ICSV14
Cairns • Australia
9-12 July, 2007
14th International Congress on Sound & Vibration

NUMERICAL STUDY OF STRUCTURAL VIBRATION INDUCED BY COMBUSTION NOISE – ONE WAY INTERACTION

Artur K. Pozarlik, Jim B.W. Kok

Laboratory of Thermal Engineering, University of Twente
P.O.Box 217, 7500 AE Enschede, The Netherlands
a.k.pozarlik@utwente.nl

Abstract

The turbulent flame in the lean combustion regime in a gas turbine combustor generates significant thermo-acoustic noise. The thermo-acoustic noise induces liner vibrations that may lead to fatigue damage of the combustion system. This phenomenon is investigated in the project FLUISTCOM using both an experimental and a numerical approach. The correlation between acoustic pressure oscillations on one side and liner vibrations on the other side is a prime interest.

In order to have better insight in the processes present in the combustion chamber, a combustion test rig was designed and manufactured at the University of Twente. One of the most important parts of the test rig is a liner with a flexible section and optical access to measure the vibration pattern and amplitude. This paper describes a flame investigated at 1.5 bar, 125 kW with premixed natural gas and air. The experimental measurements of the vibrations are done with the use of a Laser Doppler Vibrometer. CFX-Ansys was used for the transient numerical calculations of the transient combustion flow within the combustion chamber. Simultaneously, the pressure results from the near-wall region were collected and sent as initial conditions to a structural code (Ansys). Results show the one way response of the liner structure as a result of the transient pressure generated by the combustion of the gas flow.

The paper will present the numerically predicted results on the combustion field, the accompanying oscillating pressure field, and the induced structural vibration of the combustor liner. These results will be compared with the available experimental data.

1. INTRODUCTION

The life time of a typical gas turbine is mostly limited by thermal and mechanical loads occurring on the turbine blades and in the combustion chamber liner [1]. The introduction of lean premixed combustion decreases the temperature of the combustion gases, but does not extend the life time of a turbine. Even in opposite, as a consequence of high acoustic pressure oscillations, the life time of the liner is significantly reduced. During the lean combustion, the flame with its enormous thermal power amplifies acoustic pressure changes inside the combustion chamber [2,3]. The flame is an acoustic source and produces the sound waves which travel downstream of the combustion chamber and induce a vibration in the liner structure. These liner vibrations force additional changes in the acoustic pressure field inside the combustion chamber (two-way interaction). Finally, the acoustic waves propagate upstream to the flame after reflection from the combustor exit. The acoustic source is modified by the reflected waves and it starts to produce even stronger pressure fluctuations inside the combustion chamber. This behaviour leads to a higher vibration amplitude of the liner and finally to even stronger changes in the reflected waves and flame itself. This closed feedback loop is hazardous, when the acoustic eigenfrequency is close to the resonance frequency of the structure. Both processes can amplified each other and finally lead to fatigue damage of the liner.

The possibility of the liner damage should be predicted and localized before it starts to be dangerous for the whole gas turbine. Numerical approach is a reliable method to state the liner elements prone to fatigue defects. During numerical computations, the fluid and the structure domains, and interaction between them must be taken into account in order to reliable indicate the weakest points of the turbine. Nowadays, two basic ways of fluid-structure interaction modelling are recognized. The first method is based on directly solving the equations describing the fluid and the structural model within a single system. This method is called monolithical or direct approach and needs usually a development of a new code [4]. Other methods are taking advantage of already existing fluid and structural codes, by combining them in connected operation is called partitioned or iterative method. In this method the fluid and the structural solver communicate with each other with the use of an interface connection code, which assures no data transfer loss between the combined codes [5]. In this paper, the second approach is used as the combustion flow with associated phenomena and the structure vibrations are difficult to describe inside one solver. Numerical investigations are performed with the use of the CFX-10 commercial code for the reacting fluid flow calculations and the Ansys FEM package for structural deformations, respectively [6]. Both codes are coupled with the use of the MFX code available within the Ansys package. Numerical predictions are compared with the experimental data [7,8].

2. EXPERIMENTAL DATA

In order to measure vibration of the liner during combustion process, the setup available at the University of Twente is equipped with the system of windows (fig.1). This transparent system allows optical measurements of the liner velocity. The velocity amplitude is obtained from the measurements with the use of a Laser Doppler Vibrometer. The laser is fixed to a translational system in order to measure liner vibrations during combustion at different locations. The velocity amplitude is obtained from the comparison of the frequency of two laser beams, the reference beam and the scattered beam reflected by the vibrating body [8]. A flexible section with a thickness much smaller than the major part of the liner is located directly behind the transparent window (fig. 1). This thin, flexible section responds stronger to any changes in the pressure field inside the combustion chamber during the transient combustion. Pressure fluctuations inside the combustion chamber are measured with the use of the pressure transducers located along the liner walls. To prevent changes in the vibration pattern, there are no pressure sensors, thermocouples or other measurement equipments attached to the flexible wall [7,8]. The data from the pressure sensor closest to the flexible wall are here presented.

Oscillations in the lean combustion process at elevated pressure are forced by fluctuations of the fuel to air equivalence ratio during the experiment. The results of the numerical analysis are compared with the experimental data at conditions depicted in table 1.

Table 1. Operating conditions during combustion experiment.

Thermal power [kW]	Absolute pressure [bar]	Air factor [-]	Total mass flow rate [g/s]	Air preheating temperature [K]	Forcing frequency [Hz]	Forcing amplitude [%]
125	1.5	1.8	75.53	573	300	8.5

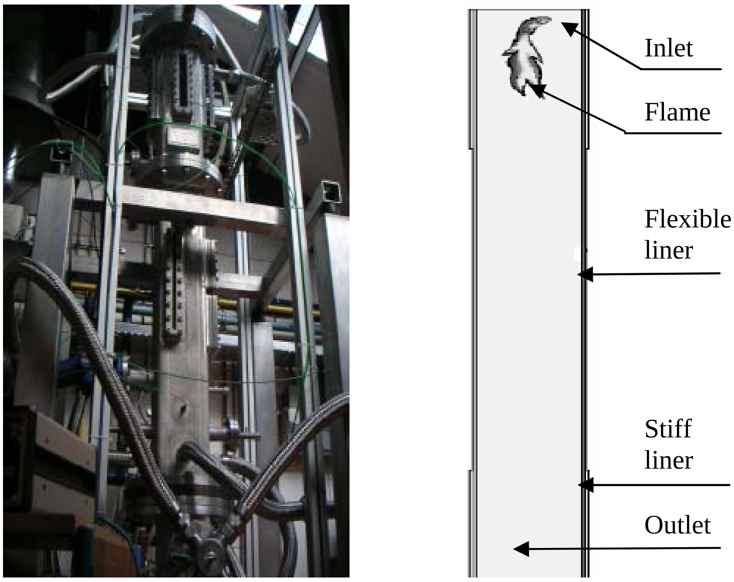

Figure 1. Experimental setup (left) and sketch of the liner geometry (right)

3. NUMERICAL DATA

Numerical calculations are performed with the use of the commercial codes from Ansys: CFX-10 for CFD analysis and the Ansys-10 FEM package for structural calculations. Additionally, Workbench-10 is used for the unstructural mesh generation and the MFX code for coupling CFX with Ansys-10.

3.1. CFD model

Numerical calculations of the reacting flow inside the combustion chamber are done for a quarter section of the initial geometry. This decreases significantly the number of elements used for CFD analysis and thereby reduces the total computational time. Results of a previous CFD analysis on the full geometry involved the flow inside the plenum and the combustion chamber served as input inlet conditions. The static average pressure is imposed at the combustion chamber outlet. Periodic conditions are prescribed on the side walls and no-slip adiabatic conditions on the cover walls. The Eddy Dissipation and Finite Chemistry Rate model is used for the combustion flow calculation. The standard k-ε model as available in CFX served for modeling the turbulence. The equivalence ratio of fuel to air is pulsated with a frequency of 300 Hz and amplitude equal to 8.5%. The results of the pressure fluctuations are collected downstream of the combustion chamber, at a position corresponding to the location of the pressure sensor during the experiment. This numerical investigation simulates the experiment carried out under the conditions depicted in table 1. A total number of 632 000 unstructural elements is used for the calculations. The non-uniform mesh with a higher element density close to the flame zone is prescribed to capture better phenomena occurring during the combustion process.

3.2. CSD model

Parallel to the CFD simulation, the numerical simulation of the combustor wall is done. The liner of the combustion setup consists of two parts connected by a sliding connection. This assures small thermal stresses during the work at elevated temperature as both liner parts can freely expand in axial directions. The only thermal stresses come from the mutual influence of the expanding plates at the flanges and at the corners of the combustion chamber. The magnitude of these stresses is too low in any place of the liner cause damage of the walls. Therefore, during calculations it is assumed that the influence of temperature on thermal stresses is minor. The material properties adequate to the uniform liner temperature (760^{o}C) are used without possibility of the wall thermal expansion in any direction. Similar to the combusting flow calculations, where only quarter part of the real combustion chamber was used, the structural model is reduced to the one liner wall. The wall is simplified further to the plate without sliding connection and holes for thermocouples and pressure transducers (located at stiff parts), as they have minor influence on the vibration amplitude. In the downstream section of the liner, the wall has much smaller thickness with comparison to the surrounding structure. This section is placed directly behind the transparent window in the pressure casing and it is vibrating at high amplitudes. Localization of the flexible part between much stiffer, rigid bodies assures well defined boundary conditions. The side parts of the liner are treated as clamped. On the inside face, the mechanical loads, i.e. pressure and shear from the CFD calculation are transferred (fig. 3).

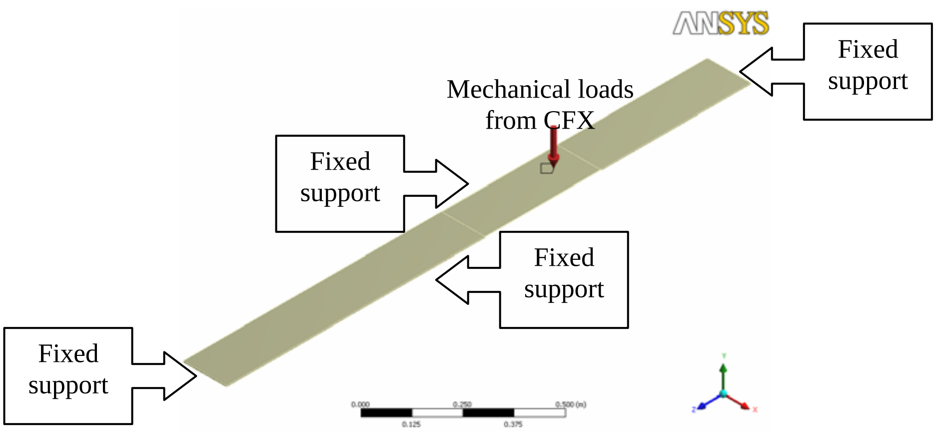

Figure 3. Boundary conditions used for structural analysis

A total number of 19 000 equally distributed SOLID92 tetrahedral elements is used for the dynamic calculation. Material properties of stainless steel 310 SS adequate to conditions at elevated temperature, together with geometrical dimensions of the wall, are depicted in table 2.

Table 2. Dimensions and material property of the liner wall

Total length [m]	**Stiff parts thickness [m]**	**Wall width [m]**	**Flexible part length [m]**	**Flexible part thickness [m]**	**Material density [kg/m^3]**	**Young's modulus [GPa]**	**Poisson's ratio [-]**
1.813	0.004	0.150	0.400	0.0015	7844	138	0.3

3.3. Coupling CFX and Ansys

Exchange of information between CFX and Ansys is possible with the use of the MFX code. The bucket search method is used as mapping algorithm [9]. Mechanical loads are transferred from the CFD code to the FEM package through the interface created on the one side of the liner structure. The global conservative interpolation method is used to interpolate results from CFD analysis to the structure face [6]. For loss less information transfer, both numerical models must be coincident in space. Information about forces is shared every time step equal to 0.3 ms. No information about displacement is sent back to the CFX code as the analysis is one-way interaction. Total calculation time is set to 0.1 s.

3.4. Modal analysis

To predict the hazardous resonant frequencies, the full model (4 walls) of the liner with the combustion chamber and the cooling passage cavities is investigated in the modal analysis. The wall and the air cavities have a temperature consistent with operating conditions. The liner flexible section determines mostly the model's eigenfrequency, thus only this section is taken into account in the analysis. The combustion and cooling air close to the wall is described with the use of the FLUID 30 acoustic elements with the ability to recognize fluid on one side and structure on the other. Regular fluid elements FLUID 30 are placed far from the structure. A model of the liner is made by SHELL63 structural elements. To prevent the direct connection of the pressure nodes from the combustion chamber and cooling passage at the structure interface, the model of the liner is divided into two equal parts. These parts together have exactly the same properties and stiffness like a non divided model. Structural parts are connected together by stiff degrees of freedom, thus any changes in one section forces the same changes in the other one (fig. 4). The walls are clamped on both ends.

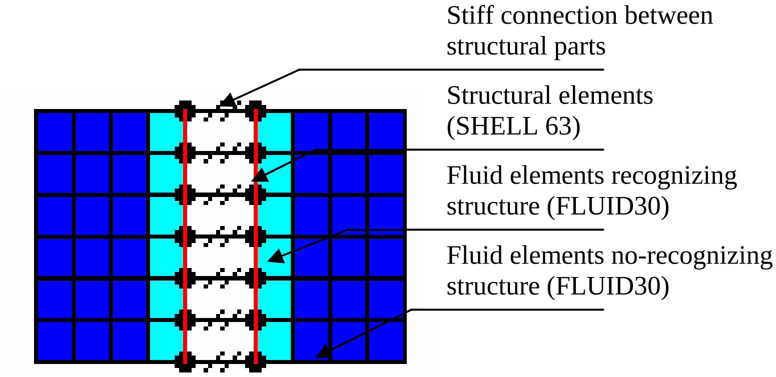

Figure 4. Connection between different elements during modal analysis

4. RESULTS

Numerical results for the pressure as a function of time, obtained during the one-way interaction between reacting flow inside the combustion chamber and chamber walls vibration, show generally good agreement with the experimental data (fig.5-6). However, some under-prediction of the amplitude by the numerical results can be noticed. It is most likely a consequence of the numerical approach used for the CFD analysis. The pressure fluctuations obtained during URANS calculations are a factor of 2 smaller than observed during the experiment (fig. 5). These fluctuations originate mostly from the changes in the main flow caused by fuel to air equivalence ratio pulsations. In the URANS approach the grid resolution and the time step used for the analysis causes a significant numerical damping of the acoustic field. A smaller time step, higher grid resolution as well as the use of different computational approach (e.g. LES) would improve resolving the acoustics but it would lead as well to a significant extension of the computational time [10]. Thereby the changes in the liner velocity are caused mostly by the changes in the mean flow and accompanying pressure variations.

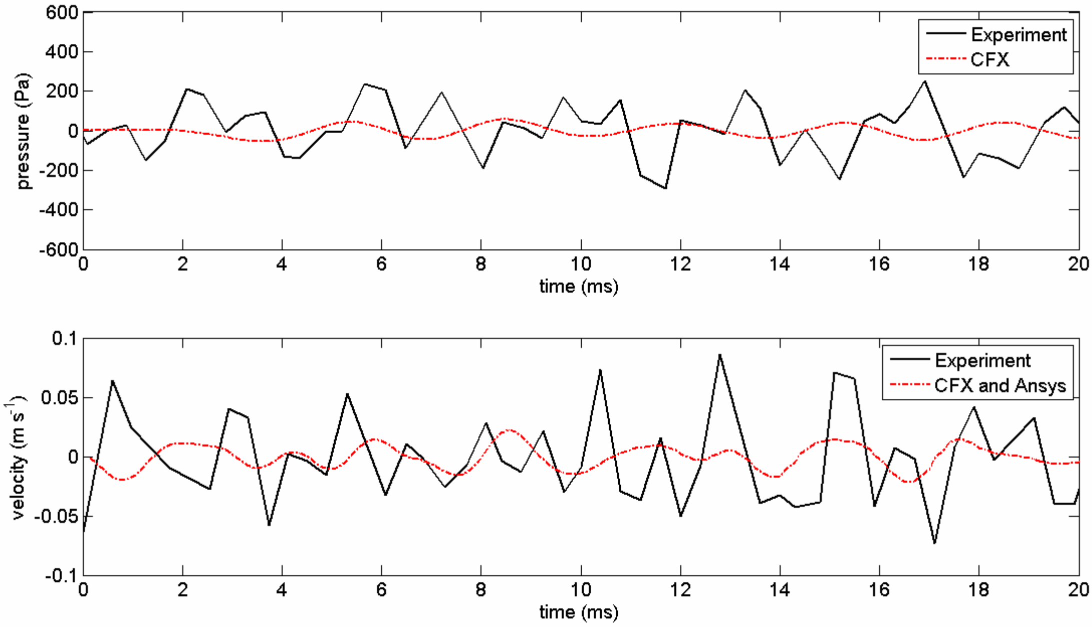

Figure 5. Time history of the change of the pressure (top) and velocity (bottom)

The sinusoidal shape of the pressure change shows that the pressure field is only affected by changes in the equivalence ratio fuel to air ratio (fig 5, top). The flame source thus is not changed by the acoustic wave which could reflect from the rigid boundaries i.e. walls. Because of one-way calculations, there are no changes in the pressure field caused by vibrations of the liner as there is no feedback from structural to fluid code [11]. Therefore a clearly sinusoidal signal of pressure change in the combustion chamber is obtained.
As a result of the smaller fluctuations in the pressure field also the magnitude of the velocity change during liner vibration is smaller (fig.5, lower part). The order of magnitude is the same as in the experiment but it could be noticed that the main trend is about twice smaller compared to the experimental data. An other factor which could have influence on the smaller velocity amplitude obtained during numerical analysis is the assumption of the uniform temperature along the liner. Temperature variations change material properties and therefore affect the amplitude of the vibrations. However, this influence is much smaller then the under-prediction of the acoustic fluctuations by the CFD code.
Vibrations of the liner induced by sinusoidal change of the pressure inside the combustion chamber show, except the forcing peak at frequency 300 Hz, also other velocity components at 320 Hz and 450 Hz (fig. 6). This is a clear indication of a non linear response. The peaks are matched well with the experimental data. The slight shift in the numerically predicted frequency is caused by one-way interaction. Due to lack of the displacement transfer to the CFD code, the wall as an additional acoustic source does not exist. Therefore the shift in the frequency with comparison to the experimental value is observed. The magnitude of the

power spectra obtained by numeric calculations and during experiment differs as a consequence of smaller velocity amplitude obtained during the computations. However, these results demonstrate that the acoustic resonance caused by mechanical forces from the fluid flow analysis is predicted correctly.

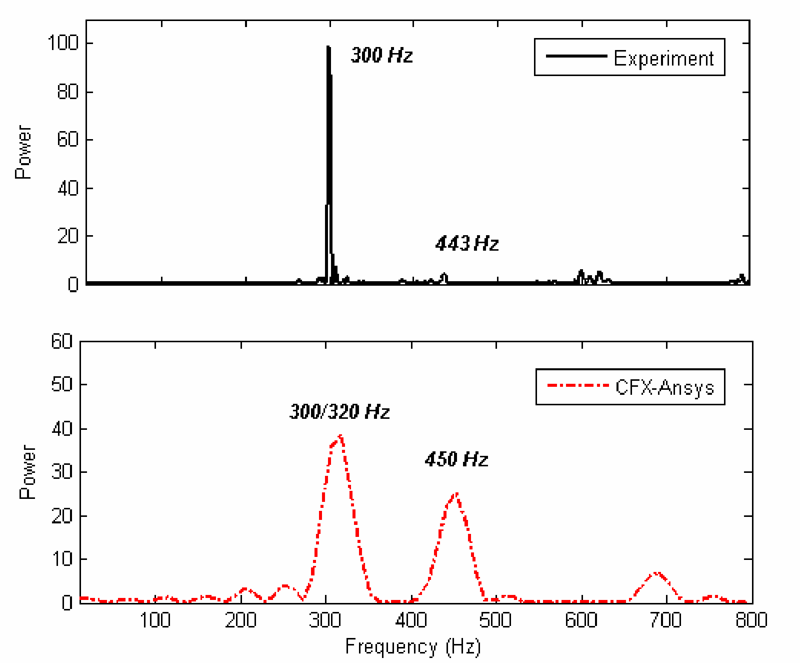

Fig. 6. FFT of the velocity: experiment (top), numeric (bottom)

Modal analysis with the use of fluid-acoustic and structural elements presents similar frequencies to the one obtained during one-way coupling (tab. 3). It could be observed that the frequency of the liner vibration is highly dependent on the mode shape. The most energetic modes are the synchronized ones, where a change in the total volume is significant (fig. 7, see encircled modes). They can induce high pressure changes inside the combustion chamber. Both synchronized modes are clearly visible on the FFT spectrum (fig. 6).

Table 3. Eigenfrequencies predicted during modal analysis

Eigenfrequency [-]	1	2/3	4	5/6	**7**	8	9	10/11	**12**
Value [Hz]	157	222	229	283	**336**	342	349	384	**448**

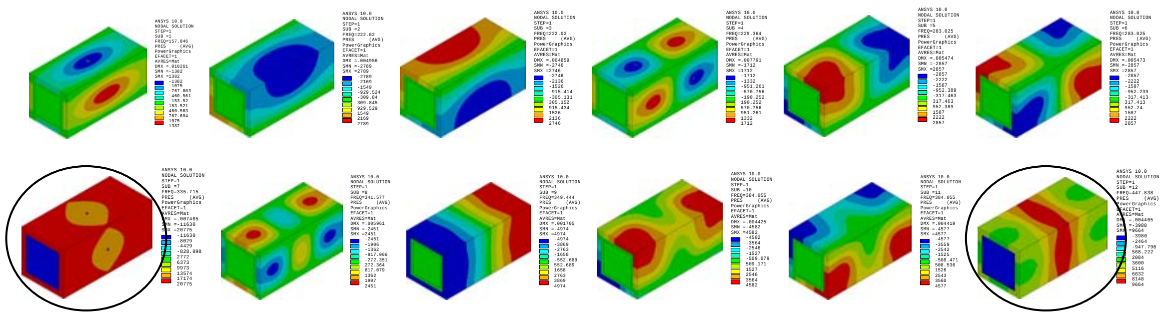

Fig.7. Eigenmodes of the fluid-structure system, synchronised modes are in the circle

5. CONCLUSIONS

Numerical investigation of the fluid-structure interaction between reacting flow inside the combustion chamber and liner wall vibration has been done. The results of the CFD analysis were successfully linked with the FEM code. Computations show under-predictions of the numerical results compared to the results obtained during experiment. It was caused by the smaller changes in the acoustic pressure amplitude predicted by the CFD code. The forcing frequency and self-excited mode observed during experiment are clearly visible on the numerical power spectra. Additional modal analysis without excitation but with the use of the acoustic-elastic model confirmed presence of the self-excited mode around 443 Hz. Further investigation of fluid-structure interaction in the combustion chamber is planned for a better understanding of the phenomena that occur.

ACKNOWLEDGMENT

This work is performed in the EU sponsored project FLUISTCOM – Fluid-Structure Interaction for Combustion System, in the Marie Curie Program, contract number MRTN-CT-2003-504183. The authors would like to thank CFX-ANSYS for the use of the code.

REFERENCES

[1] T. Tinga, J.F. van Kampen, B. de Jager, J.B.W. Kok, “Gas Turbine Combustor Liner Life Assessment Using a Combined Fluid/Structural Approach”, *Journal of Engineering for Gas Turbines and Power*, **129**, 69-79 (2007).

[2] T. Lieuwen, H. Torres, C. Johnson, B.T. Zinn, “A Mechanism of Combustion Instability in Lean Premixed Gas Trurbine Combustors”, *Journal of Engineering for Gas Turbines and Power,* **123**, 182-189 (2001).

[3] S. Ducruix, T. Shuller, D. Durox, S. Candel, “Combustion Dynamics and Instabilities: Elementary Coupling and Driving Mechanisms”, *Journal of Propulsion and Power,* **19**, No. 5, 722-733 (2003).

[4] F.J. Blom, “A monolithical fluid-structure interaction algorithm applied to the piston problem”, *Comput. Methods Appl. Mech. Engrg.* **167,** 369-391 (1998).

[5] C.A. Felippa, K.C. Park, C. Farhat, “Partitioned analysis of coupled mechanical systems”, *Comput. Methods Appl. Mech. Engrg.* **190,** 3247-3270 (2001).

[6] Ansys HTML Online Documentation, Ansys 10.0, Ansys Inc. 2005.

[7] J. van Kampen, *Acoustic pressure oscillations induced by confined turbulent premixed natural gas flames,* PhD thesis, University of Twente, Enschede, The Netherlands.

[8] R. Huls, *Acousto-Elastic Interaction in Combustion Chambers,* PhD thesis, University of Twente, Enschede, The Netherlands.

[9] Computational Nonlinear Mechanics in Aerospace Enginering, AIAA, Chapter 5 by K.Jansen, F. Shakib, T. Hughes, *Fast Projection Algorithm for Unstructured Meshes.*

[10] Z. Khatir, A.K. Pozarlik, R.K. Cooper, J.W. Watterson, J.B.W. Kok, “Numerical study of coupled fluid-structure interaction for combustion system”, *Proceedings of Conference on Numerical Methods for Fluid Dynamics,* 26-29 March 2007, Reading, UK.

[11] A.K. Pozarlik, J.B.W. Kok, “Numerical investigation of one- and two-way fluid-structure interaction in combustion systems”, *Proceedings of Int. Conf. on Computational Methods for Coupled Problems in Science and Engineering* (Coupled Problems), 21-23 May 2007, Ibiza, Spain.