Thermomechanical Design of a Gas Turbine Reheat Combustor Experiment Using FEM Analysis with the COMSOL Multiphysics® Software

F. M. Berger[1], M. Eser[1], T. Sattelmayer[1]

[1]Lehrstuhl für Thermodynamik, Technical University of Munich, Munich, Germany

Abstract

Enhanced operational flexibility and low levels of pollutant emissions are achieved with a sequential arrangement of premixed combustion stages in gas turbines for power generation. In the second - reheat - combustion stage, hot flue gases of approximately 1500K are enriched with fuel and establish a self-igniting flame - i.e. flame stabilization occurs mainly through auto-ignition. This work is part of a research project that investigates the combustion dynamics in such a reheat combustor and introduces the thermomechanical design of the combustor experiment.

Using COMSOL Multiphysics® software, a quasi-static thermomechanical analysis is carried out to ensure that the model combustor withstands gas and flame temperatures of 1500 to 2200K at high mass flow rates and allows for optical access to the combustion for diagnostic techniques at the same time. Furthermore, heat losses due to cooling must be kept at a marginal level to sustain the very temperature-sensitive auto-ignition combustion process. The resulting combustion chamber structure is made up of a water cooled steel frame with integrated quartz glass windows. The thermomechanical design target was to reduce thermal expansion of the metal parts due to high local thermal loads and allow for air-tight coupling of the metal and quartz glass structure. High temperature gradients and resulting thermal stresses are localized by the analysis procedure to optimize the structure geometry and the cooling concept.

As an academic design approach, the equations describing the uncoupled, quasi-static thermomechanical system are manually implemented into COMSOL's Weak Form PDE Tool, as indicated in Figure 1. This allows for full control and comprehension of the Finite Element analysis. Hence, the system can be solved subsequently for the thermal distribution in the structure, as well as for resulting mechanical stresses. To simplify the implementation, thermomechanical variables are defined prior to simulation. Differing material parameters are inserted with a user defined function in order to assess critical material coupling effects due to different coefficients of thermal expansion and mechanical stress resistance. The domain is discretized using COMSOL's user-defined free tetrahedral mesh option. Automatic choice of predefined COMSOL solvers is applied to obtain the solution. Thermal boundary conditions are taken from respective conjugate-heat-transfer simulations and mechanical constraints are set in a way to resolve critical parts' interaction and identify local peak stresses.

The elastic deformation of the combustion chamber is obtained for different thermal load scenarios and allows to assess the operation ability of the combustion system. Thermal stress distributions within the individual components are analyzed in detail and design

improvements for appropriate material coupling and thermomechanically stable detail design are identified. Furthermore, local cooling optimization for appropriate instrumentation with sensors - i.e. for combustion diagnostics - and optimization of wall cooling for marginal heat loss in the system are obtained. A representative distribution for the ratio of thermal stress to maximum allowable stress for the investigated metal frame is given in Figure 2.

Figures used in the abstract

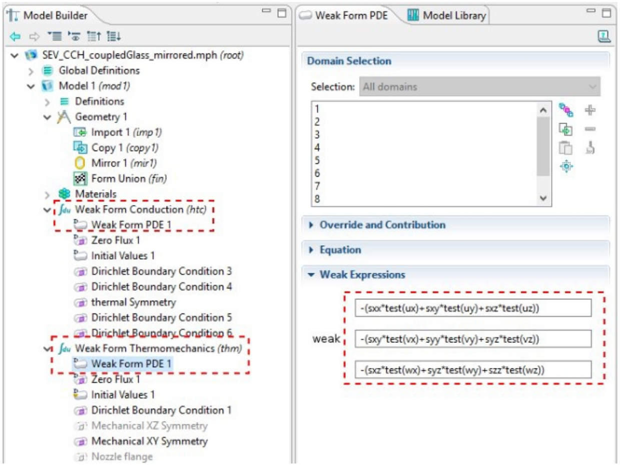

Figure 1: Implementation of the thermomechanical weak form equations into the COMSOL Multiphysics PDE tool.

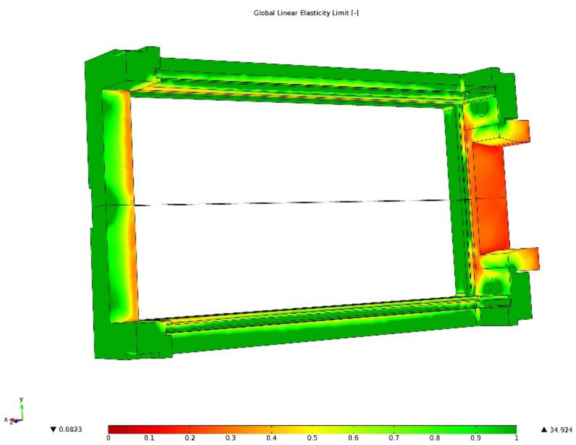

Figure 2: Representative distribution of the ratio of thermal stress to maximum tolerable stress in the frame structure.