BULK VISCOSITY DAMPING FOR ACCELERATING CONVERGENCE OF COMPRESSIBLE VISCOUS FLOW SOLVERS

Jafar S. L. Alzaeili*, Karim Mazaheri†

*Sharif University of Technology, Aerospace Department
Tehran 11635-8639
Iran
e-mail: azeili@ae.sharif.edu
Web page: http://ae.sharif.edu/~azeili

† Sharif University of Technology, Aerospace Department
Tehran 11635-8639
Iran
e-mail: mazaheri@sharif.edu
Web page: http://ae.sharif.edu/~mazaheri

Key words: Navier-Stokes Equations, Convergence Acceleration, Bulk Viscosity Damping

Abstract. In solution of the Euler equations in the steady state conditions, the error or residual waves are blamed for the reduction of convergence rate. These waves may be damped by adding a bulk viscosity term to the momentum equations. It was done to the Euler equations previously and in this work this method has been applied to the Navier-Stokes equations. In applying this method it's important to notice that these equations have actual bulk viscosity and the added artificial bulk viscosity should keep the consistency. The efficiency of this method to accelerating the convergence was shown for some problems such as low speed cavity flows. Also it has been shown that this method is independent from other accelerating techniques.

1 INTRODUCTION

In two space dimensions, we consider the time-dependent Euler equations. In numerical solution of the Euler equations for the steady state problems and also for the Navier-Stokes equations, the errors and the residuals of the conserved variables are convected, dispersed and dissipated like a scalar quantity. Therefore, it is natural to study the behavior of these waves to develop any technique which involves them, particularly in convergence acceleration methods. The error study shows that the path to convergence follows a simple, well-defined, repetitive pattern that begins almost immediately. For the first cycles, there are minor variations, but after this each cycle repeats the previous one very faithfully. A perfect correlation is maintained between the timing of the residual waves and the beating in a conventional residual history1.
The study of mathematical behavior of Euler equations shows and is convincing evidence that acoustic waves are responsible for an important part of the convergence process. One can say that all of these results are valid in the Navier-Stokes equations. It is because of convection dominate nature of these equations which makes the study of error waves in numerical solution of the Euler equations, sufficient to extend it to the Navier-Stokes equations. An effective strategy to accelerate convergence of Euler solvers in solution of steady state problems, is to damp the residual waves while they are traversing the flow field. There are several ways to do this, like residual smoothing, classic multigrid, etc., which are the most popular convergence acceleration schemes. All of these schemes have been applied to Navier-Stokes solvers successfully. It is the common method, we first develop the new ideas to Euler solvers and then extend them to Navier-Stokes solvers, and as is prenominated above it comes from the similarity in these equations. In fact the nonlinear terms in both of them are similar and only a linear term has been added to the Navier-Stokes equations, comparing to Euler equations and this term does not change the nature of equations essentially. Here one systematic way is developed to accelerate the convergence. This idea has been developed for Euler computations by Mazaheri and Roe. The idea is base on the work which was done by Ramshaw and Mousseau for accelerating the convergence of incompressible flow calculations. They added an artificial bulk viscosity term to a method based on the artificial compressibility method introduced in 1967 by Chorin. Mazaheri and Roe to do this take a slightly non-physical approach, so that the added term vanish in the steady state. In that work they constructed an artificial bulk viscosity which vanished when required, and can be tuned to damp the error waves, based on the fact that the acoustic waves are primarily responsible for the slow convergence. Physically, the most direct way to damp this component of the residual waves is by bulk viscosity damping (BVD). Thus is natural to introduce an artificial bulk viscosity to remove these waves from the flow field.

They showed that why and how bulk viscosity affects the solutions, using one dimensional analysis. Also the efficiency of bulk viscosity damping for Euler solvers in both explicit and implicit schemes was shown.

In this paper and after the discussion of bulk viscosity damping theory, we will first apply this idea to Euler solvers on unstructured grids. The BVD term is used in energy equations and a little improvement has been made. Then we will apply the bulk viscosity damping convergence accelerating method to Navier-Stokes solvers. We should note that these equations have an actual bulk viscosity term and the adding artificial bulk viscosity must keep the consistency to have accurate solution. To solve the Navier-Stokes equations a Roe based upwind method has been used to discrete the nonlinear convection terms and a standard central-difference method has been used to viscous terms. The code which has been developed to this work uses triangular, quadrilateral and hybrid grids and we test the BVD convergence accelerating in all of them for several problems. We have used some standard flow case to study this method such as cavity flow. The internal flow has been used because in these cases the effect of viscosity is more dominated than the external cases. Also some sensitivity of the bulk viscosity damping method in applying to Navier-Stokes solvers such as grids effects will be studied in the end of this paper.
2 THEORY OF THE BULK VISCOSITY DAMPING (BVD)

The bulk viscosity is a part of the viscous stress which appears in the momentum equations. The momentum equation in both Euler equations and the Navier-Stokes equations can be written as:

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = \nabla \cdot \mathbf{T}
\]  \hspace{1cm} (1)

Here, \( \mathbf{u} \) is the velocity vector, \( \rho \) is the density and \( \mathbf{T} \) is the stress tensor. For the Euler equations the stress tensor will be as:

\[
\mathbf{T} = -p \delta_{ij}
\]  \hspace{1cm} (2)

Here, \( p \) is the static pressure. For the Navier-Stokes equation the stress tensor will be as:

\[
\mathbf{T} = -p \delta_{ij} + \tau_{ij}
\]

\[
\tau_{ij} = \lambda \nabla \cdot \mathbf{u} \delta_{ij} + 2\mu \varepsilon_{ij}
\]  \hspace{1cm} (3)

Here, \( \lambda \) and \( \mu \) are, respectively, the bulk viscosity and the shear viscosity coefficients. In both Euler equations and the Navier-Stokes equations, the body forces are neglected. The strain rate \( \varepsilon \) is defined as:

\[
\varepsilon_{ij} = \frac{1}{2} \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} \right)
\]  \hspace{1cm} (4)

The bulk viscosity term, \( \lambda \nabla \cdot \mathbf{u} \delta_{ij} \), is the part of the viscous stress which is proportional to the divergence of the velocity. For positive \( \lambda \), bulk viscosity increases the internal energy proportional to square of \( \nabla \cdot \mathbf{u} \). The most important effect of this term is that it can dissipate the acoustic waves. The acoustic waves are responsible for the convergence rate, thus the dissipation of them improves the convergence rate. In other hand, the second law of thermodynamics forces that \( \mu \) and \( \lambda + 2/3 \mu \) be non negative. We here have used Stokes theorem which considers \( \lambda \) being \(-2/3 \mu\). In the Navier-stokes equations \( \mu \) is positive and according to Stokes theorem \( \lambda \) will be negative and in this case the bulk viscosity does not help in convergence rate improvement.

For incompressible flow the velocity divergence vanishes in the steady state, and adding an artificial term proportional to it will leave the steady state unchanged. This was the approach taken in Reference [4]. For compressible flow, a similar effect can be achieved using the divergence of \( \rho \mathbf{u} \), that is to say \((-\rho \nabla \cdot \mathbf{u})\). This also makes the dissipation of error waves proportional to square of their magnitudes. Consider the mass equation:
\[ \frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u}) = 0 \]  \hspace{1cm} (5)

In the steady state and time derivatives vanish and according to the mass equation the divergence of \( \rho \mathbf{u} \) vanishes, thus we can add any term proportional to it without any change in the steady state solution.

We first consider a simple set of equations which are similar to equations of flow. These equations are linearized Euler equations (acoustic equations) in one space dimension. Because of linearity of these equations, we can analyze the effect of the added term analytically. The acoustic equations are as:

\[
\begin{align*}
\rho_t + \rho_0 u_x &= 0 \\
\rho_0 u_t + p_x &= 0 \\
p_t + \rho_0 a_0^2 u_x &= 0
\end{align*}
\]  \hspace{1cm} (6)

We can consider the added bulk viscosity term, proportional to \( \rho \mathbf{u} \), which in this case will be \( \rho_0 u_x \) or \( -\rho_0 u_x \). The coefficient of this term can be considered as \( a_0 l \) where \( a_0 \) is the reference sound speed and \( l \) is an artificial bulk viscosity coefficient with the dimension of length. This constant needs to be determined by design criteria, or stability restrictions.

After adding the new term and some simplifications, we have that:

\[
\begin{align*}
\rho_0 u_t + p_x &= a_0 l \rho_0 u_{xx} \\
p_t + \rho_0 a_0^2 u_x &= 0
\end{align*}
\]  \hspace{1cm} (7)

We have added the bulk viscosity term only in momentum equation. In the next section we will add this term to energy equation and study its effect. These equations can be written in the characteristic form and by stability analyze, we find that:

\[ \frac{l}{\Delta x} \leq \frac{1-\nu}{2\nu} \]  \hspace{1cm} (8)

Here, \( \nu \) is the Courant number and \( \Delta x \) is the cell size. Thus for a simple explicit discretization, the coefficient \( l \) cannot be large compared with the mesh size. This is natural, since we have added a parabolic term to the equations. In practice, as will be seen in the next sections, the stability margin on practical stencils is slightly wider than this estimate suggests for Euler computations.

There is different in Navier-Stokes computations, these equations have a viscosity term itself. One can write the simplified equations as:
\[ \rho_0 u_x + p_x = a_0' \rho_0 u_{xx} \]
\[ p_t + \rho_0 a_0^2 u_x = 0 \]  
\[ l' = l + \frac{\mu}{a_0 \rho_0} \]  

By stability analyze similar to inviscid equations, one can find that:
\[ l \approx \Delta x - \frac{\mu}{a_0 \rho_0} \]

Thus, in viscous flow equations, the added bulk viscosity term is smaller than one which can be used for Euler computations. However, in both Euler and Navier-Stokes computations, the practical values of artificial bulk viscosity coefficient, are greater than these estimations and are different from case to case.

3 APPLICATION TO EULER EQUATIONS

In two space dimensions, the time-dependent Euler equations are:
\[ \frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix} + \frac{\partial}{\partial x} \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uH \end{pmatrix} + \frac{\partial}{\partial y} \begin{pmatrix} \rho v \\ \rho v^2 + p \\ \rho vu \\ \rho vH \end{pmatrix} = 0 \]

Here, \( \rho \) is density, \( u \) and \( v \) are velocity components, \( p \) is pressure, \( E \) and \( H \) are total internal energy and total enthalpy, respectively. There is no viscous term in these equations. After adding the artificial bulk viscosity term, the modified Euler equations will be as:
\[ \frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix} + \frac{\partial}{\partial x} \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uH \end{pmatrix} + \frac{\partial}{\partial y} \begin{pmatrix} \rho v \\ \rho v^2 + p \\ \rho vu \\ \rho vH \end{pmatrix} = \begin{pmatrix} 0 \\ (l a_0 \rho_x) \\ (l a_0 \rho_y) \\ 0 \end{pmatrix} \]

Here, the bulk viscosity term has been added to momentum equations only, the similar term can be added to energy equation. It is useful to write these equations in non dimensional form. The non dimensional form of these equations will be as:
This presentation of equations shows that the BVD term in energy equation is proportional to inverse of square of Mach number, thus is negligible except in very low Mach number flows. The BVD terms in momentum equations have a more important effect because they are proportional to inverse of Mach number and these terms have a significant effect in low Mach number flows. It is important to note that the BVD terms in both momentum and energy equations have been considered as source terms not as a part of stress tensor. The advantage of this consideration is to increase the stiffness in the equations. The BVD terms generate stiffness in equations because of their parabolic nature and their inclusion as source terms in the equations. The source terms in the equations are computed using a larger spectrum than the flux terms such as stress tensor. It is the reason of increasing the stiffness in the equations system.

We can use any popular upwind of central differencing method to calculate the fluxes passing the cells interfaces. In this work we have used Roe's upwind method which has a high performance in compressible flows. This method has low performance in low Mach number flows and the convergence rate of this method is very low. Using BVD method improve the performance of Roe's method in low mach number flows and make it an all-speed method which can solves a wide range of flows from very low Mach number flows to hypersonic flows. In this work we have applied the BVD method on triangular, quadrilateral and hybrid unstructured grids. The details of the numerical calculations for a structured grid have been given in Reference [1]. There is some difference in the numerical calculations for the unstructured grids. We have used an explicit solver to approach to steady state solution. The updating process for the mass equation is:

\[ (\rho)_{j}^{n+1} = (\rho)_{j}^{n} - \frac{\Delta t}{A} \sum_{k=1}^{n\text{faces}} F_{ik}^{n} \]  

Here, \( F_{i} \) is the flux which is crossing the cells interface and it has been computed using Roe's upwind method. The time step has been calculated using local time step method to accelerate the convergence. The updating process for momentum equations is as:
\[(\rho u)^{n+1}_j = (\rho u)^n_j - \frac{\Delta t}{A} \sum_{k=1}^{n_{faces}} F^u_{ik} \Delta s - \frac{\Delta t l}{M x A} \sum_{k=1}^{n_{faces}} (\rho_1) \Delta y \]

\[(\rho v)^{n+1}_j = (\rho v)^n_j - \frac{\Delta t}{A} \sum_{k=1}^{n_{faces}} F^v_{ik} \Delta s + \frac{\Delta t l}{M x A} \sum_{k=1}^{n_{faces}} (\rho_1) \Delta x \]

In these equations, \(\rho_1\) can be computed using equation (14).

\[\rho_1 = -\frac{1}{n_{faces}} \sum_{k=1}^{n_{faces}} F^u_{ik} \]

To ensure stability, analogy with one-dimensional case suggests that for explicit calculation, the value of \(l\) should be proportional to a local length scale, say, the average side length of the cell concerned. In the results section we will test different value for this coefficient for different cases.

4 APPLICATION TO THE NAVIER-STOKES EQUATIONS

In two space dimensions, the time-dependent the Navier-Stokes equations are:

\[
\begin{aligned}
\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \\ \rho u H \\ \rho v H \end{pmatrix} + \frac{\partial}{\partial x} \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ \rho E + \rho u H \\ \rho v H \end{pmatrix} + \frac{\partial}{\partial y} \begin{pmatrix} \rho v \\ \rho u v \\ \rho v^2 + p \\ \rho E + \rho v H \\ \rho v H \end{pmatrix} = & \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ h_x \\ \tau_{yy} \\ h_y \end{pmatrix} = \begin{pmatrix} \tau_{xx} \\ \tau_{xy} \\ \tau_{yy} \\ h_x \\ h_y \end{pmatrix}
\end{aligned}
\]

\[h_x = \tau_{xx} u + \tau_{xy} v + q_x \quad q_x = -k \frac{\partial T}{\partial x} \]

\[h_y = \tau_{yx} u + \tau_{yy} v + q_y \quad q_y = -k \frac{\partial T}{\partial y} \]

Here, \(T\) is the static temperature and \(k\) is the conduction coefficient. The relation between thermodynamics variables is determined using the ideal gas relations. After adding the artificial bulk viscosity term, the modified the Navier-Stokes equations will be as:

\[
\begin{aligned}
\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \\ \rho u H \\ \rho v H \end{pmatrix} + \frac{\partial}{\partial x} \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ \rho E + \rho u H \\ \rho v H \end{pmatrix} + \frac{\partial}{\partial y} \begin{pmatrix} \rho v \\ \rho u v \\ \rho v^2 + p \\ \rho E + \rho v H \\ \rho v H \end{pmatrix} = & \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ h_x \\ \tau_{yy} \\ h_y \end{pmatrix} + \begin{pmatrix} 0 \\ -la_0 (\rho_1) \cdot \tau_{xx} \\ -la_0 (\rho_1) \cdot \tau_{xy} \\ l a_0^2 [(-\rho_1) \cdot \tau_{xx} + (-\rho_1) \cdot \tau_{xy}] \end{pmatrix}
\end{aligned}
\]
The bulk viscosity terms are determined similar to the scheme used in Euler equations. However an important difference exists here; there is a bulk viscosity term in viscous stress tensor in the momentum equations. Here we should control the artificial bulk viscosity term to keep the consistency. The actual bulk viscosity coefficient is negative, thus as mentioned above an artificial term must be added to keep total bulk viscosity coefficient positive; positive bulk viscosity coefficient will accelerate the convergence rate. The actual bulk viscosity term is proportional to the divergence of and the artificial bulk viscosity term which has been used for compressible computations is proportional to . After some algebra one can find that:

\[ \tau_{BV} = \lambda \nabla \cdot \mathbf{u} \]
\[ \tau_{BVD} = l a_0 \nabla \cdot (\rho \mathbf{u}) \]
\[ \lambda_{BVD} = \rho l a_0 \left(1 + \frac{\mathbf{u} \cdot \nabla \rho}{\rho \nabla \cdot \mathbf{u}}\right) \]  

(19)

Here, \( \tau_{BV} \) is the actual bulk viscosity, \( \tau_{BVD} \) is the artificial one and \( \lambda_{BVD} \) is the artificial bulk viscosity coefficient which is derived comparing to the actual coefficient. For any cell computations, the summation of \( \lambda + \lambda_{BVD} \) must be positive to make the artificial term useful. \( \lambda_{BVD} \) is positive if \( \nabla \cdot (\rho \mathbf{u}) \) is positive, or according to the mass continuity equation, if \( \rho_1 \) is negative, thus in the updating process for any cell, the artificial term is used only if \( \rho_1 \) is negative in that cell. Further, \( \lambda \) is negative and to keep the total bulk viscosity positive, \( \lambda_{BVD} \) must be greater than \( |\lambda| \). We can control the amount of \( \lambda_{BVD} \) only by the length parameter \( l \). The constraint for \( l \) is keeping the stability and as it was mentioned in the theory of the BVD method, this length parameter must be in order of the cell size. For Navier-Stokes computations this value is less than the cell size and it is different from case to case. The cell size for the unstructured meshes is considered as the average of edges length.

5 RESULTS

In this section, results of using the BVD method are presented, for the Euler and Navier-Stokes computations. For Navier-Stokes computations, triangular, quadrilateral and structured meshes are used to study the grid sensitivity of the method. The standard finite volume time marching method has been used for solving the equations to the steady state solution. For the discretization of inviscid fluxes the most popular upwind Roe's method has been used and the viscous fluxes have been discretized using the standard central differencing method. In all computations the residual is considered in \( L_2 \) norm. The grids have been generated using standard frontal approach of Delaunay triangulation\textsuperscript{7}. The solutions are compared with standard benchmarks\textsuperscript{2,7,8}.
5.1 Euler Computations

For the Euler computations, we use the standard NACA0012 airfoil problem. The BVD technique is applied both in low and high Mach number. Here we study the effect of the Mach number when BVD term is added to the equations system.

First, we consider a flow with $M_\infty = 0.3$ and no angle of attack over a NACA0012 airfoil. Figure 1 shows the residual history of the solutions, for several value of BVD term. The value of BVD term is controlled by the BVD Factor $f$, which is the ratio of $l$ to the mesh size. Here we consider a constant value for BVD factor for all cells. Figure 1 shows the effect of BVD term to accelerate the convergence. For this problem as it has been shown in Reference [1] that the optimum value for the BVD factor is about 1.2. If this factor is considered to be greater than 1.2, it may cause instability in the solution.

To study the effect of Mach number, a high Mach number flow with $M_\infty = 1.2$ and with no angle of attack over NACA0012 airfoil is solved with adding BVD term. Also a low Mach number flow with $M_\infty = 0.01$ and also with no angle of attack is considered. This flow cannot be solved easily using standard compressible flow solvers, but when BVD term is added to the equations system, the solution is converged very fast. In fact most flow solvers have a very slow convergence in this region, so that many people believe they could not be used for low Mach number flows. In other words, we can develop an all-speed scheme using this method. The BVD method has no significant effect in high Mach number flows.

![Figure 1: Residual history for flow over NACA0012 airfoil with $M_\infty = 0.3$ for several value of BVD factor](image)
Equation (13) shows that the BVD term in the momentum equations in non-dimensional form are proportional to inverse of Mach number and for high Mach number flows the BVD term has no significant value and so cannot affect the convergence rate. Figure 2 shows the residual history for the flow computations over NACA0012 airfoil for $M_\infty = 1.2$ (left) and for $M_\infty = 0.01$ (right) in which a triangular mesh is used. Adding the BVD term in the supersonic case has no significant effect in the convergence rate and the improvement is negligible, but for low Mach number flow the BVD method can improve the convergence rate about 65%, which is considered as a significant improvement.

![Figure 2: Residual history for flow over NACA0012 airfoil with $M_\infty = 1.2$ and $M_\infty = 0.01$](image)

5.2 Navier-Stokes Computations

Here, the standard cavity problem is used to study the effect of BVD term in the steady state solution of the Navier-Stokes equations. Figure 3 shows three type of grids which are used in the cavity problem. Both unstructured triangular and quadrilateral meshes and rectangular structured mesh are used. We first consider a certain value for the BVD factor and test the BVD method for the all of three meshes. The cavity problem for $M_\infty = 0.3$ and $Re_\infty = 100$ has been solved over the meshes, with and without the BVD term. Figure 4 shows the residual history in all cases. An acceptable value for the BVD factor may be 1.0. According to equation (10) the BVD factor in the viscous problems is less than inviscid problems. For the Euler computations it has been shown that for a wide range of problems the optimum value for the BVD factor is about 1.2, but for Navier-Stokes computations, the experiences show that the value of 1.0 is an optimum value and greater values may cause instability in the solution. In all cases the BVD term has improved the rate of convergence about 60%. It shows that the BVD method is grid independent and work well for all type of grids which have the minimum quality to solve the Navier-Stokes equations. To show the
positive effect of the BVD term in accelerating the convergence, we use two different values for the BVD factor equal to 1.0 and 0.5. Figure 5 shows the residual history. When the value of 1.0 is used the solution is accelerated about 60%, but for value of 0.5, the solution convergence is improved about 40%.

The amount of the extra computations which is needed to apply the BVD method is about 1% of the total computations for the Euler equations and it is slightly more than this amount for the Navier-Stokes equations and it is negligible in both computations. Further, when the BVD term is used in energy equation for Navier-Stokes computations, the rate of convergence is improved about 2% and it covers the extra computations to apply BVD method.

In Reference [1] it was shown that for Euler computations the BVD acceleration method is independent of other accelerating methods such as multigrid or residual smoothing method. In the other hand, the popular methods of convergence acceleration are useful for the Navier-Stokes computations as well. In this paper it is shown that the BVD method works for Navier-Stokes computations as well as Euler computations. Thus, we can conclude that the BVD accelerating methods affect the convergence rate independently from other accelerating methods and it can be shown easily.

It is important to notice that in the all computations we have used the local time step technique which it accelerates the convergence rate too. When the BVD technique is used, we keep the same time step which has been determined using local time step technique. For each cell in the domain the time step is determined as:

\[
\Delta t = CFL \min(\Delta t_{\text{inv}}, \Delta t_{\text{vis}})
\]

\[
\Delta t_{\text{vis}} = \frac{\rho h^2}{4 \mu}, \quad \Delta t_{\text{inv}} = \frac{0.5 A}{\sum_{k=1}^{n_{\text{faces}}} (\hat{c}_k + |\hat{q}_k|) \Delta s_k}
\]

Here, \(h\) is the cell size, \(A\) is the area of the cell, \(c\) is the sound speed, \(q\) is the normal velocity and \(\Delta s\) is the length of each cell's edges. The hat terms have been calculated using Roe's averaging. For Euler computations \(\Delta t_{\text{vis}}\) is considered to have a large value.

Figure 3: Grids which are used in cavity problem, Unstructured Triangular mesh, Unstructured Quadrilateral mesh, Structured Rectangular mesh
6 CONCLUSIONS

- Bulk viscosity damping method has been introduced and briefly analyzed. This method has been applied to both Euler and Navier-Stokes computations.
- The best value of BVD factor has been found to be close to the mesh size. For Navier-Stokes computations, this value must be smaller than Euler computations.
- With an explicit scheme which has been used in this paper, we have found an
improvement in the convergence rate between 40% and 80% for Euler computations and about 60% for Navier-Stokes computations.

- The amount of extra computations which are necessary to apply the BVD method is negligible and is less than 1%.

REFERENCES