2004-01-1995

Towards Large Eddy Simulation in Internal-Combustion Engines: simulation of a compressed tumble flow.

Vincent Moureau*, Iain Barton**
*IFP, Rueil-Malmaison (France)

Christian Angelberger*, Thierry Poinsot**
**CERFACS, Toulouse (France)

ABSTRACT

The development of the Large Eddy Simulation (LES) 3D CFD code AVBP to yield a CFD tool able to predict cyclic variability in Internal Combustion (IC) engines is reported. In a first step the implementation of an Arbitrary Lagrangian Eulerian (ALE) method into AVBP is described, allowing to move solid boundaries. Then the principles and implementation of the Conditioned Temporal Interpolation (CTI) mesh management technique is described, and some specific adaptations for LES simulations are discussed. Finally a first validation of the so obtained LES IC engine code is presented by comparing predictions with findings on the square piston experiment.

INTRODUCTION

Among the technologies that will potentially be at the basis of future highly efficient, near-zero emission, cost effective powertrains, combustion processes like Controlled Auto-Ignition (CAI) or Homogeneous Charge Compression Ignition (HCCI) play a key role. They have the potential to reduce pollutant production at the source, allowing to comply with future more and more stringent emission regulations, without the need for complex and expensive after-treatment systems. One of the factors limiting at present the full exploitation of the potential of some of these new combustion techniques is the occurrence of important cycle to cycle variations in parts of the engine operating range. Cyclic variability in engines are not yet fully understood, as they are the result of an important number of phenomena hardly accessible to experimental investigations. Giving the automotive engineers the means to predict such cyclic variability, and more generally unsteady engine phenomena like warm-up or engine transients, can be viewed as a major challenge for a more effective design of the combustion engines of the future. Due to the difficulty of experimental investigations in this field, Computational Fluid Dynamics (CFD) simulations appear as the best candidate for providing such a design tool.

Today's standard in engine simulation are Reynolds Averaged Navier-Stokes (RANS) methods. They allow to accurately predict the mean characteristics of stabilised engine operation, and are therefore widely used to choose the most promising engine configuration, before going to actually test it. They thus contribute to reduce the cost of development, allowing to integrate an important number of physical knowledge into engine design. Yet these techniques are inherently not adapted to predict unsteady phenomena. A better candidate for a tool allowing to predict them are Large Eddy Simulation (LES) techniques. These techniques have proven their great potential when applied to the prediction of acoustic instability in gas turbine combustion chambers [9], and they start to be developed for piston engine applications as discussed by Celik *et al* [4].

LES and RANS techniques differ in the way they address the present impossibility to resolve all the scales present in engine flows, and especially those related to turbulence, combustion and liquid jets. RANS simulations are based on a statistical averaging to solve only the mean flow. This implies that modelling concerns the whole spectrum of scales, which in turn makes the predictivity of RANS simulations dependant on the quality of the models used. The statistical averaging also extremely complicates addressing unsteady phenomena. In LES, a spatial or temporal filtering is used to represent the large turbulent scales of the flow, which are directly resolved, while the small scales are modelled. In LES, modelling thus concerns a much smaller part of the spectrum, which leads to an improvement of predictivity as compared to RANS. LES inherently allows to address large scale unsteady phenomena, and thus has a good potential to predict engine unsteadiness.

In LES and RANS, the effect of the modelled part of the turbulent spectrum on the resolved part is assumed to be diffusive, and it is often taken into account by introducing a turbulent viscosity. The level of turbulent viscosity directly depends on the amount of modelled energy leading to high levels for RANS and far less important levels for LES. This explains the different requirements of LES and RANS in terms of numerical schemes. In RANS, the main requirement is to be robust and stable on distorted meshes. Numerical dissipation is only a second-order aspect. Most of RANS codes like KIVA [2] use upwind schemes known to be very stable, but dissipative. Inversely, in LES, vortices above and at the resolution limit must be accurately resolved, with as few numerical dissipation as possible. This implies the use of precise and energy-conserving numerical schemes, as Finite-Volume Centred-Differencing (FVCD) schemes.

LES and RANS not only differ by the requests imposed on numerics, they also imply a different simulation procedure and exploitation of results. One RANS cycle gives directly access to mean values. Realising the same objective with LES would lead to a CPU time that is way beyond reach of present supercomputers, as a potentially high number of consecutive engine cycles would have to be computed to reach a statistical convergence of the results. On the short to medium term, the objective of LES will not be to compute mean engine characteristics, a domain RANS can satisfactorily cover at reasonable CPU costs, but rather to be used in complement to RANS, addressing unsteady phenomena inaccessible to it. Experience gained with LES so far indicates that even a small number of engine cycles could be sufficient to gain insight into the occurrence of cyclic variability, which will help to suppress or limit them by design. This will necessitate however the development of engine simulation codes able to address all related phenomena like turbulence, acoustic waves, combustion and knock, as well as liquid sprays.

The objectives of the present paper is to present first developments performed to update the LES solver AVBP to be able to perform LES engine simulations, with the long term objective of realising a CFD tool able to predict cyclic variability in IC engines. Work has focused so far on conserving the numerical properties of the code when the mesh is deforming and on the mesh management using a Conditioned Temporal Interpolation (CTI) technique coupled to a high-order interpolator. The code validation is performed by comparing simulation results to a complete database of Particle Image Velocimetry (PIV) fields obtained with a square-piston experiment which generates and compresses a strong tumble motion. The influence on turbulence of assuming a 2D flow instead of simulating the 3D reality is studied, as well as the effects of the numerical schemes' accuracy.

THE AVBP CODE

The numerical developments presented in this article have all been implemented in the AVBP code which is jointly developed by CERFACS and IFP. AVBP is a parallel LES and DNS solver of the reactive multi-component compressible Navier-Stokes equations on 2D and 3D unstructured hybrid grids. The closure of these equations is done introducing a perfect-gas state equation for the mixture and a temperature dependent enthalpy tabulation for each component. A calorific capacity ratio $\overline{r}$ for the mixture is defined locally depending on the thermodynamic conditions. This aspect is very important in piston engine simulations because the theoretical thermodynamic efficiency is a function of the mean of the gas constant $\overline{r}$.

AVBP NUMERICS

The design of AVBP has been focused on guaranteeing a linear parallel efficiency : multiplying the number of processors by two divides the CPU time by the same ratio. These performances have been obtained using a mesh partitioning algorithm which balances the CPU load on each processor and suitable message-passing libraries. To minimise interface exchanges between processors, AVBP numerical schemes are based on the cell-vertex method [13] which naturally ensures a high compactness. The main convective schemes are a finite-volume Lax-Wendroff type scheme (LW) and a finite-element two-step Taylor-Galerkin scheme (TTGC) [5]. These two schemes are respectively 2^{nd} and 3^{rd} order in time and space. The diffusive scheme is a typical 2^{nd} order compact scheme. Element types handled by AVBP are triangles and quadrangles in 2D and tetrahedrons, prisms, pyramids and hexahedrons in 3D. The time integration is fully explicit to maximise accuracy.

TURBULENCE MODELS

The turbulent LES models of AVBP are a classical Smagorinsky, a filtered Smagorinsky and a Wall Adapting Linear Eddy (WALE) model. The Smagorinsky model [16] has been heavily tested and used in LES simulations. It is known to correctly predict the global turbulent quantities but also to over-dissipate when used for transient flows [14]. The filtered Smagorinsky model is similar to the Smagorinsky model but the velocity field is explicitly filtered with a higher-filter width before the evaluation of the turbulent viscosity. It has been developed to better represent the local phenomena of the flow [3]. Finally, the WALE model [3], also based on the Smagorinsky model, is used for wall-bounded flows attempting to recover the scaling laws of the wall.

IMPLEMENTATION OF AN ALE METHOD FOR MOVING WALLS

Because AVBP is a code primary written for turbo-machinery, a first step was to implement a numerical method to handle moving boundaries. This kind of methods adapted to deforming domains are of interest in many practical applications : fluid-structure interactions, free-surface flow, reciprocating engines, … In each case, the properties of the boundary movement - velocity, deformation rate, frequency - determine the more adapted. For IC engine configurations, moving boundaries are the piston and the valves that have a translation movement along a fixed axis. Body-force or Arbitrary-Lagrangian Eulerian (ALE) methods are commonly used for this type of periodic displacements.

The body-force method [19] consists in imposing a given speed on an arbitrary surface which does not necessarily coincide with mesh vertices. Complex deforming configurations can hereby easily be simulated using fixed Cartesian grids. The important drawback is that advanced boundary-modelling as imposing characteristic boundary conditions or wall laws [17] is difficult and complicated.

The ALE [1] method is an appealing alternative in which each mesh vertex *i* has a given moving speed $\dot{\mathbf{X}}_i(t)$ and the computational domain boundaries $\partial\Omega(t)$ coincide with mesh vertices. This technique is particularly adapted to deforming unstructured meshes. The vertex speed $\dot{\mathbf{X}}_i(t)$ is an input of the simulation and must therefore be calculated independently with a CTI method for example as described in the next Section.

The ALE formalism has been used in AVBP to redevelop pre-existing numerical finite-volume and finite-element convective schemes. The new schemes are very similar to the original ones and the mesh movement is simply taken into account introducing additional correcting terms due to translation and dilatation. These terms which have been rigorously determined [12], ensure that numerical properties - convergence order and precision - are the same on fixed and moving grids. Special care has been paid to the geometric conservation properties of the schemes in order to avoid the generation of spurious perturbations when deforming the grid [8]. Several simple test cases have validated the numerical method [12] and illustrated the enumerated properties.

THE CTI MESH MANAGEMENT

Given the complexity of IC engine geometry, characterised in particular by piston and valves moving at different speeds along different axes, it is essential to employ a robust mesh management technique. Its principal goal is to provide a means of moving all the mesh nodes in time in a way that ensures an accurate evolution of the moving boundaries while avoiding the occurrence of negative cell volumes.

Previous studies using an implicit solver, KIFP, have investigated different approaches to moving the mesh with and without flow calculations [20,21]. From this work the main conclusion is that standard methods of mesh movement techniques, such as the spring analogy [15], are not easily applicable to IC engine combustion chambers, where the movements of the valve and piston boundaries significantly deform the mesh. As a consequence of this, the Conditioned Temporal Interpolation (CTI) mesh management technique was developed [18]. In this section we recall the basic principles of this technique and describe its implementation into AVBP, as well as the necessary modifications for LES.

BASIC PRINCIPLES

The starting point of the CTI mesh management technique is to decompose the IC engine cycle in a number of mesh phases. Each phase is characterised by a specific and constant mesh connectivity. A start mesh is created (using standard CAD packages) at the start angle of each mesh phase. The boundaries of this start mesh are then stretched within the CAD software to yield the node positions at the crank angle corresponding to the end of the mesh phase. The final mesh obtained is called the target mesh for that respective mesh phase, and so the only difference with the start mesh and the target mesh are the node positions. The start and target meshes do not comprise negative volumes, the difficulty that faces the mesh management is to move the boundaries appropriately during the CFD simulation and to ensure that mesh quality remains acceptable.

For each mesh phase - a vector field consisting of the co-ordinate differences between the positions of the nodes in the start and target grid is stored. An absolute percentage difference field is defined as the percentage of the displacement along the path leading from the start to the target mesh. If all the nodes have a zero absolute percentage value, this would correspond to the start grid. If all nodes have a 100% absolute percentage value, this would correspond to the target grid.

The difficulty of calculating how to move all the nodes along their predefined paths is directly related to the fact that the piston and valves are moving at different speeds, which implies that the absolute percentage field changes at difference rates on different physical boundaries. To overcome this, a smoothed percentage field has to be calculated within the computational domain. Note that the adopted method for smoothing cannot ensure positive cell volumes, and some level of trial-and-error is necessary to achieve an acceptable result. This is why mesh-only simulations are very useful for initial computations.

IMPLEMENTATION IN AVBP

AVBP is a parallel solver based on mesh decomposition [10], the mesh decomposition is an important factor that required some adaptation using the CTI technique. However, the space discretisation in AVBP is node centered [13], which is advantageous for grid movement, since the mesh management requires moving the grid nodes directly. The velocity of mesh nodes is therefore easily computed and used in numerical convective schemes [12].

The actual percentage calculations used in this paper follow the work by Torres & Zolver in [18]. A relative percentage field is calculated at each time step, representing the displacement that grid points need to move from their current position to their target position. Torres & Zolver found that such a methodology using a weighted curvature term based on a zeroth-order Laplacian solver was the most generic in application. It was found to work well for multi-valve and single-valve configurations.

In order to test the implementation of the CTI methodology into AVBP, a purely mesh-only simulation of the combustion chamber of a single valve PSA XU10 engine has been performed. The mesh used in these simulations was simply a coarse RANS mesh. This allowed us to perform a fast testing of the quality of the resulting mesh management, no attempt being done to perform LES simulations at this stage. Figure 1 shows a cut through this mesh at 5°CA (degrees crank angle), 26°CA and 48°CA, obtained with a mesh-only simulation using the CTI method implemented in AVBP.

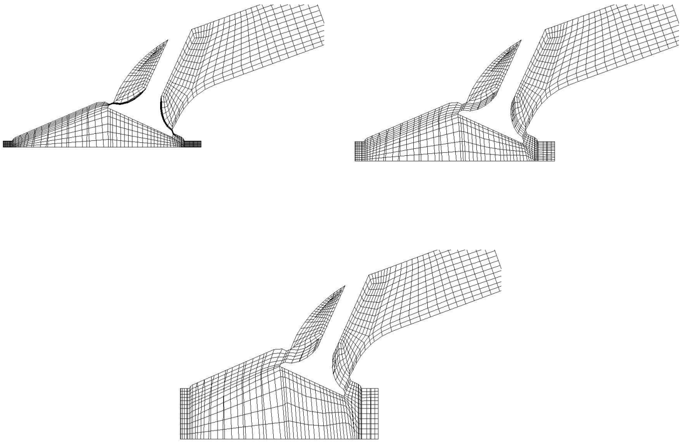

Figure 1: The grid systems at 5°CA, 26°CA and 48°CA, respectively.

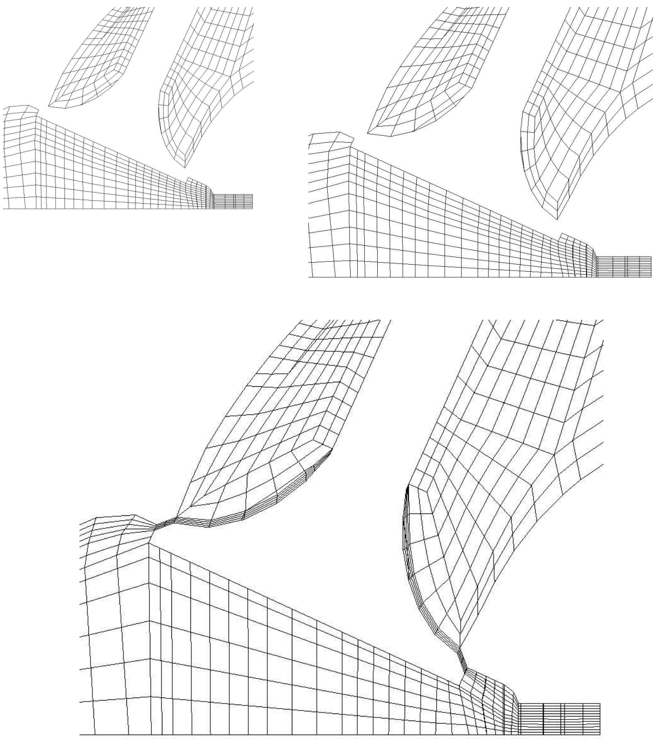

Figure 2: The grid systems at CAD of 4, 5 before the valve is opened and at 5 just after it is opened, respectively.

The simulation shown in Figure 1 and Figure 2 is part of a full engine cycle starting at 4°CA, where the valve is treated as being closed. At 5°CA the valve opens, the evolution of the grid from its closed state to its open state is shown in Figure 2 focussing on the grid in the valve region. This evolution demonstrates clearly a change in mesh phase. The entire simulation runs from 4°CA to 360°CA. It consists of 5 mesh phases. The first and last two phases have a closed valve system. The second mesh phase represents the opening of the valve and the expansion of the piston, while the third phase is essentially the closing of the valve. The last two mesh phases essentially represent the compression.

In order to cope with the partitioning of the grid, the solution of the percentage field required some care. The optimisation of the percentage field need to be done involving all the mesh partitions. While this is somewhat expensive, the increase in CPU cost was only of the order of 10% compared with simulations that did not call the CTI routine. A convergence criterion of 0.1% was set for the maximum change of percentage field calculations. For the configuration shown in Figure 1, parallelisation tests on a SGI Origin 2000 machine with a total of eight processors show a near linear increase in CPU efficiency, as shown in Figure 3. The drop from the ideal speed-up line for more than 5 processors is caused by an increase in the number of nodes on partition boundaries that require updating during the simulation.

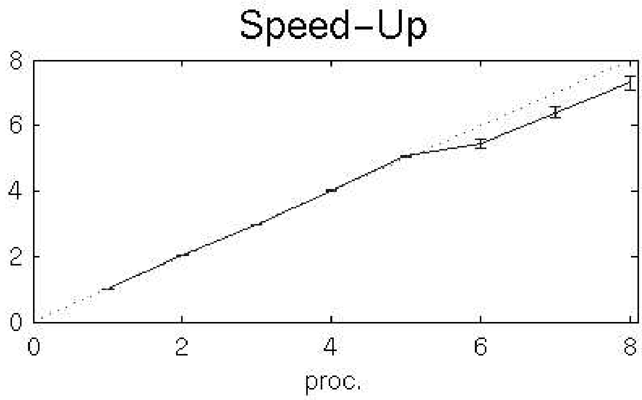

Figure 3: The parallel performance of the code running the CTI technique, showing Speed-Up vs. number of processors.

The disadvantage of the CTI approach is that a new target mesh is required whenever a moving boundary changes direction (as a new displacement field is required). Also, the paths that the grid nodes can move along are prescribed and do not change according to local deformation in the grid structure. This problem is obviously overcome, by changing the mesh topology between mesh phases, which in turn means that a "remapping" of the flow field variables has to be performed from the target grid of the preceding phase to the start grid of the current mesh phase. "Remapping" means that results are interpolated between two meshes with different grid indexing and connectivity. In the basic CTI version written for RANS applications, the remapping process is achieved by solving a least square fit interpolation of a second-order polynomial on a tight local stencil. Typically, the remapping process is required when the grid becomes too coarse or too distorted: Figure 4 shows a cut trough the XU10 grid at 55°CA, just before and just after the remapping process. As can be seen from Figure 4, the grid system in the main chamber is coarse which is caused by the previous expansion of the grid, remapping to the finer grid allows the simulation to continue without any further deterioration in the results caused by the coarseness of the grid. In fact this approach can have some benefits in particular for LES simulations and explicit code solvers. Namely, the remapping allows the user to implement the most appropriate grid system during a particular evolution of the simulation, which evidently is critical when the computational domain itself changes form.

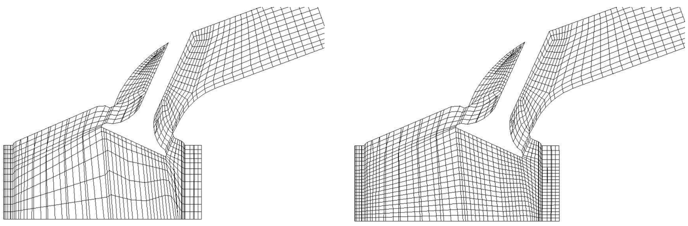

Figure 4: The grid systems at 55°CA, before remapping (left), after remapping (right).

INTERPOLATION BETWEEN MESH PHASES: ADAPTATIONS FOR LES

As explained in the previous Section and illustrated on Figure 4, the "remapping" process between two phases is of primary importance for explicit codes to keep a homogenous cell width. The interpolation must be as precise as possible not to modify the variable fields. RANS fields represent statistically mean variables, they do not contain any high-frequency structures. The interpolation scheme can therefore be limited to a simple gradient or a second-order polynomial reconstruction. However LES fields, which are a filtered realisation of the turbulent flow, contain small vortices with a frequency close to the cut-off. In this case, higher-order and larger-stencil interpolation schemes must be used.

Figure 5 illustrates the accuracy differences of three interpolators for which the stencil and the order are different. The considered simulation is a LES of a Homogenous Isotropic Turbulence (HIT). Interpolation is used to switch from a 33^3 regular mesh to a 49^3 regular mesh. Turbulent spectrums are given before and after the "remapping". These results underline that a large stencil width and a high discretisation order of the interpolation are necessary for ensuring a level of spectral quality compatible with LES simulations. Otherwise, the results obtained with the 1st order tight stencil indicate that energy is lost even at well resolved levels during interpolation. These conclusions were confirmed using different mesh refinements and coarsenings. Note that a problem that still needs to be resolved is the exchange of kinetic energy between resolved and sub-grid scale, which will be the object of future research.

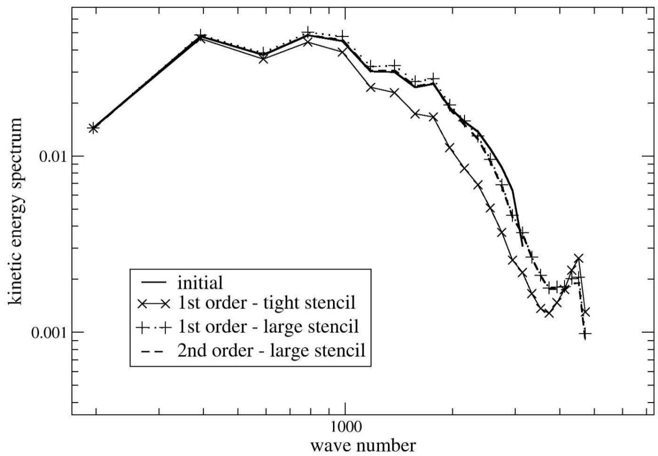

Figure 5: Spectrum comparison of interpolated LES fields from 33^3 (initial) to 49^3 meshes for a Homogenous Isotropic Turbulence flow.

FIRST APPLICATION: THE SQUARE PISTON

EXPERIMENTAL SET-UP

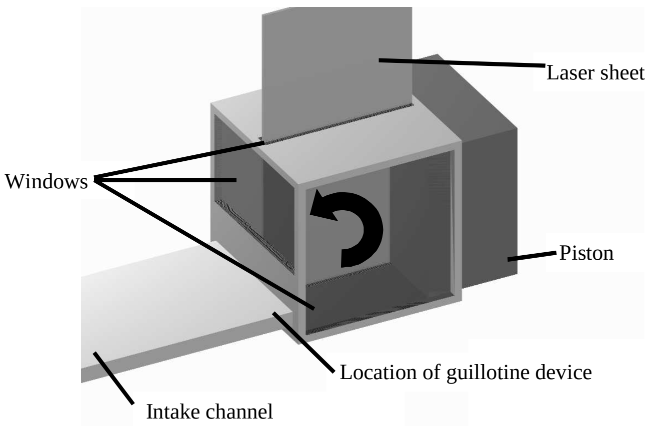

Figure 6: sketch of the square piston engine

The square piston experiment has been designed at IMFT to study the compression and the disruption of a tumble vortex [3,11]. The main goal was to better understand the flow structures and phenomena in Spark-Ignition (SI) engines. It was specifically designed with a view to validate LES simulations. The experimental set-up is shown in Figure 6: it is composed of a square piston which has a sinusoidal motion, a guillotine to close the flat intake channel, a plenum chamber at ambient pressure and multiple optical accesses for Particle Image Velocimetry (PIV) measurements. The intake channel comes out in the lower part of the chamber to generate a strong tumble motion during the intake stroke. The experiment is run with a four-stroke cycle : intake, compression, expansion and exhaust. The available data have been obtained with a volumetric compression ratio of 4 and a crankshaft speed of 206 rpm. The piston dimensions are 100x100 mm², the stroke is 75 mm. and the intake channel is 300 mm long.

More than one hundred PIV fields have been taken in the vertical symmetry plane at different crank angles. From these one can compute mean and fluctuation fields for the two speed components of the plane.

NUMERICAL SET-UP

Both 2D and 3D LES simulations have been carried out during several cycles to compute mean and fluctuating velocity profiles. A 3D mesh of 270000 vertices with prisms, tetrahedra and hexahedra elements is given on Figure 7 as an example. For 2D and 3D meshes, guillotine opening and closing are simulated and Figure 8 shows the cell deformations during these phases.

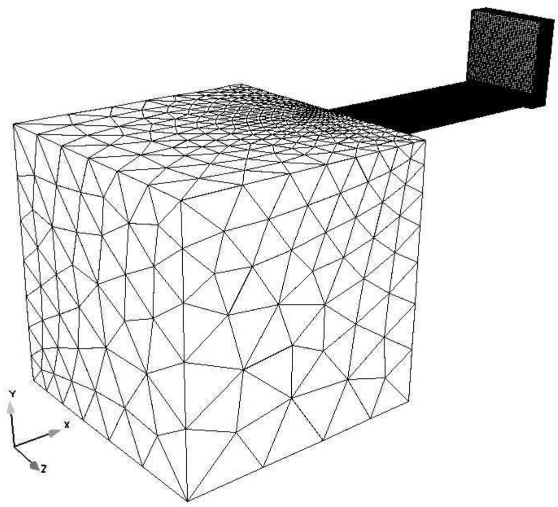

Figure 7: global view of a 3D hybrid mesh

The simulations are performed using the Smagorinsky turbulent model and a simple "law of the wall" model based on the linear/log laws. The mesh management was realised with the CTI method and the large stencil, 2nd order interpolation. The engine cycle was decomposed into 25 mesh phases. Numerical schemes described in Section "The AVBP Code" are compared to underline the effects of the convergence order.

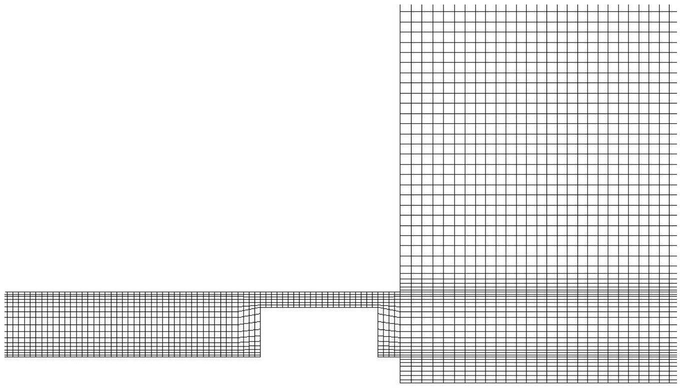

Figure 8: detail of the 2D hybrid mesh when closing the guillotine.

RESULTS

Instantaneous fields

Instantaneous fields are interesting to have a qualitative evaluation of the different simulations and to see if the results are in agreement with experimental findings. Although 2D LES simulations are not physically sound due to the three-dimensional nature of turbulence, they were used for being able to compare numerical schemes at low CPU costs. We also present their results to illustrate that performing LES in 2D meshes, which can seem tempting in terms of CPU time, will lead to erroneous results, a fact that is less evident for RANS simulations.

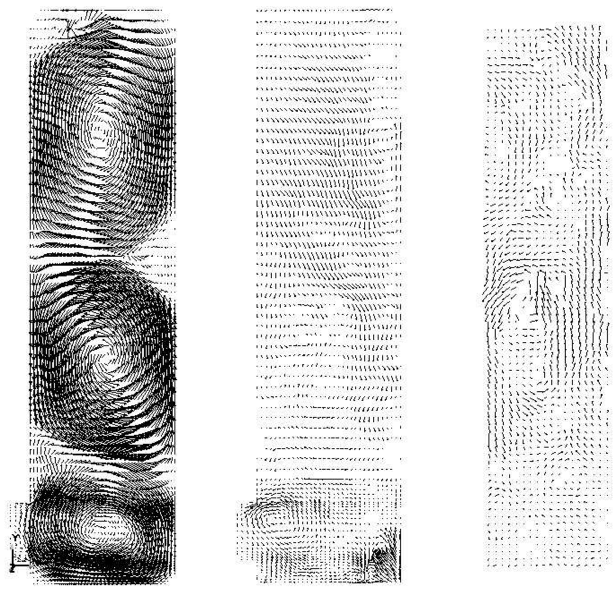

Figure 9: 2D (left), 3D (centre) and experimental (right) instantaneous velocity fields with same normalisation at Top-Dead Centre (TDC). Computational fields have been obtained with the Lax-Wendroff scheme.

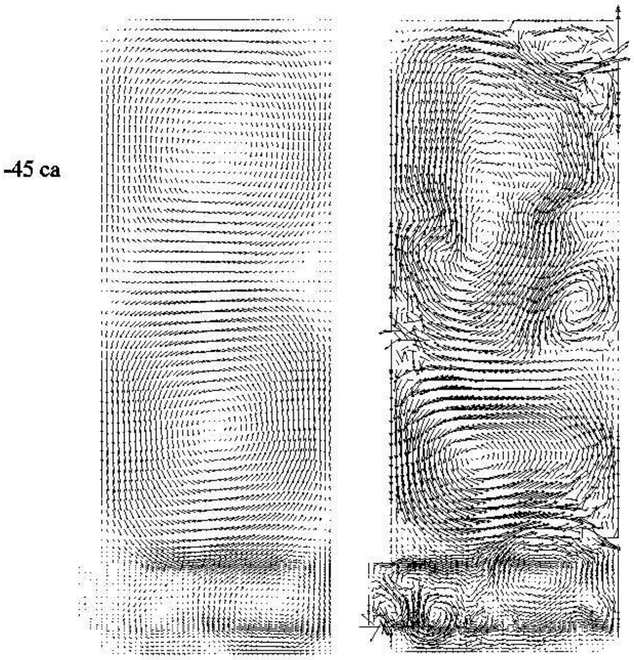

Figure 10: instantaneous velocity fields of the compression, 45° Crank Angles (CA) Before TDC (BTDC), obtained with Lax-Wendroff (left) and TTGC (right) schemes on 2D meshes. The velocity normalisation is the same.

Figure 9 presents instantaneous fields at the end of the compression stroke for 2D and 3D simulations compared to experimental fields. The first noticeable remark is about the structure of the flow. The 2D field is composed of three big contra-rotating vortices which do not appear in 3D or in the experiments. The second remark concerns the rotation speed of the vortices which is very important in 2D compared to the others. The disruption of the tumble is a typical 3D effect that could be compared to a forced Kolmogorov cascade. In 2D, the tumble stretching is not possible and the Kelvin theorem demonstrates in this case that a compression along the x-axis induces a expansion along the y-axis. This deformation is unstable and leads to a tumble break-down into smaller vortices, conserving the kinetic momentum. Therefore 2D simulations have to be discriminated for LES of turbulence compression like those found in SI engines.

Another point of importance is the effect of the numerical scheme on the flow structures. The Lax-Wendroff scheme is known to be dissipative and more accurate schemes specifically tailored for LES exist such as TTGC presented in previous Sections. A comparison of 2D instantaneous fields during the compression stroke is proposed on Figure 10. Whereas the LW velocity field is very smooth, the TTGC field includes a lot of small vortices and the presence of three big vortices is not as clear. For a given resolution TTGC allows the convection of finer structures and a potential better prediction of sub-grid turbulence.

Mean profiles

The instantaneous field comparisons have yielded qualitative remarks that can be quantitatively confirmed by plotting mean profiles. These 1D velocity profiles are extracted from mean fields as illustrated on Figure 11 and they are presented on Figure 12 to Figure 14 for different crank angles during intake and compression.

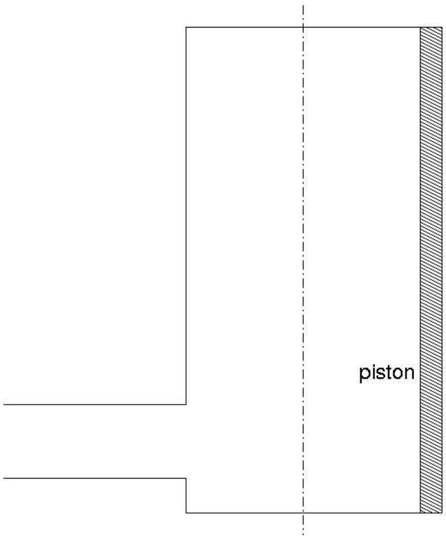

Figure 11: 1D cut in the median plane for mean profile comparisons.

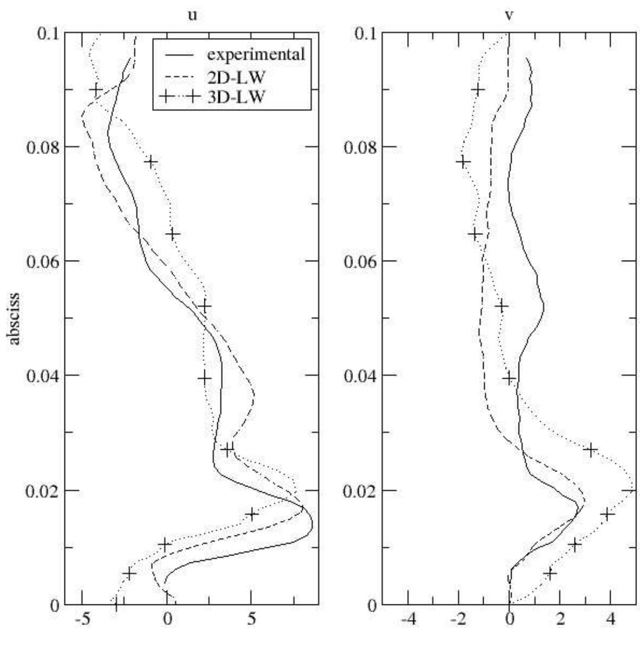

Figure 12: mean velocity profiles at 270°CA BTDC (intake).

2D mean fields have been obtained averaging the 15^{th} last cycles of a 20-cycle computation. Due to CPU costs, only six 3D cycles have been simulated and averaged. More cycles will be available in the future to compute better converged means. Concerning experimental fields, about 120 PIV realisations were available for averaging in the database.

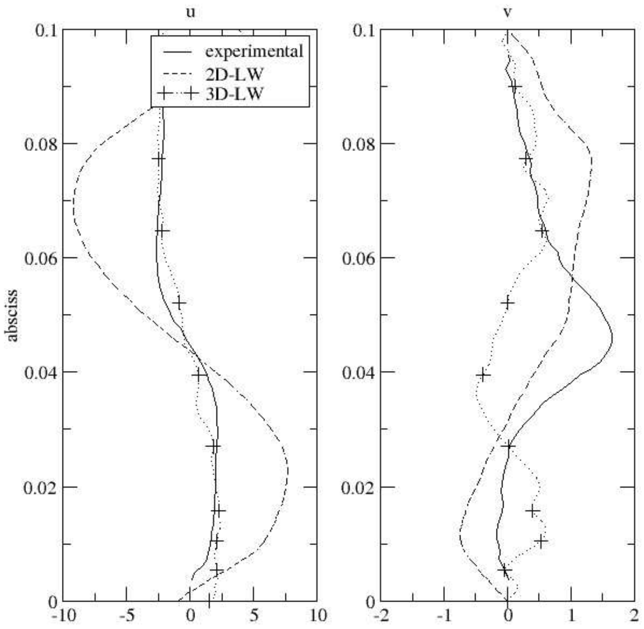

Figure 13: mean velocity profiles at 90°CA BTDC (compression).

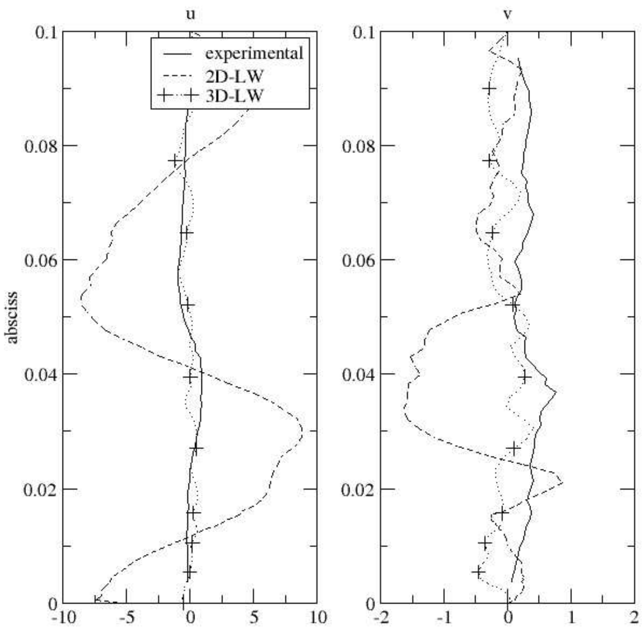

Figure 14: mean velocity profiles at TDC (end of compression).

Figure 12 shows that the intake stroke is well predicted by 2D computations. Since only few cycles have been averaged for the 3D case, it is more difficult to conclude on the predictivity. Figure 13 and Figure 14 are taken during the end of the compression stroke. Instantaneous observations have shown that this stroke can not be simulated accurately with 2D methods and mean profiles confirm these remarks. 2D profiles give too high levels of u whereas 3D profiles are in a far better agreement. At 90°CA BTDC, a big non-physical vortex occupies the totality of the 2D combustion chamber and it is broken down into three vortices at TDC.

These comparisons confirm that 2D simulations have to be discriminated for IC engine flows as expected. They are unable to predict correctly the disruption of the tumble motion. However 3D simulations seem promising and additional tests with different turbulence models and finer grids need to be performed to confirm these first results.

CONCLUSIONS & PERSPECTIVES

In this paper the developments performed to make the unstructured parallel LES CFD code AVBP applicable for IC engine applications were described, along with first results obtained on a engine-like cold flow.

An ALE methodology was implemented and tested in combinations with 2^{nd} order finite volume and 3^{rd} order finite element convective schemes. The formulation of the method was shown to move the grid without affecting numerical characteristics as precision and order of convergence, as compared to fixed grid simulations.

In order to prepare IC engine simulations, the CTI mesh management technique was then implemented into the AVBP code. A first mesh-only application to the PSA XU10 engine served in order to validate the implementation in the context of unstructured grids and domain decomposition parallelism. The additional costs for CTI were shown not to exceed 10% as compared to simulations without it. Parallel efficiency was shown not to be affected, the linear speed-up characteristic of AVBP being preserved with CTI. It was also shown that interpolation between meshes with different topologies, as needed during the simulation of an engine cycle using CTI, had to be carefully adapted for LES, in order to keep as much as possible the spectral characteristics of the resolved flow.

Finally first validations of the capabilities of the so achieved IC engine LES code AVBP were presented. Comparisons of LES simulations obtained on 2D and 3D meshes and using two convective schemes were compared with experimental findings for the square piston set-up. The simulations clearly showed that albeit the mean flow is essentially 2D during the intake phase, only a full 3D simulation is able to capture the breakdown of the tumble during piston compression. The simulations also showed that using the 3^{rd} order TTGC convective scheme allowed, for a given mesh resolution, to resolve more flow details near the cut-off scale than a 2nd order centred LW scheme, a definitive advantage for LES simulations. The mean velocity profiles obtained by phase averaging six consecutive 3D engine cycles with a standard Smagorinsky LES turbulence model combined to wall laws and using a LW scheme showed encouraging reproduction of mean experimental profiles.

Future work is presently concerned with applying a LES turbulence model based on a transport equation for sub-grid scale kinetic energy to the square piston, with the aim of improving turbulence predictions during compression. Efforts are also made to analyse more deeply the simulation outcomes, in particular by comparing predictions of velocity fluctuations over a large number of engine cycles with experimental findings. The aim is then to realise LES simulations of a real configuration.

Work is also currently under way to develop an automatic mesh management technique, adapted inflow/outflow boundary conditions for LES engine simulation, as well as a LES combustion model for premixed spark-ignited combustion. The final objective will be to demonstrate the feasibility of industrial LES engine simulations, and their ability to predict cyclic variations in SI IC engines.

ACKNOWLEDGMENTS

The authors would like to express their gratitude to Prof. Jacques Borée for kindly providing them the experimental database on the square piston. They are also indebted to Marc Zolver & Arnaud Torres for their great help in adapting the CTI technique to AVBP.

This work was made possible by the financial support of the European Commission (LESSCO2 project, contract number ENK6-CT-2002-00616) and of IFP's Techniques for Energy Applications Division.

REFERENCES

1. Amsden, A. A., Hirt, C. W., & Cook, J. L., "An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speeds," Journal of Computational Physics, Vol. 14, p 227, 1974.
2. Amsden A., O'Rourke P. & Butler T., “KIVA II: A Computer Program for Chemically Reactive Flows with Sprays,” Report LA11560-MS, Los Alamos, National Laboratory, 1989.
3. Borée, J., Maurel, S. & Bazile, R., “Disruption of a compressed vortex”, Physics of Fluids, Vol. 14, p 2543-2556, 2002.
4. Celik, I., Yavuz, I. & Smirnov, A., “Large Eddy Simulations of In-Cylinder Turbulence for IC-Engines: A Review”, Int. Journal of Engine Research, Vol. 2, No. 2, 2001.
5. Colin, O. & Rudgyard, M., "Development of High-Order Taylor-Galerkin Schemes for LES", J. of Comp. Physics, Vol. 162, pp. 338-371, 2000.
6. Ducros, F., Comte, P. & Lesieur, M., “Large-eddy simulation of transition to turbulence in a weakly compressible boundary layer over a flat plate” Journal of Fluid Mechanics, Vol. 326, p 1-36, 1996.
7. Ducros, F., Nicoud, F. & Poinsot, T., “A Wall-Adapting Local Eddy-Viscosity Model for Simulations in Complex Geometries” In Proceedings Conf. on Num. Meth. Fluid Dyn., Oxford, UK, 1998.
8. Farhat, C., Geuzaine, P. & Grandmont, C., “The Discrete Geometric Conservation Law and the Nonlinear Stability of ALE Schemes for the Solution of Flow Problems on Moving Grids” Journal of Computational Physics, Vol. 174, p. 669 - 694, 2001.
9. Lartigue G., Roux S., Benoit L., Poinsot T., Meier U., & Bérat C.. "Studies of unsteady cold and reacting flow in a swirled combustor using experiments, acoustics analysis and large eddy simulations". Proc. of the Comb. Institute, 30, 2004.
10. Lecomber, D. & Rudgyard, M., “Efficient parallel algorithms for numerical simulation” Future Generation Computer Systems, Vol. 17, p 961-967, 2001.
11. Marc, D., Borée, J., Bazile, R. & Charnay, G., “PIV and LDV measurements of tumbling vortex flow in a model square section motored engine”. Society of Automotive Engineers Paper (972834), 1997.
12. Moureau, V., Angelberger, C. & Colin, O., “On the Generalisation of High-order Schemes for Large Eddy Simulations on Moving Unstructured Meshes using an Arbitrary Lagrangian Eulerian Approach” 12th International Conference on Fluid Flow Technologies (CMFF-03), Budapest, Hungary, Sep. 3-6, 2003.
13. Rudgyard, M., “Cell Vertex Methods for Steady Inviscid Flow” Technical Report, Von Karman Institute for Fluid Dynamics, 1993.
14. Sagaut, P., “Large Eddy Simulation for Incompressible Flows, an Introduction” Springer, New York, 1998.
15. Sinha, N., Cavallo, P.A., Lee, R.A., Hosangadi, A., Kenzakowski, D.C., Dash, S.M., Affes, H. & Chu, D. “Novel CFD Techniques for In-Cylinder Flows on Tetrahedral Grids”, Society of Automotive Engineers Paper (980138), 1998.
16. Smagorinsky, J., “General Circulation Experiments with the Primitive Equations” Monthly Weather Review 91, No. 3, p 99-164, 1963.
17. Tessicini, F., Iaccarino, G., Fatica, M., Wang, M. & Verzicco, R., “Wall modeling for large-eddy simulation using an immersed boundary method” Center for Turbulence Research Annual Research Briefs, 2002.
18. Torres, A. & Zolver, M. “Combustion moteur sur maillages non-structurés, partie 1” IFP Technical Report, 53026, 2000.
19. Verzicco, R., Mohd--Yusof, J., Orlandi, P. & Haworth, D., “Large-Eddy Simulation in Complex Geometric Configurations Using Boundary Body Forces” AIAA Journal, Vol. 38, p 427, 2000.

20. Zolver, M., Klahr, D., Bohbot, J., Laget, O. & Torres, A. “Reactive CFD in Engines with a New Unstructured Parallel Solver” Oil & Gas Science and Technology, Vol. 58, p 33-46, 2003.
21. Zolver, M., Klahr, D. & Torres, A. “An Unstructured Parallel Solver for Engine Intake and Combustion Stroke Simulation” Society of Automotive Engineers Paper (2002-01-1120), 2002.

CONTACT

Dr. Christian Angelberger
IFP TAE R102
1 & 4 avenue du Bois-Préau
92852 Rueil-Malmaison (France).
Tel: +33 1 47 52 57 45
Fax: +33 1 47 52 70 68
christian.angelberger@ifp.fr