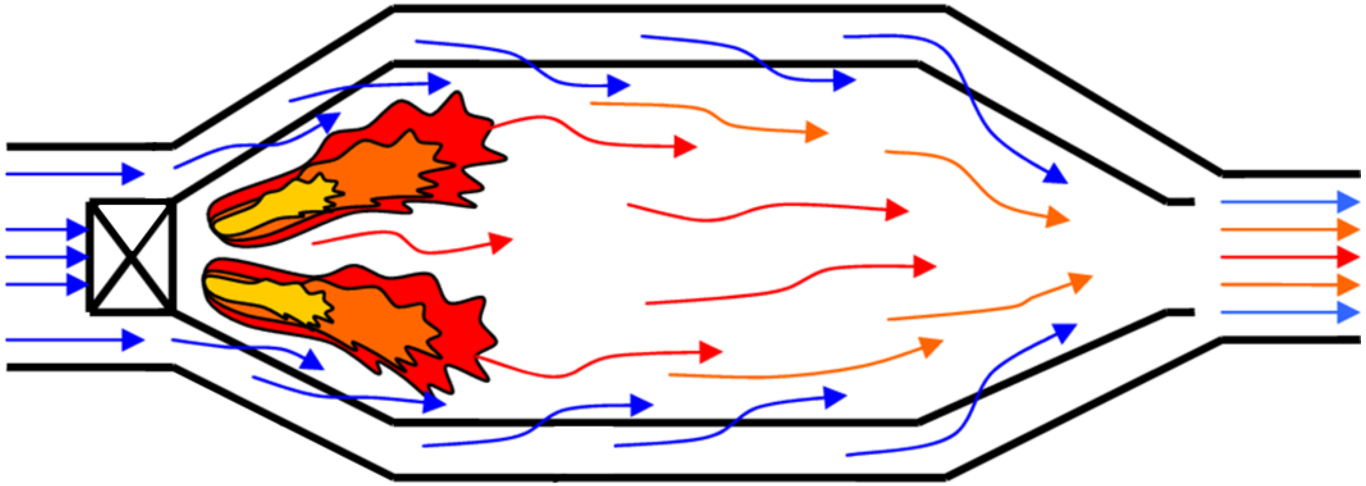
ASCENT

Progress Summary for FAA ASCENT #68:

Combustor Wall Cooling with Dirt Mitigation and a Combustor Simulator

Fabrizzio Vega, Chad Schaeffer, Reid Berdanier, Karen Thole, Stephen Lynch, Michael Barringer

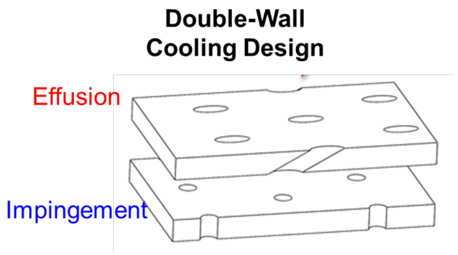
Department of Mechanical Engineering

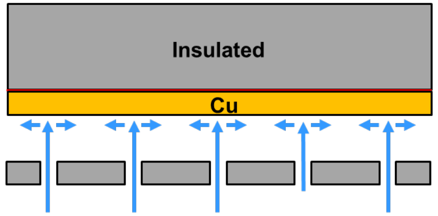


The objective is to understand durability impacts related to the combustor

Focus 1

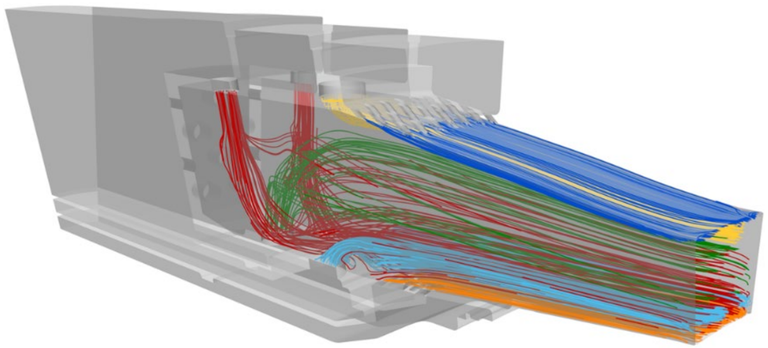
Ingestion of dirt diminishes combustor wall cooling. We will explore how dirt affects the cooling capabilities of a double-walled liner.





Focus 2

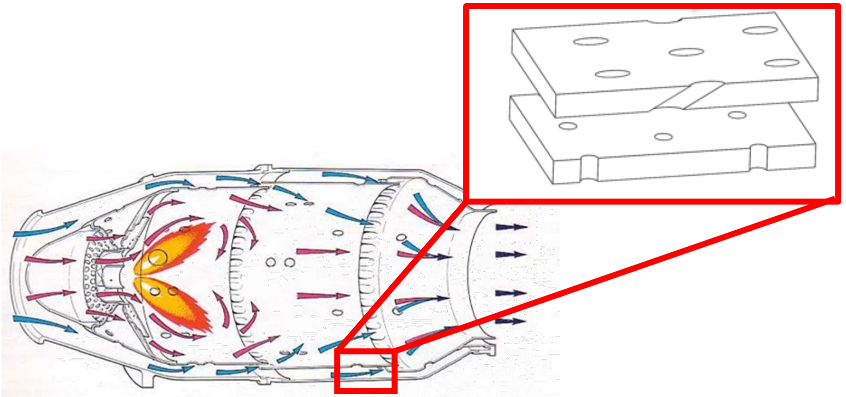
Pressure and temperature profiles exiting the combustor affect efficiency and durability of the high-pressure turbine. We developed a non-reacting profile simulator to produce a range of profiles at elevated turbulence levels entering the START lab turbine test section.

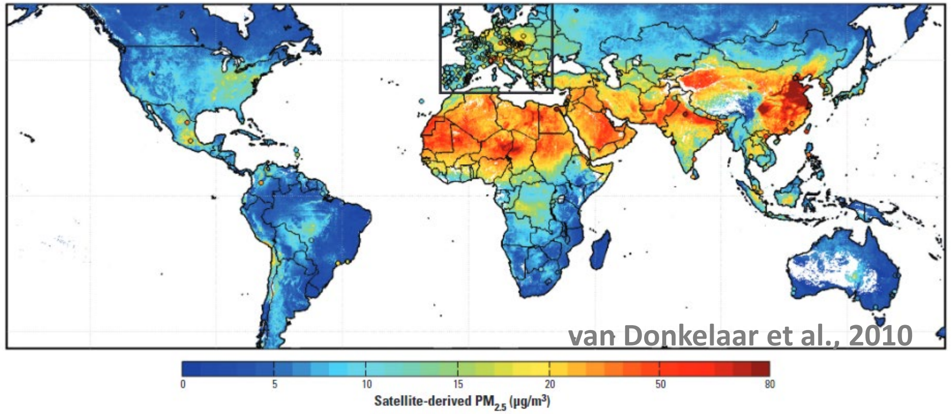


Mitigating the impacts of dirt ingestion using a triple-walled combustor liner design

Fabrizzio Vega,
Karen A. Thole,
University of Michigan
Department of Mechanical Engineering

MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

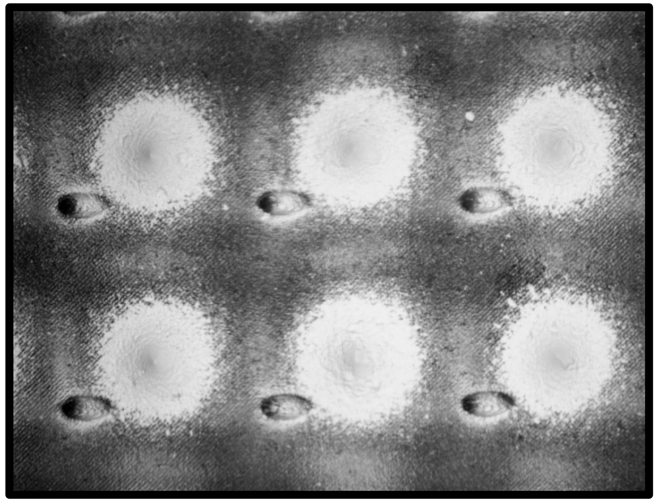




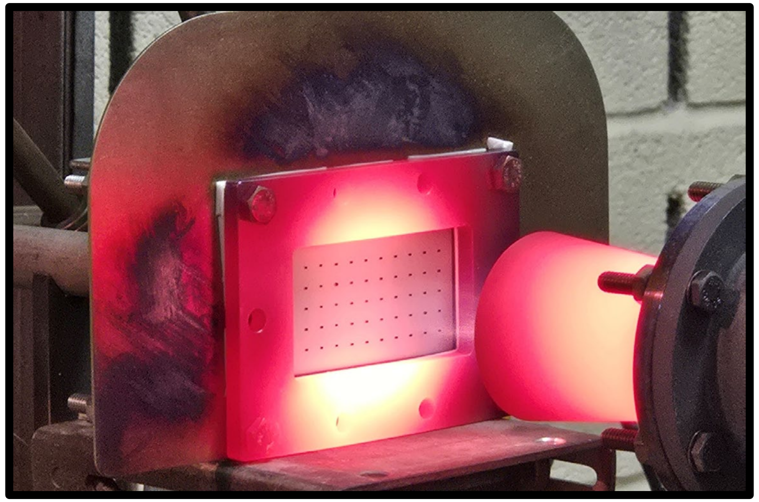
FEDERAL AVIATION ADMINISTRATION

ASCENT
AVIATION SUSTAINABILITY CENTER

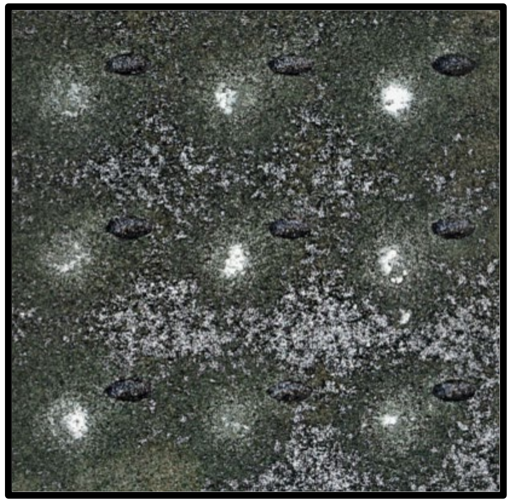
A new high-temperature test rig was created to quantify the mitigating effects of an additional combustor liner wall on dirt deposition



Room-temperature deposition

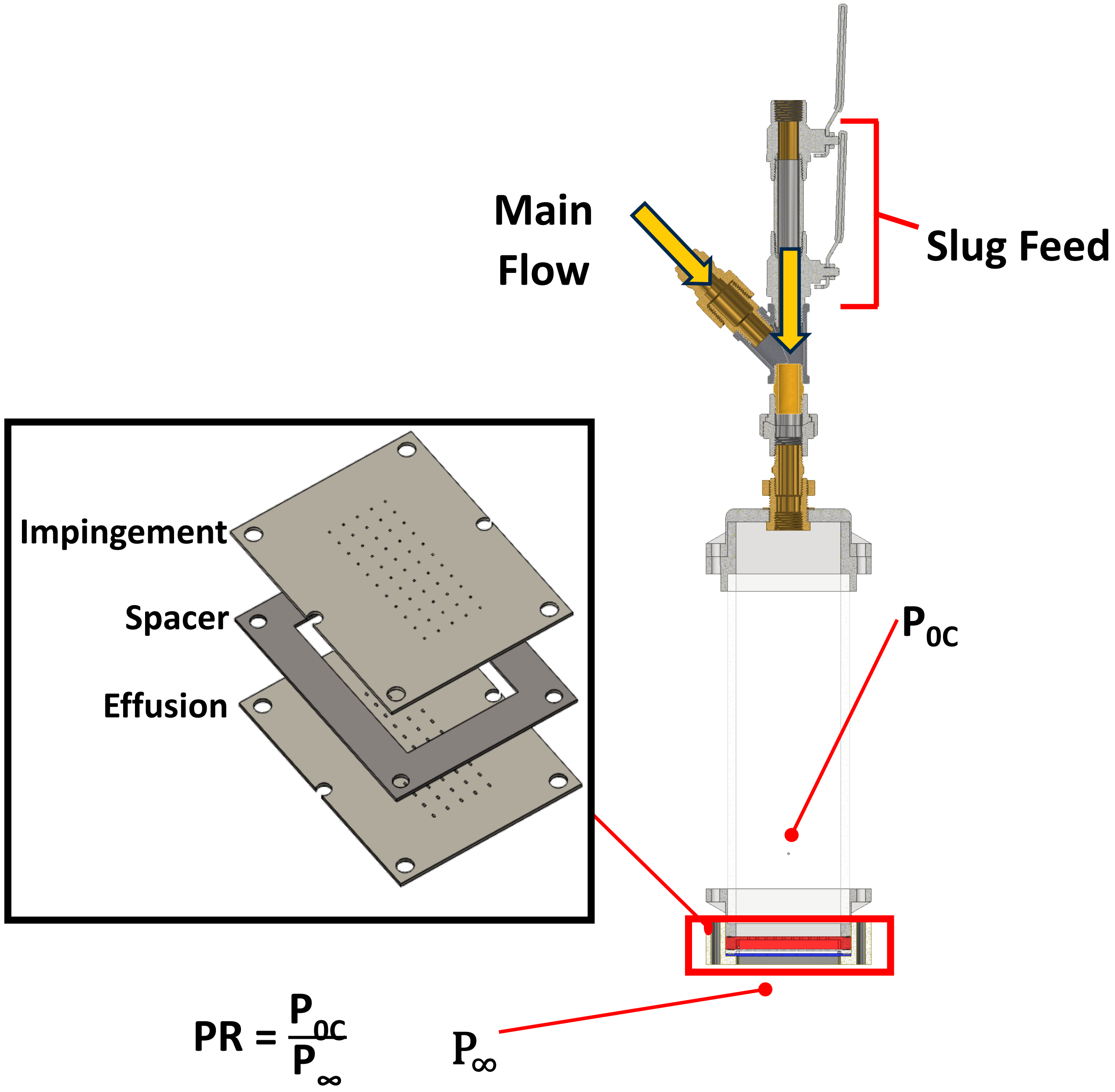


High-temperature facility

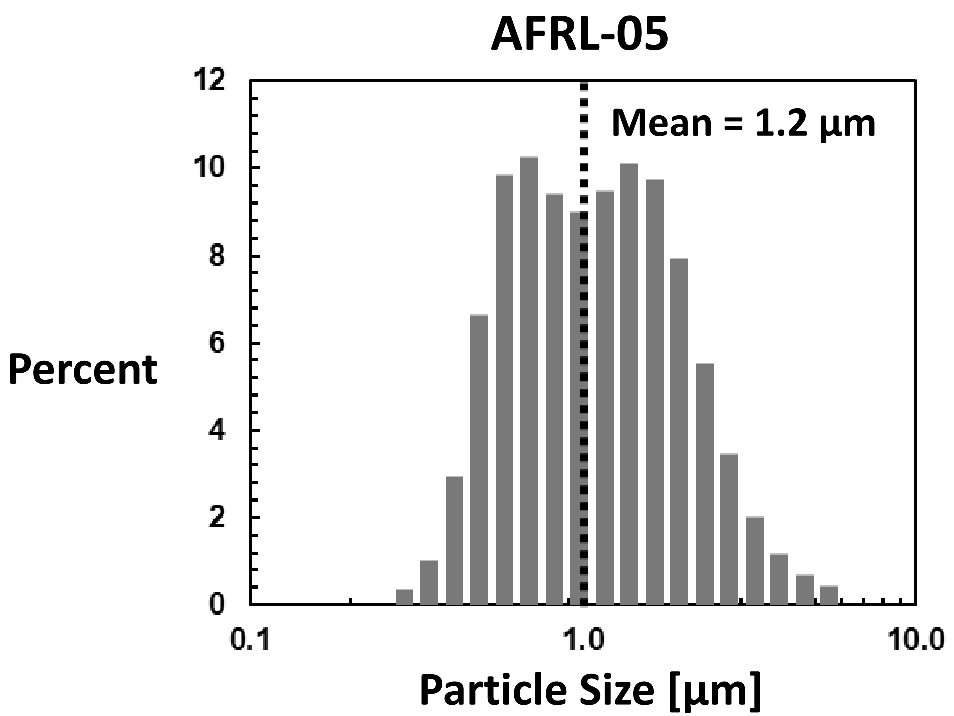


High-temperature deposition

The room temperature tests use a slug feed for injecting AFRL-05 dirt



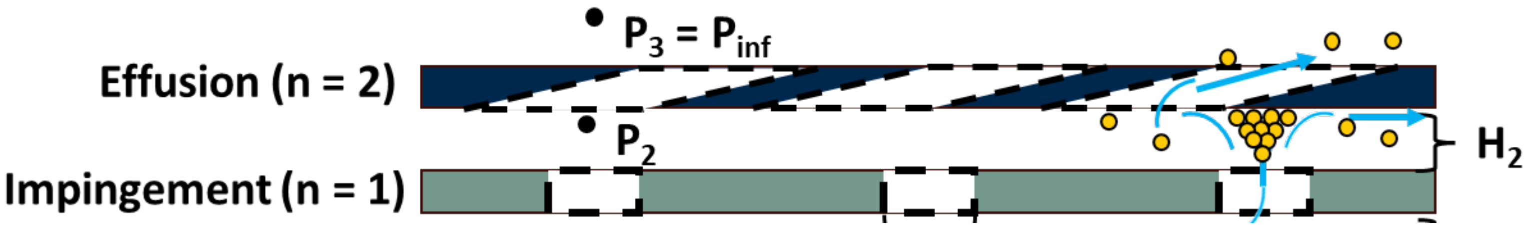
$$PR = \frac{P_{0C}}{P_\infty}$$



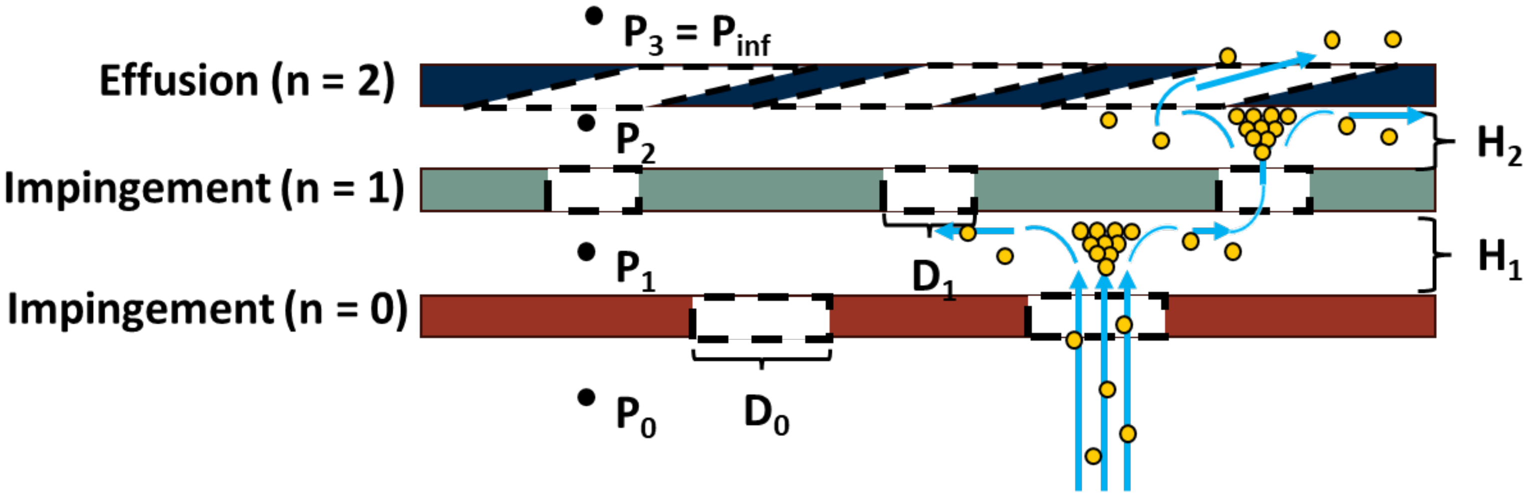
MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

The triple-wall liner was constructed by adding an upstream impingement plate

Typical Double-Wall Combustor Liner Design

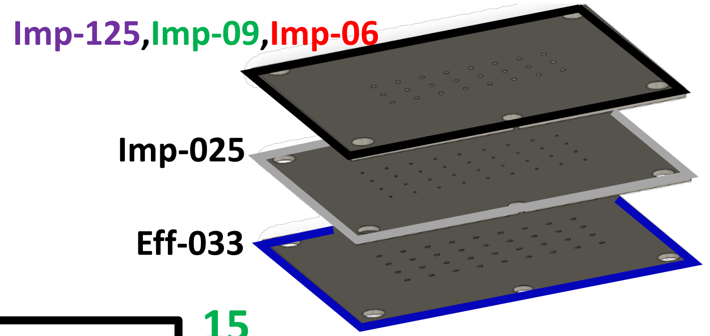


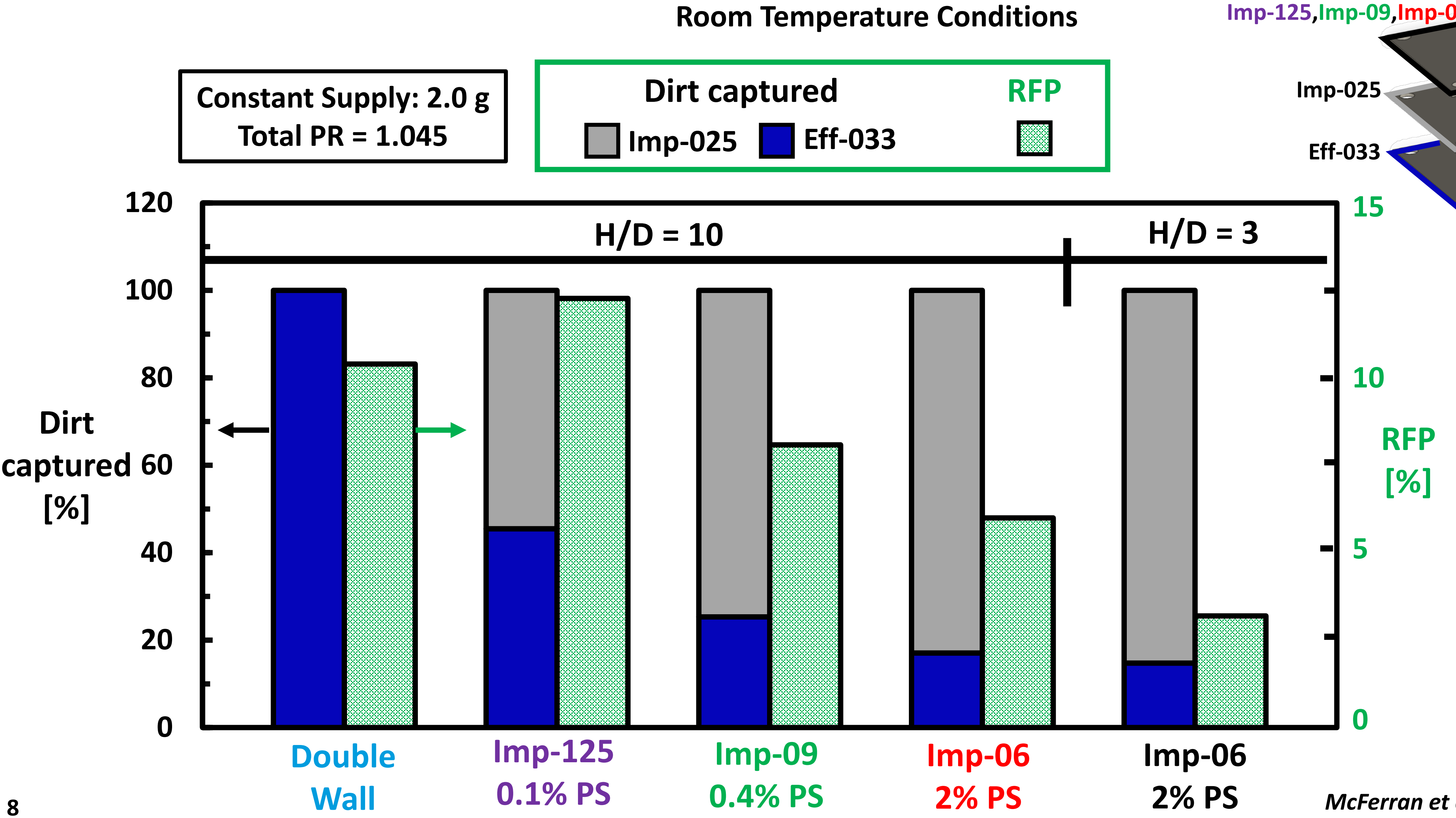
New Triple-Wall Combustor Liner Design



MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

Increasing the first plate pressure split and decreasing the plate-to-plate spacing between the first two plates reduced deposition on the effusion plate by 85%

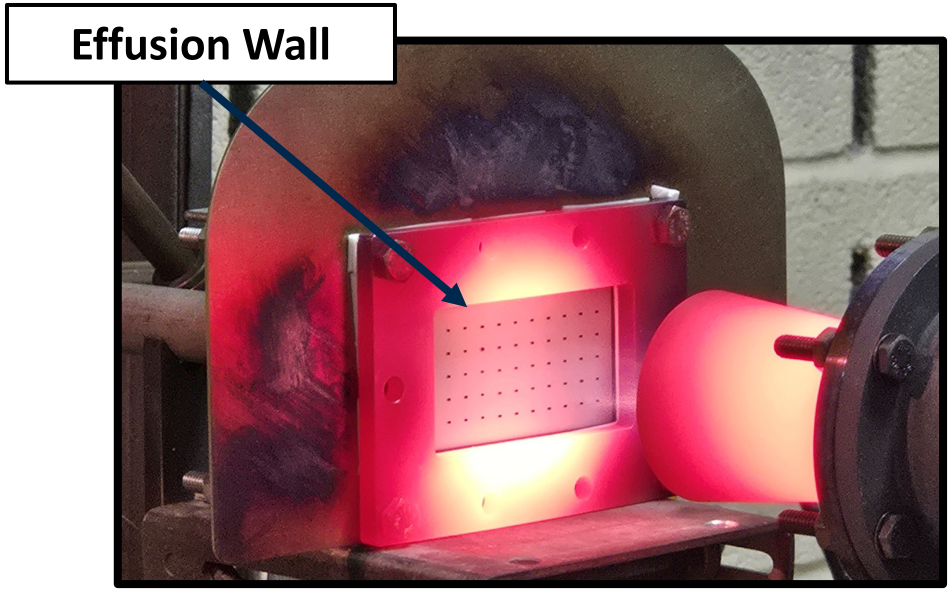


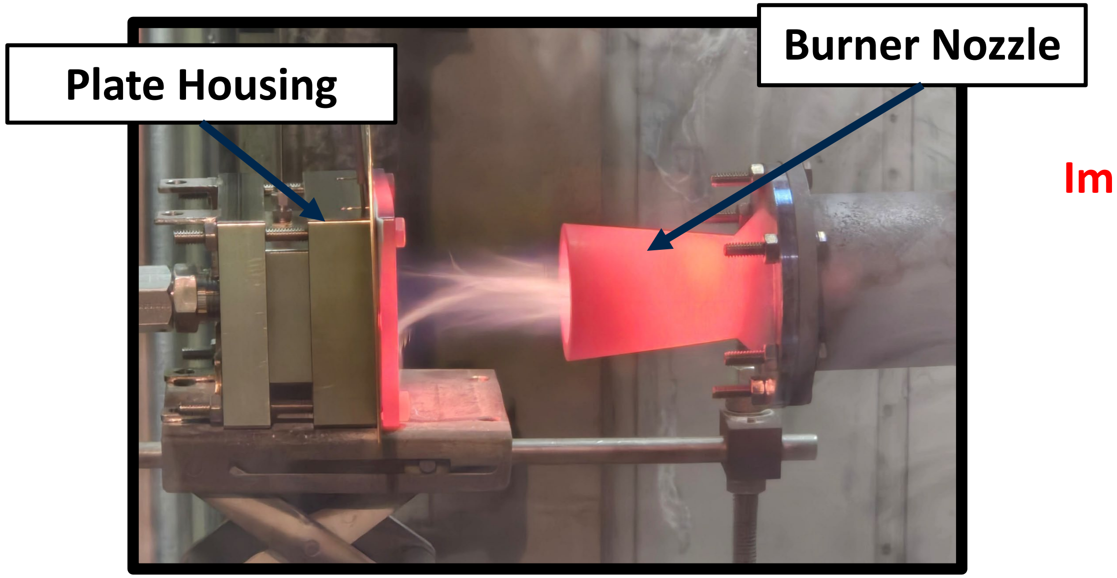


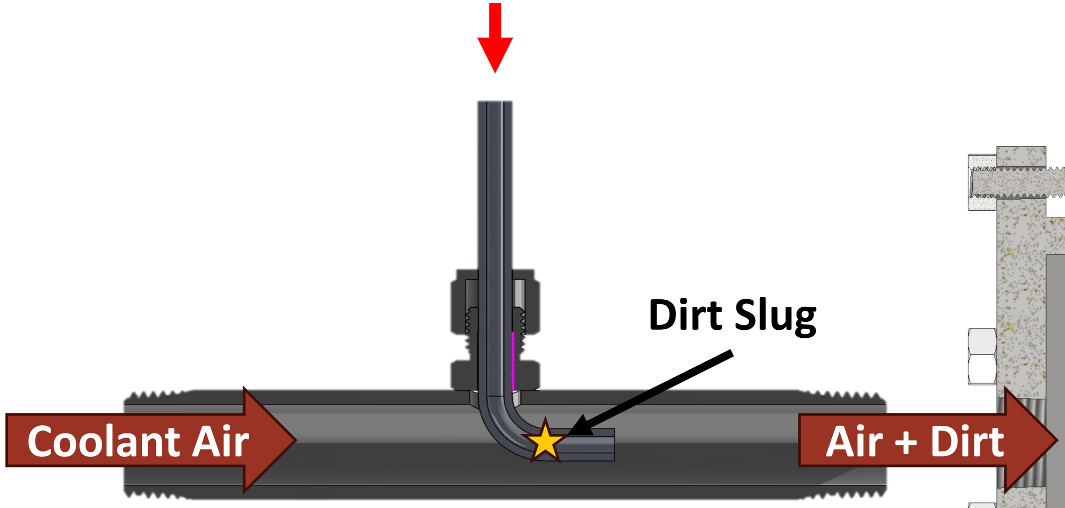
McFerran et al. 2025

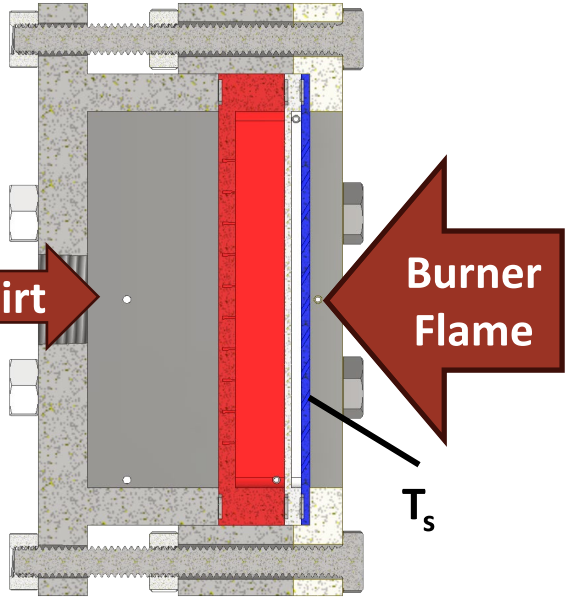
MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

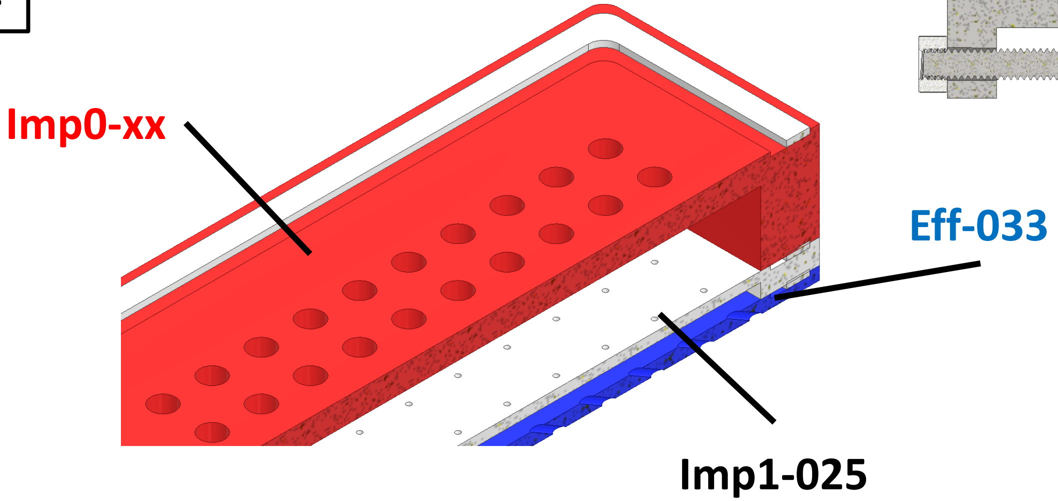
In collaboration with NASA Glenn, the construction of a high-temperature deposition testing facility is complete and features a dirt slug injection system



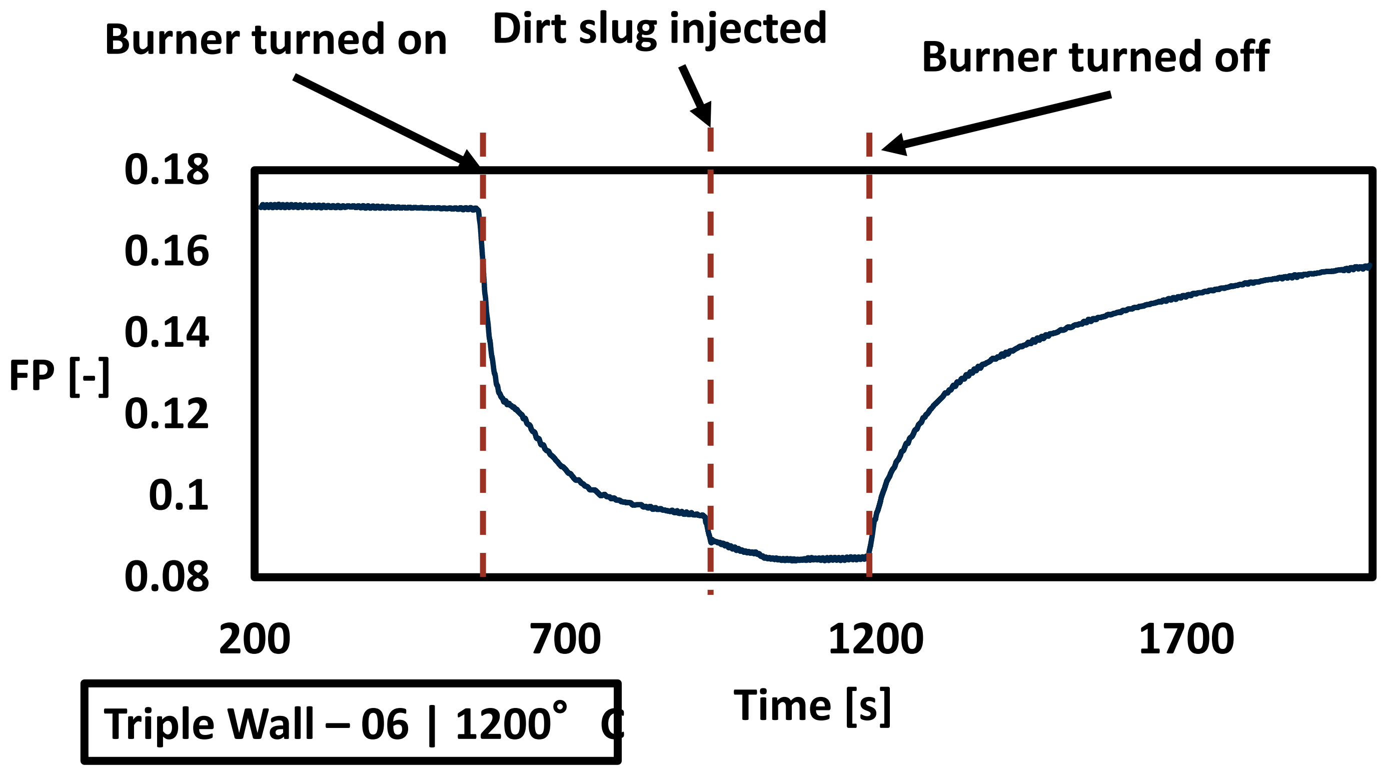


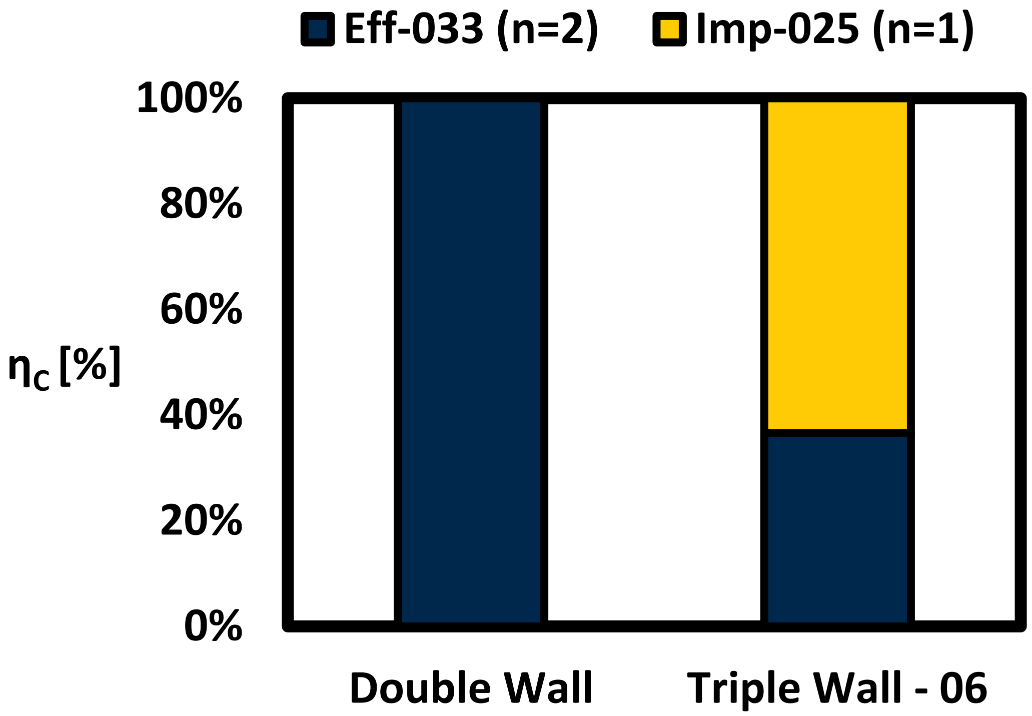






The addition of a third wall is observed to reduce deposition on the effusion plate by 63%



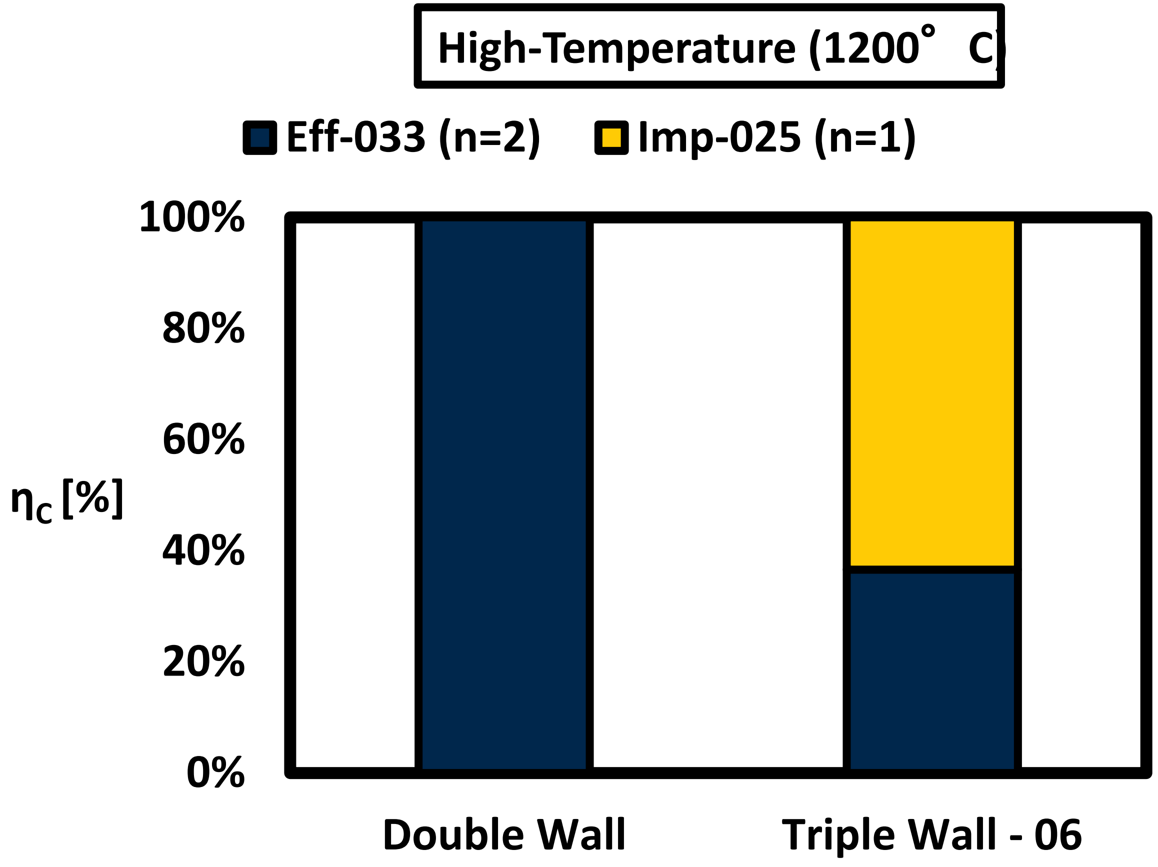


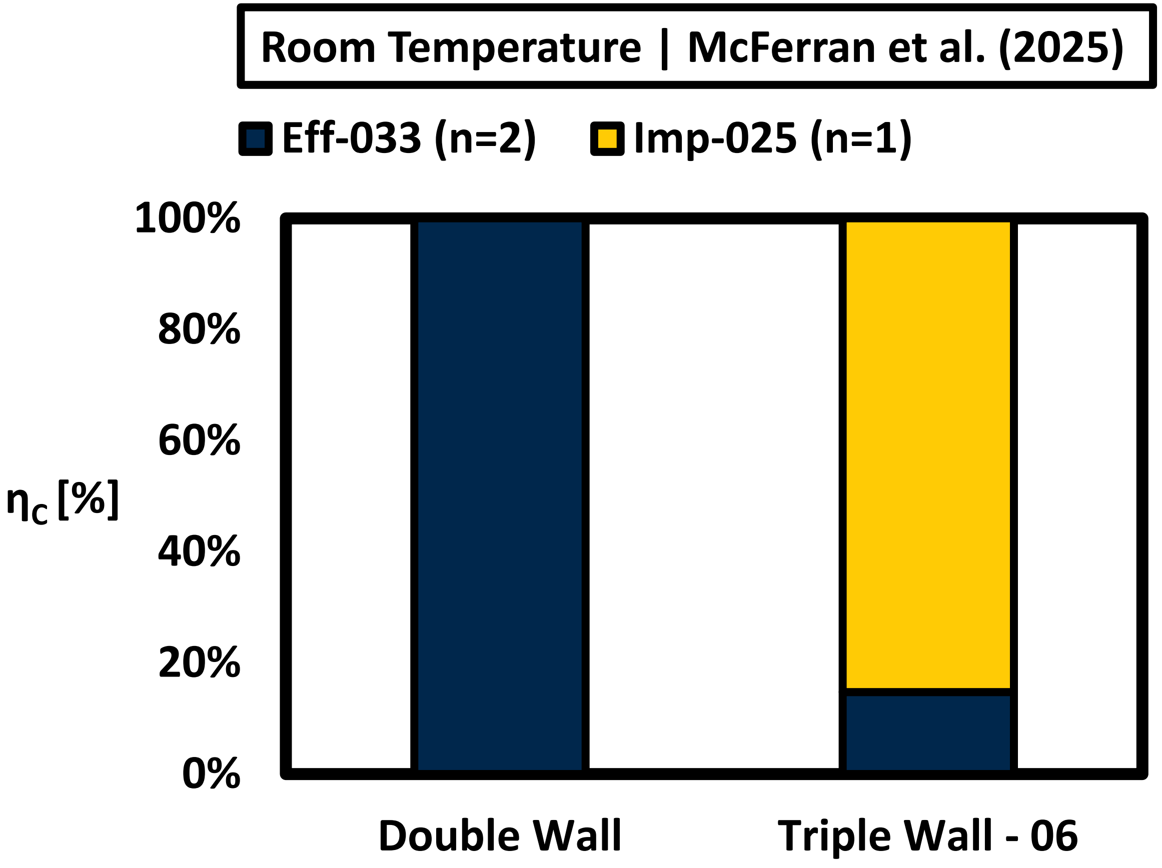
$$FP = \frac{4\,\dot{m}\sqrt{RT_{0C}}}{\pi P_{0C} N D_1^2}$$

$$\eta_{c}\,(\%) = \frac{\text{dirt captured}}{\text{total dirt captured}}$$

MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

Comparable reductions in dirt captured on the effusion wall are observed between hot and cold tests

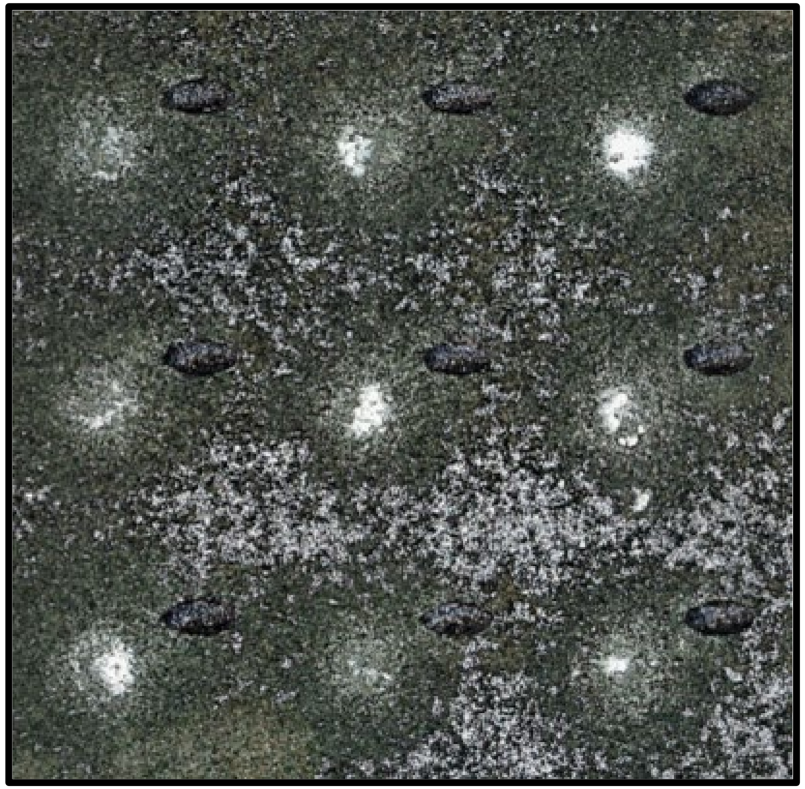




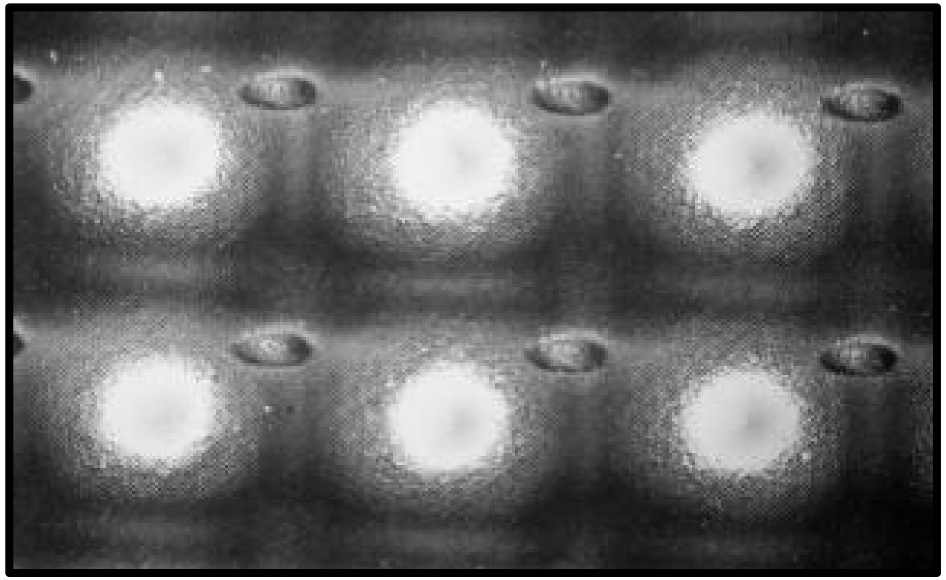
MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

For both heated and room temperature tests, the highest peaks correspond to where the adjacent jets stagnate

Triple Wall -06 (1200° C)

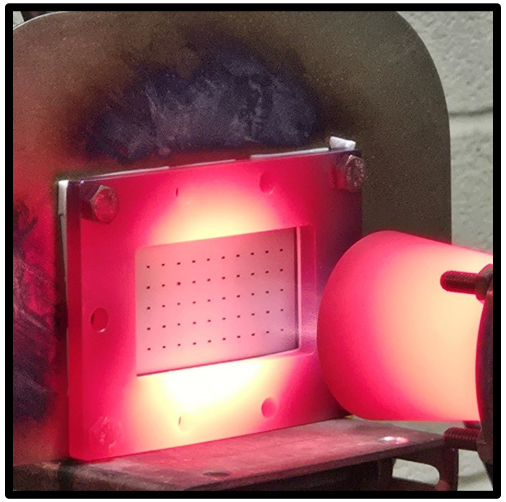


Triple Wall - 06, Room Temperature | McFerran et al. 2025

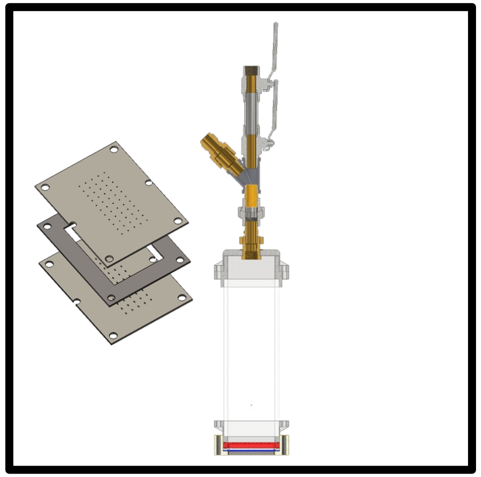


MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

The next steps for this project include more heated tests to complete the test matrix and to repeat room temperature testing



Continue high temperature deposition tests at NASA Glenn using double and triple wall geometries

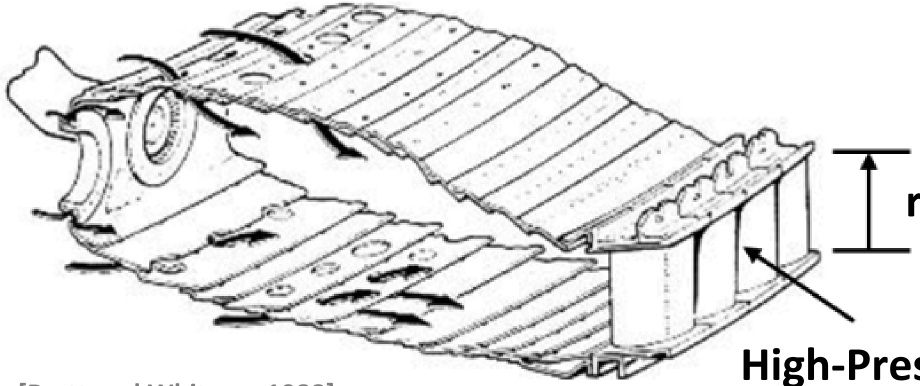


Repeat high temperature tests at room temperature using the same combustor liner geometries as for high temperature tests

MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

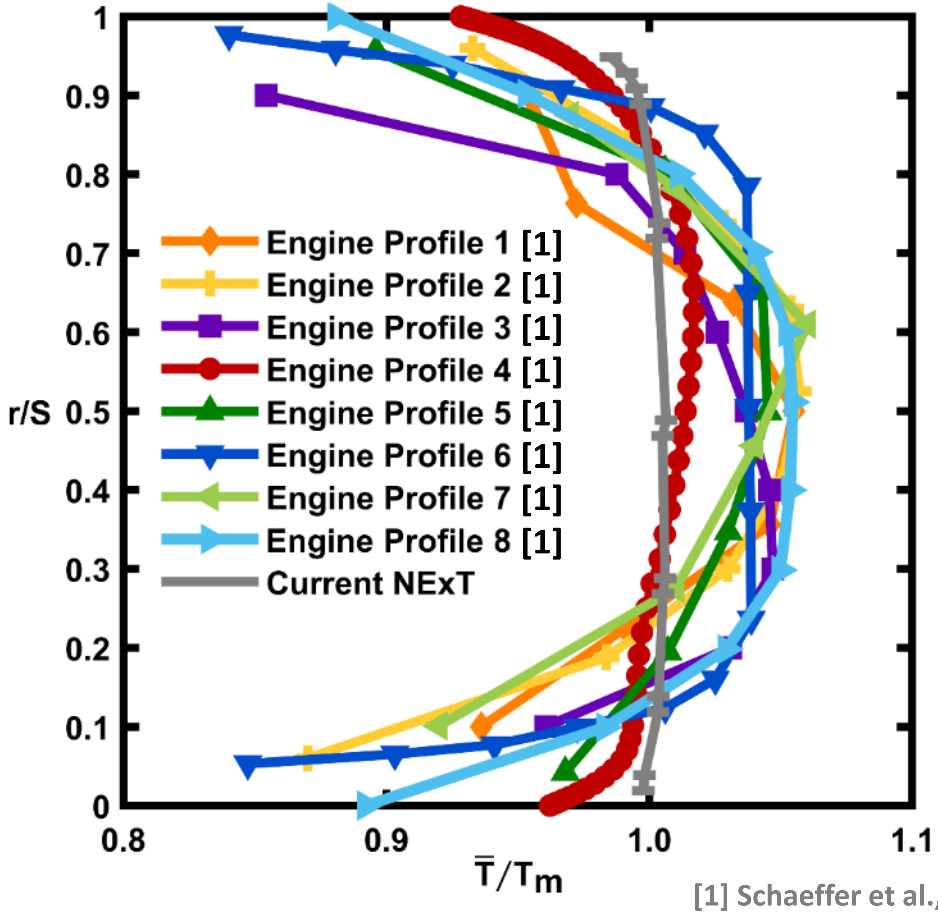
A range of profiles from literature were considered as being representative of those exiting modern engine combustors

Engine Combustor



[Pratt and Whitney, 1988]

Radial Temperature Profiles
At Combustor Exit

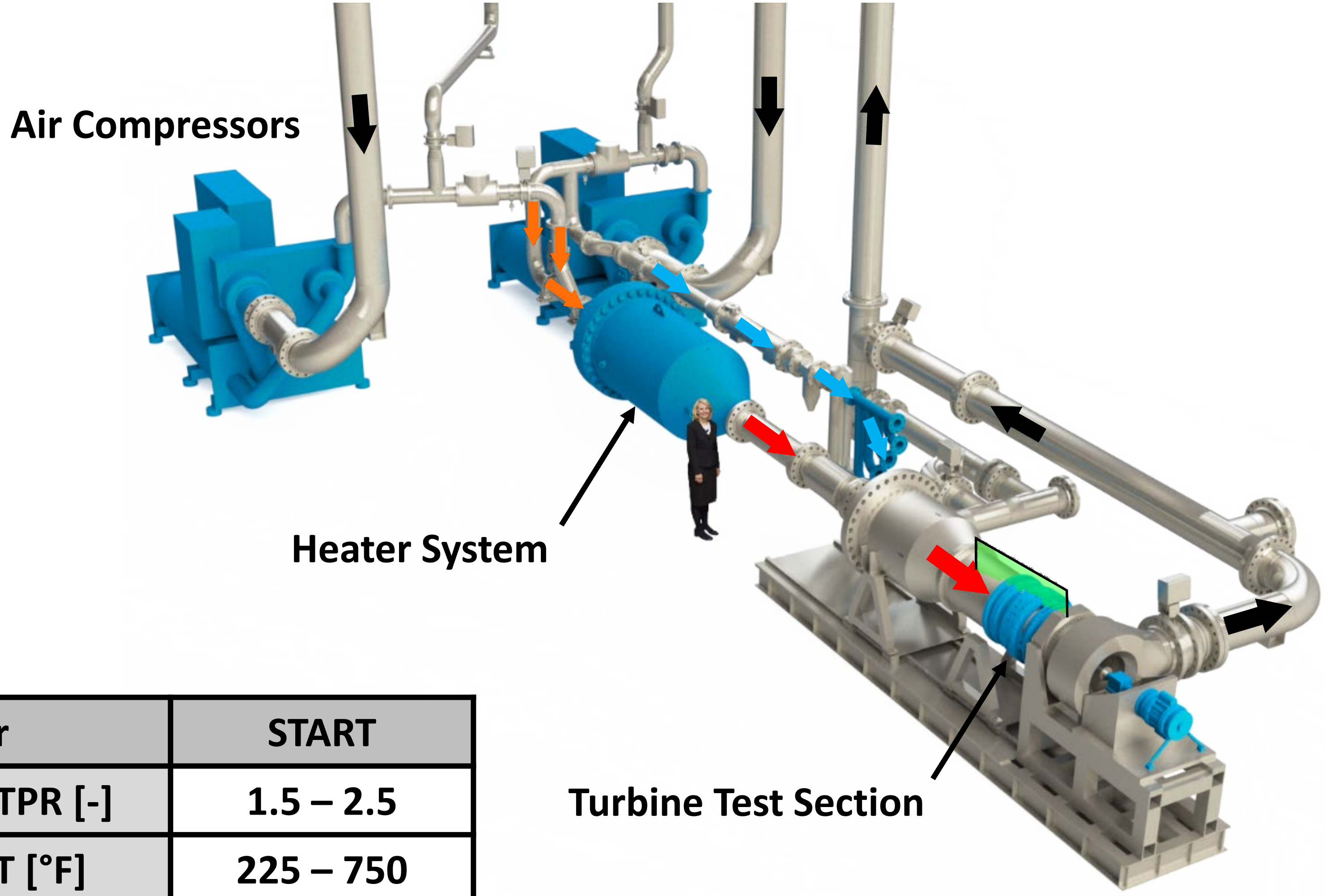


[1] Schaeffer et al., 2025

$\bar{T}$ = Circumferentially-averaged total temperature

T_m = Mass-averaged or area-averaged total temperature

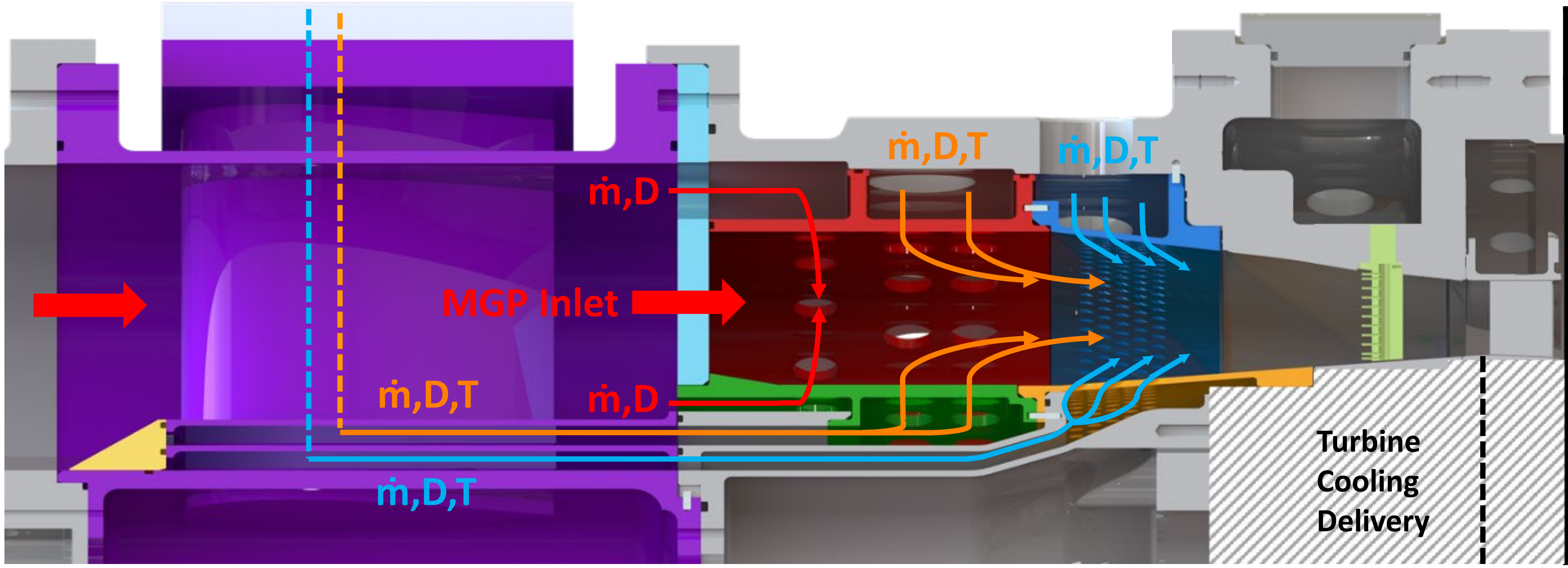
The START lab is a continuous-duration test facility with a single-stage turbine capable of replicating engine-relevant conditions using real engine hardware



Parameter	START
Pressure Ratio, TPR [-]	1.5 – 2.5
Temperature, T [°F]	225 – 750
Flowrate, ṁ [lbm/s]	25
Rotational Speed, Ω [rpm]	≤ 11,000

The simulator design was completed that incorporates a variety of control knobs that can be turned to study their effect on exit profile shape

Combustor Profile Simulator   360° Traverse



Turbine Vane Inlet →

	Parameter	Range
Row 1	$\dot{m}_1 / \dot{m}_t$ [%]	18-22*
	D_1 [in.]	0.3-0.7
Row 2,3	$\dot{m}_{2,3} / \dot{m}_t$ [%]	18-22*
	$D_{2,3}$ [in.]	0.6-0.9
	$T_{2,3}$ [°F]	40-220
Effusion	$\dot{m}_e / \dot{m}_t$ [%]	7.5-13*
	T_e [°F]	40-220

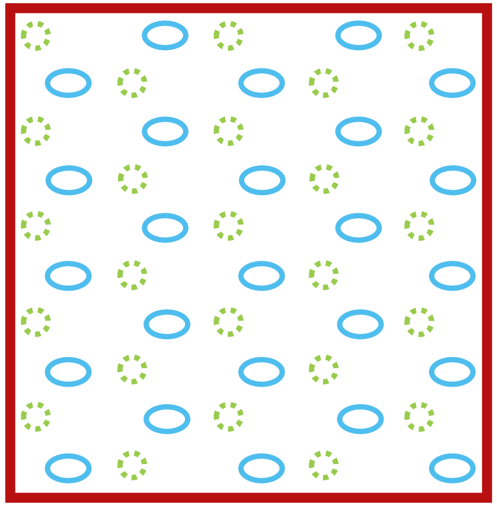
D = Diameter
ṁ = Mass flow rate
T = Temperature
1, 2, 3 = Dilution row
e = Effusion

Simulator Air Temperatures:
- Hot: 450-625° F
- Warm: 200-250° F
- Cold: 40° F

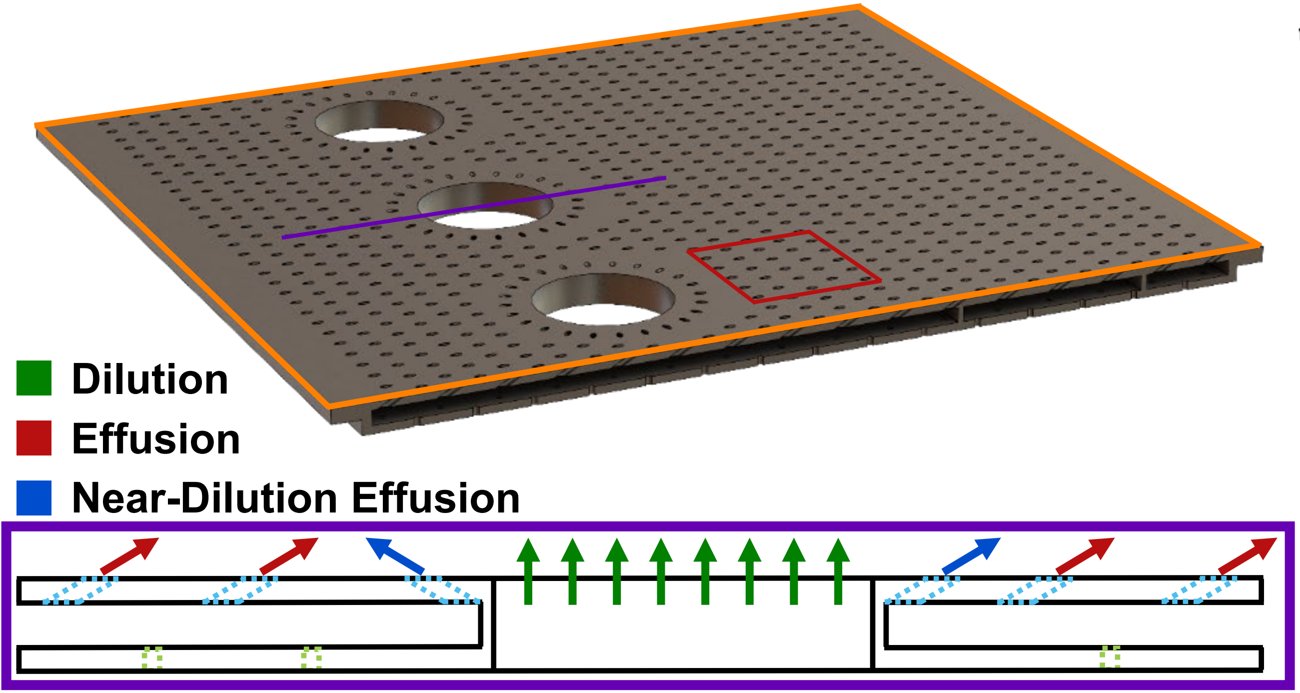
Main Gas Path (MGP) Inlet ≈ 50% of Total Flow Exiting Simulator

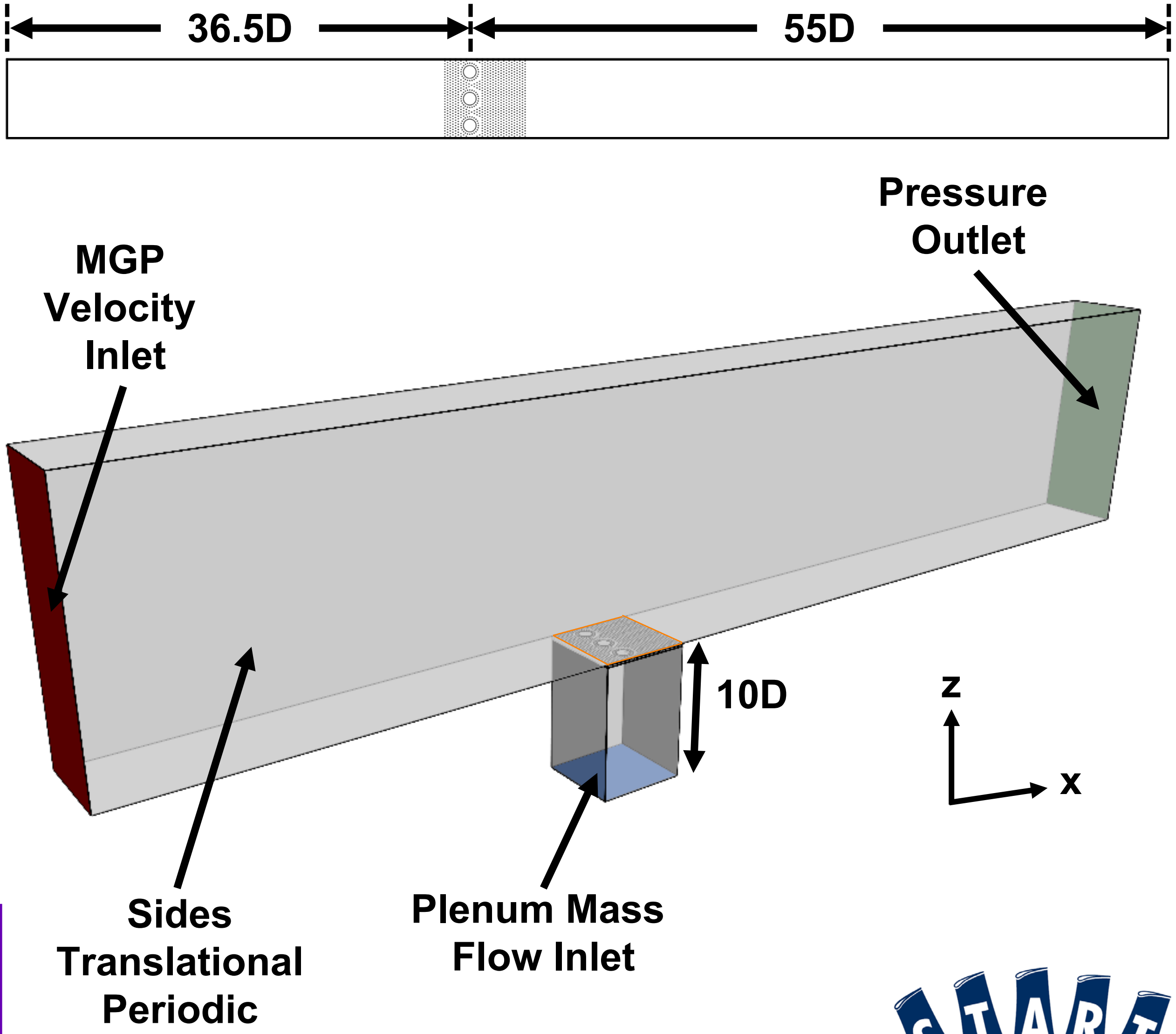
Publication From This Work
Schaeffer et al. (2024)
Journal of Turbomachinery
Turbo-24-1190

Computational benchmarking was completed using a relevant combustor liner geometry and experimental data from the study by Shrager et al. (2018)

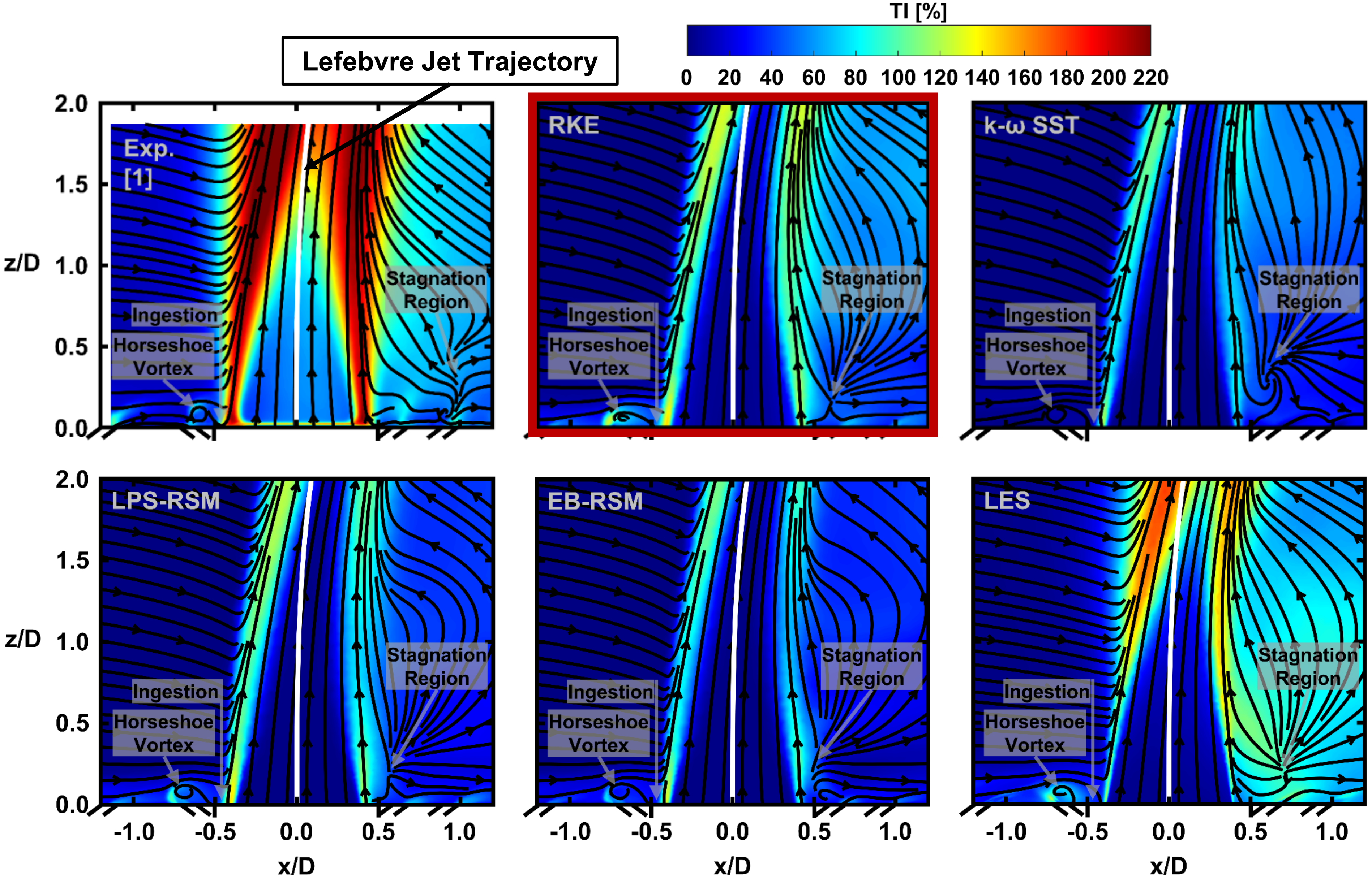


- Outward Effusion Design
- Inlet Turbulence = 0.5%
- I_{dil} = 30



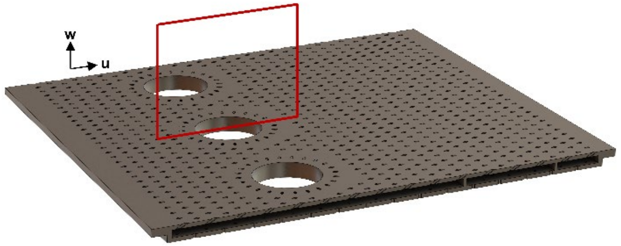


Four different RANS models and LES were compared to PIV measurements that showed underprediction of penetration depth and turbulence intensity



$$\mathrm{TI} = \frac{\sqrt{\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right)/3}}{U_\infty}$$

- Outward Effusion Design
- Inlet Turbulence = 0.5%
- I_{dil} = 30

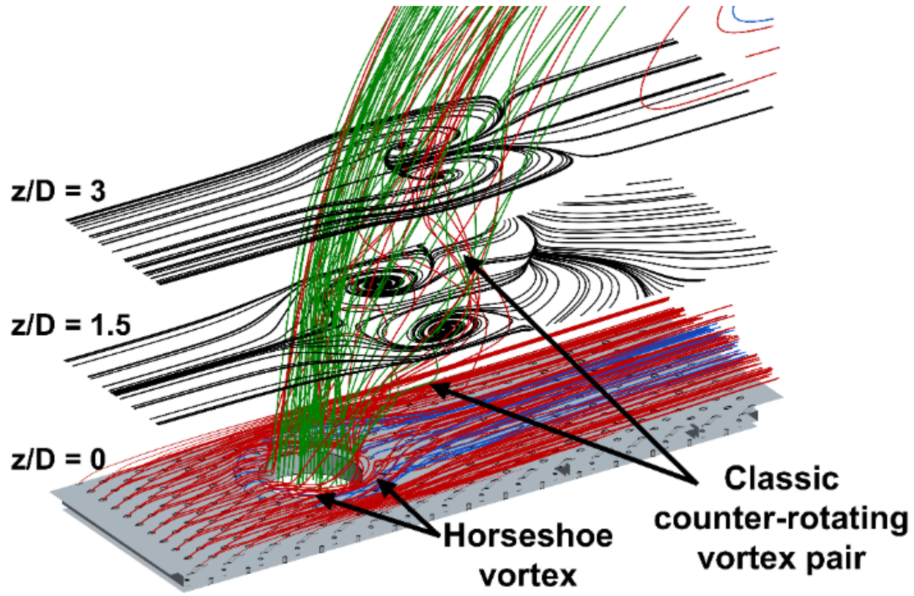


* New Publication Written
Schaeffer et al. (2025)
Journal of Turbomachinery
TURBO-25-1351 (Under Review)

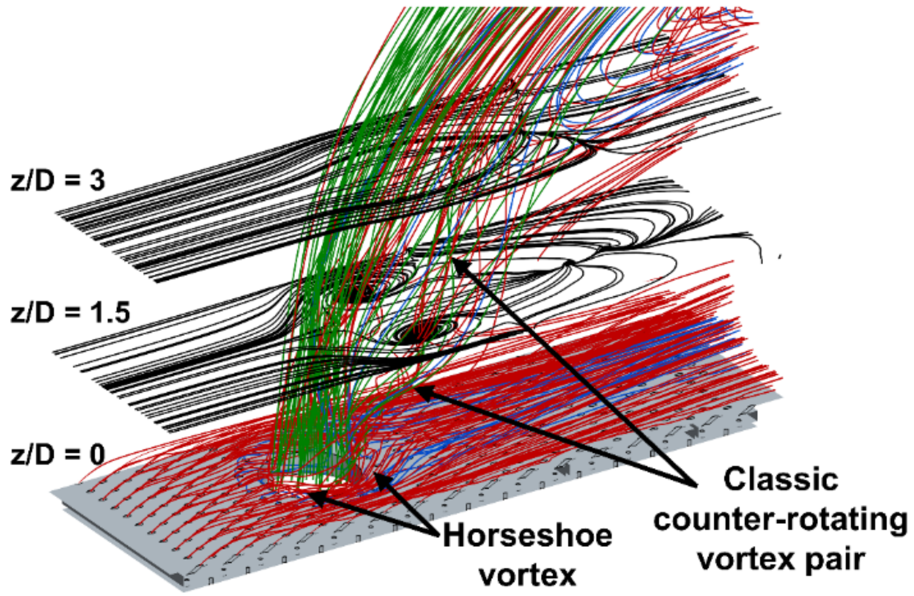
[1] Shrager et al., 2018

Both the RKE and LES models predicted similar vortex structures, but the LES predicted more entrainment and liftoff of effusion flow from the wall

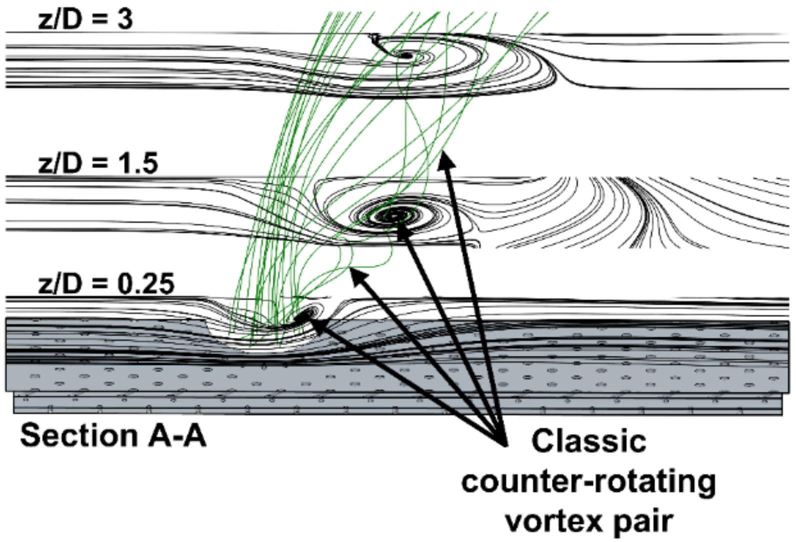
RKE



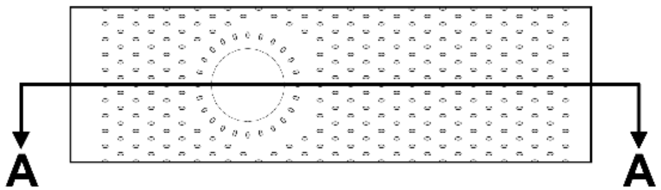
LES

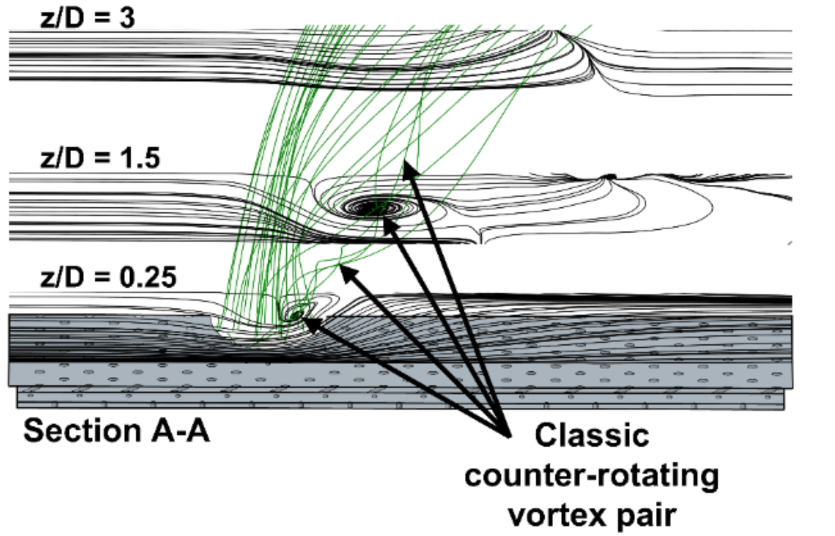


- Dilution
- Effusion
- Near-Dilution Effusion



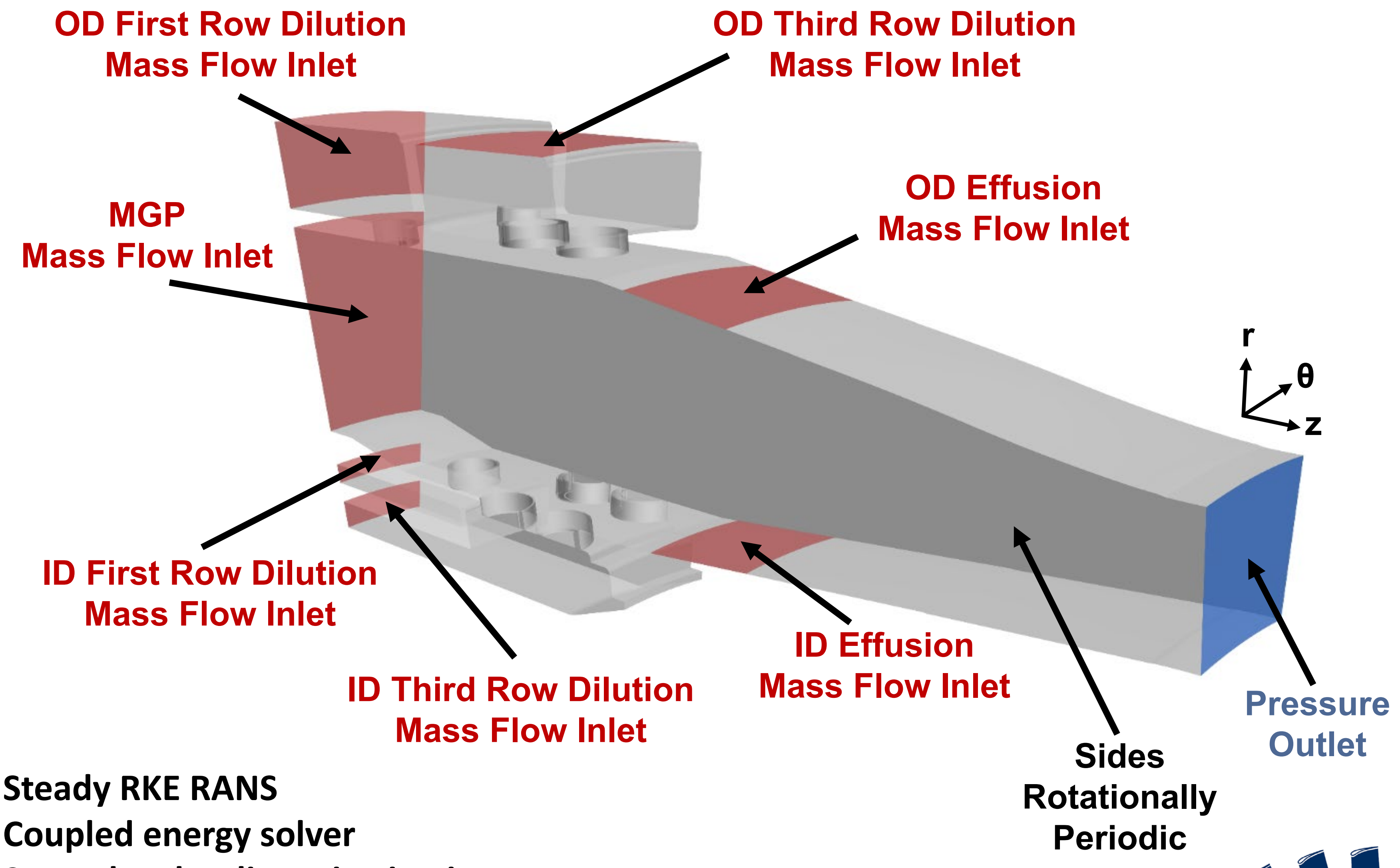
Model	I_{dil}	$\dot{m}_{dil}/\dot{m}_t$
Experiment	29.3	75%
RKE	30.3	76%
LES	28.6	74%





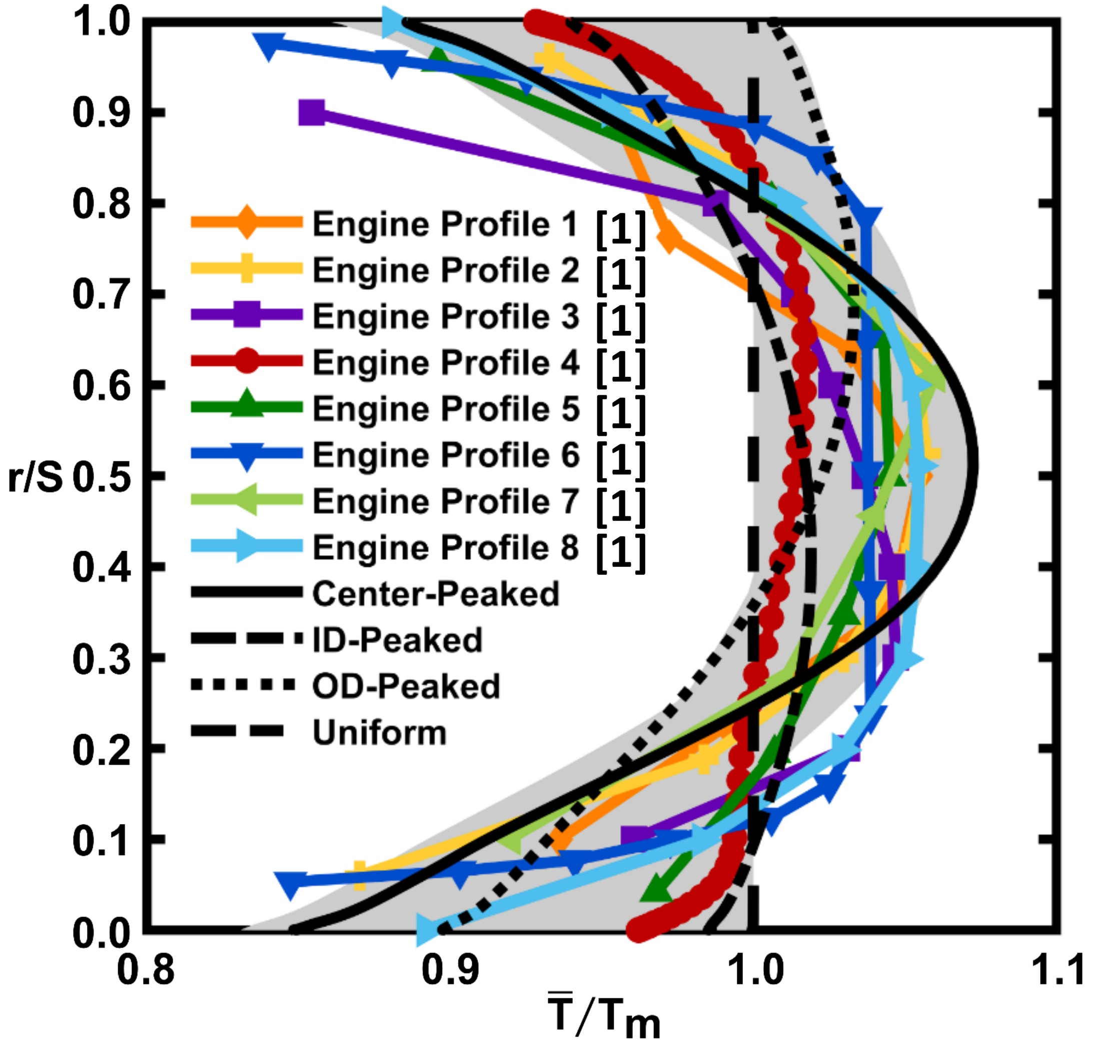
A CFD case matrix was created for the simulator geometry using a 2-level 12-factor 'Design of Experiments' and was studied using steady realizable k-ε RANS

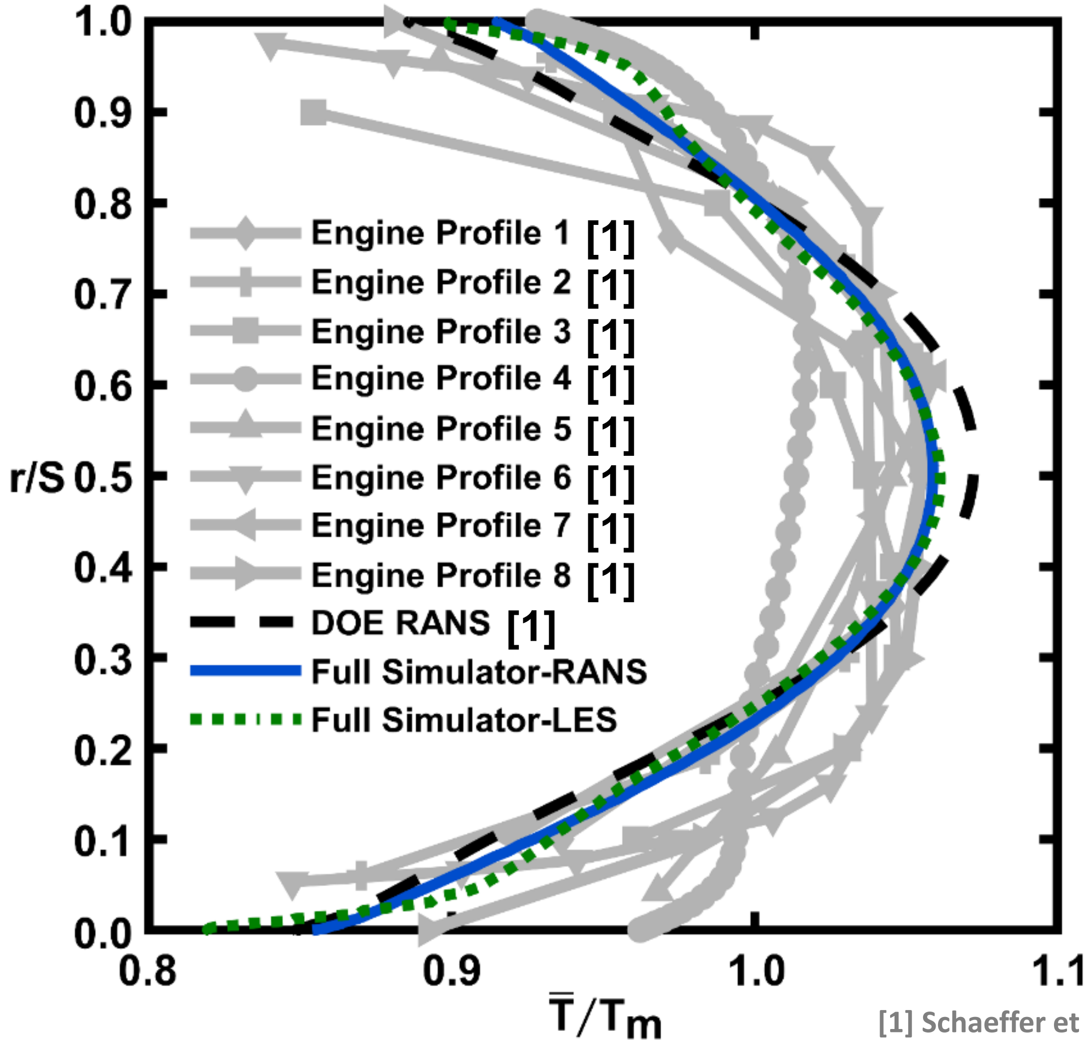
Factor Number	Variable
1	$\dot{m}_1/\dot{m}_t$ [%]
2	D_1/S [-]
3	$\dot{m}_{3,\,OD}/\dot{m}_t$ [%]
4	$\dot{m}_{3,\,ID}/\dot{m}_t$ [%]
5	$\dot{m}_{e,\,OD}/\dot{m}_t$ [%]
6	$\dot{m}_{e,\,ID}/\dot{m}_t$ [%]
7	$D_{3,\,ID}/S$ [-]
8	$D_{3,\,OD}/S$ [-]
9	$T_{3,\,OD}/T_m$ [-]
10	$T_{3,\,ID}/T_m$ [-]
11	$T_{e,\,OD}/T_m$ [-]
12	$T_{e,\,ID}/T_m$ [-]



- Steady RKE RANS
- Coupled energy solver
- Second-order discretization in space
- Viscous sublayer resolved

The CFD 'Design of Experiments' produced a wide range of profile shapes like the engine examples enabling four target profiles to be studied using RANS and LES

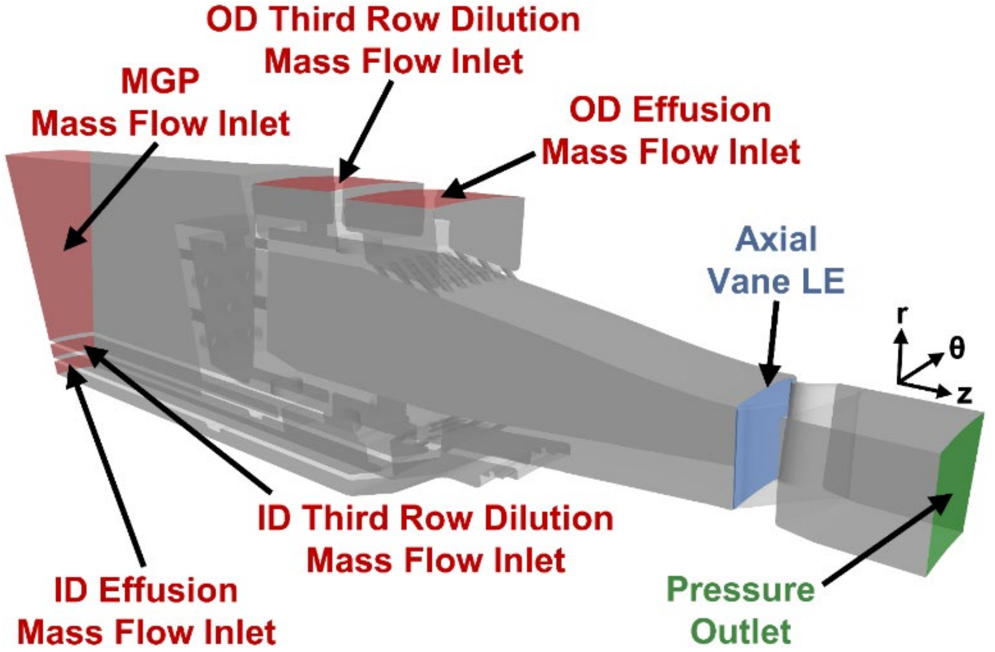


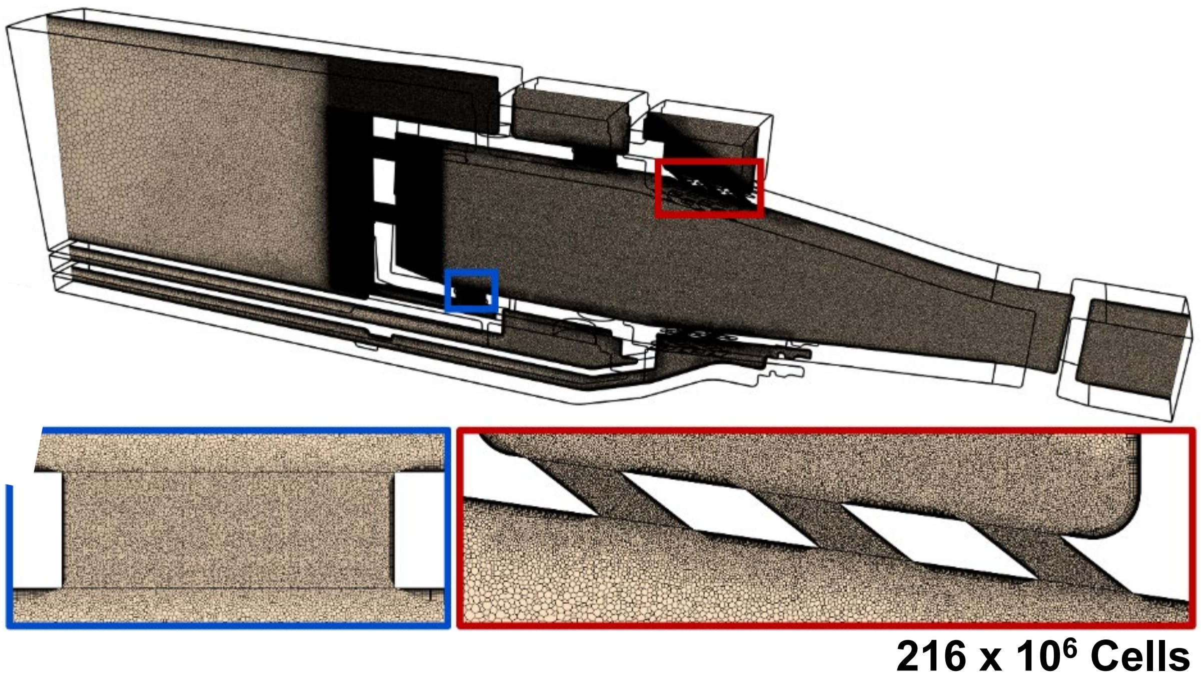


[1] Schaeffer et al., 2025

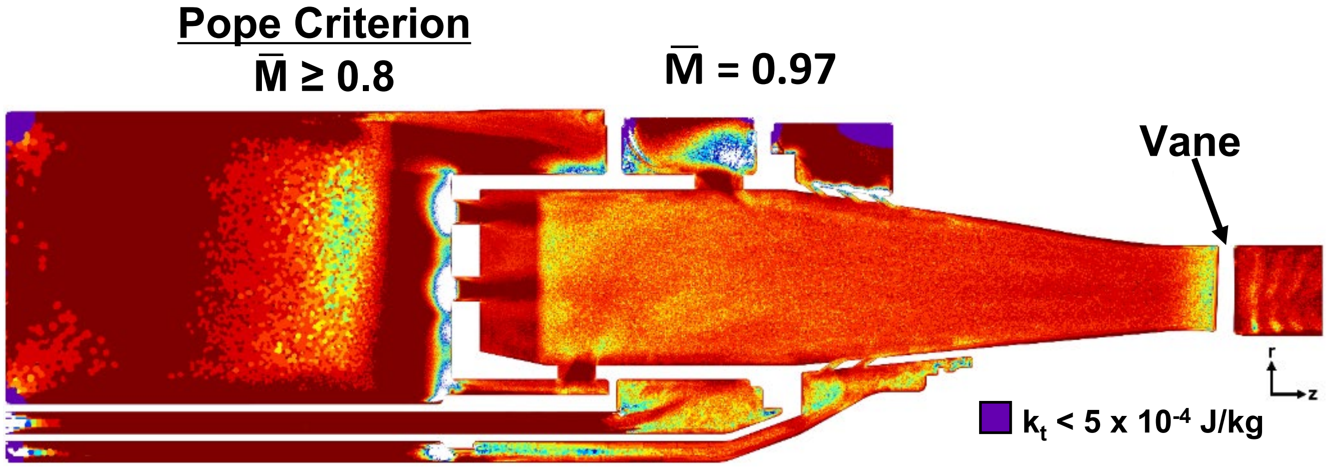
Overall, the average radial thermal profiles from RANS and LES agreed well

LES was performed on the full simulator geometry to characterize more accurate flow and thermal field predictions on the center-peaked profile configuration





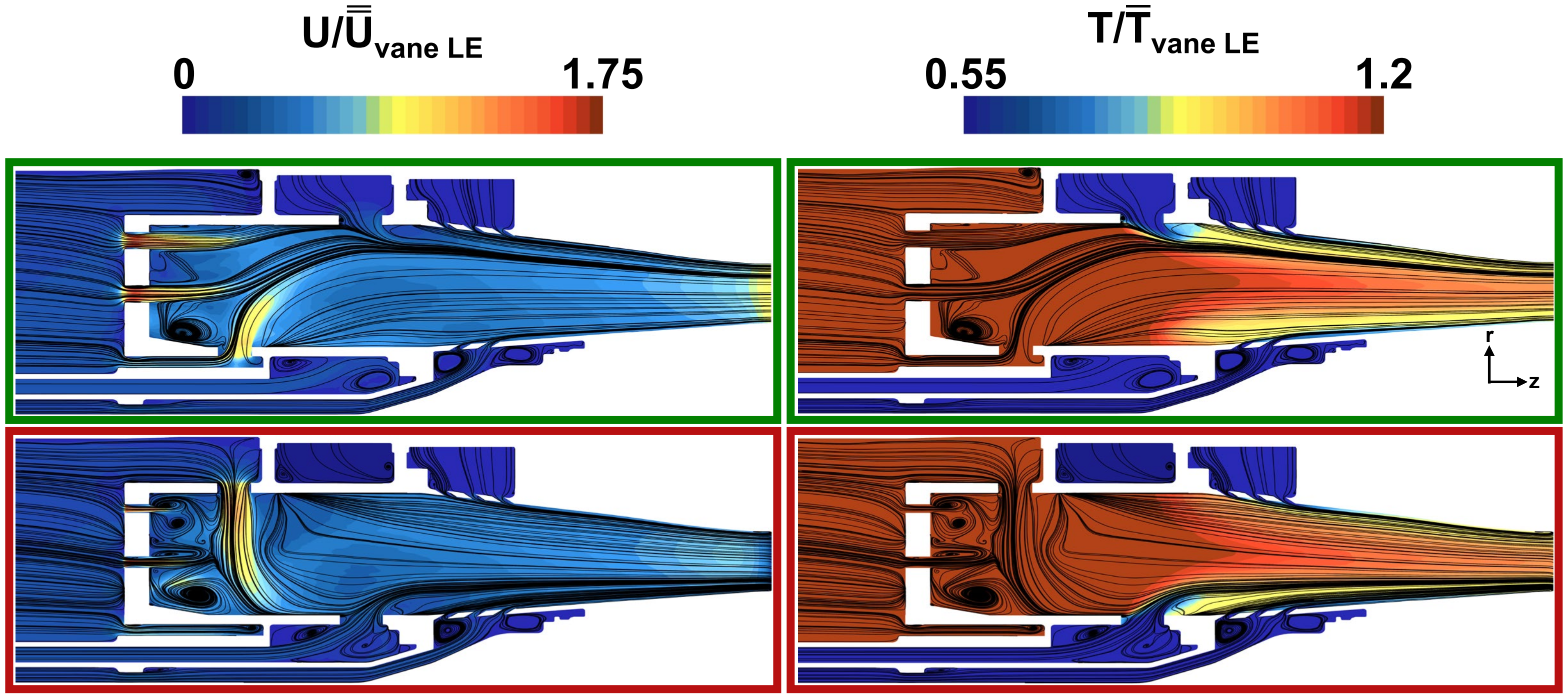
- WALE LES
- Coupled energy solver
- Second-order discretization in space, time
- Viscous sublayer resolved

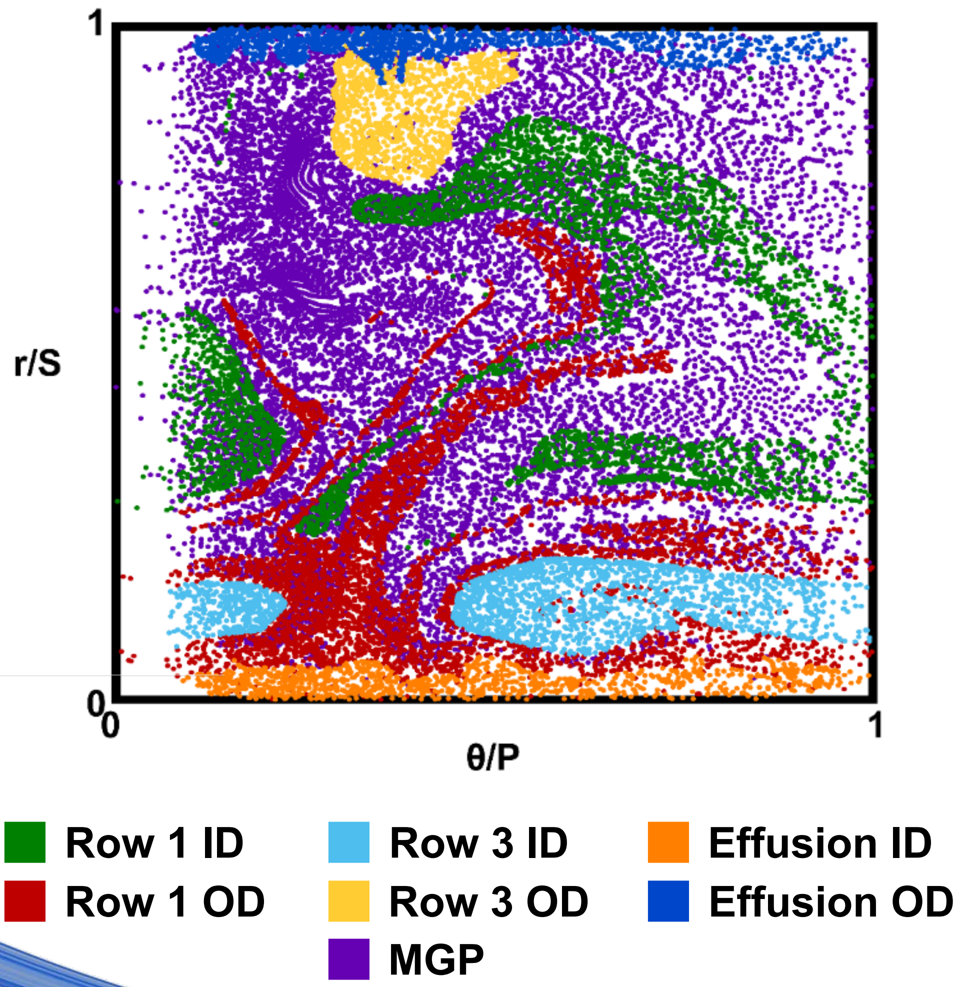


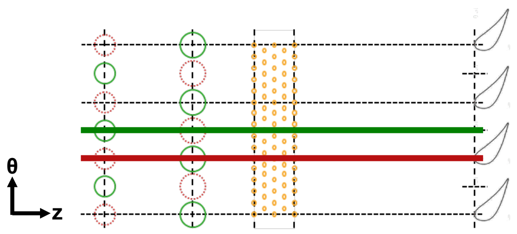
$$M = \frac{k_{res}}{k_{res}+k_{SGS}}$$

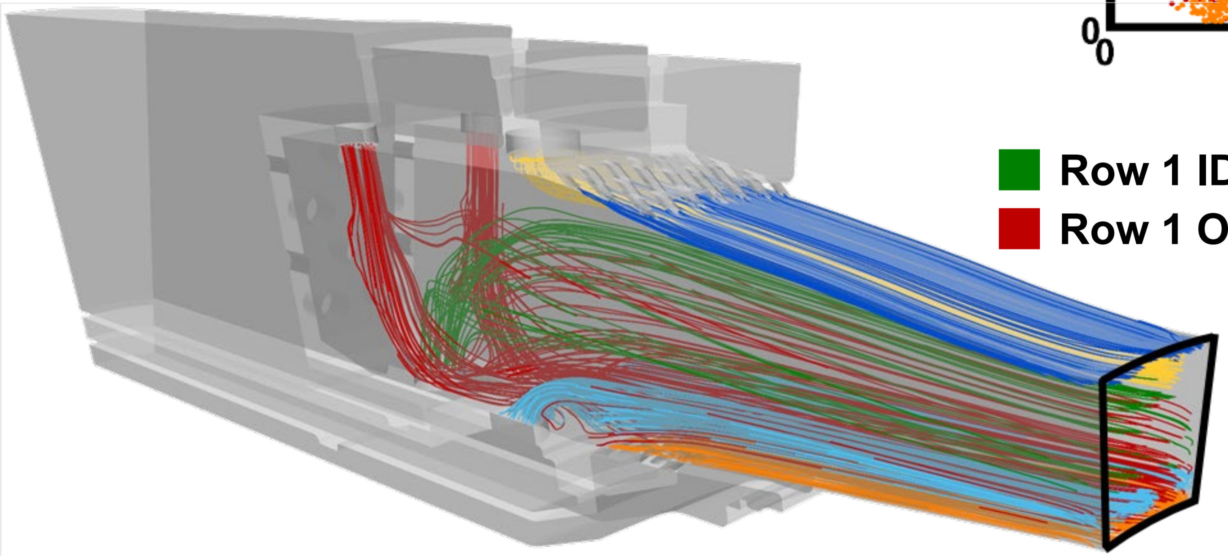
M = 0 is completely modeled
M = 1 is completely resolved

The first-row jets were found to increase the turbulence levels, while injection of dilution and effusion flow were found to tailor the profiles near the walls









* New Publication Written
Schaeffer et al. (2025)
To Be Submitted to ASME Turbo Expo 2026

The simulator hardware has been manufactured and trial assembled, while current work focuses on instrumentation, integration into the rig, and testing in 2026

ASCENT Project 068

Combustor Wall Cooling with Dirt Mitigation and Combustor Simulator

The Pennsylvania State University

PI: Reid Berdanier (PSU)

Co-PIs: Stephen Lynch (PSU), Karen Thole (UMich), Michael Barringer (PSU)

PM: Joshua Glottmann

Cost Share Partner(s): Pratt & Whitney, PSU, UMich

Students: Kyle McFerran (PSU), Chad B. Schaeffer (PSU), Fabrizzio Vega (UMich)

Objective:

Combustor Wall Cooling: Ingestion of dirt and dust lead to blockages of cooling holes which ultimately diminishes the effectiveness of combustor wall cooling. The objective of this part of the study is to explore the impact of dirt deposition on liner heat transfer, and test novel designs to reduce deposition.

Combustor Simulator: The research objective is to design a non-reacting profile simulator that produces temperature and pressure profiles representative of those entering high pressure turbines using computational fluid dynamics (CFD) simulations and then install the simulator in the START rig for experimental testing.

Project Benefits:

Combustor Wall Cooling: The expected benefit from the dirt study is to reduce the impacts of dirt deposition on turbine parts, specifically the combustor walls thereby promoting component life and safer flights.

Combustor Simulator: The expected benefit from the profile simulator study is to determine impacts of combustor exit temperature and pressure profiles at elevated turbulence on turbine stage efficiency and durability.

Research Approach:

The research approach for obtaining accurate heat transfer measurements within double-walled combustor walls:

i. Validate heat transfer measurements with and without dirt.
ii. Measure heat transfer for a range of liner designs.
iii. Quantify the sensitivity to dirt on heat transfer and flow.

The research approach for developing a profile simulator:

i. Work with a design firm to develop the simulator.
ii. Use CFD to design and determine operation of the simulator.
iii. Install and benchmark the simulator in the START rig.

Major Accomplishments (to date):

Combustor Wall Cooling: By replacing the double-wall of a combustor with a triple-wall, there is a dramatic reduction in the dirt deposition on the effusion wall, which is the critical wall in containing the combusted flows.

Combustor Simulator: CFD RANS simulations indicate that the profile simulator can achieve a wide range of profile shapes. Higher fidelity CFD using LES was performed to enable better turbulence predictions. The simulator hardware has been manufactured, and trial assembly was performed. Two new paper publications on this work were written (one under review in ASME *Journal of Turbomachinery*, one to be submitted to ASME Turbo Expo 2026).

Future Work / Schedule:

Combustor Wall Cooling: Continued work will expand the testing capabilities of the triple-wall design by considering additional dirt sizes and pressure ratios.

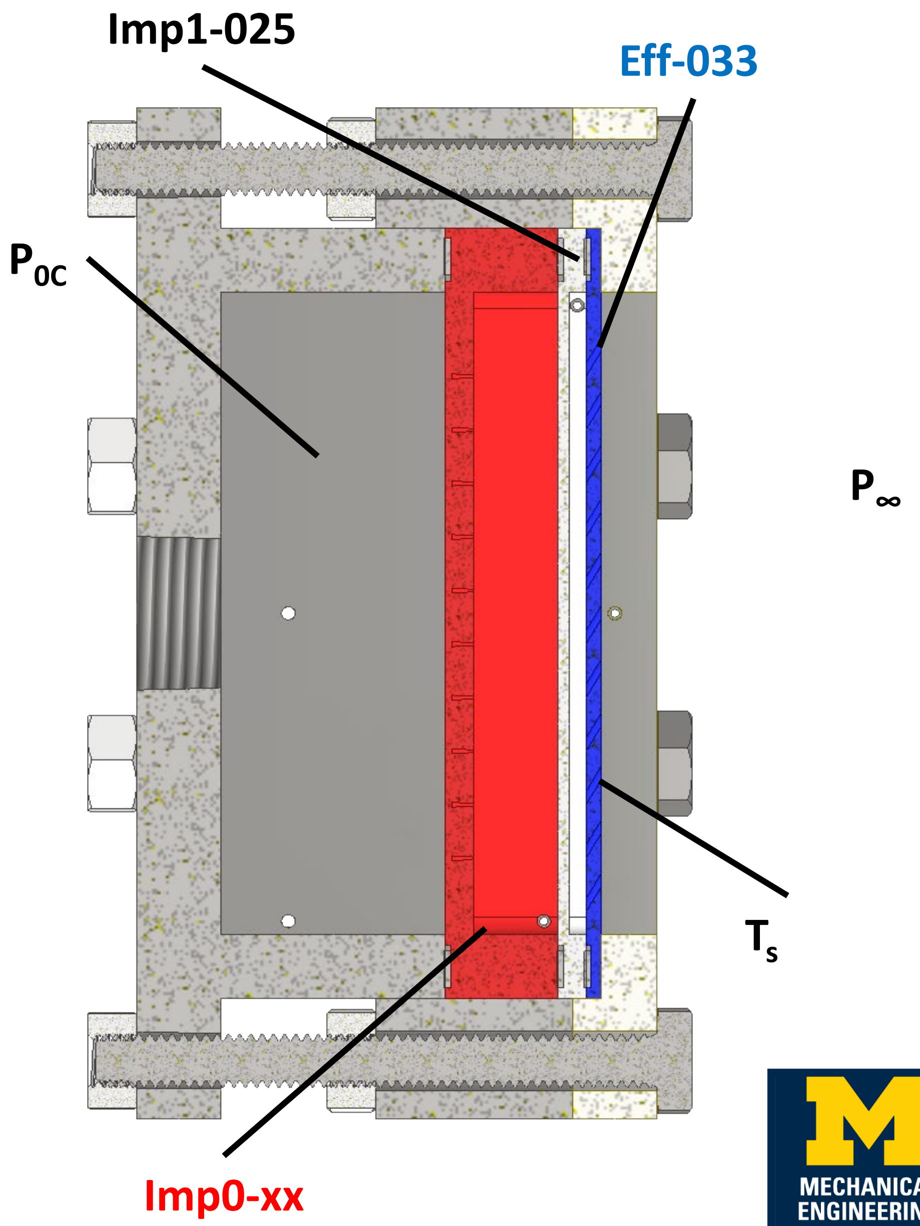
Combustor Simulator: The hardware will be instrumented and installed into the START turbine test rig, with experimental testing planned for 2026.

This research was funded by the U.S. Federal Aviation Administration Office of Environment and Energy through ASCENT, the FAA Center of Excellence for Alternative Jet Fuels and the Environment, project 068 through FAA Award Number 13-C-AJFE-PSU-132 under the supervision of Joshua Glottmann. Any opinions, findings, conclusions or recommendations expressed in this this material are those of the authors and do not necessarily reflect the views of the FAA.

A total of four combustor liner geometries are tested, varying in impingement hole diameters and plate to plate spacing

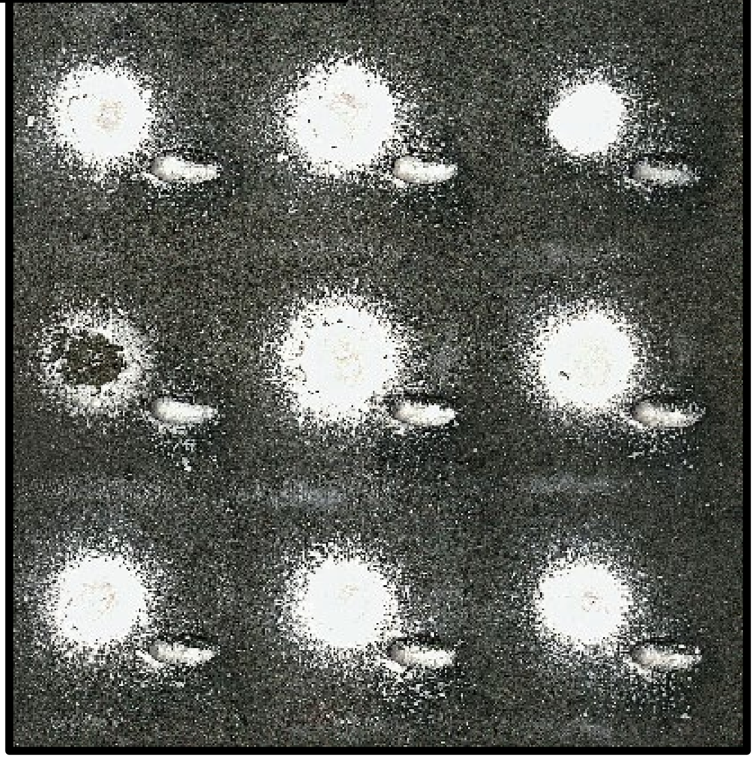
	Testing Conditions		Geometry		
	T_s [°C]	PR [-]	D_0 [in]	D_1 [in]	D_2 [in]
Combustor liner geometry	1000	1.045	0.060	0.025	0.033
	1000	1.045	0.090	0.025	0.033
	1000	1.045	0.125	0.025	0.033
	1000	1.045	-	0.025	0.033

$$PR = \frac{P_{0C}}{P_\infty}$$

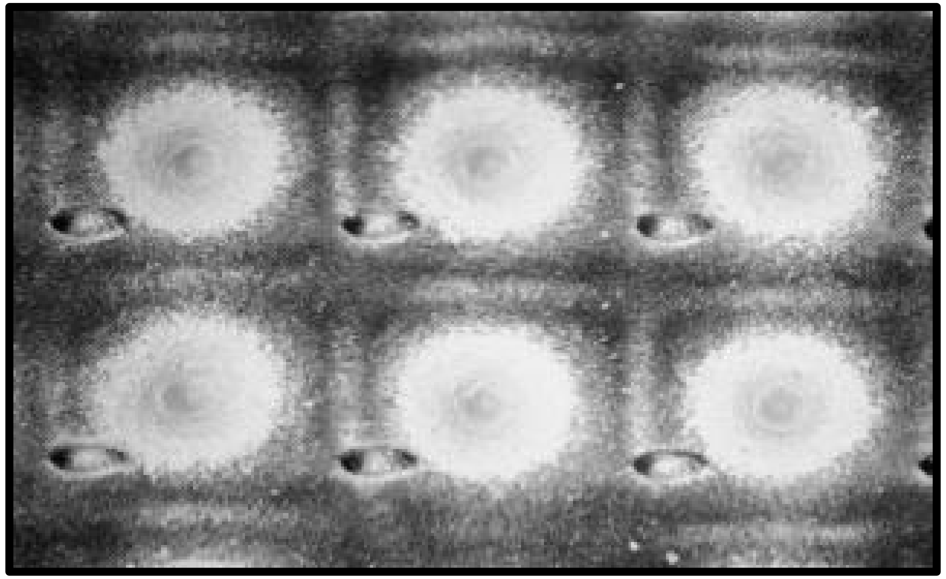


Similar dirt deposition patterns occur for heated and room temperature testing of the double wall design

Double Wall (1200° C)



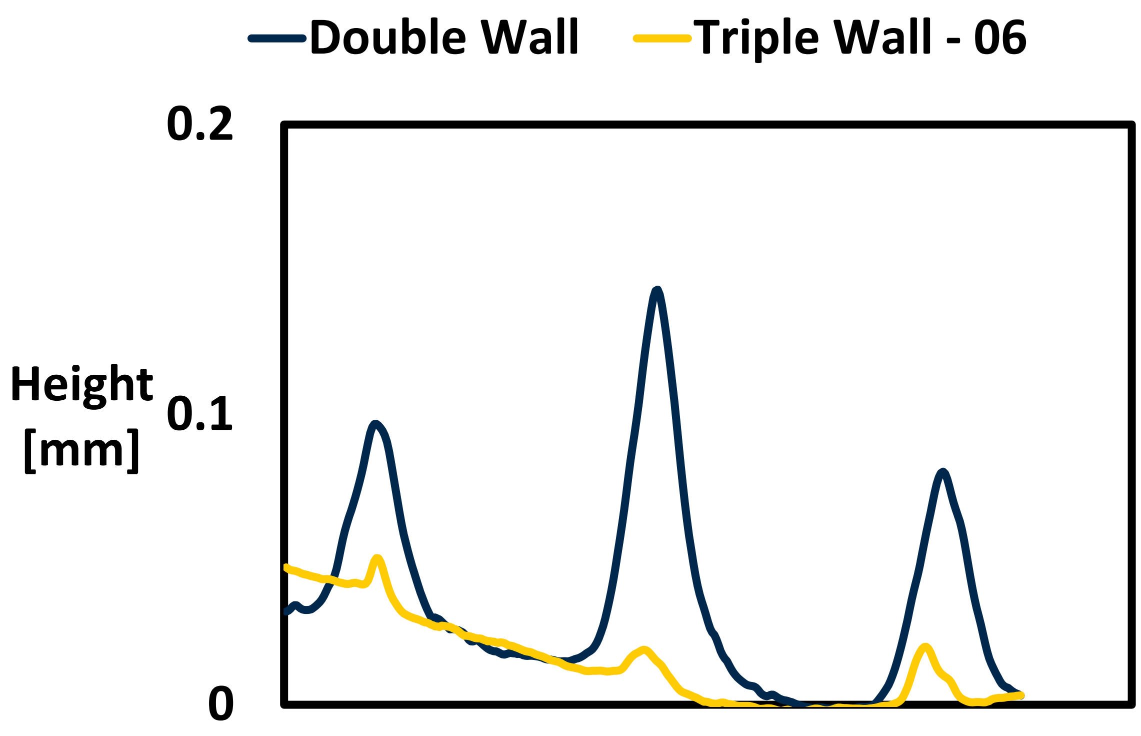
Double Wall, Room Temperature | McFerran et al. 2025



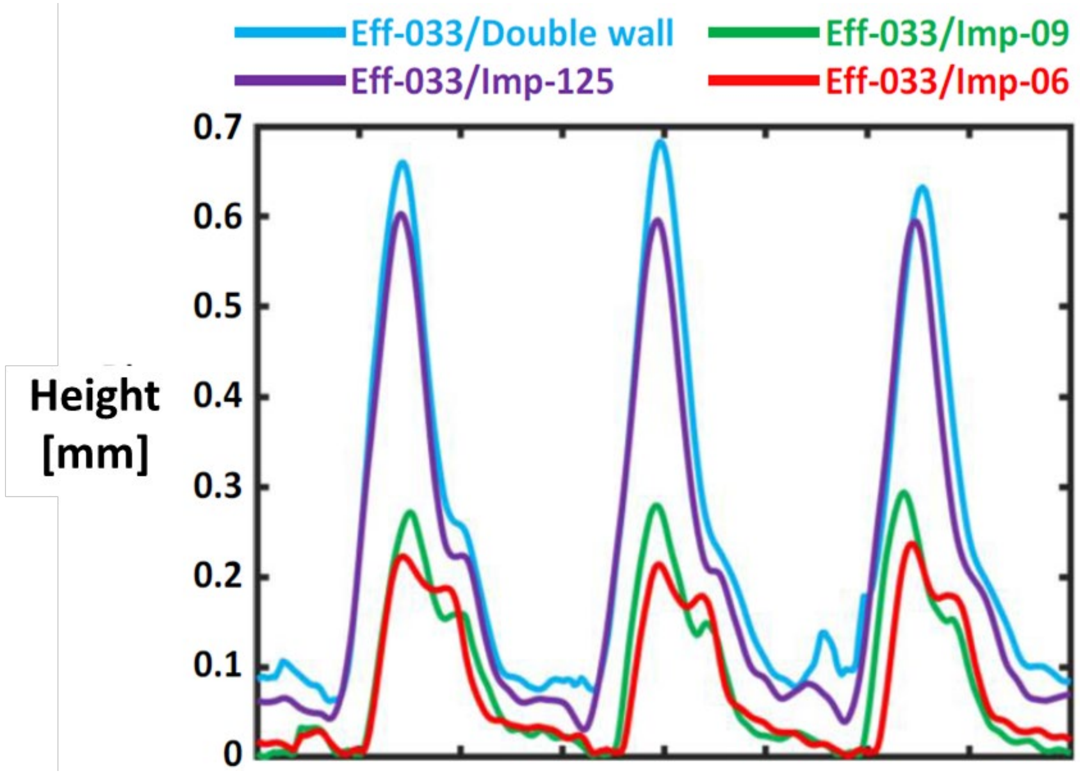
MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

The lateral average deposit heights are consistently lower for the triple wall design at both hot and cold conditions

High-Temperature (1200° C)



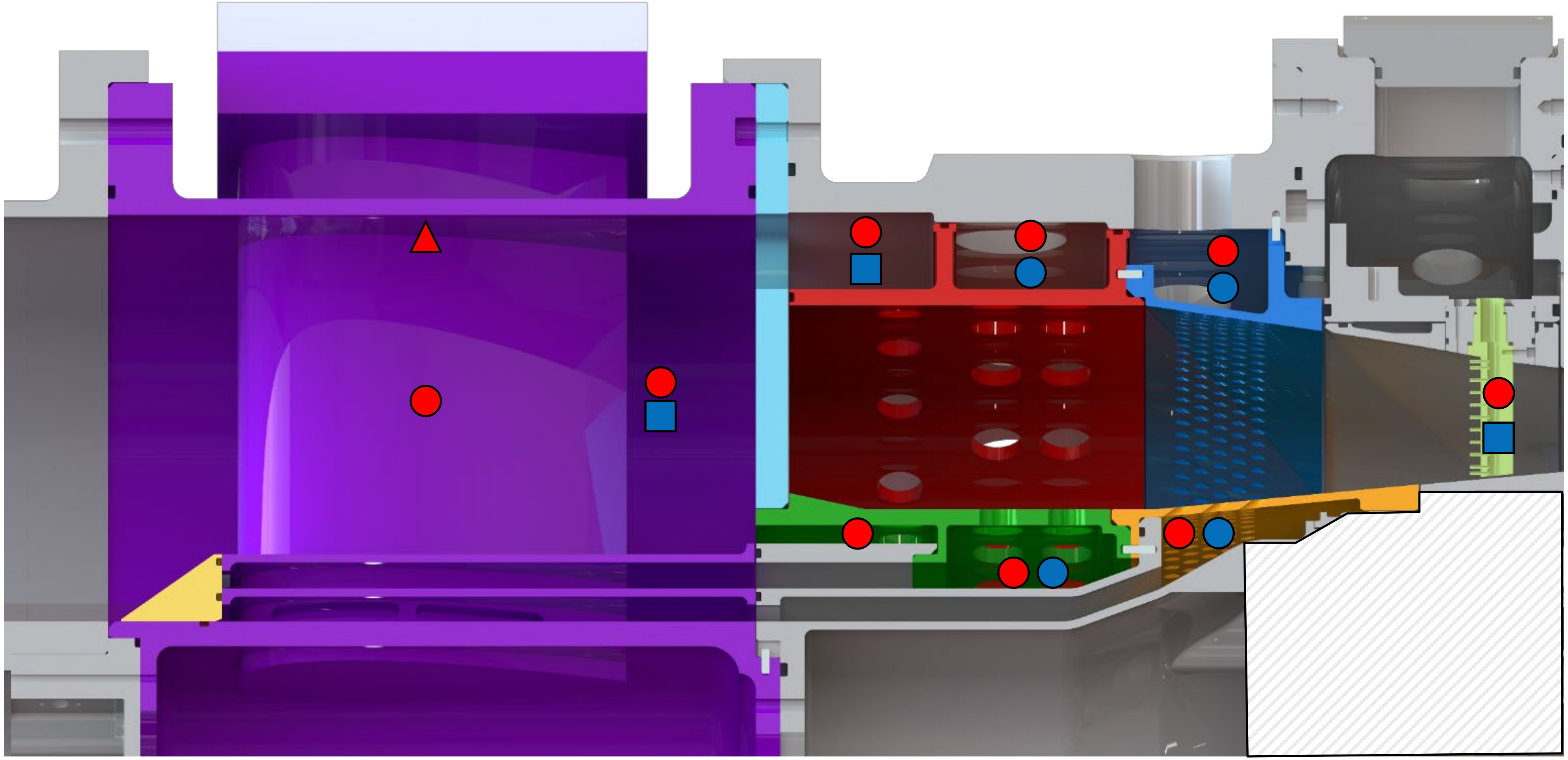
Room Temperature | McFerran et al. (2025)



Note scale differences

MECHANICAL ENGINEERING
UNIVERSITY OF MICHIGAN

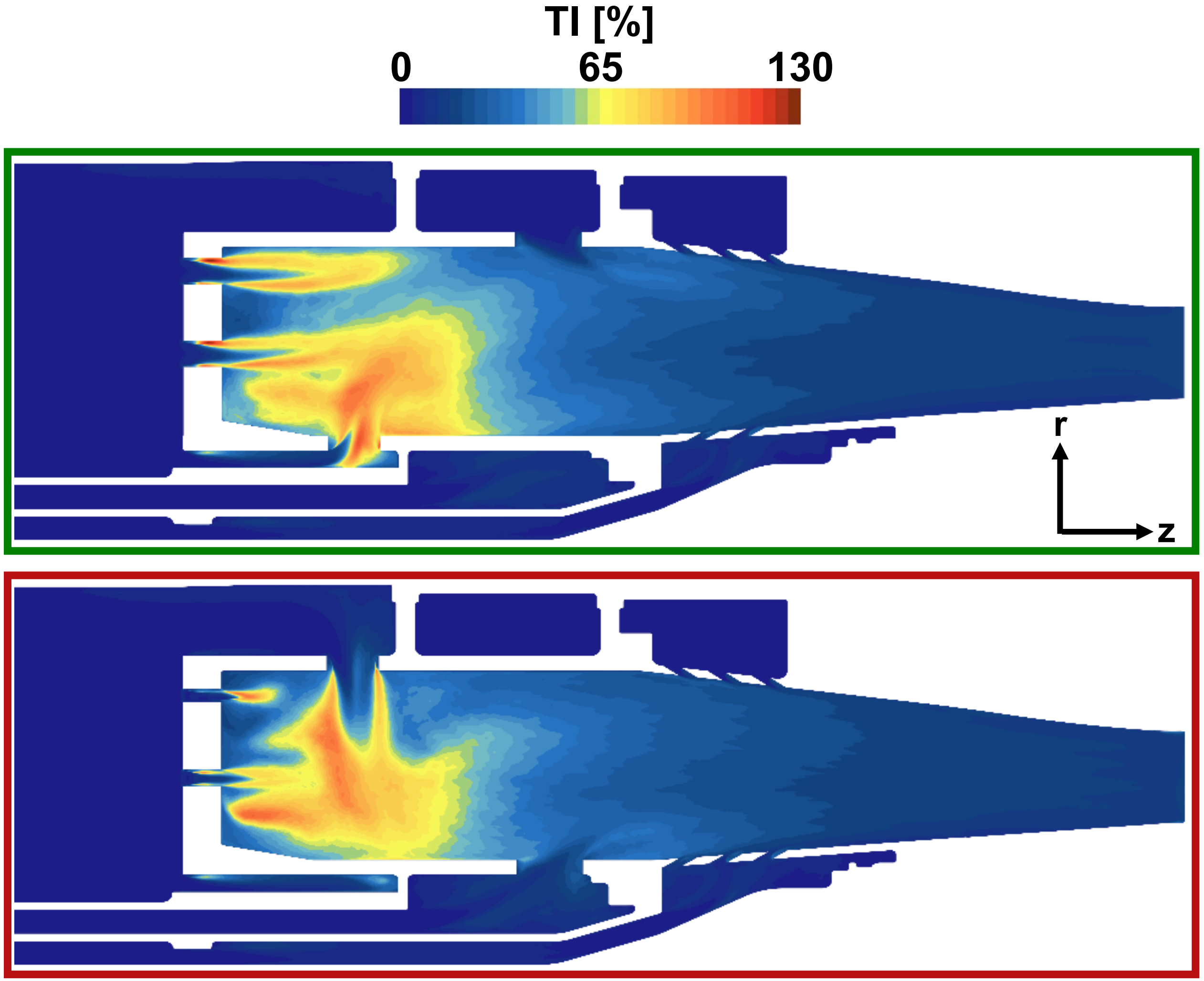
An instrumentation plan was developed within the simulator assembly to help characterize performance relative to design and CFD predictions

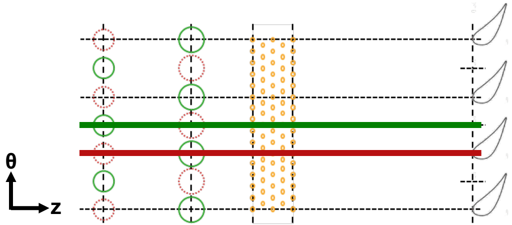


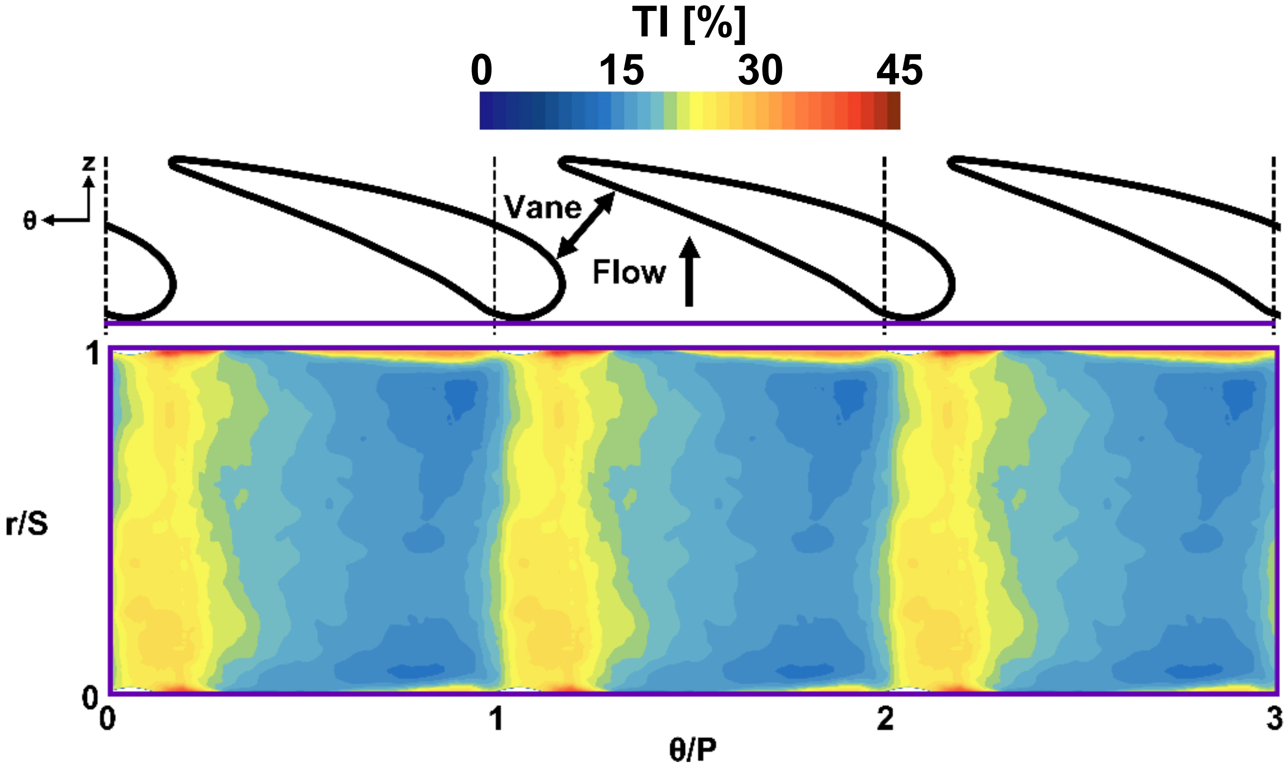
Traverse Rake

- Red circle: Thermocouple: Flow
- Red triangle: Thermocouple: Surface
- Blue circle: Plenum Pressure
- Blue square: Kiel Probe: Flow Total Pressure

The first-row jets and freestream flow were found to generate turbulence levels at the turbine inlet typical of gas turbine combustors: TI = 20-25%







$$TI = \frac{\sqrt{\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right)/3}}{\overline{\overline{U}}}$$