Abaqus/CFD – Sample Problems

Abaqus 6.10

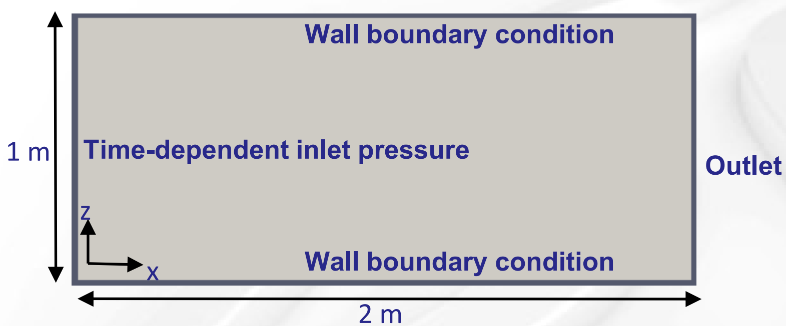
Contents

This document provides a set of sample problems that can be used as a starting point to perform rigorous verification and validation studies. The associated Python scripts that can be used to create the Abaqus/CAE database and associated input files are provided.

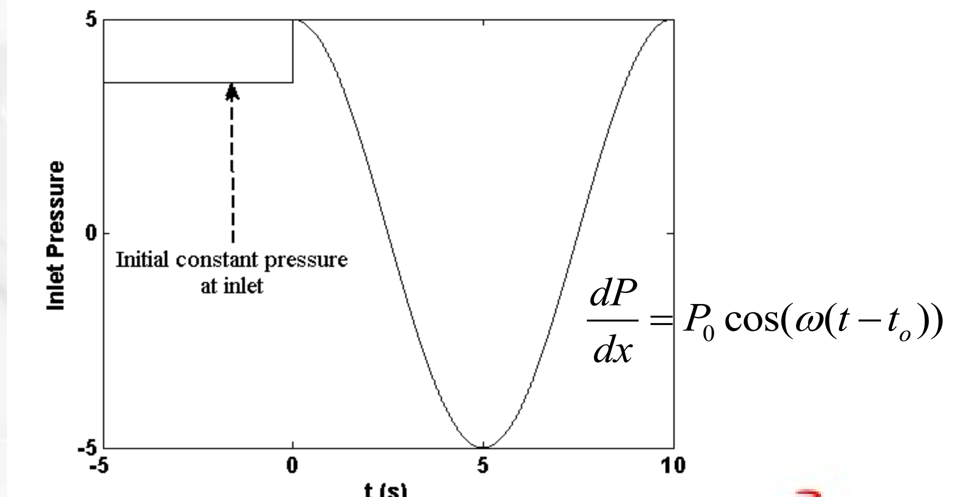
1. Oscillatory Laminar Plane Poiseuille Flow

Oscillatory Laminar Plane Poiseuille Flow

- **Overview**
  - This example compares the prediction of the time-dependent velocity profile in a channel subjected to an oscillatory pressure gradient to the analytical solution.
- **Problem description**
  - A rectangular 2-dimensional channel of width = 1m and length = 2m is considered. An oscillatory pressure gradient (with zero mean) is imposed at the inlet. The analysis is carried out in two steps. In the first analysis step, a constant pressure gradient is prescribed for the first 5 seconds of the simulation to initialize the velocity field to match that of the analytical steady-state solution. In the second analysis step, the flow is subjected to an oscillatory pressure gradient. A 40x20 uniform mesh is used for this problem. Two dimensional geometry is modeled as three dimensional with one element in thickness direction.



Schematic of the geometry used



Inlet pressure profile

Oscillatory Laminar Plane Poiseuille Flow

- **Features**
  - Laminar flow
  - Time-dependent pressure inlet
  - Multi-step analysis
- **Boundary conditions**
  - Pressure inlet
    - $t < t_o$: p = 7.024
    - $t > t_o$: $p = 10 \cdot \text{Cos}(\omega(t-t_o))$; $t_o = 5$, $\omega = \pi/5$
  - Pressure outlet (p = 0)
  - No-slip wall boundary condition on top and bottom (***V*** = 0)
- **Analytical solution**

$$\frac{dP}{dx} = -P_0 e^{i\omega t}$$

$$u(y,t) = \mathrm{Re}\left(\frac{P_o \ell_s^2}{2i\nu} e^{i\omega t}\left[1 - \frac{\cos(\kappa z - \kappa h)}{\cos(\kappa h)}\right]\right)$$

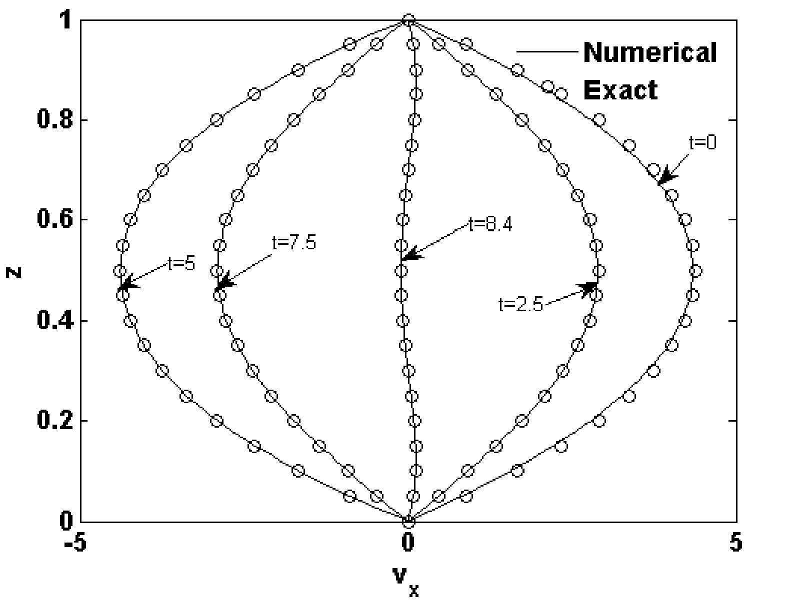
$$\kappa = \sqrt{\frac{-i\omega}{\nu}} \qquad \ell_s = \sqrt{\frac{2\nu}{\omega}}$$

- *h* is the half-channel width
- P_o is the amplitude of pressure gradient oscillation
- ω is the circular frequency

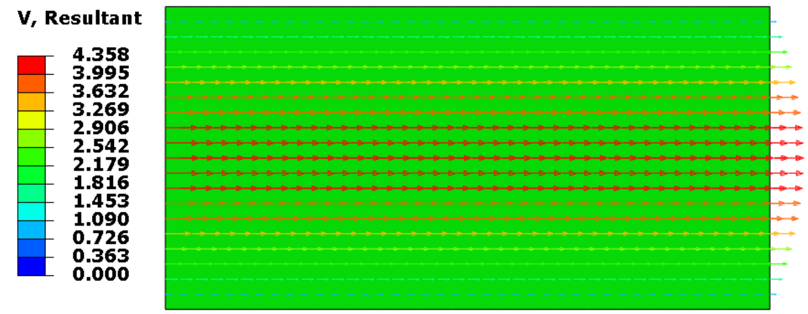
- **References**
  - Fluid Mechanics, Second Edition: Volume 6 (Course of Theoretical Physics), Authors: L. D. Landau, E.M. Lifshitz

Oscillatory Laminar Plane Poiseuille Flow

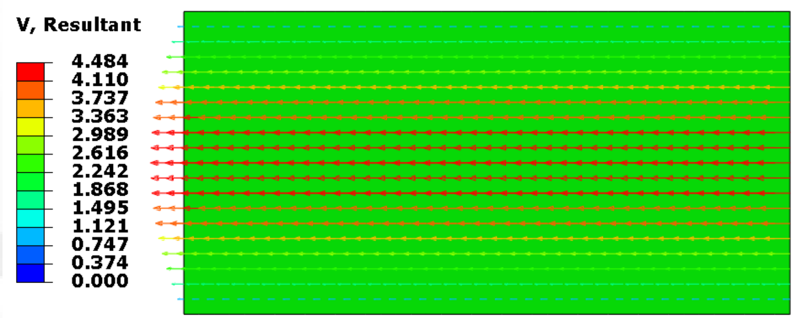
Results



Velocity profile: Comparison with analytical solution



T= 5 sec



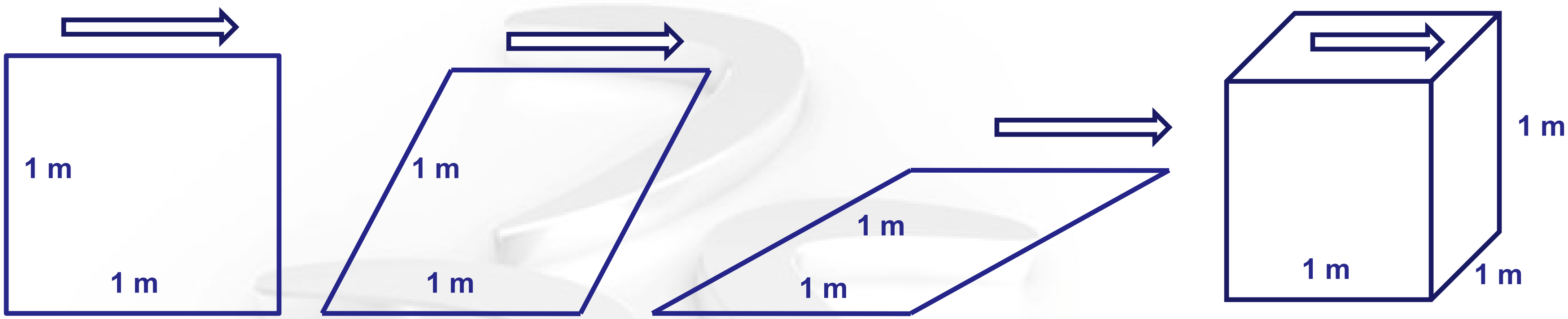
T= 10 sec

- **Files**
  - ex1_oscillatory_planeflow.py
    - ex1_oscillatory_planeflow_mesh.inp

2. Flow in Shear Driven Cavities

Flow in Shear Driven Cavities

- **Overview**
  - This sample problem compares the prediction of velocity profiles in shear-driven cavities of different shapes. 2-dimensional 90^0, 45^0 and 15^0 cavities are considered. Shear driven flow in a cubical box is also presented. Velocity profiles in each of these cases are compared against numerical results available in literature.
- **Problem description**
  - The following cavity configurations and Reynolds' numbers are presented



(a) 90^0 Cavity (b) 45^0 Cavity (c) 15^0 Cavity (d) Cubical box

- 90^0 Cavity: Reynolds number = 100, 3200; Mesh: 129x129x1
- 45^0 Cavity: Reynolds number = 100; Mesh: 512x512x1
- 15^0 Cavity: Reynolds number = 100; Mesh: 256x256x1
- Cubical box: Reynolds number = 400; Mesh: 32x32x32

Flow in Shear Driven Cavities

- **Features**
  - Laminar shear driven flow
- **Boundary conditions**
  - Specified velocity at top plane to match flow Reynolds' number
  - No-slip at all other planes (***V*** = 0)
  - Hydrostatic mode is eliminated by setting reference pressure to zero at a single node
- **References**

1. "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method"
   U. Ghia, K. N. Ghia, and C. T. Shin
   Journal of Computational Physics 48, 387-411 (1982)

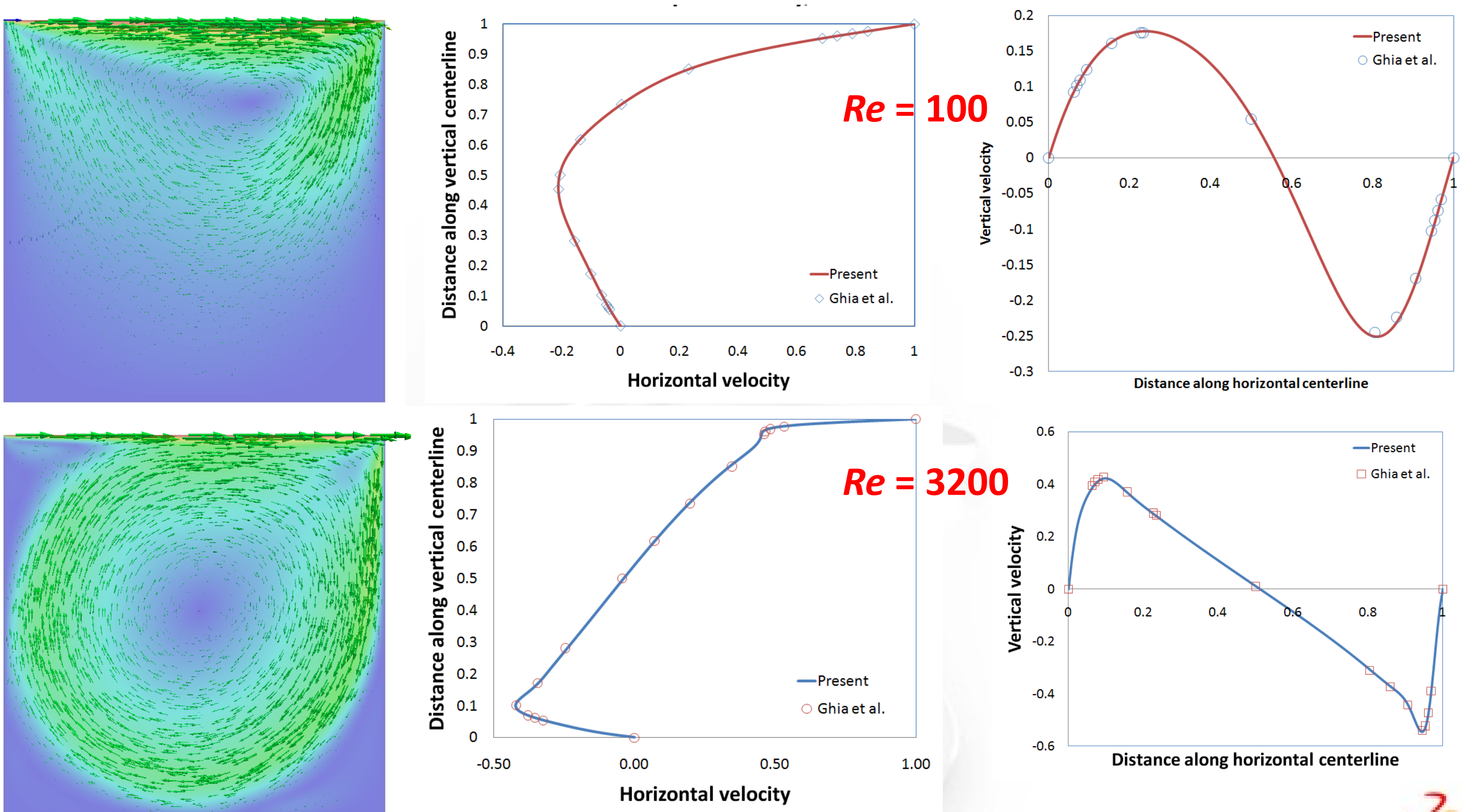
2. "Numerical solutions of 2-D steady incompressible flow in a driven skewed cavity"
   Ercan Erturk∗ and Bahtiyar Dursun
   ZAMM · Z. Angew. Math. Mech. 87, No. 5, 377 – 392 (2007)

3. "Flow topology in a steady three-dimensional lid-driven cavity"
   T.W.H. Sheu, S.F. Tsai, Computers & Fluids, 31, 911–934 (2002)

Flow in Shear Driven Cavities

90⁰ Cavity

- **Results**
  - Velocities along horizontal and vertical centerlines

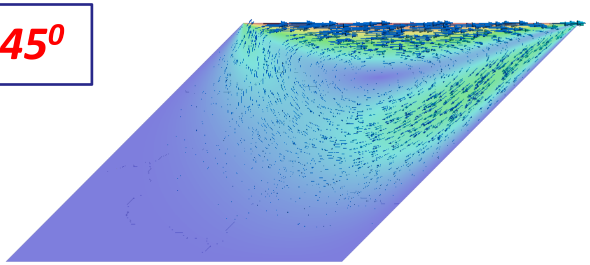


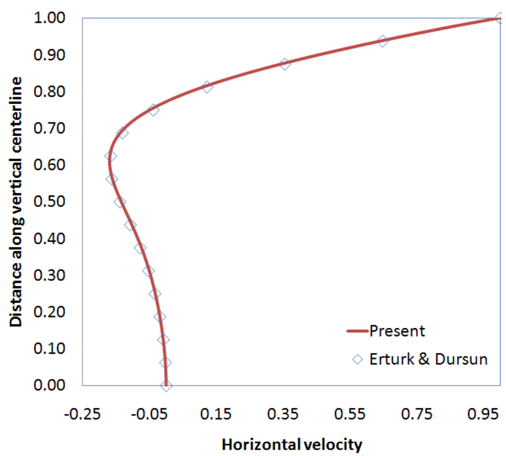
Flow in Shear Driven Cavities

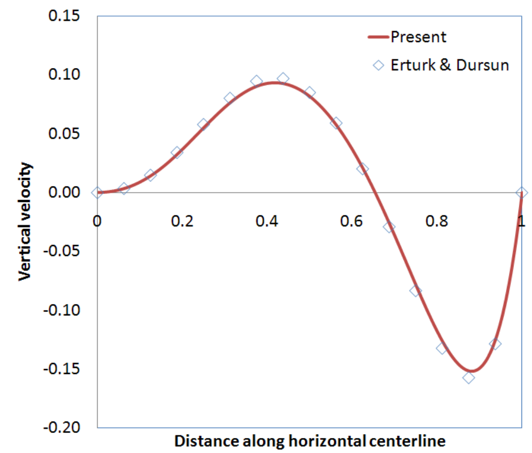
Skew Cavity (45^0 and 15^0)

- # Results
  - Velocities along horizontal and vertical centerlines

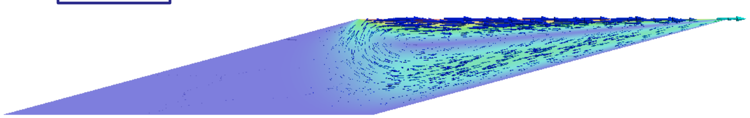
45^0

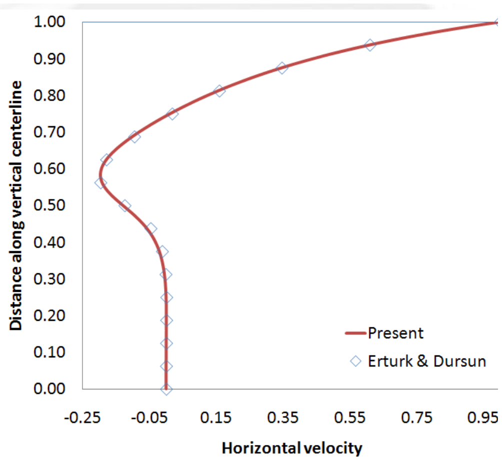


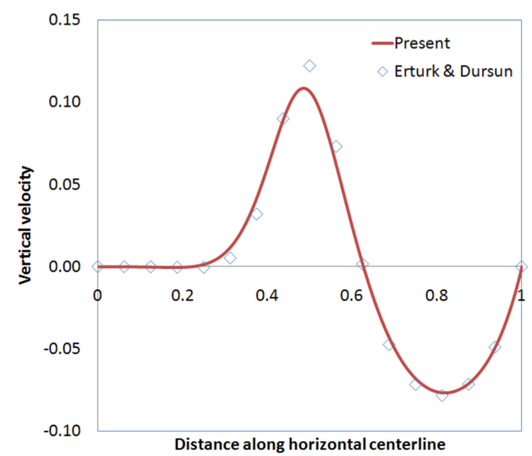




15^0



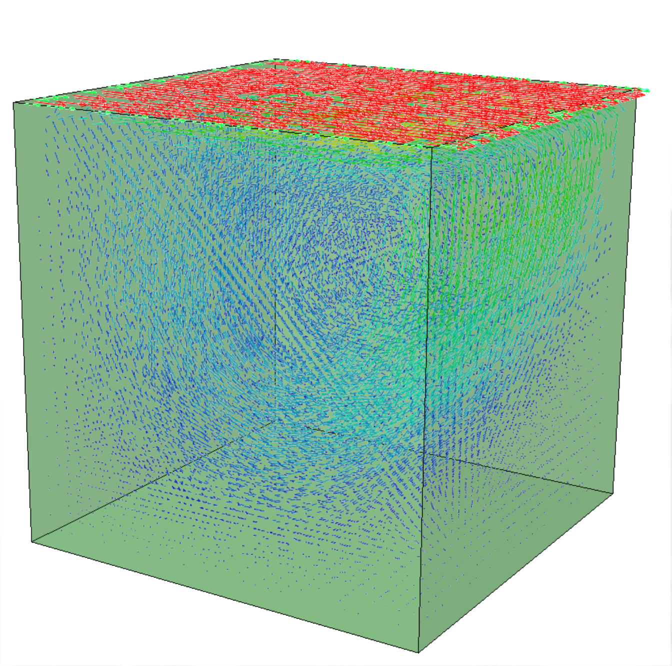




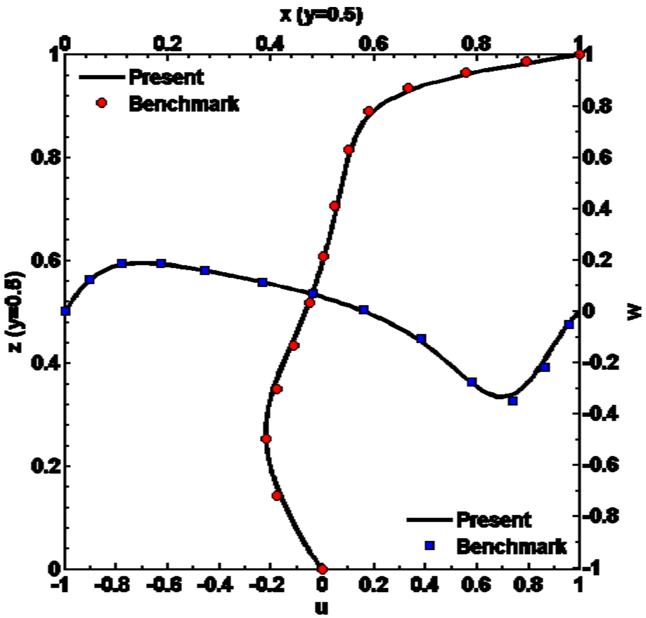
Flow in Shear Driven Cavities

Cubical Box

- **Results**



Velocity vectors



Velocities along horizontal and vertical centerlines

SIMULIA

Flow in Shear Driven Cavities

- **Files**
  - ex2_sheardriven_cavity.py
    - ex2_cavity15deg_mesh.inp
    - ex2_cavity45deg_mesh.inp
    - ex2_cubicalbox_mesh.inp
    - ex2_sqcavity_mesh.inp

3. Buoyancy Driven Flow in Cavities

Buoyancy Driven Flow in Cavities

- **Overview**
  - This sample problem compares the prediction of velocity profiles due to buoyancy driven flow in square and cubical cavities. The cavities are differentially heated to obtain a temperature gradient. Velocity profiles in each of these cases are compared against numerical results available in the literature.
- **Problem description**
  - The material properties are chosen to match the desired Rayleigh number, Ra

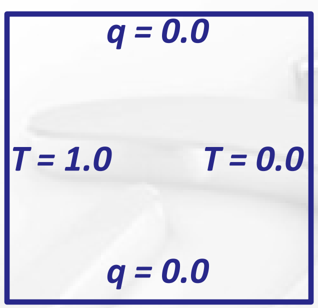
$$Ra = \frac{g\beta L^3 \Delta T}{\upsilon\alpha}$$

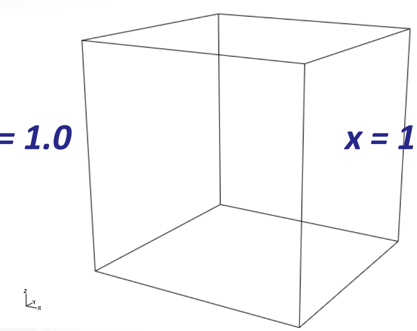
υ, Kinematic viscosity

α, Thermal diffusivity

β, Thermal expansion coefficient

g, Acceleration due to gravity





  - Square Cavity: Rayleigh number = 1e3, 1e6
  - Cubical Cavity: Rayleigh number = 1e4

Buoyancy Driven Flow in Cavities

- **Features**
  - Buoyancy driven flow
  - Boussinesq body forces
- **Boundary conditions**
  - No-slip velocity boundary condition on all the planes ($\boldsymbol{V} = 0$)
  - Specified temperatures
  - Hydrostatic mode is eliminated by setting reference pressure to zero at a single node
- **References**

1. “Natural Convection in a Square Cavity: A Comparison Exercise”
   G. de Vahl Davis and I. P. Jones
   International Journal for Numerical Methods in Fluids, **3**, 227-248, (1983)

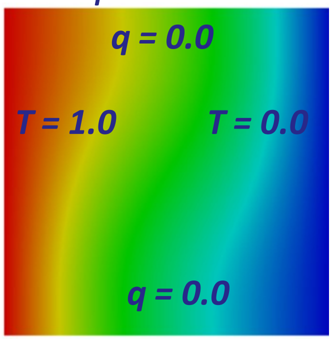
2. "Benchmark solutions for natural convection in a cubic cavity using the high-order time–space method"
   Shinichiro Wakashima, Takeo S. Saitoh
   International Journal of Heat and Mass Transfer, 47, 853–864, (2004)

Buoyancy Driven Flow in Cavities

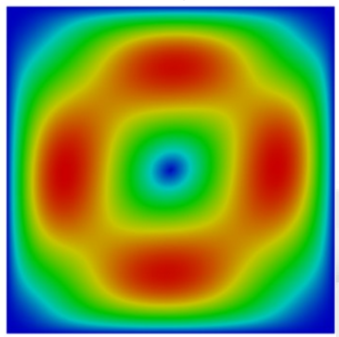
2D Square Cavity

- **Results**

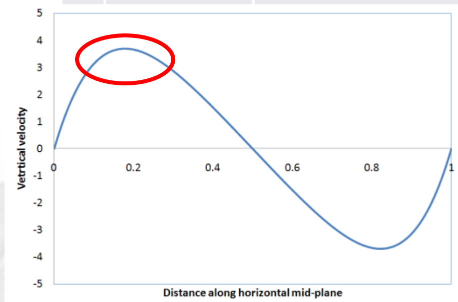
Temperature



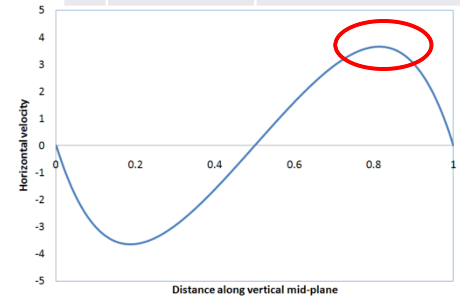
Velocity



	Present	Benchmark (de Vahl Davis)
u	3.695	3.697
x	0.1781	0.178

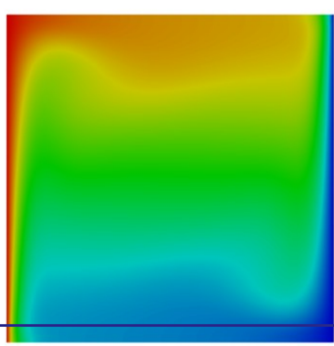


	Present	Benchmark (de Vahl Davis)
u	3.654	3.649
x	0.8129	0.813

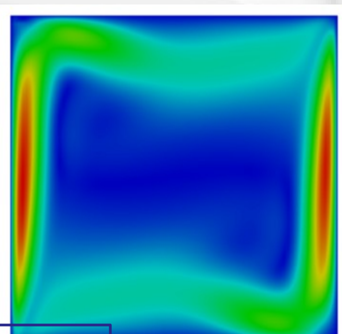


Ra = 1000, Pr = 0.71

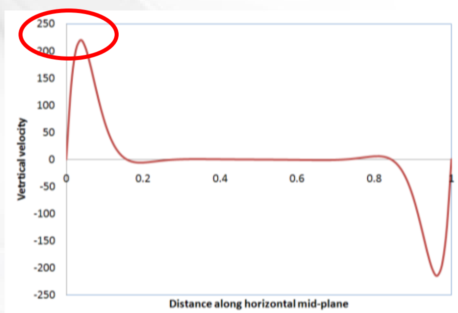
Temperature



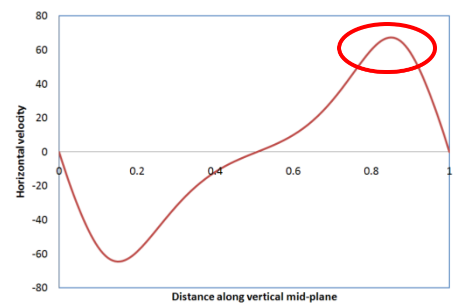
Velocity



	Present	Benchmark (de Vahl Davis)
u	219.747	219.36
x	0.0375	0.0379



	Present	Benchmark (de Vahl Davis)
u	65.9	64.63
x	0.85	0.85



Ra = 1.0e6 , Pr = 0.71

SIMULIA

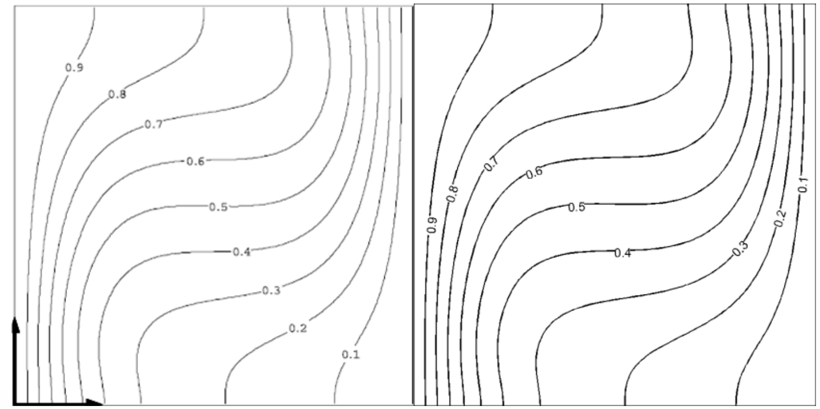
Buoyancy Driven Flow in Cavities

Cubical Box

- **Results**

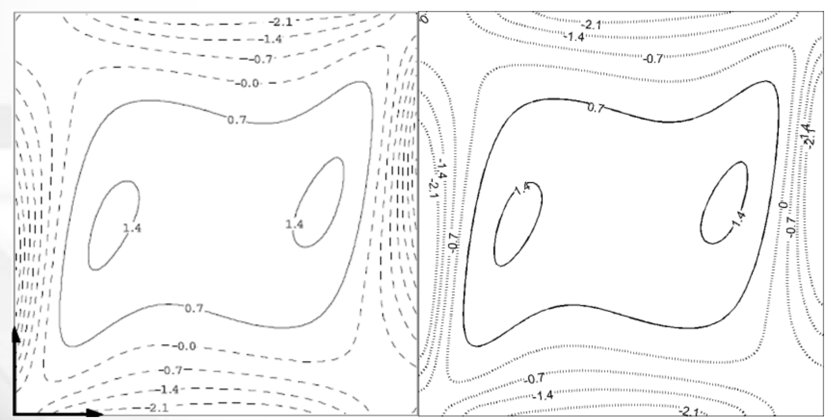
Ra = 1.0e4, Pr = 0.71

	$\omega_2(0,0,0)$	$U_{1_{\max}}(0.5,0.5,z)$	$U_{3_{\max}}(x,0.5,0.5)$
Benchmark (Wakashima & Saitoh (2004))	1.1018	0.1984 (z = 0.8250)	0.2216 (x = 0.1177)
Abaqus/CFD	1.1017	0.1986	0.2211
Error	-0.009%	0.1%	0.2%



Benchmark Present

Temperature contours at the mid plane (y=0.5)



Benchmark Present

Vorticity contours at the mid plane (y=0.5)

SIMULIA

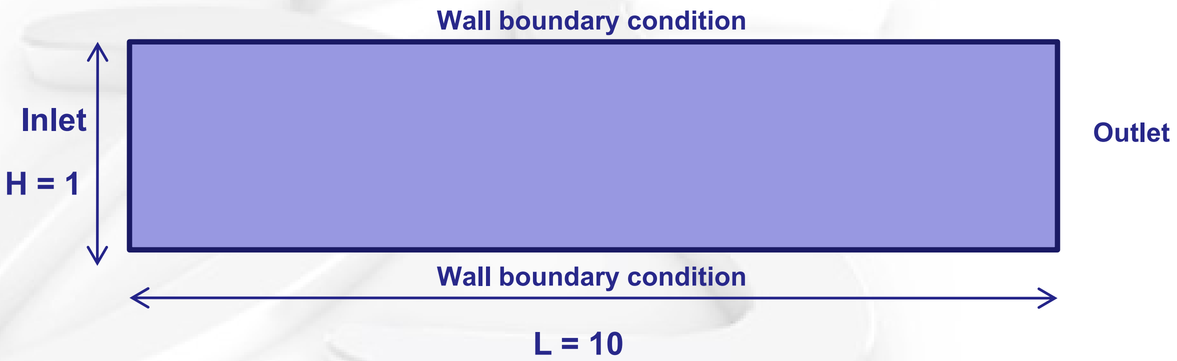
Buoyancy Driven Flow in Cavities

- **Files**
  - ex3_buoyancydriven_flow.py
    - ex3_sqcavity_mesh.inp
    - ex3_cubicbox_mesh.inp

4. Turbulent Flow in a Rectangular Channel

Turbulent Flow in a Rectangular Channel

- **Overview**
  - This example models the turbulent flow in a rectangular channel at a friction Reynolds number = 180 and Reynolds number (based on mean velocity) = 5600. The one equation Spalart-Allmaras turbulence model is used. The results are compared with direct numerical simulation (DNS) results available in the literature as well as experimental results. Abaqus/CFD results at a friction Reynolds number = 395 and Reynolds number (based on mean velocity) = 13750 are also presented.
- **Problem description**
  - A rectangular 2-dimensional channel of width = 1 unit and length = 10 units is considered. A pressure gradient is imposed along the length of the channel by means of specified pressure at the inlet and zero pressure at the outlet. The pressure gradient is chosen to impose the desired friction Reynolds number for the flow.



Turbulent Flow in a Rectangular Channel

- **Problem description (cont'd)**
  - The Reynolds number is defined as $Re = \rho U_{av} H/\mu$
  - The friction Reynolds number is defined as $Re_\tau = \rho u_\tau (H/2)/\mu$, where u_τ is the friction velocity defined as

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}} \qquad \tau_w = \text{wall shear stress}$$

- **Features**
  - Turbulent flow
  - Spalart-Allmaras turbulence model
- **Boundary conditions**
  - No-slip velocity boundary condition on channel walls(***V*** = 0)
  - Set through thickness velocity components to zero
  - Distance function = 0, Kinematic turbulent viscosity = 0 at No-slip velocity boundary condition
- **Mesh**
  - Mesh – 50 (Streamwise) X 91 (Normal) X 1 (Through thickness)
  - y^+ at first grid point ~ 0.046

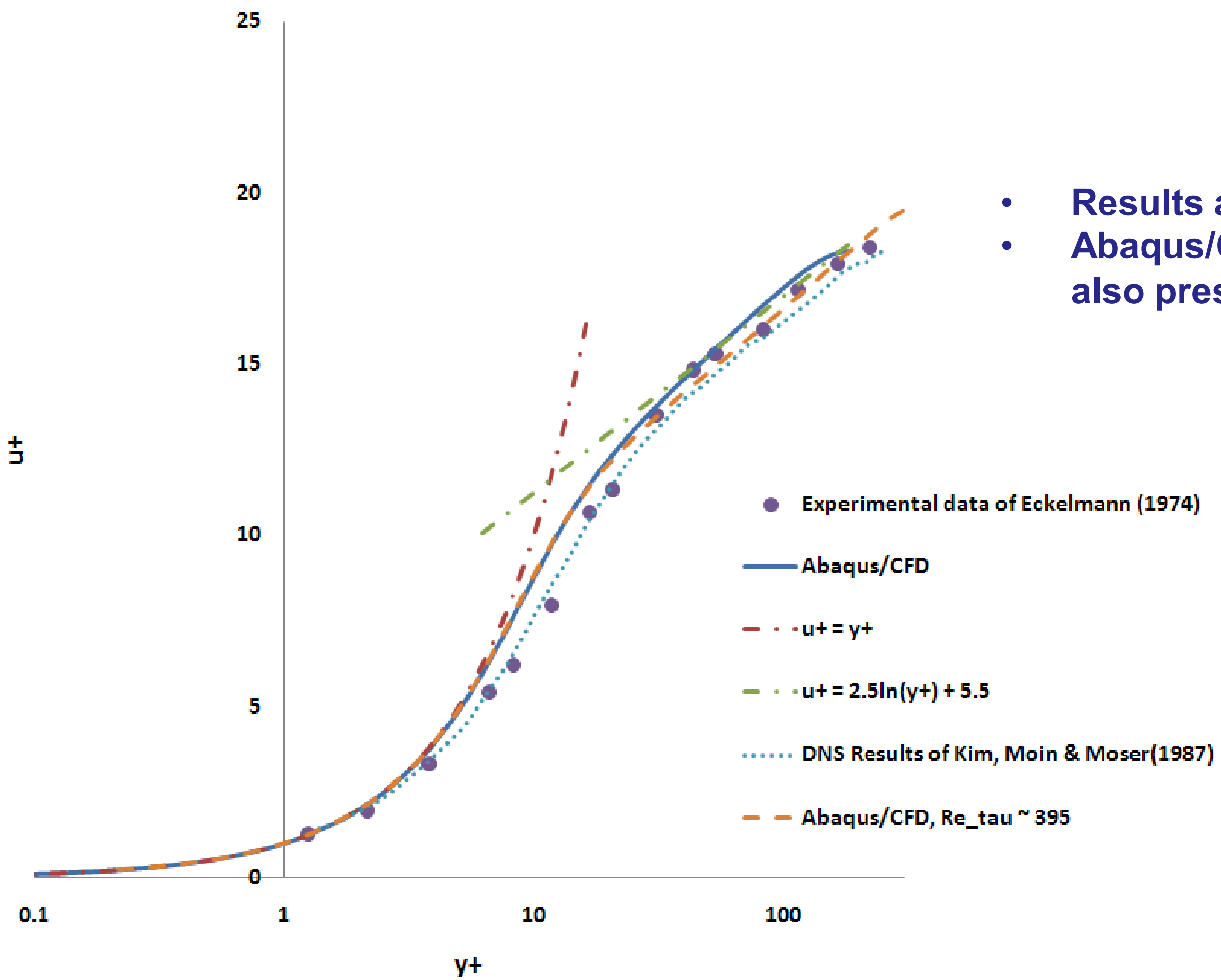
Turbulent Flow in a Rectangular Channel

- **References**

1. "Turbulence statistics in fully developed channel flow at low Reynolds number"
   J. Kim, P. Moin and R. Moser
   Journal of Fluid Mechanics, **177**, 133-166, (1987)
2. "The structure of the viscous sublayer and the adjacent wall region in a turbulent channel flow"
   H. Eckelmann
   Journal of Fluid Mechanics, **65**, 439, (1974)

Turbulent Flow in a Rectangular Channel

- **Results**
  - “Law of the wall”

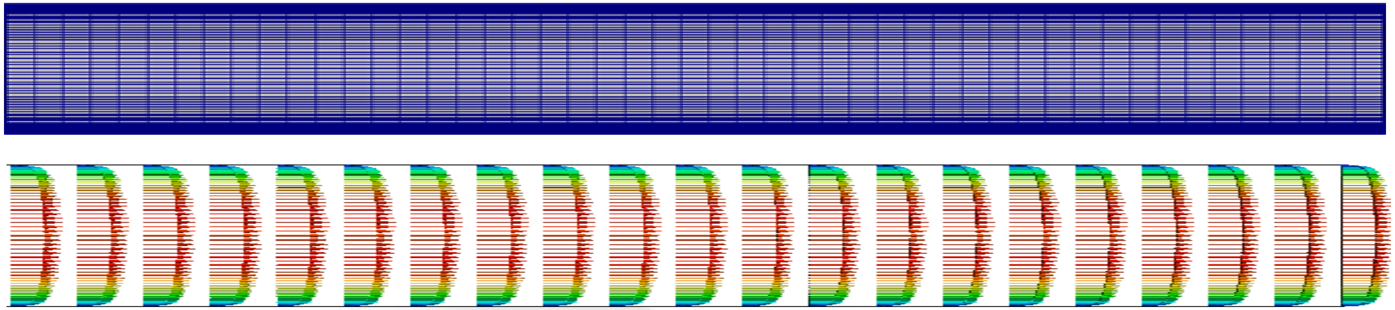


$$, y^+ = \frac{u_\tau y}{\nu}$$

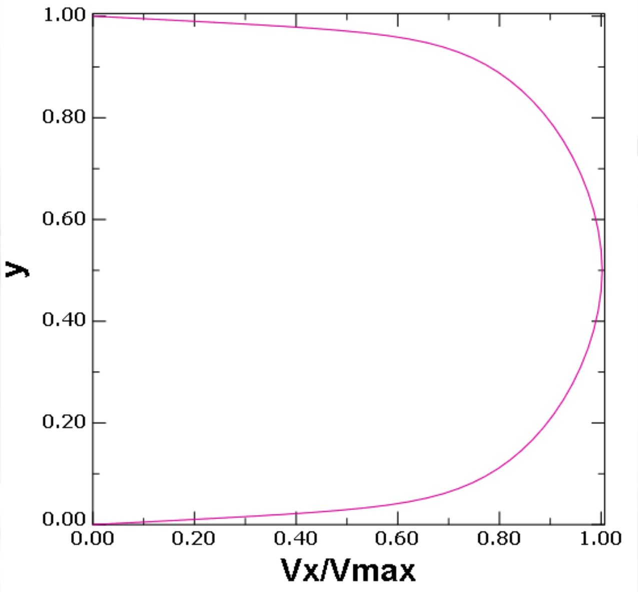
- **Results are presented at $Re_\tau = 180$**
- **Abaqus/CFD results at $Re_\tau = 395$ is also presented**

Turbulent Flow in a Rectangular Channel

- **Results**



Velocity profile



Turbulent Flow in a Rectangular Channel

- **Files**
  - ex4_turb_channelflow.py
    - ex4_turb_channelflow_mesh.inp

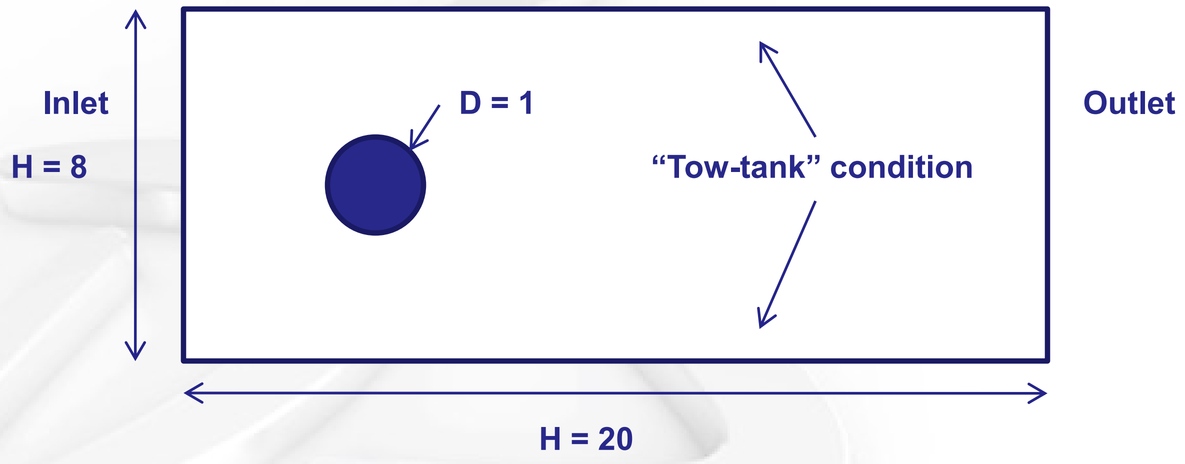
- **Note**
  - *Abaqus/CFD results at a friction Reynolds number ~ 395 and Reynolds number (based on mean velocity) ~ 13750 can be obtained by setting a pressure gradient of 0.0573*

SIMULIA

5. Von Karman Vortex Street Behind a Circular Cylinder

Von Karman Vortex Street Behind a Circular Cylinder

- **Overview**
  - This example simulates the Von Karman vortex street behind a circular cylinder at a flow Reynolds number of 100 based on the cylinder diameter. The frequency of the vortex shedding is compared with results available in the literature.
- **Problem description**
  - The computational domain for the vortex shedding calculations consists of an interior square region (-4 < x < 4; -4 < y < 4) surrounding a cylinder of unit diameter. The domain is extended in the wake of cylinder up to x = 20 units



Von Karman Vortex Street Behind a Circular Cylinder

- **Features**
  - Unsteady laminar flow
  - Reynolds number = $\rho U_{inlet} D/\mu$

- **Fluid Properties**
  - Density = 1 unit
  - Viscosity = 0.01 units

- **Boundary conditions**
  - Tow tank condition at top and bottom walls:
    - U = 1, V = 0
  - Inlet velocity: U_{inlet} = 1, V = 0
  - Set through thickness velocity components to zero
  - Outlet: P = 0
  - Cylinder surface: No-slip velocity boundary condition (***V*** = 0)

Von Karman Vortex Street Behind a Circular Cylinder

- **References**

1. Transient flow past a circular cylinder: a benchmark solution",
M. S. Engelman and M. A. Jamnia
International Journal for Numerical Methods in Fluids, **11**, 985-1000, (1990)

Von Karman Vortex Street Behind a Circular Cylinder

Results

	Total number of element	Strouhal Number*	Δt
Coarse	1760	0.1749	0.03
Fine	28160	0.1735	0.0075

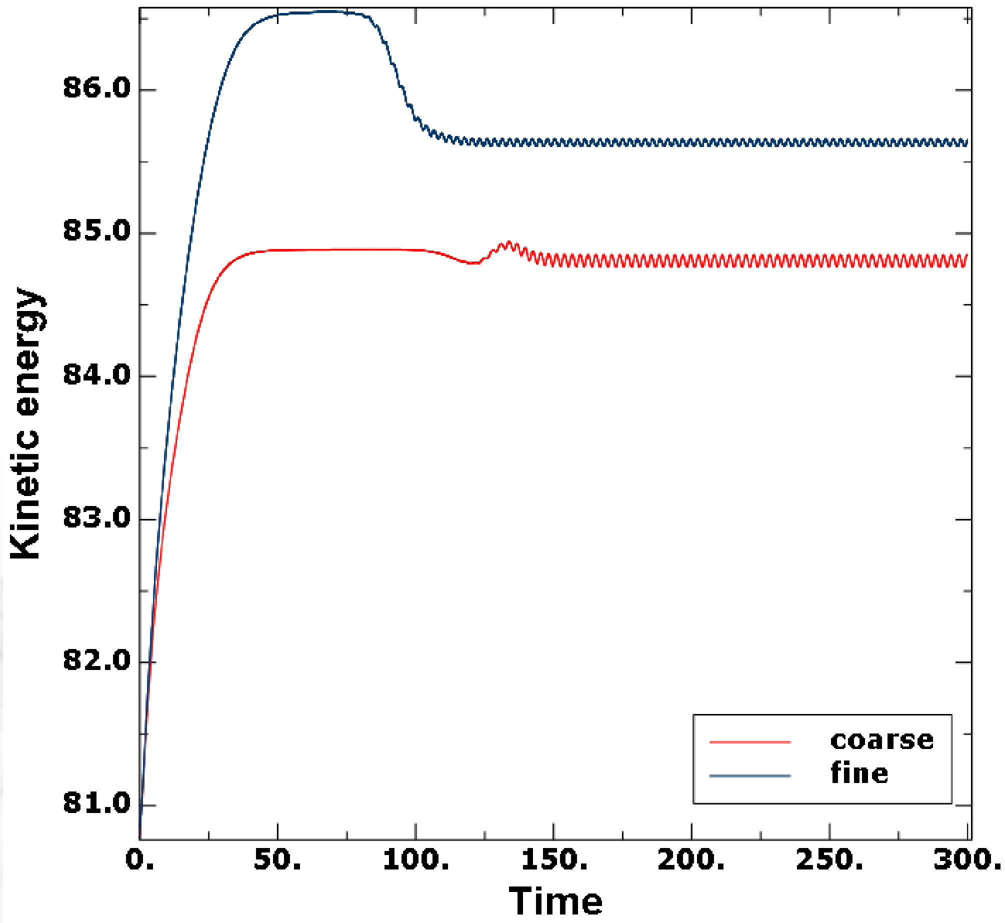
** St = fD/U_{inlet} , f is the frequency of vortex shedding*

- The results are consistent with the Strouhal numbers reported in the reference (0.172-0.173)
- The frequency of the vortex shedding is obtained as the half of the peak frequency obtained by a FFT of kinetic energy plot (from t = 200 sec to 300 sec)

Von Karman Vortex Street Behind a Circular Cylinder

Results

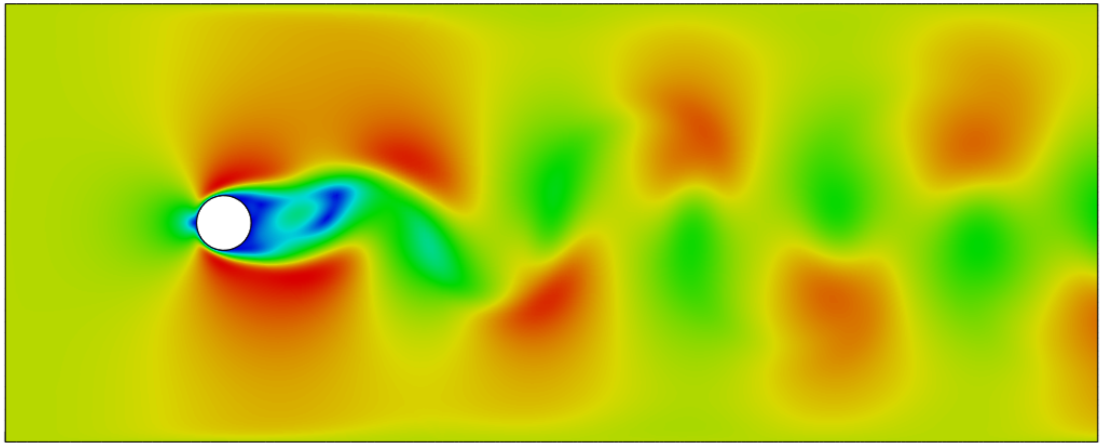
- Kinetic energy



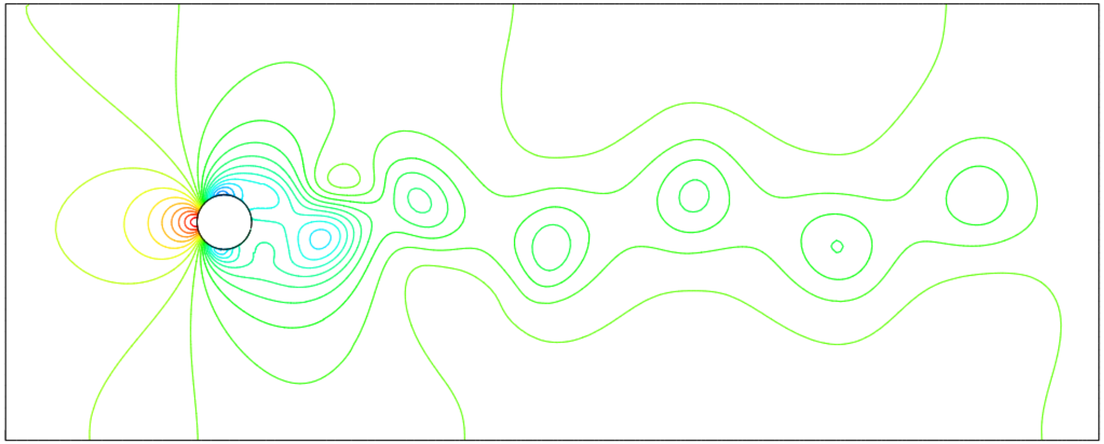
SIMULIA

Von Karman Vortex Street Behind a Circular Cylinder

Results



Velocity contour plot at t = 300 sec

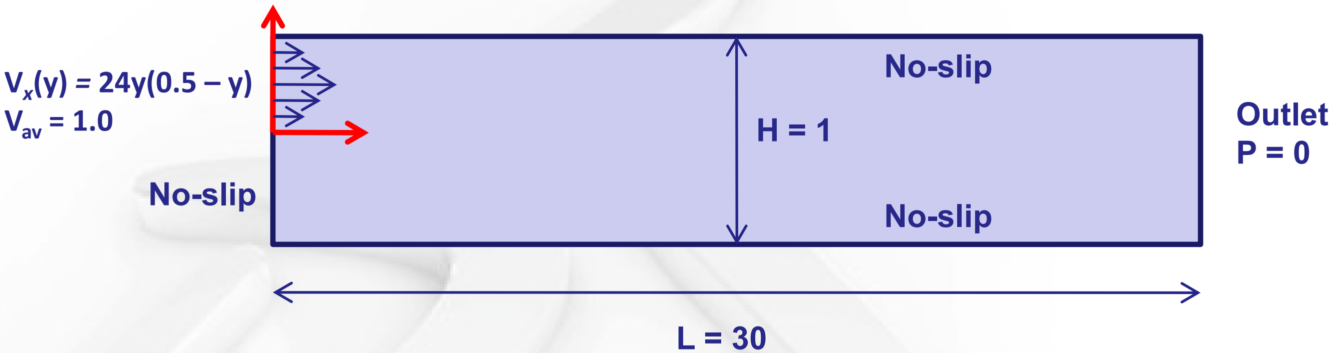


Pressure line plot at t = 300 sec

6. Flow Over a Backward Facing Step

Flow Over a Backward Facing Step

- **Overview**
  - This example simulates the laminar flow over a backward facing step at a Reynolds number of 800 based on channel height. The results are compared with numerical as well as experimental results available in literature.
- **Problem description**
  - The computational domain for the flow calculations consists of an rectangular region (0 < x < 30; -0.5 < y < 0.5). The flow enters the solution domain from 0< y < 0.5 while -0.5 < y < 0.0 represents the step. A parabolic velocity profile is specified at the inlet.



$$\mathrm{Re} = \frac{\rho V_{av} H}{\mu}$$

Flow Over a Backward Facing Step

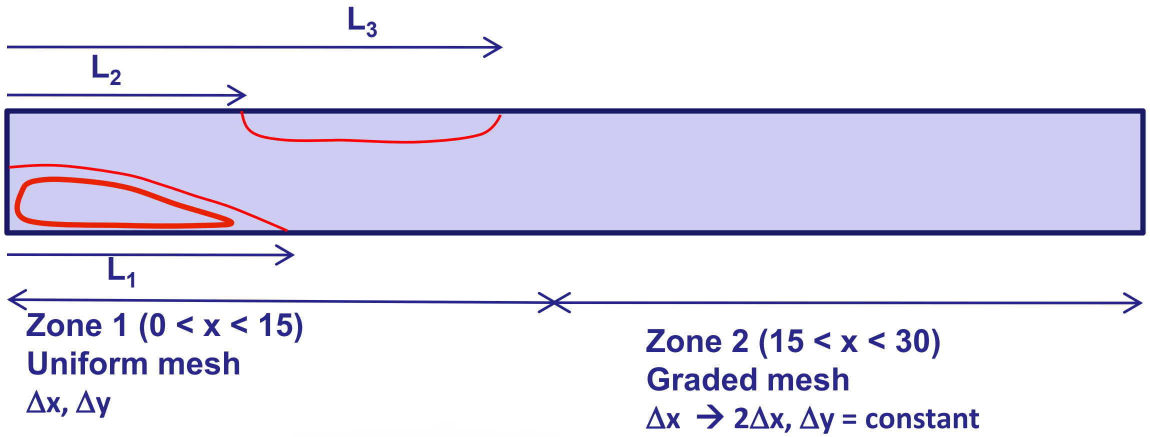
- **Features**
  - Steady laminar flow
- **Fluid Properties**
  - Density = 1 unit
  - Viscosity = 0.00125 units (the viscosity is chosen so as to set the flow Reynolds number to 800)
- **Boundary conditions**
  - Set through thickness velocity components to zero
  - No-slip velocity boundary condition at top and bottom walls
    - $V_x = 0, V_y = 0$
  - Inlet velocity: Parabolic velocity profile - $V_x = f(y)$
    - $V_y = 0$
  - Outlet: P = 0
  - No-slip velocity boundary condition at the step boundary
    - $V_x = 0, V_y = 0$

Flow Over a Backward Facing Step

- **References**
  - "A test problem for outflow boundary conditions – Flow over a backward-facing step"
    D. K. Gartling, International Journal for Numerical Methods in Fluids
    Vol 11, 953-967, (1990)

Flow Over a Backward Facing Step

Results



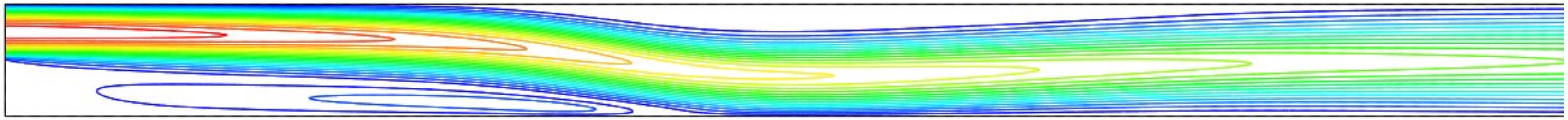
	Mesh (Across the channel x along the channel length)	L_1 Length from the step face to the lower re-attachment point	L_2 Length from the step face to upper separation point	L_3 Length of the upper separation bubble
Gartling (1990)	40x800	6.1	4.85	10.48
Abaqus/CFD	Fine 80x1200x1 (Zone 1) 80x832x1 (Zone 2)	5.9919	4.9113	10.334
Abaqus/CFD	Medium 40x600x1 (Zone 1) 40x416x1 (Zone 2)	5.7471	4.8379	10.101
Abaqus/CFD	Coarse 20x300x1 (Zone 1) 20x208x1 (Zone 2)	4.5018	3.9659	8.7748

- Elements used by Gartling (1990) were biquadratic in velocity and linear discontinuous pressure elements. In contrast, the fluid elements in Abaqus/CFD use linear discontinuous in velocity and linear continuous in pressure.

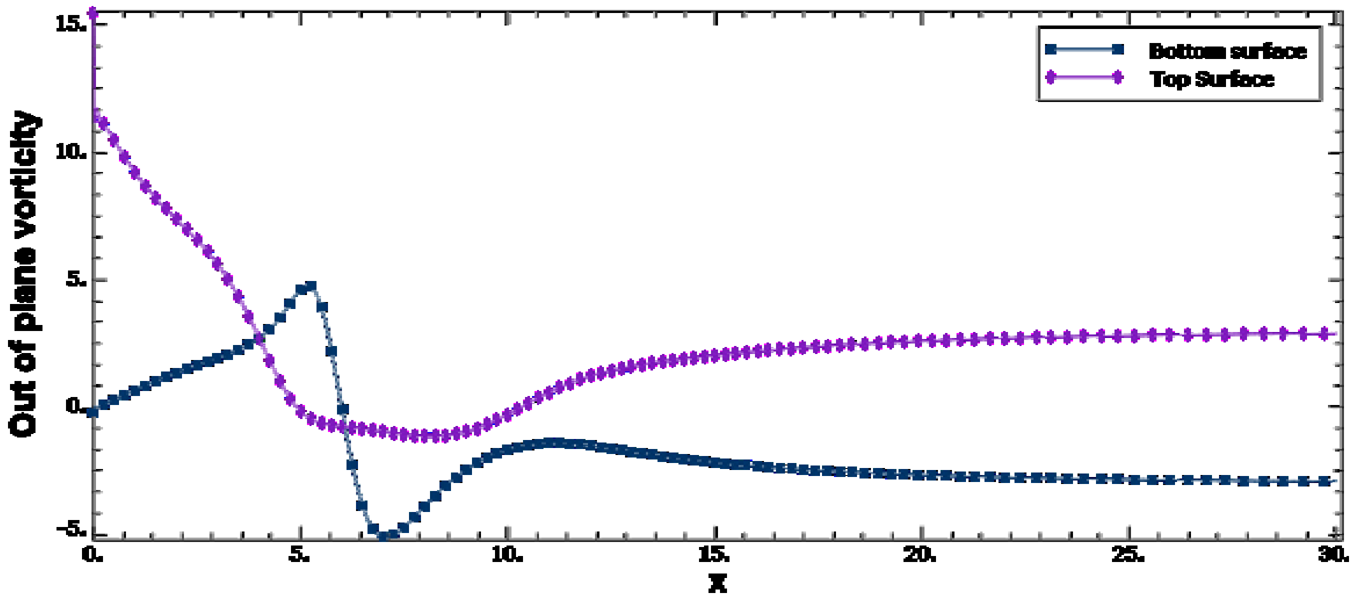
Flow Over a Backward Facing Step

Results

Pressure line plot (0 < x < 30)



Velocity line plot (0 < x < 15)

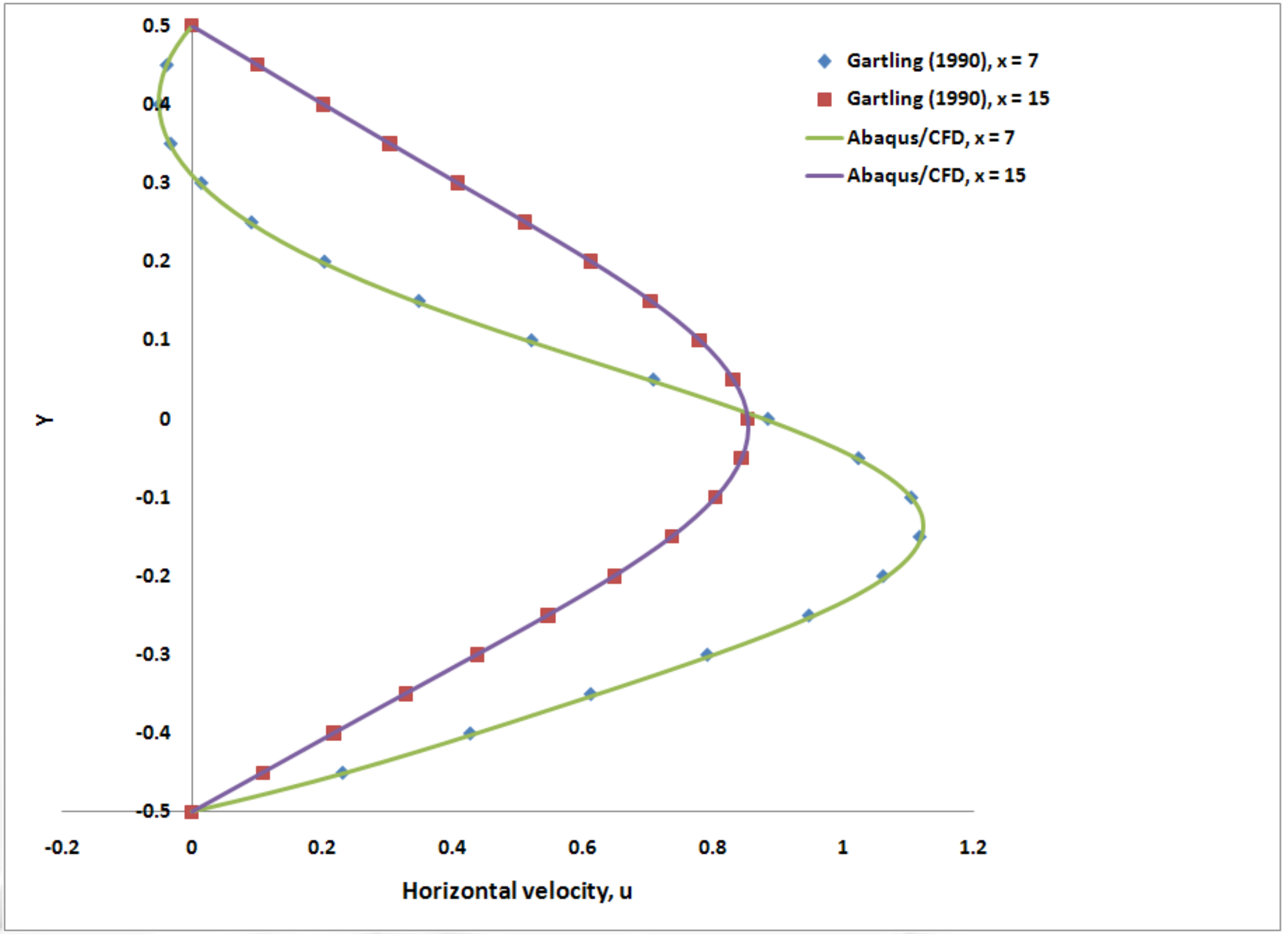


Out of plane vorticity (0 < x < 30)

Flow Over Backward Facing Step

Results

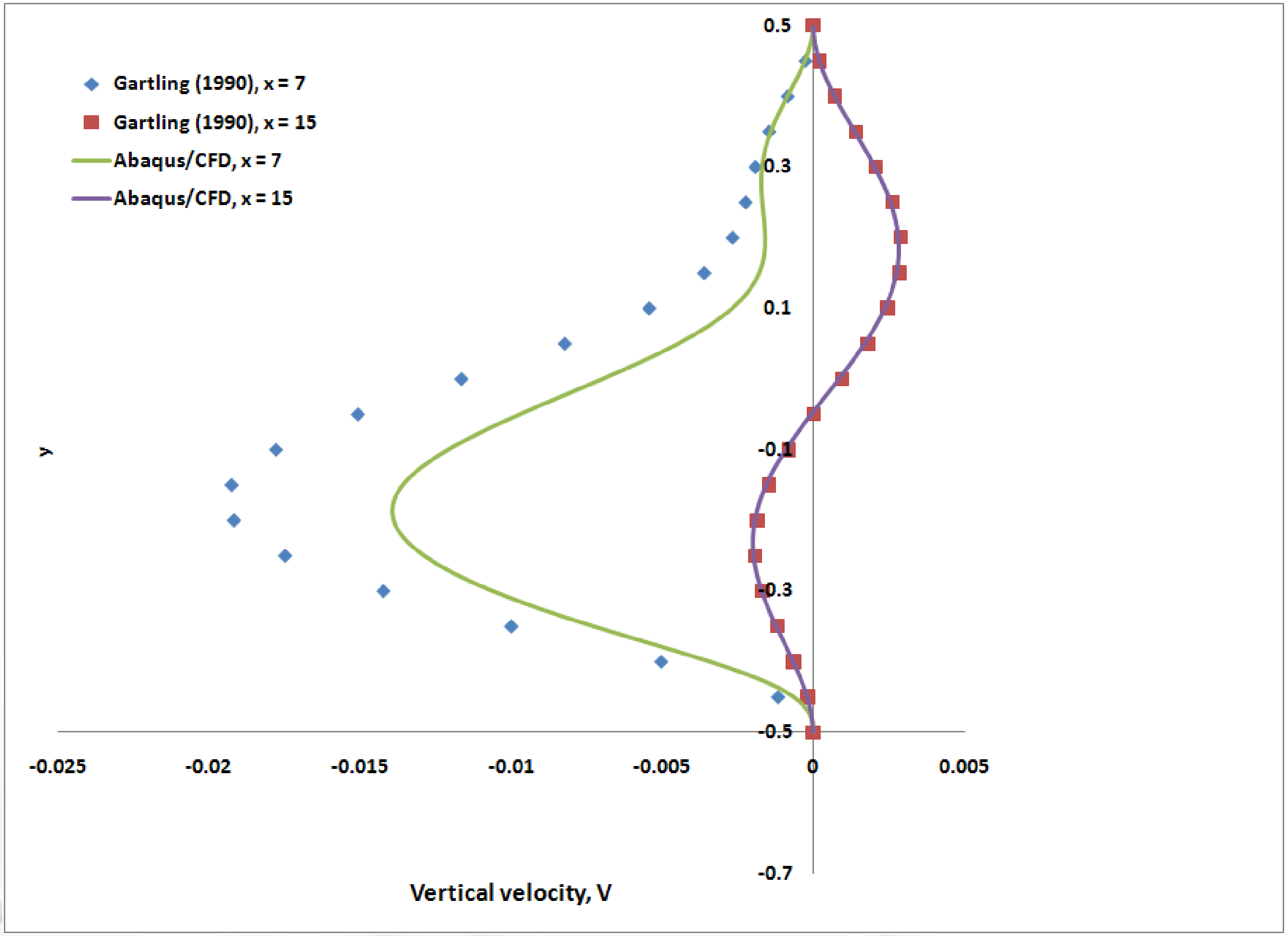
- **Horizontal velocity at x = 7 & x = 15**



Flow Over Backward Facing Step

Results

- Vertical velocity at x = 7 & x = 15



SIMULIA

Flow Over Backward Facing Step

- **Files**
  - ex6_backwardfacingstep.py
    - ex6_backwardfacingstep_coarse.inp
    - ex6_backwardfacingstep_medium.inp
    - ex6_backwardfacingstep_fine.inp
  - coarse_parabolic_inlet_velocity.inp
  - medium_parabolic_inlet_velocity.inp
  - fine_parabolic_inlet_velocity.inp

- **Note**
  - *The models require a parabolic velocity profile at the inlet. This needs to be manually included as boundary condition in the generated input file.*
  - *The parabolic velocity profile required is provided in files coarse_parabolic_inlet_velocity.inp, medium_parabolic_inlet_velocity.inp and fine_parabolic_inlet_velocity.inp for coarse, medium and fine meshes, respectively.*