Validation of Internal Flow Prediction Codes for Solid Propellant Rocket Motors

H. Tuğrul TINAZTEPE, Mine E. YUMUŞAK
ROKETSAN Missiles Industries Inc.
Ankara-Samsun Karayolu 40.km
PO. Box 30 Elmadağ
06780 Ankara
TURKEY

François VUILLOT
ONERA
BP 72-29 Avenue de la Division Leclerc
F-92322 Chatillon Cedex
FRANCE

Flow structure of inside solid propellant rocket motors has an important role in obtaining better performance and reliability. Recent developments in computational fluid dynamics lead to the solid propellant motor manufacturers use more internal flow solvers before static testing. The codes intended to be used for predicting the flow inside solid rocket motors should be validated by means of several test cases. In many cases internal flow field measurements can not be obtained due to very complex structure of the field. For this reason a support project of formerly AGARD has been completed between ROKETSAN and ONERA in order to validate the computational results by comparing different codes for different test cases. The outcomes of this study are presented in this paper. The aim of this project was to improve analysis capabilities for complex flow fields inside solid propellant rocket motors by validating 2-D planar and axisymmetric flow solvers. Two different flow solvers are validated for steady-state operation prediction of solid propellant rocket motor solutions and compared with ONERA’s computations for the test cases proposed by ONERA. Analytical solutions are also performed for simplified test cases and compared with the computational results. Pressure, Mach number, entropy of the flow field obtained from different codes are presented and the residual convergence criteria, mass flow rate and head end pressure evaluation are discussed with grid requirements for a better computational result.

NOMENCLATURE

\[ U, V = x, y \text{ components of velocity vector} \]
\[ v_{\text{inj}} = \text{injecting velocity} \]
\[ m = \text{mass flow rate} \]
\[ \rho = \text{density} \]
\[ P = \text{static pressure} \]
\[ T = \text{static temperature} \]
\[ S = \text{entropy} \]
\[ H = \text{total enthalpy} \]

Subscripts

\[ \text{inj} = \text{injecting quantity} \]
\[ \text{ex} = \text{exit quantity} \]
\[ \text{ref} = \text{reference quantity} \]
\[ \text{nd} = \text{non-dimensional quantity} \]
\[ \text{th} = \text{theoretical quantity} \]

1.0 INTRODUCTION

The internal flow field plays an important role in several aspects of the performance and the design of a propulsion system such as solid propellant rocket motors. The nozzle and combustion chamber flow constitutes one of the most important aspects for solid rocket motor design. The nozzle gas dynamics calculations are sensitive to the flow entering the nozzle and hence performance prediction requires a detailed knowledge of the chamber flow field. Indeed, better knowledge of internal flow field should make possible to optimize solid propellant grain geometry and thermal insulation thickness needed for good structural loading [1, 2], and to better control current complex geometrical effects (of solid propellant grains) on combustion product flow field.

Recently, much attention has been given to numerical simulation of internal aerodynamics of solid rocket motors. Due to the recent progress in computing power, this field is developing rapidly. Analysis of the motor’s internal flow using the techniques of Computational Fluid Dynamics (CFD) would also be extremely useful in the study of motor phenomena not measurable through static motor tests.

Prediction of rocket motor performance is currently performed with assumption of one-dimensional flow [3], and it has been demonstrated that this simplified internal analysis is able to give reasonable accuracy on total performance prediction (total impulse, specific impulse, burn time etc.). However, most solid propellant rocket motors use complex internal geometry. Flow simulations performed at different injection directions [4-5] have shown that the flow in mass injected chambers is highly two-dimensional. Therefore, in rocket motors, the two dimensional effects must be accounted for in order to assess their influence on performance. In order to validate the results of ROKETSAN codes; analytical solutions for simple test cases and computational results of ONERA for complex test cases are used. This is the subject of the present work that has been conducted at ROKETSAN with support from ONERA. The objective of the present study is to validate the newly developed computational tool for assessing the internal flow field of two dimensional or axisymmetric propulsion systems using Euler equations.

2.0 NUMERICAL SCHEME

2.1 Governing Equations

The Euler equations in either planar or axisymmetric coordinate systems read as follows:

\[ \frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \delta K = 0 \]  

where \( \delta = 0 \) in Cartesian coordinates, \( \delta = 1 \) in cylindrical coordinates.

\[ Q = \begin{bmatrix} \rho \\ \rho U \\ \rho V \\ \rho E \end{bmatrix}, \quad F = \begin{bmatrix} \rho U \\ P + \rho U^2 \\ \rho UV \\ \rho UH \end{bmatrix}, \quad G = \begin{bmatrix} \rho V \\ \rho UV \\ P + \rho V^2 \\ \rho VH \end{bmatrix}, \quad K = \begin{bmatrix} \rho V/y \\ \rho UV/y \\ \rho V^2/y \\ \rho VH/y \end{bmatrix} \]

For the axisymmetric case “y” corresponds the distance from the symmetry axis.

2.2 ONERA Solver “SIERRA”

The SIERRA code is a research code dedicated to internal flows inside solid rocket motors as well as to unsteady regimes that can develop on the motor acoustic modes. It is based on a cell centered,
finite volume, explicit predictor-corrector Mac Cormack’s [8] scheme, in its original unsplit form. Artificial viscosity can be added in the form of second and fourth order terms that are adapted to the flow by sensors reacting to the pressure and/or the velocity fields (Jameson’s type [6]). The code is multi-block and can be applied to a variety of geometry. Several boundary conditions are available and can be chosen by the user.

The 2-D unsteady Navier-Stokes equations are solved in their compressible form and in the physical domain. Turbulence modeling is limited to algebraic models, based on sub-grid scale modeling. A two-phase flow solver, based on an Eulerian (or two fluids) approach, is also available.

### 2.3 ROKETSAN Solver “IBSE2D”

The IBSE2D code is an explicit code for the analysis of inviscid internal flows inside 2-D and axisymmetric geometry. It uses a cell vertex, finite difference, explicit multistage Runge-Kutta solution scheme based on the work of Jameson’s [6]. The number of stages in the scheme can be selected, but a four-stage scheme is used. The governing equations are discretized in a structured Cartesian grid. A spatially varying time step (local time step) can be used to accelerate the convergence rate.

Implicit residual smoothing (IRS) can be used with the multistage scheme at elevated CFL (Courant-Fredrich-Lewy) numbers. IRS can also maintain stability and thus increase the convergence rate. The code uses second-order (central) finite differences throughout, and requires an artificial dissipation term to prevent odd-even decoupling. A fourth difference term is used for this purpose. The code also uses a second difference artificial viscosity term for shock capturing. Second difference artificial viscosity term is multiplied by a second difference of the pressure that is designed to detect shocks. In this study two different codes of ROKETSAN were validated and compared with ONERA results, but only the explicit code details are presented here. Similar results were obtained from the implicit code.

### 3.0 BOUNDARY CONDITIONS

The numerical boundary-condition treatment is achieved with classical methods. Mainly four types of boundary conditions may be encountered in solid propellant rocket motor internal flow problems as illustrated in Figure 1:

- **a)** head end and nozzle inert wall boundary
- **b)** subsonic/supersonic exit outflow boundary
- **c)** symmetry axis on the upper boundary
- **d)** solid propellant injecting surface boundary

![Figure 1: Boundary Conditions for Solid Propellant Rocket Motors.](image)

Because of its inviscid modeling, a slip condition is used for the inert wall boundary. For converging/diverging nozzle outflow is supersonic, a first-order extrapolation can be utilized.
For subsonic outflow conditions, exit pressure is specified. All of the flow quantities except normal component of velocity at the symmetry axis have the same values at the first node away from the centerline. Solid propellant is burning over the surface in normal direction, therefore tangential velocity can be taken as zero and mass flow rate and temperature of the gas phase combustion products are specified.

4.0 TEST CASES AND RESULTS

Four test cases are selected starting from simple to more complex motor geometry as summarized in Table 1. 1st test case is 2-D planar and analytical solution is used as a reference data. Axisymmetric version of this test case is also used to compare different computational results. 2nd test case is a channel connected to a nozzle in which capability of the codes to simulate both subsonic and supersonic flow is tested. 3rd test case is mainly used to test capability of codes to simulate sudden change of properties from propellant grain end to nozzle entrance. 4th test case includes motor head end cavity and reverse flow in this region. This final test case includes most of the flow features of an axisymmetric solid rocket motor and its internal flow character.

Table 1: Summary of Test Cases

<table>
<thead>
<tr>
<th>Test Case</th>
<th>Geometry</th>
<th>Objective to test</th>
<th>Flow Type</th>
<th>Injection from</th>
</tr>
</thead>
<tbody>
<tr>
<td>1A</td>
<td>analytical solution</td>
<td>2-D Planar</td>
<td>Propellant</td>
<td></td>
</tr>
<tr>
<td>1B</td>
<td>axisymmetric formulation</td>
<td>Axisym.</td>
<td>Propellant</td>
<td></td>
</tr>
<tr>
<td>2A</td>
<td>analytical solution of 1-D nozzle</td>
<td>Axisym.</td>
<td>Head end</td>
<td></td>
</tr>
<tr>
<td>2B</td>
<td>Nozzle entrance</td>
<td>Axisym.</td>
<td>Propellant</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Propellant grain to nozzle</td>
<td>Axisym.</td>
<td>Propellant</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Reverse flow at head end and to simulate full motor</td>
<td>Axisym.</td>
<td>Propellant</td>
<td></td>
</tr>
</tbody>
</table>

4.1 Test Case 1

A solid rocket motor grain port with a planar geometry has been used. For this test case, propellant burning from the side is modeled as a 2D planar porous wall. The aim was to verify the numerical solutions by comparing them with the analytical solutions of axial and transverse profiles of flow variables and entropy along the axis (centerline). Total length and the height are 581 mm and 20 mm respectively. Rate of injection from bottom wall is taken as constant over the surface with 2.42 kg/s/m². Exit plane pressure is 1.5 bar. Injected fluid is taken as air (cold flow test) at 303K. 51 x 16 grid points are used as given in Figure 2.

![Figure 2: Geometrical Representation of 2D Porous-Walled Duct.](image-url)
Taylor’s Solution [9] gives the theoretical velocity and entropy change along the injecting wall as:

\[ u = \frac{\pi}{2} \frac{x}{h} v_{\text{inj}} \cos \left( \frac{\pi}{2} \frac{y}{h} \right) \]  
\hspace{1cm} (2. a) 

\[ v = - v_{\text{inj}} \sin \left( \frac{\pi}{2} \frac{y}{h} \right) \]  
\hspace{1cm} (2. b) 

\[ \Delta S_{\text{th}} = \frac{1}{T} \left( \frac{\pi}{2} n \right)^2 \frac{L}{2} = \frac{1}{8T} \left( \pi v_{\text{inj}} \right)^2 \left( \frac{L}{h} \right)^2 \]  
\hspace{1cm} (2. c) 

According to Equation 2.a the maximum axial velocity occurs at the symmetry axis side of the exit, and its non-dimensional value is : \( u_{\text{nd}} = 0.1826 \), where \( v_{\text{nd}} = 0 \). and from Equation 2.b the maximum transverse velocity occurs at the injection wall side of the exit, and its non-dimensional value is \( v_{\text{nd}} = 0.0040 \), where \( u_{\text{nd}} = 0 \). Substitution of the known values into Equation 2.c gives non-dimensional entropy change along the injecting wall as: \( \Delta S_{\text{th}} = 0.01667 \).

### Table 2: Comparison of IBSE2D, SIERRA and Analytical Solutions at Exit Plane for Test Case 1.A

<table>
<thead>
<tr>
<th></th>
<th>( u_{\text{sym}} )</th>
<th>( \Delta S_{\text{inj}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>0.1826</td>
<td>0.01667</td>
</tr>
<tr>
<td>IBSE2D</td>
<td>0.1796</td>
<td>0.01636</td>
</tr>
<tr>
<td>SIERRA</td>
<td>0.1790</td>
<td>0.01608</td>
</tr>
</tbody>
</table>

Axial velocity profiles are given in Figure 3.a for planar solution and Figure 3.b for axisymmetric solution at different axial locations of grids. The gradients are resolved in a similar accuracy. The differences near the injecting wall and symmetry boundaries are due to cell-center topology of SIERRA and post processing technique used in this study.

![Figure 3: Non-Dimensional Axial Velocity (\( u_{\text{nd}} \)) versus Transverse Distance at Head-End (I=2), Mid-Plane (I=25), Aft-End (I=50) for IBSE2D and SIERRA.](image-url)
Figure 4 presents convergence histories toward steady-state solutions in terms of the ratio of the outlet mass flow rate to the injected mass flow rate. Convergence in mass balance (injected to exit ratio) shows the steady state reached in an engineering level of accuracy in 1/3 of total number of iterations. Oscillations in head end pressure are observed as mass flow rate oscillates. SIERRA results are given for exit to injected mass flow rate ratio of 0.999 and IBSE2D results are given for 0.997. Figure 4 shows the convergence history of the maximum of density residual in the solution domain. It is observed that the computational results of the field solutions presented above belong to a fully converged state (nearly to the machine accuracy).

4.2 Test Case 2

Second test case is a lab-scale-motor for the verification of mass injection boundary condition in an axisymmetric geometry and used for the validation of 2D flow stability computations [1]. This test case simulates a “complete motor” mode: the combustion products are injected laterally on the propellant surface and taken out from a nozzle. It corresponds to a cylindrical port motor of chamber length 170 mm and inner radius of 45 mm. It has a converging diverging nozzle of throat radius of 16.77 mm with a radius of curvature of 30 mm. Overall motor length is \( L = 270 \) mm. Rate of injection is 11.39 kg/s/m\(^2\) with
specific heat ratio of 1.14, gas constant of 299.5 J/kg/K and flame temperature of 3387 K. Reference pressure is taken as 1.0 bar. For the computations 99 x 16 grid points are used as shown in Figure 6.

![Figure 6: Computational Grid of Test Case 2.](image)

Results of inlet and exit Mach numbers of the nozzle, stagnation density and pressure are given in Table 3 for test case 2A. In this test case mass inflow is from head end boundary only.

<table>
<thead>
<tr>
<th>Test Case</th>
<th>( M_{\text{inlet}} )</th>
<th>( M_{\text{exit}} )</th>
<th>( P_o )</th>
<th>( \rho_o )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical Solution</td>
<td>0.0831</td>
<td>0.01096</td>
<td>2.4692</td>
<td>2.4692</td>
</tr>
<tr>
<td>IBSE2D</td>
<td>0.0829</td>
<td>0.01087</td>
<td>2.4755</td>
<td>2.4756</td>
</tr>
<tr>
<td>SIERRA</td>
<td>0.0828</td>
<td>-</td>
<td>2.4561</td>
<td>-</td>
</tr>
</tbody>
</table>

Test Case 2B corresponds to injection from sidewalls. Results of Mach number for Test case 2B are given in Figure 7. Although IBSE2D is an explicit code, convergence acceleration can be achieved by an implicit residual smoothing method. This method allows CFL numbers up to 4 for a stable solution. CFL number is taken as 3.9 with implicit residual smoothing. Although the flow contours are exactly the same for two different CFL numbers, implicit residual smoothing method accelerates the convergence as shown in Figure 8.a.

![Figure 7: Constant Mach Lines of IBSE2D for Different CFL Numbers.](image)
The density residual given in Figure 8.a started at a level of $10^1$ and fully converged solution (up to machine accuracy) is reached at a level of $10^{-20}$. The number of iterations required for engineering analysis is 10,000 for CFL of 0.9. This number of iteration may not be necessary since the head-end pressure and the mass flow rate reached to their nearly steady-state values after iteration $10^4$ respectively as shown in Figure 8.b and Figure 8.c.

Figure 8: Residual History of IBSE2D for Different CFL Numbers.

Figure 9.a and 9.b compare the steady state velocity profiles in the chamber ($x/L = 0.3148$) obtained with the Taylor’s Solution [4,9]. The agreement is good for both axial and radial velocity profiles.

(a) Axial Velocity  
(b) Radial Velocity

Figure 9: Computational and Analytical Velocity Profiles at $x/L = 0.3148$.

4.3 Test Case 3

This test case has the same properties of Test Case 2B, propellant grain aft end and nozzle entrance step add more complexity to the problem as well as the throat diameter is smaller than Test Case 2 resulting a high chamber pressure. Rate of injection from propellant surface is 16 kg/s/m². 121 x 16 grid points are
used as shown in Figure 10.

Figure 10: Computational Grid of Test Case 3.

Figure 11 shows the agreement of Mach values of IBSE2D and SIERRA flow solvers for this test case. Non-dimensional pressure along the motor axis is shown in Figure 12.a. The convergence criteria is based on the ratio of exit to the injected mass flow rate is less than 0.997 and density residual is less than $10^{-18}$.

Evolution of mass flow ratio is given in Figure 12.b.

Figure 11: Constant Mach Lines of IBSE2D and SIERRA.

( Range 1: $M_{\text{min}} = 0.002$, $M_{\text{max}} = 0.050$, $\Delta M = 0.01$,
Range 2: $M_{\text{min}} = 0.050$, $M_{\text{max}} = 3.08$, $\Delta M = 0.1$ )

Figure 12: Non-Dimensional Centerline Pressure Results and Injected Mass Flow Rate and Exit Mass Flow Rate Histories of IBSE2D.


4.4 Test Case 4

Total length and the inside radius are 270 mm and 20 mm. Injection rate from propellant surface is 19.6 kg/s/m². Reference pressure is taken as 1 bar. Injected gas specific heat ratio and gas constant are 1.22 and 287 J/kg/K. Flame temperature is 3149K. For the first calculations, 176 x 15 grid points are used as given in Figure 13. Effect of grid refinement is also analysed and results are given in the following paragraphs. Mach number and pressure results computed by IBSE1D and SIERRA are given in Figure 14-15.

Figure 13: Computational (Coarse) Grid of Test Case 4.

Figure 14: Test Case 4; Constant Mach Lines of IBSE2D and SIERRA.

( \( M_{\min} = 0.03 \), \( M_{\max} = 2.85 \), \( \Delta \text{Mach} = 0.085 \))

Figure 15: Test Case 4; Constant Pressure Lines of IBSE2D and SIERRA.

( Range 1: \( P_{\min} = 1.065 \), \( P_{\max} = 18.46 \), \( \Delta P = 0.8 \),
   Range 2: \( P_{\min} = 18.46 \), \( P_{\max} = 18.95 \), \( \Delta P = 0.04 \) )
5.0 GRID REFINEMENT

Coarse grid (175x15) solution for test case 4 is plotted against fine grid (463x43) solution for the same test case, and then a comparison is done regarding computed flow parameters such as; Mach number, entropy and pressure and time requirement for convergence to decide the mesh sensitivity of the solvers. The below given Mach contour plot shows that no improvement on Mach number values is reached with the fine grid computation of IBSE2D when compared with its coarse grid computation. Similarly, the computation with a fine grid causes an almost negligible change in pressure values as shown in Figure 16. The pressure in the subsonic regime computed with fine grid is a little high almost with a constant $\Delta P$ of 0.064. The constant entropy lines are given in Figure 17. Recall that all other test cases with relatively coarse grids satisfies the Equation of Crocco (i.e. constant entropy lines follow stream lines) very well in the subsonic flow regimes but after transonic region all the constant entropy lines having a great tendency of attaching the nozzle wall. The reason of this result is the diffusive effect of the numerical scheme over the coarse mesh.

In particular, the grid density found to have no influence on the vorticity field over most of the main flow regions of the chamber. However, the vorticity field between propellant aft surface and nozzle entrance significantly depends on the mesh size. This is illustrated in Figure 18. Indeed, only the fine grid solution permits to capture the rotational jet structure close to the surface. Moreover, the jet structure originating from aft corner of the propellant is maintained in fine grid solution while the coarse grid diffuse this shear layer.
Figure 18: Effect of the Grid Density on the Vorticity Field.

Figure 19 and 20 present velocity vectors for the different grids at both head-end region and the nozzle entrance region. The grid has an influence on the shape of flow in the dead zone between the conical grain surface and the head-end [1-2]. In the coarse grid solution the circulation near the wall can not be resolved, while with the fine grid solution a stable vortex can be captured. Between injecting wall and inert wall there is a transfer of a kinetic energy which either supplies a small rapid and unstable vortex for the coarse grid or a big slow and stable vortex for the fine grid [2]. At the nozzle entrance region, there is always a vortex, which seems to be independent on the grid size. This result came out because of the flapping nature of the flow in this region.

Figure 19: Velocity Vectors at Head-End Region.
Validation of Internal Flow Prediction Codes for Solid Propellant Rocket Motors

Figure 20: Velocity Vectors at Nozzle Entrance Region.

The conclusion reached after this study is that there is not much influence of the grid density on the iso-Mach number lines and isobars lines. The most marked influence of the grid size can be clearly seen on the vorticity fields. The fine grid allows a better definition of the flow at the head-end and nozzle entrance region.

Also for fine grid solution, travelling of transient vortices cause to increase residuals for a steady state convergence, therefore, it took longer time than expected compared with a coarse grid solution as shown in Figure 21. The same behaviour has been observed in ONERA computations.

Figure 21: Residual History of IBSE2D for Coarse (176x15) and Fine Grid (463x43).

6.0 CONCLUSION

During this study, the existing internal flow solvers of ROKETSAN were tested for various test cases ranging from simple to complex solid propellant rocket motor geometry. The solutions were compared with the available analytical solutions and with the solutions obtained at ONERA.
IBSE2D flow solver is in good agreement with SIERRA in terms of Mach number values. But it does not have as good agreement as fine grid, in prediction of pressure values in subsonic region for test case 4 with coarse grid configuration. This discrepancy may be attributed to the dissipative effect of the numerical scheme on coarse grids. Although pressure variation along the axial distance matches exactly with result of SIERRA computation in most of the test cases, small difference in pressure values exists for subsonic regions. Error in computation of exit mass flow rate is less than %1 of injection mass flow rate. This error is in acceptable limits as other computations. As a result of the validation study, IBSE2D flow solver is capable of predicting 2D/axisymmetric flows in solid propellant rocket motors with good accuracy in terms of mean flow field and with a sufficient accuracy in terms of vortical details.

7.0 REFERENCES


SYMPOSIA DISCUSSION – PAPER NO: 29

Discusser’s Name: Alba Lalitha Ramaswamy

Question:
Did you compare your predictions with measured data?

Author’s Name: H. Tugrul Tinaztepe

Author’s Response:
The scope of our study was to compare computational results. ONERA’s code SIERRA results have already been compared with experimental data and published previously. The references are stated in the paper.
This page has been deliberately left blank

Page intentionnellement blanche