19th International Symposium of Ballistics, 7–11 May 2001, Interlaken, Switzerland

COMPUTATION OF MUZZLE FLOW FIELDS USING UNSTRUCTURED MESHES

A.B. Crowley, and J. Szmelter

Cranfield University (RMCS), Shrivenham, Swindon, Wiltshire. SN6 8LA, U.K.

ABSTRACT

This paper examines the feasibility of using a new unstructured meshes based code in the computation of muzzle blast flow fields. Most of the codes employed so far have used a finite difference or finite volume approach on structured meshes. The use of unstructured meshes allows for easy body fitted mesh generation with concentration of points in key arrays. This may be computationally advantageous and may alleviate some of the problems experienced by codes based on non-body fitted meshes, particularly for very complex shapes. The principal aim of the study was to assess if unstructured meshes based methods are practical for the extension to three-dimensional computations. The performance of two selected codes for structured and unstructured meshes is compared.

INTRODUCTION

Recent advances in computational fluid dynamics (CFD) techniques have led to the development of a number of codes [1–3] for the computation of intermediate ballistics flow fields during projectile launch. There is considerable interest in understanding the details of such flow fields as they affect the projectile launch and the dynamic loading of projectile, and sabot if present. Anomalous loadings have also been recorded by projectile mounted pressure sensors. Moreover the conditions are somewhat hostile, and the presence of instrumentation may affect the details of the flow field, as demonstrated in [3]. For these reasons CFD provides an attractive option for enhancing our knowledge of the complex flows involved.

Most of the codes employed so far have used a finite difference or finite volume approach on structured meshes, and some high resolution schemes have experienced problems in following the collapse of the shock bottle as the venting flow of propellant product gases weakens. In this regime there is a bow shock around the tail of the projectile as the venting gas accelerates past the projectile, which interacts with the receding Mach disc.

It may be therefore advantageous to explore an alternative approach – solvers based on body fitted computational meshes. In this paper the application of triangular meshes is investigated. Such a choice allows for easy modelling of complex shapes and in contrast to structured non-body fitted meshes allows for improved representation of the projectile geometry. Moreover, triangular meshes lend themselves well to adaptive mesh techniques, such as mesh movement and dynamic remeshings. Boundary conditions for body-fitted meshes can be applied directly. The advantages and limitations of the unstructured meshes approach will be highlighted in view of the potential generalisation to a fully three-dimensional tetrahedral mesh based solver.

NUMERICAL TECHNIQUES

Two codes were chosen for this study. They solve 2D flow, which is a mixture of two gas phases with different γ's. They differ very substantially in their principles, therefore any comparisons which are made are only of the qualitative nature. However, such comparison already permits one to observe the general features of methodologies which they utilize.

Structured Meshes Code

The MBIB2 code [1] is a two-phase flow code developed originally for the study of particulate impact on the muzzle brake during the intermediate ballistics phase of projectile launch. It is assumed that the highly transient compressible, but inviscid, flow is symmetric about the barrel axis. Turbulence is not included in the current code. A moving projectile may be included in the simulation if required, and has been included in this study.

The code solves the finite volume form of the equations of conservation of mass, momentum and energy for the gas phase using a cell-centred Eulerian approach. If a particulate phase is present, mass and momentum equations for this phase are added. Zalesak's multi-dimensional flux corrected transport algorithm [4] is employed to limit the overshoots and undershoots commonly associated with the numerical computation of shock waves. The low order flux is calculated using a first order exact 1D Riemann solver, and the higher order flux is predicted using a second-order central difference scheme based on a predictor-corrector method.

It is assumed that both gases, that is the propellant gas efflux and air, satisfy the ideal gas law with the appropriate gas constant and that the mixture properties are given by appropriate weighted averages.

A number of calculations have been undertaken to validate this code against published results in [3], and have demonstrated that the results from this code are in good agreement with experimental data.

Unstructured Meshes Code

The two-dimensional Euler equations written in the Eulerian, Cartesian coordinates are solved by an in-house code, which is a modification of the code described in [5]. It is assumed that both gas phases flow with the same velocity, with each of two gases occupying the volume fraction. A cell vertex finite volume discretisation in space and multistage Runge-Kutta discretisation in time are employed. The code operates on triangular meshes. The mesh generation is based on the advancing front technique. The dynamic, moderate mesh movement during the time-dependent calculation uses a spring analogy concept [6]. This means that the mesh is interpreted as a spring network where each edge of each triangle represents a spring with stiffness inversely proportional to the length of that edge. The grid points along the outer boundary of the mesh are fixed and the instantaneous locations of the points on the surface of projectile's segments are either prescribed or specified by the solution of trajectory equations. The position of the interior nodes is found by solving the static equilibrium equations that result from a summation of forces at each node in both the x and y directions. When cells become too distorted, remeshing is performed using the advanced front technique. The code is general and allows for any geometrical shape and mutually moving bodies to be considered. It has been extensively validated for a range of steady-state cases. Unsteady flow and moving mesh capabilities have been validated for the NACA 0012 oscillating The dynamic remeshing permits the adaptive tracking of flow features such as shocks, with error estimation based on the pressure gradient.

NUMERICAL EXAMPLES

Open Shock Tube

This standard test case was chosen in order to asses the general qualitative agreement between the two codes, bearing in mind the difference between axisymmetric and two-dimensional equations as being used by the structured and unstructured meshes codes respectively. Single gas (air) phase only was tested with the following internal conditions: barrel radius = 7.60 cm, static pressure = 3.492×10^5 Pa, static density = 2.8116 kg/m^3, static temperature = 432.55 K, Mach number = 0.8104 and axial velocity = 337.932 m/s; and external conditions: barrel radius = 15.2 cm, static pressure = 1.013×10^5 Pa, static density = 1.225 kg/m^3, static temperature = 288.00 K. A 100x100 Cartesian grid was used in the structured meshes code. The computational mesh of 3709 points and 7293 triangular cells used for the calculation is shown in Fig. 1. The representative pressure contour plot obtained from the unstructured mesh calculation at the time 1505 ms is shown in Fig. 2. Features such as the Mach disc are clearly present and in the correct position. There are no distinct differences in the contour plots obtained by the structured code. A history of pressure changes at two sample points (locations: axial = 30.500 cm, radial =11.249 cm and axial = 41.800 cm, radial =19.745 cm), obtained from the structured meshes code are presented in Fig. 3 and show consistent agreement with the results taken from [1]. Corresponding history of pressure changes obtained from the unstructured meshes two-dimensional code is shown in Fig. 4. As expected the values of pressure are much higher than those obtained by the axisymmetric solver. However, the consistency in shape and time positions with the results shown in Fig. 3 is easily noticeable. Comparisons of pressure history in other sample points confirmed this.

81 mm Mortar

An experimental study of overpressure histories in the muzzle flow-field of an 81 mm mortar system was undertaken recently as part of an ongoing study of gun-break signatures. Twenty rounds were fired on full charge, and the pressure histories were recorded at six different positions in the resulting flow-field, at distances between 10 and 30 calibre (D) from the muzzle, on rays at angles of 5°, 30° and 60° from the forward extended barrel axis. A detailed description of this example can be found in [3].

In the unstructured body fitted meshes this problem requires a combination of the dynamic remeshing and mesh movement in order to reflect the position of moving projectile. Examples of triangular meshes obtained at different computational times are shown in Fig. 5 (mesh movement) and Fig. 6 (remeshing). Pressure contour plots at two selected times are shown in Figure 7. They correspond to the results with inlet conditions taken from the internal ballistics calculation of gun tube venting after shot exit. The time of 1.75 ms after the start of venting corresponds to the maximum extent of the shock bottle, which begins to shrink thereafter. A clear bow shock is formed around the tail of the projectile while it is enveloped in the expanding gas. Comparison of experimental results with unstructured meshes code was not conducted, since quantitative comparisons with two-dimensional code are not possible, as illustrated in the first example.

CONCLUSIONS

An initial study of the potential of the unstructured meshes approach to modelling of muzzle blast has been conducted, and was directed mainly to overcome difficulties in dynamic remeshing and mesh movement, specific to the body fitted techniques. It has been observed that special treatments are necessary in cases when two objects in the field come to direct or close contact during the projectile movement. This is a common feature of the muzzle blast problem and although specific treatments for the presented numerical examples have been found there remains a need to search for a new more general approach. The somewhat unexpected finding was a relatively fast computational running time of the unstructured mesh based programs by comparison with the structured meshes code. This is due to much fewer computational points being required by unstructured meshes code, since the flexibility of the mesh generation allows for the computational points to be placed in more optimal manner. Further qualitative comparisons are necessary, for which an axisymmetric version of the unstructured meshes code in being developed.

Home
Table of Contents

REFERENCES

1. J.C.T. Wang & G.F. Widhopf, "Numerical simulation of blast flow-fields using a high resolution TVD finite volume scheme", *Computers & Fluids,* 18,103–137, (1990).
2. R. Cayzac, E. Carette, T. Alziary de Roquefort, C. Vaglio, J. Brossard, "Intermediate ballistics computations and validations", *Ballistics '98, 17th International Symposium on Ballistics,* Midrand, South Africa, (1998).
3. A.B. Crowley, "Muzzle flow field evolution", *EFBP '2000,* (2000)
4. S.T. Zalesak, "Fully Multidimensional Flux-Corrected Transport Algorithms for Fluids", *J. Comp. Phys.* 31, 335–362 (1979).
5. J. Szmelter, S. Abdullah, "A Trajectory Prediction for Segmented Projectiles Using CFD Code", *Journal of Battlefield Technology, Vol. 2, No. 2,* July 1999.
6. J.T. Batina, "Unsteady Euler Algorithm with Unstructured Dynamic Mesh for Complex-Aircraft Aerodynamic Analysis", *AIAA Paper 89-1189*, April 1989.

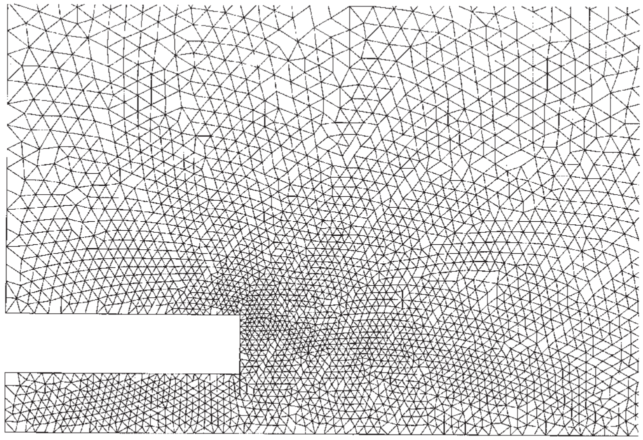

Figure 1. Shock tube – unstructured mesh.

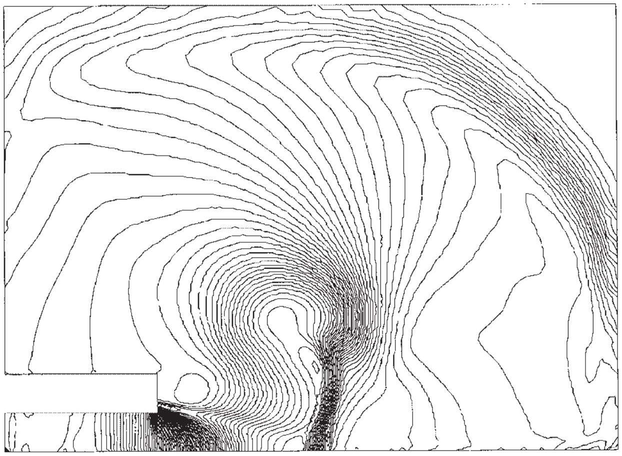

Figure 2. Shock tube pressure contours at time = 1505 μs.

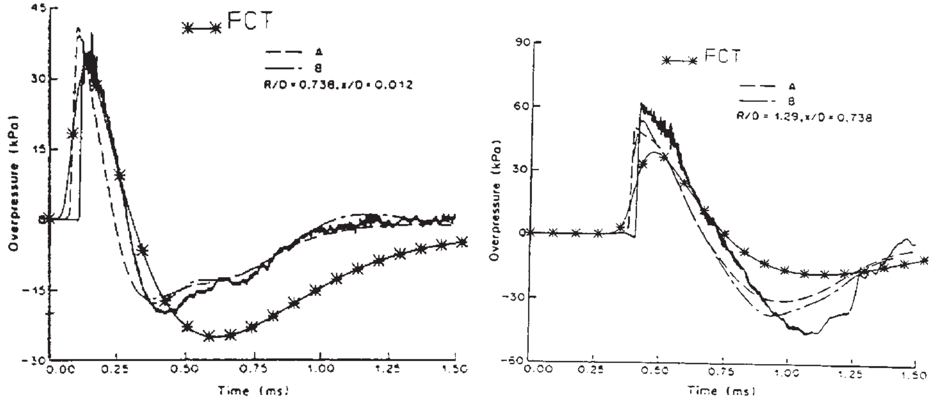

Figure 3. Pressure history – structured meshes axis symmetric.

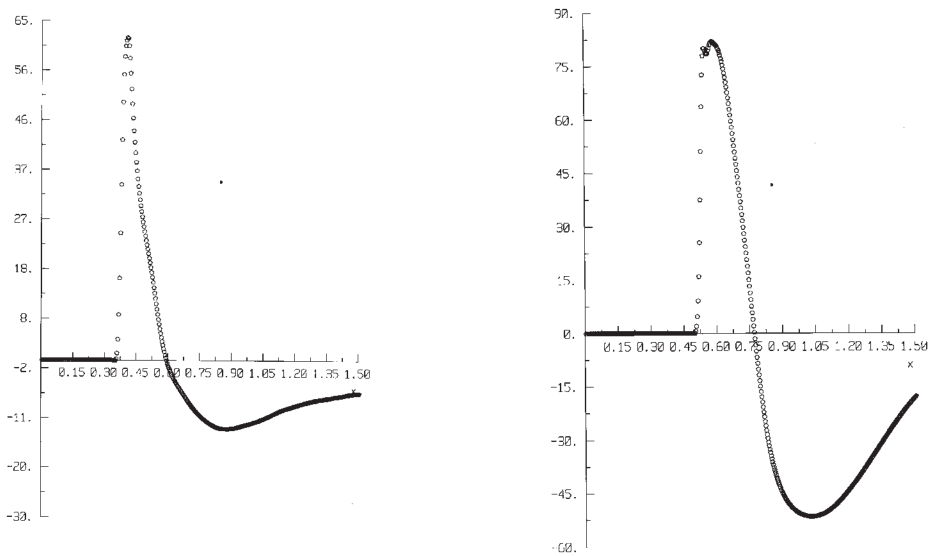

Figure 4. Pressure history unstructured meshes – 2-D.

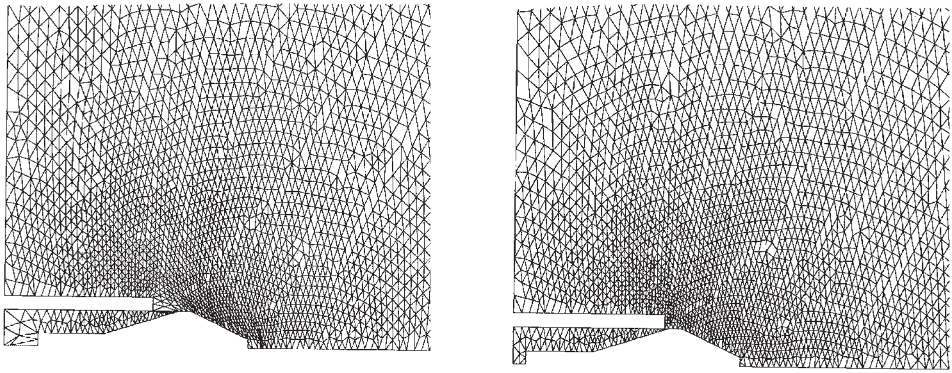

Figure 5. Details of the mesh movement before and after (exaggerated for illustration purposes).

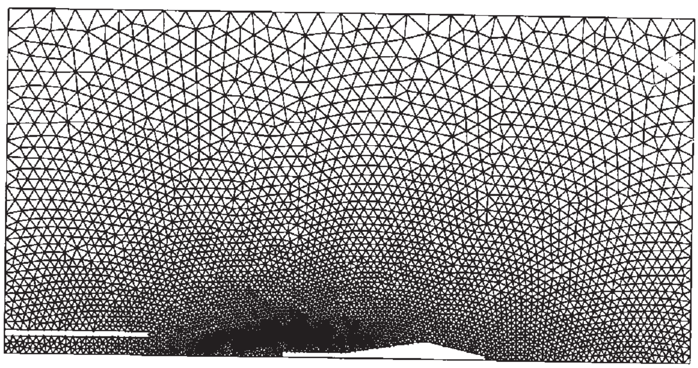

Figure 6. Remeshing of unstructured grid.

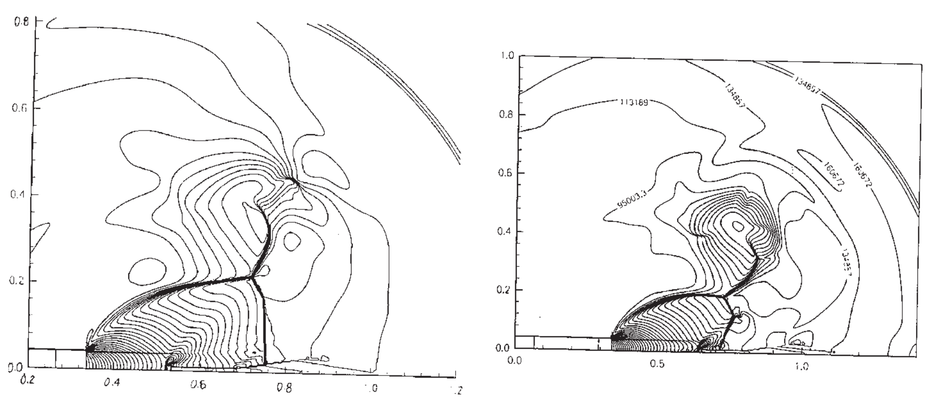

Figure 7. Pressure contour plots at 1.75 ms (left) and 2.15 ms (right) after shot exit.