# Viscous flow in pipes and channels. Computational Fluid Dynamics

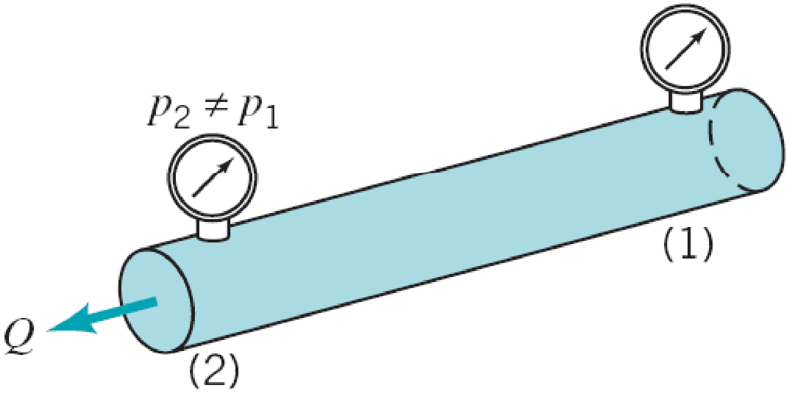
Lecture 6

# Content

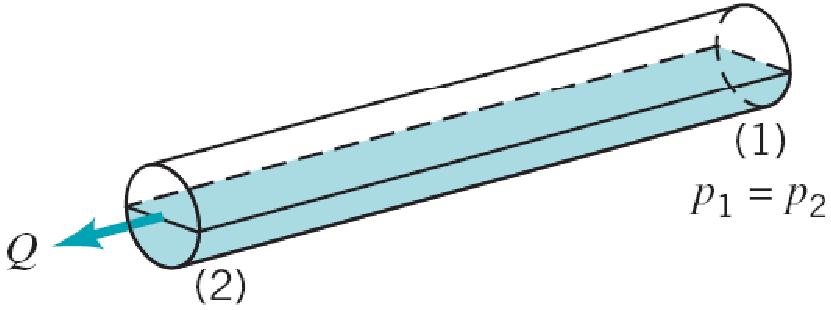
- Laminar and turbulent flow
- Entrance region
- Flow in a pipe
- Channels of non-circular cross-section
- Circuit theory for fluidic channels
- Computational fluid dynamics

# General characteristic of Pipe flow

- pipe is **completely filled** with water
- main driving force is usually a **pressure gradient** along the pipe, though **gravity** might be important as well



**Pipe flow**



**open-channel flow**

# Important facts from Fluid Dynamics

$$\boxed{\mathrm{Re} = \frac{\rho u d}{\mu}}$$

- Reynolds number
  - Can be interpreted as a ratio of inertia and viscous forces

# Laminar or Turbulent flow

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$

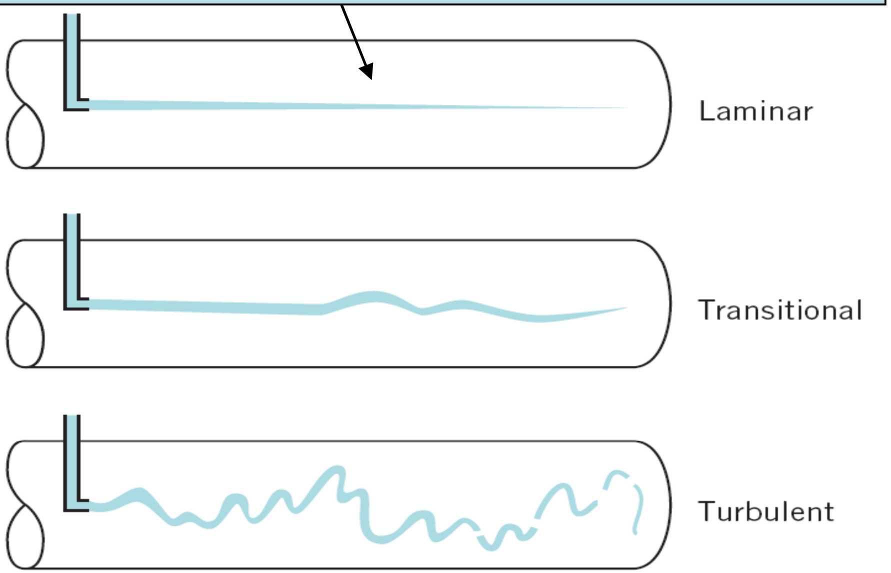
Reynolds number: can be interpreted as a ratio of inertia and viscous forces

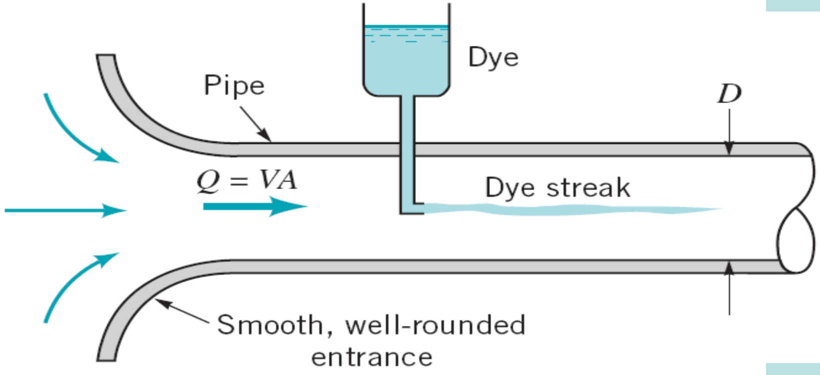
well defined streakline, one velocity component

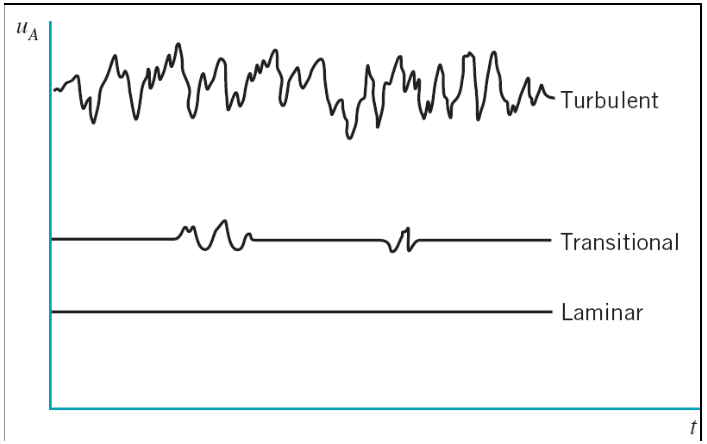
$$\boldsymbol{V} = u\hat{\boldsymbol{i}}$$

$\mathrm{Re} < 2100$

$\mathrm{Re} > 4000$







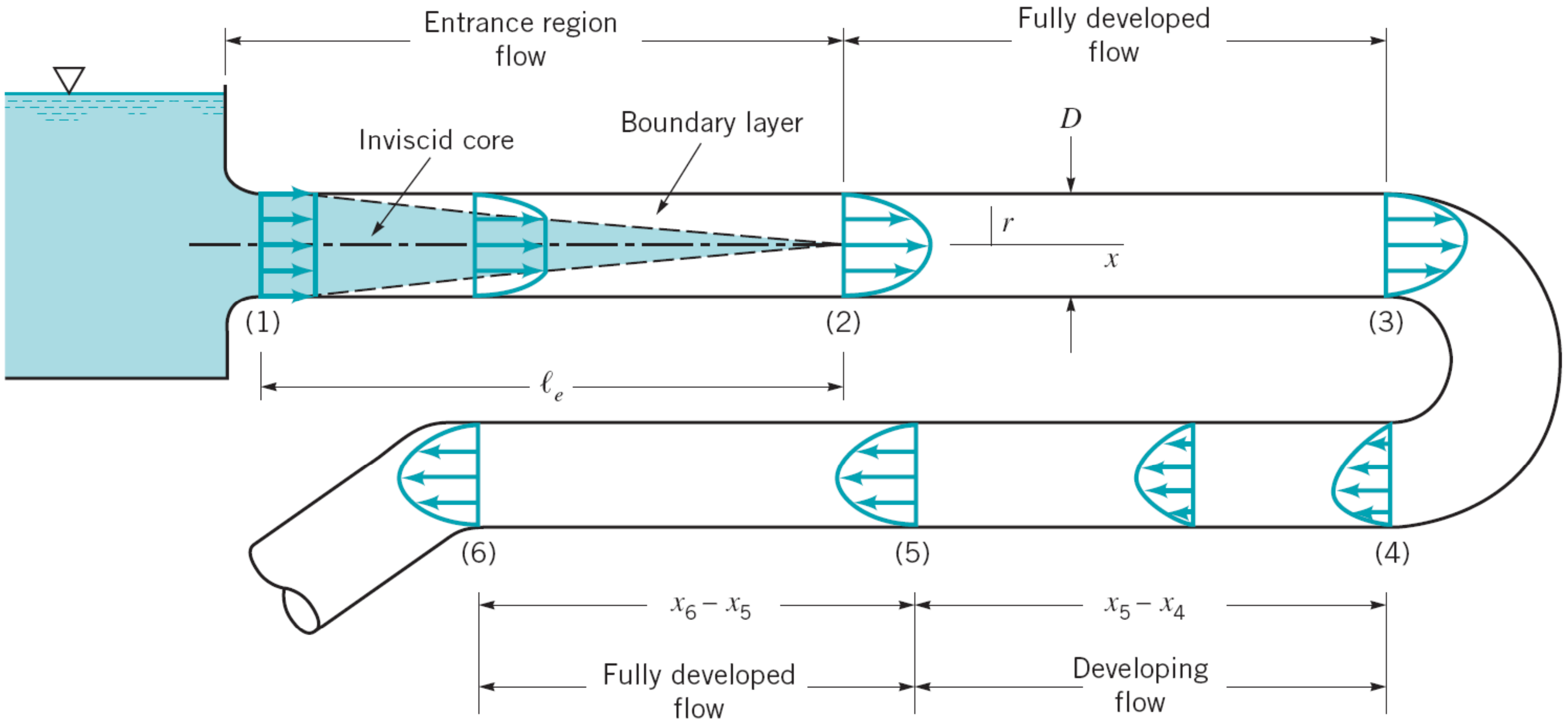
velocity along the pipe is unsteady and accompanied by random component normal to pipe axis

$$\boldsymbol{V} = u\hat{\boldsymbol{i}} + v\hat{\boldsymbol{j}} + w\hat{\boldsymbol{k}}$$

# Laminar or Turbulent flow

- In this experiment water flows through a clear pipe with increasing speed. Dye is injected through a small diameter tube at the left portion of the screen. Initially, at low speed (Re <2100) the flow is laminar and the dye stream is stationary. As the speed (Re) increases, the transitional regime occurs and the dye stream becomes wavy (unsteady, oscillatory laminar flow). At still higher speeds (Re>4000) the flow becomes turbulent and the dye stream is dispersed randomly throughout the flow.

# Entrance region and fully developed flow



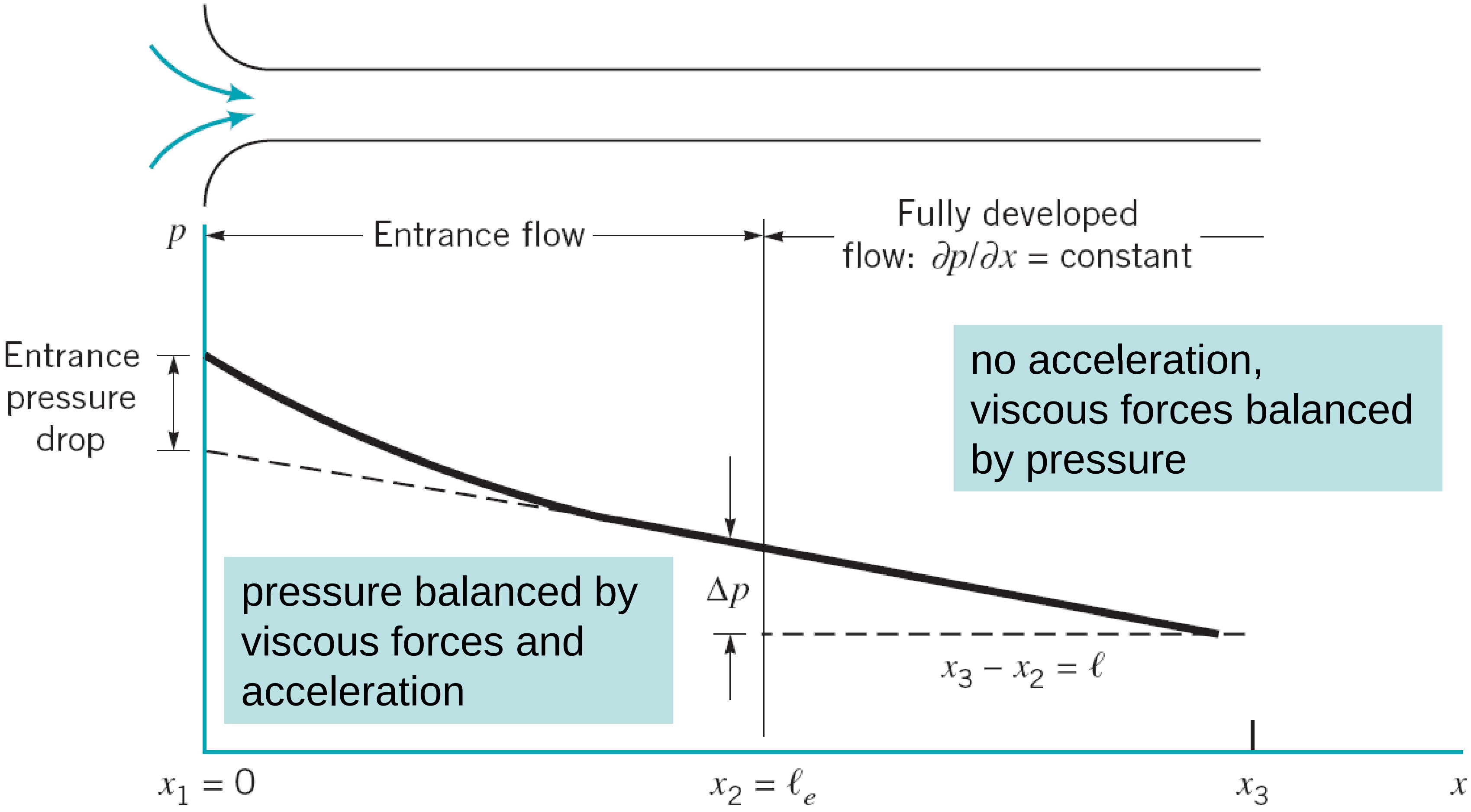
- fluid typically enters pipe with nearly uniform velocity
- the length of entrance region depends on the Reynolds number

dimensionless entrance length →

$$\frac{l_e}{D} = 0.06\,\mathrm{Re} \qquad \text{for laminar flow}$$

$$\frac{l_e}{D} = 4.4\left(\mathrm{Re}\right)^{1/6} \qquad \text{for turbulent flow}$$

# Pressure and shear stress



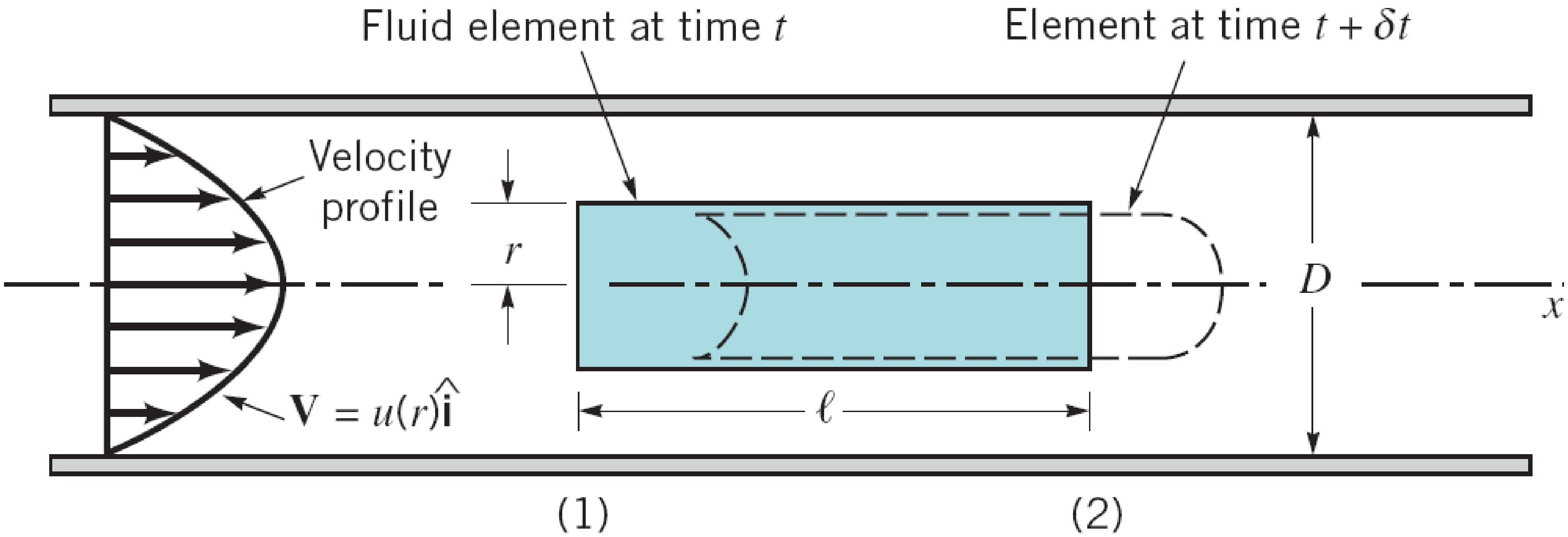
no acceleration, viscous forces balanced by pressure

pressure balanced by viscous forces and acceleration

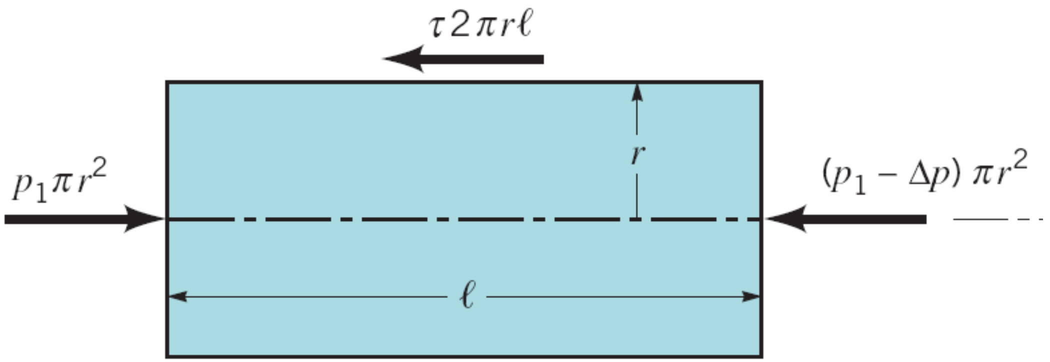
# Fully developed laminar flow

- we will derive equation for fully developed laminar flow in pipe using 3 approaches:
  - from 2nd Newton law directly applied
  - from Navier-Stokes equation
  - from dimensional analysis

# 2nd Newton's law directly applied



# 2nd Newton's law directly applied

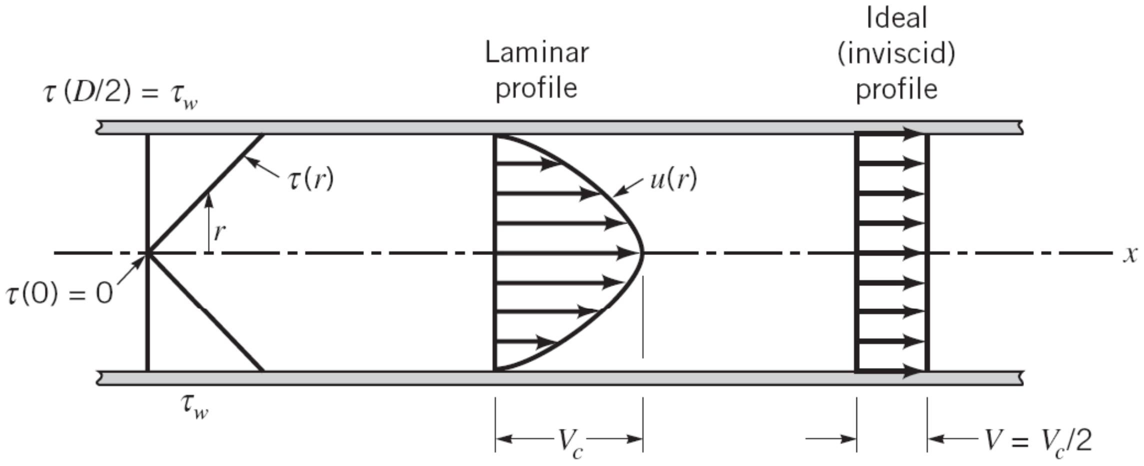


$$p_1\pi r^2 - (p_1 - \Delta p)\pi r^2 - \tau 2\pi r l = 0$$

$$\frac{\Delta p}{l} = \frac{2\tau}{r} \quad \Rightarrow \quad \tau = Cr, \text{ at } r = D/2 \text{ stress is maximum } \tau_w \text{ \textbf{wall shear stress}}$$

$$\tau = \frac{2\tau_w r}{D} \text{ and } \Delta p = \frac{4l\tau_w}{D}$$

doesn't depend on radius



# 2nd Newton’s law directly applied

for Newtonian liquid: $\tau = -\mu \frac{du}{dr}$ $\quad$ $\tau = \left( \frac{\Delta p}{2l} \right) r$

$$\frac{du}{dr} = -\left( \frac{\Delta p}{2\mu l} \right) r$$

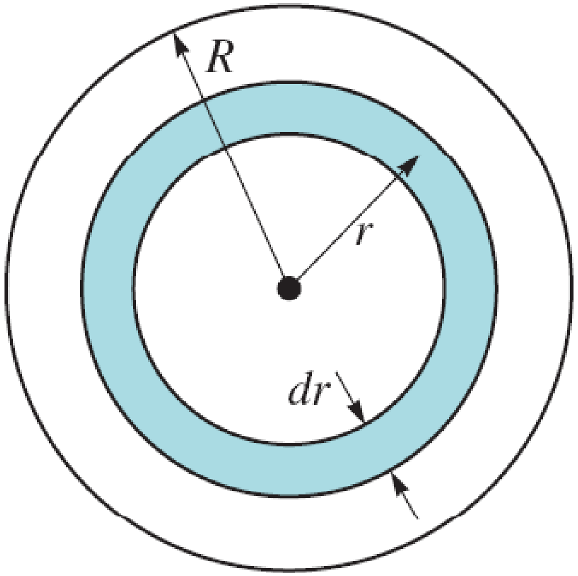
$$u = -\left( \frac{\Delta p}{4\mu l} \right) r^2 + C_1$$

boundary condition: $u = 0$ at $r = D/2 \Rightarrow C_1 = \left( \frac{\Delta p}{16\mu l} \right) l$

$$u(r) = \left( \frac{\Delta p D^2}{16\mu l} \right) \left[ 1 - \left( \frac{2r}{D} \right)^2 \right]$$

Flow rate:

$$Q = \int u dA = \int_0^{D/2} u(r) 2\pi r dr = \frac{\pi D^4 \Delta p}{128\mu l}$$



$dA = 2\pi r\ dr$

# 2nd Newton's law directly applied

- if gravity is present, it can be added to the pressure:

$$\frac{\Delta p-\rho g l \sin\theta}{l}=\frac{2\tau}{r}$$

$$V=\frac{\left(\Delta p-\rho g l \sin\theta\right)D^2}{32\mu l}$$

$$Q=\frac{\pi\left(\Delta p-\rho g l \sin\theta\right)D^4}{128\mu l}$$

# Navier-Stokes equation applied

$$\nabla \cdot \mathsf{V} = 0$$

$$\frac{\partial \boldsymbol{V}}{\partial t} + \mathsf{V} \cdot \nabla \mathsf{V} = -\frac{\Delta p}{\rho} + \boldsymbol{g} + v \nabla^{2} \boldsymbol{V}$$

in cylindrical coordinates:

$$\frac{\partial p}{\partial x} + \rho g \sin \theta = \mu \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u}{\partial r} \right)$$

boundary conditions:

$$\left.\frac{\partial u}{\partial r}\right|_{0} = 0;\ u(R) = 0$$

- The assumptions and the result are exactly the same as Navier-Stokes equation is drawn from 2nd Newton law

# Creeping flow in microchannels

- Let’s consider flow with very small Re numbers

$$\nabla \cdot \mathrm{V} = 0$$

$$\frac{\partial \boldsymbol{V}}{\partial t} + \mathrm{V} \cdot \nabla \mathrm{V} = -\frac{\Delta p}{\rho} + \boldsymbol{g} + v\nabla^2 \boldsymbol{V}$$

- DV/Dt part (non-linear) can be neglected leading to Stokes equation:

$$\nabla p = v\nabla^2 \vec{\boldsymbol{V}} + \frac{\vec{\boldsymbol{F}}}{\rho}$$

- equation is linear, and therefore is reversible. change of the velocity direction at the boundary will lead to change of the velocity direction in the whole domain

# Dimensional analysis applied

$$\Delta p = F\left(V, l, D, \mu\right)$$

$$\frac{D\Delta p}{\mu V} = \phi\left(\frac{l}{D}\right)$$

assuming pressure drop proportional to the length:

$$\frac{D\Delta p}{\mu V} = \frac{Cl}{D} \qquad \Rightarrow \qquad \frac{\Delta p}{l} = \frac{C\mu V}{D^2}$$

$$\boxed{Q = AV = \frac{(\pi / 4C)\Delta p D^4}{\mu l}}$$

# Dimensional analysis of pipe flow

- **major loss** in pipes: due to viscous flow in the straight elements
- **minor loss**: due to other pipe components (junctions etc.)

## Major loss:

$$\Delta p = F(V, D, l, \varepsilon, \mu, \rho)$$

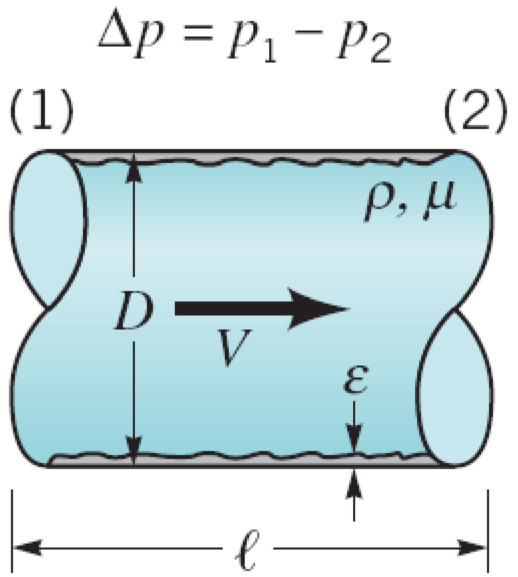
roughness

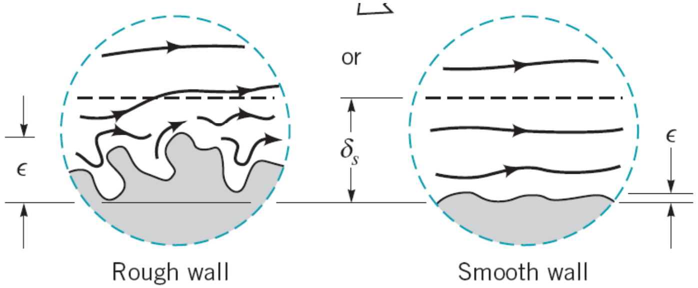
- those 7 variables represent complete set of parameters for the problem

$$\frac{\Delta p}{\frac{1}{2}\rho V^2} = \tilde{\phi}\left(\frac{\rho V D}{\mu}, \frac{l}{D}, \frac{\varepsilon}{D}\right)$$

as pressure drop is proportional to length of the tube

$$\frac{\Delta p}{\frac{1}{2}\rho V^2} = \frac{l}{D}\phi\left(\mathrm{Re}, \frac{\varepsilon}{D}\right)$$





# Dimensional analysis of pipe flow

friction factor

$$\frac{\Delta p}{\frac{1}{2}\rho V^2} = \frac{l}{D}\phi\left(\text{Re}, \frac{\varepsilon}{D}\right) \qquad \overset{\text{friction factor}}{\downarrow}\; f = \frac{\Delta p D}{\frac{1}{2} l \rho V^2}$$

$$f = \phi\left(\text{Re}, \frac{\varepsilon}{D}\right) \quad \text{and} \quad \Delta p = f\frac{l}{D}\frac{\rho V^2}{2}$$

- for fully developed laminar flow in a circular pipe:

$$f = 64/\text{Re}$$

- for fully developed steady incompressible flow (from Bernoulli eq.):

$$h_{L\,major} = \frac{\Delta p}{\rho g} = f\frac{l}{D}\frac{V^2}{2g}$$

# Non-circular ducts

- Reynolds number based on hydraulic diameter:

$$\mathrm{Re}_h = \frac{\rho V D_h}{\mu} \qquad D_h = \frac{4A}{P}$$

cross-section

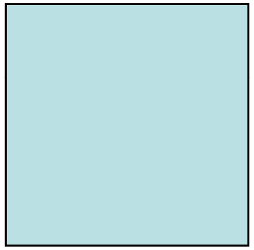
wetted perimeter

- Friction factor for noncircular ducts:

for fully developed laminar flow: $f = C / \mathrm{Re}$

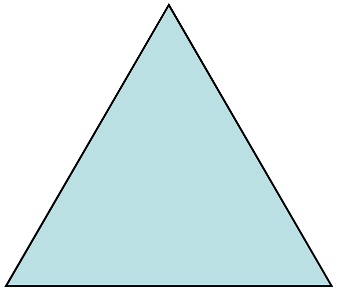
$$S\Delta p = \tau_w \mathrm{P} L; \qquad f = \frac{\tau_w}{\left(\dfrac{\rho V^2}{2}\right)}$$

$$\Delta p = f \frac{\mathrm{P} l}{4S} \frac{\rho V^2}{2} = f \frac{l}{D_h} \frac{\rho V^2}{2}$$



$$D_h = a$$

$$f = \frac{56.8}{\mathrm{Re}_{Dh}}$$

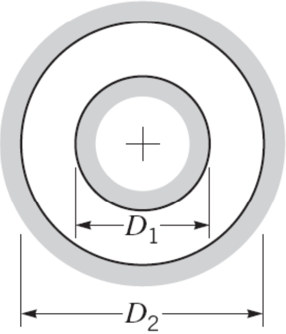
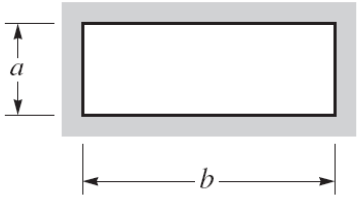


$$D_h = a/\sqrt{3}$$

$$f = \frac{53.2}{\mathrm{Re}_{Dh}}$$

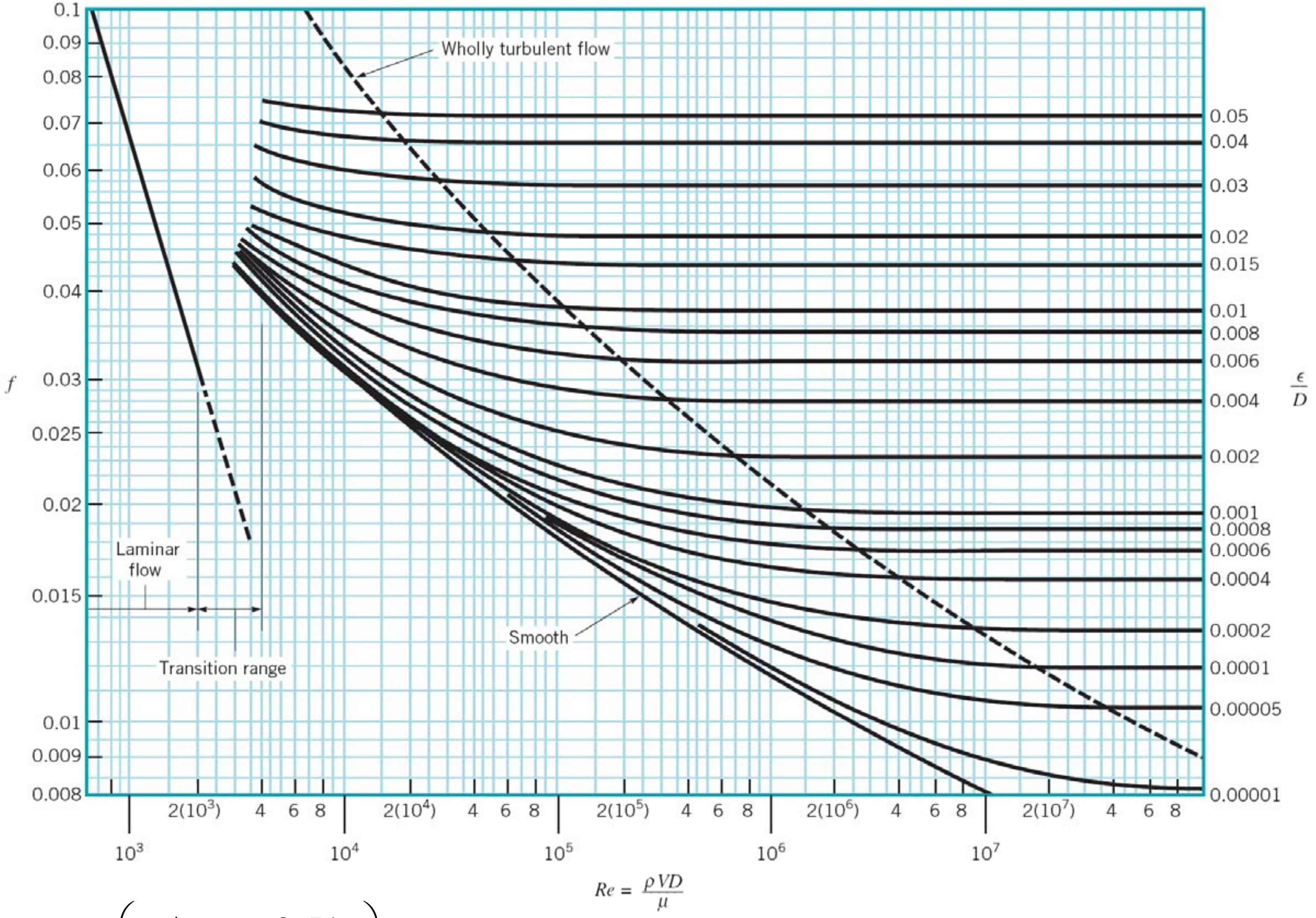
# Non-circular ducts

- Friction factor for noncircular ducts: $f = C / \mathrm{Re}$

| **Shape** | **Parameter** | $C = f\,\mathrm{Re}_h$ |
|---|---|---|
| I. Concentric Annulus<br>$D_h = D_2 - D_1$<br><br> | $D_1/D_2$ | |
| | 0.0001 | 71.8 |
| | 0.01 | 80.1 |
| | 0.1 | 89.4 |
| | 0.6 | 95.6 |
| | 1.00 | 96.0 |
| II. Rectangle<br>$D_h = \dfrac{2ab}{a + b}$<br><br> | $a/b$ | |
| | 0 | 96.0 |
| | 0.05 | 89.9 |
| | 0.10 | 84.7 |
| | 0.25 | 72.9 |
| | 0.50 | 62.2 |
| | 0.75 | 57.9 |
| | 1.00 | 56.9 |

# Moody chart

Friction factor as a function of Reynolds number and relative roughness for round pipes



$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\mathrm{Re}\sqrt{f}}\right)$$

Colebrook formula

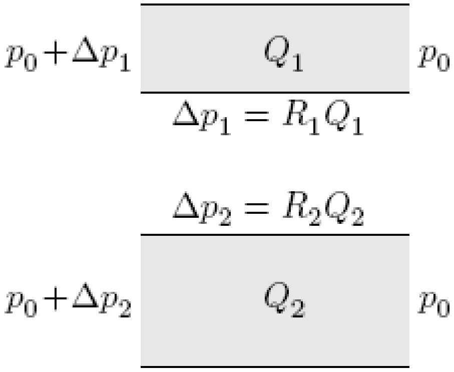
# Equivalent circuit theory

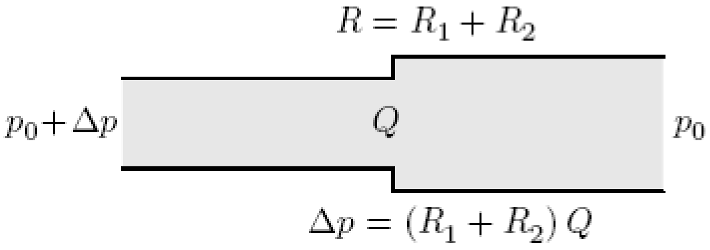
flow: $$\Delta p = R \times Q$$

$$\Updownarrow \qquad \Updownarrow \qquad \Updownarrow$$

electricity: $$V = R \times I$$

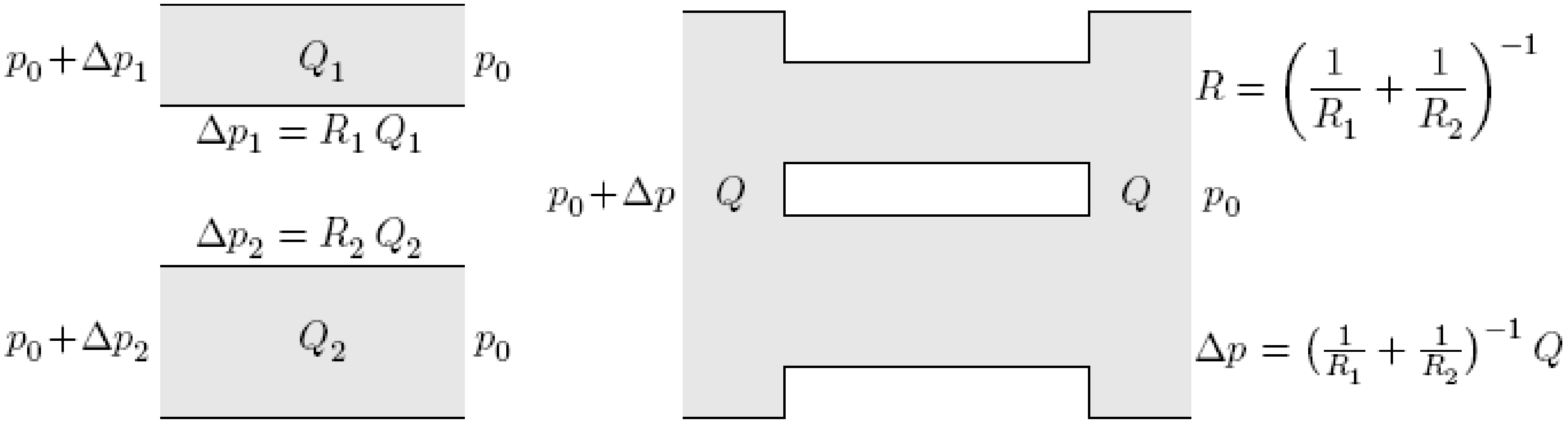
- channels connected in series





# Equivalent circuit theory

- channels connected in parallel



# Compliance

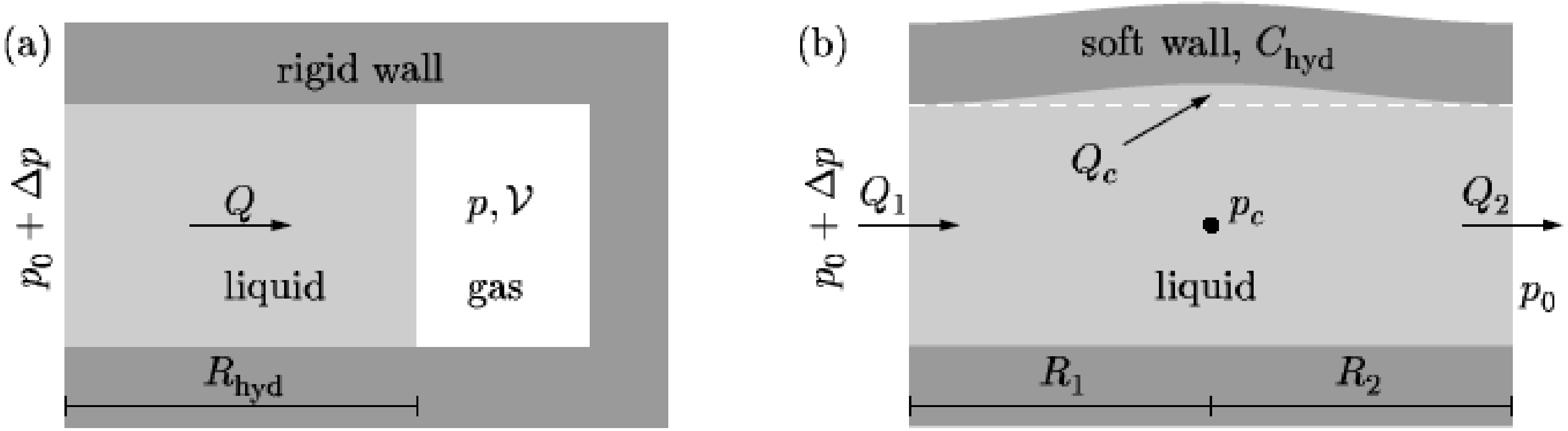
- compliance (hydraulic capacitance):

Q – volume ***V***/time

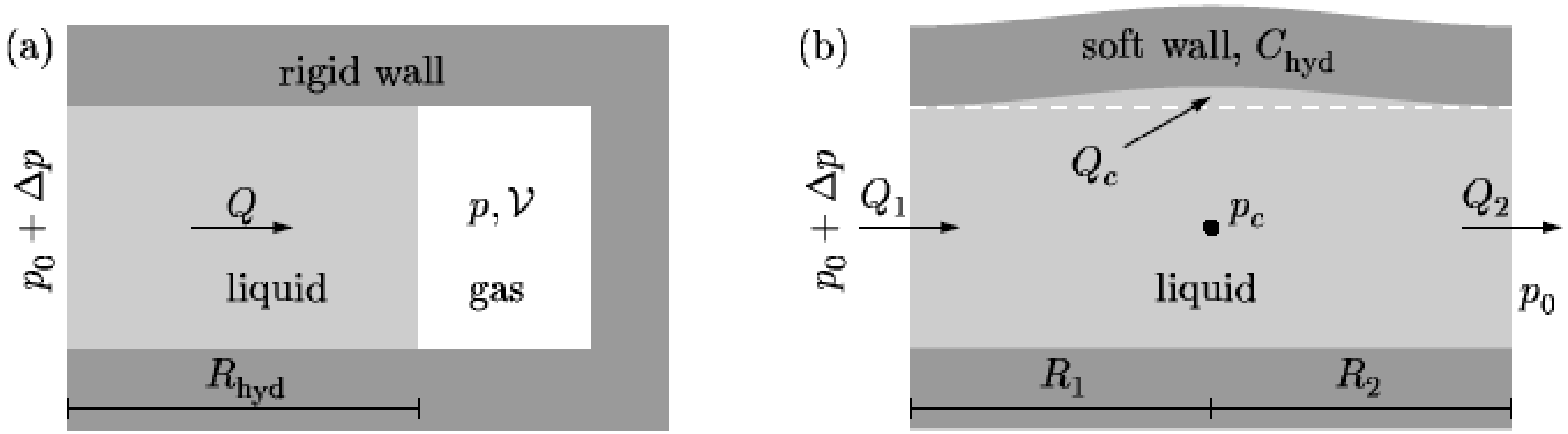
I – charge/time

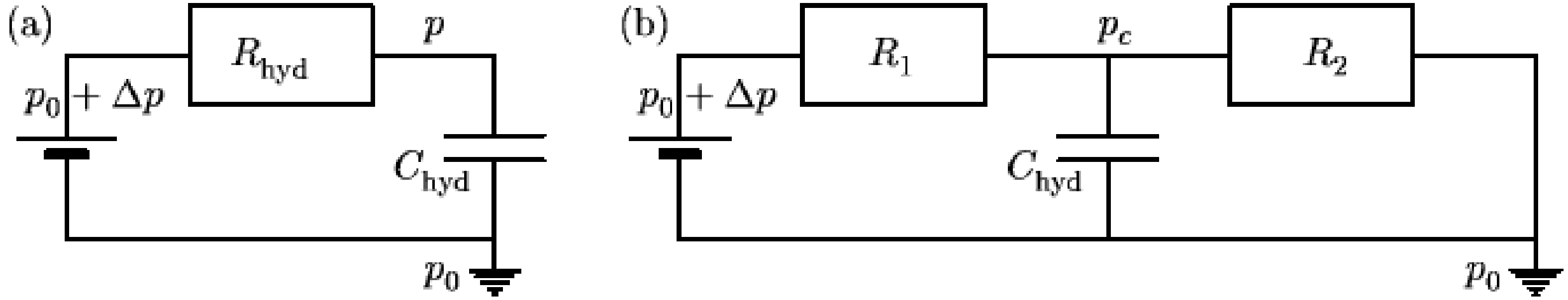
flow: $$C_{hyd} = -\frac{dV}{dp}$$

electricity: $$C = \frac{dq}{dU}$$



# Equivalent circuits





# Pipe networks

- Serial connection

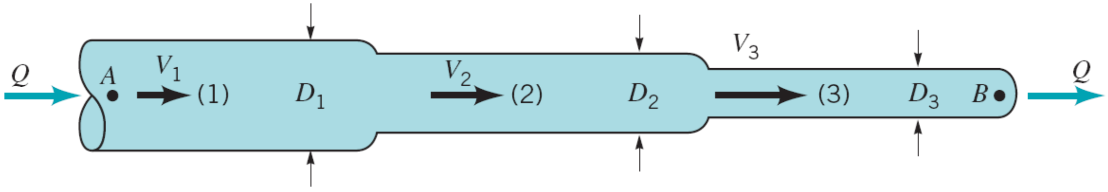
$$Q_1 = Q_2 = Q_3$$

$$h_{L_{A-B}} = h_{L_1} + h_{L_2} + h_{L_3}$$

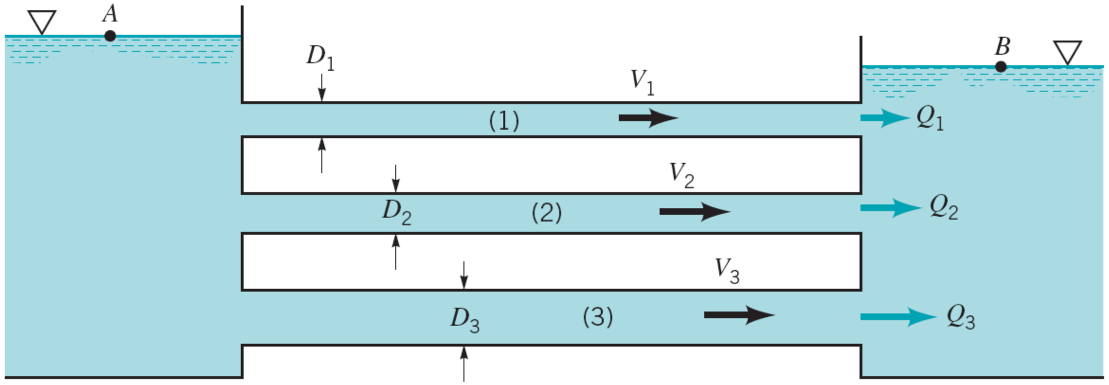
- Parallel connection

$$Q = Q_1 + Q_2 + Q_3$$

$$h_{L_1} = h_{L_2} = h_{L_3}$$



(a)



(b)

# COMPUTATIONAL FLUID DYNAMICS

# Introduction

- Computational fluid dynamics applications:
  - Aerodynamics of aircraft and vehicles
  - Hydrodynamics of ships
  - Microfluidics and biosensors
  - Chemical process engineering
  - Combustion engines and turbines
  - Construction: External and internal environment
  - Electric and electronic engineering: heating and cooling of circuits

- Advantages of CFD approach
  - Reduction of time and costs
  - Ability to do controlled experiment under difficult and hazardous condition
  - Unlimited level of detail
  - Possibility to couple several physical processes (momentum/mass/energy transfer, electrical/magnetic fields etc.)

# Governing equations for fluid dynamics

- <u>Mass conservation:</u>
  mass change in a volume is equal to the net rate of flow

$$\frac{\partial \rho}{\partial t}+div(\rho\vec{u})=0 \xrightarrow{\text{Uncompressible fluid}} div(\vec{u})=0$$

- <u>Moment conservation: Navier-Stokes equation</u>

$$\rho(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z})=-\frac{\partial p}{\partial x}+\rho g_x+\mu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2})$$

$$\rho(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z})=-\frac{\partial p}{\partial y}+\rho g_y+\mu(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2})$$

$$\rho(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z})=-\frac{\partial p}{\partial z}+\rho g_z+\mu(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2})$$

# Discretization of the equations

- To obtain the solution the continuous non-linear equations are discretized and converted to algebraic equations.
- Discritization techniques:
  - Finite difference
  - Finite volume (finite element)
  - Boundary elements

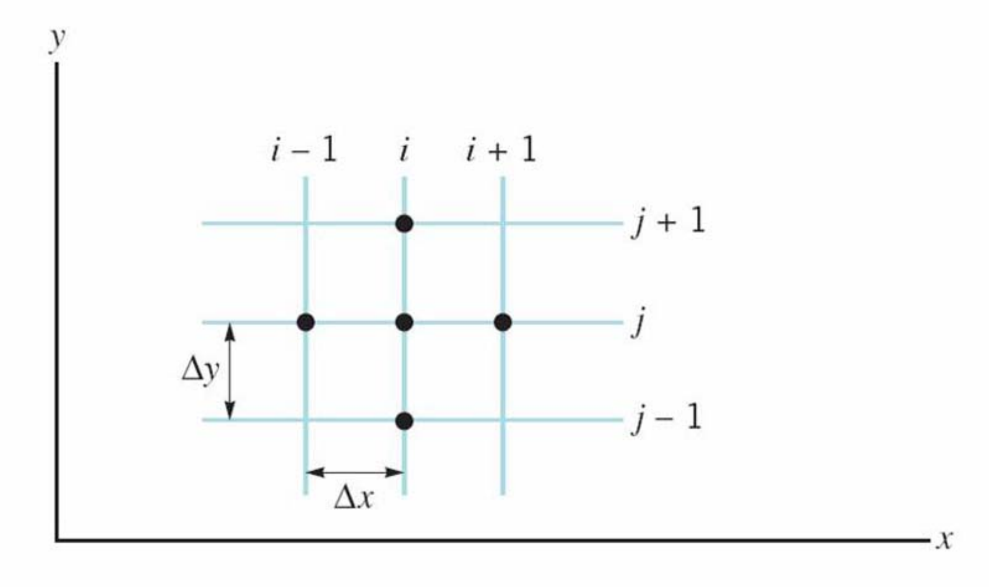
# Discretization Techniques

- Finite difference
  - Differential equations are converted to algebraic through the use of Taylor series expansion

$$u_{i+1,j} = u_{i,j} + \left(\frac{\partial u}{\partial x}\right)_{i,j} \frac{\Delta x}{1!} + \left(\frac{\partial^2 u}{\partial x^2}\right)_{i,j} \frac{\Delta x^2}{2!} + \ldots$$

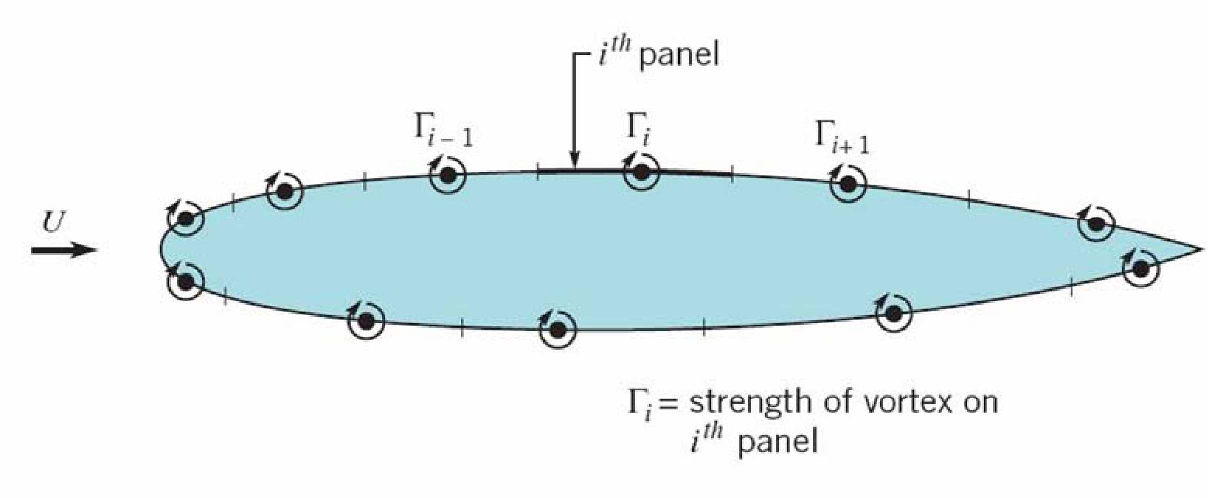
$$\left(\frac{\partial u}{\partial x}\right)_{i,j} = \frac{u_{i+1,j} - u_{i,j}}{\Delta x} + O(\Delta x)$$

- How shall we represent the second derivative?



# Discretization Techniques

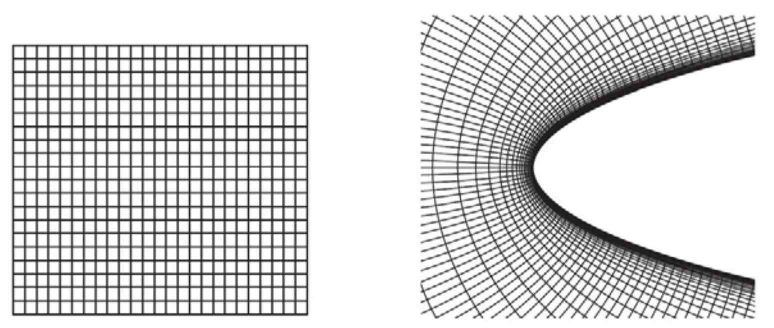
- Finite element
  - Similar to finite difference method, continuous functions are replaced by piecewise approximations valid on particular grid element
- Finite volume
  - Control Volume form of NSE is used on every grid element
- Boundary element method
  - Boundary is broken into discrete segments (panels), appropriate singularities (sources, sinks, doublets, vortices) are distributed along the segments



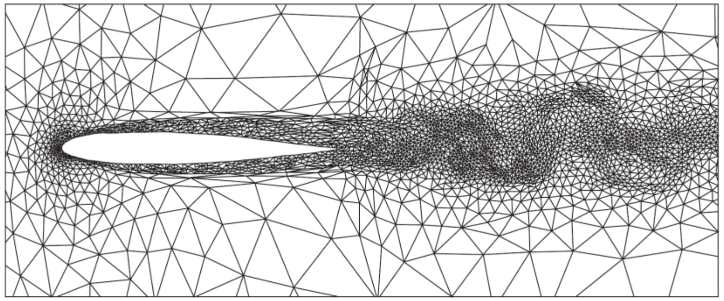
# CFD Methodology

finite element method:

1. Choose appropriate physical model
2. Define the geometry.
3. Define the **mesh** (grid): flow field is broken into set of elements
    - **Mesh could be structured (regular pattern) or unstructured. Other types could be hybrid (several structured elements), moving (time dependent)**

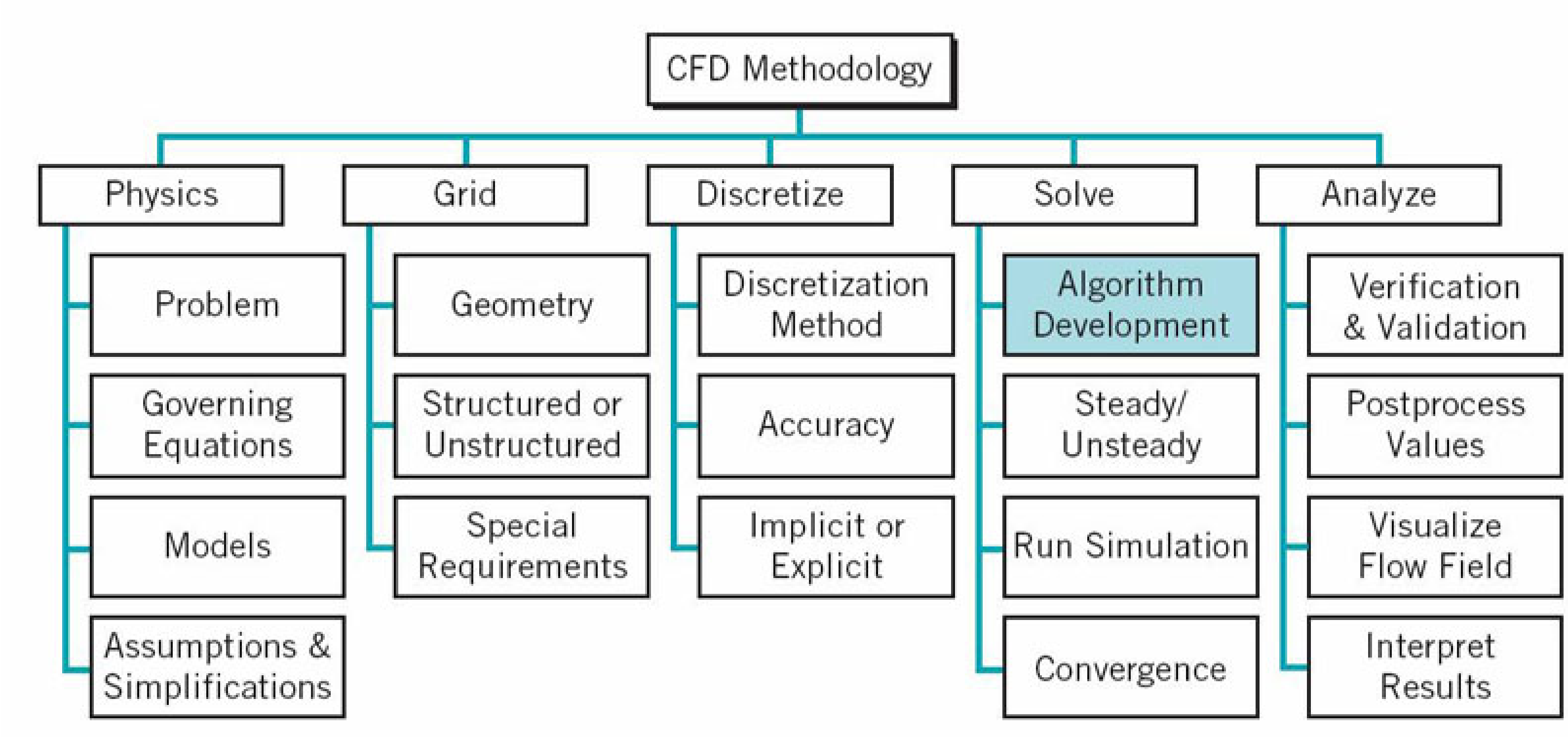

(a)

(b)



4. Define the **boundary conditions**
5. **Solving:** conservation equations are (mass, momentum and energy) are written for every element and solved.
6. **Postprocessing**: solution is visualized and hopefully understood

# CFD Methodology



CFD Methodology

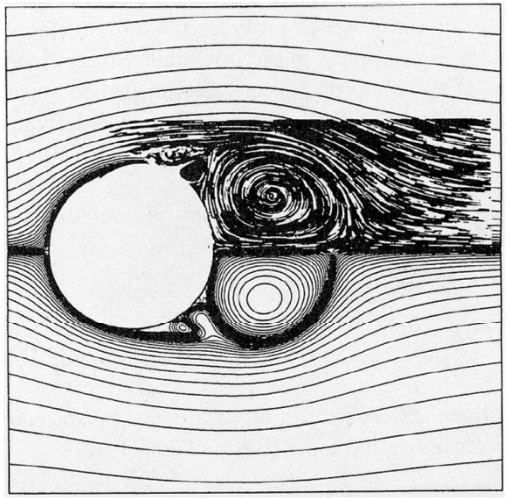
Physics
- Problem
- Governing Equations
- Models
- Assumptions & Simplifications

Grid
- Geometry
- Structured or Unstructured
- Special Requirements

Discretize
- Discretization Method
- Accuracy
- Implicit or Explicit

Solve
- Algorithm Development
- Steady/ Unsteady
- Run Simulation
- Convergence

Analyze
- Verification & Validation
- Postprocess Values
- Visualize Flow Field
- Interpret Results



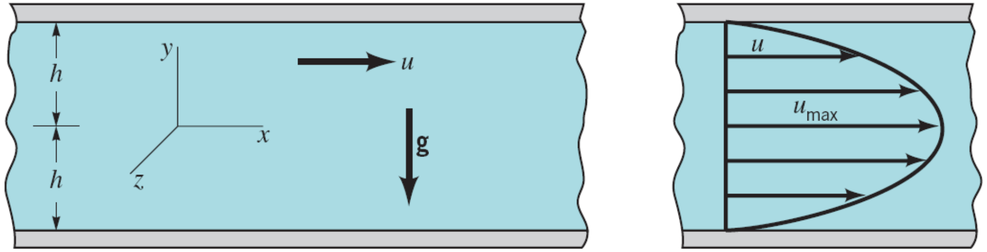
- Common problems:
  - Convergence issues
  - Difficulties in obtaining the quality grid and managing the resources
  - Difficulties in turbulent flow situations
- Verification
  - Using other techniques

# Flow in a 2D pipe

- Can be solved analytically:

Boundary condition (no slip) $u(h) = u(-h)$

Velocity profile $$u = \frac{1}{2\mu}\frac{\partial p}{\partial x}(y^2 - h^2)$$



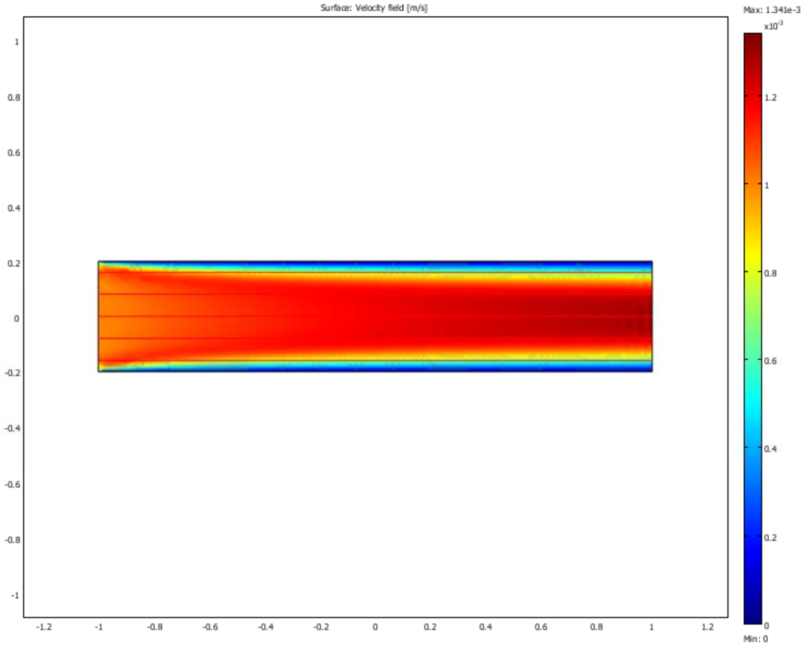
(a) (b)

---

Problem:
Solve analytically and numerically and compare.

Parameters
V=0.001m/s; h=0.2m; m=$10^{-3}$ Pa*s

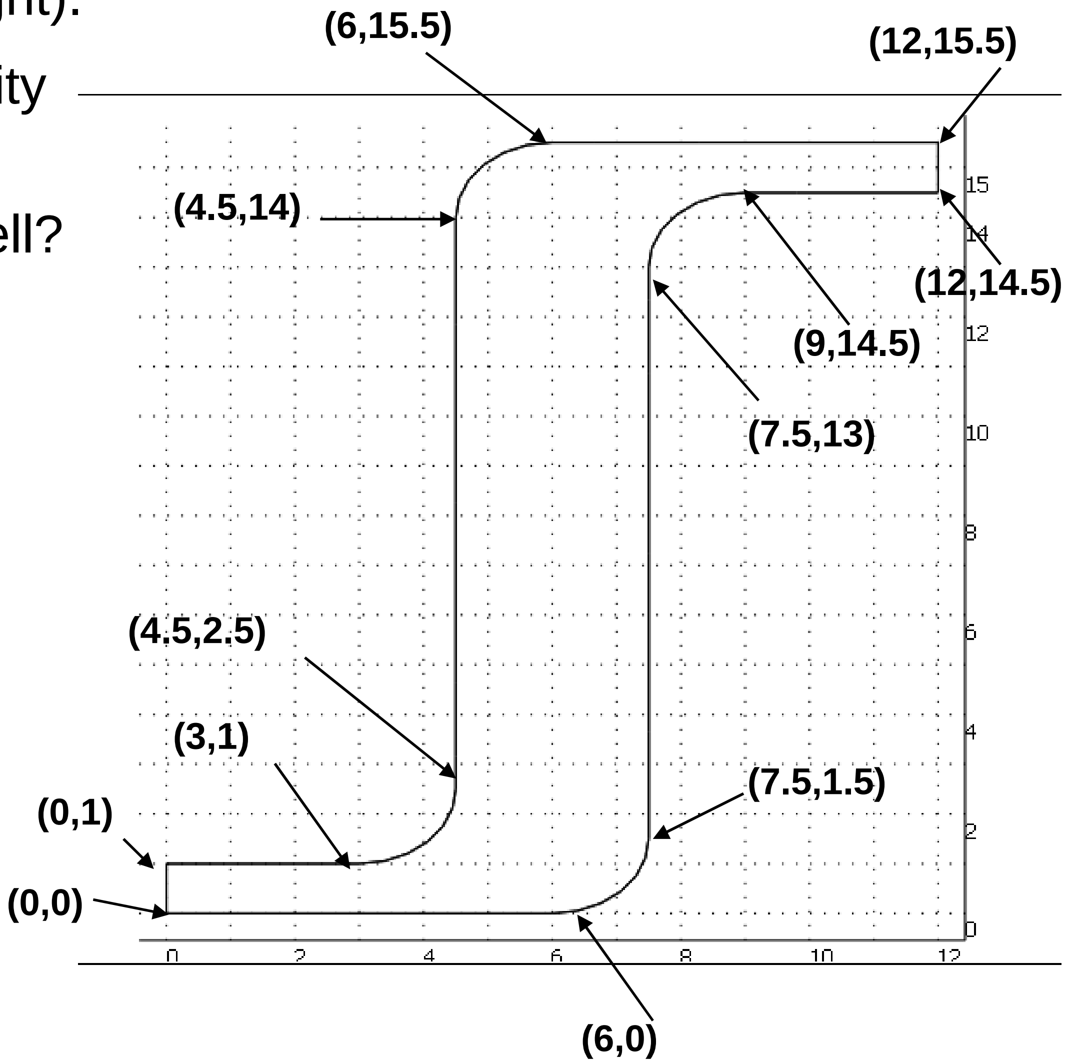
Questions:
- What is the Reynolds number?
- What is the calculated pressure drop in the pipe? What is the entrance pressure drop?
- Is laminar flow fully developed?



# S-cell Problem (home work)

- Calculate velocity field in an S-cell (3D fluidic cell, 50mm height).
- What would flow field uniformity across a 1mmx1mm array

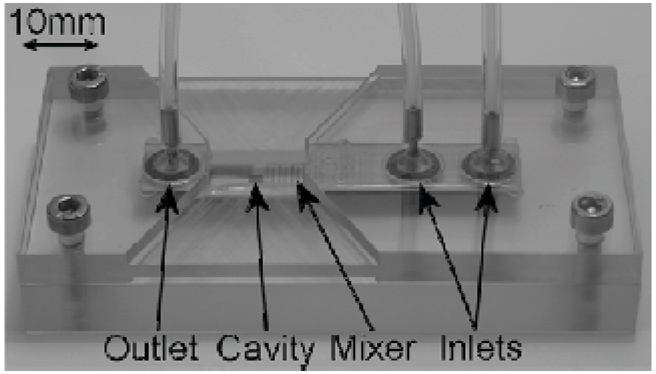
  spotted in the middle of the cell?



**Data:**
Flow rate: 100ul/min
Liquid: water, T=298K
Channel height (z): 50um
Channel width: 1mm in/outlet;
3mm chamber
Curvature radius: 1.5mm

# Problems

- Ethanol solution of a dye (μ=1.197 mPa·s) is used to feed a fluidic lab-on-chip laser. Dimension of the channel are L=122mm, width w=300um, height h=10 um. Calculate pressure required to achieve flow rate of Q=10μl/h.



- 8.7 A soft drink with properties of 10 ºC water is sucked through a 4mm diameter 0.25m long straw at a rate of 4 cm3/s. Is the flow at outlet laminar? Is it fully developed?

- Calculate total resistance of a microfluidic circuit shown. Assume that the pressure on all channels is the same and equal Δp.