HiSST: 4th International Conference on
High-Speed Vehicle Science & Technology

22–26 September 2025, Tours, France

Mesh Regularization for Hypersonic Flow and Fluid-Structure Coupling in Atmospheric Reentry

Jérôme Breil[1], Vivien Loridan[2], Laurent Muscat[3], Florent Nauleau[4], Marina Olazabal[5]

Abstract

The paper presents a structured mesh regularization strategy based on the line sweeping method, which has been modified to enable the enforcement of wall orthogonality and refinement of the mesh near the wall. The method enhances mesh regularity and is integrated into the CEA inhouse aerothermal simulation code, allowing seamless application to both steady and unsteady computations. The proposed framework is demonstrated to be effective in regularizing structured meshes, resolving tangled cell issues, and enforcing wall orthogonality in hypersonic simulations. The method is also shown to be effective in a fluid-structure interaction case, where the thermal protection system undergoes ablation in an unsteady simulation. The results demonstrate that the method significantly enhances the robustness and accuracy of hypersonic reentry simulations, particularly in the presence of strong shocks, thin boundary layer and wall deformations.

Keywords: *Hypersonic Flow, ablation, regularization, Arbitrary Lagrangian-Eulerian (ALE)*

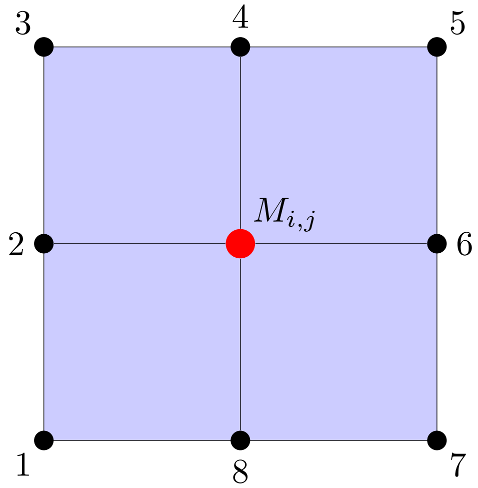
1. Introduction

The study of hypersonic flow and fluid-structure coupling during atmospheric reentry is a complex and challenging task that requires accurate numerical simulations. The boundary layer, a thin region near the wall, plays a crucial role in predicting aerodynamic heating, shear stresses, and pressure loads, which directly influence vehicle design and material integrity. However, the extreme conditions encountered during hypersonic flight, such as high Mach numbers and intense shock waves, make the accurate resolution of the boundary layer particularly challenging for computational fluid dynamics (CFD) solvers.

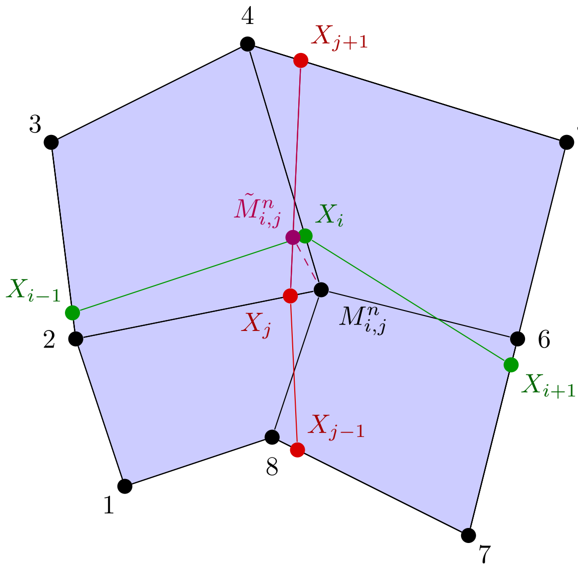
In structured-block aerodynamic codes, ensuring mesh regularity is important to accurately capture strong shocks and prevent numerical instabilities due to discontinuities. Unlike unstructured meshes, structured grids offer advantages in terms of computational efficiency, memory access patterns, and solver convergence. However, their block-wise nature can lead to alignment issues at block interfaces, which must be corrected to maintain solution accuracy. Additionally, when simulating flow over complex geometries or in the presence of moving boundaries, structured meshes require dynamic techniques to preserve orthogonality and smoothness.

To address these challenges, a structured mesh regularization method based on line sweeping [1] was implemented, where nodes are adjusted along the structured grid lines. This method was then adapted to take into account the constraint of orthogonality of the meshes to the wall for the geometries treated. Following the regularization step, normal-to-wall adaptation was applied to refine the mesh following an exponential law, ensuring proper boundary layer resolution [2]. Unlike external preprocessing tools, this method is directly integrated into the simulation code, allowing seamless application to both steady and unsteady computations.

[1] *CEA-Cesta, 15 avenue des Sablières–CS60001, 33116 Le Barp Cedex, France, jerome.breil@cea.fr*
[2] *CEA-Cesta, 15 avenue des Sablières–CS60001, 33116 Le Barp Cedex, France, vivien.loridan@cea.fr*
[3] *CEA-Cesta, 15 avenue des Sablières–CS60001, 33116 Le Barp Cedex, France, laurent.muscat@cea.fr*
[4] *CEA-Cesta, 15 avenue des Sablières–CS60001, 33116 Le Barp Cedex, France, florent.nauleau@cea.fr*
[5] *CEA-Cesta, 15 avenue des Sablières–CS60001, 33116 Le Barp Cedex, France, marina.olazabal@cea.fr*



(a) Node to be moved in red, stencil for the Line-Sweeping method in black.



(b) Line-Sweeping regularization method in 2D.

Fig 1. Presentation of the Line-Sweeping regularization method in 2D with the stencil in order to move a node.

In this paper, several test cases will be presented to demonstrate the effectiveness of the method. First, the ability of the method to regularize structured meshes and resolve tangled cell issues, improving overall mesh quality, will be shown. In the context of hypersonic simulations, it will be illustrated how the method enforces wall orthogonality and corrects mesh misalignments which can arise from block assembly in meshing tools. Such misalignments can lead to numerical instabilities near shock waves. Finally, the method will be applied to a fluid-structure interaction case, where the thermal protection system undergoes ablation in an unsteady simulation. The mesh will dynamically adapt, in an Arbitrary Lagrangian-Eulerian (ALE) framework, to the receding wall while preserving orthogonality and boundary layer resolution. These results will demonstrate that the structured-block mesh regularization and adaptation strategy significantly enhances the robustness and accuracy of hypersonic flow and reentry simulations, particularly in the presence of strong shocks and wall deformations.

2. Regularization, adaptation and refinement of the mesh

2.1. Line-Sweeping regularization method

The Line-Sweeping method developed by Jin Yao [3] is a local iterative geometric technique used to regularize a structured mesh, ensuring a uniform smoothing of the mesh. The method involves repositioning each node based on the position of its neighbors until convergence of the regularization or until an acceptable solution is obtained.

The stencil used to move a node is depicted in Figure 1.(a), where the red node represents the one that needs to be repositioned. In Figure 1.(b), the Line-Sweeping method is illustrated in 2D. The black nodes correspond to the mesh at iteration n. Six fictitious points are calculated, with point X_{i-1} located in the middle of the branch of points 1 to 3. The other green points are also placed in the middle of the vertical branches. Red nodes like X_{j+1} are placed in the middle of the horizontal branches. The calculated point, $\tilde{M}^n_{i,j}$ is at the intersection of the red and green branches formed by the points X_i, X_j. Finally, the damping parameter θ is used to place the new node $M^{n+1}_{i,j}$ according to the equation (1).

Here, the node will be on the segment $[\tilde{M}^n_{i,j}; M^n_{i,j}]$ depending on the value of θ.

$$M_i^{n+1} = \theta M^n_{i,j} + (1-\theta)\tilde{M}^n_{i,j} \tag{1}$$

When dealing with non-uniform mesh on the wall, weights are used to propagate a non-uniform distribution of the nodes on the wall. The weights are such that they respect the ratio between the nodes on the wall, and using another damping parameter, a uniform mesh can be obtained far from the wall.

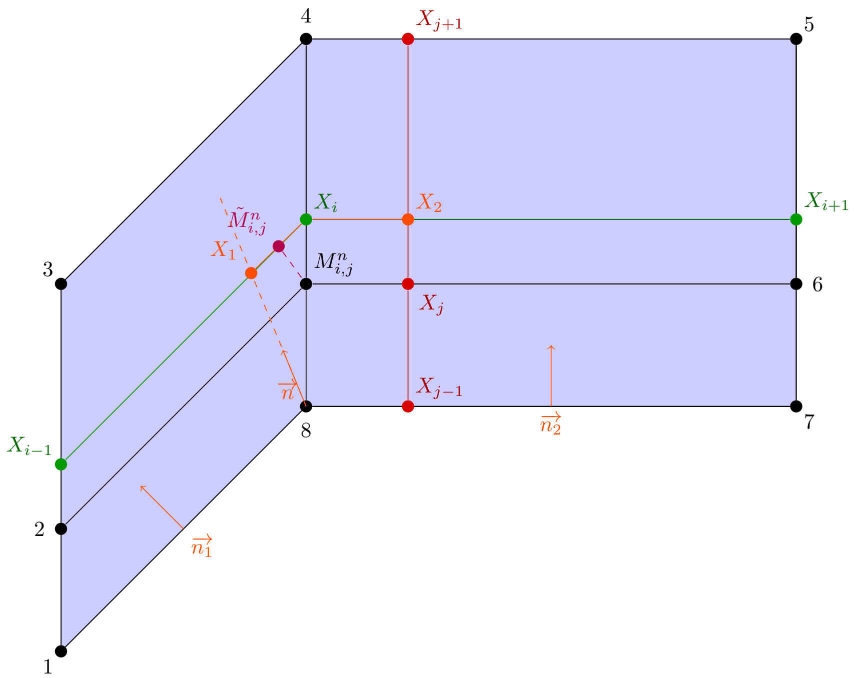


Fig 2. Line-Sweeping method with orthogonality criterion in 2D.

2.2. Line-Sweeping method adapted to the orthogonality constraint

To impose the orthogonality of the meshes on the wall, the Line-Sweeping method was modified to obtain orthogonal mesh on the wall. The stencil used to move a node is the same as for the Line-Sweeping 2D method shown in Figure1.(a). To find the new position of the node, two points are calculated, denoted X_1 and X_2 in Figure 2. The point X_2 is placed by the Line-Sweeping 2D method. To find the position of point X_1, the normals $\overrightarrow{n_1}$ and $\overrightarrow{n_2}$ are calculated relative to the segments of the wall as shown in the diagram, then the sum of these two vectors $\overrightarrow{n}$ is chosen to evaluate the point X_1. The point X_1 is then the intersection between the line carried by the vector $\overrightarrow{n}$ passing through the node $n°8$, and the branch of the X_i represented in green. These two points make it possible to define a new branch in orange and contained on the green branch. Finally, a coefficient $\alpha \in [0, 1]$ controls whether the new node will be closer to point X_1 or point X_2, but still on the orange branch. An example of the calculation of the α coefficient will be illustrated in the part presenting the results obtained using this method of regularization. For the calculation of the coefficient α, parameters related to the shape of the wall can be taken into account. In the context of this study, it is calculated according to the height of the node in the y direction and allows the constraint of orthogonality to the wall to be relaxed as the nodes is far from it. In Figure 2 the orthogonalization is done with respect to the adjacent mesh line.

2.3. Refinement using exponential law

The β law is a refinement law with an exponential function, commonly used in boundary layer simulation to obtain a very small cell close to the wall and few cells far from it. This can be applied in a context of

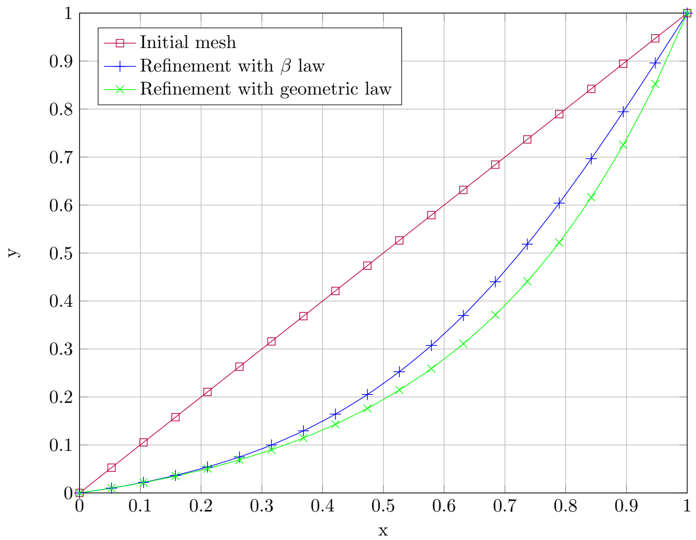


Fig 3. Comparison of the refinement of a 1D mesh by the Beta law and by the geometric law for a first mesh size imposed at 10^{-2}.

structured meshes to obtain a refinement of the meshes around the boundary layer. To apply this law on a 1D mesh, the nodes concerned are iterated over as follows:

$$x_i = x_1 + f_n(x_{N_x+1} - x_1) \tag{2}$$

with

$$f_n = 1 + \beta\frac{1-e^p}{1+e^p} \tag{3}$$

where $p = z(1-n)$, $z = log(r)$, $r = \frac{\beta+1}{\beta-1}$ and $n = \frac{i-1}{N_x}$. Here N_x is the number of cells and x_i is the i-th node of the mesh for $i \in 1, ..., N_x + 1$. $\beta \in [1.00001, 1.01]$ acts as an expansion coefficient. For low value of β, the mesh will be refined on the wall. The points x_1 and x_{N_x+1} at the edge of the domain are fixed.

Using a Newton method, it is possible to impose the size of the first cell and find the associated β coefficient depending on the total number of cells, as well as the length of the domain. This represents an advantage since it is possible to control the size of the mesh close to the wall where the boundary layer develops. In Figure 3, an example of refinement of a 1D mesh is presented with a first cell size imposed at 10^{-2} and is compared to a more classical geometric law.

3. Application to hypersonic flow simulations

3.1. Untangle a double ellipsoid geometry

The double ellipsoid geometry is a common test case for hypersonic flow simulations. The initial mesh of the geometry is generated using a structured grid, which leads to a tangled mesh (Figure 4.(a)). The purpose of this section is to demonstrate the ability of the line-sweeping method to untangle meshes as a regularization method. The line-sweeping method is applied to the tangled mesh of the double ellipsoid geometry, and the nodes are moved along the lines of the structured grid as shown in Figure 4.(a)-(g). This step improves the mesh quality by reducing the skewness of the cells and increasing the orthogonality of the mesh near the wall. Finally, the exponential law is applied to the regularized mesh, and the cells near the wall are refined. The resulting mesh has a higher resolution near the wall. The resulting mesh is suitable for aerodynamic flow simulations at Mach 8.2 (Figure 4.(h)), and can be used to study the hypersonic flow around the double ellipsoid geometry.

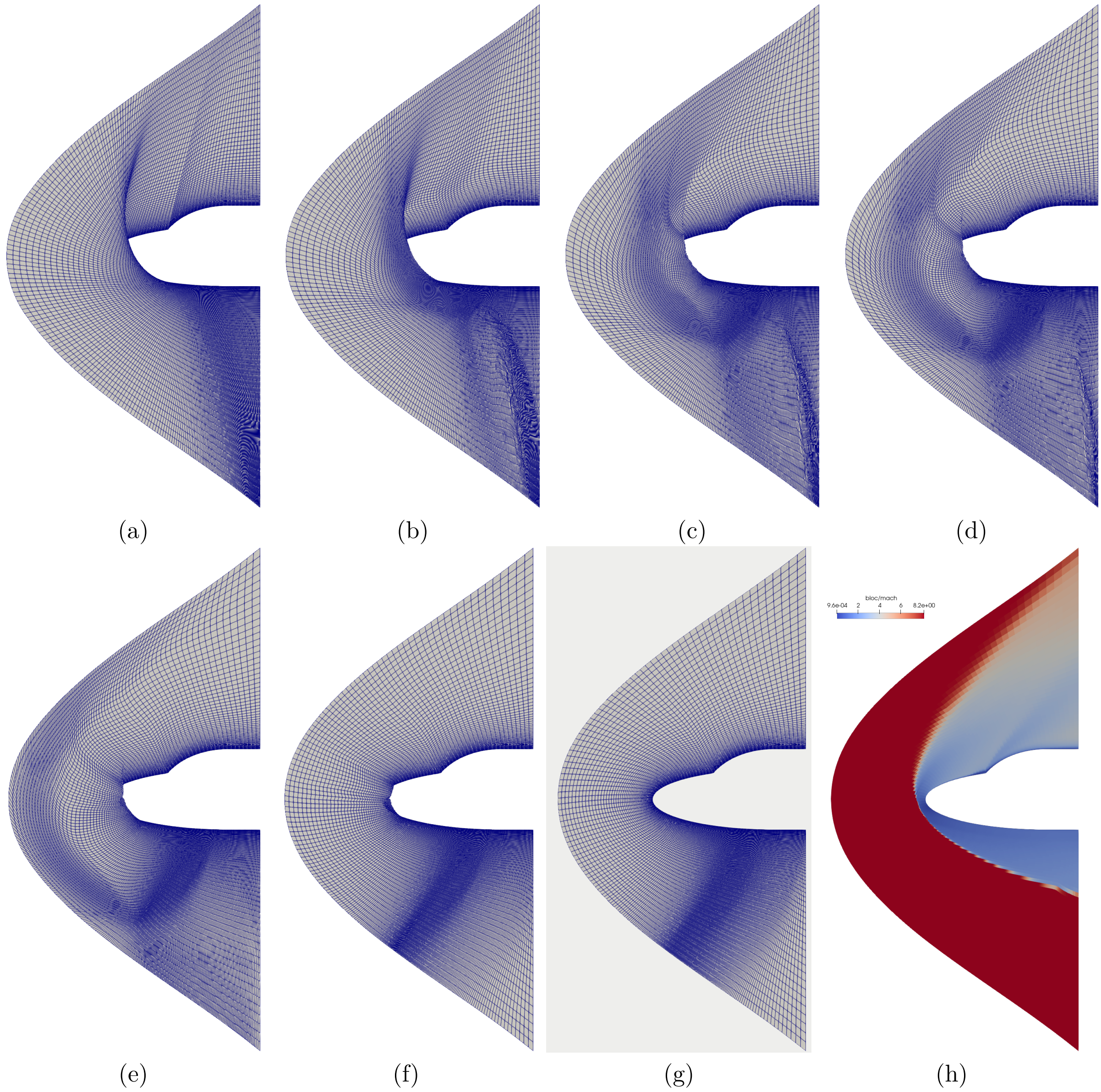


Fig 4. Regularization of a tangled mesh: The initial mesh (a) is tangled and cannot be used for aerodynamic calculations. The line-sweeping method is applied to untangle the mesh, and then a regularization step is performed to improve the mesh quality (b)-(f). Finally, the exponential law is used to refine the mesh near the wall (g). The resulting mesh is then suitable for aerodynamic calculations at Mach 8.2 (h).

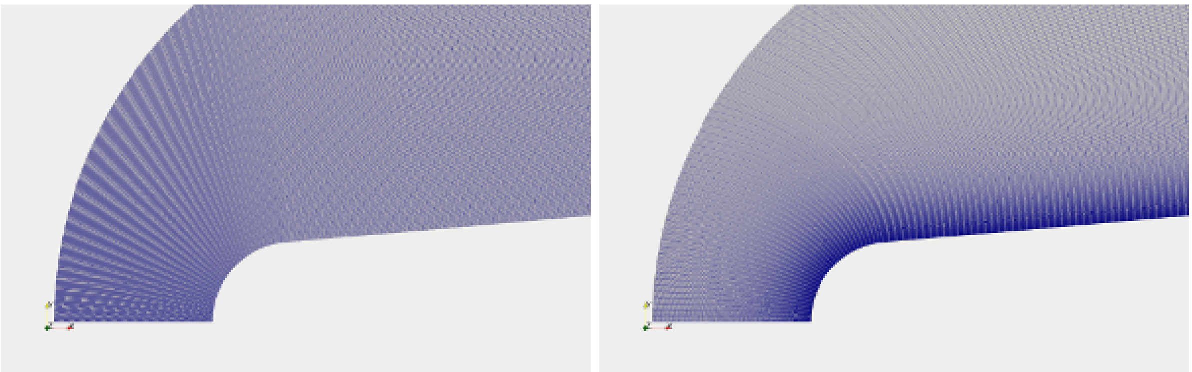
3.2. Case of a Cone/Cylinder/Flare geometry

The case of a Cone/Cylinder/Flare geometry (CCF) as presented in [5] is now considered. The shape of this object has been designed for wind tunnel experiments to study hypersonic boundary layer.

The boundary conditions are given in the Table 1. The initial mesh generated by the ICEM meshing tool shows straight lines on the mesh, which is a result of the blocking technique. When the regularization method is applied to the mesh, the effect of the method is clearly visible on the nose of the CCF geometry, as shown in Figure 5. A straight line on the mesh due to the blocking technique can be observed on the initial mesh obtained from the ICEM meshing tool. After regularization, the mesh is smooth and

Table 1. Boundary conditions of the reference flow along the CCF geometry.

Parameters	Values
M_∞	6
Re_∞	$5.57\ 10^6$
p_∞	164.0 Pa
ρ_∞	$0.02175\ \mathrm{kg.m^{-3}}$
T_p	300 K

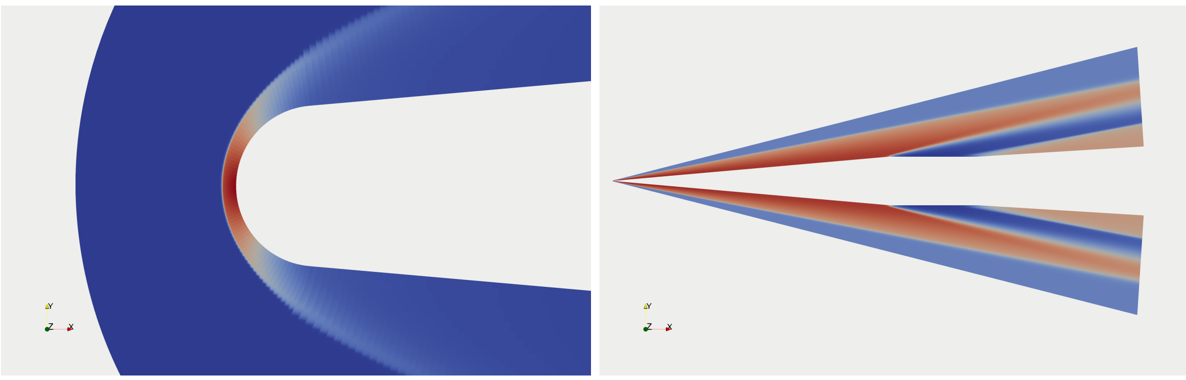


(a) Initial mesh from ICEM (b) Mesh after regularization

Fig 5. Zoom on the nose of the geometry.

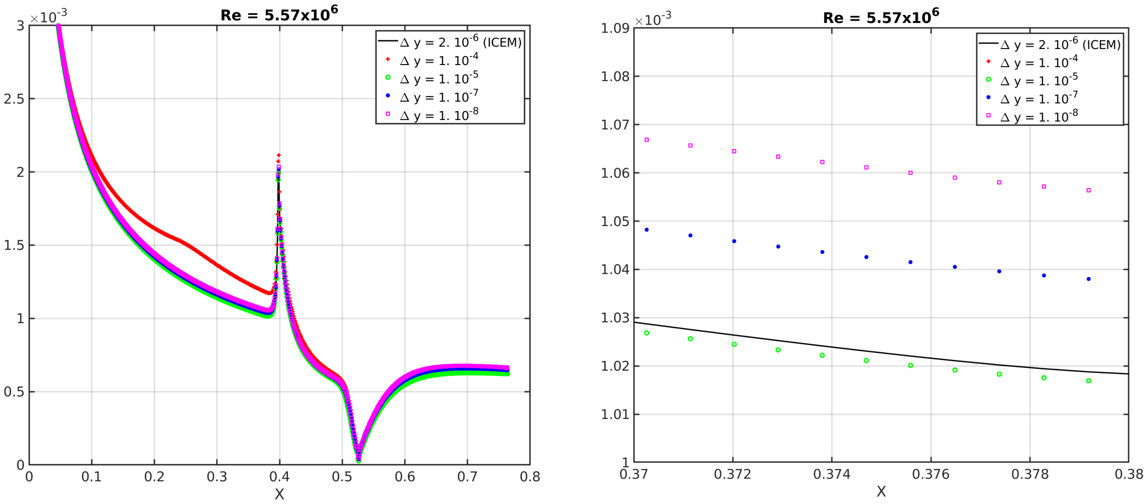
orthogonal to the wall.

Figure 6 shows the pressure around the CCF geometry, highlighting the shock at the top of the nose and the shock wave around the global geometry. Figure 7 presents the skin friction coefficient on the CCF geometry for different sizes of the first cell. The main advantage of the method is the control of the size of the first cell. Figure 7.(a) shows that if the size of the first cell is too large (here $1 \times 10^{-4}m$), the correct solution cannot be captured. On the other hand, if the size is too small, the number of iterations to converge increases dramatically (8000 instead of 1500 for smaller size of first cell). Moreover, if the first cell size is too small and the number of cells does not change, the resolution of the boundary layer may decrease as shown in Figure 7.(b).



(a) Zoom on the nose (b) Shock around the geometry

Fig 6. Pressure around CCF.



(a) Skin friction coefficient on the wall of CCF

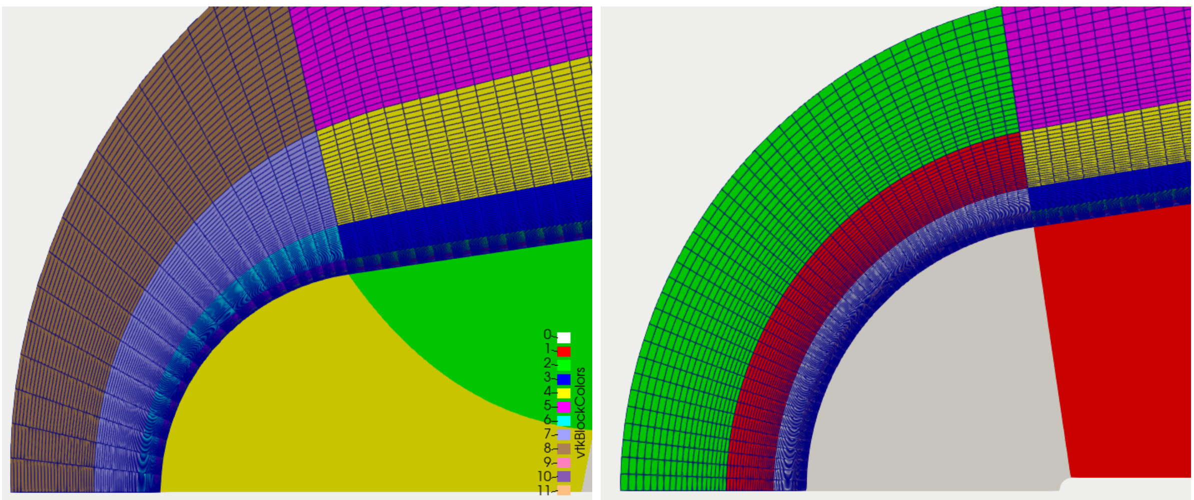
(b) Zoom on the skin friction coefficient

Fig 7. Skin friction coefficient.

3.3. Case of an ablating geometry

The case of an ablating geometry is a challenging problem for structured mesh regularization and adaptation. In this case, the thermal protection system undergoes ablation during an unsteady simulation, and the mesh must adapt dynamically to the receding wall while preserving orthogonality and boundary layer resolution.

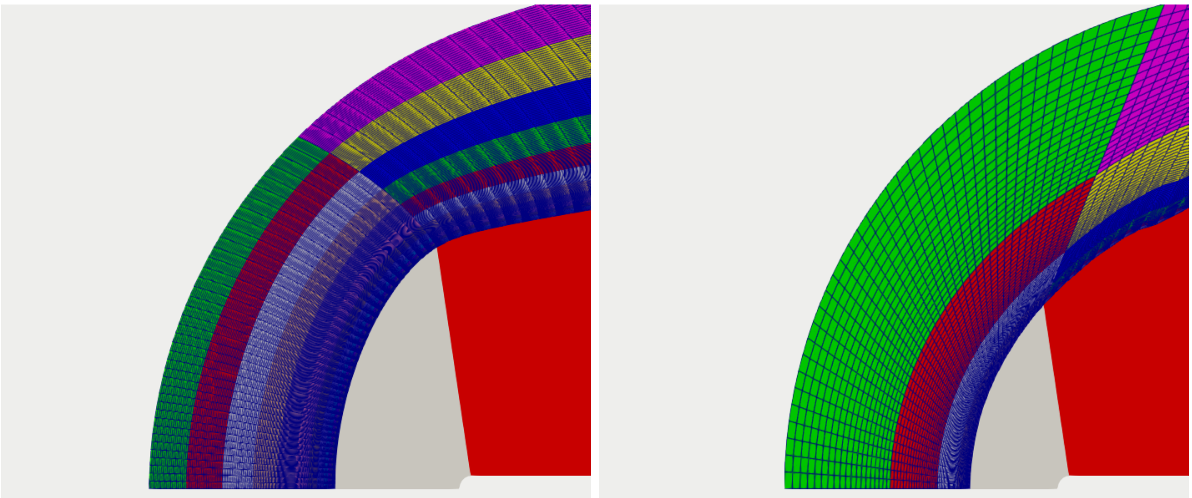
To address this challenge, our structured-block mesh regularization and adaptation strategy is applied to a fluid-structure interaction case. Two different meshes were generated, as shown in Figure 8. The mesh generated using ICEM was specifically built for this ablation problem, with the solid and fluid meshes moving during the simulation. The mesh generated using Gmsh was created without any anticipation of mesh movement.



(a) ICEM mesh

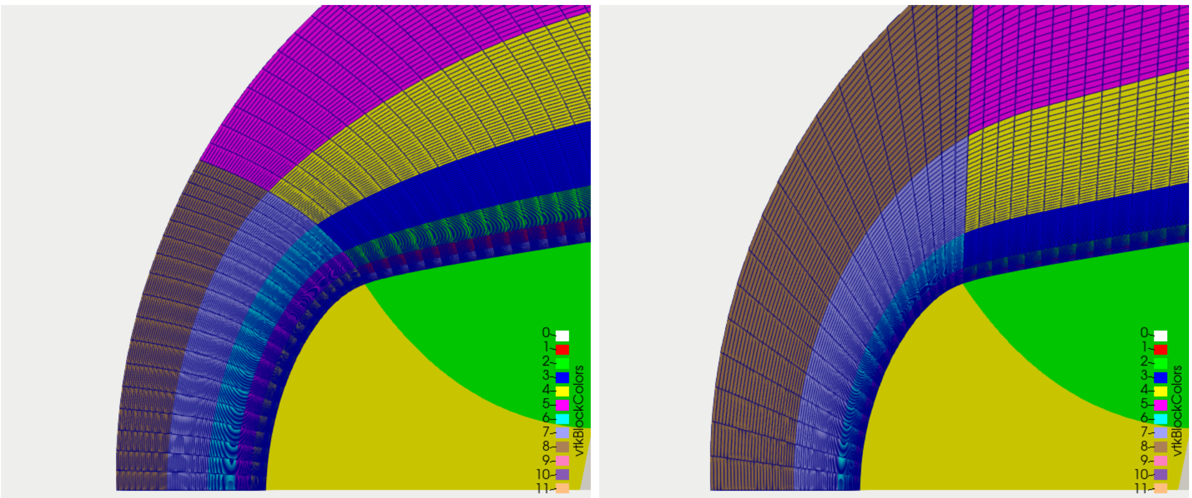
(b) Gmsh mesh

Fig 8. Two different meshing tools are used to generate initial mesh: ICEM (a) and Gmsh (b).



(a) with regularization (b) without regularization

Fig 9. Case of an ablating geometry with (a) and without (b) mesh regularization and refinement for the Gmsh mesh.



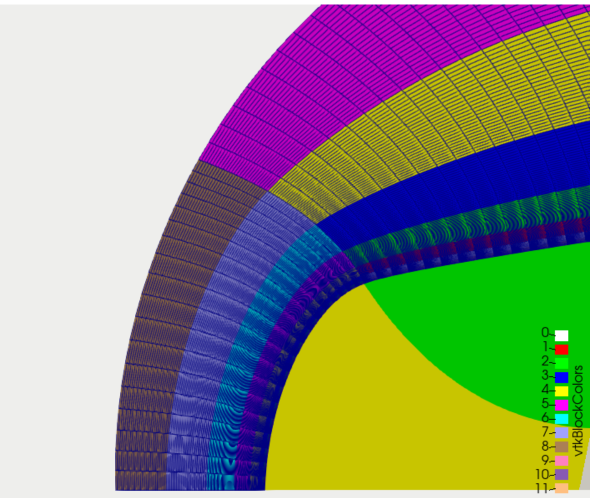
(a) with regularization (b) without regularization

Fig 10. Case of an ablating geometry with (a) and without (b) mesh regularization and refinement for the ICEM mesh.

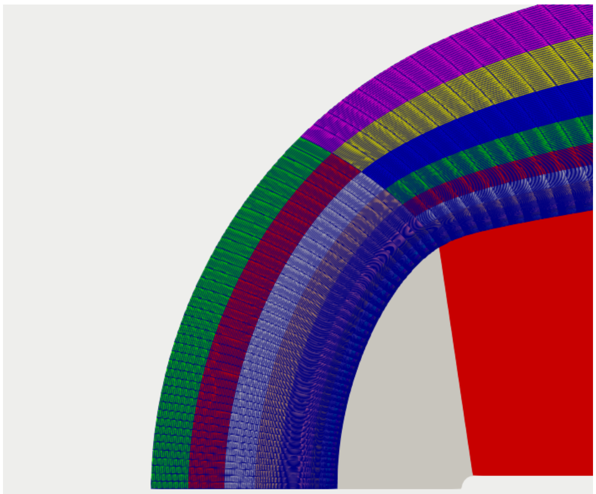
In the case where the initial mesh was generated using the Gmsh software, the same simulation was run with and without regularization. The results of the simulation are shown in Figure 9. Figure 9.(b) shows the final mesh without regularization, while Figure 9.(a) shows the mesh after regularization and refinement using the exponential law. Without regularization, the cells near the wall of the thermal protection are skewed, whereas orthogonality is maintained close to the wall with regularization. The results of the simulation without regularization are totally different from the results where orthogonality is imposed close to the wall. Simulation without regularization for long time simulation can also stop due to tangled cells. The results demonstrate that our structured-block mesh regularization and adaptation strategy significantly enhances the robustness and accuracy of hypersonic flow and reentry simulations.

The results of the hypersonic reentry simulations using the ICEM mesh are shown in Figure 10. Figure 10.(b) shows the final mesh without regularization, while Figure 10.(a) shows the mesh after regularization. During this simulation the wall is moving and the mesh generated using ICEM is orthogonalized using the line-sweeping method with an orthogonality criterion. The exponential law is then used to refine the mesh near the wall during ablation. The results demonstrate that our structured-block mesh regularization can handle hypersonic flow and ablation during reentry simulation, particularly in the presence of strong shocks and wall deformations. The geometry of the thermal protection at the end of the simulation with or without regularization is exactly the same and can be superimposed.

The comparison of the regularization results for ICEM and Gmsh meshes are presented in Figure 11. This shows that the results using our structured-block mesh regularization and adaptation strategy produce the same results using two different meshes. The mesh for reentry simulation can be constructed without anticipation of wall movement. The geometry of the thermal protection at the end of the simulation, shown in Figure10.(b) (ICEM mesh without regularization) and Figure11 (comparison of ICEM mesh and Gmsh mesh with regularization and refinement), is identical and can be superimposed.



(a) ICEM mesh



(b) Gmsh mesh

Fig 11. Case of an ablating geometry comparison of ICEM (a) and Gmsh (b) meshes with regularization and refinement.

4. Conclusion

In this paper, a structured mesh regularization method was presented based on line sweeping, where nodes are adjusted along the structured grid lines. This method enhances mesh regularity and enables the enforcement of wall orthogonality, a crucial requirement for accurate flux computation. Following the regularization step, normal-to-wall adaptation was applied to refine the mesh following an exponential law, ensuring proper boundary layer resolution.

The effectiveness of the method was demonstrated through several test cases. First, it was shown that the method could regularize structured meshes and resolve tangled cell issues, improving overall mesh quality. In the context of hypersonic simulations, it was illustrated how the method enforces wall orthogonality and corrects mesh misalignments which can arise from block assembly in meshing tools. Such misalignments can lead to numerical instabilities near shock waves. The method was also successfully applied to a CCF geometry, and using refinement techniques, the effect of the first cell size on the skin friction coefficient was controlled. Finally, the method was applied to a fluid-structure interaction case, where the thermal protection system undergoes ablation in an unsteady simulation. The mesh dynamically adapts, in an Arbitrary Lagrangian-Eulerian (ALE) framework, to the receding wall while preserving orthogonality and boundary layer resolution. The structured-block mesh regularization and adaptation strategy significantly enhances the robustness and accuracy of hypersonic flow and reentry simulations, particularly in the presence of strong shocks and wall deformations. The method can be applied to complex geometries like the double ellipsoid, and can be easily integrated into existing simulation codes. The ability to untangle meshes, refine the boundary layer, and take into account ablation makes the method a powerful tool for hypersonic flow simulations.

One possible perspective for the 3D case is the extension of the structured mesh regularization method to three-dimensional grids. This would involve adapting the 3D line sweeping technique [6, 7] and developing new techniques for enforcing orthogonality and boundary layer resolution in three dimensions. The ability to handle complex 3D geometries and to take into account ablation would make the method a valuable tool for a wide range of 3D ALE simulations [8].

References

[1] Yao, J. *A Mesh Relaxation Study and Other Topics*, LLNL-TR-637101, (2013).

[2] Roche, C. Breil, J. and Olazabal, M. *Mesh regularization of ablating hypersonic vehicles.* Proceedings of the ECCOMAS conference, (2022).

[3] Yao, J. *An Efficient Line-Sweep Mesh Relaxation Method with Locally Conservative Interface Reconstruction.* Lawrence Livermore National Laboratory, (2011).

[4] Roy, J. C. and Blottner, F. G. *Methodology for Turbulence Model Validation: Application to Hypersonic Flows.* Journal of Spacecraft and Rockets, 40, 313-325, (2003).

[5] Esquieu, S. Benitez, E. Schneider, S. P. and Brazier, J.-Ph. *Flow and Stability Analysis of a Hypersonic Boundary Layer over an Axisymmetric Cone Cylinder Flare Configuration* AIAA Forum Aviation, (2019).

[6] Breil, J., Damour, G. Guisset, S. and Colaïtis A. *3D mesh regularization within an ALE code using a weighted line sweeping method.* Comput. Fluids, 292, 106591, (2025).

[7] Damour, G. Guisset, S. Breil J. *3d mesh regularization based on a weighted line sweeping method.* IMR, (2024).

[8] Guisset, S., Damour, G. and Breil, J. *Cell-centered indirect Arbitrary Lagrangian-Eulerian numerical strategy for solving 3D gas dynamics equations.* J. Comput. Phys., 505, 4, 112903 (2024).