Thermal Modeling of an Advanced Geared Turbofan for Distributed Engine Control Application

Jonathan Kratz (NASA GRC)

Dennis Culley (NASA GRC)

George Thomas (N&R Engineering)

AIAA Propulsion & Energy Forum

10 July 2018

Outline

Summary

- Presents a method for modeling the dynamic thermal environment of a gas turbine engine with an eye toward control system component reliability as it relates to the implementation of distributed engine control.
- Application is to a conceptual N+3 generation geared turbofan.
- The resulting model is shown to run in real-time within a multi-model simulation environment that demonstrates the ability to interact with hardware to drive test equipment

Outline

- Background/Motivation
- Thermal Modeling Methodology
- Thermal Modeling Techniques
- Application to an Advanced Geared Turbofan
- Real-Time Capabilities
- Summary

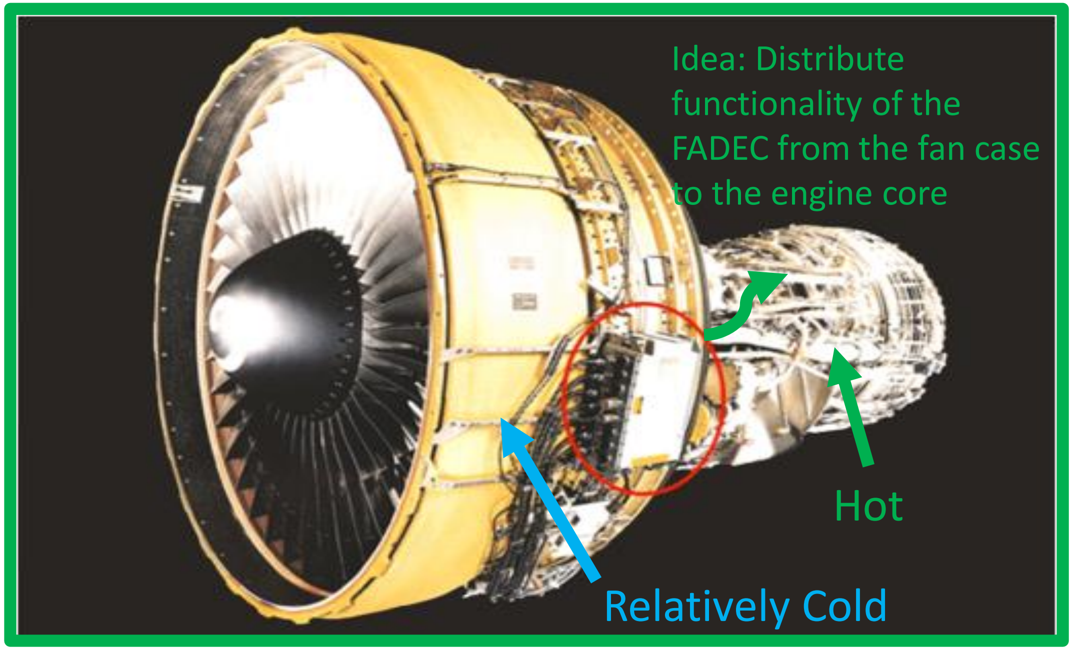
Background: Distributed Engine Control

Current Control Approach

- Centralized architecture performed through a full authority digital engine controller (FADEC)
- Constrains the control system topology and limits capability

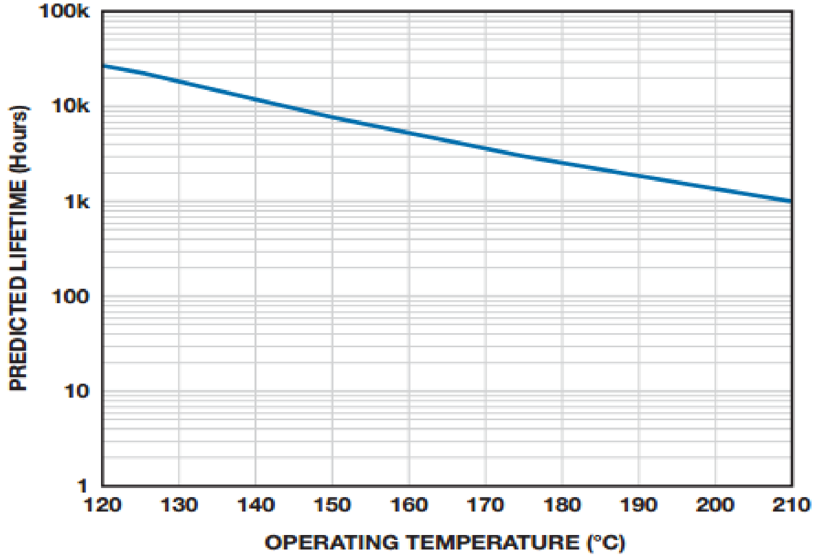
Distributed Engine Control

- Hardware-driven strategy that adds flexibility
- Modularizes the control system and distributes control functions to smart nodes located across the engine
- Utilizes a light-weight digital communication network



- Some Potential Benefits: Reduce weight, reduce volume impact, alleviate obsolescence and certification issues, enable more advanced control

Background: High Temperature Electronics



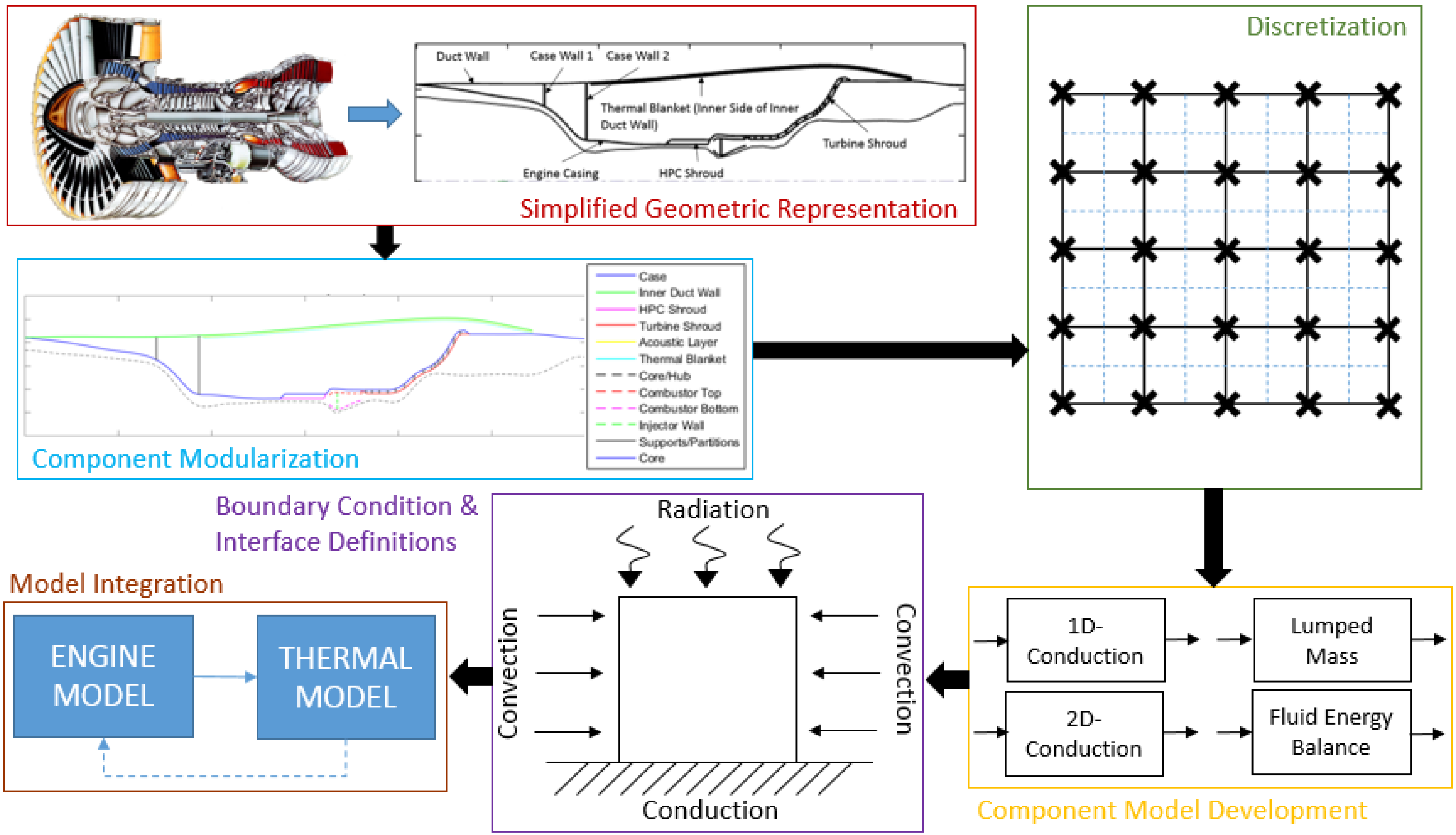
Example of the reliability vs. temperature relationship for an electronic device http://www.analog.com/library/analogdialogue/archives/46-04/high_temp_electronics.pdf

- Desire to mount smart nodes on the engine core
- Challenging thermal environment
  - State-of-art material for internal gas path exceed 1500˚C
  - Common consumer electronics operate reliably <70˚C, sometimes 150˚C
  - Inverse relationship between temperature and electronic reliability
- High-Temp electronics
  - Silicon-On-Insulator (SOI): Up to 300˚C (225˚C near term)
  - Silicon Carbide: 500˚C +

- Important considerations: Max & min temperature (steady-state), rate of change in temperature (dynamic), & temperature cycling (dynamic)
- Objective: Develop a thermal model of the relevant engine structure to estimate the environment in which DEC electronics will be placed + develop re-useable modeling tools + develop capability to use the model and or its results to drive test equipment

"High temperature is relative"

Thermal Modeling Methodology



Thermal Modeling Techniques

Structure

- 2-D Finite Difference Method (FDM) - More significant components (engine casing, shrouds, duct wall, etc.)

$$\frac{\partial T}{\partial t} = \frac{1}{\rho C_p} \nabla \cdot (k \nabla T) \rightarrow \frac{\partial T}{\partial t} = \frac{1}{\rho C_p}\left[\left(\frac{\partial k}{\partial r} + \frac{k}{r}\right)\frac{\partial T}{\partial r} + k\frac{\partial^2 T}{\partial r^2} + \frac{\partial k}{\partial z}\frac{\partial T}{\partial z} + k\frac{\partial^2 T}{\partial z^2}\right]$$

T = temperature, t = time, ρ = density, C_p = heat capacity, k = thermal conductivity, r = radial direction coordinate, z = axial direction coordinate

  - Model the component as a cylindrical shell of constant radius and thickness
  - Discretized and then solved using a 2-D implicit scheme

- Lumped Capacitance – Less significant components (core components – compressor and turbine blades)

$$\frac{\partial T}{\partial t} = \frac{1}{mC_p}[\underbrace{hA(T_F - T)}_{\text{Convection}} + \underbrace{uA(T_R - T)}_{\text{Radiation}}]$$

m = effective mass, A = surface area, T_F = temperature of convecting fluid, T_R = temperature of radiating body, h = convection heat transfer coefficient, u = radiation heat transfer coefficient

Thermal Modeling Techniques

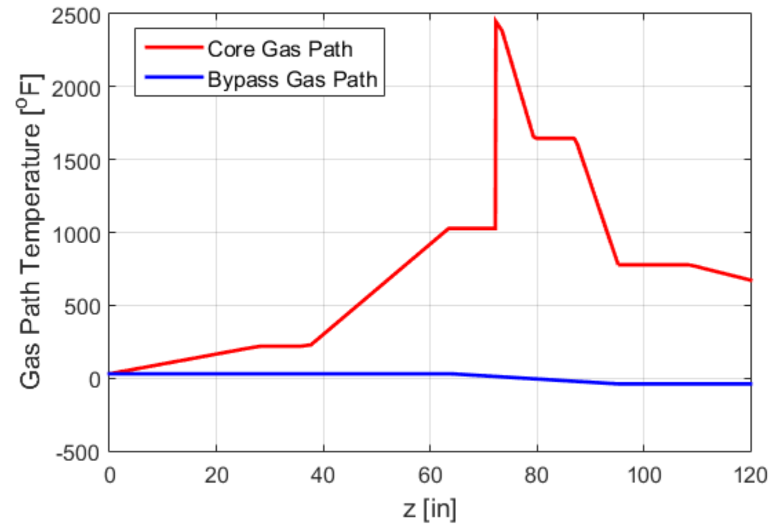
Flow Paths & Voids

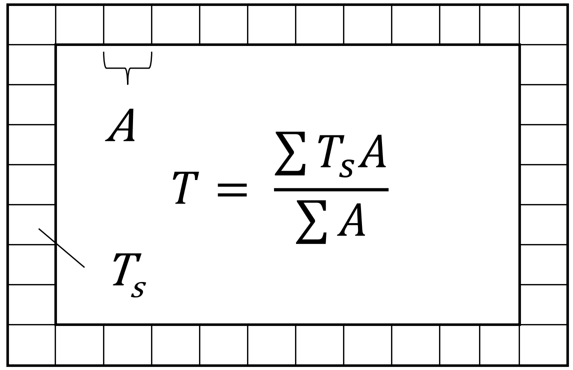
- Engine simulation data – used for gas paths and some bleed flows
- Fluid Energy Balance – used for bleed flows of significant heat transfer and relatively low mass flow

$$Q = \sum_{out}(\dot{m}C_pT) - \sum_{in}(\dot{m}C_pT)$$

T = temperature, C_p = heat capacity,
$\dot{m}$ = mass flow rate, Q = heat

- Average of Surroundings – used for closed volumes with no forced air flow



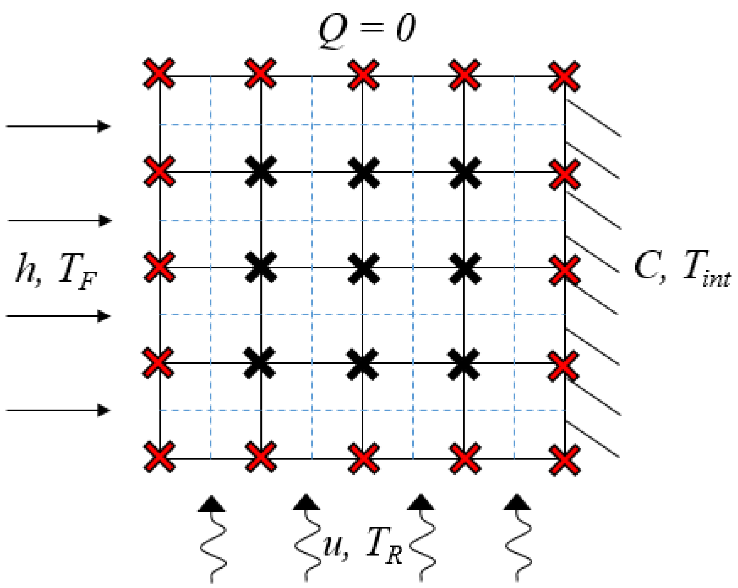


A = surface area of element exposed to the enclosed fluid
T_s = surface temperature of the element

Thermal Modeling Techniques

Boundary Conditions

- Conduction
  - Thermal capacitance of each boundary node is computed to enable conduction boundary conditions to be applied at the interface of 2 solid components
- Convection
  - Forced and natural convection are considered
  - Relations used for the coefficient *h* are generic and tunable
    - Tuning variables were set based on guidance from studies conducted at NASA and information found in literature
- Radiation
  - Written in a linear form
  - The coefficient *u* is a strong function of temperature and is updated each time-step of the simulation.
  - Relations for *u* assumes radiation between reflective concentric cylinders
  - Assumed radiation only occurs between parallel surfaces

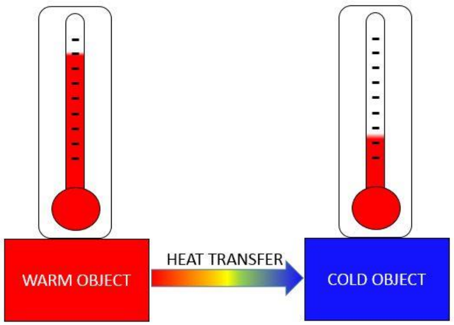


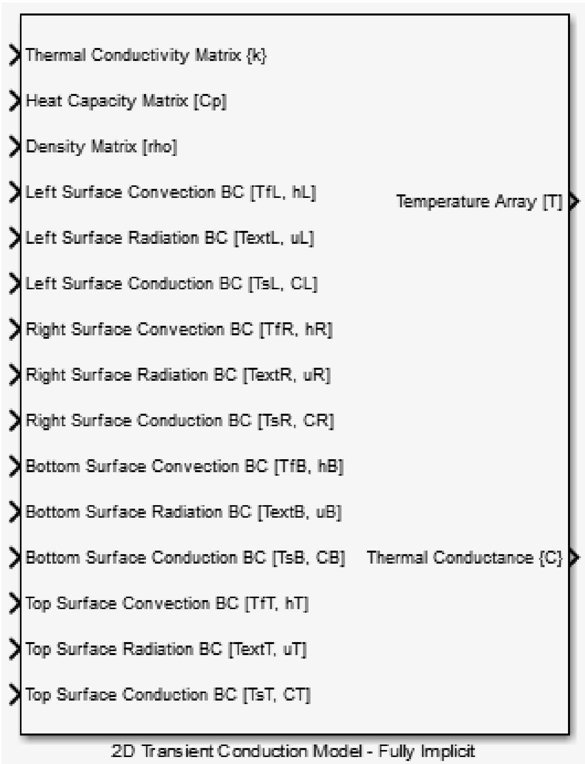
Q = heat
h = convection heat transfer coefficient
T_F *= fluid temperature*
u = radiation heat transfer coefficient
T_R *= temperature of radiating body*
C = thermal capacitance
T_{int} *= temperature at the interface between 2 solids*

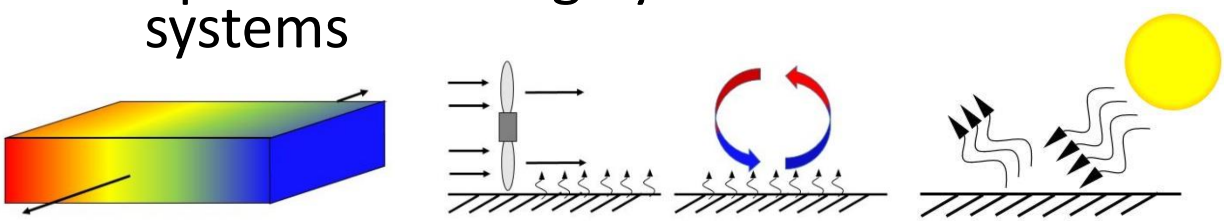
Thermal Modeling Techniques

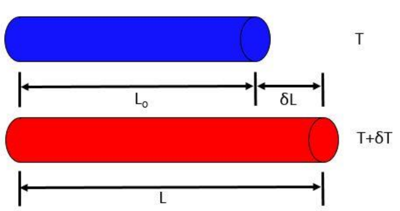
Thermal System Analysis Toolbox (TSAT)

- Library of tools developed in the MATLAB/Simulink environment
- Topics modeled
  - Conduction
  - Convection
  - Radiation
  - Deformation
  - Air Properties
  - Fluid Heat Transfer
  - General Tools
- Provides building blocks for building up and modeling dynamic thermal systems







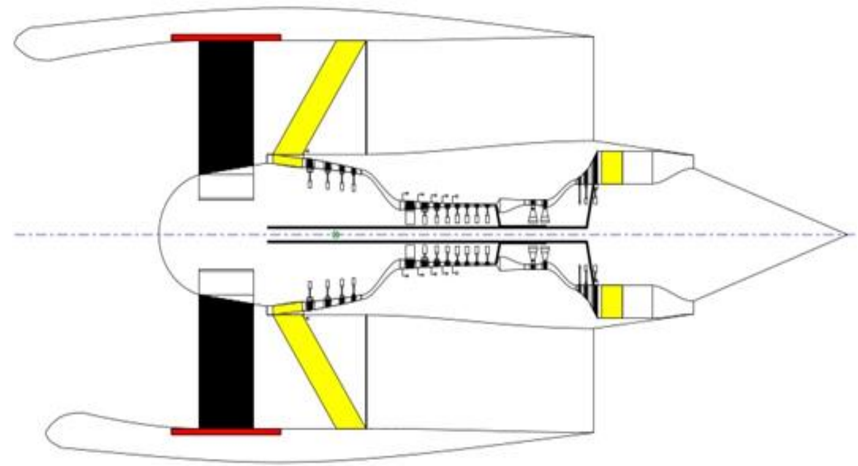


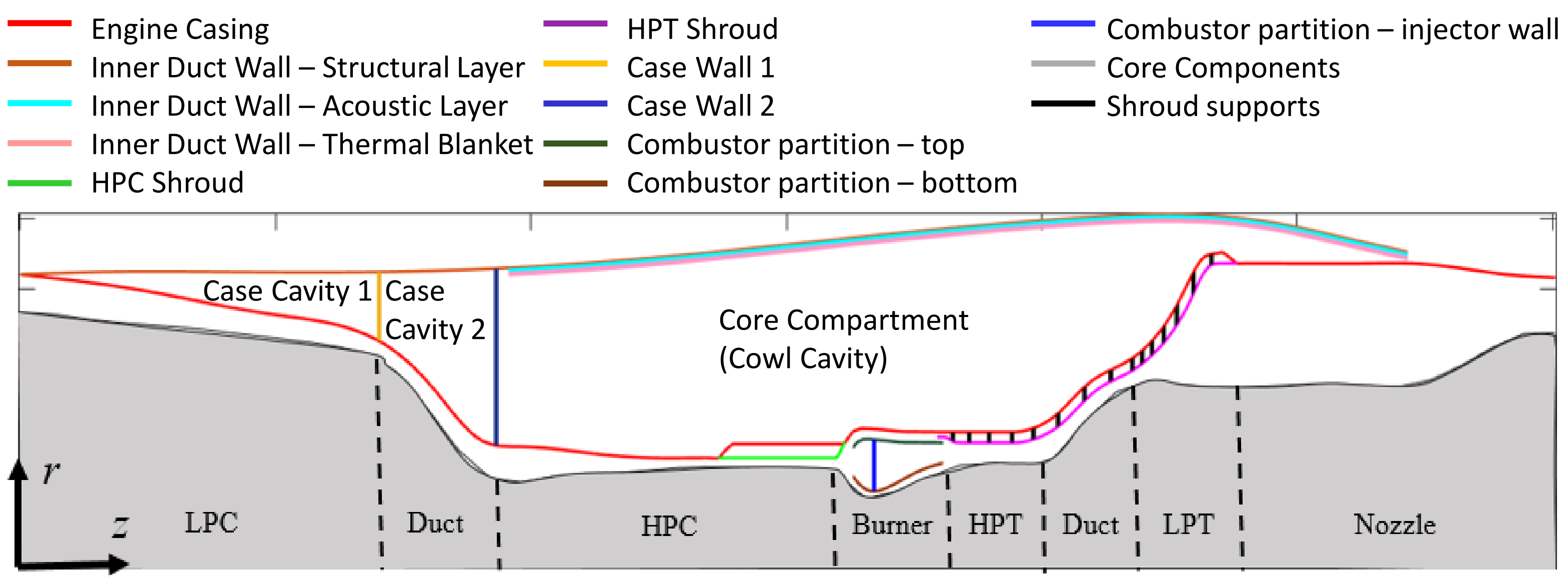
Download Link: https://github.com/nasa/TSAT

Application: The Engine

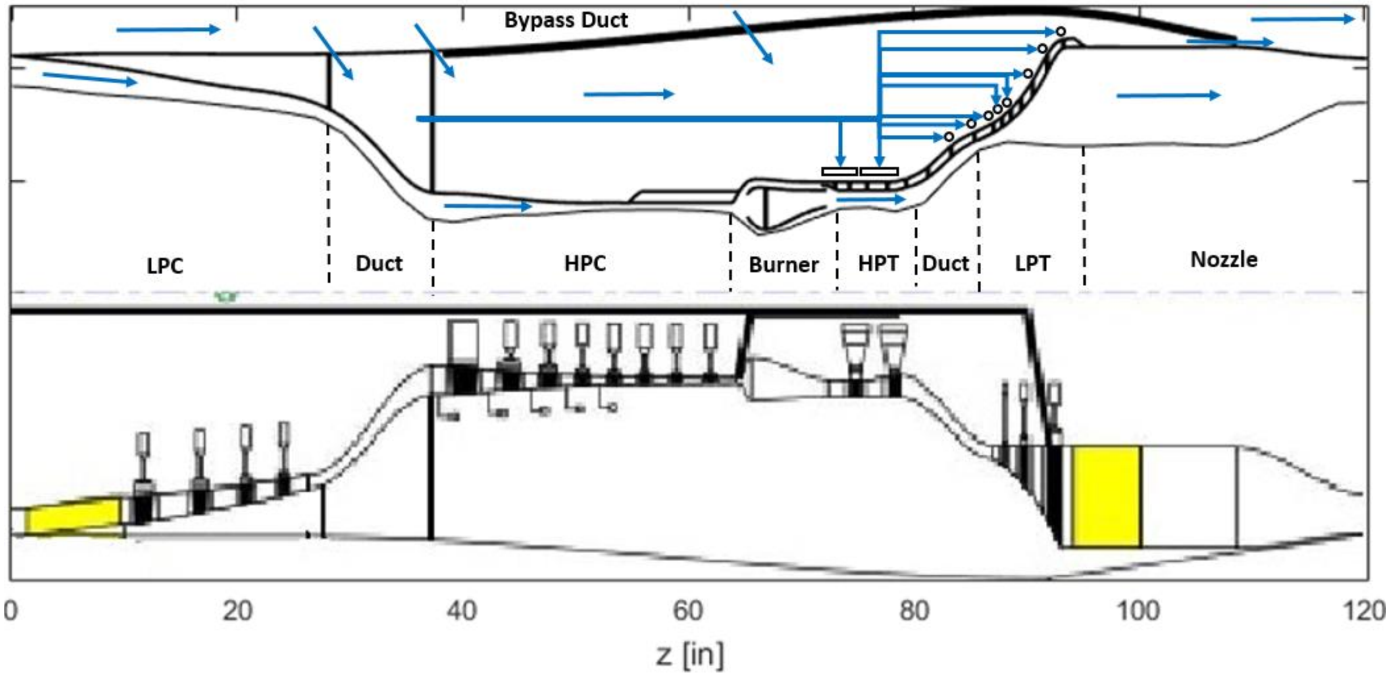
Advanced Geared Turbofan 30,000lb$_f$ (AGTF30)

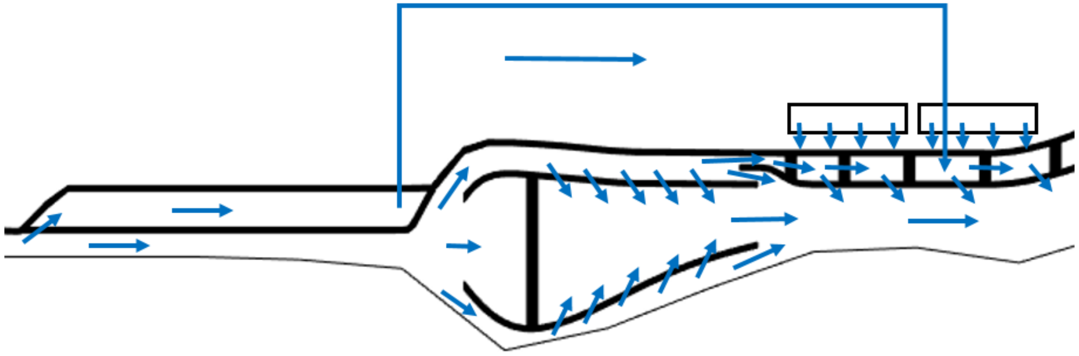
- Based on the NASA N+3 NPSS reference engine
- 3rd generation geared turbofan
- Features a compact gas turbine (CGT) and a variable area fan nozzle
- Capable of producing 30,000lb$_f$ of thrust at the sea-level static condition

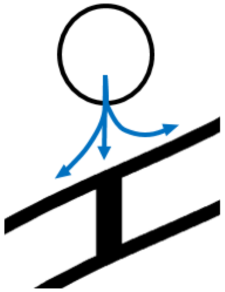




Application: The Engine Air Flow

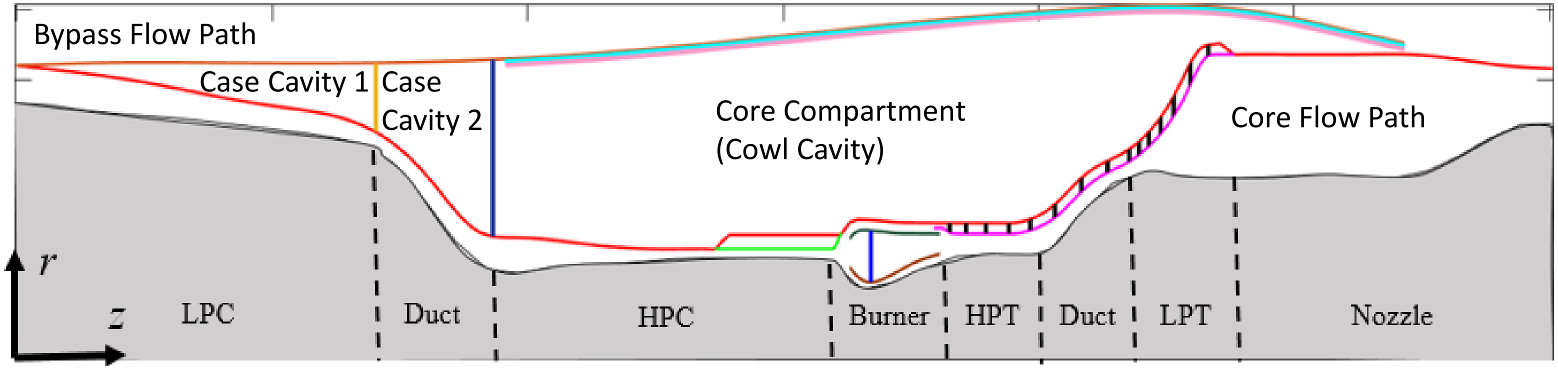






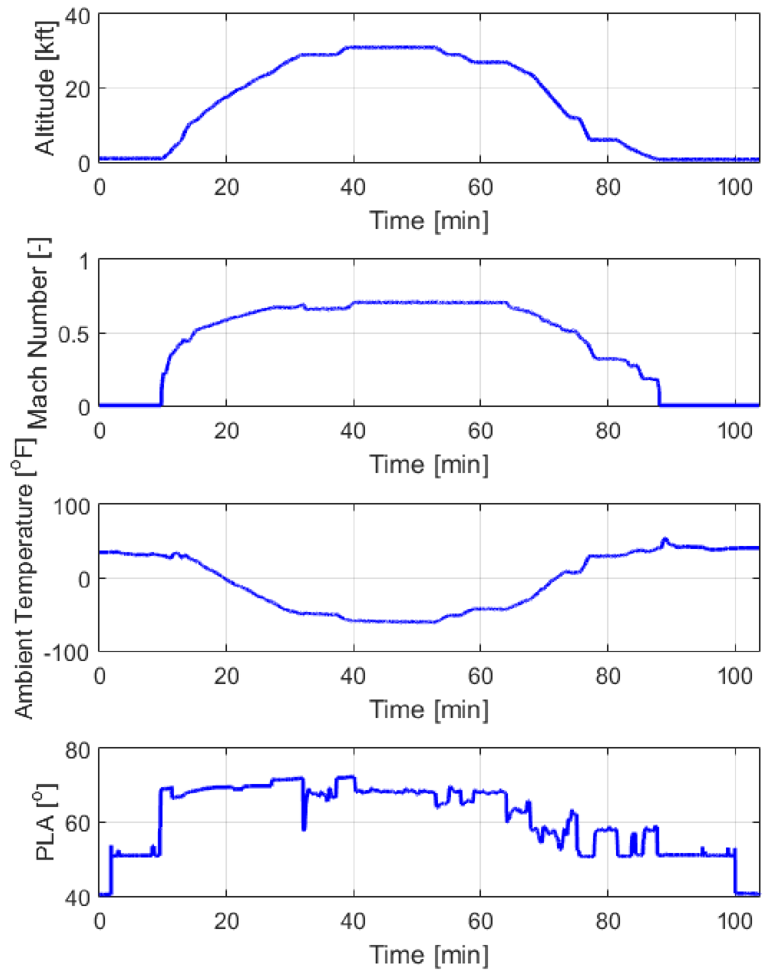
Application: Modeling

- Structures
  - All components except the "core components" were modeled with 2-D FDM
    - Geometry was approximated as a cylindrical shell of the components average radius and thickness
    - Various levels of discretization were investigated
  - Core components utilized a lumped capacitance model
- Important cavities and voids
  - Bypass flow path, core flow path, and case cavity 2 temperatures were driven by the engine model simulation
  - Case cavity 1 temperature was approximated as the average of its surrounding structure
  - Core compartment temperature was approximated using the fluid energy balance method
- Boundary conditions and interfaces between models were defined



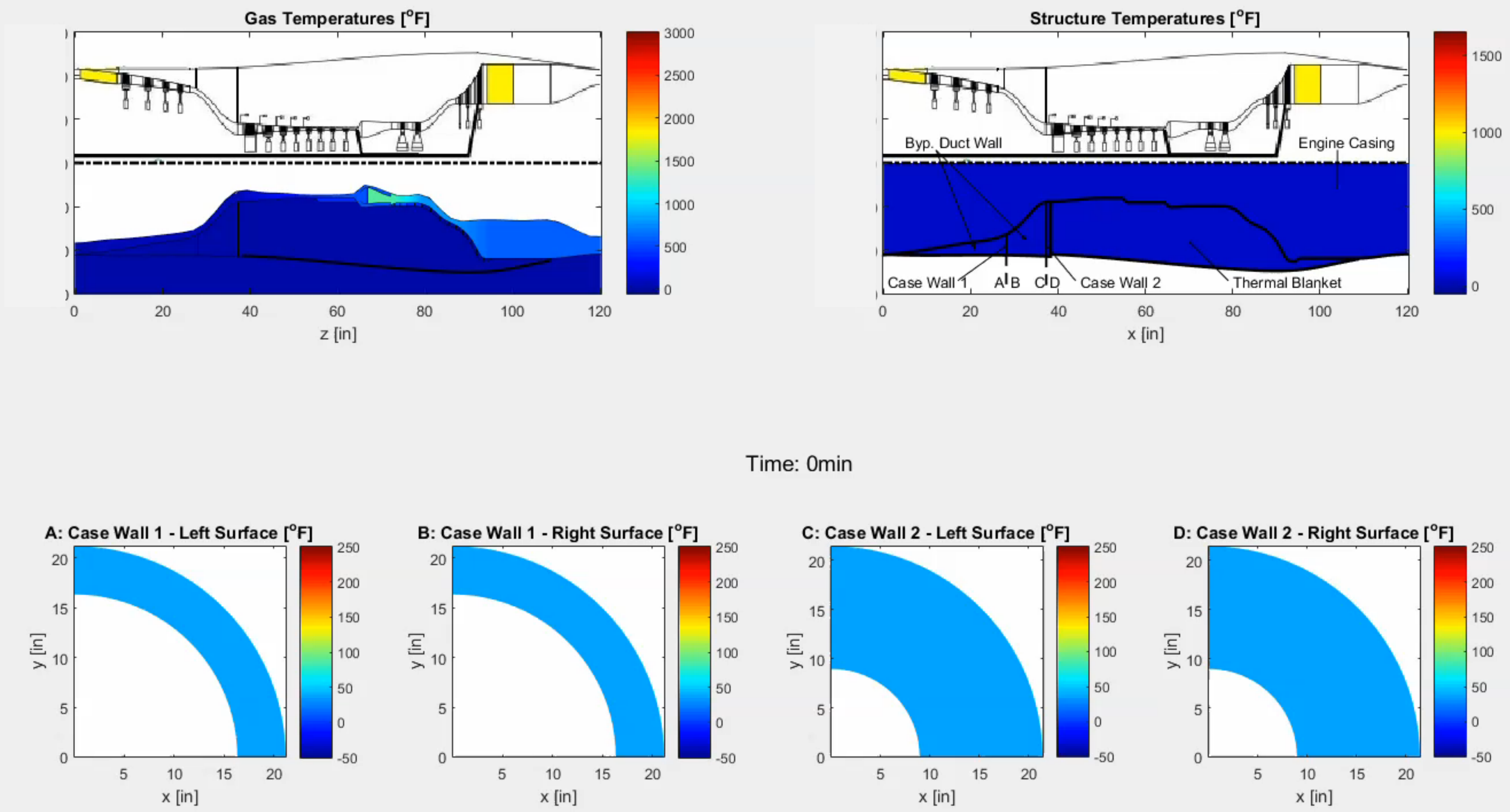
Application: Flight Profile

- Flight profile constructed from real data
- The starting and ending destinations are unknown but is representative of a ~250 mi flight
  - Cleveland, Ohio to Washington D.C. or Las Vegas, Nevada to Los Angeles, California
- At the start of the simulation all structures are initialized at ambient temperature
- After the flight, the thermal simulation is extended to investigate heat soak back (modeling details are not provided here for the sake of time – see the paper)



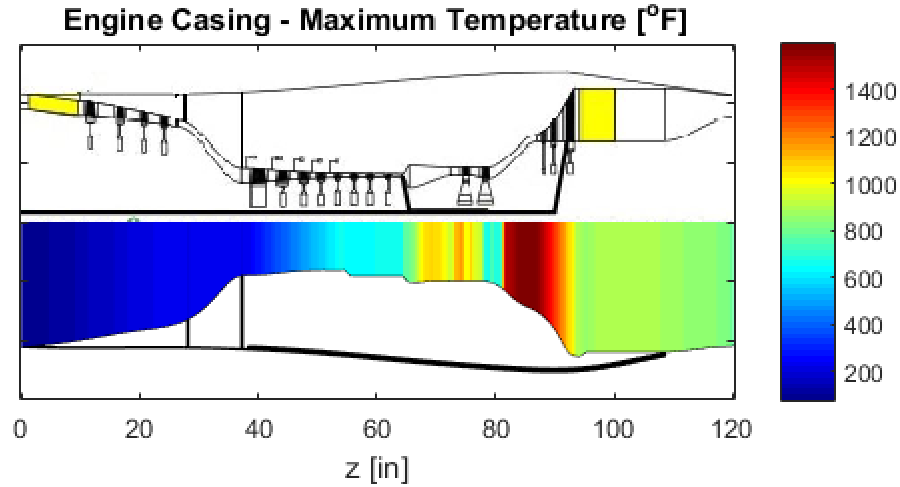
PLA = power lever angle

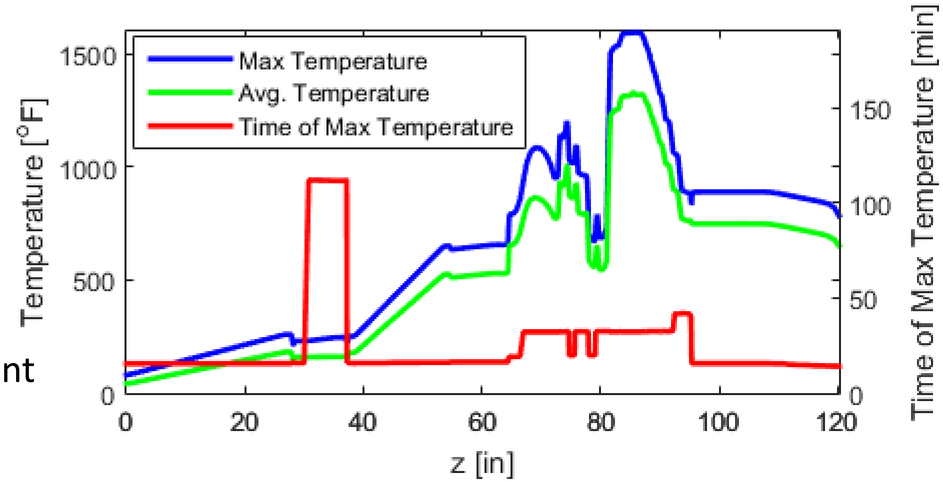
Application: Results



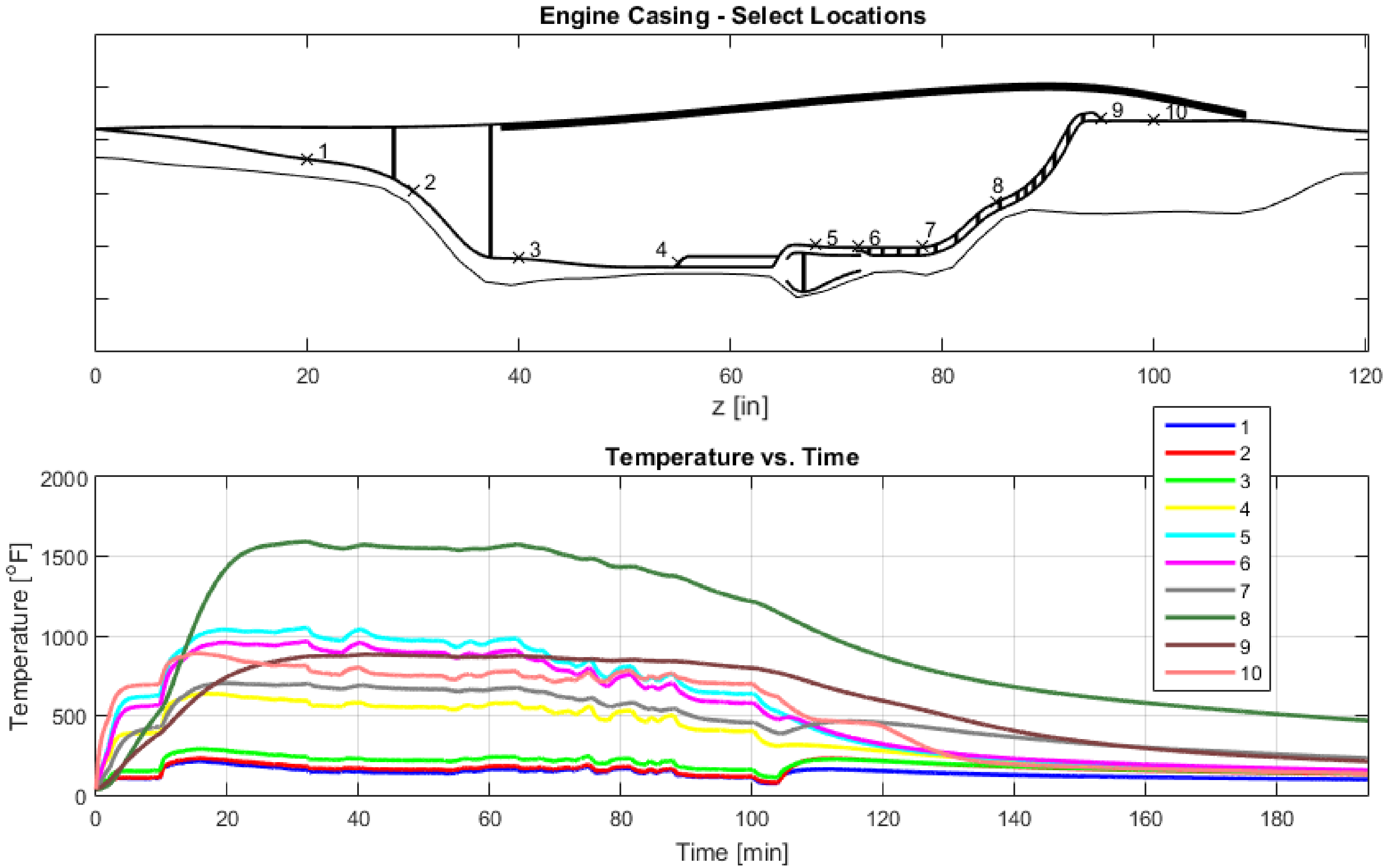
Application: Results

- Compartments
  - Case Cavity 1: Up to 125˚F
  - Case Cavity 2: Up to 155 ˚F
  - Core Compartment (Cowl Cavity): Up to 650 ˚F (400 ˚F upstream of the inter-turbine duct)
- Structure
  - Engine Casing: Up to 1580 ˚F (1100 ˚F outside the inter-turbine duct)
  - Inner Duct Wall (Thermal Blanket): Up to 600 ˚F (350 ˚F upstream of the inter-turbine duct)
  - Case Wall 1: Up to 220 ˚F
  - Case Wall 2: Up to 220 ˚F
- Observations
  - Rate of change in temperature
    - Compartments: -2.5 ˚F/sec – 2.5 ˚F/sec
    - Structures: -2.5 ˚F/sec - 5 ˚F/sec
  - Maximum temperatures occurred at different parts of the engine during different times including: the cold startup of the engine, during climb, during cruise, and during heat soak back



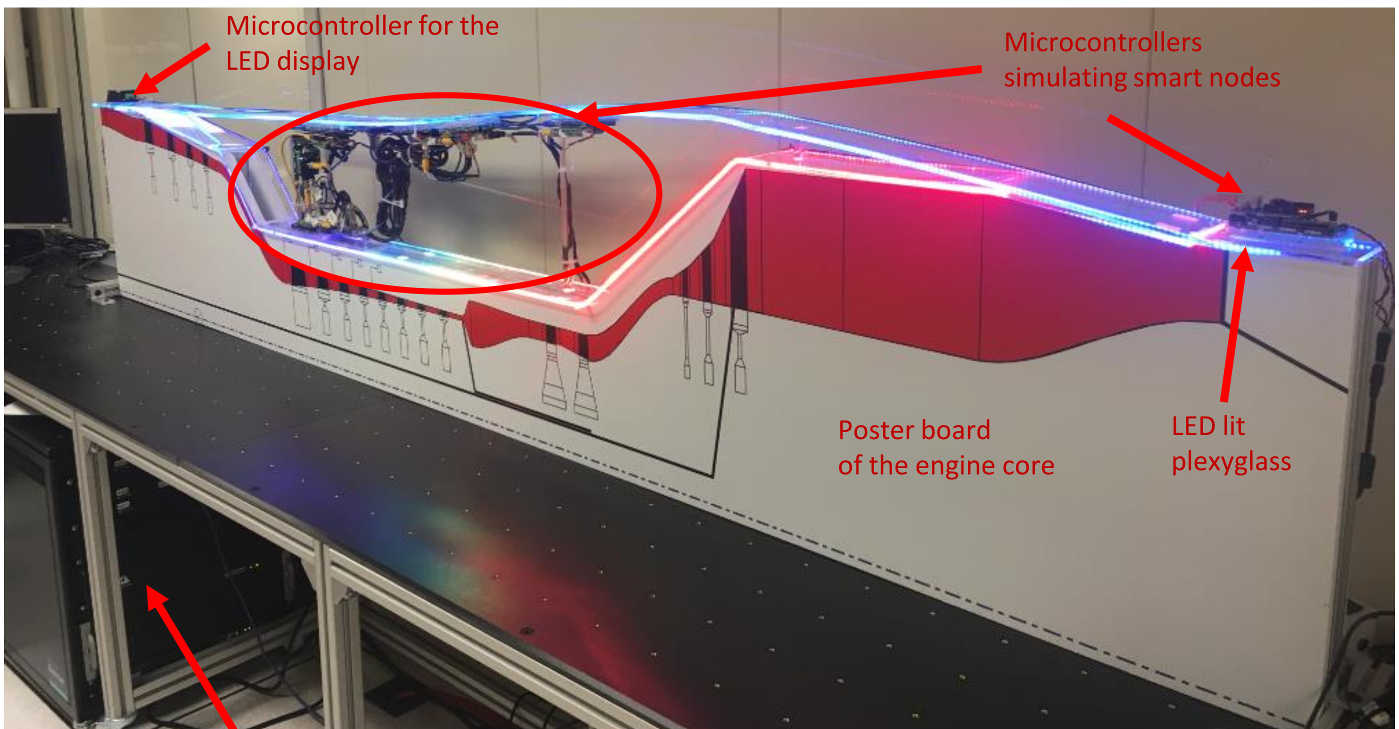


Application: Results



Real-Time Capabilities

- Model runs faster than real-time
- Migrated to the hardware-in-the-loop (HIL) system known as the Decentralized Engine Control System Simulator (DECSS)
- Integrated in a multi-model simulation including a physical network and simulated smart nodes
- Used the model to drive a real-time, full-size LED display
  - Illustrates the ability to interact with test equipment



Summary

- Motivation for high level thermal modeling pertaining to distributed engine control has been discussed
- A thermal modeling methodology for gas turbine engines has been proposed
- An application of modeling methodology has been illustrated
- Results from the application have been presented and discussed
- Real-time capabilities have been demonstrated with eye toward hardware testing

Acknowledgements

- Funded by the Transformational Tools and Technology (TTT) project under the Aerospace Research Mission Directorate (ARMD)
- NASA civil servants & contractors who contributed in some way to this effort: Sanjay Garg, Scott Jones, Jonathan Litt, Jeffryes Chapman, Vikram Shyam, Paht Juangphanich, Ram Bhatt, Jerry Lang, James DiCarlo, Joe Grady, Dan Paxson, and Shane Sowers
- The Distributed Engine Control Working Group (DECWG®) for providing input and guidance related to this work

Questions?

Contact Information:

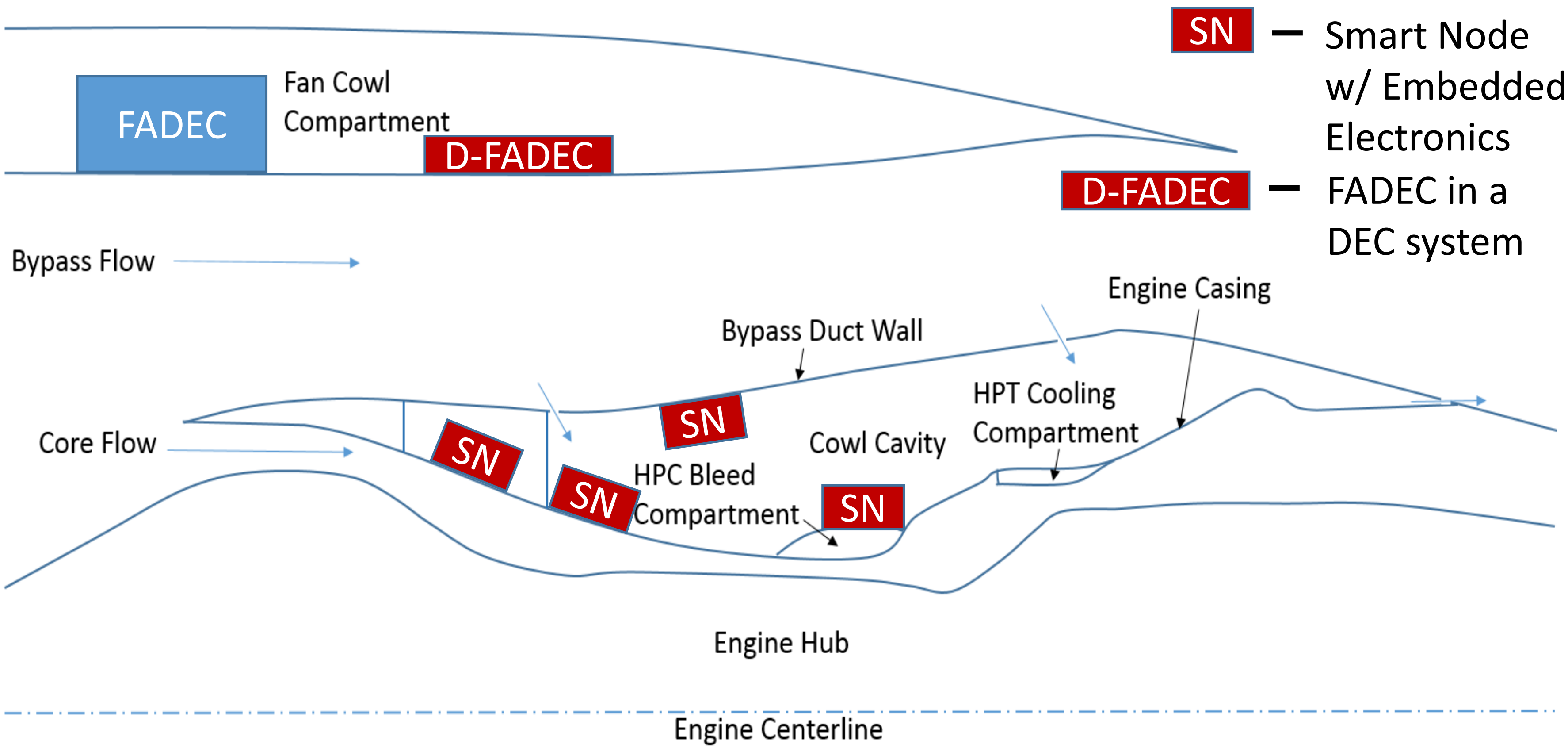
- Jonathan Kratz – jonathan.kratz@nasa.gov
- Dennis Culley – dennis.e.culley@nasa.gov
- George Thomas – george.l.thomas@nasa.gov

TSAT Link:

https://github.com/nasa/TSAT

EXTRA SLIDES

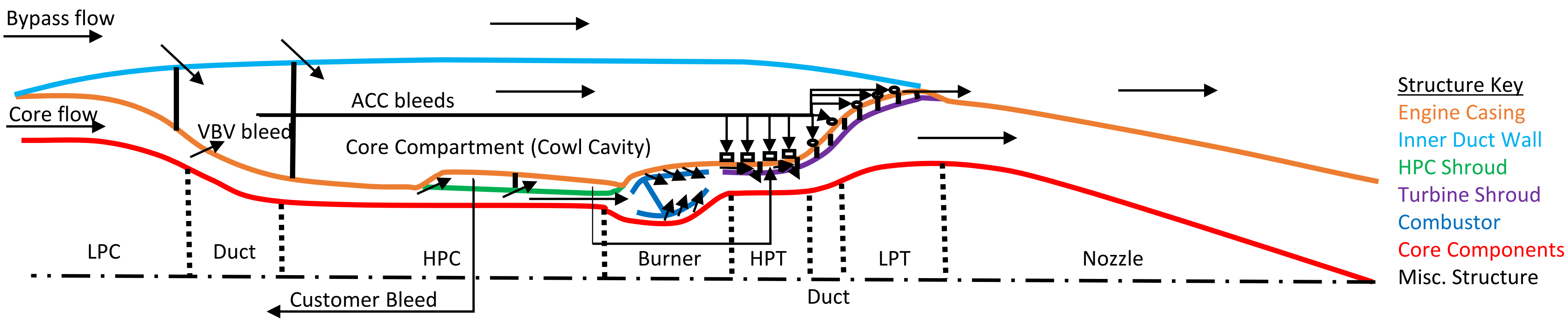
Background: Distributed Engine Control



Where can we mount hardware?

- SN mounting surfaces could include those exposed to the cowl cavity or case compartments (engine casing, bypass duct wall, & various supports structures)

Thermal Modeling Methodology



What part(s) of the engine are we interested in modeling?

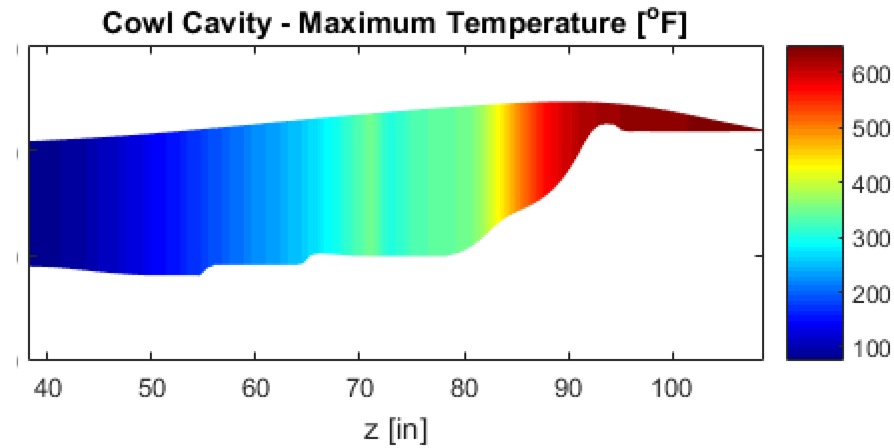
- Any potential mounting structure for a smart node
- Any compartment in which a smart node could be mounted
- Any structure or flow path that could have significant implications on the thermal environment of a potential mounting locations

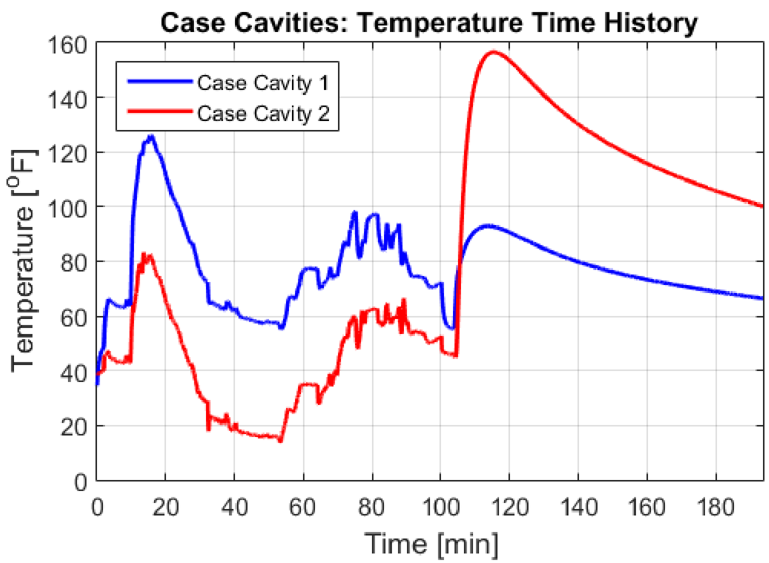
Needs?

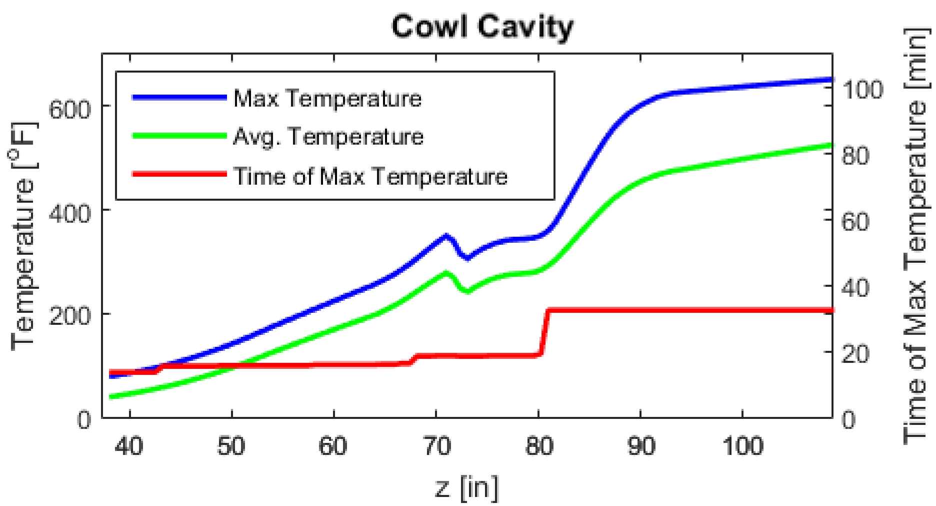
- Geometry/dimensions, secondary air system, other heat transfer/cooling mechanisms inherent in the design

Application: Results

- Max T: 125˚F during take-off (case cavity 1), 155 ˚F during heat soak (case cavity 2), 400 ˚F during climb (applicable portion of the cowl cavity)
- Max Increasing dT/dt: 1 ˚F/sec (case cavity 1 & 2), 2.5 ˚F/sec (applicable portion of the cowl cavity)
- Max Decreasing dT/dt: 0.5 ˚F/sec (case cavity 1), 2 ˚F/sec (case cavity 2), 2.5 ˚F/sec (applicable portion of the cowl cavity)
- Observations
  - Temperature rises several hundred degrees through the cowl cavity (core compartment)
  - Case cavity 1 shows dampening effects comparted to the temperature response of case cavity 2

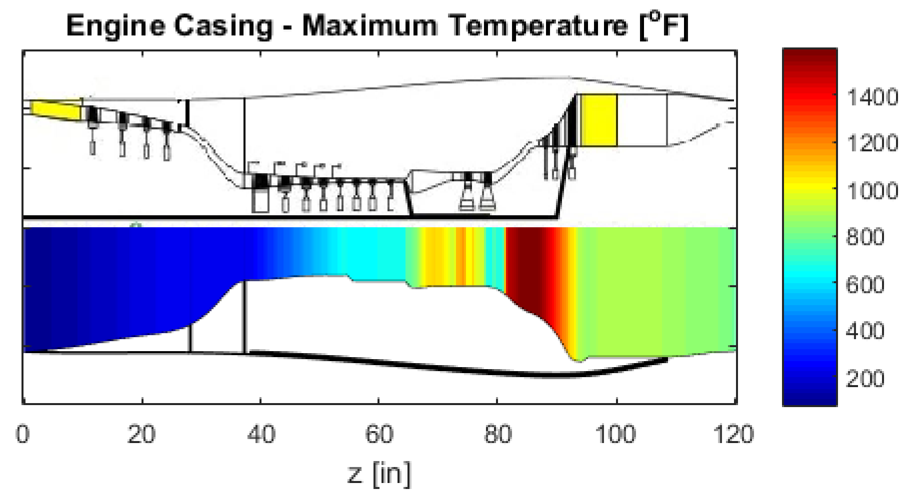


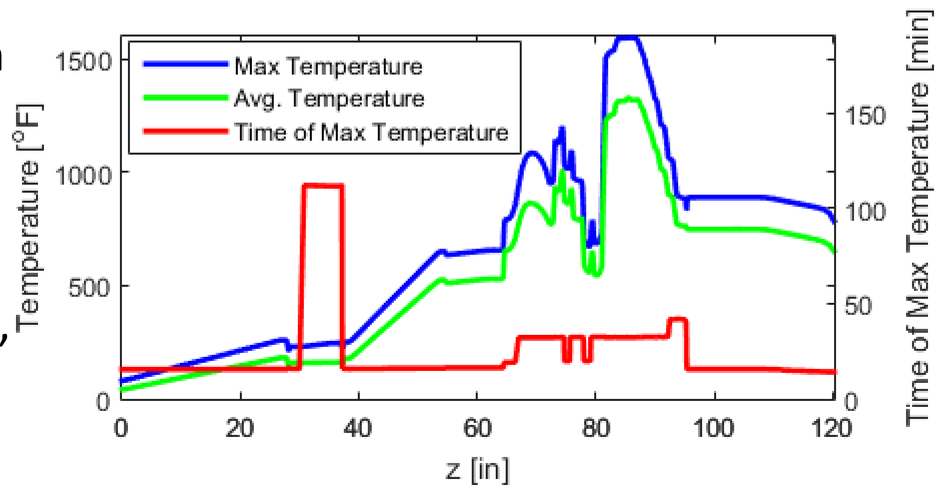




Application: Results

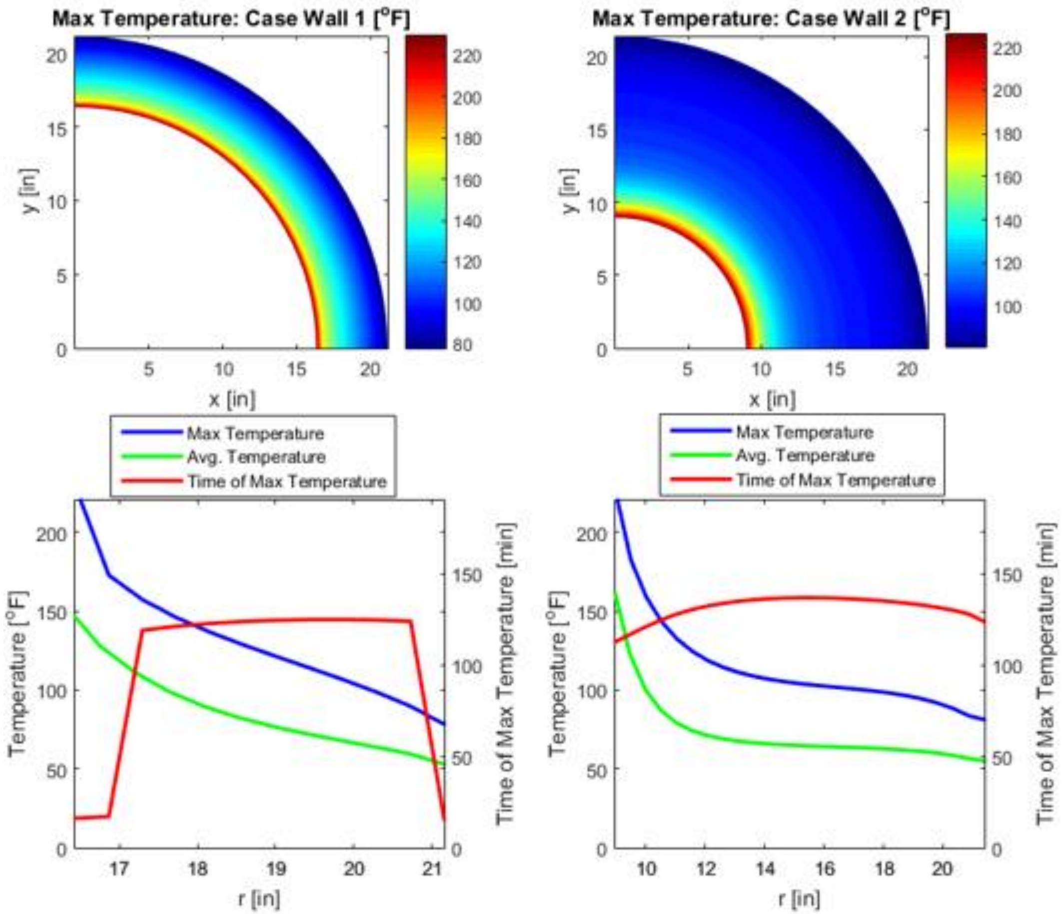
- Max T: ~1580 ˚F (~1100 ˚F neglecting inter-turbine duct)
- Max Increasing dT/dt: 5 ˚F/sec (neglecting nozzle)
- Max Decreasing dT/dt: 2.5 ˚F/sec (neglecting nozzle)
- Observations:
  - Max temperature occurs in the inter-turbine duct region due high temperatures from the aggressive cycle design and lack of active cooling
  - Max temperatures for different locations are shown to occur during climb, cruise, and heat soak





Application: Results

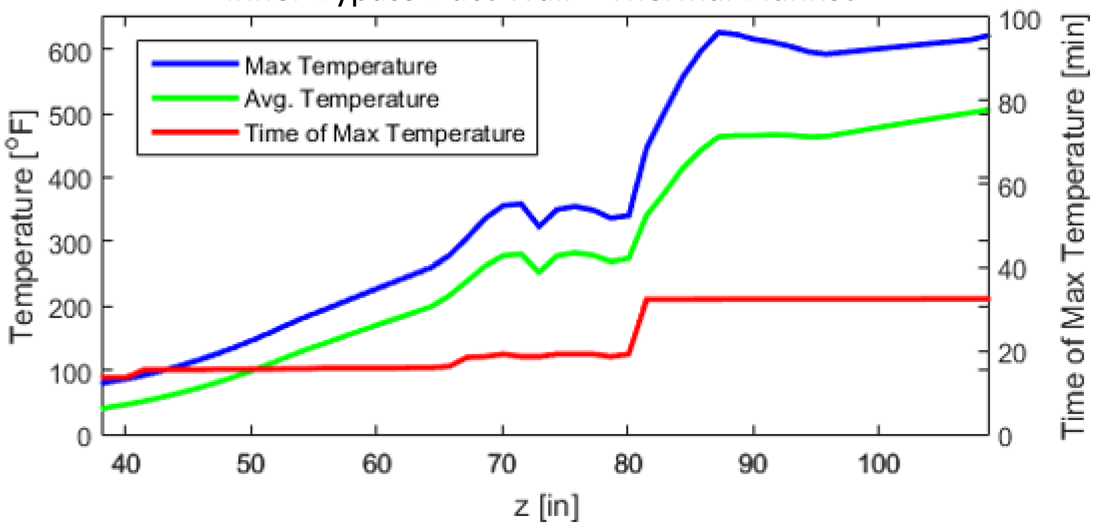
- Max T: ~220 ˚F
- Max Increasing dT/dt: ~1.3 ˚F/sec (at engine case), ~0.3 ˚F/sec (away from the engine case)
- Max Decreasing dT/dt: ~0.7 ˚F/sec (at engine case), ~0.1 ˚F/sec (away from the engine case)
- Observations:
    - Max temperature occur during heat soak, and to a lesser extent take-off



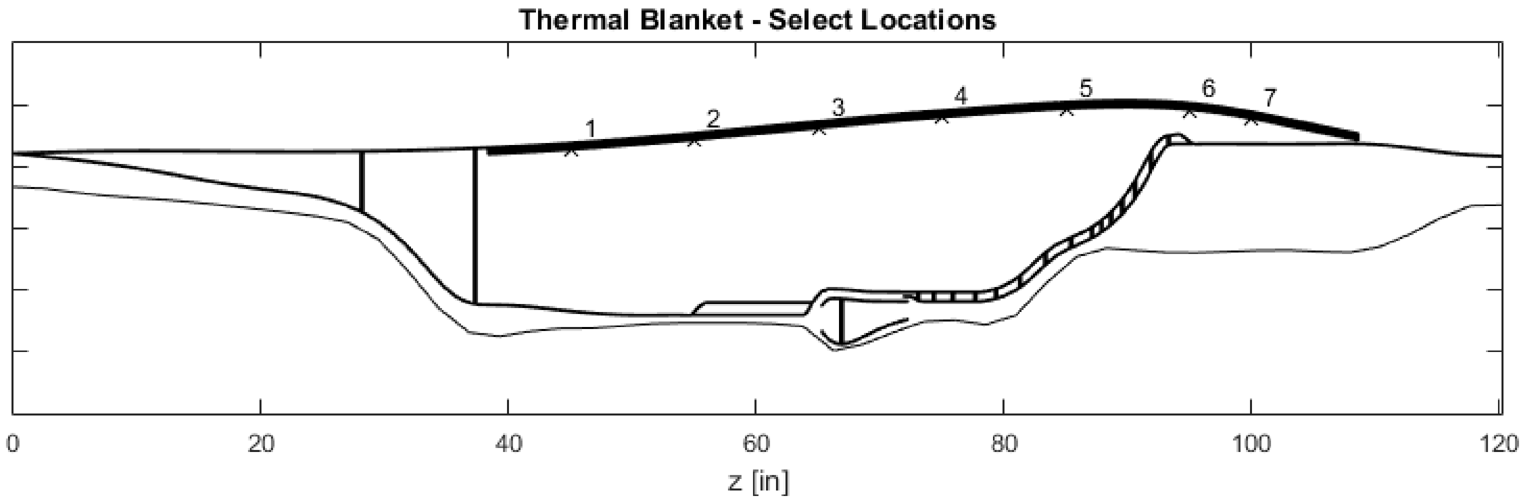
Application: Results

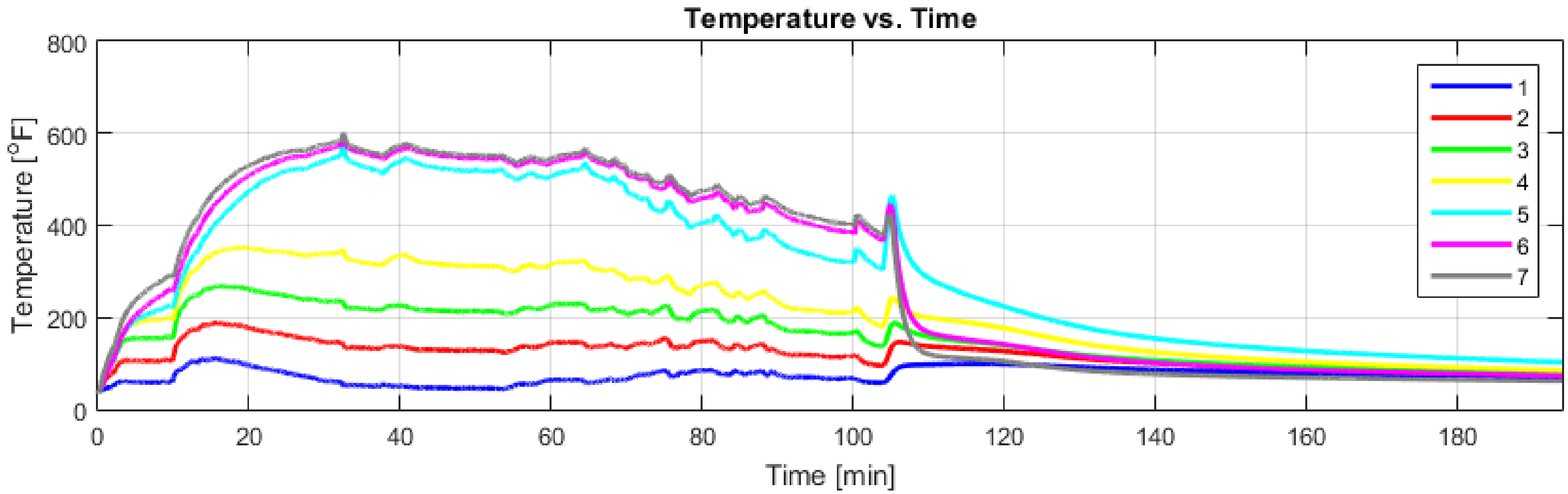
- Max T: ~600 ˚F (~350 ˚F for the region upstream of the inter-turbine duct)
- Max Increasing dT/dt: ~2 ˚F/sec
- Max Decreasing dT/dt: ~1.5 ˚F/sec
- Observations:
  - Max temperatures reached during climb to shortly after reaching cruise

Inner Bypass Duct Wall – Thermal Blanket

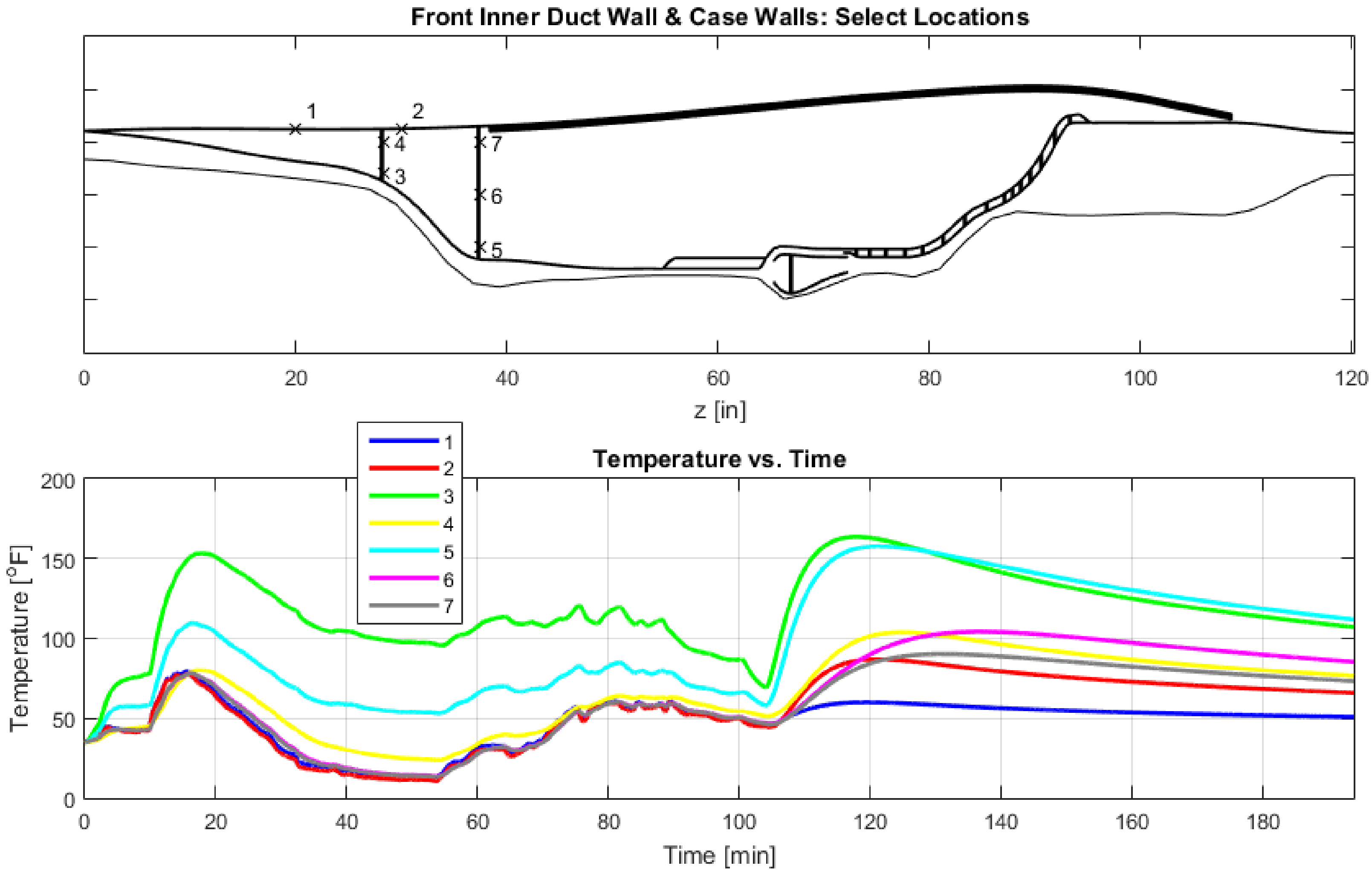


Application: Results

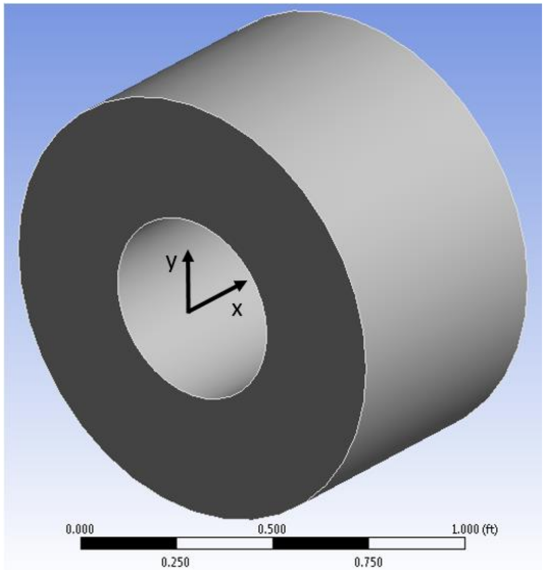


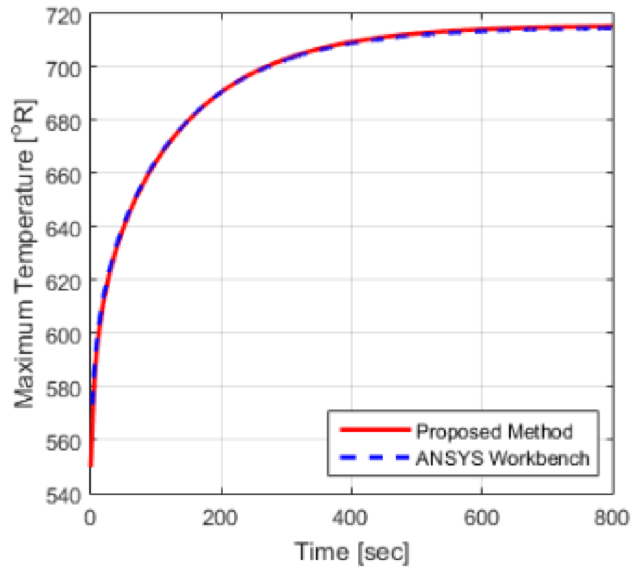


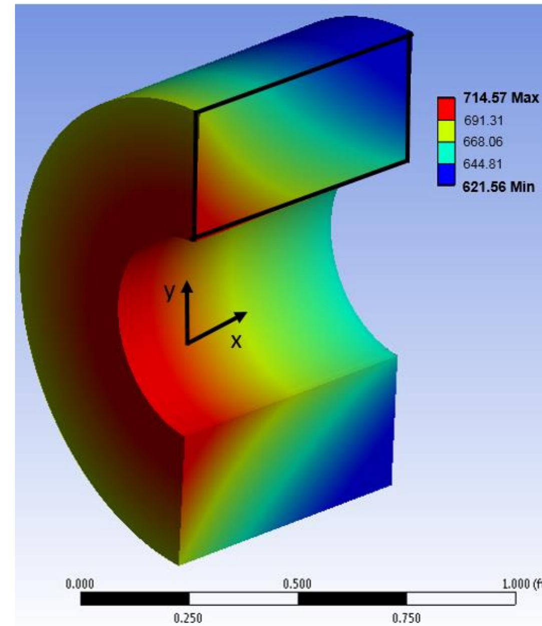
Application: Results



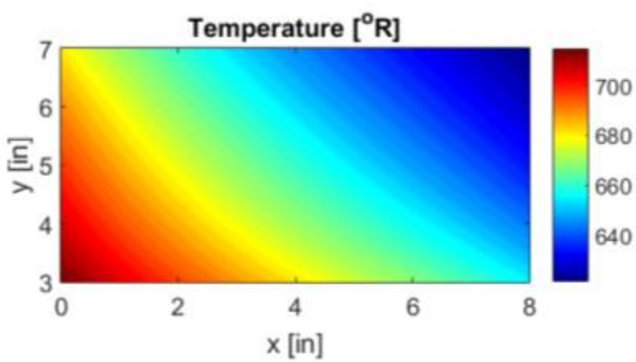
Extra Slides: ANSYS Verification Example







Proposed Method →



ANSYS Workbench ↑