

# NUMERICAL INVESTIGATIONS OF FLUID STRUCTURE INTERACTION BETWEEN UNSTEADY FLOW AND VIBRATING LINER IN A COMBUSTION CHAMBER

Z. KHATIR<sup>\*1</sup>, A.K. POZARLIK<sup>†</sup>, J.B.W. KOK<sup>†</sup>, R.K. COOPER<sup>\*</sup> and J.W.  
WATTERSON<sup>\*</sup>

<sup>\*</sup>School of Mechanical and Aerospace Engineering, Queen's University Belfast  
Ashby Building, Stranmillis Road  
Belfast BT9 5AH, Northern Ireland, U.K.  
e-mail: z.khatir@qub.ac.uk/zkhatir@hotmail.com

<sup>†</sup>Laboratory of Thermal Engineering, University of Twente  
P.O. Box 217, 7500 AE Enschede, The Netherlands  
e-mail: a.k.pozarlik@utwente.nl

**Key words:** Coupled Computational Fluid Dynamics-Computational Structural Dynamics, Combustion Application, Computational Investigation.

**Abstract.** Numerical investigations of fluid structure interaction between unsteady flow and vibrating liner in a combustion chamber are undertaken. The computational study consist of two approaches. Firstly, a partioned procedure consists of coupling the LES code AVBP for combustion modelling with the FEM code CaluliX for structural dynamic analysis. The CFD code CFX together with the FEM Ansys package are then used. Results of unsteady fluid structure interaction applied to combustion system are presented and compare well with experimental results.

## 1 INTRODUCTION

The ability to predict the stability of a given burner is the centre of many studies which can be experimental [1, 2] or numerical [3, 4, 5]. The present study is towards better understanding of transient combustion and its coupling with combustion wall vibration.

Motivated by the above concerns, numerical investigations on the coupling between CFD and CSD are undertaken. After specifying the experimental set-up, the next Section will be devoted to the computational procedures. Two numerical methods are presented. One consists of a partioned scheme for coupling the CFD research code AVBP [6] with the FEM code CaluliX [7]. The other approach uses the following commercial codes from ANSYS: CFX-10 for the CFD modelling and the Ansys FEM for CSD analysis [8]. Finally, Section 3 presents the results of numerical computations which contribute to validate the proposed investigations by comparison with experimental data.

---

<sup>1</sup>Correspondence: Rock Deformation Research, School of Earth Sciences, University of Leeds, Leeds LS2 9JT, U.K. Email: zkhatir@hotmail.com

## 2 EXPERIMENTAL AND COMPUTATIONAL SPECIFICATIONS

### 2.1 Experimental set-up

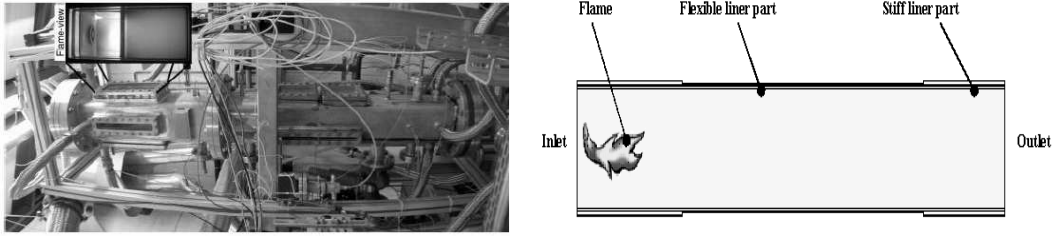


Figure 1: (Left) Experimental setup; (Right) Schematic of the combustion chamber geometry.

In order to measure vibration of the liner during the combustion process, the setup is equipped with the system of windows as shown in Figure 1. The part of the liner placed directly behind this transparent system is much thinner than the other liner elements. This flexible section responds more strongly to transient changes of the pressure inside the chamber during the transient combustion. The vibrations of the liner are measured by laser Doppler vibrometer. The characteristics of the flexible liner and the operating conditions during combustion experiment are depicted in Tables 1 and 2 respectively.

<i>Property</i>	<i>Value</i>	<i>Property</i>	<i>Value</i>
Length	$L$ 0.400 m	Young's modulus	$E$ 138 GPa
Width	$W$ 0.150 m	Poisson's ration	$\nu_s$ 0.3
Thickness	$t$ 0.0015 m	Density	$\rho_s$ 7844 kg/m <sup>3</sup>

Table 1: Dimensions and material properties of the flexible structural liner.

<i>Power</i>	<i>Absolute</i>	<i>Air factor</i>	<i>Total mass flow</i>	<i>Air preheating</i>
<i>[kW]</i>	<i>pressure [bar]</i>	<i>[-]</i>	<i>[g/s]</i>	<i>temperature [K]</i>
125	1.5	1.8	75.53	573

Table 2: Operating conditions during combustion experiment.

### 2.2 Computational models

#### 2.2.1 Coupled AVBP-CaluliX modelling

A LES fully compressible explicit code is used to solve the multi-species Navier-Stokes equations on hybrid grids. Subgrid stresses are described by the classical Smagorinsky model [6]. The flame/turbulence interaction is modelled by the Dynamic Thickened Flame (DTF) model which accounts for both mixing and combustion. Unsteady calculations are performed following operating conditions in Table 2 ([3]). The pressure loads from AVBP are used as inputs in the FEM code CaluliX. Care is taken to accurately transfer the pressure loads from CFD to CSD as a structured mesh is used for CSD using a volume spline interpolation [4, 5].

### 2.2.2 Coupled CFX-Ansys modelling

URANS is the numerical approach used for combustion flow. A quarter of the chamber is computed to save computational time. The reacting flow is solved using an Eddy Dissipation and Finite Rate Chemistry combustion models with  $k-\epsilon$  turbulence model as available in CFX. The mass fraction ratio of the fuel to air ratio in the inlet flow is pulsed with frequency 300 Hz with amplitude equal to 8.5%. Simultaneously, the transient structural deformations of the wall are computed as in Figure 2. As a result of the modular liner design no significant thermal stresses are generated. Calculations are performed with material properties adequate to temperature equal to 760°C. Exchange of information between CFX and Ansys is made possible by the MFX code.

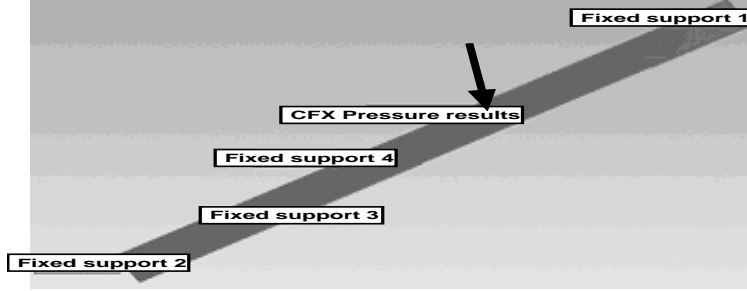


Figure 2: Boundary conditions used for solid model.

## 3 RESULTS

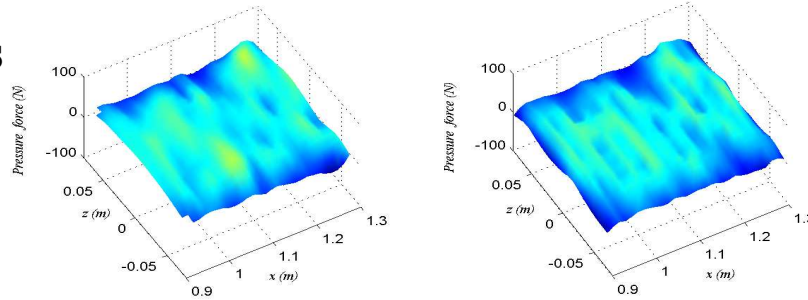


Figure 3: Isosurface of pressure forces at time  $t = 0.885$  s: (Left) CFD AVBP forces; (Right) CSD CaluliX forces after transfer using volume spline interpolation [4, 5].

The isosurface of AVBP and CalculiX forces on the flexible liner is plotted in Figure 3 at  $t = 0.885$  s. The distribution for both cases is very similar. The sum of the CFD and CSD forces on the flexible liner is identical, as are the 1<sup>st</sup> and 2<sup>nd</sup> moments. This confirms an accurate data transfer between CFD and CSD. Results predicted by numerical calculations show good agreement with experiment as seen in Figures 4– 5.

## 4 CONCLUSIONS

1. Computational investigations of coupled CFD-CSD applied to combustion system have been performed, and show good agreement with experiment.
2. Further investigations are still being undertaken to provide a better understanding of FSI in combustion systems.

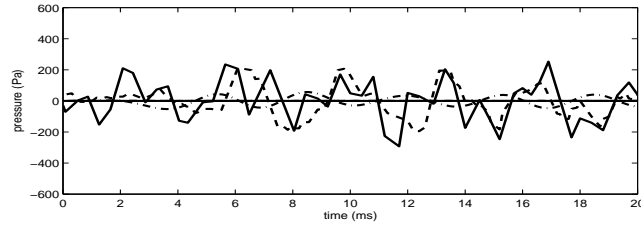


Figure 4: Pressure fluctuations on flexible liner: (—) Experiment; (- - -) AVBP and (-.-.-) CFX numerical simulations.

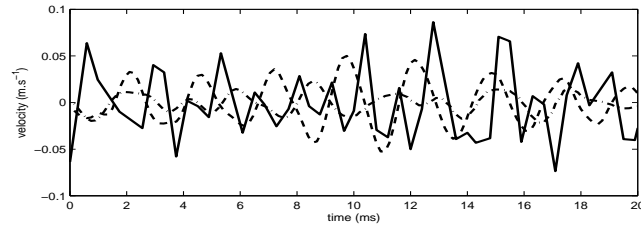


Figure 5: Vibrating liner's velocity: (—) Experiment; (- - -) Coupled AVBP-CalculiX and (-.-.-) Coupled CFX-Ansys computations.

## ACKNOWLEDGEMENTS

- EC Marie Curie RTN project FLUISTCOM, contract MRTN-CT-2003-504183.
- A.X. Sengissen, P. Mauro: CERFACS, Toulouse, France.
- R. Huls: University of Twente, Enschede, The Netherlands.

## REFERENCES

- [1] Huls R. Acousto-Elastic Interaction in Combustion Chambers, PhD thesis, University of Twente, Enschede, The Netherlands, ISBN: 90-365-2336-2, 2006.
- [2] Pozarlik A.K. and Kok J.B.W. The interaction of combustion pressure oscillation and liner vibrations, *13<sup>th</sup> International Congress on Sound and Vibration (ICSV13)*, Vienna, Austria, July 2-6, 2006.
- [3] Sengissen A.X. Simulation aux grandes échelles des instabilités de combustion: vers le couplage fluid/structure, PhD thesis, TH/CFD/06/30, INP Toulouse, France, 2006.
- [4] Khatir Z. Fluid structure interaction computation with application to combustion system, *European Marie Curie RTN FLUISTCOM Workshop on Fluid Structure Interaction - Numerical Aspects and Coupling Software with Applications*, Queen's University Belfast, Belfast, UK, January 25-26, 2007.
- [5] Khatir Z. Computation of fluid structure interaction with application to three-dimensional combustion System, *4<sup>th</sup> International Conference on Computational Fluid Dynamics (IC-CFD4)*, Ghent - Belgium, July 10-14, 2006.
- [6] The AVBP code: A parallel CFD code for reactive unsteady flow simulations on hybrid grids. <http://www.cerfacs.fr/cfd/avbp.html>
- [7] CALCULIX: A Free Software - Three-Dimensional Structural Finite Element Program. <http://www.calculix.de>
- [8] Ansys HTML Online Documentation, Ansys 10.0, Ansys Inc. 2005.