Numerical Investigation of Burning Processes in a Small Gas Turbine Combustor

Alex Dolnik - Rafael

Supervisor: Prof. Yeshayahu Levy - Technion
Studies done in this research

- CFD simulation for 2 combustor configurations - “A” and “B”.
- Validation against experimental data
- Parametric study of combustion chamber
- Optimization of temperature field at combustion chamber outlet
Motivation and Objectives

- Use CFD to explore and understand burning processes in combustion chamber of jet engine
- Develop and test numerical tools and strategies for combustion chamber design
- Apply CFD for parametric study and improve combustor characteristics
Overview of combustion chamber
Mesh generation

- Unstructured mesh
- Low aspect ratio cells
- Automatic mesh generation
- Mesh flexibility and controllability
- Easy integration with CAD tool
In order to validate numerical model, simulation of combustor “A” was carried out and results were compared with experimental data.

3 meshes were created to assure solution is mesh insensitive.

5 different swirl angles were simulated.

Main fuel mass flow rate deviation checked.
Regular Mesh - 240,000 cells
Boundary conditions

- Air mass inlet
- Fuel mass inlet
- Main pressure outlet
- Bearing pressure outlet
- Periodic boundaries
- Wall boundaries
3-step kinetic mechanism

- Introduction of fuel decomposition reaction allows to simulate burning of premixed (lubrication) fuel
  
  1. $JP10 \rightarrow C_7H_{16}$
  2. $C_7H_{16} + 7.5O_2 \rightarrow 7CO + 8H_2O$
  3. $CO + 0.5O_2 \rightarrow CO_2$

- Burning efficiency of lubrication fuel is in a good agreement with early estimation based on experimental data (~50%)
“Main” fuel

\[ \eta_{\text{burn}} = 0.998 \]

“Lubrication” fuel

\[ \eta_{\text{burn}} = 0.545 \]
Combustor “A” simulation results
Temperature field at Combustor “A” outlet

Experimental data

Simulation results
Total Temperature Distribution at Combustor Outlet - Hub

- Experimental data
- Regular Fuel Flow
- Increased (+5%) Fuel Flow
- Reduced (-5%) Fuel Flow

Experiment – Simulation Comparison
Experiment – Simulation Comparison

Total Temperature Distribution at Combustor Outlet - Mid

- **Experimental data**
- **Regular Fuel Flow**
- **Increased (+5%) Fuel Flow**
- **Reduced (-5%) Fuel Flow**

![Graph showing total temperature distribution at combustor outlet](image-url)
Total Temperature Distribution at Combustor Outlet - Tip

- Experimental data
- Regular Fuel Flow
- Increased (+5%) Fuel Flow
- Reduced (-5%) Fuel Flow
Swirl angles variation

0 deg

4 deg

8 deg

12 deg

16 deg
Swirl angles variation

T04 Profile vs Swirl Angle

Total Temperature, [K]

R / Rmax

0 deg
4 deg
8 deg
12 deg
16 deg
Damage to vaporizers in Combustor “A”

Damaged vaporizer

Simulation results
Possible reason for vaporizers damage
Combustion products leakage from primary zone
Combustor “B” simulation

Vaporizer was redesigned to protect it from thermal damage.
Combustor “B” simulation
Combustor “B” simulation

T04 Radial Profiles - Combustor "A" vs Combustor "B"

Total Temperature, [K]

R / Rmax

Combustor "A"

Combustor "B"
Dilution zone redesign

“new” combustor

optimized combustor
Temperature profile optimization

T04 Profile Optimization

- Optimal Profile
- Block A
- 1
- 3
- 6

Total Temperature, [K]

R / R_{max}
Temperature profile optimization

Optimized Combustor

Combustor “B”
Conclusions

- CFD model was successfully applied to simulate combustion chamber
- Simulation results come with good agreement with data measured in experiments
- Simulations were able to predict different design problems that were faced during experiments
- Parametric study was carried out to check influence of swirl angle and fuel deviation on temperature profile at combustion chamber outlet
- Optimization study was carried out to improve the temperature profile for combustor “B”