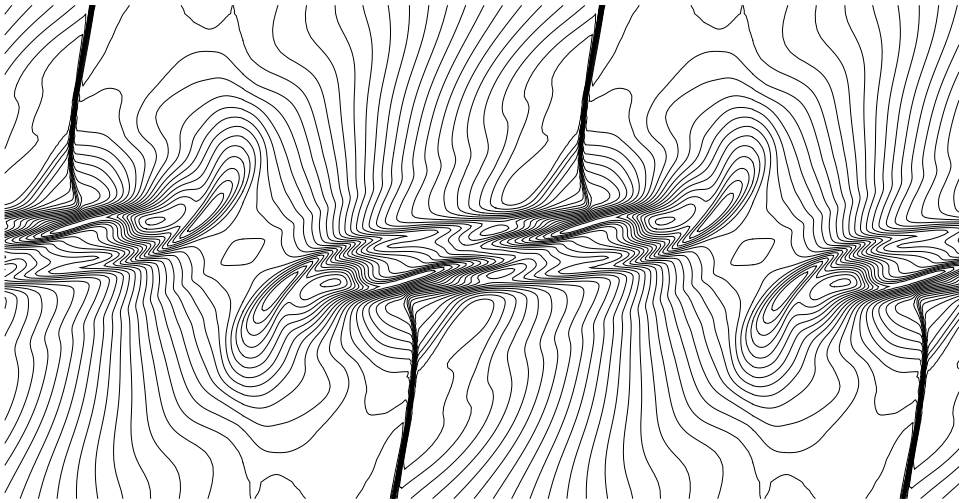
AIAA

# Implicit Space-Time Residual Distribution Method for Unsteady Viscous Flow

Jiří Dobeš[1,2], Mario Ricchiuto[1], Árpád Csík[1] and Herman Deconinck[1,]

[1]*von Karman Institute for Fluid Dynamics, Sint-Genesius-Rode, Belgium*

[2]*Czech Technical University, Dept. of Technical Mathematics, Prague, Czech Republic*



**16th AIAA Computational Fluid Dynamics Conference**

**June 23–26, 2002/Orlando, Florida**

# Implicit Space-Time Residual Distribution Method for Unsteady Viscous Flow

Jiří Dobeš[1,2]*, Mario Ricchiuto[1]†, Árpád Csík[1]‡ and Herman Deconinck[1,§]
[1] *von Karman Institute for Fluid Dynamics, Sint-Genesius-Rode, Belgium*
[2] *Czech Technical University, Dept. of Technical Mathematics, Prague, Czech Republic*

**Multidimensional upwind residual distribution schemes have been developed over the last years as a monotonicity preserving spatial discretization method for hyperbolic conservation laws on unstructured grids composed of triangles or tetrahedra. More recently a space-time formulation has been proposed by several authors to perform time accurate unsteady inviscid simulations. In the present paper we extend the use of the space-time schemes to viscous laminar computations. The viscous terms are introduced in the discretization through a Petrov-Galerkin approach consistent with the underlying residual distribution method. Grid convergence studies on scalar models are presented showing second order of accuracy of the proposed approach. The method is then applied to the laminar Navier-Stokes equations. Promising results are obtained. We also investigate explicit and implicit iterative techniques for the solution of the implicit system of equations obtained with the proposed algorithms. For viscous problems, the implicit approach shows a dramatic speed-up of the computation, however for inviscid high speed flows explicit iterations can be competitive.**

## Introduction

THE Multidimensional Upwind Residual Distribution Schemes (RDS) have emerged in the last years as an appealing alternative to Finite Volume (FVM) and Finite Element (FEM) methods. Thanks to their inherent dissipative character and to a residual property, positive second order schemes can be designed on compact stencil of an arbitrary unstructured grid without any tuning of the numerical dissipation. Moreover, the multidimensional nature of the upwinding procedure dramatically reduces the dependence of the results on the mesh quality.[1,2]

Unfortunately, unlike in FVM, for RDS spatial and temporal discretization cannot be decoupled. As in FEM the coupling arises from a non-diagonal mass matrix. For steady computations the mass matrix can be lumped, without affecting the accuracy of the method. For unsteady problems one has to take into account the full mass matrix. However, being this matrix non-positive definite, the resulting scheme is non-positive. In the past a Flux Corrected Transport technique has been tried to solve the monotonicity problem.[3,4] Unfortunately, this approach has shown a lack of robustness and it is extremely unsatisfactory from the theoretical point of view. Recently,[5–7] the space-time approach has been introduced, which has shown promising theoretical properties and numerical results.

The main idea of the space-time approach is to reformulate the schemes including time in the set of independent variables and then apply the numerical tools developed for steady state computations to a space-time residual. By doing this an implicit system of equations is obtained at each time step. In the present work, as in[5] this implicit system has been considered as a steady problem in a pseudo time variable $\tau$ in $d+1$ dimensions (being $d$ the number of space dimensions and time the additional one) and solved within each time-step using standard Residual Distribution Schemes in space-time. The positivity of the schemes transforms into linear stability. Moreover, if the space-time mesh obeys to certain geometrical constraints, the multidimensional upwinding allows to naturally decouple the solution of the past time level from the future one. In particular, by solving for two time levels at once, the time step relative to the last space-time slice can be arbitrary large, resulting in a linearly unconditionally stable time marching procedure.[5]

In[5,7,8] results have been shown for scalar models and for the Euler equations. Here we will follow more closely the work done in[5,8] extending it to viscous computations. This is achieved through a consistent Petrov-Galerkin treatment of all terms (time derivatives, convective terms and dissipative terms). For linear problems this leads to a standard RDS discretization for the inviscid part and to a Galerkin method for the viscous terms. It turns out, that the time levels are not fully decoupled, but some coupling through the space-time mesh geometry exists.

Convergence studies are presented for scalar Advection-Diffusion and viscous Burgers equations,

*Ph.D. candidate
†Ph.D. candidate
‡Ph.D. candidate
§Professor, AIAA member

space-time. The method is then applied to the laminar Navier-Stokes equations.

The paper is organized as follows: the Space-Time RDS method of[5] is shortly recalled, to explain its main ideas and how the decoupling of time levels is achieved. Then the discretization of the viscous part is explained and convergence results are presented. Finally results for the Navier-Stokes equations are shown. Steady-state in pseudo-time is reached by conventional time-integration. A comparison between explicit and implicit pseudo-time integration techniques is also performed, with particular emphasis on the viscous case.

## Space-Time RDS for Scalar Conservation Laws

Consider a scalar conservation law in quasi-linear form

$$\frac{\partial u}{\partial t} + \lambda(u) \cdot \nabla u = 0 \tag{1}$$

to be solved on a space-time domain $\Omega_{st} = \Omega_s \times [0, t_{max}]$. Following[5,8] we rewrite last equation in the following space-time notation

$$\lambda_{st}(u) \cdot \nabla_{st} u = 0, \tag{2}$$

where $\lambda_{st} = (\lambda, 1)$ and $\nabla_{st} = (\nabla, \partial/\partial t)$.

Given an initial solution $u_0(x, y, 0)$ at time $t = 0$, we will compute the unknown quantity $u_1(x, y, t_1)$ at time $t_1 > 0$ by solving the steady equation (2) on the space-time slice $\Omega_s \times [0, t_1]$ by means of standard RDS.

In particular, given an initial discretization of the space domain $\Omega_s$ composed of linear finite elements (space intervals in 1D, triangles in 2D), we build a space-time grid on $\Omega_s \times [0, t_1]$ composed of linear finite elements in space-time (triangles in 1D space and tetrahedra in 2D space). For any space-time element $T$, we define the element residual as

$$\phi^T = \int_T \lambda_{st}(u) \cdot \nabla_{st} u \, dT$$

The nodal residuals are then computed as

$$\phi_i^T = \beta_i^T \phi^T.$$

with $\beta_i^T$ a distribution coefficient. The steady solution of (2) is reached by marching in a pseudo-time variable $\tau$, *i.e.*

$$S_i \frac{\mathrm{d}u_i}{\mathrm{d}\tau} + \sum_{T \in \Omega_i} \beta_i^T \phi^T = 0 \tag{3}$$

Positivity and/or accuracy of the discretization are the requirements one takes into account for the design of the distribution coefficient $\beta_i^T$. Details on the general properties of RDS can be found in the references given in the introduction. Here we will focus our attention on the so-called Multidimensional Upwinding ($\mathcal{MU}$) property.

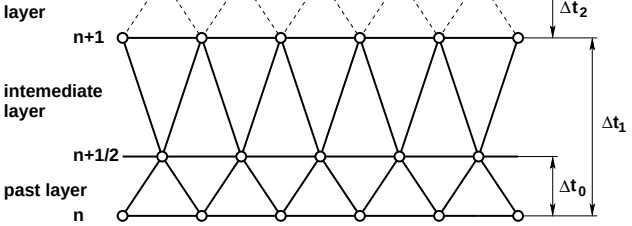



**Fig. 1 Space-time mesh for 1D case**

For a node $i$ in the element $T$, we define the nodal normal $\mathbf{n}_i$ as the inward pointing vector normal to the face of $T$ scaled by the length (surface in a tetrahedron) of the face. We then define the upwind parameter $k_i = \lambda_{st}(u) \cdot \mathbf{n}_i / D$, where $D$ is the number of independent variables ($D = d + 1$ in space-time if $d$ is the number of space dimensions). It can be easily checked that $k_i < 0$ if $i$ is an upwind node of $T$, while $k_i > 0$ if $i$ is a downwind node. Hence, $\mathcal{MU}$ is simply obtained by requiring $\beta_i^T = 0$ if $k_i < 0$.

As explained extensively in,[5,8] the $\mathcal{MU}$ allows to build space-time grids which guarantee a decoupling of the past solution from the future one so that a time marching procedure can be achieved. To understand how this is done in practice, consider the lower row of space-time triangles in figure 1. For the 1D case of equation (1) with constant speed $\lambda$ one can easily check that $k_i < 0$ for all the nodes at time level $n$ provided that the following condition holds:[5,8]

$$\Delta t_0 \leq \frac{\Delta x}{2|\lambda|}. \tag{4}$$

If the space-time mesh respects constraint (4), the solution at time level $n$ will be preserved allowing a true time marching scheme. However, a strict restriction on the magnitude of the time-step is an undesirable feature of an implicit method like the space-time RD. For this reason in[5,8] a second layer of space-time elements of arbitrary time width has been added which allows to increase the effective time-step of the computation $\Delta t$ still retaining the decoupling of the solution at time $t^n$ from the one at $t^{n+1}$. Once this is done, the use of a positive RD scheme translates into unconditional stability of the time-marching procedure for arbitrary time-steps.

In this paper, we use the following multidimensional upwind RDS schemes: the linear positive N scheme, the linear second order LDA scheme and the non-linear B (blended) scheme. Details can be found in the references.[1,2]

## Space-Time RDS for Scalar Viscous Problems

Provided that the distribution coefficients $\beta_i^T$ in equations (3) are bounded, it can be shown that RDS are equivalent to a Petrov-Galerkin discretization with

$$w_i^{PG} = \varphi_i + \sum_{T\in\Omega_i} \Pi^T \alpha_T, \qquad \alpha_T = \beta_i^T - \frac{1}{d+1}, \quad (5)$$

being $\Pi^T$ the characteristic function of element $T$. This FEM interpretation of the residual distribution method allow to easily include in a consistent way dissipative terms in the discretization.

In particular, consider the scalar problem

$$\frac{\partial u}{\partial t} + \lambda(u)\cdot\nabla u = \nabla\cdot(\mu\nabla u) \quad (6)$$

where $\mu$ is a diffusivity coefficient. We want to solve equation (6) on a space-time domain $\Omega_{st} = \Omega_s \times [0, t_{max}]$. The equation can be written in space-time notation as

$$\lambda_{st}(u)\cdot\nabla_{st} u = \nabla_{st}\cdot(\mu\nabla u, 0). \quad (7)$$

Note the zero in vector on the RHS corresponding to the absence of physical time dissipation $\mu\partial^2 u/\partial t^2$.

We add a pseudo-time dependency and apply the space-time Petrov-Galerkin discretization to

$$\frac{\partial u}{\partial \tau} + \lambda_{st}(u)\cdot\nabla_{st} u = \nabla_{st}\cdot(\mu\nabla u, 0).$$

The discretization procedure consists of the following steps:

- The term containing the pseudo-time derivative is multiplied by a mass matrix. The mass matrix is lumped, as it is usual for steady state computations.
- The discretization of the convective terms reduces to standard space-time RDS by construction of the Petrov-Galerkin test function.
- The discretization of the viscous term on the RHS reduces to a central Galerkin discretization in the linear case. We linearize locally the viscosity coefficient in the non-linear case and then we use the same Galerkin discretization.

After some algebra one can obtain the following semi-discrete equation for node $i$

$$S_i \frac{\mathrm{d}u_i}{\mathrm{d}\tau} = -\sum_{T\in\Omega_i} \beta_i^T \phi^T - \sum_{T\in\Omega_i} \mu \sum_{j\in T} \frac{\mathbf{n}_j^S\cdot\mathbf{n}_i^S}{(d+1)^2 S_T} u_j, \quad (8)$$

where $\mathbf{n}_j^S$ are *spatial components* of normal $\mathbf{n}_j$. For residual distribution schemes for which the distribution coefficients can be unbounded (e.g. the N scheme), we replace in equation (8) $\beta_i^T\phi^T$ by $\phi_i^T$ which is always well defined.

With reference to figure 1, we observe that there is no contribution to the viscous residual of nodes at time level $n+1/2$ from nodes at different time levels.

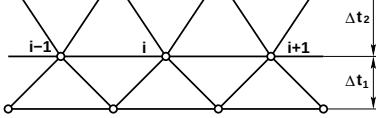



**Fig. 2 Grid stencil of viscous residual for 1D problem**

The same holds for nodes at level $n+1$. This is due to the absence of the time dissipation term $\mu\partial^2 u/\partial t^2$ and it allows to keep the decoupling of the solution guaranteed by the mesh geometry and by the $\mathcal{MU}$ for the inviscid residual.

Unfortunately, the spatial components of the normals are scaled by the time-step of the corresponding time layer. This leads to the fact that the viscous residual for a node depends on time step $\Delta t$ of *both surrounding layers.*

Consider for example the 1D linear advection-diffusion equation. Consider three neighbor nodes $i-1$, $i$ and $i+1$ at time level $n+1/2$ surrounded by elements with time-step $\Delta t_1$ and $\Delta t_2$ respectively (see fig. 2). The contribution of the viscous term to the residual for node $i$ is given by

$$\mu\frac{\Delta t_1 + \Delta t_2}{2\Delta x}(u_{i-1} - 2u_i + u_{i+1}). \quad (9)$$

The knowledge of $\Delta t_1$ and $\Delta t_2$ is needed to discretize the equations.

As a consequence, the time-layers are not fully decoupled, but there is some coupling through the space-time mesh geometry. In particular, for nodes in the last time level $(n+1)$ the knowledge of the timestep $\Delta t_2$ (see figure 1) is needed. Fortunately, we know the grid geometry. In particular, in the linear case we know a priori the next time-step $\Delta t_2$, while for non-linear problems we can compute iteratively $\Delta t_2$ during the pseudo-time iterations. Note also that, because of the linearity of the method, this dependence on the mesh geometry is linear, even for nonlinear problems.

Second order of accuracy is expected in space and time if a residual distribution scheme with bounded distribution coefficients is used. In this case, in fact, a linear consistent P-G discretization is used in space-time.

With reference to figure 1, we define the mesh stretch ratio $Q$ as the ratio of time-steps of intermediate layer to time-step of past layer:

$$Q = \frac{\Delta t_1 - \Delta t_0}{\Delta t_0}. \quad (10)$$

We can achieve arbitrarily large time-steps by changing this value according to our needs, retaining the unconditional stability of the time marching procedure if a positive Residual Distribution Scheme is used.

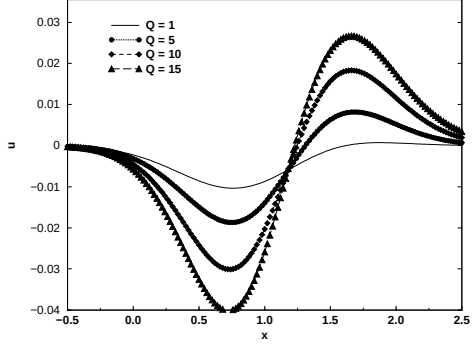



**Fig. 3 1D advection-diffusion equation, N scheme, $u_{exact} - u_{numeric}$**

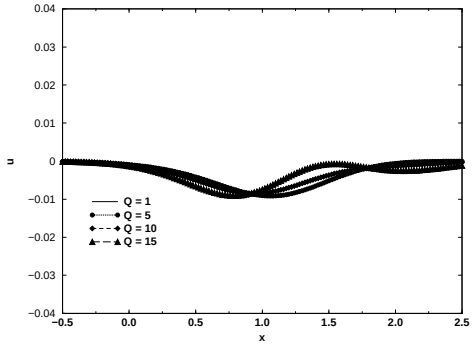



**Fig. 4 1D advection-diffusion equation, LDA scheme, $u_{exact} - u_{numeric}$**

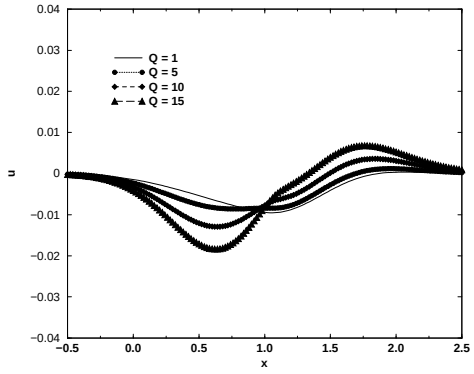



**Fig. 5 1D advection-diffusion equation, B scheme, $u_{exact} - u_{numeric}$**

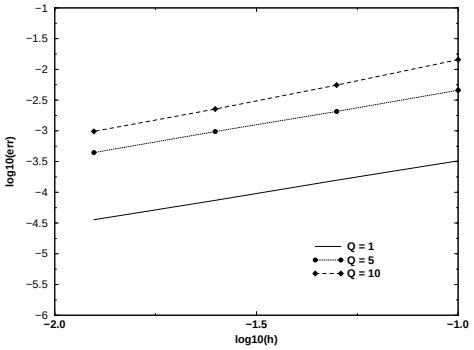



**Fig. 6 1D advection-diffusion equation, N scheme, convergence study**

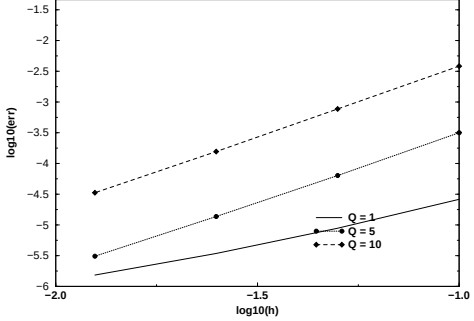



**Fig. 7 1D advection-diffusion equation, LDA scheme, convergence study**

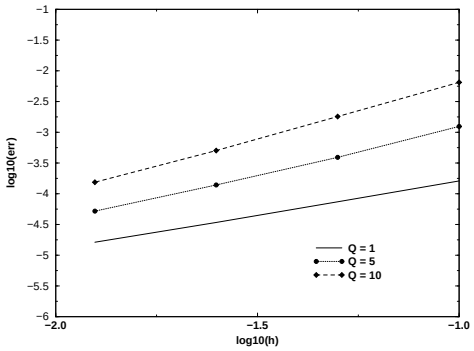



**Fig. 8 1D advection-diffusion equation, B scheme, convergence study**

| Q | N | LDA | B |
|---|---|---|---|
| 1 | 1.06 | 1.36 | 1.10 |
| 2.5 | 1.14 | 1.61 | 1.36 |
| 5 | 1.12 | 2.22 | 1.52 |
| 7.5 | 1.22 | 2.24 | 1.71 |
| 10 | 1.29 | 2.28 | 1.80 |
| 12.5 | 1.19 | 2.14 | 1.69 |
| 15 | 1.25 | 2.17 | 1.73 |

**Table 1 Convergence ratios for 1D advection-diffusion equation**

## Scalar Viscous Problem – Numerical Results

As a first test case, the 1D linear advection-diffusion of a step was chosen. The equation

$$u_t + au_x = \mu u_{xx} \tag{11}$$

has been solved on a domain $\Omega_{st} = [-5, 5] \times [0, 1]$, with initial conditions $u_0(x) = 1$ for $x < 0$ and $u_0(x) = 0$ for $x > 0$. In the computations we took $a = 1$ and $\mu = 0.1$. This problem admits the analytical solution

$$u(t, x) = \frac{1}{2}\mathrm{erf}\left(\frac{x - at}{\sqrt{4\mu t}}\right). \tag{12}$$

In figures 3, 4, 5 the error $(u_{exact} - u_{numeric})(x)$ is plotted for various values of the parameter $Q$ and for all the schemes used in this paper. As expected, the LDA and B scheme show a much smaller error than

| | | | |
|---|---|---|---|
| 1 | 0.96 | 1.80 | 1.18 |
| 2.5 | 0.97 | 1.63 | 1.30 |
| 5 | 0.93 | 2.15 | 1.33 |
| 7.5 | 0.92 | 1.89 | 1.29 |
| 10 | 0.92 | 1.73 | 1.26 |
| 12.5 | 0.88 | 1.59 | 1.18 |
| 15 | 0.87 | 1.51 | 1.14 |

**Table 2 Convergence ratios for 1D Viscous Burgers equation**

the N scheme. To compute the order of accuracy of the discretization, a convergence study has been performed. In this case we took the larger domain $\Omega_{st} = [-10, 10] \times [0, 1]$ and we set as initial conditions the exact solution at $t = 10$. The error has been computed as

$$\epsilon = \|u - u_{exact}\|_{L_2} = \iint (u_{num} - u_{exact})^2 \, \mathrm{d}x \, \mathrm{d}t \approx \approx \sqrt{\frac{\sum (u(x_i, t_j) - u_{exact}(x_i, t_j))^2}{\#nodes}}. \tag{13}$$

The resulting grid convergence hystories are plotted in figures 6, 7, 8 as a function of $Q$ for the N, LDA and B scheme respectively. The computed orders of accuracy are summarized in table 1.

We can see that the LDA scheme gives the expected second order of accuracy. Also the nonlinear B scheme gives almost second order of accuracy.

A similar test was performed on the 1D viscous Burgers equation. The initial solution chosen is that of a steepening profile. Convergence ratios are presented in table 2. The lower order of accuracy observed for the B scheme is caused by the very steep gradient forming in the solution.

## Space-Time RDS for Navier-Stokes Equations

The space-time formulation of system residual distribution schemes is a straightforward extension of the space-time formulation of the scalar schemes.[5,8]

Consider the well known conservative formulation of Navier-Stokes equations

$$\frac{\partial \mathbf{w}}{\partial t} + \nabla(\mathbf{f}, \mathbf{g}) = \nabla(\mathbf{r}, \mathbf{s}). \tag{14}$$

The system is closed by constitutive relations and equation of state. One can rewrite this system in space-time form and add a pseudo-time dependence

$$\frac{\partial \mathbf{w}}{\partial \tau} + \nabla_{st}(\mathbf{w}, \mathbf{f}, \mathbf{g}) = \nabla_{st}(\mathbf{0}, \mathbf{r}, \mathbf{s}). \tag{15}$$

The discretization can easily be obtained through the consistent Petrov-Galerkin approach described in the previous section. Roe's linearization[9] is used wherever the Jacobians are needed.

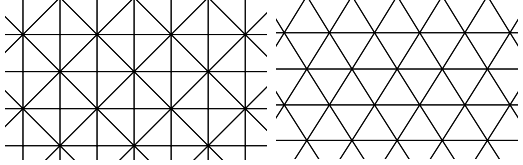


**Fig. 9 Diamond and isotropic mesh pattern**

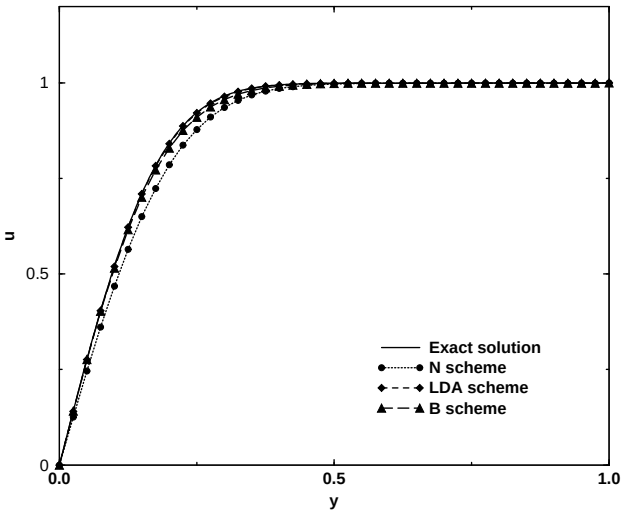



**Fig. 10 First Stokes problem**

## Navier-Stokes Equations – Numerical Results

The first test case we present is the so-called First Stokes problem.[10] Consider an infinite plate immerged in a fluid at rest. At $t = 0$ the plate is instantiniously set into motion with a velocity $u_\infty$ in its plane. The exact solution of the problem is

$$u = -u_\infty \mathrm{erf}\left(\frac{y}{2\sqrt{\nu t}}\right), \tag{16}$$

where $\nu$ is the kinematic viscosity of the fluid (assumed to be constant). The problem has been solved on a unit square discretized by $40 \times 40$ mesh with a diamond pattern (see zoom in fig. 9). The space-time mesh stretch ratio $Q$ was set to 12. In figure 10 the solutions obtained with the N, LDA and B scheme are shown. One can see a very good agreement with the analytical solution.

We consider then a transonic vortex pairing problem in a mixing layer.[11] The problem consist of a shear layer defined by two free streams with velocity profiles $u = \tanh(2y)/2$. To these velocity profiles we superimpose the velocity perturbations

$$v' = \sum_{k=1}^{2} a_k \cos(2\pi k x / L_x + \phi_k) \exp(-y^2/b) \tag{17}$$

with parameters $a_1 = 0.01$, $a_2 = 0.05$, $\phi_1 = \phi_2 = \pi/2$, $b = 10$. The problem has been solved inside a rectangular area $L_x \times L_y$ with $L_x = 30$ and $L_y = 100$.

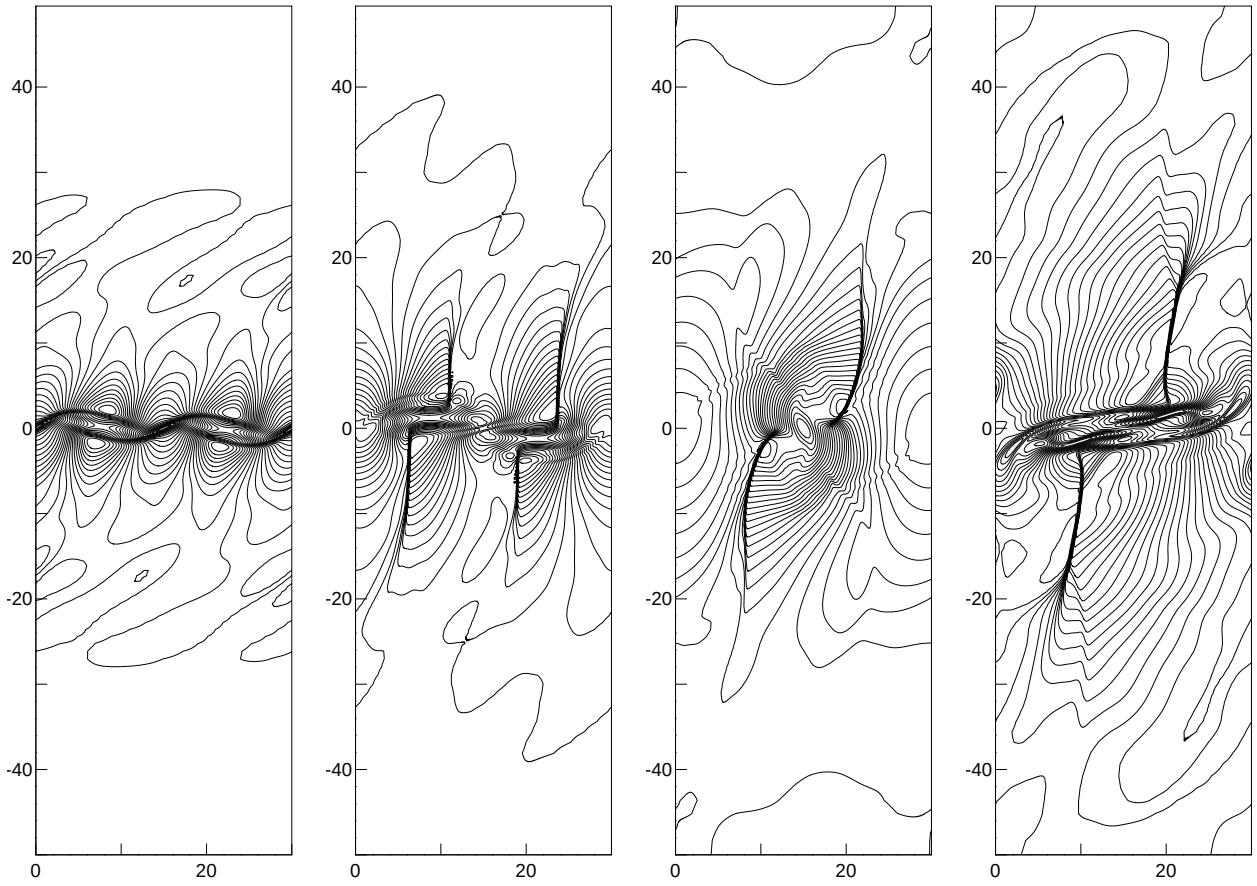



**Fig. 11 Isolines of normalized temperature. Figures for $t$=40, 80, 120, 160**

The top and bottom boundaries are treated as inviscid walls, and periodic boundary conditions are set on the left and right boundary. The kinematic viscosity in the freestreams is set to $\nu_\infty = 10^{-3}$, corresponding to a Reynolds number $Re = 1000$. Sutherland's law for viscosity has been used in the computations. The speed of sound (and hence density) in the initial solution is determined from the assumption of constant stagnation enthalpy

$$a^2 = a_1^2 + \frac{\gamma - 1}{2}(u_1^2 - u) \tag{18}$$

and $Ma_\infty = 0.8$. Constant static pressure is assumed across whole flow-field. The grid used consists of an isotropic triangulation (see fig 9) stretched in the $y$-direction using the mapping

$$y = \frac{L_y}{2}\frac{\sinh(b_y \eta)}{\sinh(b_y)}, \tag{19}$$

where $\eta \in (-1, 1)$ and $b_y = 3.4$. The mesh contains 201×201 nodes. The computation has been run with the B scheme.

The solution at times $t = 40, 80, 120, 160$ are shown in figures 11, where we plot 30 levels of isolines of the temperature. Comparing with the fourth order results of,[11] our method shows all features of flow-field. We judge the results very satisfactory considering that our method is at most second order accurate.

## Pseudo-time Integration

The steady-state solution is found by marching in pseudo-time. Because we are not interested in the transient state, we can use the fastest and the most robust method. Although explicit pseudo-time integration works well, for stiff problems the pseudo-time step restriction is too severe and the computation becomes too long. We implemented also implicit time-stepping with the backward Euler method in pseudo-time.

The explicit local time step integration can be written as

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \frac{\Delta\tau_i}{S_i}\mathbf{R}_i^n. \tag{20}$$

where $\Delta\tau_i$ is computed as

$$\Delta\tau_i = CFL \cdot \Delta\tau_{i0}$$

with $\Delta\tau_{i0}$ the maximum pseudo time-step allowed by the positivity of the N scheme in the linearized case. The simplest implicit method, linear Euler backward method, is obtained by evaluating the residual at the

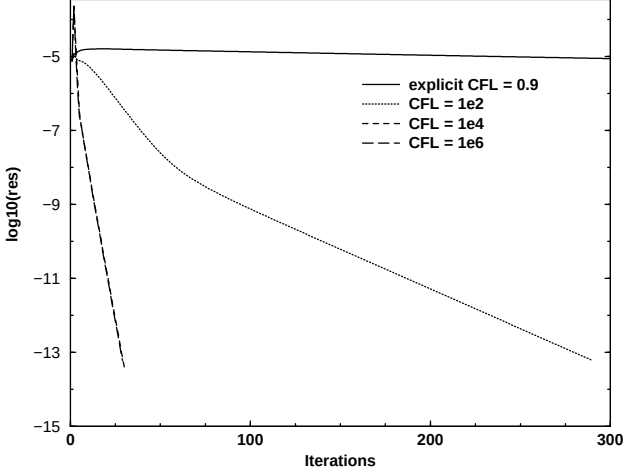



**Fig. 12 Convergence of method for one physical time-step for 2D Navier-Stokes equations in terms of iterations.**

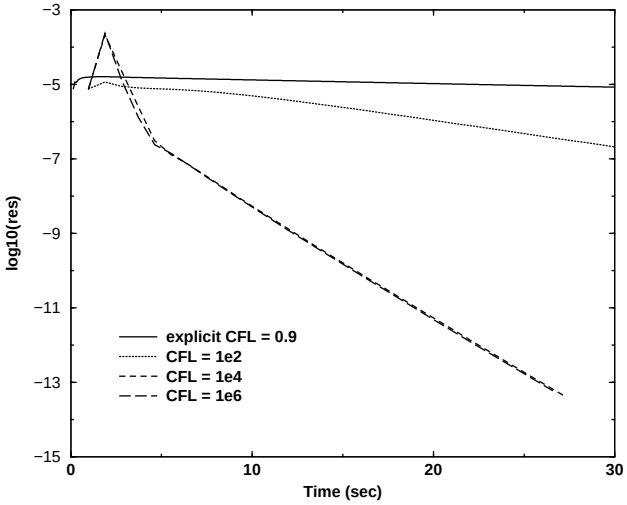



**Fig. 13 Convergence of method for one physical time-step for 2D Navier-Stokes equations in terms of CPU time.**

next time level. The residual in the time level $n+1$ is extrapolated using the first order approximation

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \frac{CFL \cdot \Delta\tau_{i0}}{S_i} \left( \mathbf{R}_i^n + \frac{\partial \mathbf{R}}{\partial \mathbf{U}} (\mathbf{U}^{n+1} - U^n) \right). \tag{21}$$

This leads to large system of equations with sparse matrix for the unknown $\Delta\mathbf{U} = \mathbf{U}^{n+1} - \mathbf{U}^n$. The method is unconditionally stable in the linear case.

The Jacobian matrix $\partial\mathbf{R}/\partial\mathbf{U}$ is analytically computed by a Picard approximation, i.e. the distribution coefficient (matrix) and the Jacobians are assumed to be independent on the solution vector.

The resulting system is then solved by GMRES method with ILU(0) preconditioning. The PETSc algebraic library has been used in all the computations.[12]

The method was tested also on scalar problems, but concern is related to the Navier-Stokes equations. The first test case we consider, simple enough to be able to compute it also with the explicit procedure, was similar to the first Stokes problem, but in a channel. The fluid is in the rest in the channel and suddenly the walls start to move. In fig 12 the decay of residual in terms of iterations during one pseudo-time step is plotted. We can immediately see, how fast is the implicit time integration compared to the explicit one. This is also due to space-time nature of the problem, in which the new solution we look for is relatively close to the initial condition in pseudo-time. More important is the convergence of the method in terms of CPU time (fig 13). The method was working for *CFL* numbers up to $10^{10}$, but setting *CFL* higher then approximatively $10^5$ did not change the speed significantly.

As a second test case we considered the Double Mach Reflection of Woodward and Colella.[13] The problem involves the inviscid computation of a Mach 10 shock (not shown here) reflecting on a wedge. From this type of computation one could expect problems due to the strong discontinuities present in the solution and to the approximations made in the derivation of system matrix. As a matter of fact, the implicit method converges faster in terms of pseudo-time iterations, but not in terms of CPU time.

As a final remark, note that the computation of the vortex pairing in a mixing layer took approx. 4 days on Linux workstation equipped with CPU Pentium 4 running on 1.7 GHz and 512 MB of memory. The computation with the explicit pseudo-time integration would not be possible.

## Conclusions

In the work the use of space-time RDS has been extended to the solution of viscous problems. We derived a consistent treatment of convective and dissipative terms through a Petrov-Galerkin interpretation of the RDS. The presented approach as been tested on scalar equations, where second order of accuracy for the LDA scheme and almost second order of accuracy for the B scheme have been measured. The method has been applied to the solution of the laminar Navier-Stokes equations. Promising results have been obtained.

A performance comparison between implicit and explicit pseudo-time integration techniques has been presented. For viscous computations implicit pseudo-time integration over-performs the explicit one of an order of 50 in terms of CPU time. Worse performances have of the implicit integrator have been observed for inviscid problems containing strong discontinuities.

## References

[1] Paillère, H., *Multidimensional Upwind Residual Distribution Schemes for the Euler and Navier-Stokes Equations on Unstructured Grids*, Ph.D. thesis, Université Libre de Bruxelles, von Karman Institute for Fluid Dynamics, Jun 1995.

Multidimensional Upwind Residual Distribution Schemes and Applications in Aeronautics," *AAIA*, Vol. 2328, 2000.

[3]Ferrante, A., "Solution of the Unsteady Euler Equations Using Residual Distribution and Flux Corrected Transport," Tech. rep., von Karman Institute for Fluid Dynamics, Belgium, Chausée do Waterloo 72, B-1640 Rhode Saint Genèse, Belgium, Jun 1997.

[4]Hubbard, M. and Roe, P., "Compact High-Resolution Algorithms for Time-Dependent Advection on Unstructured Grids," *International Journal for Numerical Methods in Fluids*, Vol. 33, 2000.

[5]Csík, A. and Deconinck, H., "Space Time Residual Distribution Schemes for Hyperbolic Conservation Laws on Unstructured Linear Finite Elements," *International Journal for Numerical Methods in Fluids*, Vol. 40, 2002, pp. 573–581.

[6]Abgrall, R. and Mezine, M., "Construction of Second Order Accurate Monotone and Stable Residual Distributive Schemes: the Unsteady Case," RC-MAB Report 2001-33, Universite Bordeaux 1, 2001.

[7]Mezine, M. and Abgrall, R., "Upwind Multidimensional Residual Schemes for Steady and Unsteady Flows," *ICCFD2, International Conference on Computational Fluid Dynamics 2*, Sydney, Australia, 15-19 July 2002.

[8]Csík, A., Ricchiuto, M., Deconinck, H., and Poedts, S., "Space-Time Residual Distribution Schemes for Hyperbolic Conservation Laws," *15th AIAA Computational Fluid Dynamics Conference*, Anaheim, California, Jun 2001, AIAA 2001-2617.

[9]Deconinck, H., Roe, P. L., and Struijs, R., "A Multidimensional Generalization of Roe's Flux Difference Splitter for the Euler Equations," *Computers and Fluids*, Vol. 22, 1993, pp. 215–222.

[10]Schlichting, H., *Boundary-Layer Theory*, Mc-Graw-Hill, 1979, ISBN 0-07-055334-3.

[11]Yee, H. C., Sandham, N. D., and Djomehri, M. J., "Low-Dissipative High-Order Shock-Capturing Methods Using Characteriscic-Based Filters," *Journal of Computational Physics*, Vol. 150, 1999, pp. 199–238.

[12]Balay, S., Gropp, W. D., McInnes, L. C., and Smith, B. F., "PETSc home page," http://www.mcs.anl.gov/petsc.

[13]Woodward, P. and Colella, P., "The Numerical Simulation of Two-Dimensional Fluid Flow with Strong Shocks," *Journal of Computational Physics*, 1984, pp. 115–173.