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

- 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

Discretization

Simplified Geometric Representation

Component Modularization

Boundary Condition & Interface Definitions

Model Integration

ENGINE MODEL

THERMAL MODEL

Radiation

Convection

Conduction

1D-Conduction

2D-Conduction

Lumped Mass

Fluid Energy Balance

Component Model Development
Thermal Modeling Techniques

Structure

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

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

\( T = \) temperature, \( t = \) time, \( \rho = \) 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)

\[
\frac{\partial T}{\partial t} = \frac{1}{m C_p} \left[ h A (T_F - T) + u A (T_R - T) \right]
\]

Convection \hspace{1cm} \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

\[ Q = \sum_{\text{out}} (\dot{m}C_pT) - \sum_{\text{in}} (\dot{m}C_pT) \]

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

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

\[ T = \frac{\sum T_s A}{\sum A} \]

\( A = \) 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.

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

A: Case Wall 1 - Left Surface \( \left[ ^\circ F \right] \)

B: Case Wall 1 - Right Surface \( \left[ ^\circ F \right] \)

C: Case Wall 2 - Left Surface \( \left[ ^\circ F \right] \)

D: Case Wall 2 - Right Surface \( \left[ ^\circ F \right] \)

Time: 0 min
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

Engine Casing - Select Locations

Temperature vs. Time

Temperature [°F]

Time [min]
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
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 compared 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 to 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: \( \sim 600 \, ^\circ\text{F} \) (\( \sim 350 \, ^\circ\text{F} \) for the region upstream of the inter-turbine duct)
- Max Increasing \( \frac{dT}{dt} \): \( \sim 2 \, ^\circ\text{F}/\text{sec} \)
- Max Decreasing \( \frac{dT}{dt} \): \( \sim 1.5 \, ^\circ\text{F}/\text{sec} \)
- Observations:
  - Max temperatures reached during climb to shortly after reaching cruise

Inner Bypass Duct Wall – Thermal Blanket
Application: Results

Thermal Blanket - Select Locations

Temperature vs. Time
Application: Results

Front Inner Duct Wall & Case Walls: Select Locations

Temperature vs. Time

- Temperature [°F]
- Time [min]

Legend:
- 1
- 2
- 3
- 4
- 5
- 6
- 7
Proposed Method → ANSYS Workbench