HiSST: International Conference on High-Speed Vehicle Science Technology

26–29 November 2018, Moscow, Russia

Unsteady simulation of the flow in a hypersonic airbreathing vehicle air intake during cowl opening using a conservative overlapping mesh CFD technique

Marc Ferrier[1] *Philippe Grenard*[2]

Abstract

An original moving mesh CFD technic has been used to compute the flow around a hypersonic vehicle forebody and into its air intake during the engine cowl opening. This technic computes the intersection between a static background (the vehicle forebody) and a moving overlapping mesh (the cowl) at each time step. This ensure a continuity relationship between both meshes and the conservative property of the method. This work is in progress and, so far, only Euler simulations have been conducted. Results show the transient flow during cowl opening. When completely open, the air intake is started.

Keywords: *hypersonic vehicle, CFD, moving meshes*

Nomenclature

Latin

CFD – Computational Fluid Dynamics

BM – Background Mesh

OM – Overlapping Mesh

1. Introduction

A hypersonic vehicle cannot be operated from Mach 0 using a single scramjet engine. Instead, to provide the primary acceleration, a combined cycle based on rockets or turbojets has to be used. During this primary phase, for aerodynamic reasons, the scramjet flow path might be closed by an engine cowl. When the cowl opens, the flow in the air intake can be subject to transient phenomena that can lead to intake blockage [1]. It is then important to be able to simulate properly this unsteady flow. In this study, a generic hypersonic vehicle is considered. For this vehicle, the cowl opens in less than one second, which invalidates the quasi-steady approach, i.e., the steady simulation of different opening positions of the cowl with a complete remeshing of the geometry for each position. Instead, a real unsteady approach is considered, which requires an adaptable mesh during the simulation. This is achieved by using a conservative overlapping mesh technique.

As the CHIMERA technic [2, 3], the technic used in this study needs two meshes: a static background mesh and a moving overlapping mesh, but contrary to it, it does not necessitate interpolation between these two meshes. Indeed, a new mesh is rebuilt at each time step by computing intersection between the background and the envelop of the foreground mesh. If the faces velocity are also given, as well as the volume evolution of cut cells (see section 2), then the conservative property of the technique is ensured. The movement of the foreground mesh can be prescribed by a kinematic screw defining both translation and rotation. It can also be directly computed by the CFD by integrating efforts and momentum on the different boundaries defining the envelop of the moving mesh.

The work presented here is still in progress. For the moment, only Euler simulations have been conducted on the forebody of the vehicle for one flight point. The movement of the cowl is prescribed; efforts on

[1] *ONERA, Chemin de la Hunière, Palaiseau, France, marc.ferrier@onera.fr*
[2] *ONERA, Chemin de la Hunière, Palaiseau, France, philippe.grenard@onera.fr*

the cowl are not taken into account. The meshing methodology and the results are presented in section 3.

2. The moving mesh CFD methodology

In order to realize CFD computations with a moving body, we consider two meshes. The first one is static and corresponds to the main flow. It will be referred to as the ”background” mesh (BM). The second one can move with a solid body movement and corresponds to the moving part of the CFD (here, the cowl of the vehicle). It will be referred to as the ”overlapping” mesh (OM). The closed surface containing the overlapping mesh, corresponding to its contour, is called the ”polyhedron”. The principle of the methodology is to cut the background mesh by the polyhedron and rebuild cells connectivities between cut cells of the background mesh to internal cells of the overlapping mesh.

The intersection of the background mesh by the polyhedron is computed in a few steps:

- collisions of elements (point very close to a face for example) are detected in order to avoid numerical precision errors ;
- elements with collisions detected are moved by a small random displacement until they do not collide with any other elements ;
- the faces of the polyhedron are splitted in multiple ”subfaces”, each corresponding to the intersection of that face with a background volume cell ;
- each subface thus corresponds to a background mesh cell : it is then added to the list of faces of the background mesh ;
- each cut face of the background mesh is rebuild to keep only the ”visible” part of the face. Hided faces are removed from cells definition ;
- the connectivity between overlapping cells and background cells is rebuild through the use of each subface definition.

Once the mesh is rebuilt with that procedure, the new faces (coming from the polyhedron) can have a velocity vector defined by the global solid body movement of the overlapping mesh. Equations of fluid mechanics must thus be altered in order to take into account the different fluxes linked to face movement as well as volume regression.

The integral conservative form of fluid mechanics equations of a conservative variable q reads:

$$\frac{\partial}{\partial t}\left(\int_{\Omega(t)} q d\Omega\right) + \int_{\partial\Omega(t)} q\left(\underline{U} - \underline{V_f}\right)\cdot \underline{da(t)} = \text{Diffusion + source terms}$$

Once spatially discretized, this can be written as:

$$\frac{\partial}{\partial t}(q\Omega(t)) + \sum_i q\left(\underline{U} - \underline{V_{fi}}\right)\cdot \underline{A_{fi}} = \text{Diffusion + source terms}$$

which finally reads :

$$\Omega\frac{\partial q}{\partial t} + \boxed{q\frac{\partial\Omega(t)}{\partial t}} + \sum_i q\left(\underline{U}\boxed{-\underline{V_{fi}}}\right)\cdot \underline{A_{fi}} = \text{Diffusion + source terms}$$

Comparing to classical fixed mesh equations, the two highlighted terms are to be added in order to ensure conservativity. The blue term corresponds to the volumic rate of change of the cell whereas the red term is linked to the face movement which transfer some quantity of the q variable from one cell to another.

With a uniform and constant q field, with no diffusion nor sources, those two terms must thus verify the following equation in order to ensure conservativity:

$$\frac{\partial \Omega(t)}{\partial t} = \sum_i \underline{V_{fi}} \cdot \underline{A_{fi}}$$

3. Application to the cowl opening of a hypersonic vehicle

The computational case is the opening of the engine cowl of the generic hypersonic vehicle depicted on Fig. 1. This vehicle is intended to be released from a booster at hypersonic speed. During the acceleration phase, the cowl is closed in order to reduce the drag of the composite (booster and vehicle). When the release speed is reached, the cowl is open and the engine is started.For this study, the separation Mach number is taken equal to 7.1 and the cowl opens in 7ms.

Fig 1. CAD modeling of the vehicle.

3.1. CAD and mesh modeling

In this study, only the forebody of the vehicle, including the engine air intake, is calculated, and for symmetry reasons, only half a vehicle is considered. Fig. 2 shows the computational domain; the colored plane in the background corresponds to the symmetry plane. On this figure, several positions of the cowl are depicted, from closed to totally open. The internal flowpath (the engine flowpath) is visible in Fig. 3. For this simulation, the path includes the air intake and the very beginning of the combustor.

This case is particularly difficult to handle because the two solids are continuously in contact during the movement. On these contact surfaces, faces of both meshes are coplanar which makes the intersection computation difficult.

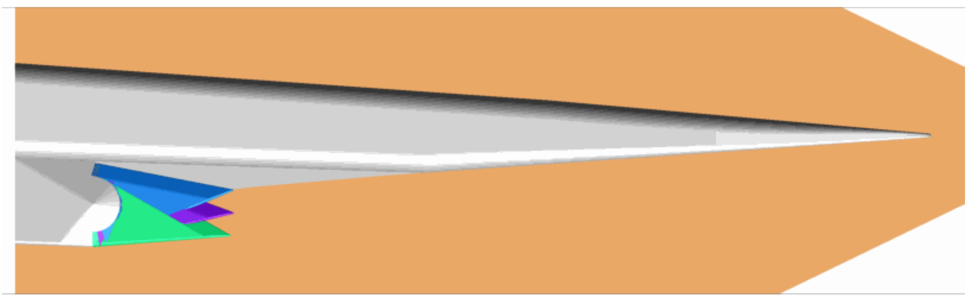

Fig 2. Computational domain. Different positions of the engine cowl are shown.

The meshing methodology is illustrated on Fig. 4. Tetrahedral cells are used. The BM and the OM contain respectively 5.4 and 0.5 million of cells. The BM is defined as the mesh of the computational domain without the cowl (top of the figure). The cells of this mesh will be cut by the surface mesh of the OM and the OM volume will be subtracted from the background. For the moment, only Euler

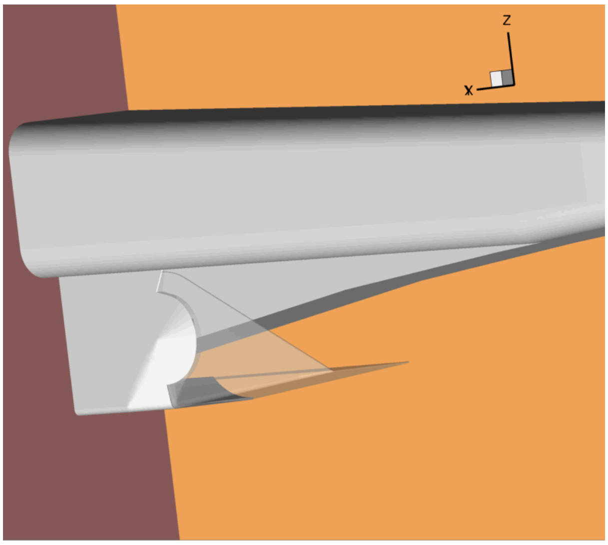

Fig 3. Engine flowpath of the vehicle.

simulations have been considered, then the OM that is subtracted to the background is the cowl itself (middle of the figure). Indeed, in case of RANS simulations, prismatic cells are generally added to the mesh close to the wall boundaries to compute properly the boundary layer. These cells lay down the walls along their largest dimension: they are aligned with the boundaries. For a moving mesh, the most convenient way to ensure this property is to include them into the OM, which means that the OM is, at least partially, made of fluid domains (i.e. the domains including the prismatic cells). For an Euler simulation, an isotropic mesh is sufficient at the wall boundaries. Such a mesh will result directly from the cutting of the BM by the cowl surface. In other words, for this Euler simulation, only the mesh of the solid volume of the cowl is needed and the surface of the OM will be seen as wall boundaries by the background.

It is noticeable, on Fig. 4, that the entire cowl is meshed (whereas only half a forebody is considered for the background). This is also clearly visible on Fig. 6. This procedure allows to limit cases where faces of background and overlapping meshes are coincident. Even if this case is taken into account by the algorithm, it is time consuming and potentially source of problems.

Some particular zones of local mesh refinement are visible on the background, especially on Fig. 5. These zones correspond to the path of the moving leading edge of the cowl. It is indeed more coherent that the size of a cutting cell is similar to the size of the cut cell. If not, then the cutting procedure of a large background cell by small overlapping cells will result in a new large cell and the local refinement is lost.

Finally, the resulting background mesh is visible on Fig. 7 & 8. It is seen that the cutting procedure results in polyhedral cells, and then this technic is only accessible to solver that can handle such meshes.

3.2. Numerics and physics modeling

In this study, air is considered as a thermally and calorically perfect gaz and Euler equations are used to model the gaz dynamic. The boundary conditions of the computational domain are:

- supersonic inlet,
- supersonic outlet for the external flow,
- supersonic/pressure outlet (depending on the characteristics curves) for the internal flow.

The simulation is initialized with a velocity equal to zero in accordance with flow conditions in the internal

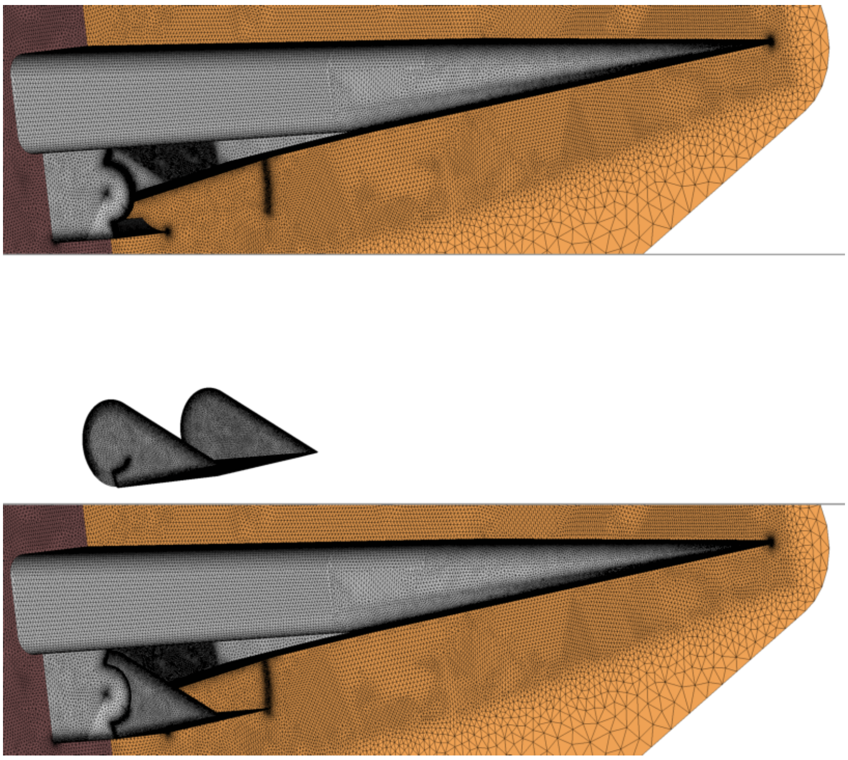

Fig 4. Top: background mesh. Middle: cowl mesh. Bottom: assembly of background and cowl meshes.

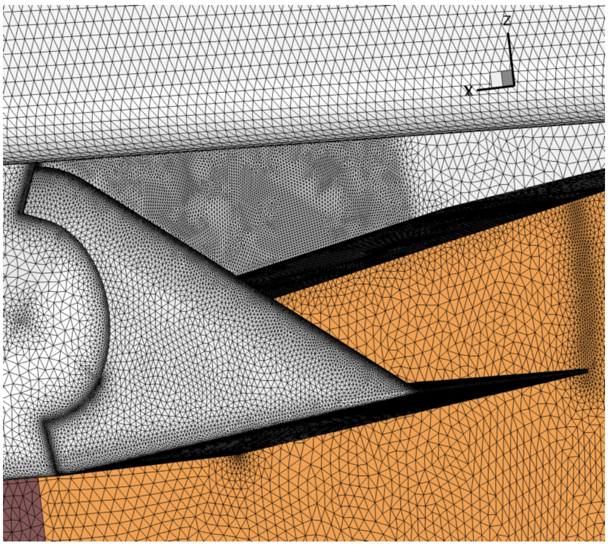

Fig 5. Zoom on the cowl from the inside of the computational domain.

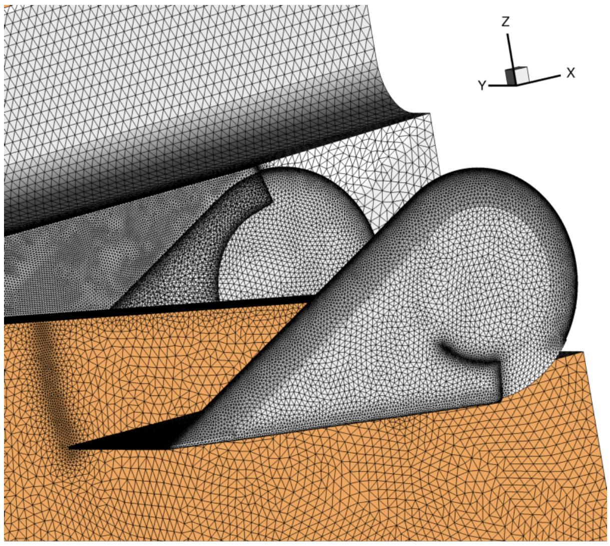

Fig 6. Zoom on the cowl from the outside of the computational domain.

flow path as long as the cowl is closed. A first computation is first converged with a closed cowl. Then the cowl is open with a prescribed constant rotation velocity. During the cowl opening, the velocity in the internal flow path is progressively increasing from zero to supersonic Mach numbers, and accordingly, the internal outlet boundary is switching from pressure outlet to supersonic outlet condition.

Numerical schemes are:

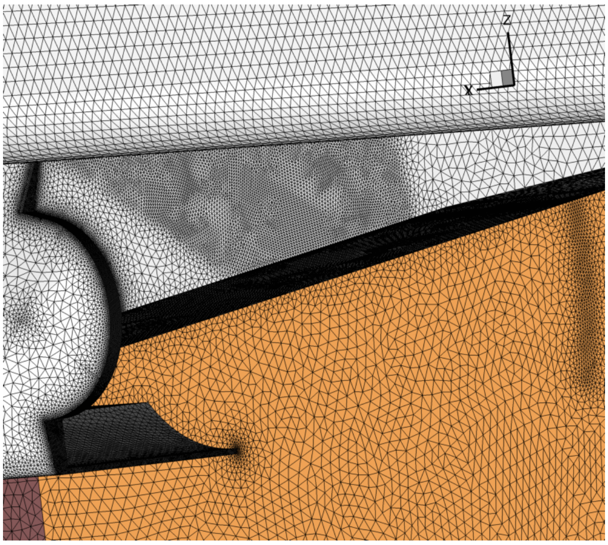

Fig 7. Background mesh before subtraction of the mask.

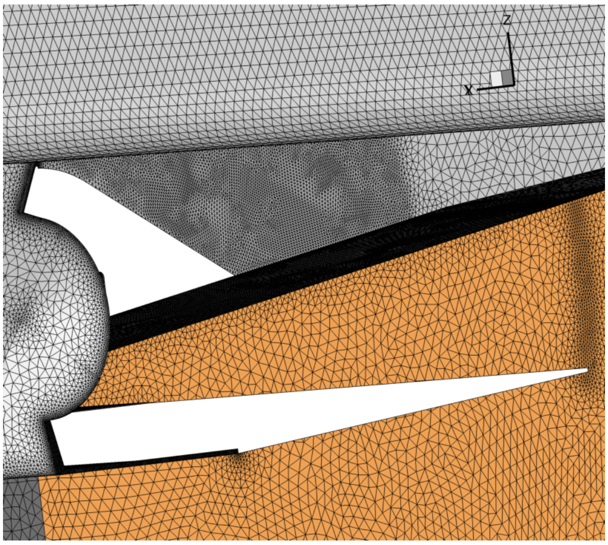

Fig 8. Background mesh after subtraction of the mask.

- Time integration: implicit Euler scheme.
- Euler fluxes scheme: HLLC.
- 2nd order in space: limited MUSCL type interpolation.

3.3. Results

The computation is performed on 448 Intel Xeon « Broadwell » cores. 15000 time steps of 1 μs are used to complete this simulation. The restitution time is approximately 17 hours, which is not excessive despite of the intersection procedure done at each time step. This procedure represents between 2 and 17% of the duration of a time step, depending on the number of cells to intersect in a sub-domain.

Results of the simulation are shown on Fig. 9 for different time steps and beginning with a converged solution with a closed cowl (a). Subsonic regions are colored in blue. In (b) the cowl starts to open and the flow becomes partially supersonic in the air intake. As long as the cowl is not entirely open, a step, visible in (c) and (d), is present at the interface between the cowl and the engine. This step, in presence of supersonic flow, results in a normal shock wave whose intensity decreases as the step regresses. In the same time, the oblique shock visible at the windward side of the closed cowl weakens as it progressively lays in the flow direction under the external compression ramp of the forebody. At time step (e) the cowl motion is stopped. At this moment, the air intake is completely started. The computation is run 1000 more iterations until reaching the result visible in (f). The internal flow path has not changed compared to (e), contrary to the flow under the cowl that did not adapt instantaneously its new position.

4. Conclusion

An original moving mesh CFD technic has been used to compute the flow around a hypersonic vehicle forebody and into its air intake during the cowl opening. This technic computes the intersection between a static BM (the vehicle forebody) and a moving OM (the cowl) at each time step. This ensures a continuity relationship between both meshes and the conservative property of the method.

So far, only Euler computations have been conducted on a tetrahedral mesh. In these simulations, the OM is the volume mesh of the cowl. The intersection is computed between the background and the

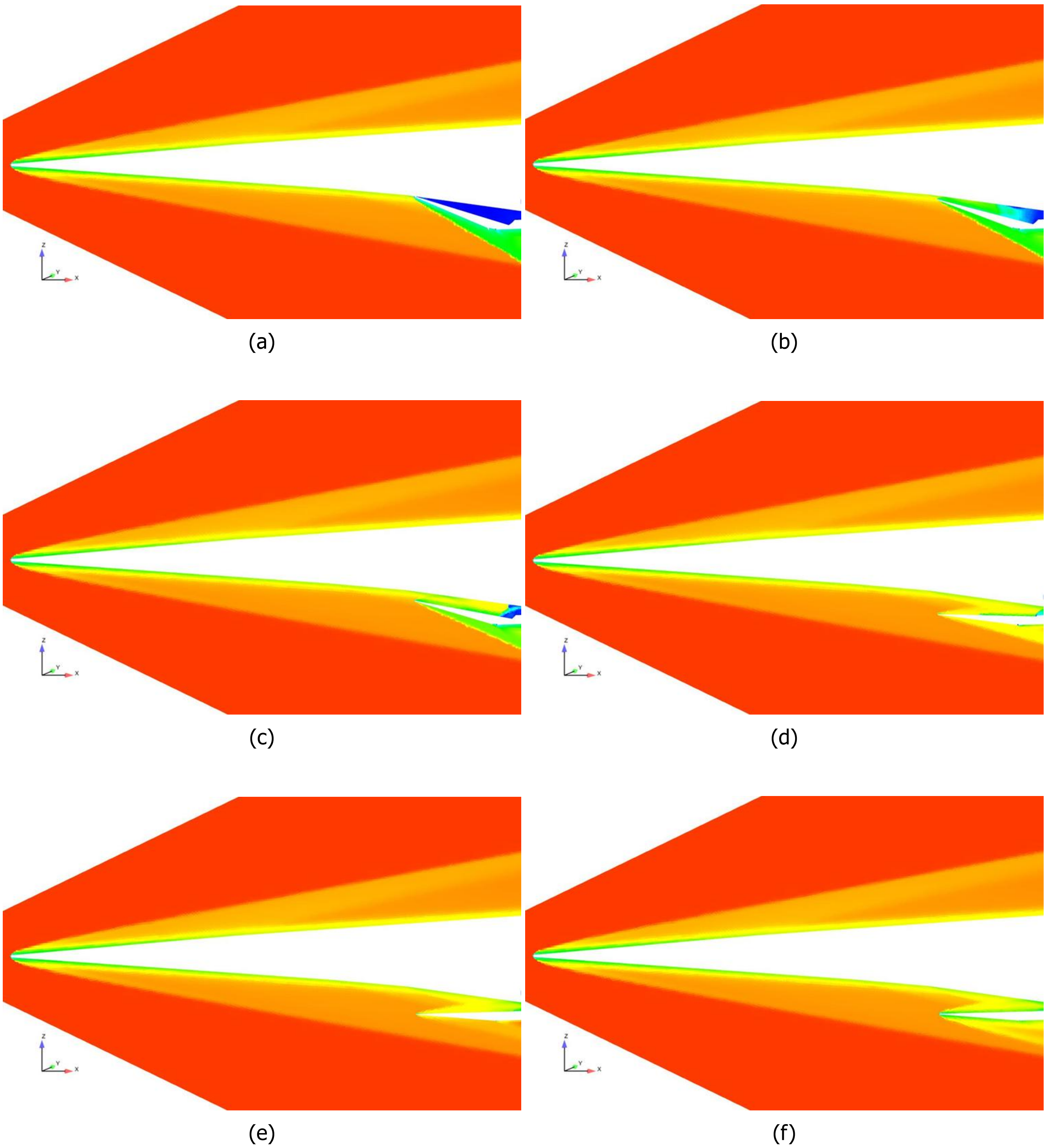

Fig 9. Mach field at different instants of the cowl opening.

envelop of the OM, then the cowl is subtracted from the background domain. The difficulty of the case lies in the fact that some faces of both meshes are continuously coincident during the movement.

Results show the transient flow in and around air intake during cowl opening. When completely open, the air intake is started.

This result must now be confirmed by RANS simulations. This requires to add prismatic cells at wall boundaries into the BM and the OM, which means that the OM will contain fluid domains. This represents an additional difficulty for this case again because of coincident surfaces.

References

[1] H. Ogawa, A.L. Grainger, and R.R. Boyce. Inlet starting of high-contraction axisymmetric scramjets. 16th AIAA/DLR/DGLR International Space Planes and Hypersonic Systems and Technologies Conference. AIAA 2009-7401, October 2009. Bremen, Germany.

[2] B. Landmann and M. Montagnac. A highly automated parallel chimera method for overset grids based on the implicit hole cutting technique. *International Journal for Numerical Methods in Fluids*, 66(6):778–804, June 2011.

[3] T. Renaud, M Costes, and S. Péron. A computation of goahead configuration with chimera assembly. *Aerospace Science and TechnologyAerospace Science and Technology*, 19(1):50–57, June 2012.