Simulation of flow over a bridge using OpenFOAM

Sahl Abdelsayed

University of Sheffield

May 6, 2019

Outline

Introduction

What is OpenFOAM?

OpenFOAM (Open source Field Operation And Manipulation) is a free, open-source, customisable CFD numerical solver.

Introduction

What is OpenFOAM?

OpenFOAM (Open source Field Operation And Manipulation) is a free, open-source, customisable CFD numerical solver.

We need some context...

A bit of context

In 1879, Tay bridge in Dundee Scotland has collapsed due to a violent storm [1].

A bit of context

Tacoma Narrows Bridge has collapsed in an unexpected manner in 1940.

More about the Tacoma Bridge

The initial theory of failure was that the bridge collapsed due to resonance.

More about the Tacoma Bridge

The initial theory of failure was that the bridge collapsed due to resonance.

This theory turned out to be incorrect and the failure was due to aeroelastic flutter [2].

More about the Tacoma Bridge

The initial theory of failure was that the bridge collapsed due to resonance.

This theory turned out to be incorrect and the failure was due to aeroelastic flutter [2].

Engineering aim of this project is..

Improving the fundamental understanding of flow past a bridge by carrying out the appropriate simulations.

More about the Tacoma Bridge

The initial theory of failure was that the bridge collapsed due to resonance.

This theory turned out to be incorrect and the failure was due to aeroelastic flutter [2].

Engineering aim of this project is..

Improving the fundamental understanding of flow past a bridge by carrying out the appropriate simulations.

As we all know, simulations bring about numerous advantages.

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.
- Run and analyse simulations on Re=20, Re=50, Re=100.

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.
- Run and analyse simulations on Re=20, Re=50, Re=100.

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.
- Run and analyse simulations on Re=20, Re=50, Re=100.

Assumptions

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.
- Run and analyse simulations on Re=20, Re=50, Re=100.

Assumptions

- Flow is laminar.

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.
- Run and analyse simulations on Re=20, Re=50, Re=100.

Assumptions

- Flow is laminar.
- Flow is incompressible.

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.
- Run and analyse simulations on Re=20, Re=50, Re=100.

Assumptions

- Flow is laminar.
- Flow is incompressible.
- Air is a Newtonian fluid.

Aims and assumptions

Project aims

- Simplify the geometry to a simple 2D cylinder model.
- Run and analyse simulations on Re=20, Re=50, Re=100.

Assumptions

- Flow is laminar.
- Flow is incompressible.
- Air is a Newtonian fluid.
- Flow is transient, single phase, and isothermal.

Geometry

The geometry was drawn on an application called **gmsh**.

Geometry

The geometry was drawn on an application called **gmsh**.

The cylinder has a diameter of 1m.

Geometry

The geometry was drawn on an application called **gmsh**.

The cylinder has a diameter of 1m.

The outline of the 2D cylinder is drawn shown in Figure 3.1.

Geometry

The geometry was drawn on an application called **gmsh**.

The cylinder has a diameter of 1m.

The outline of the 2D cylinder is drawn shown in Figure 3.1.

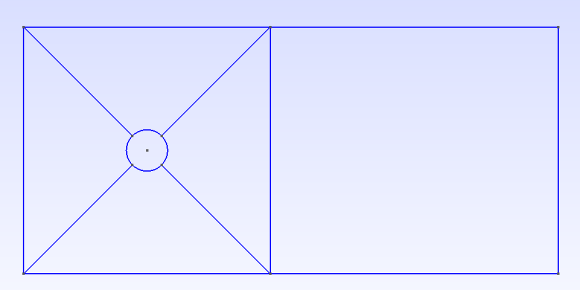


Figure 3.1: Shows the outline of the cylinder

Geometry

The next stage involved extruding the geometry.

Geometry

The next stage involved extruding the geometry.

Extrusion is important as it will add surfaces as shown in Figure 3.2.

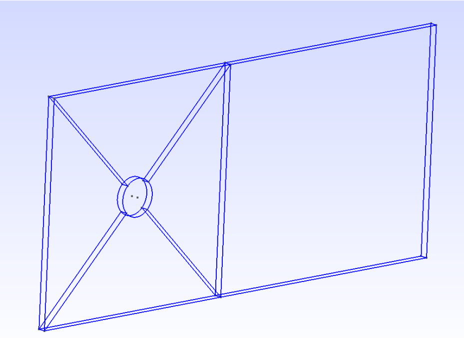


Figure 3.2: Shows the extruded cylinder.

Geometry

After the extrusion the geometry was meshed.

Geometry

After the extrusion the geometry was meshed.

The result obtained are shown in Figure 3.3.

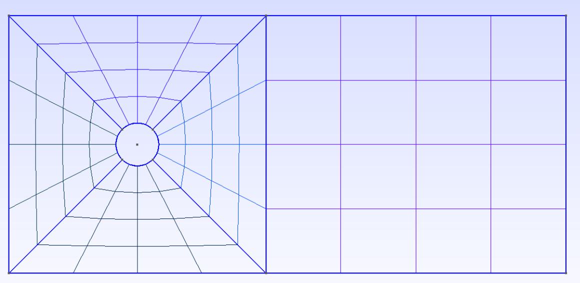


Figure 3.3: Shows the cylinder with the applied meshing.

Geometry

Afterwards, the surfaces were labelled as shown on Figure 3.4

Geometry

Afterwards, the surfaces were labelled as shown on Figure 3.4

This is an important step as it plays a big role in setting the boundary conditions.

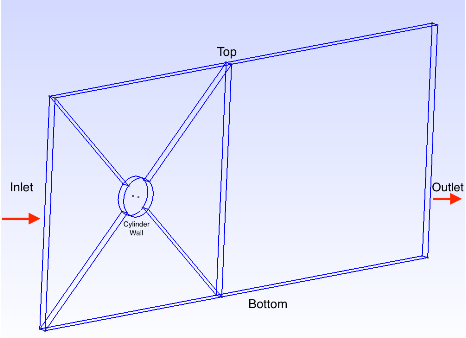


Figure 3.4: Shows the final geometry with labelled surfaces

Boundary conditions

Boundary conditions

Velocity boundary conditions:

Boundary conditions

Velocity boundary conditions:

- $1\ \mathrm{m\,s^{-1}}$ in the x-direction at the inlet, to be able to apply non-dimensionalisation.

Boundary conditions

Velocity boundary conditions:

- $1\ \mathrm{m\,s^{-1}}$ in the x-direction at the inlet, to be able to apply non-dimensionalisation.
- $0\ \mathrm{m\,s^{-1}}$ at cylinder wall.

Boundary conditions

Velocity boundary conditions:

- $1\ \mathrm{m\,s^{-1}}$ in the x-direction at the inlet, to be able to apply non-dimensionalisation.
- $0\ \mathrm{m\,s^{-1}}$ at cylinder wall.
- Slip condition on top and bottom.

Boundary conditions

Velocity boundary conditions:

- $1\ \mathrm{m\,s^{-1}}$ in the x-direction at the inlet, to be able to apply non-dimensionalisation.
- $0\ \mathrm{m\,s^{-1}}$ at cylinder wall.
- Slip condition on top and bottom.
- Zero gradient at the outlet.

Boundary conditions

Velocity boundary conditions:

- $1\ \mathrm{m\,s^{-1}}$ in the x-direction at the inlet, to be able to apply non-dimensionalisation.
- $0\ \mathrm{m\,s^{-1}}$ at cylinder wall.
- Slip condition on top and bottom.
- Zero gradient at the outlet.
- Back and front wall were set as empty.

Boundary conditions

Pressure boundary conditions:

Boundary conditions

Pressure boundary conditions:

- The following surfaces were set as zero gradient:

Boundary conditions

Pressure boundary conditions:

- The following surfaces were set as zero gradient:
  - Top and bottom
  - Inlet
  - Cylinder wall

Boundary conditions

Pressure boundary conditions:

- The following surfaces were set as zero gradient:
  - Top and bottom
  - Inlet
  - Cylinder wall
- Back and front walls were set as empty

Boundary conditions

Pressure boundary conditions:

- The following surfaces were set as zero gradient:
  - Top and bottom
  - Inlet
  - Cylinder wall
- Back and front walls were set as empty
- Outlet was set to 0

Boundary conditions

Pressure boundary conditions:

- The following surfaces were set as zero gradient:
  - Top and bottom
  - Inlet
  - Cylinder wall
- Back and front walls were set as empty
- Outlet was set to 0

To understand why outlet was set to 0, the Navier-Stokes equation should be studied.

Navier-Stokes equations

[1] ρ is the density and $\overrightarrow{\mathbf{V}}$ is the velocity vector.

Navier-Stokes equations

The mass continuity equation can be written as [1]

[1] ρ is the density and $\overrightarrow{\mathbf{V}}$ is the velocity vector.

Navier-Stokes equations

The mass continuity equation can be written as [1]

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \overrightarrow{\mathbf{V}}) = 0 \tag{3.1}$$

Since the flow is incompressible Equation 3.1 can be written as

[1] ρ is the density and $\overrightarrow{\mathbf{V}}$ is the velocity vector.

Navier-Stokes equations

The mass continuity equation can be written as [1]

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \overrightarrow{\mathbf{V}}) = 0 \tag{3.1}$$

Since the flow is incompressible Equation 3.1 can be written as

$$\nabla \cdot (\overrightarrow{\mathbf{V}}) = 0 \tag{3.2}$$

[1] ρ is the density and $\overrightarrow{\mathbf{V}}$ is the velocity vector.

Navier-Stokes equations

Navier-stokes momentum equation can be expressed as [2]

$$\frac{\partial(\rho\overrightarrow{\mathbf{V}})}{\partial t} + \rho\nabla\cdot(\overrightarrow{\mathbf{V}}\,\overrightarrow{\mathbf{V}}) = -\nabla\overline{P} + \mu(\nabla^2\overrightarrow{\mathbf{V}}) \qquad (3.3)$$

Dividing by ρ on both sides Equation 3.3 can be written as [3]

$$\frac{\partial(\overrightarrow{\mathbf{V}})}{\partial t} + \nabla\cdot(\overrightarrow{\mathbf{V}}\,\overrightarrow{\mathbf{V}}) = -\nabla\overline{p} + \nu(\nabla^2\overrightarrow{\mathbf{V}}) \qquad (3.4)$$

[2] ρ is the density, $\overrightarrow{\mathbf{V}}$ is the velocity vector, $\overline{P}$ is the pressure, and μ is the dynamic viscosity.

[3] $\overline{p}$ is the pressure per unit density and ν is the kinematic viscosity.

Non-dimensionalisation

Definition

Non-dimensionalisation is the process of eliminating units by suitable substitution.

Non-dimensionalisation facilitated setting the Reynolds number of each flow.

Non-dimensionalisation

For instance, Reynolds can be calculated by the following expression. [4]

[4] D is the diameter of the cylinder and u is the velocity in the x-direction

Non-dimensionalisation

For instance, Reynolds can be calculated by the following expression. [4]

$$Re = \frac{uD}{\nu} \tag{3.5}$$

[4] D is the diameter of the cylinder and u is the velocity in the x-direction

Non-dimensionalisation

For instance, Reynolds can be calculated by the following expression. [4]

$$Re = \frac{uD}{\nu} \tag{3.5}$$

Rearranging the equation to make ν subject of the formula Equation 3.5 becomes

[4] D is the diameter of the cylinder and u is the velocity in the x-direction

Non-dimensionalisation

For instance, Reynolds can be calculated by the following expression. [4]

$$Re = \frac{uD}{\nu} \tag{3.5}$$

Rearranging the equation to make ν subject of the formula Equation 3.5 becomes

$$\nu = \frac{uD}{Re} \tag{3.6}$$

[4] D is the diameter of the cylinder and u is the velocity in the x-direction

Non-dimensionalisation

For instance, Reynolds can be calculated by the following expression. [4]

$$Re = \frac{uD}{\nu} \tag{3.5}$$

Rearranging the equation to make ν subject of the formula Equation 3.5 becomes

$$\nu = \frac{uD}{Re} \tag{3.6}$$

It is already established that u$= 1\,\mathrm{m\,s^{-1}}$ and D$= 1\mathrm{m}$

[4] D is the diameter of the cylinder and u is the velocity in the x-direction

Non-dimensionalisation

For instance, Reynolds can be calculated by the following expression. [4]

$$Re = \frac{uD}{\nu} \tag{3.5}$$

Rearranging the equation to make ν subject of the formula Equation 3.5 becomes

$$\nu = \frac{uD}{Re} \tag{3.6}$$

It is already established that u$=$ $1\,\mathrm{m\,s^{-1}}$ and D$=$ 1m

Hence...

[4] D is the diameter of the cylinder and u is the velocity in the x-direction

Non-dimensionalisation

For instance, Reynolds can be calculated by the following expression. [4]

$$Re = \frac{uD}{\nu} \tag{3.5}$$

Rearranging the equation to make ν subject of the formula Equation 3.5 becomes

$$\nu = \frac{uD}{Re} \tag{3.6}$$

It is already established that u= $1\,\mathrm{m\,s^{-1}}$ and D= 1m

Hence...

$$\nu = \frac{1}{Re} \tag{3.7}$$

[4] D is the diameter of the cylinder and u is the velocity in the x-direction

Non-dimensionalisation

Non-dimensionalisation

The required viscosity can now be calculated using Equation 3.7.

Non-dimensionalisation

The required viscosity can now be calculated using Equation 3.7.

Therefore, to obtain Reynolds number of 20, 50, and 100.

Non-dimensionalisation

The required viscosity can now be calculated using Equation 3.7.

Therefore, to obtain Reynolds number of 20, 50, and 100.

Kinematic viscosity should be set at 0.05, 0.02, and 0.01 respectively.

Nature of the forces acting

[5] A is the surface area

Nature of the forces acting

Two of the fundamental forces acting in any aerodynamic problems are:

[5] A is the surface area

Nature of the forces acting

Two of the fundamental forces acting in any aerodynamic problems are:

- Lift F_L, which acts in the y-direction
- Drag F_D, which acts in the x-direction

[5] A is the surface area

Nature of the forces acting

Two of the fundamental forces acting in any aerodynamic problems are:

- Lift F_L, which acts in the y-direction
- Drag F_D, which acts in the x-direction

Having obtained the forces, the coefficient of lift C_L and coefficient of drag C_D can be calculated. [5]

[5] A is the surface area

Nature of the forces acting

Two of the fundamental forces acting in any aerodynamic problems are:

- Lift F_L, which acts in the y-direction
- Drag F_D, which acts in the x-direction

Having obtained the forces, the coefficient of lift C_L and coefficient of drag C_D can be calculated. [5]

$$C_L = \frac{2F_L}{\rho u^2 A} \tag{4.1}$$

$$C_D = \frac{2F_D}{\rho u^2 A} \tag{4.2}$$

[5] A is the surface area

Time traces

To be able to study the variation of coefficient of lift and drag along time, non-dimensional time $\hat{t}$ was required. It can be calculated using Equation 4.3

Time traces

To be able to study the variation of coefficient of lift and drag along time, non-dimensional time $\hat{t}$ was required. It can be calculated using Equation 4.3

$$\hat{t} = \frac{tu}{D} \tag{4.3}$$

Time traces

To be able to study the variation of coefficient of lift and drag along time, non-dimensional time $\hat{t}$ was required. It can be calculated using Equation 4.3

$$\hat{t} = \frac{tu}{D} \tag{4.3}$$

We know that velocity u=$1\,\mathrm{m\,s^{-1}}$ and diameter D= 1m hence..

$$\hat{t} = t \tag{4.4}$$

Time traces were then obtained by plotting coefficient of lift and coefficient of drag vs non-dimensional time.

Vortex shedding frequency and Strouhal number

Vortex shedding frequency and Strouhal number

By applying Fourier transform to the coefficient of lift time trace obtained, the vortex shedding frequency f can be obtained.

Vortex shedding frequency and Strouhal number

By applying Fourier transform to the coefficient of lift time trace obtained, the vortex shedding frequency f can be obtained.

Having obtained the vortex shedding frequency, Strouhal number can be calculated using Equation 4.5

$$St = \frac{fD}{u} \tag{4.5}$$

Being given that u= $1\,\mathrm{m\,s^{-1}}$ and D= 1m Equation 4.5 simplifies to

$$St = f \tag{4.6}$$

The entire methodology process was done on MATLAB.

Velocity profile at Re=20

Stable flow and short separation length.

Velocity profile at Re=20

Stable flow and short separation length.

Pressure profile at Re=20

Pressure profile at Re=20

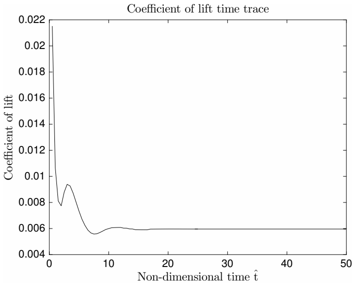
Constant difference in pressure, which creates constant force.

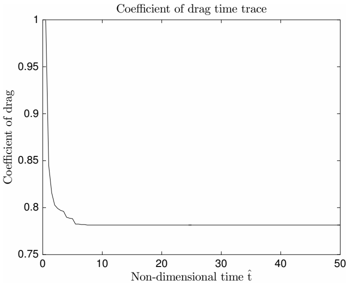
Pressure profile at Re=20

Constant difference in pressure, which creates constant force.

Time traces obtained at Re=20.

Having obtained the forces, time traces were plotted as shown below

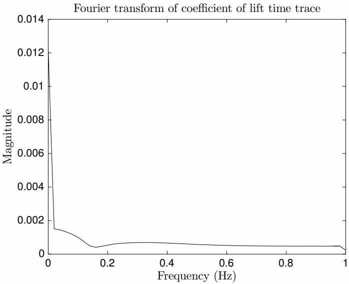




$C_L = 0.006$
$C_D = 0.7815$

Vortex shedding and Strouhal Number at Re=20

Applying Fourier transformation to coefficient of lift time trace we get..



No peaks observed, hence no vortex shedding

Velocity profile at Re=50

Velocity profile at Re=50

Initial oscillation occurs and larger separation length.

Velocity profile at Re=50

Initial oscillation occurs and larger separation length.

Pressure profile at Re=50

Pressure profile at Re=50

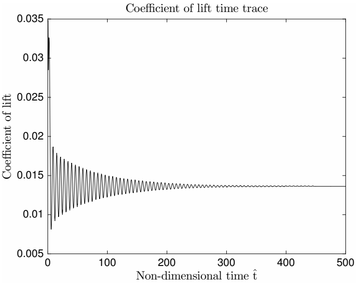
Slight fluctuation in pressure creates a slightly varying force.

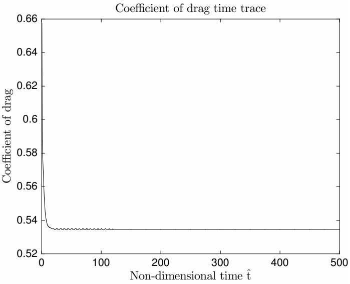
Pressure profile at Re=50

Slight fluctuation in pressure creates a slightly varying force.

Time traces obtained at Re=50

The time traces now can be plotted.



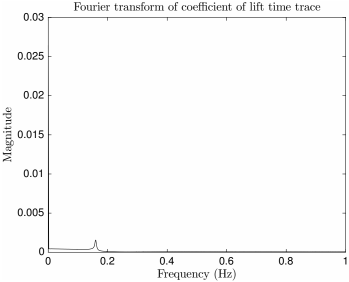


$C_L = 0.0136$

$C_D = 0.5345$

Vortex shedding and Strouhal Number Re=50

Applying Fourier transformation to coefficient of lift time trace we get..



$$\mathrm{St} = \mathrm{f} = 0.16$$

Velocity profile at Re=100

Velocity profile at Re=100

Airflow constantly oscillating.

Velocity profile at Re=100

Airflow constantly oscillating.

Pressure profile at Re=100

Pressure profile at Re=100

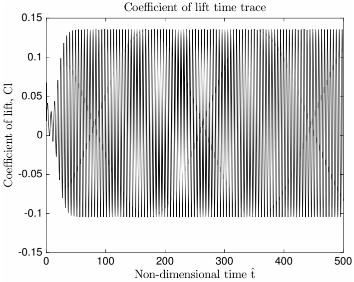
Fluctuation in pressure creates a fluctuating force.

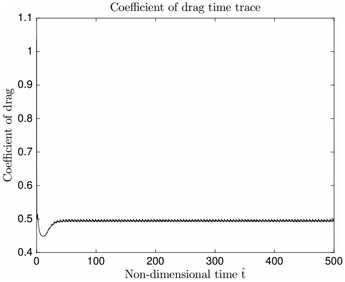
Pressure profile at Re=100

Fluctuation in pressure creates a fluctuating force.

Time traces obtained at Re=100

Time traces obtained are shown below.



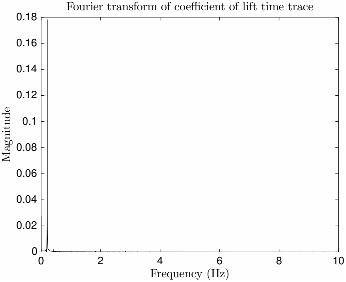


$C_L = 0.136 \leq C_l \leq$ -0.1048

$C_D = 0.493$

Vortex shedding and Strouhal number at Re=100

Applying Fourier transformation to coefficient of lift time trace we get..



St = f = 0.202

Velocity profiles

As Reynold's number increases:

Velocity profiles

As Reynold's number increases:

- Velocities at top and bottom of cylinder increases.

Velocity profiles

As Reynold's number increases:

- Velocities at top and bottom of cylinder increases.
- Seperation length increases.

Velocity profiles

As Reynold's number increases:

- Velocities at top and bottom of cylinder increases.
- Seperation length increases.

Beyond Reynold's number of 50, vortex shedding starts occuring.

Velocity profiles

As Reynold's number increases:

- Velocities at top and bottom of cylinder increases.
- Seperation length increases.

Beyond Reynold's number of 50, vortex shedding starts occuring.

Pressure profiles

For Re=20, Re=50, Re=100:

Pressure profiles

For Re=20, Re=50, Re=100:

Relative pressure per unit density at leading edge was 0.74, 0.619, and 0.542 m^2s^{-2} respectively.

Pressure profiles

For Re=20, Re=50, Re=100:

Relative pressure per unit density at leading edge was 0.74, 0.619, and 0.542 m^2s^{-2} respectively.

The pressure difference on leading and trailing edge generates drag force.

Pressure profiles

For Re=20, Re=50, Re=100:
Relative pressure per unit density at leading edge was 0.74, 0.619, and 0.542 m^2s^{-2} respectively.

The pressure difference on leading and trailing edge generates drag force.

While, the pressure difference on top and bottom of the cylinder generates the lift force.

Pressure profiles

For Re=20, Re=50, Re=100:

Relative pressure per unit density at leading edge was 0.74, 0.619, and 0.542 m^2s^{-2} respectively.

The pressure difference on leading and trailing edge generates drag force.

While, the pressure difference on top and bottom of the cylinder generates the lift force.

Fluctuation in pressure occurred in Reynolds number beyond 50.

Time traces and vortex shedding

	C_L	C_D	frequency f/Hz	Strouhal Number St
Re20	0.006	0.7815	none	none
Re50	0.0136	0.5345	0.16	0.16
Re100	0.136 $\leq C_l \leq$ -0.1048	0.493	0.202	0.202

Table 1: Shows the summary of results obtained.

Time traces and vortex shedding

	C_L	C_D	frequency f/Hz	Strouhal Number St
Re20	0.006	0.7815	none	none
Re50	0.0136	0.5345	0.16	0.16
Re100	$0.136 \leq C_l \leq -0.1048$	0.493	0.202	0.202

Table 1: Shows the summary of results obtained.

It can be established:

Time traces and vortex shedding

	C_L	C_D	frequency f/Hz	Strouhal Number St
Re20	0.006	0.7815	none	none
Re50	0.0136	0.5345	0.16	0.16
Re100	$0.136 \leq C_l \leq$ -0.1048	0.493	0.202	0.202

Table 1: Shows the summary of results obtained.

It can be established:

- As Reynolds number increases C_D decreases.
- As Reynolds number increases C_L increases.
- Vortex shedding starts forming from Reynolds number 50.
- As Reynold's number increases Strouhal number increases.

Next Step?

Next Step?

Further work to be done:

Next Step?

Further work to be done:

- Explore Turbulence models.
- Set Variable boundary conditions.
- Attempt to use a real bridge geometry.

OpenFOAM vs ANSYS

	OpenFOAM	**ANSYS**
Advantages	1) Free license and open source 2) Improves understanding of flow	1) Good user interface 2) More matured
Disadvantages	1) Lack of user interface 2) No structured learning guidelines	1) Expensive license 2) Can be used as a black box

References I

[1] **Biezma, María Victoria and Schanack, Frank**, "Collapse of steel bridges," *Journal of performance of constructed facilities*, vol. 21, no. 5, pp. 398–405, 2007.

[2] **Green, Daniel and Unruh, William G**, "The failure of the tacoma bridge: A physical model," *American journal of physics*, vol. 74, no. 8, pp. 706–716, 2006.

Thank you for your undivided attention.