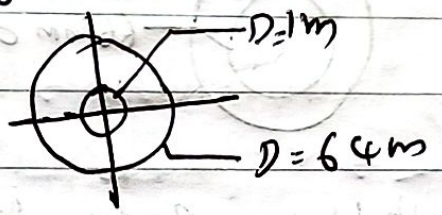
Flow over cylinder Geometry

Geometry:
Analysis type :- 2D
Units - m
XY plane - Sketching - sketching → Look at face
Circle → Draw
→ Dimensions



Modelling → Concept → Surface from sketches
Select the Sketch 1 as Base object → Apply
Operation - Add Frozen →
Generate

XY plane - New sketch → Sketch 2.
Sketching - Line → Draw vertical line through centre

Tool - Face split
Target face - select the surface
Tool Geometry - Select sketch 2 (on the line) → Apply.
Generate
Save
Close Geometry.

Flow over cylinder

Mesh

Physics preference - CFD, Solver preference - Fluent
Generate
Select all edges except the split lines
Mesh - insert - sizing
Type - no: of divisions
Set no: of divisions to 96
Select the split lines as edges
Mesh - insert - sizing
Type - no: of divisions
Set no of divisions to 96
Bias type ———
← Set Bias factor 460
Reverse bias - select lower split line - apply
Generate
Mesh - insert - face meshing
Select both faces as Geometry - Apply
Generate
Create named selection for inlet, outlet, wall
Select Body selection.
Select the geometry full as body and give name as fluid
Generate

Bias factor = ratio b/w largest and smallest element

If the divisions are not uniform, change mesh behaviour from left to hard.

To check mesh quality.

~~Change Mesh~~

Select Quality → Set Mesh metric to skewness.

Check if Max value is less than 0.9.

Save project

Close mesh

Setup

2D, double precision. → start.

General:

Solver - pressure based

Time - Transient

~~2D space~~ - Planar

Models

Viscous model - Laminar - OK

Materials:

Double click air - Set density to 1

Viscosity $8.333e^{-3}$

Change/create - close

Cell zone conditions:

Select fluid

Change type to fluid
Click edit → material name 'air' - apply - close

Boundary conditions :
inlet :- Momentum - Set velocity specification method to components
x velocity 1
y velocity 0
Apply → close

Outlet :- ~~Momentum~~ pressure outlet
Momentum - Gauge pressure 0

Wall :- → Stationary wall, No slip → close.

Solution :
Methods - scheme - SIMPLE
Pressure - Second order
Momentum - Second order upwind
Transient formulation - First order implicit

Report definition
Right click Report definition
- New - force report - Drag
- Edit Drag coefficient - Set name to cd

Set Force Vector as

X	Y	Z
1	0	1

Set wall under Zones ~~-OK~~

~~Unsetl~~ Unseled Report file

Tick only Report Plot

-OK

Similary ~~de~~ create a report definition for Drag force

Initialization

- Standard initialization

Gauge pressure 0, X velocity 0, Y velocity 0.

- Initialize.

Calculation Activities:

Autosave every (Time step) → edit

→ save Data Every 1. change Time Step to Flow time

→ Browse.

Autosave Case / Data file: Flow_over_a_cylinder.dat.gz

add it

To save only data file & not case file.

gz - compressed state.

OK.

~~Ok~~. Append file name with - Flow time

~~OK~~ Decimal places in file name 6

OK

~~Solution Animation~~

Results → Graphics → Contours → Contours of Velocity
→ Save / Display.

Zoom the display to the inner circle
Untick Auto Range
Min Max
0 2

Colourmap ~~Edit~~ Options - Change type ~~from~~ to float
Apply - Close

Click title bar ☰ on right for display type time
- Save / Display

Solution Animation under Calculation Activities
Name - animation1
Record after every 10 time step
Storage type - In Memory
Select contour - 1 from Animation object
Animation view - select - Use Active
OK

Run calculation :

Number of time step to 1000

Time step size to ~~0.001~~ ~~0.01~~ 0.1

Max iteration per time step 20

Save File

File - Write → Case & Data

File of type : Legacy Compressed ~~Data~~ Case / Data files

(*.cas.gz *.dat.gz) (we record option)

File name : flow_over_a_cylinder_ini-cas.h5

- OK

Calculate.

Results - Animations - Playback

- Select animation-1

- Select play button

- Write / Record Format → video file

- Write.

File will be same where we have set the working directory

(Ansys Project - SOLVE - mesh-animation-1)

Results - vectors. - skip → 10

Another type of initialization:
Initial values. Gauge pressure 0
X velocity 1
Y velocity 0

~~Patch~~
Domain - Adapt - Manual
Cell Registers - ~~Region~~ New - Region

In Region register -
Xmin(m) Xmax(m)
0 32

Ymin(m) Ymax(m)
-5 5

Shape - Quad
Name region_0
Save/Display
Close - close

Under Initialisation - Patch
Variable → X velocity
Value 0.25.
Select region_0
Patch - close

~~Vorticity~~

contours – contour_1 → Contours of vorticity magnitude

Calculation Activities – Solution Animations.
Record after every 1 time step.
Storage type in memory
Select contour 1.
Adjust the view by zooming.
Click Use active
Run calculation
Calculate.