INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS
Int. J. Numer. Meth. Fluids 2008; **56**:1343–1349
Published online 17 December 2007 in Wiley InterScience (www.interscience.wiley.com). DOI: 10.1002/fld.1701

Numerical study of coupled fluid–structure interaction for combustion system

Z. Khatir[1,*,†], A. K. Pozarlik[2], R. K. Cooper[1], J. W. Watterson[1] and J. B. W. Kok[2]

[1]*School of Mechanical and Aerospace Engineering, Queen's University Belfast, Ashby Building, Stranmillis Road, Belfast BT9 5AG, Northern Ireland, U.K.*
[2]*Laboratory of Thermal Engineering, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands*

SUMMARY

The computation of fluid–structure interaction (FSI) problems requires solving simultaneously the coupled fluid and structure equations. A partitioned approach using a volume spline solution procedure is applied for the coupling of fluid dynamics and structural dynamics codes. For comparative study, two commercial packages for combustion and structural analysis are used. Results of numerical investigations of FSI between unsteady flow and vibrating liner in a combustion chamber are presented and show good agreement with experimental data. Copyright © 2007 John Wiley & Sons, Ltd.

Received 12 April 2007; Revised 26 October 2007; Accepted 30 October 2007

KEY WORDS: fluid–structure interaction; energy transfer; combustion chamber

1. INTRODUCTION

The numerical simulation of fluid–structure interaction (FSI) problems occurs in many engineering and scientific applications, ranging from airfoil oscillations or aero-hydrodynamics, blade flutter analysis in turbomachinery, to power generation in the design of gas turbine combustors [1, 2].

Unlike monolithical approaches where the fluid and structural equations are solved in a single computational domain, partitioned procedures [3] use independent techniques for the fluid and structure subdomains and exchange data along the fluid–structure interface.

In this article, the computation of FSI for a combustion system is carried out, with a focus in enforcing the interface boundary condition by transferring the fluid loads to the structure. Various methods have been developed for the problem of data transfer with non-matching meshes.

*Correspondence to: Z. Khatir, Rock Deformation Research, School of Earth Sciences, University of Leeds, Leeds LS2 9JT, U.K.
†E-mail: zkhatir@rdr.leeds.ac.uk

Contract/grant sponsor: European Marie Curie RTN; contract/grant number: MRTN-CT-2003-504183

A detailed review of data transfer methods is given in [4]. The volume spline interpolation method by Hounjet and Meijer [5] used for the data transfer in this paper is suitable for the transfer of both load and displacement.

2. FLUID–STRUCTURE COUPLING

2.1. Principle

The evaluation of the structure load induced by the fluid and the displacement of the fluid grid induced by the structural motion needs to satisfy the requirement of conservation of work. The principle of virtual work is usually used to ensure conservation of energy. For this purpose, a linear transformation is sought. If the displacement $\delta\mathbf{x}_\mathrm{f}$ of the fluid grid is expressed by means of the structural displacement $\delta\mathbf{x}_\mathrm{s}$ using the transformation matrix $[G]$ such that

$$\delta\mathbf{x}_\mathrm{f}=[G]\delta\mathbf{x}_\mathrm{s} \tag{1}$$

then the requirement for conservation leads to a corresponding matrix for the transformation of forces:

$$\mathbf{F}_\mathrm{s}^\mathrm{T}\delta\mathbf{x}_\mathrm{s}=\mathbf{F}_\mathrm{f}^\mathrm{T}\delta\mathbf{x}_\mathrm{f}=\mathbf{F}_\mathrm{f}^\mathrm{T}[G]\delta\mathbf{x}_\mathrm{s} \tag{2}$$

$$\mathbf{F}_\mathrm{s}=[G]^\mathrm{T}\mathbf{F}_\mathrm{f} \tag{3}$$

where $\mathbf{F}_\mathrm{f}$ and $\mathbf{F}_\mathrm{s}$ denote the fluid and structural loads, respectively.

2.2. Volume spline interpolation

Volume spline interpolation is a very simple method to interpolate non-uniformly spaced data. The method belongs to the category of scattered interpolation methods using radial basis functions $[\phi:\mathbb{R}^3\times\mathbb{R}^3\to\mathbb{R}]$ and is well suited for transferring both load and displacement [5].

Consider a set of N unordered support points $\mathbf{x}_i=(x_i,y_i,z_i)(i=1,\ldots,N)$ with values of $f(\mathbf{x})$ at these points equal to $f_i\in\mathbb{R}(i=1,\ldots,N)$, and the interpolation function defined as

$$s(\mathbf{x})=\sum_{i=1}^{N}\alpha_i\phi_i(\mathbf{x})+\mathscr{P}(\mathbf{x}) \tag{4}$$

with $\phi_i(\mathbf{x})=\|\,\mathbf{x}-\mathbf{x}_i\,\|$. Here, α_i are coefficients and $\mathscr{P}(\mathbf{x})$ is a global polynomial function of total degree at most $K-1$. The coefficients $\{\alpha\}$ and the polynomial functions $\mathscr{P}(\mathbf{x})$ are chosen to satisfy the N fitting conditions $s(\mathbf{x}_i)=f(\mathbf{x}_i)(i=1,\ldots,N)$ and the constraints $\sum_{i=1}^{N}\alpha_i Q(\mathbf{x}_i)=0$, for all polynomials Q of degree at most $K-1$.

Let $\{P_1,P_2,\ldots,P_k\}$ be a basis for the polynomials of total degree $K-1$. Assuming that the degree K of the polynomials to be 1, in which case the polynomial $\mathscr{P}(\mathbf{x})$ is just a constant α_{N+1}, then the corresponding systems of equations that need to be solved to find the coefficients $\alpha_i,(i=1,\ldots,N+1)$ are

$$\begin{pmatrix}\boldsymbol{\Phi} & P^\mathrm{T}\\ P & 0\end{pmatrix}\begin{pmatrix}\boldsymbol{\alpha}\\ \alpha_{N+1}\end{pmatrix}=\begin{pmatrix}\mathbf{f}\\ 0\end{pmatrix} \tag{5}$$

DOI: 10.1002/fld

where $\mathbf{\Phi}_{ij}=\phi(\|\mathbf{x}_i-\mathbf{x}_j\|), i, j=1,\ldots,N$; $P_{ij}=1, i=1, j=1,\ldots,N$ with $\boldsymbol{\alpha}=(\alpha_1,\ldots,\alpha_N)^{\mathrm{T}}$ and $\mathbf{f}$ is the vector of function values at the centres, $\mathbf{f}=(f(\mathbf{x}_1),\ldots,f(\mathbf{x}_N))^{\mathrm{T}}$. The matrix $\mathbf{\Phi}$ being full and ill conditioned for large N, care should be taken when solving the linear system. The use of preconditioning and iterative solution methods enables the study of large size problems [6].

2.3. CFX and Ansys coupling

Exchange of information between CFX and Ansys is possible with the use of the MFX code. A globally conservative interpolation method for data transfer, together with a bucket search algorithm for mapping a node to an element face as available in Ansys 10, is used. During interpolation, the total force balance over the surface is preserved [7]. For lossless data transfer between numerical codes, the exchange information is only possible between faces with the same global coordinates. The information about mechanical forces is transferred from the CFX analysis to the structural code every time step.

3. COMBUSTION APPLICATION

The ability to predict the stability of a given burner is the centre of many studies, which can be experimental [2] or numerical [8–11]. The turbulent flame in the lean combustion regime in a gas turbine combustor generates significant acoustic pressure oscillations which induce liner vibrations that may lead to fatigue damage of the combustion system. The correlation between pressure oscillations and liner vibrations is of a prime interest. The performances of two commercial computational fluid dynamics (CFD) codes and computational structures dynamics (CSD) codes are contrasted. The techniques mentioned in the previous section are applied here for the numerical investigations of FSI between unsteady flow and vibrating liner in a combustion chamber, and consist of coupling the CFD research code AVBP [12] with the finite element method (FEM) code CalculiX [13], and the commercial codes from ANSYS: CFX-10 for the CFD modelling and the Ansys FEM package for CSD analysis [7]. Results are presented and validated against experimental data. The vibration of the liner during the combustion process is measured at a section that is designed thin and flexible.

This flexible section responds strongly to transient changes of the pressure inside the chamber during the transient combustion. The vibrations of the liner are measured by a laser Doppler vibrometer. To avoid disturbing the free liner vibration pattern, the pressure sensor is located about 0.5 m above the point at which vibrations are recorded. In Figure 1, pressure and velocity variations at measurement points are depicted. The characteristics of the flexible liner and the operating conditions during the experiment are depicted in Table I.

3.1. Numerical results

Firstly, the large Eddy simulation (LES) fully compressible explicit code AVBP is used to solve the multi-species Navier–Stokes equations on hybrid grids [12]. Subgrid stresses are described by the classical Smagorinsky model. The flame/turbulence interaction is modelled by the dynamic thickened flame model, which accounts for both mixing and combustion. The numerical scheme uses third-order spatial and third-order explicit time accuracy. The boundary condition treatment is based on a multi-species extension of the NSCBC method, for which the acoustic impedance can

DOI: 10.1002/fld

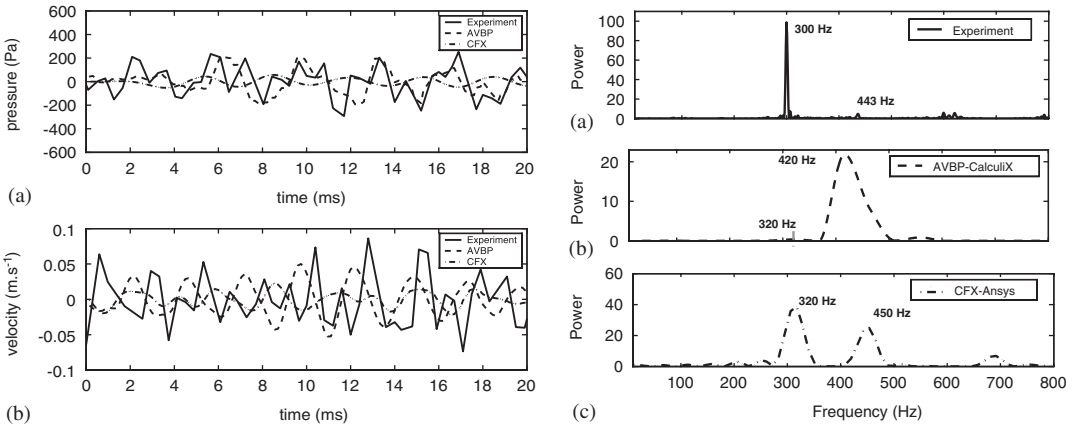

Figure 1. (Left) (a) Pressure variations on vibrating liner: (——) experiment; (- - - -) AVBP and (-.-.-.-.) CFX numerical simulations; (b) velocity fluctuations on vibrating liner: (——) experiment; (- - - -) coupled AVBP–CalculiX and (-.-.-.-.) coupled CFX–Ansys computations. (Right) Power spectra of liner's velocity: (a) (——) experiment; (b) (- - - -) Coupled AVBP–CalculiX and (c) (-.-.-.-.) coupled CFX–Ansys computations.

Table I. Data used for numerical calculations.

Length	Width	Thickness	Young's modulus	Poisson's ratio
Flexible liner				
0.400 (m)	0.150 (m)	0.0015 (m)	138 (GPa)	0.3
Power	Abs. pressure	Air factor	Mass flow	Air preheating temperature
Combustion flow				
125 (kW)	1.5 (bar)	1.8	75.53 (g/s)	573 (K)

be controlled. The walls are handled using a dynamic logarithmic law-of-the-wall formulation for velocity and temperature: the thermal treatment mimics the convective losses due to the cooling channel with a conjugate approach, imposing a heat resistance and prescribing the cooling air temperature. A full description can be found in [1, 8]. Runs are performed on very fine grids of the order of 2.7 millions tetrahedra (element sizes of the order of 1 mm), with time steps in the order of 1.5×10^{-7} s and a maximum Courant–Friedrichs–Lewy (CFL) number equal to 0.7. Unsteady calculations are performed following operating conditions as in Table I. The pressure loads from AVBP are used as inputs in the FEM code CalculiX and are accurately transferred using the volume spline interpolation [10, 11].

In a parallel line of investigation, the URANS numerical approach is used for combustion flow prediction. To save computation time and to increase the number density of elements within the calculated domain, the model was reduced to a quarter section of the real combustion chamber, with periodic boundary conditions. A total number of 632 000 unstructured elements, mostly placed in the flame and recirculation region, are used for calculations. The near-wall region is created

DOI: 10.1002/fld

with the use of prism elements to avoid generating highly distorted tetrahedral elements at the face. The thickness of the first prism element is equal to 0.2 mm. The maximal value of y^+ in a neighbourhood of the investigated wall is smaller than 3.2 with an average value of 0.9. The average CFL number during transient calculations was equal to 29. The CFX code is an implicit scheme thus is stable even during calculations with high CFL number. Previous investigation with the different mesh resolution but average CFL number equal to 1.7 shows minor influence on the obtained pressure field inside the combustion chamber. For reduction of the computational effort, calculations with higher CFL number are performed. The reaction flow is solved with the use of the Eddy dissipation/finite rate chemistry combustion model [7]. This way, the reaction rate at each step can be limited by turbulent mixing or chemical kinetics. For turbulence, the k-epsilon model as available in CFX is used. The velocity and turbulence profiles at the inlet are taken from steady-state calculations of the full setup geometry. The static average pressure is imposed at the combustion chamber outlet. Adiabatic, no-slip walls are assumed and the influence of the pressure from the cooling passage is neglected. The investigated flame is a premixed natural gas flame under conditions presented in Table I. The mass fraction ratio of the fuel to air ratio in the inlet flow is pulsated with frequency 300 Hz and amplitude equal to 8.5%. Simultaneously, the transient structural deformations of the wall are computed. As a result of the modular liner design no significant thermal stresses are generated. Calculations are performed at the typical liner temperature during operating conditions, which is equal to 760°C (Table I). Damping of the vibrating wall is not included in the physical model in the calculation. The time step and the total calculation time are fixed to 0.15 and 100 ms, respectively, in URANS, whereas the LES signal is sampled at 0.15 ms intervals during a simulated time of 21 ms (i.e. 140 samples). Results shown in Figure 1 indicate that the numerical and experimental results compare well. Numerical pressure and velocity fluctuations on the thin vibrating liner are well predicted. The coupled AVBP–CalculiX gives a good approximation of the wall velocity and pressure amplitudes, whereas they are under-predicted with CFX–Ansys. This is due to the smaller fluctuations in the acoustic pressure inside the combustion chamber predicted by CFX in comparison with the experimental results. In the URANS approach, the high grid resolution and the large time step used for the analysis make it difficult to observe fluctuations in the acoustic pressure inside the combustion chamber. A smaller time step (CFL much smaller than 1) would enable to capture this phenomenon at the cost of increasing the (already long) calculation time. Thus, changes in the liner velocity are mostly caused by changes in the mean flow resulting from the equivalence ratio fluctuations. The forcing frequency (300 Hz) and self-excited mode (443 Hz) are clearly noticeable on spectra, with both numerical values matching the experimental values. These results demonstrate that the numerics capture the impact of forcing on the acoustic resonance of the test rig with a reasonable accuracy. Finally, Figure 2(a) reports the time histories of the works—W_f and W_s performed by the fluid and structure subsystems on the fluid–structure interface during the AVBP–CalculiX computations. These works are evaluated as $W_\mathrm{f} = \sum_n W_\mathrm{f}^{n+1}$ and $W_\mathrm{s} = \sum_n W_\mathrm{s}^{n+1}$, where W_f^{n+1} and W_s^{n+1} are the works performed by the fluid and structure subsystems at a point located on the interface during the time interval $[t^n, t^{n+1}]$ (i.e. corresponding to the evaluation point in the experiment). The solution procedure exhibits little if any violation of the conservation of energy transfer at the interface (in Figure 2(a), the curves for W_s and W_f are undistinguishable). Figure 2(b) displays the time history of this minor relative violation of energy transfer, $|W_\mathrm{s} - W_\mathrm{f}|/|W_\mathrm{s}|$, for the algorithm. It shows that initially, the maximum recorded energy-transfer violation is about 0.47% of the work performed by the structure subsystem on the interface. Afterwards, the relative energy-transfer violation oscillates between 0.01 and 0.7% with a mean of 0.3%, which demonstrates again the

DOI: 10.1002/fld

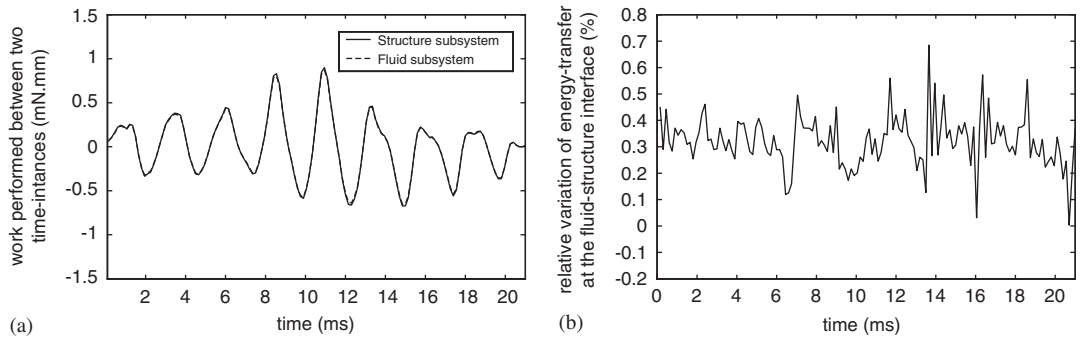

Figure 2. (a) Computed time histories of the works performed on the interface by the structure and fluid subsystems and (b) time history of the instantaneous relative violation of energy transfer on the interface with coupled AVBP–CalculiX computations.

good energy-transfer behaviour of the solution procedure. These results show that the error of the volume spline procedure is of order $O(h^2)$ and is consistent with previous studies [14].

4. CONCLUSION

Numerical investigations of FSI applied to a combustion system have been performed. A partitioned approach using a volume spline solution procedure for the data transfer between LES and FEM codes has shown to accurately conserve energy. Computations using a URANS and FEM packages have revealed small under-prediction of the results. In summary, the results agree well with experimental data for the prediction of velocity and pressure wall fluctuation as well as for the self-excited mode in the combustion chamber. Further investigations are still being undertaken to provide a better understanding of FSI in combustion systems.

ACKNOWLEDGEMENTS

This work is carried out within the European Marie Curie RTN project FLUISTCOM through contract number MRTN-CT-2003-504183. CFX-Ansys is acknowledged for the use of their code. We thank the referees for their comments.

REFERENCES

1. Sengissen AX, Poinsot TJ, Van Kampen JF, Kok JBW. Response of swirled non-premixed burner to fuel flow rate modulation. *Complex Effects in Large Eddy Simulations*, Limassol, Cyprus, September 2005.
2. Huls R. Acousto–elastic interaction in combustion chambers. *Ph.D. Thesis*, University of Twente, The Netherlands, ISBN: 90-365-2336-2, 2006.
3. Felippa CA, Park KC, Farhat C. Partioned analysis of coupled mechanical systems. *Computational Methods in Applied Mechanics and Engineering* 2001; **190**:3247–3270.
4. Forsythe N. A partitioned approach to fluid–structure interaction for artificial heart valves. *Ph.D. Thesis*, Queen's University Belfast, Northern Ireland, U.K., 2006.
5. Hounjet MHL, Meijer JJ. Evaluation of elastomechanical and aerodynamic data transfer methods for non-planar configurations in computational aeroelastic analysis. *NLR-TP-95690-U*, National Aerospace Laboratory NLR, Amsterdam, The Netherlands, 1995.

DOI: 10.1002/fld

6. Barba LA, Leonard A, Allen CB. Advances in viscous vortex methods—meshless spatial adaption based on radial basis function interpolation. *International Journal for Numerical Methods in Fluids* 2005; **47**:387–421.
7. Ansys CFX/FEM HTML Online Documentation, Ansys 10.0. Ansys Inc., 2005.
8. Sengissen AX. Simulation aux grandes échelles des instabilités de combustion: vers le couplage fluid/structure, *Ph.D. Thesis*, TH/CFD/06/30, INP Toulouse, France, 2006.
9. Pozarlik AK, Kok JBW. The interaction of combustion pressure oscillation and liner vibrations. *Thirteenth International Congress on Sound and Vibration* (*ICSV13*), Vienna, Austria, 2–6 July 2006.
10. Khatir Z. Computation of fluid structure interaction with application to three-dimensional combustion system. *Fourth International Conference on Computational Fluid Dynamics* (*ICCFD4*), Ghent, Belgium, 10–14 July 2006.
11. Khatir Z. Fluid structure interaction computation with application to combustion system. *EC Marie Curie RTN Fluistcom Workshop on Fluid Structure Interaction*, Queen's University Belfast, U.K., 25–26 January 2007.
12. The AVBP code. http://www.cerfacs.fr/cfd/avbp_code.php.
13. CALCULIX: a free software 3D FEM program. http://www.calculix.de.
14. Cordero-Gracia M, Ribollés P, Valero E, Gómez M. A volume spline interpolation tool for elastomechanical and aerodynamic data transfer problems. *Proceedings of the IASTED International Conference on Applied Simulation and Modelling*, Rhodes, Greece, June, 2004; 265–269.

DOI: 10.1002/fld