An Innovative Computational Fluid Dynamics Solver Based On the Original Form of the Conservation Laws for Fluid Flow

Gafar Elamin‡, Frederick Ferguson*
Center for Aerospace Research North Carolina A&T State University

An innovative and robust algorithm capable of solving a variety of complex fluid dynamic problems is developed. This so-called, Integro-Differential Scheme, (IDS) is designed to overcome known limitations of established schemes. The IDS implements a smart approach in transforming 3-D computational flowfields of fluid dynamic problems into their 2-D counterparts, while preserving their physical attributes. The strength of IDS rests on the implementation of the mean value theorem to the integral form of the conservation laws. This process transforms the integral equations into a finite difference scheme that lends itself to efficient numerical implementation. Preliminary solutions generated by IDS demonstrated its accuracy in terms of its ability to capture flowfield physics. In this paper, the results of applying the IDS to two problems; namely, the flow over a flat plate, and the shock/boundary layer interaction problem, are documented and discussed. In both cases, the results showed very good agreement with the physical expectation for the problem. In an effort to validate the IDS, its solution to the shock/boundary layer interaction problem was compared to that generated by the NASA-GRC Conservation Element Solution Element scheme. The results obtained by IDS are comparable if not better than the GRC results.

Nomenclature

ρ	=	Density
u	=	Velocity in the x-direction
v	=	Velocity in the y-direction
e	=	Internal energy
E	=	Total energy
T	=	Temperature
V	=	Magnitude of velocity vector
q_x	=	Heat flux in the x-direction
q_y	=	Heat transfer in the y-direction
p	=	Pressure
μ	=	Dynamic (absolute) viscosity

‡ PhD. Student, *Center for Aerospace Research*, Greensboro, North Carolina, 27411, AIAA Student Member.
* Director, *Center for Aerospace Research*, Greensboro, North Carolina, 27411, AIAA Senior Member.

τ_{xx} = Normal stress in the x direction

τ_{yy} = Normal stress in the y direction

τ_{xy} = Shear stress in the y direction exerted in a plane perpendicular to x direction

k = Thermal conductivity

ρ_∞ = Density at freestream conditions

M_∞ = Mach Number at freestream conditions

p_∞ = Pressure at freestream conditions

a_∞ = Speed of sound at freestream conditions

u_∞ = X-component of the velocity at freestream conditions

T_∞ = Temperature at freestream conditions

Re_L = Reynolds number based on a characteristic length, L

Pr = Prandtl number

γ = Ratio of c_p to C_v

I. Introduction

The boundary value problems involved with non-linear vertical equations of the elliptic-hyperbolic type governing fluid flow are as complicated as to make analytic methods virtually impossible. In addition, aerospace designers are currently demanding a solution to problems under conditions that cannot be duplicated with existing experimental facilities. Hence, the only way to obtain reasonable, complete information on fluid flows and their characteristics lies in computational fluid dynamics (CFD). As known, there are many well-established numerical schemes in the literature; they all have their strengths and weaknesses. Even though, these schemes have led to significant improvements in the art of CFD, they are not adequate to handle the existing demands, and they have many drawbacks. For instance, the efficiency and expense of running different configurations by a given CFD code are the main concerns of aerospace communities. In addition, the inability of the existing CFD codes to facilitate inexperienced users with limited training is a serious setback. The key goal, therefore, is to improve the efficiencies of CFD tools thus making these communities more productive. This research focuses on the development of a simple, robust, efficient, and accurate numerical framework that is capable of solving a variety of complex fluid dynamics problems and overcoming several limitations of well-established schemes. The new scheme is built with extensive physics considerations and has the following features:

1. The scheme is based on a smart integration of the traditional finite volume and finite difference schemes and therefore guarantees the conservation properties throughout the domain by the first and the formulation simplicity by the latter.
2. The integral form of the physical conservation laws is used, rather than the differential form. As such, the scheme has the potential to capture the realities of flow physics more efficiently. The differential form follows from the integral form under the additional assumption that the physical solution is smooth, an assumption that is difficult to realize numerically in a region of rapid change, such as a boundary layer or a shock.
3. The scheme guarantees the numerical solution continuity, since it focuses on the flux quantity's conservation, rather than manipulating the primitive flow variables. The fluxes are continuous, and primitive variables are not.
4. An accurate accounting of the mass, momentums, and energy fluxes is considered within the control volume and through its surfaces. The mean value theorem is used to evaluate the rate of change of fluxes at the

center of the control volume, rather than the traditional extrapolating or interpolating of the node's values at the neighboring cells in typical finite volume schemes. The extrapolating and interpolating processes are generally time-consuming and may result in numerical smearing.

5. A consistence averaging process is maintained in finite difference formulation; again the mean value theorem is used to evaluate the derivatives. In the traditional finite difference schemes, derivatives at the mesh points are expressed in terms of mesh values of dependent variables by using finite difference approximations. The accuracies of these approximations, especially those of higher order accuracies, are generally excellent as long as dependent variables vary slowly across a mesh interval, but may not be adequate if these variables vary too rapidly. Thus, in a high-gradient region, (for example), in a shock wave, accuracy may demand the use of an extremely fine mesh, which, in turn, may cause a prohibitively high computing cost. The rate of change of fluxes at each mesh point is evaluated as an average of the rates of change at the centers of the four neighboring control volumes that share a node.

II. The Integro-Differential Scheme

The Integro-Differential Scheme is a smart integration of the traditional finite volume and finite difference schemes. It relied on the coupled behavior of discretized cells and their corresponding nodes. And the numerical process is conducted in two alternating fashions. A typical control volume, illustrated in Figure 1, describes the numerical details associated with the finite volume formulation. Similarly, numerical details associated with the finite difference formulation are described through the use of Figure 2.

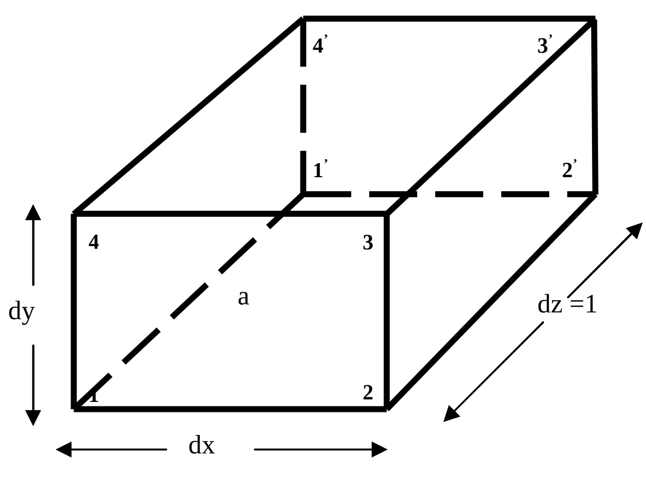


Figure 1. Finite Volume Representation

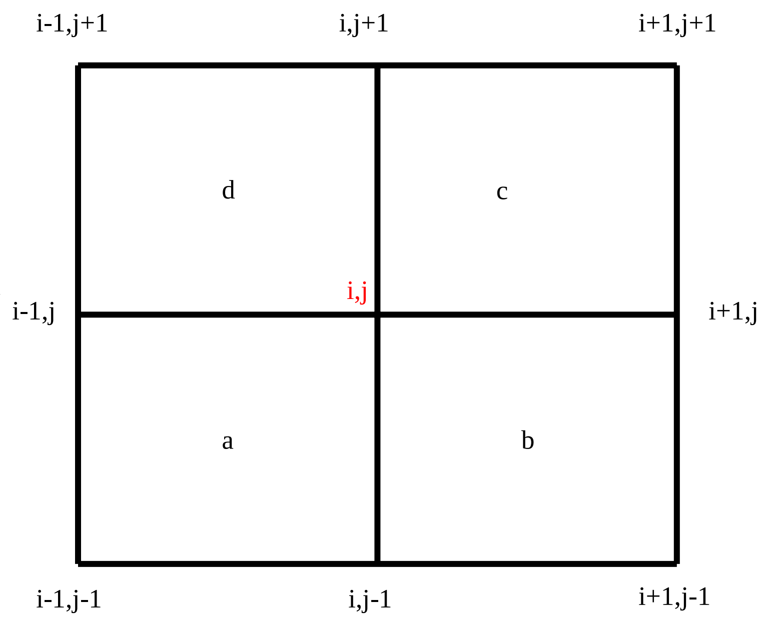


Figure2. Finite Difference Representation

A. The Governing Equations

When defining any numerical solution to a fluid dynamic problem, the conservation laws must be satisfied. As known in fluid dynamics, the conservation laws can be applied in two basic forms: the differential form and the integral form. However, experience has shown that when the integral form of the conservation laws is applied to fluid dynamics problems, high fidelity numerical solutions can be obtained. As such, the development of the Integro-Differential Scheme outlined in this research is based only on the integral formulation of the conservation laws. Mathematically, the conservation laws, namely, mass, momentum and energy, can be expressed by the following equations,

1. Conservation of Mass

$$\iiint_v \frac{\partial \rho}{\partial t} dv + \oiint_s \rho \overline{V} d\overline{s} = 0 \tag{1}$$

In Equation (1) ρ, v, t, represent density, the volume, and time, respectively.. The symbol $d\overline{s}$ represents the surface of the control volume and is given by the following vector:

$$d\overline{s} = dydz\,\overline{i} + dxdz\,\overline{j} + dxdy\,\overline{k} \tag{2}$$

Also, the vector $\overline{V}$ represents fluid velocity and can be expressed as

$$\overline{V} = u\,\overline{i} + v\,\overline{j} + w\,\overline{k} \tag{3}$$

2. Conservation of Momentum

$$\frac{\partial}{\partial t} \iiint_v \rho \overline{V} dv + \oiint_s \left(\rho \overline{V} . d\overline{s}\right) \overline{V} = -\oiint_s P d\,\overline{s} + \oiint \hat{\tau} d\overline{s} \tag{4}$$

where P is the pressure and the symbol, $\hat{\tau}$, represents a tensor that defines the various components of the local viscous stresses. This tensor can be described by the following equation:

$$\hat{\tau} = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix} \tag{5}$$

where, the symbols $\tau_{xx}, \tau_{xy}, \tau_{yy}, \tau_{yx}, \tau_{zx}, \tau_{zy}$ and τ_{zz} are the local shear stress components.

3. Conservation of Energy

$$\frac{\partial}{\partial t} \iiint_v \rho E dv + \oiint_s \rho E \overline{V} . d\overline{s} = -\oiint_s P \overline{V} . d\overline{s} + \oiint_s \hat{\tau} . \overline{V} d\overline{s} + \oiint_s \dot{\overline{q}} d\overline{s} \tag{6}$$

The vector $\dot{q}$ represents the rate of heat conducted per unit area through the surface of the control volume. In general, the vector $\dot{q}$ can be rewritten in coordinate form such that

$$\bar{\dot{q}}_{vis} = \dot{q}_x \bar{i} + \dot{q}_y \bar{j} + \dot{q}_z \bar{k} \tag{7}$$

where $\dot{q}_x$, $\dot{q}_y$, and $\dot{q}_z$ represent the rate of heat conducted per unit area in *x* ,*y*, and *z*, respectively. As stated in the literature, the thermal conduction is proportional to the temperature gradient. Mathematically, this relation can expressed as,

$$\dot{q}_x = -k \frac{\partial T}{\partial x} \tag{8}$$

$$\dot{q}_y = -k \frac{\partial T}{\partial y} \tag{9}$$

$$\dot{q}_z = -k \frac{\partial T}{\partial z} \tag{10}$$

where *k* is the thermal conductivity, and the minus sign accounts for the fact that the heat is transferred in the opposite direction of the temperature gradient.

In equation (6) *E* represents the total energy per unit mass is the summation of the internal energy *e*, and the kinetic energy.

$$E = \left(e + \frac{V^2}{2} \right) \tag{11}$$

B. The Integro-Differential Scheme Implementation

The numerical process of this scheme consists of the following steps:

1. The governing equations were employed on each small control volume to formulate their finite difference form. This task is accomplished by applying the mean value theorem to the integral form of the equations to evaluate the rate of change of mass, momentum, and energy at the center of each control volume.
2. The average value of the rate of change of mass, momentum, and energy for each four neighbor cells is calculated, which represents the rates of change at the center node of those cells.
3. Taylor's series expansion is employed to update the solution at the centered node.

III. Results

A. The Supersonic flow over a Flat Plate Problem

The supersonic flow over a flat plate is a classical fluid dynamic problem, and it has received considerable attention from many researchers, However, it has no exact analytical solution. Even though it can be claimed that some traditional techniques can solve this problem, the results obtained from these techniques are reasonably good for certain applications. Their approximate nature is extremely limiting in terms of flight condition and geometry. The innovative CFD solver was employed to solve this problem. As in Figures 3 through 6, the results showed very good agreement with the physical expectation of viscous flow over a flat plate.

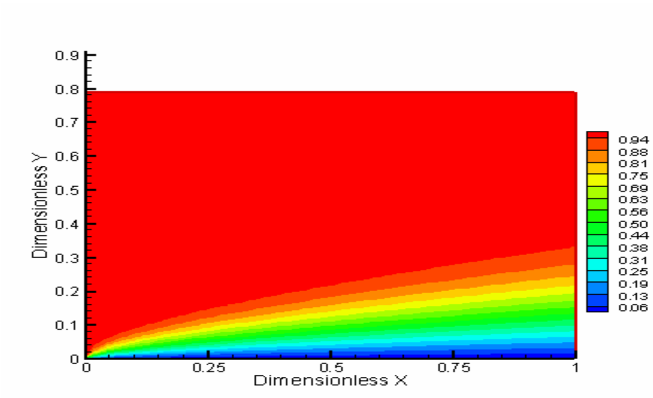


Figure 3. u Component of the Velocity Distribution

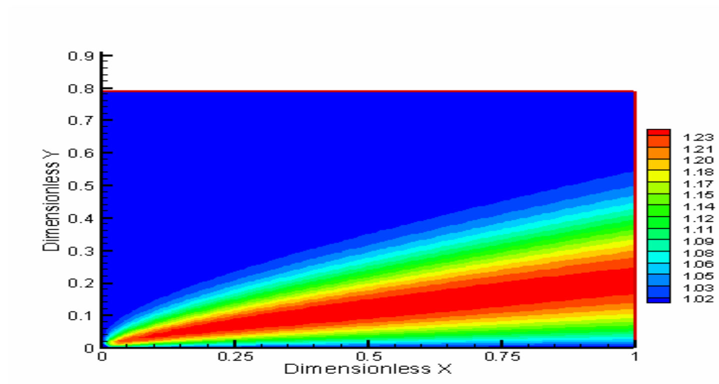


Figure4. Temperature Distribution

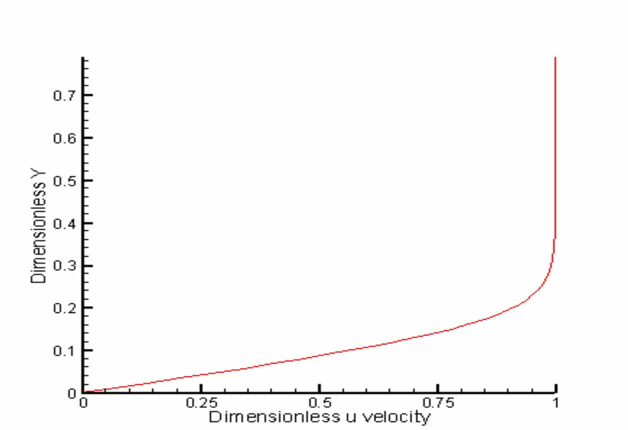


Figure 5. u Component of the Velocity Profile

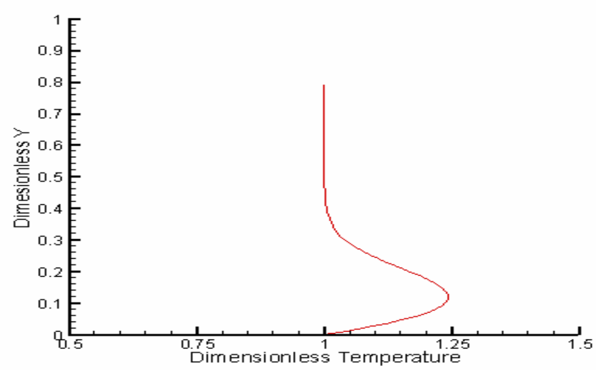
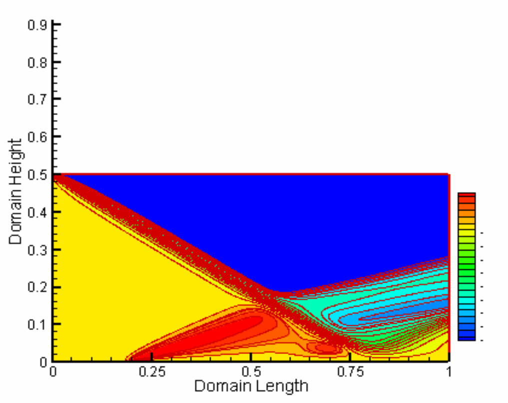


Figure 6. Temperature Profile

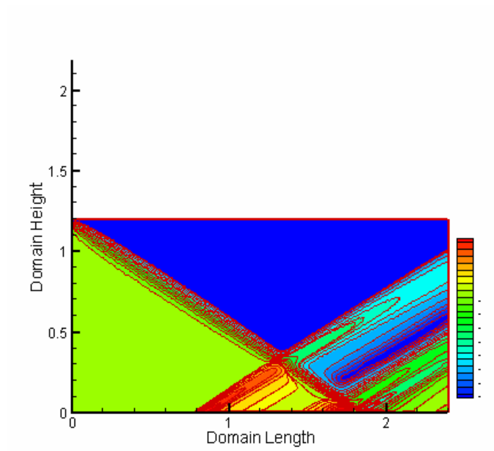
B. The Shock/Boundary Layer Interaction Problem

The shock/boundary layer interaction phenomenon is of considerable importance from both theoretical and practical points of view. This phenomenon has a large impact on the design of hypersonic vehicles for the presence of extended recirculation regions, intense local heating, and a loss of efficiency of the aerodynamic control surfaces.

The complexity of the phenomenon and its importance in the design of a hypersonic vehicle requires an understanding of the controlling effects and their characterization. In the last few decades, studies dealt with the shock wave/boundary layer interaction problem, mainly from an experimental point of view. More recently, numerical investigations of this problem have also been possible due to the massive increase in computer capabilities. Moreover, this problem has become a benchmark of testing new numerical methods. The innovative solver is employed to solve this problem. In an effort to validate the results, the same problem was solved using the NASA-GRC second order conservation element-solution element scheme. Figures 7 through 8 depicted a comparison between the results obtained by the new innovation and the NASA-GRC second order scheme results.

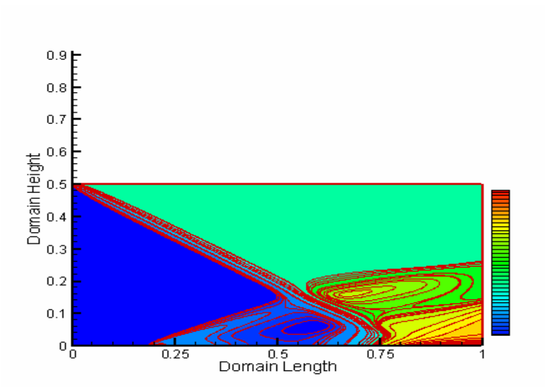


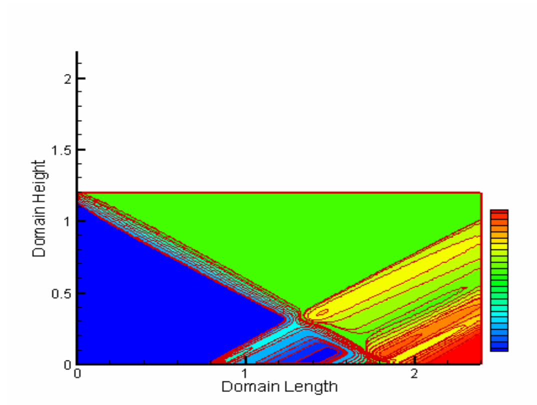
(a) The Innovative Scheme Solution



(b) NASA-GRC Scheme Solution

Figure 7. The Normal Component of the Velocity Distribution





(a) The Innovative Scheme Solution **(b) NASA- GRC Scheme Solution**

Figure 8. The Pressure Distribution

IV. Conclusion

A new numerical scheme for solving equations that govern fluid dynamics problems was developed. The innovation is called the integro-differential scheme. The scheme name depicts exactly what it says by combining the integral form of the conservation laws to formulate the governing equations and transforming them in a suitable differential form for the finite difference representation. The concept of the control volume was considered when calculating the integrations and the finite difference held for the numerical implementation of the scheme.. Moreover, the scheme was built on a very simple and strong mathematical foundation with extensive physics considerations. These considerations solidified the belief that the scheme is robust, efficient, and capable of solving a variety of complex fluid dynamics problems

In the stage of the scheme development, an unsteady, compressible, viscous flow over a flat plate problem was considered. The Integro-Differential Scheme was implemented using FORTRAN as a programming language and the solution results were visualized using the TECPLOT application. The results showed very good agreement with the physical expectation of viscous flow over a flat plate. Because the shock/boundary layer interaction problem has become a benchmarck to test new numerical methods for viscous flow since MaxCormack's work of 1971, the new scheme was employed to solve it. The results of this problem were obtained and visualized using TECPLOT. As a validation, the same problem was solved using the NASA-GRC second order conservation element-solution element scheme. The TECPLOT visual comparsion for the shock/boundary layer interaction problem showed that the integro-differential scheme results are comparable if not better than the NASA-GRC second order scheme results.

References

[1] J. D. Anderson, Jr., "*Computational Fluid Dynamics-The basis with applications*", McGraw-Hill, Inc., 1995.

[2] T. J. Chung, "*Computational Fluid Dynamics*", Cambridge University Press, 2002.

[3] Stephen Akwaboa, "*Navier-Stokes Solver for a Supersonic Flow over a Rearward-Facing Step*", M. S. Thesis, (Department of Mechanical Engineering, North Carolina A & T State University, Greensboro, 2004).

[4] S. C. Chang, X. Y. Wang, and C. Y. Chow, "*The space-time Conservation Element and Solution Element method: A new high-resolution and genuinely multidimensional paradigm for solving conservation law*", Journal of Computational Physics, Vol. 156, 1999, PP 89-136.

[5] Zeng-Chan Zhang, S. T. John Yu, and Sin-Chung. Chang, "*The space-time Conservation Element and Solution Element Method for Solving the Two and Three Dimensional Unsteady Euler Equations Using Quadrilateral and Hexahedral Meshes*", Journal of Computational Physics, Vol. 175, 2002, PP 168-199.

[6] Sin-Chung. Chang, "*The Method of space-time Conservation Element and Solution Element- A New Approach for Solving the Navier-Stokes and Euler Equations*", Journal of Computational Physics, Vol. 119, 1995, PP 295-324.

[7] Moujin Zhang, S. T. John Yu, and Sin-Chang, "*Solving the Navier-Stokes Equations by the CESE Method*". AIAA paper no. 2004-0075, 2004.

[8] X. Y. Wang, "*Computational Fluid Dynamics Based on the method of Space-Time Conservation Element and Solution Element*", Ph.D. dissertation, (Department of Aerospace Engineering, University of Colorado, Boulder, 1995).

[9] Klaus A. Hoffmann and Steve T. Chiang, "*Computational fluid dynamics for engineers*", Engineering Education System, March 1995.

[10] Anderson, John D., "Modern *Compressible Flow With Historical Perspective*", 2nd ed., 1990 McGraw-Hill Publishing Company, 1995.

[11] NASA GLENN Research Center: www.grc.nasa.gov/www/k-12/airplane/atmosmet.html

[12] John D. Anderson, Jr., "*Fundamental of Aerodynamics*", 3rd ed. Published by McGraw-Hill, 2001.

[13] David C., Wilcox, "*Basic Fluid Mechanics*", 1st ed., 1997 by DCW Industries, Inc.

[14] G., Moretti, and M., Abbet, "*A Time-Dependent Computational Method for Blunt-Body Flows*", AIAA Journal, vol. 4, no. 12, 1966, pp. 2136-2141.

[15] D. A., Anderson, J. C., Tannehill, and R. H., Pletcher, "*Computational Fluid Dynamics and Heat Transfer*", Hemisphere Publishing Co./McGraw-Hill Book Co., 1984.