CFD MODELING OF SYNGAS COMBUSTION IN GAS TURBINE CONDITIONS

Abdallah Abou-Taouk & Lars-Erik Eriksson
Division of Fluid Dynamics
Department of Applied Mechanics

20100415
Outline

- Introduction
- Background and Motivation
- Objectives
- Industrial Relevance
- Combustion Modeling
- CFD Modeling
- Summary
Background and Motivation

• The aim of the project is to investigate swirl-stabilized flames, mainly with syngas fuel, with respect to combustion- and turbulence modeling.

• In the gas turbine community the development of combustor technology is currently following the general trend towards fuel-flexibility and increased use of bio fuels.

• Improved combustion modeling of flames → improved combustion design with respect to combustion efficiency and emissions.
Objectives

• The goal is to improve the predictions for the complex chemistry-turbulence interactions.
• Derivation of new multi-step global mechanisms for syngas flames, coupling these multi-step mechanisms with an established turbulence interaction model
• CFD analysis for the SIT syngas burner test rig at LTH/Combustion Physics
• Validation of CFD against experimental data
Industrial Relevance

- Siemens: The SIT syngas burner is important from an environmental point of view since syngas fuel gives the opportunity and flexibility to run on both bio- and fossil fuels.
- Volvo Aero: Maintaining and developing the afterburner competence
- Improvement of currently used CFD tools, RANS or URANS-based techniques, both in terms of combustion modeling and turbulence modeling.
Combustion Modeling Methodology

- Multistep global reaction mechanisms
  - Key to efficient coupling with CFD codes
  - Robust optimization of parameters
  - Comparison with standard detailed mechanism
  - CANTERA software
Combustion Modeling

- CANTERA PSR code is able to handle detailed mechanisms, eg GRI Mech 3.0, etc
- CANTERA PSR code has difficulty with global mechanisms (numerical reasons)
- New PSR code developed for multi-step global mechanisms (FORTRAN code)
- Comparisons with detailed mechanism done for methane/air PSR
  - P=1 atm, Tin=295K, fi=0.7
  - Meredith 3-step global scheme (done)
  - Meredith 5-step global scheme (done)
  - Westbrook Dryer 2-step global scheme (ongoing)
Combustion Modeling

Meredith 3-step scheme:
\[ 2CH_4 + 3O_2 \rightarrow 2CO + 4H_2O \]
\[ 2CO + O_2 \rightarrow 2CO_2 \]
\[ 2CO_2 \rightarrow 2CO + O_2 \]

Meredith 5-step scheme:
\[ 2CH_4 + O_2 \rightarrow 2CO + 4H_2 \]
\[ 2H_2 + O_2 \rightarrow 2H_2O \]
\[ 2CO + O_2 \rightarrow 2CO_2 \]
\[ 2H_2O \rightarrow 2H_2 + O_2 \]
\[ 2CO_2 \rightarrow 2CO + O_2 \]
The results show good agreement between the detailed and the 3-step mechanism, while the 5-step mechanism for methane shows somewhat larger differences.
Combustion Modeling – Near Future

• Comparisons between WD2 mechanism and detailed mechanism for methane/air PSR.
• Find acceptable detailed mechanism for syngas + methane/air mixtures
• Set up optimization algorithm in MATLAB
• Optimize global scheme for selected cases
CFD Modeling

- Analyses are carried out in two different meshes
- The first results have been obtained for the fine mesh, including steady-state RANS analysis as well as unsteady hybrid URANS/LES analysis
- Softwares used:
  - Meshing: ICEM CFD
  - CFD: ANSYS CFX
CFD Modeling – Geometry

- Burner geometry file received from SIT (SIT syngas burner)
**CFD Modeling – Geometry**

– The geometry consists of three systems: RPL, Pilot, and Main
CFD Modeling – Meshing

- Mesh 1: ~8M tetra cells
CFD Modeling – Meshing

- A lot of funny work! (2 month meshing)
- Mesh 2: ~7.5M hexa cells
CFD Modeling – Method

- Code: ANSYS CFX 11
- Compressible flow, Methane gas
- Steady state: Turbulence Model k-ω SST
- Transient: Turbulence Model SAS SST
- Finite Rate Chemistry and Eddy Dissipation Model
- Reaction: Westbrook Dryer 2-reaction mechanism for methane
- In the start of the project CFD on Mesh nr1 have been done in order to understand the flow path of this complex system
- The major focus regarding the CFD analyses have been done on the fine mesh
CFD Modeling – Boundary Conditions

– Inlet
  – Total Temperature
    – Main: 650 K
    – Pilot: 650K
    – RPL: 300K
  – Mass flow rates are set so the φ-number is equal to
    – Main: 0.48
    – Pilot: 0.48
    – RPL: 1.6

– Outlet
  – Constant static pressure – 101325Pa

– Walls
  – All the walls are set to adiabatic
CFD Modeling – Results

Steady state: $k-\omega$ SST
CFD Modeling – Results

Steady state: $k-\omega$ SST
CFD Modeling – Results

\[ \varphi = \frac{z \cdot 4}{(1-z)Y_{O_2}} \]

\[ z = Y_{CH_4} + \frac{16}{28} Y_{CO} + \frac{16}{44} Y_{CO_2} \]

Steady state: k-\omega SST
CFD Modeling – Results

Transient: SAS SST model
Timestep: 7e-6s
Data extracted every 10 timestep during 500 timesteps
CFD Modeling – Results

Transient: SAS SST model
Timestep: 7e-6s
Data extracted every 10 timestep during 500 timesteps
CFD Modeling – Results

Transient; SAS SST model
Timestep: 7e-6s
Data extracted every 10 timestep during 500 timesteps
CFD Modeling – Near Future

- Validation of CFD against experimental data
- Modify WD2 according to the results obtained in the PSR
- Implement the global mechanism for syngas
- CFD-calculations in the in house code VOLSOL++ at Volvo Aero
Summary

- CANTERA PSR works well for detailed kinetics
- New PSR code developed for multi-step global mechanisms
- Comparisons with detailed mechanism done for methane/air PSR
- The meshing work is finalized
- First results obtained for steady-state RANS analysis as well as unsteady hybrid URANS/LES analysis
Thank you…

- Questions?