INSTITUTE FOR COMPUTATIONAL FLUID DYNAMICS (ICFD)

Benchmark Problems in Fluid Flow Modelling

by

M.F. Webster and P.K. Sweby

Numerical Analysis Report 18/85

KEY WORDS: Benchmarks Compressible flow Incompressible flow
Computational fluid dynamics

This work forms part of the research programme of the Institute for Computational Fluid Dynamics at the Universities of Oxford and Reading.

INSTITUTE FOR COMPUTATIONAL FLUID DYNAMICS (ICFD)

Benchmark Problems in Fluid Flow Modelling

by

M.F.Webster and P.K.Sweby

Numerical Analysis Report 18/85

KEY WORDS: Benchmarks Compressible flow Incompressible flow
Computational fluid dynamics

This work forms part of the research programme of the Institute for Computational Fluid Dynamics at the Universities of Oxford and Reading.

CONTENTS

1 Introduction

This compendium of benchmark problems is mainly the result of an ICFD one day workshop held at Reading University on 27 September 1984. The views expressed there are embodied in the guidelines and problems that follow. The framework adopted is split into two sections. The first contains a list of proposed benchmarks with an annotation to denote those adopted. The second section is in the form of an appendix and contains the full details of those benchmarks of which the ICFD has some experience. Where appropriate, attention is drawn to programs in the ICFD Software Base (denoted SB) which incorporate benchmark problems.

The main speakers at the workshop were N.E.Hoskin(AWRE,Aldermaston), A.G.Hutton(CEGB,Berkeley),P.L.Roe(Cranfield),M.Andrews(I.C.London), C.L.Farmer(AEE,Winfrith) and P.K.Sweby(ICFD,Oxford and Reading). The ICFD is grateful to all the participants at the workshop but particular thanks must go to Dr Hoskin for his continual advice and encouragement in this area.

The first section of this report is organised in the following format. Those benchmarks that are incorporated in the appendix section are assigned a letter and number. Compressible problems are marked with a "C#" and incompressible problems with a "D#". Those benchmarks that do not fall into this catagorisation are absent from the appendix and, where possible, a source of reference is provided for such problems. The format used in the appendix section is self-explanatory.

Clearly there is scope for further benchmark problems. For example, the Industrial Advisory Committee to the ICFD has suggested future topics to cover might include permeable media flows, two-phase flows and flows with dominant source terms. Hence where it is considered there is a need for a benchmark model problem an entry has been made indicated with a status "? ? ?".

First edition.

2 Guidelines

The concensus of opinion of participants at the workshop was that there are certain key features underlying the choice of suitable benchmark model problems. These may be summarised in the following adopted guidelines:

2.1. Within a class of problems there is a need to identify benchmarks that describe simple flow situations. In addition, model problems should be capable of overtesting troublesome constraints and highlight the choice of variables and the statement of the equations.

Particular aspects of model capabilities will depend upon the type of flow problem. A graduated approach adopting a sequence of model problems with an increasing scale of complexity is recommended i.e. (linear → nonlinear),
(scalar → system),
(1-D → 2-D → 3-D).

2.2. Problems should be unambiguously stated in a detailed and precise manner with all the mathematical and numerical parameters specified.

2.3. Ideally an analytic solution should be available to the model problem. Failing this there should at least be some analysis available to establish the behaviour of the solution.

2.4. Model problems should be based upon physically recognisable situations.

2.5. The numerical scheme/solvers should be tested for:
(a) error growth and stability;
(b) efficiency, robustness and relative cost;
(c) grid convergence.

3 Compressible Flow Problems (C).

3.1 1-D Unsteady Flow.

3.1.1 Plane Geometry : Scalar.

3.1.1.1 Linear constant coefficient advection (C1)

Equation: $u_t + u_x = 0$

Data: (a) Square pulse.

(b) Sin^2 pulse. (Many use Gaussian as alternative)

3.1.1.2 Linear variable coefficient advection (C2)

Equation: $u_t + a(x)u_x = 0$

Data: as above.

3.1.1.3 Inviscid Burgers' equation (C3)

Equation: $u_t + (1/2\ u^2)_x = 0$

Data: (a) Shifted square pulse.

(b) Unshifted $\sin^2$ pulse.

3.1.2 Plane Geometry : systems

3.1.2.1 Euler equations (Conservation form)

$$\begin{pmatrix} \rho \\ \rho u \\ e \end{pmatrix}_t + \begin{pmatrix} \rho u \\ p + \rho u^2 \\ u(e + p) \end{pmatrix}_x = 0$$

3.1.2.1.1 Single shock (C4)

Should be tested on both a uniform and a variable grid.

3.1.2.1.2 Single expansion (backward-facing shock data) (C5)

3.1.2.1.3 Open shock tube (C6)

(a) Sod's Problem.

(b) Weak Shock & Strong Expansion.

3.1.2.1.4 **Blast Tube Problem (C7)**

3.1.2.1.5 **Constant acceleration piston**

Ref: Courant and Friedrichs, "Supersonic Flow and Shock Waves", Springer (1976).
Objective: To demonstrate accuracy of shock formation and subsequent modelling.

3.1.3 **Polar Geometry : systems**

3.1.3.1 **Shock reflected from origin (C8)**

Both cylindrical and spherical formulation of equations.

3.2 **2-D Unsteady Flow**

3.2.1 **Plane Geometry : scalar**

3.2.1.1 **1-D model problems on non-aligned 2-D mesh**

3.2.1.1.1 **Linear constant coefficient advection**

Refer to 1-D problem C1.

3.2.1.1.2 **Inviscid Burgers' equation**

Refer to 1-D problem C3.

3.2.1.2 **Variable coefficient 2-D advection (C9)**

(a) Rotating cone.
(b) Rotating key.

3.2.1.3 **Inviscid Burgers' equation (C10)**

Equation: $\underline{u}_t + \underline{\nabla}(\frac{1}{2}|\underline{u}|^2) = 0$.

Data: (a) Cone.
(b) Key.

3.2.2 **Plane Geometry : systems**

3.2.2.1 **Euler Equations (conservation form)**

3.2.2.1.1 **1-D model problems on non-aligned 2-D mesh**

3.2.2.1.1.1 **Single plane shock**

Refer to 1-D problem C4.

3.2.2.1.1.2 **Single expansion**

Refer to 1-D problem C5.

3.2.2.1.1.3 **Open shock tube**

Refer to 1-D problem C6.

3.2.2.1.1.4 **Blast Tube Problem**

Refer to 1-D problem C7.

3.2.2.1.2 **Mach 3 Wind Tunnel with Forward facing step (C11)**

3.2.2.1.3 **Double Mach reflection of a strong shock**

Ref: Courant and Friedrichs, "Supersonic Flow and Shock Waves", Springer (1976).
Objective: Representation of shock reflection and shock/contact discontinuity interactions in 2-D.

3.2.2.1.4 **Richtmyer-Meshkov instability problem**

Ref: D.L.Youngs, Physics 12 D (1984).
Objective: Representation of non-planar shock.

3.2.2.1.5 **Spherical blast wave (Sedov problem) in (x,y)**

Ref:Los Alamos Report, LA-10112-C, "Workshop on Accurate, Monotonic Methods for Multi-dimensional Rezoners", Asilomar State Park, Monterey, California, June(1983).
Objective: Representation of shock discontinuity and maintenance of spherical symmetry.

3.2.3 **Polar geometry :systems**

3.2.3.1 **Propagation of a plane shock through an irregular 2-D grid**

Ref: AWRE Aldermaston (D.L.Youngs).
Objective: Test extension of difference methods to irregular grids (possibly non-orthogonal).

3.2.3.2 **Spherical shell**

Ref: AWRE Aldermaston (D.L.Youngs).
(a) Constant applied inward radial velocity.
Objective: To test maintenance of spherical symmetry.
(b) Translational velocity (in (r,θ)).
Objective: Test accuracy of 2-D (r,θ) calculations.

3.2.3.3 Spherical blast wave (Sedov problem) in (r,θ)

Ref: Asilomar workshop (1983).
Objective: Representation of shock discontinuity and maintenance of spherical symmetry.

3.3 1-D Steady Flow

3.3.1 Plane Geometry : systems

3.3.1.1 Euler equations (conservation form)

3.3.1.1.1 Laval Nozzle problem (C12)

3.4 2-D Steady Flow

3.4.1 Plane Geometry : systems

3.4.1.1 Euler equations

3.4.1.1.1 Channel flow over a bump (C13)

3.4.1.1.2 Flow over NACA0012 aerofoil

Ref: M.F.Paisley, University of Oxford.

4 Incompressible Flow Problems (D).

4.1 Laminar Flow.

4.1.1 Linear advection of a rotating cone in an incompressible velocity field(D1)

Equation: $m_t + \underline{u}.\underline{\nabla} m = 0$

fluid property (mass) m,

velocity field $\underline{u}$ (given).

4.1.2 Diffusion-convection transport in an incompressible velocity field (D2).

Equation: $\underline{u}.\underline{\nabla} T - Pe^{-1}\nabla^2 T = 0$

temperature T,

velocity field $\underline{u}$ (given),

Peclet number Pe.

4.1.3 Shallow water equations.

4.1.3.1 2-D Problems.

Equations: $\varphi_t + \nabla(\underline{u}\varphi) = 0$

$\underline{u}_t + \underline{\nabla}(\underline{uu}) + f\underline{v} + \nabla\varphi = 0$

velocity field $\underline{u} = (u_1, u_2)$,

velocity field $\underline{v} = (-u_2, u_1)$,

geopotential φ,

coreolis force f.

4.1.3.1.1 Grammeltvedt problem

Status: ? ? ?
Ref: A.Grammeltvedt,"A survey of finite-difference schemes for the primitive equations for a barotropic fluid",Monthly Weather Review,V97,no5,(1969).

4.1.4 St. Venant Equations.

4.1.4.1 1-D Problems.

Equations: $A_t + Q_x = 0$

$$Q_t + (Q^2/A)_x + gA(h_x + (Q|Q|)/K^2) = 0$$

total mass flow Q,
cross-sectional area A,
river surface height h,
friction parameter K.

4.1.4.1.1 River Flow problem (D3)

4.1.4.2 2-D Problems.

4.1.4.2.1 Constricted channel

Status: ? ? ?
Ref: P.Samuels,HR,Wallingford.

4.1.5 Navier-Stokes equations

$$\nabla.\underline{u} = 0$$

$$\underline{u}_t + \underline{u}.\underline{\nabla u} - Re^{-1}\nabla^2\underline{u} + \underline{\nabla}p = \underline{f}$$

velocity field $\underline{u}$,

pressure field p,

body force vector $\underline{f}$ (assumed zero),

Reynolds number $Re = \varrho UD/\mu$.

4.1.5.1 Driven cavity flow

Ref: T.J.Chung,"Finite Element Analysis in Fluid Dynamics", McGraw-Hill,(1978).
Objective: To solve 2-D Navier-Stokes equations for the driven cavity problem over a range of Re.

4.1.5.2 Plane channel flow over a forward-facing step (D4)

4.1.5.3 Plane channel flow over a backward-facing step (D5)

4.1.5.4 Plane channel flow over a symmetrical step constriction (D6)

4.1.5.5 Vortex shedding behind a cylinder

Ref: Gresho et al. I.J.Num.Meth.Fluids,V4,(1984).
Objective: To solve 2-D Navier-Stokes equations for the flow over a cylinder in a channel for a range of Re.

4.1.6 Steady Buoyancy Influenced Flows.

4.1.6.1 Mixed convection.

Equations: $\underline{\nabla}.\underline{u} = 0$

$$\underline{u}.\underline{\nabla u} - Re^{-1}\nabla^2\underline{u} + \underline{\nabla}p = \frac{Gr}{Re^2}T$$

$$\underline{u}.\underline{\nabla}T - Pe^{-1}\nabla^2 T = 0$$

Grashof number $Gr = g\varrho^2\beta(T_1 - T_0)D^3/\mu^2$,

Peclet number $Pe = \varrho c_p UD/k$,

T_1, T_0 reference temperatures.

4.1.6.1.1 Convection in an infinite slot with transpiration across a wall

Ref: CEGB,Berkeley (A.G.Hutton).
Objective: Comparison of numerical and theoretical solutions.

4.1.6.2 Natural convection.

Equations: $\underline{\nabla}.\underline{u} = 0$

$$\underline{u}.\underline{\nabla u} - Pr\nabla^2\underline{u} + \underline{\nabla}p = RaPrT$$

$$\underline{u}.\underline{\nabla}T - \nabla^2 T = 0$$

Rayleigh number $Ra = g\beta(T_1 - T_0)D^3/(\nu\kappa)$,

Prandtl number $Pr = \nu/\kappa$,

kinematic viscosity $\nu = \mu/\varrho$,

thermal diffusivity $\kappa = k/\varrho c_p$,

T_1, T_0 reference temperatures.

4.1.6.2.1 Natural convection in a square cavity

Ref: G.De Vahl Davis and I.P.Jones, I.J.Num.Meth.Fluids,V3,no3 (1983).
Objective: To solve steady 2-D natural convection flow of a Boussinesq fluid for a range of Ra (10^3,10^4,10^5,10^6).

4.1.6.2.2 **Steady natural convection in a porous media in an annular cavity**

Ref: A.Castrejon,Report no.PDR/CFJU IC/9,Comp.Fluid Dyn.Unit,Imp.Coll. London (1983).
Objective: To solve steady 2-D natural convection flow in porous media in an annular cavity for Ra (10^3,10^4,10^5,10^6).

Appendix I

Details of Specimen Compressible Flow Problems

C1. 1-D Linear constant coefficient advection.

Equation: $u_t + u_x = 0$

Computational domain: [0,1]

Mesh spacing: $\Delta x = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.1 \\ 0.5 \\ 0.9 \end{cases}$

Boundary conditions: Periodic

Output time: T = 6.12

Initial data: a) Square pulse:

$$u_0 = \begin{cases} 0 & x < 0.25 \\ 1 & x \in [0.25,0.5] \\ 0 & x > 0.5 \end{cases}$$

b) Sine squared pulse:

$$u_0 = \begin{cases} 0 & x < 0.25 \\ \sin^2 \pi(4x-1.5) & x \in [0.25,0.5] \\ 0 & x > 0.5 \end{cases}$$

Notes:

Initial data should be projected on the computational grid via the trapezium rule approximation to

$$U_k^0 = \frac{1}{\Delta x} \int_{x_k-\Delta x/2}^{x_k+\Delta x/2} u_0(x)\, dx$$

namely

$$U_k^0 = \frac{u_0(x_{k+1/2}) + u_0(x_{k-1/2})}{2\Delta x}$$

C2. 1-D Linear variable coefficient advection.

Equation: $u_t + a(x)u_x = 0$

Propagation speed a(x): $(1 + x^2)^{-1}$

Computational domain: [0,1]

Mesh spacing: $\Delta x = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.1 \\ 0.5 \\ 0.9 \end{cases}$

Boundary conditions: Periodic

Output time: T = 6.12

Initial data: a) Square pulse:

$$u_0 = \begin{cases} 0 & x < 0.25 \\ 1 & x \in [0.25, 0.5] \\ 0 & x > 0.5 \end{cases}$$

b) Sine squared pulse:

$$u_0 = \begin{cases} 0 & x < 0.25 \\ \sin^2 \pi(4x-1.5) & x \in [0.25, 0.5] \\ 0 & x > 0.5 \end{cases}$$

Notes:

Initial data should be projected on the computational grid via the trapezium rule approximation to

$$U_k^0 = \frac{1}{\Delta x} \int_{x_k-\Delta x/2}^{x_k+\Delta x/2} u_0(x)\, dx$$

namely

$$U_k^0 = \frac{u_0(x_{k+1/2}) + u_0(x_{k-1/2})}{2\Delta x}$$

C3. 1-D Inviscid Burgers' equation.

Equation: $u_t + (\frac{1}{2}u^2)_x = 0$

Computational domain: $[0,1]$

Mesh spacing: $\Delta x = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.1 \\ 0.5 \\ 0.9 \end{cases}$

Boundary conditions: Cauchy

Initial data: a) Shifted square pulse:

$$u_0 = \begin{cases} -0.25 & x < 0.25 \\ 0.75 & x \in [0.25,0.5] \\ -0.25 & x > 0.5 \end{cases}$$

Output time: T = 0.2 & 1.4

b) Sine squared pulse:

$$u_0 = \begin{cases} 0 & x < 0.25 \\ \sin^2 \pi(4x-1.5) & x \in [0.25,0.5] \\ 0 & x > 0.5 \end{cases}$$

Output time: T = 0.1 & 0.7

Notes:

Initial data should be projected on the computational grid via the trapezium rule approximation to

$$U_k^0 = \frac{1}{\Delta x} \int_{x_k-\Delta x/2}^{x_k+\Delta x/2} u_0(x)\, dx$$

namely

$$U_k^0 = \frac{u_0(x_{k+1/2}) + u_0(x_{k-1/2})}{2\Delta x}$$

C4. 1-D Euler equations: single shock.

Equations:

$$\begin{bmatrix} \rho \\ \rho u \\ e \end{bmatrix}_t + \begin{bmatrix} \rho u \\ p + \rho u^2 \\ u(e + p) \end{bmatrix}_x = 0$$

Gas constant: $\gamma = 1.4$

Computational domain: [0,1] (1-D shock tube)

Mesh spacing: $\Delta x = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \\ \text{Geometric grid with initial} \\ \text{spacing above and ratio 1.05} \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.15 \\ 0.30 \end{cases}$

Boundary conditions: Transparent at both ends.

Output time: $T = 0.144$

Initial data:

$$\begin{bmatrix} \rho \\ \rho u \\ e \end{bmatrix} = \begin{cases} \begin{bmatrix} 1.00000 \\ 1.59099 \\ 3.76563 \end{bmatrix} & x < 0.5 \\ \begin{bmatrix} 0.26230 \\ 0.00000 \\ 0.25000 \end{bmatrix} & x > 0.5 \end{cases}$$

C5. 1-D Euler equations: single expansion.

Equations:

$$\begin{pmatrix} \rho \\ \rho u \\ e \end{pmatrix}_t + \begin{pmatrix} \rho u \\ p + \rho u^2 \\ u(e + p) \end{pmatrix}_x = 0$$

Gas constant: $\gamma = 1.4$

Computational domain: [0,1] (1-D shock tube)

Mesh spacing: $\Delta x = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \\ \text{Geometric grid with initial} \\ \text{spacing above and ratio 1.05} \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.15 \\ 0.30 \end{cases}$

Boundary conditions: Transparent at both ends

Output time: $T = 0.144$

Initial data:

$$\begin{pmatrix} \rho \\ \rho u \\ e \end{pmatrix} = \begin{cases} \begin{pmatrix} 1.00000 \\ 0.18820 \\ 2.51771 \end{pmatrix} & x < 0.5 \\[2ex] \begin{pmatrix} 0.19307 \\ 1.84656 \\ 0.57916 \end{pmatrix} & x > 0.5 \end{cases}$$

C6. 1-D Euler equations: Open Shock Tube.

Equations:

$$\begin{bmatrix} \rho \\ \rho u \\ e \end{bmatrix}_t + \begin{bmatrix} \rho u \\ p + \rho u^2 \\ u(e + p) \end{bmatrix}_x = 0$$

Gas constant: (a) $\gamma = 1.4$
(b) $\gamma = 5/3$

Computational domain: [0,1] (1-D shock tube)

Mesh spacing: $\Delta x = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \\ \text{Geometric grid with initial spacing above and ratio 1.05} \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.15 \\ 0.30 \end{cases}$

Boundary conditions: Transparent at both ends

Output time: (a) T = 0.144
(b) T = 0.018

Initial data:

$$\text{(a)} \quad \begin{bmatrix} \rho \\ \rho u \\ e \end{bmatrix} = \begin{cases} \begin{bmatrix} 1.00000 \\ 0.00000 \\ 2.50000 \end{bmatrix} & x < 0.5 \\[2em] \begin{bmatrix} 0.12500 \\ 0.00000 \\ 0.25000 \end{bmatrix} & x > 0.5 \end{cases}$$

$$\text{(b)} \quad \begin{bmatrix} \rho \\ \rho u \\ e \end{bmatrix} = \begin{cases} \begin{bmatrix} 1.00000 \\ 0.00000 \\ 10000.0 \end{bmatrix} & x < 1/3 \\[2em] \begin{bmatrix} 0.00100 \\ 0.00000 \\ 0.01000 \end{bmatrix} & x > 1/3 \end{cases}$$

C7. 1-D Euler equations: Blast Tube Problem.

Equations:

$$\begin{bmatrix} \rho \\ \rho u \\ e \end{bmatrix}_t + \begin{bmatrix} \rho u \\ p + \rho u^2 \\ u(e+p) \end{bmatrix}_x = 0$$

Gas constant: $\gamma = 1.4$

Computational domain: [0,1] (1-D shock tube)

Mesh spacing: $\Delta x = \begin{cases} 0.00500 \\ 0.00250 \\ 0.00125 \\ \text{Geometric grid with initial spacing above and ratio 1.05} \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.015 \\ 0.030 \end{cases}$

Boundary conditions: Neumann at both ends

Output times: T = 0.01, 0.016, 0.026, 0.028, 0.030, 0.032, 0.034, 0.038

Initial data:

$$\begin{bmatrix} \rho \\ \rho u \\ e \end{bmatrix} = \begin{cases} \begin{bmatrix} 1.00000 \\ 0.00000 \\ 2500.00 \end{bmatrix} & x < 0.1 \\ \begin{bmatrix} 1.00000 \\ 0.00000 \\ 0.02500 \end{bmatrix} & .1 < x < .9 \\ \begin{bmatrix} 1.00000 \\ 0.00000 \\ 250.000 \end{bmatrix} & x > 0.9 \end{cases}$$

C8. 1-D systems, Polar geometry: Shock reflected from origin.

Equations:

$$\begin{bmatrix} \varrho \\ \varrho u \\ e \end{bmatrix}_t + r^{-2}\left[r^2 \begin{bmatrix} \varrho u \\ p + \varrho u^2 \\ u(e + p) \end{bmatrix}\right]_r = \begin{bmatrix} 0 \\ 2p/r \\ 0 \end{bmatrix}$$

Gas constant: $\gamma = 5/3$

Computational domain: [0,1]

Mesh spacing: $\Delta r = 0.01$

Mesh ratio, $\lambda = \Delta t/\Delta r$: $\lambda = 0.6$

Boundary conditions: Neumann conditions at origin

Dirichlet conditions at r = 1:

$$\begin{bmatrix} \varrho \\ \varrho u \\ e \end{bmatrix} = \begin{bmatrix} (1+t)^2 \\ -(1+t)^2 \\ \frac{1}{2}(1+t)^2 \end{bmatrix}$$

Output time: T = 0.6

Initial data:

$$\begin{bmatrix} \varrho \\ \varrho u \\ e \end{bmatrix} = \begin{bmatrix} 1 \\ -1 \\ .5 \end{bmatrix}$$

C9. Variable coefficient 2-D advection.

Equation: $u_t + a(x,y)u_x + b(x,y)u_y = 0$

Propagation speeds : $a(x,y) = -\cos(\theta)$
$b(x,y) = \sin(\theta)$

where $\theta = \tan^{-1}((y-.5)/(x-.5))$

Computational domain: $[0,1]x[0,1]$

Mesh spacing: $\Delta x,\ \Delta y = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.1 \\ 0.5 \\ 0.9 \end{cases}$

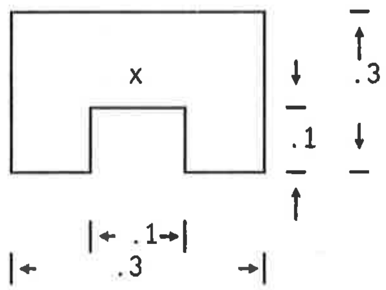
Boundary conditions: Periodic

Output time: $T = \pi/2$ and 5π

Initial data:

a) Circular Cone, base radius .15, unit height at apex, centered at (.75,.75) initially

b) Key, unit height, centered at (.75,.75) initially



C10. 2-D Inviscid Burgers' Equation.

Equation: $\mathbf{u}_t + \nabla(\frac{1}{2}|\mathbf{u}|^2) = 0.$

where $\mathbf{u} = (u,v)^T$

Computational domain: [0,1]x[0,1]

Mesh spacing: $\Delta x,\ \Delta y = \begin{cases} 0.01 \\ 0.02 \\ 0.04 \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta x$: $\lambda = \begin{cases} 0.1 \\ 0.5 \\ 0.9 \end{cases}$

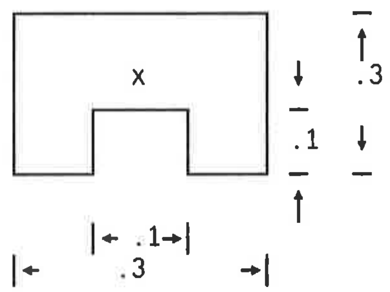
Boundary conditions: Periodic

Output time: $T = \pi/2$ and 5π

Initial data:

a) Circular Cone, base radius .15, unit height at apex, centered at (.75,.75) initially

b) Key, unit height, centered at (.75,.75) initially



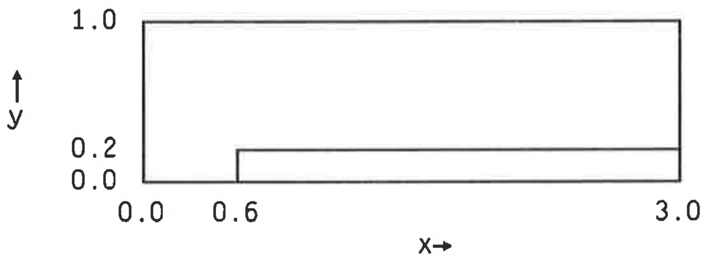
C11. 2-D Euler Equations: Mach 3 Wind Tunnel with Step.

Equations:

$$\begin{bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{bmatrix}_t + \begin{bmatrix} \rho u \\ p + \rho u^2 \\ \rho uv \\ u(e + p) \end{bmatrix}_x + \begin{bmatrix} \rho v \\ \rho uv \\ p + \rho v^2 \\ v(e + p) \end{bmatrix}_y = 0$$

Gas constant: $\gamma = 1.4$

Computational domain:



Mesh spacing: $\Delta = \Delta x = \Delta y = \begin{cases} .0500 \\ .0250 \\ .0125 \end{cases}$

Mesh ratio, $\lambda = \Delta t/\Delta$: $\lambda = .4$

Boundary conditions: Neumann on $y = 1$
$y = 0,\ 0 < x < .6$
$y = .2,\ .6 < x < 3$

Initial conditions (inflow) on x=0

Transparent (outflow) on x=3

Output : Density contours

Output times: T = 1.0,1.5,2.0,2.5,3.0,3.5,4.0

Initial data:

$$\begin{bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{bmatrix} = \begin{bmatrix} 1.4 \\ 4.2 \\ 0.0 \\ 8.8 \end{bmatrix}$$

C12. 1-D Steady Flow : Euler Equations - Laval Nozzle.

Equations:

$$\varrho_t + (\varrho u)_x = 0$$

$$(\varrho u)_t + (p+\varrho u^2)_x - (p/A)A_x = 0$$

with

$$p = \varrho/\gamma(1-\tfrac{1}{2}(\gamma-1)u^2)$$

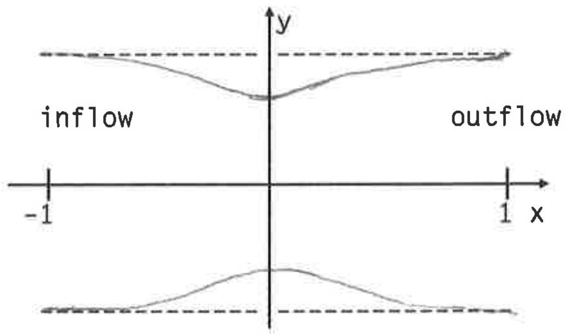
where $A = A(x)$ is the cross-sectional area of the nozzle

p, ϱ are the physical pressure and density multiplied by the cross-sectional area

Computational Domain: $x \in [-1,1]$

Nozzle geometry: $y(x) = 1 - 0.1(1+\cos\pi x)$ $\quad -1 \leq x \leq 1$

$A(x) = \pi y(x)^2$



Mesh: 16 or 32 cells, with those at outflow being 3 times those at throat

Boundary conditions: subsonic inflow $p = (\varrho/\gamma)^{\gamma}$

subsonic outflow

$$p = p_\infty = [1+\tfrac{1}{2}(\gamma+1)M_\infty^2]^{(\gamma/1-\gamma)}$$

Cases: $M_\infty = 0.4$ subcritical

$M_\infty = 0.6$ supercritical

Output: Mach number distribution $M = u/a$ $(a^2=1-\tfrac{1}{2}(\gamma-1)u^2)$

Analytic Solution: Liepmann & Roshko

C13. 2-D Steady Flow : Euler Equations - Channel flow over a bump.

Equations:

$$\rho_t + (\rho u)_x + (\rho v)_y = 0$$

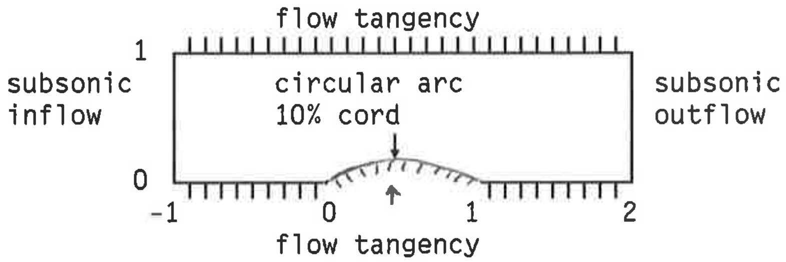
$$(\rho u)_t + (p+\rho u^2)_x + (\rho uv)_y = 0$$

$$(\rho v)_t + (\rho uv)_x + (p+\rho v^2)_y = 0$$

with

$$p = \rho/\gamma(1-\tfrac{1}{2}(\gamma-1)u^2+v^2)$$

Computational Domain:



Mesh: 64 x 16 cells ratio 1:3 away from bump

Boundary conditions:

subsonic inflow $p = (\rho/\gamma)^{\gamma}$
$v = 0$

subsonic outflow

$$p = p_\infty = [1+\tfrac{1}{2}(\gamma+1)M_\infty^2]^{(\gamma/1-\gamma)}$$
$$v = 0$$

flow tangency elsewhere

Cases:

$M_\infty = 0.5$ subcritical
flow symmetrical

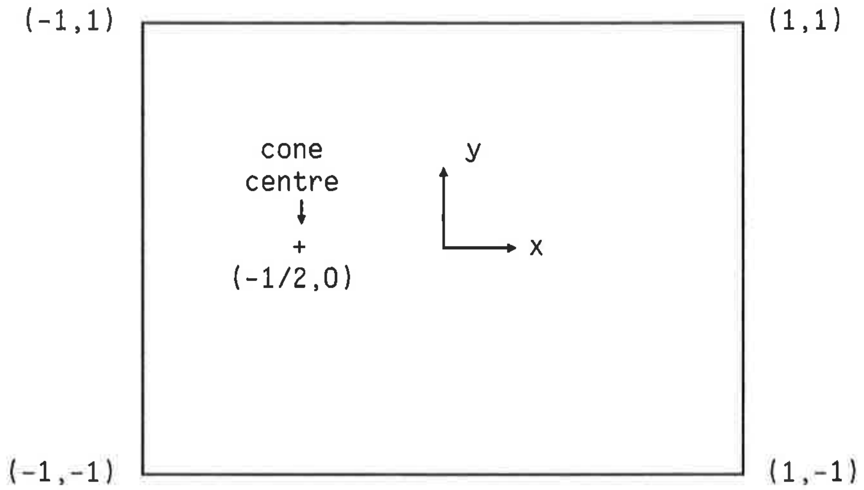
$M_\infty = 0.675$ supercritical
flow not symmetric

Output: Mach number distribution $M = u/a$ along bottom wall
$(a^2=1-\tfrac{1}{2}(\gamma-1)(u^2+v^2))$

Appendix II

Details of Specimen Incompressible Flow Problems

D1. Linear advection of a rotating cone in an incompressible velocity field.



Objective: Accurate numerical prediction of linear advection over a long time scale in an l_2-sense.

Geometry: $[-1 \leq x \leq 1]$, $[-1 \leq y \leq 1]$.

Boundary conditions: Periodic.

Output times: After first half revolution and any integral number thereafter.

Output: Plot fluid property m, with m_{max} and m_{min}, and l_2-error.

Initial data: Cone, centred at (-1/2,0) where r = 0, then
(a)linear straight-sided: $m = 0$ for $r > 1/4$, $m = 1 - 4r$ for $r \leq 1/4$;

(b)Sin^2: $m = 0$ for $r > 1/4$, $m = Sin^2 4\pi r$ for $r \leq 1/4$;

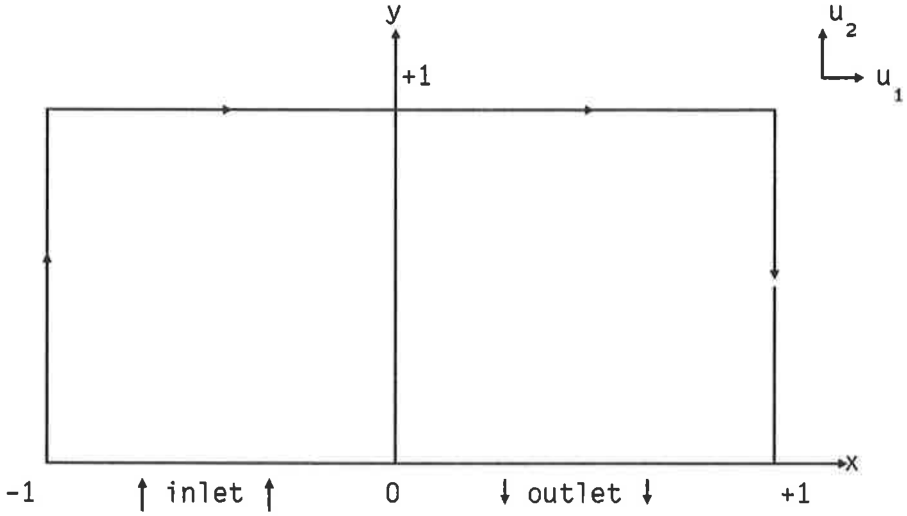
(c)Gaussian: $m = \exp(-20r^2)$ (smooth).

Time step: $\Delta t = 0.02$ (i.e. 50 time steps per revolution).

Mesh choices: Finite element shape functions
(a) 1600 Bilinears on rectangles,
(b) 2048 Linears on triangles.

Ref: A.Priestley and K.W.Morton,"On Characteristic Galerkin and Lagrange-Galerkin Methods",Oxford University Rep.No.85/9,(1986).

D2. Diffusion-convection transport in an incompressible velocity field.



Objective: Solve the 2-D transport equation for the passive scalar T in the rectangular domain.

Domain: $[-1 \le x \le 1, 0 \le y \le 1]$.

Specified: velocity field $\underline{u} = (u_1, u_2)$ as

$$u_1 = 2y(1-x^2) \quad \text{and} \quad u_2 = -2x(1-y^2).$$

Boundary conditions: On all but outlet

$$T = 1 + \tanh[\alpha(2x + 1)] \quad \text{on } y = 0,\ -1 \le x \le 0,$$

$$T = (1 - \tanh\alpha) \text{ on } \begin{cases} x = -1,\ 0 < y < 1, \\ y = 1,\ 0 \le x \le 1, \\ x = 1,\ 0 \le y < 1, \end{cases} \quad \text{where } \alpha = 10.$$

At outlet $y = 0$, $0 < x < 1$ any appropriate treatment e.g. $\frac{\partial T}{\partial n} = 0$.

Required: A graphical comparison between the T-profile and the calculated outlet profile for Pe = 10,100,500,10^3 and 10^6. These results should also be given in tabular form at x-increments of 0.1. At least one complete set of results should be for a regular mesh.

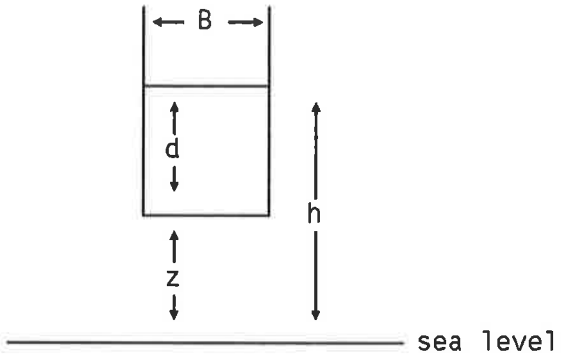
Ref: R.M.Smith and A.G.Hutton, I.J.Num.Heat Trans., V5, pp439-461, (1982).

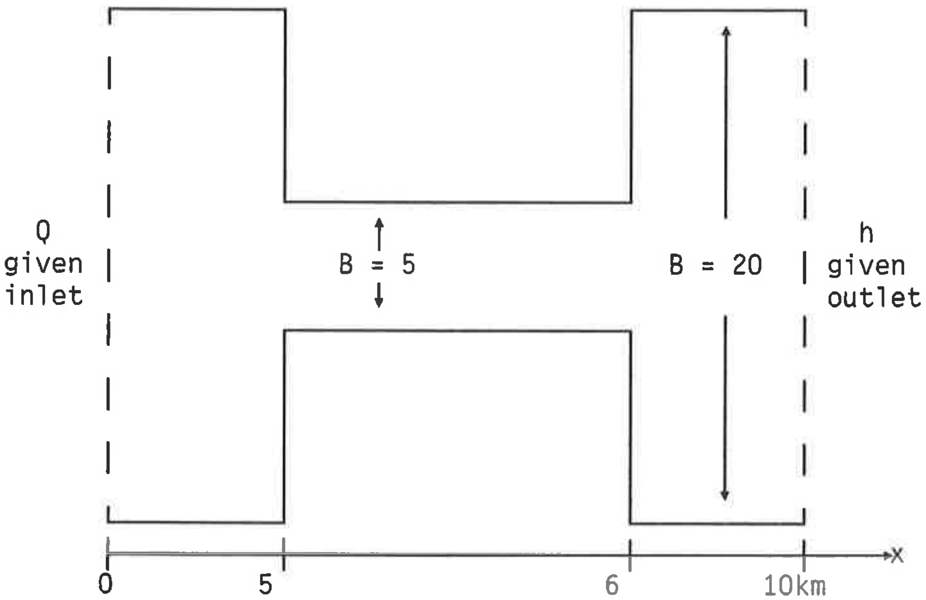
SB Program: PETGAL (Author B.W.Scotney).

Appendix II

D3. River flow problem.

Cross-sectional view





Vertical view

Objective: (a) To obtain a solution for large β;
(b) To observe the effect of the constriction on the flow;
(c) To capture essence of solution on a coarse mesh.

Variables:

Total mass flow	Q	units	m^3s^{-1}
cross-sectional area	A		m^2
river breadth	B		m
river surface height	h=z+d		m
bottom height	z		m
depth	d		m
space variable	x		km
time	t		ks.

Geometry: $[0 \leq x \leq 10]$, $[0 \leq t \leq 86.4]$.
$z = \beta(10-x)$, with slope β where $0.1 \leq \beta \leq 4$.
B = {5m for $5 \leq x \leq 6$, 20m otherwise}.
A = Bd (flat bottom, vertical sided channel).

Boundary conditions: $h = 4m$ at $x = 10$.

At $x = 0$, Q rises linearly from 20 to $200 m^3 s^{-1}$ over
(a)12 hours and then falls off over 12 hours,
or (b) 2 hours and then falls off over 12 hours.

Output times: Every 2 hours.

Output: Q, h, velocity Q/A.

Initial data: $d = 4$ for all x.

$Q = 20$ for all x.

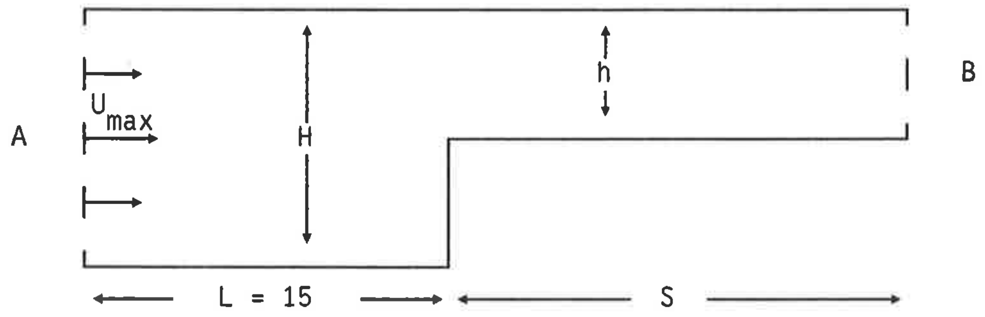
K^2 chosen such that $h_x + (Q|Q|)/K^2 = 0$

initially and is fixed for all time.

Mesh sizes: (a) $\Delta x = 1km$, $\Delta t = 2.4ks$ (Coarse);
(b) $\Delta x = 100m$, $\Delta t = 0.3ks$ (Fine).

Ref: A.Priestley,"Numerical Approximation of Nonlinear Hyperbolic equations",MSc Dissertation,University of Oxford,(1984).

D4. Plane channel flow over a forward-facing step.



Objective: To find a steady-state solution of the Navier-Stokes equations for laminar flow over a forward-facing step.

Boundary conditions: A fully-developed laminar flow profile is assumed at the inlet (A). Any appropriate outlet boundary condition may be applied.

Geometry: L = 15 fixed. S variable.
H = 1.0 and h = 0.5.

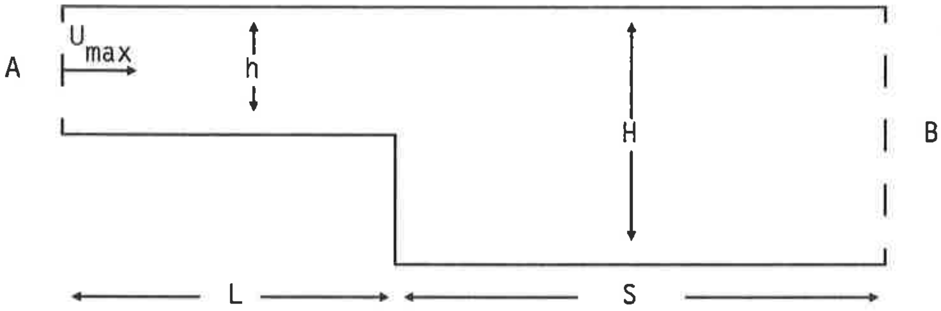
Reynolds Number: defined as $Re = \varrho U_{max} H/\mu$.

Required: Profiles at x = 5,10,15,20,25 for Re = 50,150,500.

Ref: R.M.Smith,"Report on VI Meeting IAHR Working Group on Refined Modelling of Flows",Kernforschung,Karlsruhe,March(1983).
CEGB,UK Rep.Nos. TRRD/B/PS/271/M83,TPRD/B/PS/292/M83.

SB Program: FDVORST (Author M.F.Webster).

D5. Plane channel flow over a backward-facing step.



Objective and boundary conditions: Similar to D4.

Geometry: L = 3 fixed. S = 19 fixed.
(a) H = 1.0 and h = 0.5, (b) H = 1.5 and h = 1.0.

Calculations: Cases (a) and (b) for Re = 50,150.

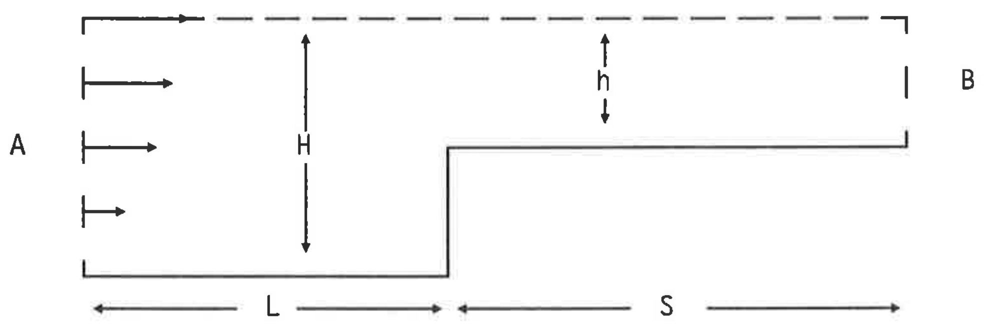
Reynolds Number: defined as $Re = \varrho U_{max}(H-h)/\mu$.

Required: At steady-state
- (a) streamline plots, with enlargement in recirculation zone;
- (b) pressure level plots;
- (c) max and min u_i along lines x = 0.8,2,4,8,12;
- (d) wall shear stress τ_w and $\int_0^x \tau_w dx$ at x as in (c);
- (e) state computer type, storage required, CPU and real time used, and number of unknowns;
- (f) indicate increase of computational effort required to improve accuracy e.g. present results for two different computational grids that are compatible with the physical problem.

Ref: GAMM Workshop,"Notes on Numerical Fluid Mechanics",Vieweg-Verlag Series,Nice,Jan.(1983).

SB Program: FDVORST (Author M.F.Webster).

D6. Plane channel flow over a symmetrical step constriction.



Objective: Similar to D4.

Boundary conditions: Similar to D6 with the additional symmetry conditions on the channel centreline.

Geometry: L = 2, S = 2.
H = 1.0 and h = 0.5.

Mesh sizes: $\Delta x = \Delta y = \{1/10, 1/20, 1/40, 1/60, 1/80\}$.

Reynolds Number: defined as $Re = \rho U_{mean} H/\mu$.

Required: Streamline plots for Re =0,10,50,100,500,2000.

Ref: S.C.R.Dennis and F.T.Smith,"Steady flow through a channel with a symmetrical constriction in the form of a step",Proc.R.Soc.Lond., A372,pp393-414,(1980).

SB Program: FDVORST (Author M.F.Webster).