CI2 – Inviscid Strong Vortex Shock Wave Interaction

Contact: Chongam Kim, chongam@snu.ac.kr

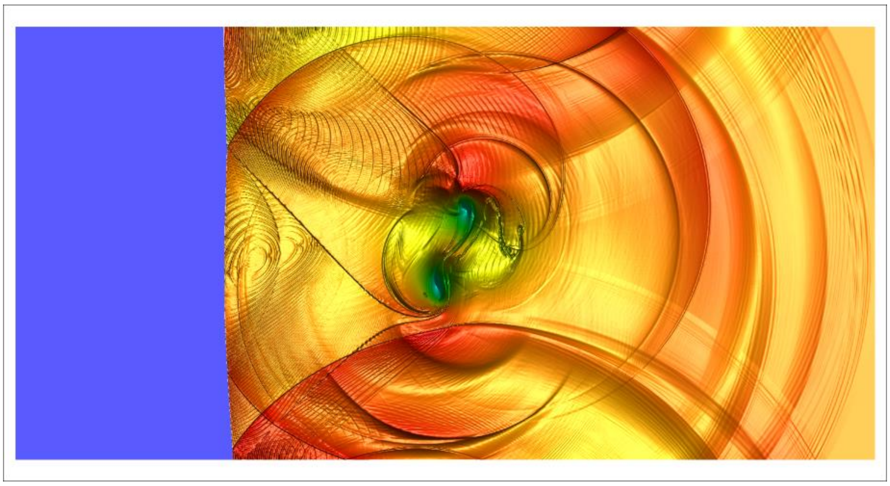

Figure 1. Density Contour of the Problem

Overview

This case is designed to verify that a scheme can capture complex physical phenomena resulting from the interaction between a strong vortex and a shock wave. This is a two-dimensional unsteady inviscid flow including multiple shock discontinuities. When the strong vortex and the strong shock wave encounter, considerable distortion of shock structure occurs, followed by the generation of linear and non-linear waves propagating onto the downstream flow fields. Two distinctive physical phenomena can be observed in this problem. First, the strong vortex is split into two separate vortical structure due to the compression effects of the shock passage. The post-shock vortical structure depends strongly on the relative strength of the shock and the vortex. Second, cylindrical acoustic wave structure appears on the downstream side of the stationary shock. The sound waves centered on the moving vortex core are partly cut off by the shock wave. As a result, alternating expansion and compression regions are observed.

Verification

Participants should compute the VI1 vortex transport test case to verify the inviscid implementation of each code. The verification test case should be run with the shock-capturing scheme adopted for the calculation of the strong vortex and shock wave interaction. The verification results should be included in the submission.

Computational Mesh

Four types of meshes, namely RT (Regular Triangle), IT (Irregular Triangle), RQ (Regular Quadrilateral), and M (Mixed), are provided according to the category of participants' solvers. Three categories are considered according to overall discretization strategy:

Category A. FEM (Finite Element Method)-type solvers

- DG, CG, FR, CPR, Hybrid Schemes, and so on

Category B. FVM (Finite Volume Method)-type solvers

Category C. FDM (Finite Difference Method)-type solvers

Each category uses a different set of meshes. Each mesh is named after its type and the reciprocal of its mesh size. The set of meshes for each category is provided as follows.

Category A. FEM-type solvers

. RT (Regular Triangle): RT50, RT100, RT150, RT200, RT250 and RT300

. IT (Irregular Triangle): IT50, IT100, IT150, IT200, IT250 and IT300

. RQ (Regular Quadrilateral): RQ50, RQ100, RQ150, RQ200, RQ250 and RQ300

. M (Mixed): M50, M100, M150, M200, M250 and M300

Category B. FVM-type solvers

. RT (Regular Triangle): RT50, RT100, RT200, RT300, RT400 and RT500

. IT (Irregular Triangle): IT50, IT100, IT200, IT300, IT400 and IT500

. RQ (Regular Quadrilateral): RQ50, RQ100, RQ200, RQ300, RQ400 and RQ500

. M (Mixed): M50, M100, M200, M300, M400 and M500

Category C. FDM-type solvers

. RQ (Regular Quadrilateral): RQ50, RQ100, RQ200, RQ300, RQ400 and RQ500

All types of meshes are provided in GMSH script files (.geo) for unstructured-grid-based solvers. Participants can generate appropriate GMSH grid files (.msh) by using the script files and the GMSH program. Read the instructions (Readme.txt) provided in the mesh files. In addition, RQ type meshes are provided in CGNS grid files (.cgns) for structured-grid-based solvers. Participants can directly use the CGNS grid files without any pre-processing.

At the initial state, the stationary shock is exactly aligned with the meshes. No local adaptive mesh refinement around the shock is applied. Substantial grid perturbation effects may be observed near the shock wave, particularly when the vortex is passing through the shock wave.

Problem Description

To model the flow physics of the strong vortex-shock interaction, the flow is assumed to be governed by the non-dimensionalized 2-D Euler equations. The system is closed by the equation of state for air (an ideal gas with the ratio of specific heats, $\gamma = 1.4$).

Initially, the flow fields contains a stationary shock with $M_s = 1.5$ and a strong vortex with $M_v = 0.9$. The shock is located at $x = 0.5$, and the center of the vortex is located at the point $(x_c, y_c) = (0.25, 0.5)$. The upstream flow quantities are specified by $(\rho_u, u_u, v_u, p_u) = (1.0,\ M_s\sqrt{\gamma},\ 0.0,\ 1.0)$, except for the vortex. The vortex rotates counter-clockwise with the angular velocity given below.

$$v_\theta = \begin{cases} v_m \dfrac{r}{a} & \text{if} \quad r \le a, \\ v_m \dfrac{a}{a^2 - b^2}\left(r - \dfrac{b^2}{r}\right) & \text{if} \quad a \le r \le b, \\ 0 & \text{if} \quad r > b. \end{cases}$$

Here, r is the distance from the vortex core (x_c, y_c), and $(a, b) = (0.075, 0.175)$. v_m is the maximum angular velocity, which occurs at $r = a$. We take $M_v = v_m/\sqrt{\gamma}$ as a measure of the vortex strength. The flow quantities of the downstream (ρ_d, u_d, v_d, p_d) (i.e. on the right side of the stationary shock) are determined from the upstream quantities with the stationary shock condition. Detailed information on the initialization process is given in the appendix.

The left boundary at $x = 0$ and the right boundary at $x = 2$ are considered as a supersonic inlet and a subsonic outlet, respectively. The upper and lower sides are treated as wall boundary. The target time for numerical simulation is $t = 0.7$. See Fig. 2 for the summary of the problem setup.

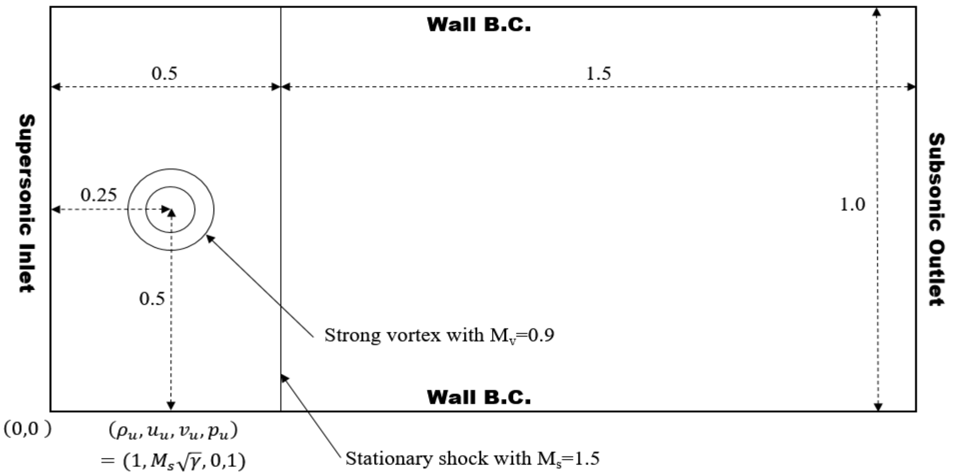

Figure 2. Problem Description

Data Submission

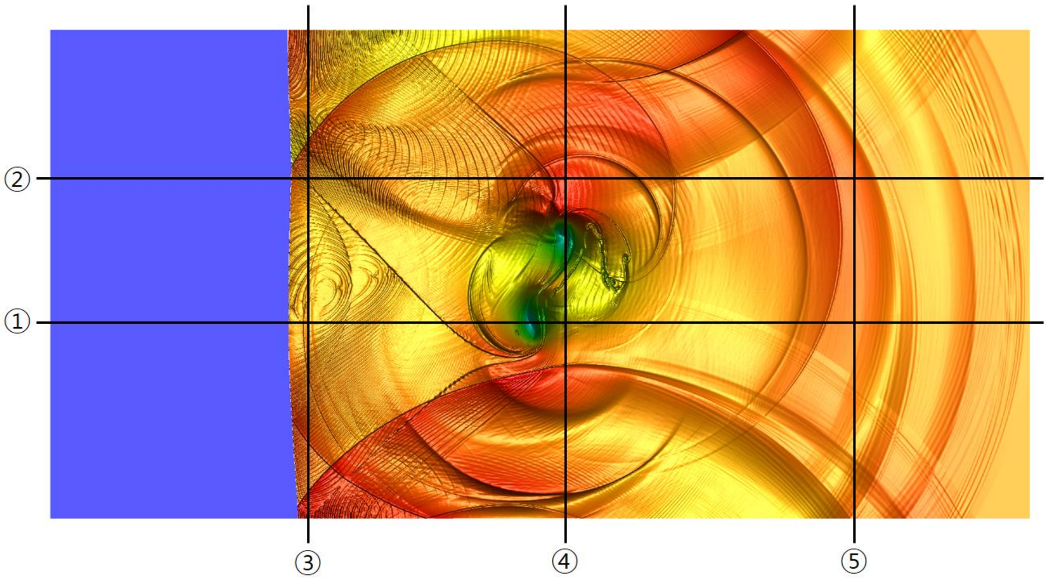

Figure 3. Selected Lines to Evaluate Computed Solutions ($t = 0.7$)

This flow is highly unsteady and complex with multiple shocks, which poses severe restriction on conventional order tests. The quality of submitted solutions is thus evaluated by reference solutions obtained on extremely fine meshes.

For some or all of the given mesh types (RT, IT, RQ and M), participants should use all six different grid sizes ($h = \frac{1}{50}, \frac{1}{100}, \frac{1}{150}, \frac{1}{200}, \frac{1}{250}, \frac{1}{300}$ for Category A., and $h = \frac{1}{50}, \frac{1}{100}, \frac{1}{200}, \frac{1}{300}, \frac{1}{400}, \frac{1}{500}$ for Category B. and C.) and submit their results as follows. Five specific lines are considered as shown in Fig. 3. Specific description for each line is given below. According to the category of each solver, participants should submit both cell-averaged solutions and/or higher-order solutions at equidistant points P_i along five lines. Higher-order solutions are only for those who use the Category A. solvers.

①, ②: $P_i = (x_i, \alpha + \varepsilon)$, where $x_i = \frac{h}{2} + (i-1) * h$, $h = \frac{2}{N}$, $i = 1, \ldots, N$

③, ④, ⑤: $P_i = (\beta + \varepsilon, y_i)$, where $y_i = \frac{h}{2} + (i-1) * h$, $h = \frac{1}{N}$, $i = 1, \ldots, N$

Here, $\varepsilon = 0.0001$ to avoid the overlap with cell interfaces.

①: $(\alpha, N) = (0.4, 8000)$, ②: $(\alpha, N) = (0.7, 8000)$,

③: $(\beta, N) = (0.52, 4000)$, ④: $(\beta, N) = (1.05, 4000)$, ⑤: $(\beta, N) = (1.65, 4000)$.

. Cell-averaged solutions: $\Pi_0 \rho_k|_{P_i}$, where $P_i \in T_k$ for every methods of Category A., B., and C.

. Higher-order solutions: $\Pi_m \rho_k|_{P_i}$, where $P_i \in T_k$ for $\mathbb{P}_m$-approximated methods of Category A.

ρ_k is the density distribution on the cell T_k, and Π_m indicates a projection onto a polynomial space of degree m.

Moreover, participants should submit two contour images of the Schlieren variable given as:

$$Sch = \frac{\ln(1+\|\nabla\rho\|)}{\ln 10}.$$

The first image should cover the entire domain [0, 2]x[0, 1], and the second image should cover the domain of [0.9, 0.12]x[0.33 0.63] for resolving vortex structures. Both images are 50 equally spaced Schlieren contours in grayscale from 0.05 to 2.4, where the darker colors indicate the higher values. Participants should provide cell-averaged values and/or sub-cell distributions. When submitting contour images, specify (cell-averaged and/or sub-cell) contours and describe detailed plotting procedures.

Appendix: Initial Flow Condition

Implement S1 to S6 step-by-step in order to initialize the computational flow fields.

S1. Calculate the downstream conditions (ρ_d, u_d, v_d, p_d) by using the condition for the stationary normal shock and the given upstream conditions $(\rho_u, u_u, v_u, p_u) = (1.0, M_s\sqrt{\gamma}, 0.0, 1.0)$, as follows.

$$\frac{\rho_u}{\rho_d} = \frac{u_d}{u_u} = \frac{2 + (\gamma - 1)M_s^2}{(\gamma + 1)M_s^2}, \qquad \frac{p_d}{p_u} = 1 + \frac{2\gamma}{\gamma + 1}(M_s^2 - 1), \qquad v_d = 0.0,$$

with $M_s = 1.5$ and $\gamma = 1.4$

S2. Initialize the computational domain outside the vortex with the given upstream conditions (ρ_u, u_u, v_u, p_u) and the computed downstream conditions (ρ_d, u_d, v_d, p_d).

S3. Calculate the velocity field $(u_{vor},\ v_{vor})$ inside the vortex by superposing the upstream velocity conditions $(u_u,\ v_u)$ and the tangential velocity field (v_θ) given below. Here, r is the distance from the vortex core $(x_c, y_c) = (0.25, 0.5)$, and $(a, b) = (0.075, 0.175)$ is used. v_m is the maximum tangential velocity, which occurs at $r = a$. We take $M_v = v_m/\sqrt{\gamma}$ as a measure of the vortex strength, and $M_v = 0.9$ in this computation. The vortex rotates counter-clockwise with the angular velocity given below.

$$v_\theta(r) = \begin{cases} v_m \dfrac{r}{a}, & \text{if } r \leq a, \\ v_m \dfrac{a}{a^2 - b^2}\left(r - \dfrac{b^2}{r}\right), & \text{if } a \leq r \leq b, \\ 0, & \text{if } r > b, \end{cases}$$

and therefore

$$u_{vor}(r) = u_u + \text{x} - \text{component of } v_\theta,$$
$$v_{vor}(r) = v_u + \text{y} - \text{component of } v_\theta.$$

S4. In order to calculate the temperature field $(T_{vor}(r))$ inside the vortex, integrate the following ODE obtained from the normal momentum equation with the centripetal force, as follows.

$$\frac{dT_{vor}(r)}{dr} = \frac{\gamma - 1}{R\gamma}\frac{v_\theta(r)^2}{r} \quad \rightarrow \quad \int_r^b \frac{dT_{vor}(r)}{dr} dr = \int_r^b \frac{\gamma - 1}{R\gamma}\frac{v_\theta(r)^2}{r} dr$$

Here, R is the gas constant. The above relation is integrated using $v_\theta(r)$ in S3 with $T_{vor}(b) = T_u$, where the temperature at the upstream state of T_u is determined by the ideal gas law $p = \rho RT$ with $R = 1$.

S5. Using the isentropic relation provided below, calculate the density and the pressure field $(\rho_{vor},\ p_{vor})$ inside the vortex. ρ_u, p_u and T_u are the upstream conditions as references.

$$\rho_{vor}(r) = \rho_u \left(\frac{T_{vor}(r)}{T_u}\right)^{\frac{1}{\gamma-1}}, \qquad p_{vor}(r) = p_u \left(\frac{T_{vor}(r)}{T_u}\right)^{\frac{\gamma}{\gamma-1}}$$

S6. Using the computed density, velocity, and pressure fields, $(\rho_{vor}(r), u_{vor}(r), v_{vor}(r), p_{vor}(r))$, initialize inside the vortex.