THREE DIMENSIONAL H/P ADAPTIVE DISCONTINUOUS GALERKIN METHOD FOR HIGH SPEED COMPRESSIBLE FLOWS

Konstantinos T. Panourgias¹ and John A. Ekaterinaris²

¹Department of Mechanical Engineering and Aeronautics
University of Patras
Rio, Achaia, Greece
e-mail: panourgias@iacm.forth.gr

²Aerospace Engineering
Embry-Riddle Aeronautical University
Daytona Beach, FL, USA
e-mail: ekaterij@erau.edu

Keywords: High-order methods, discontinuous Galerkin, limiting, h/p-adaptivity, unstructured meshes.

Abstract: High-order discontinuous Galerkin (DG) discretizations possess features making them attractive for high-resolution computations in three-dimensional flows that include strong discontinuities and embedded complex flow features. A key element, which could make the DG method more suitable for computations of these time-dependent flows in complex domains, is application of limiting procedures that ensure sharp and accurate capturing of discontinuities for unstructured mixed-type meshes. A unified limiting procedure for DG discretizations in unstructured three-dimensional meshes is developed. A total variation bounded (TVB) limiter is applied in the computational space for the characteristic variables. The performance of the unified limiting approach is shown for different element types employed in mixed-type meshes and for a number of standard inviscid flow test problems including strong shocks to demonstrate the potential of the method. Furthermore, increased order of expansion and adaptive mesh refinement is introduced in the context of it h/p–adaptivity in order to locally enhance resolution for three-dimensional flow simulations that include discontinuities and embedded complex flow features.

1 INTRODUCTION

Developments of high-order numerical methods for unstructured meshes could offer significant advantages for the simulation of complex high-speed flows, compressible turbulence [1], and high-speed combustion [2] in non-trivial geometries of interest to practical applications. The discontinuous Galerkin (DG) [3, 4, 5, 6], the spectral volume (SV) [5, 7], and the spectral difference (SD) [5, 8, 9] methods have shown promise for high-resolution computations of complex flows because they have a compact stencil, and retain the design order of accuracy even for meshes of moderate quality that would often result from grid generation over complex three-dimensional configurations. The potential applications of these methods for high-resolution simulations of practical flow problems can further be enhanced with the use of mixed-type meshes and solution adaptive schemes.

Despite of the advantages that high-order methods possess compared to traditional second-order accurate finite-volume methods, and uniformly high-order, finite-difference methods, such as ENO [10] and WENO [11], the higher computational cost prevents their more widespread use in practical applications. Solution adaptive refinement strategies of h-, p-, or h/p-type can reduce computing time for high-resolution simulations of complex flows without compromising numerical accuracy. The h/p finite element framework [12] has been adopted for the implementation of DG finite element [13] method. The key elements of this framework that are exploited in this work, are the hierarchical bases, and the collapsed coordinate transformations. Utilization of these features allows to perform all required for the implementation of the DG method operations for the standard cubical element. Modal, hierarchical bases are constructed in the computational space over a unit cube using tensor products and transformed back to the physical space of hexahedra, prisms, or tetrahedra using a collapsed coordinates system [12]. Modal bases facilitate mixed element implementation and adaptive mesh refinement with non-conforming faces.
The DG method is a mixture of the finite element and finite volume methods, which approximates the solution within each element as an expansion of basis functions, while the approximate solutions are discontinuous across the element interfaces. Thus, an arbitrary domain $\Omega$ is decomposed into an ensemble of non-overlapping elements, $\Omega = \Omega_1 \cup \Omega_2 \cdots \cup \Omega_e$, where $e$ is the number of elements in the mesh. For the DG method, however the expansion of the approximate solution is local for each element in $\Omega$ and no continuity is imposed on the element interfaces.
2.1 Spatial discretization of the Euler equations

For every element, the weak formulation of the Euler equations is formed by multiplying the conservative form of the governing equations with a weighting function \( w_i(x) \) and integrating over the element \( \Omega_m \):

\[
\int_{\Omega_m} w_i(x) \left[ \frac{\partial U(x,t)}{\partial t} + \nabla F(U) \right] d\Omega_m = 0, \quad m = 1, 2, \ldots, e
\]

where \( F = (f_x(U), f_y(U), f_z(U)) \) contains the flux vectors, \( U = [\rho, \rho u, \rho v, \rho w, E]^T \) is the state variables vector, \( \rho \) is the density, \( u, v, w \) are the Cartesian velocity components, \( E \) is the total energy, and the pressure \( p \) is related to state variables through the equation of state \( p = \gamma - 1 \left[ E - 0.5\rho(u^2 + v^2 + w^2) \right] \). As in other finite element methods, after substitution of \( U \) with an approximate solution \( U_h \), and integration by parts obtain:

\[
\frac{d}{dt} \int_{\Omega_m} w_i(x) U_h d\Omega_m = \int_{\Omega_m} \nabla w_i F(U_h) d\Omega_m - \oint_{\partial\Omega_m} w_i(x) F(U_h)n ds,
\]

where \( n = (n_x, n_y, n_z) \) is the outward normal vector to the faces of the element. For the DG method, \( U_h \) in eqn (2), represents the finite-element approximation of the solution within the element without continuity requirements at the element faces. This approximation is a combination of the polynomial basis functions \( b_j \) of degree \( k \) at most:

\[
U_h = \sum_{i=1}^{N} c_i(t) b_i(x),
\]

In this expansion, the solution coefficients, \( c_i(t) \), are the degrees of freedom to be advanced in time for each variable in the element, and in our implementation \( b_i(x) \) represent tensor products of the Jacobi polynomial basis functions, which in the Galerkin context, belong in the same set as the weighting functions \( w_i(x) \) in eqn (2). When the basis functions are polynomials \( P^k \) of degree \( k \) the most, then for smooth flow problems the optimal order of accuracy that can be achieved is \( k + 1 \). The term \( F(U_h)n \) in eqn (2) is replaced with a suitable numerical flux \( H(U_{h+}, U_{h-}) \), where \( U_{h+} \) is the exterior and \( U_{h-} \) is the interior of the element solution. The local Lax-Friedrichs (LLF) flux has been used.

2.2 Implementation of the DG method

The implementation of the DG method for arbitrary shape elements (hexahedra, tetrahedra etc.) of the physical domain \( \Omega \) is carried out in the computational space, denoted by \( \Omega_{st} \), for the standard cubical element configuration (see Fig. 1). Every element of the physical space, linear or higher-order, can be transformed to the standard cubical element. The transformation maps that are applied in order to transfer elements to the canonical cubical element of the standard computational space are obtained by utilizing the collapsed coordinate system [12].

![Figure 1: Limiting of \( P_1 \) expansions on the standard cubical element of the computational domain.](image-url)
Fine resolution of discontinuities and complex flow structures, which appear in time dependent compressible flows, is achieved apart from the higher-order approximations (p-adaptivity), by the use of adaptive mesh refinement (AMR/h-refinement). During the solution evolution, elements identified for refinement are split into four (2d problems solved with 3d meshes) or eight (purely 3d problems) smaller and self-similar elements and the solution of the original element is projected to the finite element space of the new generated elements. At the same time, a de-refinement procedure is performed by merging to an upper level already split elements located at regions that the flow structures of interest (shocks, vortices etc) have passed and there is no longer need to have fine resolution. This dynamical change of the mesh density (refinement/de-refinement) is performed at a specified number of time steps to ensure that a well-refined mesh exists for adequate capturing of flow structure that require high resolution. This dynamic AMR approach has been implemented for hexahedral and prismatic elements using the hierarchical quad- and oct-tree data structure. For time integration, the explicit, total variation diminishing (TVD), third-order accurate Runge-Kutta (TVD-RK3) method [29, 30] is used. The TVD-RK3 explicit method provides stable time integration but has severe time stability limitations (CFL $\sim 1/p^2$, where $p$ is the polynomial order). In this work, the time step limitations resulting for small size elements and higher-order approximations were offset by the parallel implementation with suitable mesh partitioning. In addition, implicit time marching schemes [31]–[36], can be employed especially for viscous flow computations.

3 UNIFIED LIMITING

The construction of the basis functions and the implementation of the DG method in the computational domain for the standard cubic element, allows straightforward numerical treatment of mixed-type meshes in a unified way. The solution coefficients $c_i(t)$ in eqn (3) are defined in the computational space and are unique for every element. A Taylor series expansion [21, 22] reveals that in the absence of discontinuities the solution coefficients are estimates of the solution derivatives. Thus, limiting of the solution coefficients amounts to limiting the solution derivatives in both $\mathbb{P}_1$ adaptive and hierarchical limiting. The monotonicity of the solution is preserved in a TVB in the mean (TVBM) sense [18], by applying a slope limiting procedure to all types of elements. For every element all limiting operations are performed in $\Omega$, over the standard configuration, where it is imposed that the slope of the solution at the element edges does not exceed the variation of the mean solution across them. Every element is checked for limiting, separately in each direction, by using a modified form of the TVB limiter proposed by Cockburn and Shu [18, 19]. The modified TVB limiter is:

$$\bar{m}(a_1,a_2,\ldots,a_n) = \begin{cases} a_1 & \text{if } |a_1| \leq ML^2_{x,y,z} + b, \\
(m(a_1,a_2,\ldots,a_n)) & \text{otherwise}. \end{cases}$$  

(4)

The parameter $M \geq 0$, which represents the second derivative of the solution [20, 18], for each characteristic field is not set to an a priori fixed value and plays the role of a discontinuity detector. It is estimated using finite difference like approximation in each direction of the computational domain. For our modified limiter, the constant $b$ is introduced, as a lower threshold in order to apply the limiter in a quite narrow band, where shocks appears. For all problems the $b$ is set $10^{-3}$. Furthermore, $L_{x,y,z}$ in eqn (4), is not considered to be the physical space element edge length ($\Delta x$) as in the original limiter, but represents the characteristic length of the element in the physical space. The function $m(a_1,a_2,\ldots,a_n)$, is the usual minmod function:

$$m(a_1,a_2,\ldots,a_n) = \begin{cases} s \min_{1 \leq j \leq n} |a_j| & \text{if } \text{sgn}(a_1) = \text{sgn}(a_2) = \ldots = \text{sgn}(a_n) = s, \\
0 & \text{otherwise}. \end{cases}$$  

(5)

and it is applied to limit the solution coefficients (solution moments), in the characteristic space. For $p$-adaptive limiting of arbitrary order of approximation $\mathbb{P}_k$, $k \geq 1$, only the $\mathbb{P}_1$ expansion of the solution can be used for elements flagged for limiting in order to preserve monotonicity. Then, if the solution at an element has been limited, the approximation order at that element is set to $P = 1$. For elements neighboring limited elements, a higher-order of approximation could be used.

3.1 Limiting for hexahedral elements

For a $\mathbb{P}_1$ approximation over a hexahedral element, the solution expansion (see eqn (3)), has the form:

$$u_h = c_0^h b_0 + c_1^h b_1 + c_2^h b_2 + c_3^h b_3 + c_4^h b_4 + c_5^h b_5 + c_6^h b_6 + c_7^h b_7.$$  

(6)
For the standard cubical element of the computational space, the basis functions \( b_j^h, j = 0, \ldots, 7 \) in eqn (6), where the superscript has been added to denote element type, hexahedron in this case, are tensor products of the Jacobi basis functions relative to the local Cartesian system \((\eta_1, \eta_2, \eta_3)\) and they are given by:

\[
\begin{align*}
    b_0^h &= 1, & b_1^h &= \eta_1, & b_2^h &= \eta_2, & b_3^h &= \eta_3, \\
    b_4^h &= \eta_1\eta_2, & b_5^h &= \eta_1\eta_3, & b_6^h &= \eta_2\eta_3, & b_7^h &= \eta_1\eta_2\eta_3.
\end{align*}
\]

(7a)

(7b)

The coefficients \( c_j^h \) are modified by the TVB limiter as shown below. It is observed that the bases \( b_4^h, b_5^h, b_6^h \), and \( b_7^h \) in eqn (6) are non-linear and include higher than first-order polynomials. It has been found in numerical experiments that for regular elements the values of these coefficients are always much smaller than the coefficients of the linear part. Therefore, the solution coefficients \( c_2^h, c_5^h, c_6^h, c_7^h \) could remain unchanged or set to zero if a regular hexahedral element is flagged for limiting. For sheared or distorted elements these coefficients, \( c_2^h, c_5^h, c_6^h, c_7^h \), could be modified with the same procedure used for the modification of the coefficients \( c_1^h \), \( c_4^h \) and \( c_3^h \) considering additional solution control points.

The solution coefficient \( c_1^h \) corresponds to the \( u_{\eta_1} \) solution derivative therefore the coefficient \( c_1^h \) is limited along the \( \eta_1 \) direction. For doing so, the \( \mathbb{P}_1 \) part of the solution at points \( A_1 \) and \( A_2 \) located the centers of the faces (see Fig. 1) and on the axis along the \( \eta_1 \) direction are considered.

According to eqn (6), at the face centers the local deviation of the solution with respect to the mean is:

\[
\begin{align*}
    U_{A_1} &= u^h(-1, 0, 0) - c_0^h = -c_1^h, \\
    U_{A_2} &= u^h(1, 0, 0) - c_0^h = c_1^h.
\end{align*}
\]

(8a)

(8b)

Limiting of the solution along the \( \eta_1 \) direction is performed by limiting the central point values \( U_{A_1} \) and \( U_{A_2} \), using the TVB limiter of eqn (4), as follows:

\[
\begin{align*}
    \tilde{U}_{A_1} &= -m(-U_{A_1}, c_{0,(i+1,j,k)}^h - c_{0,(i,j,k)}^h, c_{0,(i,j,k)}^h - c_{0,(i-1,j,k)}^h), \\
    \tilde{U}_{A_2} &= m(U_{A_2}, c_{0,(i+1,j,k)}^h - c_{0,(i,j,k)}^h, c_{0,(i,j,k)}^h - c_{0,(i-1,j,k)}^h),
\end{align*}
\]

(9a)

(9b)

where \( c_{0,(i,j,k)}^h \) denotes the mean solution for the element \((i, j, k)\) in the computational domain. Then the new limited value of \( c_1^h \) is recomputed as follows:

\[
\tilde{c}_1^h = \frac{\tilde{U}_{A_2} - \tilde{U}_{A_1}}{2}.
\]

(10)

Along the \( \eta_2 \) and \( \eta_3 \) directions the solution coefficients \( c_2^h \) and \( c_3^h \) are limited using a similar procedure as for the \( c_1^h \) coefficient. Using the same approach the limiting procedure for prismatic and tetrahedral elements is performed at the computational space.

4 NUMERICAL EXAMPLES

4.1 Convergence with AMR

The first test problem is the convection of an isentropic vortex in a three-dimensional domain discretized with prismatic elements. For this problem, the exact solution is smooth and convergence of dynamic AMR can be demonstrated. The vortex strength is \( \beta = 5 \) and the free stream conditions \( \rho = 1, p = \frac{1}{\gamma}, u = 0.5 \).

<table>
<thead>
<tr>
<th>AMR level</th>
<th>( \mathbb{P}_1-L_2 ) error</th>
<th>rate of converg.</th>
<th>( \mathbb{P}_2-L_2 ) error</th>
<th>rate of converg.</th>
<th>( \mathbb{P}_3-L_2 ) error</th>
<th>rate of converg.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.524 × 10^{-2}</td>
<td>2.07</td>
<td>1.22 × 10^{-3}</td>
<td>3.12</td>
<td>3.91 × 10^{-4}</td>
<td>4.1</td>
</tr>
<tr>
<td>2</td>
<td>3.362 × 10^{-3}</td>
<td>2.07</td>
<td>1.39 × 10^{-4}</td>
<td>3.12</td>
<td>2.27 × 10^{-5}</td>
<td>4.1</td>
</tr>
<tr>
<td>3</td>
<td>8.150 × 10^{-4}</td>
<td>2.05</td>
<td>1.66 × 10^{-5}</td>
<td>3.09</td>
<td>1.57 × 10^{-6}</td>
<td>3.98</td>
</tr>
</tbody>
</table>

Table 1: \( L_2 \) norm of the error in density and convergence rates of numerical solutions computed with dynamic AMR

A uniform coarse background mesh (with increasing element size depending on the order of expansion) with
prismatic elements is constructed. The vortex is set as initial condition and the region that the vortex occupies is refined dynamically by one or more levels as the vortex is convecting. Numerical solutions obtained with $P_1$, $P_2$, and $P_3$ expansions are expected to reach second, third, and fourth-order rate of convergence with respect to the level of dynamic mesh refinement. The density $L_2$ norm of errors and the convergence rate of density with respect to the exact solution for different levels of refinement are shown in Table I.

4.2 Shock density perturbation interaction

An important one-dimensional test case involving shocks, regions of rapid variation of the numerical solution, and smooth parts of solution is the shock-entropy interaction case of Osher and Shu used also in [21]. The initial conditions for this test problem are:

$$(\rho, u, p) = \begin{cases} 
3.857143, -0.920279, 10.3333 & \text{if } -10 \leq x \leq 0 \\
1 + 0.2\sin(5x), -3.549648, 1.0 & \text{if } 0 < x \leq 10.0530964 \\
1.0, -3.549648, 1.0 & \text{if } x > 10.0530964 
\end{cases}$$

This problem was solved first with a fine mesh using $P_1$ approximation. A uniform mesh with prismatic elements and element size $h_{P_1} = 0.005$ was used. Next $P_2$ and $P_3$ expansions with element size $h_{P_2} = 0.03$ and $h_{P_3} = 0.07$ respectively were employed. In addition, dynamic $h$-refinement was used in regions where shocks develop. The domain was filled with prismatic elements and dynamic AMR [37] was applied at the elements where shocks were detected. The computed density gradient was used as AMR criterion. It was found however that for this problem and other problems involving shocks only the limited elements could be used as AMR. Note that the element size for the numerical solutions with different order of expansions was selected so that the numerical error in the smooth parts of the flow is approximately of the same order with the $P_1$ expansion numerical solution ($0.005^2 = 0.000025$) on a finer uniform mesh. A mesh size proportional to the order of expansion is pursued therefore for $P_2$ expansions $h_{P_2} = 0.03$ ($0.03^3 = 0.000027$) and for $P_3$ expansions $h_{P_3} = 0.07$ ($0.07^4 = 0.000024$) was selected. For $P_2$ and $P_3$ expansions the refined (limited) elements have $P_1$ approximation. For the $P_2$ approximation, three levels of refinement are used so that the element size in regions with steep gradients (limited elements) is approximately equal to element size of the fine mesh used for the $P_1$ numerical solution whereas for the $P_3$ expansion four levels of $h$-refinement are used. The comparisons of the density obtained from the $P_2$ ($P_1$ at limited elements) and $P_3$ ($P_1$ at limited elements) numerical solutions with the density distribution obtained from the fine mesh numerical solution with $P_1$ expansion are shown in Figs. 2-(a) and 2-(b), respectively. It appears that the numerical solution obtained with coarser but dynamically refined meshes at the shocks and with $P_2$ and $P_3$ solutions recovers the results obtained from $P_1$ numerical solution computed with a finer mesh. The agreement is quite good both in regions of shocks and the smooth but rapidly varying flow features.
4.3 Capturing shock interactions with different element types and AMR

The flow at \( M = 3 \) in a channel with a step [38], a time dependent flow with complex shock interactions, is used to demonstrate shock capturing with different element types. For this two dimensional flow, a mesh with tetrahedral elements is inherently not oriented with the plane of the flow and requires use of three-dimensional limiting procedures. For a mixed-type mesh with hexahedral and prismatic elements, the elements were intentionally sheared so that the lateral faces are not normal to the plane of the flow. Prismatic elements were clustered in the region of the singular corner to diminish build up of entropy layer on the lower wall [38]. The prismatic mesh was interfaced with hexahedral mesh for the rest of the domain. Both meshes and the limited elements for these computations are depicted in Fig. 3-(a). Computed density fields obtained on these meshes are shown in Fig. 3-(b). In both computations, the slip line at the upper wall Mach stem is visible but roll-ups do not develop due to insufficient resolution. Furthermore, in both cases a large Mach stem is visible at the lower wall due to insufficient refinement at the singular corner.

![Figure 3: (a) The mesh and the limited elements at t=4.0 for M=3 flow in a tunnel with a step. Top mixed-type mesh with prisms and hexahedra, bottom mesh with tetrahedra. (b) Computed density at t=4.0: top mixed-type mesh with prisms and hexahedra, bottom mesh with tetrahedra.](image)

Next, a solution was computed with dynamic adaptive mesh refinement. The computation started from an initial mesh with \( h_i = 1/20 \), much coarser than the mesh used to compute the flow with hexahedral elements on a fixed mesh where the element size was \( h = 1/80 \). The initial mesh was dynamically adapted during the course of the computation. Cells were divided in regions where discontinuities have been detected while the mesh was re-refined to up a coarser level once the discontinuities moved away. The density gradient of the computed solution was used as AMR criterion. Five levels of dynamic adaptive mesh refinement were performed. The smallest cell size after five levels of dynamic AMR was \( (1/20)/2^5 = 1/640 \). The smallest cells were at the locations where shocks were detected. The adapted mesh at final time \( t = 4.0 \) and the computed density are shown in Figs. 4 and 5, respectively. In Fig. 4, the limited elements are also shown. It can be seen that limiting is confined in regions of discontinuities and steep flow gradients. The computations on fixed meshes had 23,000 hexahedral and prismatic elements and 90,000 tetrahedra and average element size \( h_{hex} = 1/80 \), \( h_{prism} = 1/160 \), and \( h_{tetra} = 1/100 \). The final adapted mesh had approximately 80,000 elements and smallest element size \( h_{hex} = 1/640 \) in regions with very large density gradients, and with a reasonable increase of the computational cost, quite better resolution was achieved. Comparing the \( P_1 \) numerical solution computed with dynamic AMR with the \( P_5 \) numerical solution [39] and subcell limiting it appears equivalent resolution of the shock interactions and fine scale structures have been achieved. For the part of the flow with the vortices instead of AMR, \( p \)-adaptivity could be applied. The result obtained from a computation with three levels of refinement and \( p \)-type adaptive refinement in the smooth parts of the flow field is shown in Fig. 6. For this computation, the solution expansion increased from \( P_1 \) to \( P_2 \) and \( P_3 \) in the region of the vortical structures. It can be seen that quite good convection of the vortices can be achieved on a sparser mesh when the order of accuracy increases in the vortical flow region. The computed vorticity magnitude was used as criterion for performing \( p \)-adaptivity dynamically. The limited elements as well as the elements where higher-order expansions were employed are marked in Fig. 6. It was shown in [13] that for vortex convection local
increase of order of expansion results in error reduction according to the order of the scheme e.g for a $P_1$-$P_2$-$P_3$ solution the rate of convergence is approximately four.

4.4 Transonic flow over the ONERA M6 wing

Transonic inviscid flow at $M = 0.84$ over the ONERA M6 wing [40] at angle of incidence $\alpha = 3^\circ$ was computed using a mesh with tetrahedral elements and $P_1$ (second-order) approximation. The computed solution reached steady-state and convergence was achieved in the $L_2$ norm of the residuals for all quantities. The computed pressure distribution on the wing surface and the symmetry plane is shown in Fig. 7-(a) for the $P_1$ numerical solution. The transonic shock structures are captured within a cell even though the mesh is not aligned with the shocks. The limited cells are superimposed in Fig. 7-(a) to demonstrate that apart from the shocks limiting is also performed in other regions where steep gradients exist, such as the leading edge region. It is observed in Fig. 7-(a) (see pressure contours on slices) that because the off–body tetrahedra grow up in size rapidly, the shock resolution away from the surface deteriorates. Next, a coarse mesh with layers of prismatic elements on the surface was constructed and a numerical solution with dynamic mesh refinement was obtained. The surface mesh provides resolution for satisfactory geometry definition of the leading edge and the tip region while the resolution in the region where shocks are expected to occur is rather coarse. The adapted surface mesh and the computed surface pressure distribution are shown in Fig. 8 and 7-(b), respectively. Two levels of AMR were sufficient to confine the shock in a narrow region of smaller cell size. The computed surface pressure and the off surface shocks structure (see Fig. 7-(b)) demonstrate that the resolution obtained with AMR is comparable to the resolution obtained with a fine hexahedral mesh. The computed surface pressure distributions are compared with the measurements in Fig. 9. The dynamically refined prismatic/tetrahedral mesh has less than half the number of cells of the fine hexahedral mesh and almost one third of the total degrees of freedom compared to the fine hexahedral mesh (noting that $P_1$ hexahedra have 8 degrees of freedom while prisms have 6 degrees of freedom for $P_1$ approximation).

5 CONCLUSIONS

A discontinuous Galerkin method suitable for three-dimensional computations on unstructured mixed-type meshes was presented. Limiting of linear terms of $P_1$ expansion is performed in order to resolve accurately the shocks and contact discontinuities. For higher-order approximations, the order of accuracy drops at the discontinuities at $P_1$ without compromising the solution accuracy. The limiter is performed to the unit cubical element of the computational space, making the limiting procedure applicable for all element types. The use of h-refinement (AMR) at regions with discontinuities and p-adaptation at regions with smooth solution offers significant increase of the solution accuracy with lower computational cost.

Figure 4: Limited elements and final adapted mesh at t=4.0 in a tunnel with a step.
Figure 5: Computed solution at $t=4.0$ with hexahedral initial mesh using dynamic AMR for $M=3$ flow in a tunnel with a step: top density, bottom entropy measure.

Figure 6: Computed density and limited elements at $t=4.0$ with combined dynamics AMR and p-type refinement in the region of roll-ups after the Mach stem; The order of expansion is indicated: $P_1$ elements (blue), $P_2$ (yellow), and $P_3$ (red).
Figure 7: (a) Surface pressure distribution over the ONERA M6 wing at $M=0.84$ and incidence $\alpha = 3^\circ$; $P_1$ numerical solution. The limited elements are superimposed, (b) Surface pressure distribution over the ONERA M6 wing obtained with two levels of AMR of the prismatic elements.

Figure 8: Two levels of refinement at the shock for the mesh with prismatic elements. (a) Upper surface of the wing, (b) Symmetry plane.

Figure 9: Comparison of the computed surface pressure coefficient distribution with the experiment for the ONERA M6 wing at $M=0.84$ and incidence $\alpha = 3^\circ$; $P_1$ numerical solution with two levels of AMR of the prismatic elements.
REFERENCES


