SCHOOL OF ENGINEERING
AERO406 Advanced Fluid Mechanics and Aerodynamics

Viscous Flow between Concentric Cylinders: Theoretical and Computational Fluid Dynamic Solutions

Coursework

Michael Parker

December 2024

Dr. Zhihui Li

School of Engineering
University of Liverpool
Brownlow Hill, Liverpool, L69 3GH

Table of Contents

1. Introduction

This report presents a comparison between the analytic and computational solutions to the Couette flow, a viscous flow with two surfaces moving at different velocities. The initial experiment conducted by Maurice Marie Alfred Couette was used to investigate the viscosity of fluids, it was further advanced in a paper by Sir Geoffrey Ingram Taylor which was used to prove the no slip boundary condition was the correct condition for viscous flows. This flow is also an excellent test of a computational fluid solver as it has an exact solution from the Navier stokes allowing for an understanding of accuracy. The aim of this report is to derive the exact analytic solutions and compare them against a computational approach understanding the caveats of such an approach and any deviations from the analytical solution. (Science Direct, 2024)

2. Theory

The following derivations will start with the Navier stokes in cylindrical (Polar) Coordinates taken from the Schaum's Fluid Dynamic textbook (Potter, 2008). Within this section the boundary conditions will be discussed along with the key assumptions for each test case, the final solution will be displayed within this chapter however the full handwritten derivation including all steps can be found within the appendix.

Mass Conservation

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0$$

Momentum Balance in r-direction

$$\rho\left[\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta} + u_z\frac{\partial u_r}{\partial z} - \frac{u_\theta^2}{r}\right] = -\frac{\partial p}{\partial r} + \mu\left[\frac{\partial^2 u_r}{\partial r^2} + \frac{1}{r}\frac{\partial u_r}{\partial r} + \frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2} + \frac{\partial^2 u_r}{\partial z^2} - \frac{u_r}{r^2} - \frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta}\right]$$

Momentum Balance in θ-direction

$$\rho\left[\frac{\partial u_\theta}{\partial t} + u_r\frac{\partial u_\theta}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_\theta}{\partial \theta} + u_z\frac{\partial u_\theta}{\partial z} - \frac{u_r u_\theta}{r}\right] = -\frac{1}{r}\frac{\partial p}{\partial \theta} + \mu\left[\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r}\frac{\partial u_\theta}{\partial r} + \frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial \theta^2} + \frac{\partial^2 u_\theta}{\partial z^2} - \frac{u_\theta}{r^2} - \frac{2}{r^2}\frac{\partial u_r}{\partial \theta}\right]$$

Momentum Balance in z-direction

$$\rho\left[\frac{\partial u_z}{\partial t} + u_r\frac{\partial u_z}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta} + u_z\frac{\partial u_z}{\partial z}\right] = -\frac{\partial_p}{\partial z} + \mu\left[\frac{\partial^2 u_z}{\partial r^2} + \frac{1}{r}\frac{\partial u_z}{\partial r} + \frac{1}{r^2}\frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2}\right] - \rho g_z$$

2.1. Problem Setup

Each of the boundary conditions use the same geometry an inner cylinder with an outer cylinder concentric to it. The inner cylinders radius is $1 \times 10^{-3}m$ whilst the outer cylinder has a radius of $2 \times 10^{-3}m$. The flow is steady incompressible and laminar, with no pressure gradient forming in the axial spatial direction (z), finally there are no body forces acting.

2.2. Boundary Condition 1

For condition one the inner cylinder is rotating at $100\ rad\ s^{-1} = \omega$, whilst the outer cylinder remains stationary.

2.2.1. Assumptions

A. Steady state $\partial t = 0$
B. Symmetric across $\theta\ \partial\theta = 0$
C. No flow in r or z direction $V_r = V_z = 0$
D. Symmetry in z direction $\partial z = 0$
E. Incompressible, Newtonian Fluid, Constant μ , ρ
F. Non Slip Boundary Condition
G. No pressure gradient across z $\frac{\partial p}{\partial z} = 0$
H. No body forces acting on system

2.2.2. Derivation Results

$$V_\theta(r) = -\frac{\omega \times r}{\left(\frac{{R_o}^2}{{R_i}^2} - 1\right)} + \frac{\omega}{r \times \left(\frac{1}{{R_i}^2} - \frac{1}{{R_o}^2}\right)}$$

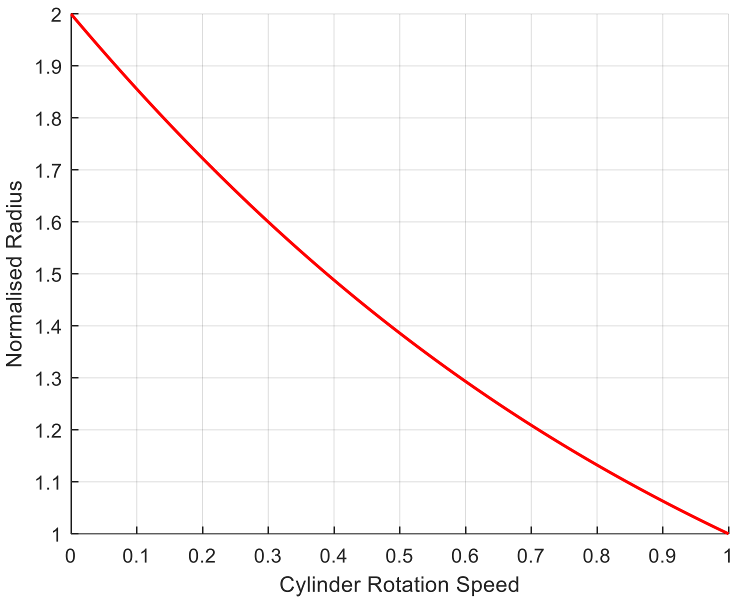


Figure 1 Analytic Results for Boundary Condition 1

2.3. Boundary Condition 2

For condition two the inner cylinder is stationary, whilst the outer cylinder translates along the axial direction(z) at $0.1\,\frac{m}{s} = U_0$.

2.3.1. Assumptions

A. Steady state $\partial t = 0$
B. Symmetric across $\theta\ \partial\theta = 0$
C. No flow in r or θ direction $V_r = V_\theta = 0$
D. Symmetry in z direction $\partial z = 0$
E. Incompressible, Newtonian Fluid, Constant μ , ρ

F. Non Slip Boundary Condition

G. Constant Pressure $\partial_p = 0$

H. No body forces acting on system

2.3.2. Derivation Results

$$V_z(r) = \frac{U}{ln\left(\frac{R_o}{R_i}\right)} \times ln\left(\frac{r}{R_i}\right)$$

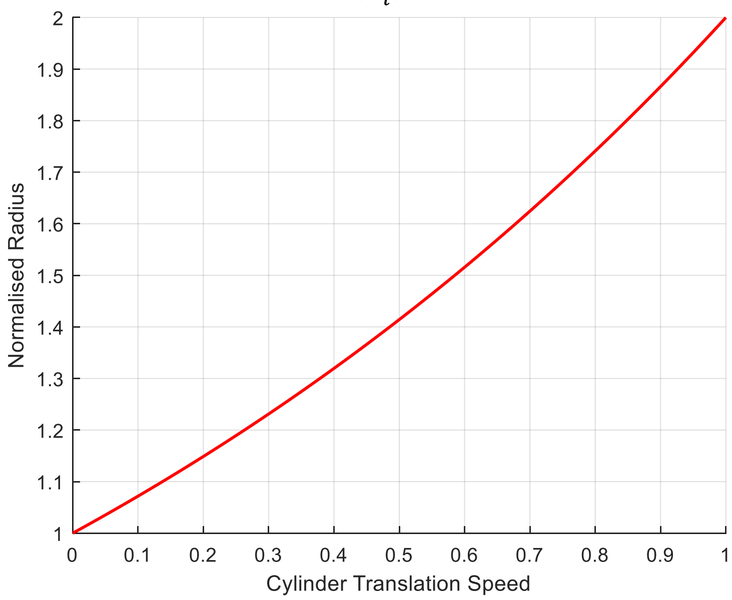


Figure 2 Analytic Results for Boundary Condition 2

2.4. Boundary Condition 3

For condition three the inner cylinder is rotating at 100 $rad\ s^{-1} = \omega$, whilst the outer cylinder translates along the axial direction(z) at 0.1 $\frac{m}{s} = U_0$.

2.4.1. Assumptions

Carried over from the calculation of Boundary Condition 1 and 2.

2.4.2. Derivation Results

$$V_m(r) = \left| \sqrt{\left(-\frac{\omega \times r}{\left(\frac{{R_o}^2}{{R_i}^2} - 1\right)} + \frac{\omega}{r \times \left(\frac{1}{{R_i}^2} - \frac{1}{{R_o}^2}\right)} \right)^2 + \left(\frac{U}{ln\left(\frac{R_o}{R_i}\right)} \times ln\left(\frac{r}{R_i}\right) \right)^2} \right|$$

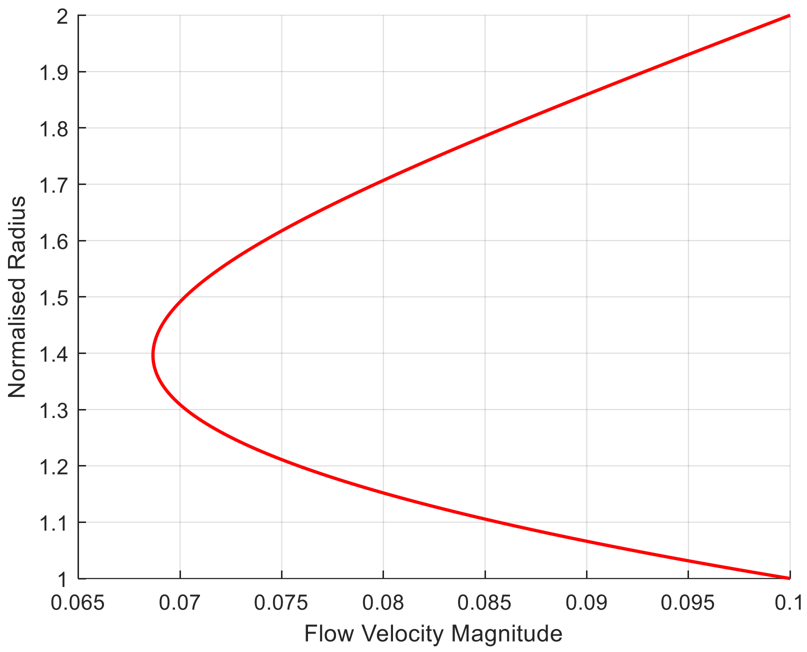


Figure 3 Analytic Results for Boundary Condition 3

3. Simulation

Each of the three boundary conditions use the same mesh file split into 3 sections, Inner Cylinder Outer Cylinder and Farfield. For this experiment there is no flow coming into the body so no inlet was required however it was important to set the farfield as a pressure outlet to prevent the build up of pressure along the axial spatial direction. Both the inner and outer plate were set as heat flux markers to provide the no slip conditions. For the mesh a rectilinear construction was used to increase the mesh density at the surfaces and the density at 0.05m as this was the point the measurements where taken from.

3.1. Mesh Study

A Mesh study was conducted to understand how refined a mesh had to be to collect the most accurate results for the experiment whilst maintaining a quick run time. Three types of mesh where created a coarse mesh, medium mesh and a refined mesh, the number of nodes and elements can be found in Table 1, the measurement of accuracy was a comparison between the average deviation from analytical results of boundary condition 1 and the time taken to run the simulation.

Table 1 Mesh Setup for Mesh Refinement Study

Mesh Type	Number of Elements	Number of Nodes
Coarse	20262	18650
Medium	65452	60800
Fine	157389	156980

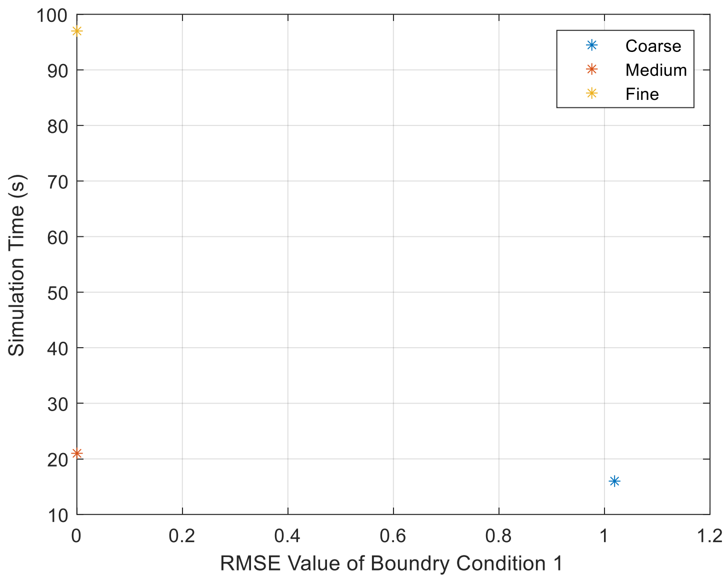


Figure 4 RMSE Analysis of boundary condition one with Three mesh types.

From Figure 4 it was determined that the coarse mesh was unacceptable with an average deviation value above 1, the difference in average deviation between the medium and fine mesh was negligible however the time taken to simulate was over 3 times for the fine mesh. Following this analysis the medium mesh was chosen for the remainder of the experiment.

3.2. CFD Setup

For each of the boundary conditions the main setup was the same with only the specific boundary conditions changing for each analysis. The main configurations can be found below in Table 2.

Table 2 configuration Parameters for CFD Setup

Configuration Parameter	Description
Solver Method	INC_NAVIER_STOKES
Spatial Scheme	WEIGHTED_LEAST_SQUARES
CFL Number	1000
CFL Adapt	0.1, 2.0, 10.0, 1e10, 0.001
Max Iterations	250
Linear Solver	FGMRES
Convective Numerical Method	FDS
Time Discretization	EULER_IMPLICIT
Convergence Criteria	REL_RMS_PRESSURE

3.3. Dynamic Mesh Definitions

Shown below is the specific setup for each test case and how the dynamic aspect of the test cases within the simulation were achieved.

Table 3 Dynamic Mesh Definitions for Boundary Conditions

	Boundary Condition 1	Boundary Condition 2	Boundary Condition 3
SURFACE_MOVEMENT	MOVING_WALL	MOVING_WALL	MOVING_WALL, MOVING_WALL

MARKER_MOVING	INNER_CIRCLE	OUTER_CIRCLE	INNER_CIRCLE, OUTER_CIRCLE
SURFACE_TRANSLATION_RATE	0 0 0	0 0 0.1	0 0 0 0 0 0.1
SURFACE_ROTATION_RATE	0 0 100	0 0 0	0 0 100 0 0 0

4. Results

To compare the accuracy of the simulation against analytical results an RMSE evaluation was conducted measuring the average difference between values predicted by the simulation against the known values, it provides an estimation of how well the model is able to predict the target values, the closer a value is to zero the more accurate the model is.

4.1. Boundary Condition 1

Shown below in Figure 5 is the comparison between the analytical values and the su2 data, as mentioned above in the derivation section the normalised radius is plotted against the cylinder rotation speed. As can be seen from the plot the data overlays each other with an RMSE value of 2.9×10^{-4}. This shows the accuracy of the simulation and how little improvement can be made to the su2 model.

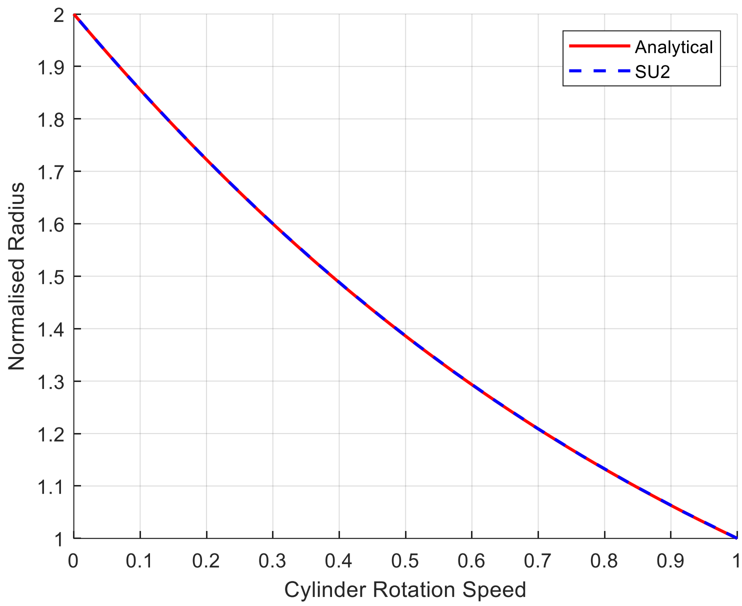


Figure 5 Analytical Vs Simulation results for Boundary Condition 1.

4.2. Boundary Condition 2

The accuracy of the second boundary condition is similar to the first with both lines being plotted there is no visible difference as seen in Figure 6. With the RMSE value for this analysis being 1.8×10^{-4} it again shows that the simulation is an excellent representation of the analytical solution.

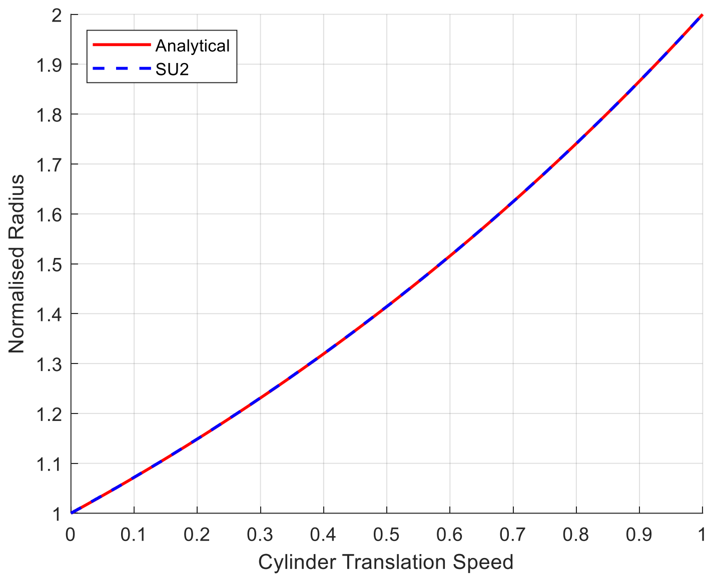


Figure 6 Analytical Vs Simulation results for Boundary Condition 2.

4.3. Boundary Condition 3

This is the first condition where a deviation could be seen between the data when plotted, however the maximum difference between the velocity magnitude data was 3×10^{-4} and the RMSE value was 2×10^{-4}. Even with this deviation the simulation model is an extremely accurate representation of the analytical solutions.

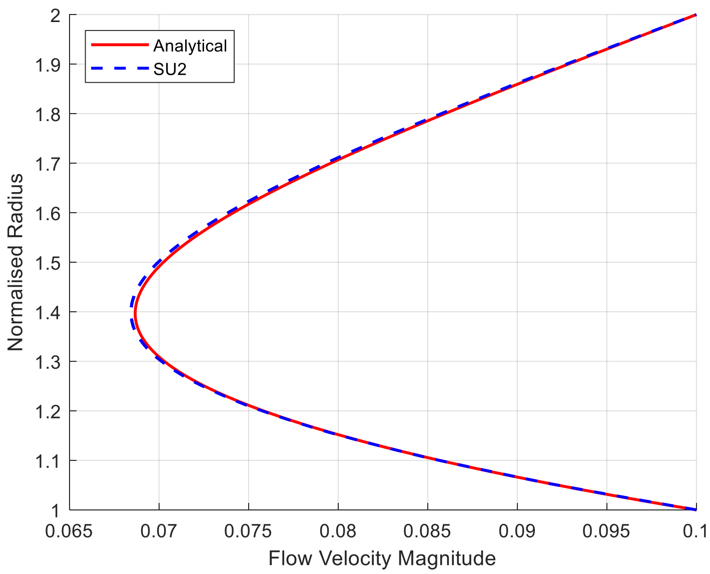


Figure 7 Analytical Vs Simulation results for Boundary Condition 3

5. Discussion

The analytical and computational solutions to the Couette flow are in close agreement with little improvement possible. Through further mesh refinement at the target area the small discrepancies may be removed. Further steps this evaluation could take could be to look at the shear stress acting through the fluid to understand how the different boundary conditions affect the stresses. Secondly the experiment could move away from the ideal conditions set and move towards more realistic conditions looking to understand the differences.

References

Li, Z., 2024. *Viscous Flow Coursework.* [Online]
Available at: www.canvas.liverpool.ac.uk/courses/76853/assignments/285846
[Accessed 16 November 2024].

Potter, M., 2008. *Fluid Mechanics.* 3rd ed. Michigan: McGraw.

Science Direct, 2024. *Couette Flow.* [Online]
Available at: https://www.sciencedirect.com/topics/physics-and-astronomy/couette-flow#:~:text=The%20Couette%20flow%20consists%20of,planes%20with%20different%20linear%20velocities.
[Accessed 20 November 2024].

Appendix

Boundary Condition 1 Derivation

Boundary condition 1 derivation

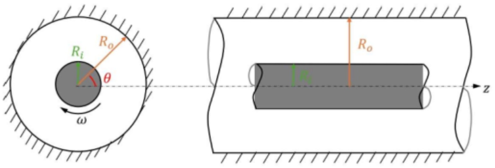


Fig. 1 Boundary condition I for concentric cylinder flow

Assumptions

a) steady state $dt = 0$
b) symmetric across θ $d\theta = 0$
c) no flow in r or z direction $V_r = V_z = 0$
d) symmetry in z direction $dz = 0$
e) incompressible, newtonian fluid constant μ and ρ
f) non slip boundary condition
g) no pressure gradient across z $\frac{dP}{dz} = 0$,
h) no body forces act on system

Boundary condition

$V_\theta = 0$ at $r = R_o$
$V_\theta = \omega r$ at $r = R_i$

Mass conservation

$$\underbrace{\frac{1}{r}\frac{\cancelto{0}{\partial(r v_r)}}{\partial r}}_{c,\ V_r = 0} + \underbrace{\frac{1}{r}\frac{\cancelto{0}{\partial v_\theta}}{\partial \theta}}_{b,\ d\theta = 0} + \underbrace{\frac{\cancelto{0}{\partial v_z}}{\partial z}}_{c,\ V_z = 0} = 0$$

$\therefore = 0$

Momentum balance in r

$$\underbrace{\rho\left[\cancelto{0}{\frac{\partial v_r}{\partial t}} + \cancelto{0}{v_r\frac{\partial v_r}{\partial r}} + \cancelto{0}{\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}} + \cancelto{0}{v_z\frac{\partial v_r}{\partial z}} - \frac{v_\theta^2}{r}\right]}_{c,\ V_r = 0} = -\frac{\partial p}{\partial r} + \mu\left[\underbrace{\cancelto{0}{\frac{\partial^2 v_r}{\partial r^2}} + \cancelto{0}{\frac{1}{r}\frac{\partial v_r}{\partial r}} + \cancelto{0}{\frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2}} + \cancelto{0}{\frac{\partial^2 v_r}{\partial z^2}} - \cancelto{0}{\frac{v_r}{r^2}}}_{c,\ V_r = 0} - \underbrace{\cancelto{0}{\frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta}}}_{b,\ d\theta = 0}\right]$$

$$\therefore \quad -\rho \frac{v_\theta^2}{r} = -\frac{\partial p}{\partial r}$$

momentum balance in θ

$$\rho\left[\cancelto{0}{\frac{\partial v_\theta}{\partial t}} + \underbrace{v_r \cancelto{0}{\frac{\partial v_\theta}{\partial r}} + \frac{v_\theta}{r}\cancelto{0}{\frac{\partial v_\theta}{\partial \theta}} + v_z \cancelto{0}{\frac{\partial v_\theta}{\partial z}} - \cancelto{0}{\frac{v_r v_\theta}{r}}}_{c,\ V_r = V_z = 0}\right] = -\frac{1}{r}\cancelto{0}{\frac{\partial p}{\partial \theta}} + \mu\left[\frac{\partial^2 v_\theta}{\partial r^2} + \frac{1}{r}\frac{\partial v_\theta}{\partial r} + \frac{1}{r^2}\cancelto{0}{\frac{\partial^2 v_\theta}{\partial \theta^2}} + \cancelto{0}{\frac{\partial^2 v_\theta}{\partial z^2}} - \frac{v_\theta}{r^2} - \frac{2}{r^2}\cancelto{0}{\frac{\partial v_r}{\partial \theta}}\right]$$

a, $\partial t = 0$ b, $\partial\theta = 0$ b, $\partial\theta = 0$ b, $\partial\theta = 0$ d, $\partial z = 0$ c, $V_r = 0$

$$0 = \mu\left(\frac{\partial^2 V_\theta}{\partial r^2} + \frac{1}{r}\frac{\partial V_\theta}{\partial r} - \frac{V_\theta}{r^2}\right)$$

$$0 = \frac{\partial^2 V_\theta}{\partial r^2} + \frac{1}{r}\frac{\partial V_\theta}{\partial r} - \frac{V_\theta}{r^2}$$

$$0 = \frac{d^2 V_\theta}{d r^2} + \frac{1}{r}\frac{d V_\theta}{d r} - \frac{V_\theta}{r^2}$$

momentum balance in z

$$\rho\left[\cancelto{0}{\frac{\partial v_z}{\partial t}} + v_r \cancelto{0}{\frac{\partial v_z}{\partial r}} + \frac{v_\theta}{r}\cancelto{0}{\frac{\partial v_z}{\partial \theta}} + v_z \cancelto{0}{\frac{\partial v_z}{\partial z}}\right] = -\cancelto{0}{\frac{\partial p}{\partial z}} + \mu\left[\cancelto{0}{\frac{\partial^2 v_z}{\partial r^2}} + \cancelto{0}{\frac{1}{r}\frac{\partial v_z}{\partial r}} + \cancelto{0}{\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}} + \cancelto{0}{\frac{\partial^2 v_z}{\partial z^2}}\right] - \cancelto{0}{\rho g_z}$$

c, $V_z = 0$ g, $\frac{\partial p}{\partial z} = 0$ c, $V_z = 0$ h, $g_z = 0$

$$\therefore = 0$$

momentum balance in θ

$$0 = \frac{d^2 V_\theta}{dr^2} + \frac{1}{r}\frac{dV_\theta}{dr} - \frac{V_\theta}{r^2}$$

Euler cauchy form

$$0 = m^2 + (a-1) + b$$

where $a=1$, $b=-1$

$$0 = m^2 + (1-1) + -1$$

$$0 = m^2 - 1$$

$m^2 = 1 \therefore m = \pm 1$

$\therefore$ Solution form

$$V_\theta = C_1 r^1 + C_2 r^{-1}$$

use boundry conditions

$V_\theta = 0$ at $r = R_o$

$$0 = C_1 R_o + \frac{C_2}{R_o}$$

$$C_1 = -\frac{C_2}{R_o^2}$$

Plug C_1 in

$V_\theta = \omega r$ at $r = R_i$

$$\omega_{R_i} = C_1 R_i + \frac{C_2}{R_i}$$

$$\omega R_i = \frac{-C_2 R_i}{R_o^2} + \frac{C_2}{R_i}$$

$$\omega R_i = C_2\left(\frac{-R_i}{R_o^2} + \frac{1}{R_i}\right)$$

$$C_2 = \frac{\omega R_i}{\left(\frac{1}{R_i} - \frac{R_i}{R_o^2}\right)}$$

Plug C_1 and C_2 back into

$$V_\theta = C_1 r + \frac{C_2}{r}$$

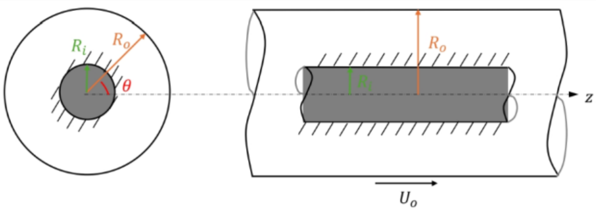
$$V_\theta = \frac{-C_2 . r}{R_o^2} + \frac{\omega R_i}{\left(\frac{1}{R_i} - \frac{R_i}{R_o^2}\right) r}$$

$$V_\theta = -\frac{\left(\frac{\omega R_i}{\frac{1}{R_i} - \frac{R_i}{R_o^2}}\right) r}{R_o^2} + \frac{\omega R_i}{\left(\frac{1}{R_i} - \frac{R_i}{R_o^2}\right) r}$$

$$\boxed{V_\theta(r) = -\frac{\omega . r}{\left(\frac{R_o^2}{R_i^2} - 1\right)} + \frac{\omega}{r\left(\frac{1}{R_i^2} - \frac{1}{R_o^2}\right)}}$$

Boundary Condition 2 Derivation

Boundary condition 2 Derivation



Assumptions

a) steady state $dt = 0$
b) Symmetric across θ $d\theta = 0$
c) no flow in r or θ direction $V_r = V_\theta = 0$
d) Symmetry in z direction $dz = 0$
e) incompressible, newtonian fluid constant μ and ρ
f) non slip boundary condition
g) constant pressure, $dP = 0$
h) no body forces act on system

Boundary condition

$V_z = 0$ at $r = R_i$
$V_z = U$ at $r = R_o$

Mass conservation

$$\underbrace{\frac{1}{r}\cancel{\frac{\partial(rV_r)}{\partial r}}^{\to 0}}_{c,\ V_r = 0} + \underbrace{\frac{1}{r}\cancel{\frac{\partial V_\theta}{\partial \theta}}^{\to 0}}_{c,\ V_\theta = 0} + \underbrace{\cancel{\frac{\partial V_z}{\partial z}}^{\to 0}}_{d,\ V_z = 0} = 0$$

$\therefore = 0$

Momentum balance in r

$$\rho\underbrace{\left[\cancel{\frac{\partial v_r}{\partial t}}^{\to 0} + \cancel{v_r\frac{\partial v_r}{\partial r}}^{\to 0} + \cancel{\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}}^{\to 0} + \cancel{v_z\frac{\partial v_r}{\partial z}}^{\to 0} - \cancel{\frac{v_\theta^2}{r}}^{\to 0}\right]}_{c,\ V_r = V_\theta = 0} = \underbrace{-\cancel{\frac{\partial p}{\partial r}}^{\to 0}}_{g,\ dP = 0} + \mu\underbrace{\left[\cancel{\frac{\partial^2 v_r}{\partial r^2}}^{\to 0} + \cancel{\frac{1}{r}\frac{\partial v_r}{\partial r}}^{\to 0} + \cancel{\frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2}}^{\to 0} + \cancel{\frac{\partial^2 v_r}{\partial z^2}}^{\to 0} - \cancel{\frac{v_r}{r^2}}^{\to 0} - \cancel{\frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta}}^{\to 0}\right]}_{c,\ V_r = V_\theta = 0}$$

$\therefore = 0$

Momentum balance in θ

$$\rho\underbrace{\left[\cancel{\frac{\partial v_\theta}{\partial t}}^{\to 0} + \cancel{v_r\frac{\partial v_\theta}{\partial r}}^{\to 0} + \cancel{\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}}^{\to 0} + \cancel{v_z\frac{\partial v_\theta}{\partial z}}^{\to 0} - \cancel{\frac{v_r v_\theta}{r}}^{\to 0}\right]}_{c,\ V_r = V_\theta = 0} = \underbrace{-\cancel{\frac{1}{r}\frac{\partial p}{\partial \theta}}^{\to 0}}_{g,\ dP = 0} + \mu\underbrace{\left[\cancel{\frac{\partial^2 v_\theta}{\partial r^2}}^{\to 0} + \cancel{\frac{1}{r}\frac{\partial v_\theta}{\partial r}}^{\to 0} + \cancel{\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}}^{\to 0} + \cancel{\frac{\partial^2 v_\theta}{\partial z^2}}^{\to 0} - \cancel{\frac{v_\theta}{r^2}}^{\to 0} - \cancel{\frac{2}{r^2}\frac{\partial v_r}{\partial \theta}}^{\to 0}\right]}_{c,\ V_r = V_\theta = 0}$$

$\therefore = 0$

E

momentum balance in Z

$$\rho\left[\underset{a,\ dt=0}{\cancelto{0}{\frac{\partial v_z}{\partial t}}} + \underbrace{v_r \cancelto{0}{\frac{\partial v_z}{\partial r}} + \frac{v_\theta}{r}\cancelto{0}{\frac{\partial v_z}{\partial \theta}}}_{c,\ v_r = v_\theta = 0} + v_z \underset{d,\ dz=0}{\cancelto{0}{\frac{\partial v_z}{\partial z}}}\right] = -\underset{g,\ dP=0}{\cancelto{0}{\frac{\partial p}{\partial z}}} + \mu\left[\frac{\partial^2 v_z}{\partial r^2} + \frac{1}{r}\frac{\partial v_z}{\partial r} + \frac{1}{r^2}\underset{b,\ d\theta=0}{\cancelto{0}{\frac{\partial^2 v_z}{\partial \theta^2}}} + \underset{d,\ dz=0}{\cancelto{0}{\frac{\partial^2 v_z}{\partial z^2}}}\right] - \underset{h,\ g=0}{\cancelto{0}{\rho g_z}}$$

$$\therefore \mu\left(\frac{\partial^2 v_z}{\partial r^2} + \frac{1}{r}\frac{\partial v_z}{\partial r}\right) = 0$$

$$= \mu \frac{1}{r}\frac{d}{dr}\left(r\frac{dv_z}{dr}\right) = 0$$

$$\frac{d}{dr}\left(r\frac{dv_z}{dr}\right) = 0$$

$$\int^{r} \frac{d}{dr}\left(r\frac{dv_z}{dr}\right)$$

$$r\frac{dV_z}{dr} = C_1$$

$$\frac{dV_z}{dr} = \frac{C_1}{r}$$

$$\int^r \frac{dV_z}{dr} = \frac{C_1}{r}$$

$$\therefore V_z(r) = C_1 \ln(r) + C_2$$

apply boundary conditions

at $r = R_i$, $V_z = 0$

$$0 = C_1 \ln(R_i) + C_2$$

$$C_2 = -C_1 \ln(R_i)$$

at $r = R_o$, $V_z = U$

$$U = C_1 \ln(R_o) + C_2$$

$$= C_1 \ln(R_o) - C_1 \ln(R_i)$$

$$U = C_1 \ln\left(\frac{R_o}{R_i}\right)$$

$$C_1 = \frac{U}{\ln\left(\frac{R_o}{R_i}\right)}$$

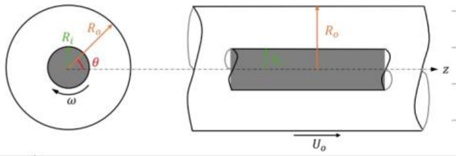
Substitute C_1 back into $V_z(r) = C_1 \ln(r) + C_2$

$$\boxed{V_z(r) = \frac{U}{\ln\left(\frac{R_o}{R_i}\right)} \ln\left(\frac{r}{R_i}\right)}$$

Boundary Condition 3 Derivation

Boundary condition 3 derivation

Assumptions carried over from BC_1 and BC_2



Velocity magnitude (V_m)

$$V_m(r) = \left|\sqrt{V_\theta^2 + V_z^2}\right|$$

$$V_\theta(r) = -\frac{\omega \cdot r}{\left(\frac{R_o^2}{R_i^2} - 1\right)} + \frac{\omega}{r\left(\frac{1}{R_i^2} - \frac{1}{R_o^2}\right)}$$ from BC1 derivation

$$V_z(r) = \frac{U}{\ln\left(\frac{R_o}{R_i}\right)} \ln\left(\frac{r}{R_i}\right)$$ from BC2 derivation

$$V_m(r) = \left|\sqrt{\left(-\frac{\omega \cdot r}{\left(\frac{R_o^2}{R_i^2} - 1\right)} + \frac{\omega}{r\left(\frac{1}{R_i^2} - \frac{1}{R_o^2}\right)}\right)^2 + \left(\frac{U}{\ln\left(\frac{R_o}{R_i}\right)} \ln\left(\frac{r}{R_i}\right)\right)^2}\right|$$

Boundary Condition 1 Configuration File

% ------------- DIRECT, ADJOINT, AND LINEARIZED PROBLEM DEFINITION ------------%

%

% Solver type (EULER, NAVIER_STOKES, RANS,

%   INC_EULER, INC_NAVIER_STOKES, INC_RANS,

```
%            NEMO_EULER, NEMO_NAVIER_STOKES,
%            FEM_EULER, FEM_NAVIER_STOKES, FEM_RANS, FEM_LES,
%            HEAT_EQUATION_FVM, ELASTICITY)
SOLVER= INC_NAVIER_STOKES
%
RESTART_SOL= NO
%
SYSTEM_MEASUREMENTS= SI
%
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%% BLOCK 1 - PHYSICAL MODEL
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%
% ---------------- INCOMPRESSIBLE FLOW CONDITION DEFINITION ------------------%
%
% Density model within the incompressible flow solver.
% Options are CONSTANT (default), BOUSSINESQ, or VARIABLE. If VARIABLE,
% an appropriate fluid model must be selected.
INC_DENSITY_MODEL= CONSTANT
%
% Solve the energy equation in the incompressible flow solver
INC_ENERGY_EQUATION = NO
%
% Initial density for incompressible flows
% (1.2886 kg/m^3 by default (air), 998.2 Kg/m^3 (water))
INC_DENSITY_INIT= 1.0000
%
```

% Initial velocity for incompressible flows (1.0,0,0 m/s by default)

INC_VELOCITY_INIT= (0.0, 0.0, 0.0)

%

% Non-dimensionalization scheme for incompressible flows. Options are

% INITIAL_VALUES (default), REFERENCE_VALUES, or DIMENSIONAL.

% INC_*_REF values are ignored unless REFERENCE_VALUES is chosen.

INC_NONDIM= DIMENSIONAL

%

% List of inlet types for incompressible flows. List length must

% match number of inlet markers. Options: VELOCITY_INLET, PRESSURE_INLET.

%INC_INLET_TYPE= NONE

%

% Fluid model (STANDARD_AIR, IDEAL_GAS, VW_GAS, PR_GAS,

%   CONSTANT_DENSITY, INC_IDEAL_GAS, INC_IDEAL_GAS_POLY, MUTATIONPP, SU2_NONEQ, FLUID_MIXTURE, COOLPROP)

FLUID_MODEL= INC_IDEAL_GAS

%

% Viscosity model (SUTHERLAND, CONSTANT_VISCOSITY, POLYNOMIAL_VISCOSITY).

VISCOSITY_MODEL=CONSTANT_VISCOSITY

%

MU_CONSTANT= 1.716E-5

%

%~~~~~~%

%% BLOCK 2 - SPATIAL DISCRETISATION

%~~~~~~%

%

% ----------------------- FLUX DISCRETISATIONN ---------------------------%

% Convective numerical method (JST, JST_KE, JST_MAT, LAX-FRIEDRICH, CUSP, ROE, AUSM,

```
%                        AUSMPLUSUP, AUSMPLUSUP2, AUSMPLUSM, HLLC, TURKEL_PREC,
%                        SW, MSW, FDS, SLAU, SLAU2, L2ROE, LMROE)
CONV_NUM_METHOD_FLOW= FDS
%
% Monotonic Upwind Scheme for Conservation Laws (TVD) in the flow equations.
%           Required for 2nd order upwind schemes (NO, YES)
MUSCL_FLOW= YES
%
% Slope limiter (NONE, VENKATAKRISHNAN, VENKATAKRISHNAN_WANG,
%               BARTH_JESPERSEN, VAN_ALBADA_EDGE)
SLOPE_LIMITER_FLOW= NONE
%
% Numerical method for spatial gradients (GREEN_GAUSS, WEIGHTED_LEAST_SQUARES)
NUM_METHOD_GRAD= WEIGHTED_LEAST_SQUARES
%
% -------------------- STREAMWISE PERIODICITY DEFINITION --------------------%
%
% Generally for streamwise periodictiy one has to set MARKER_PERIODIC= (<inlet>, <outlet>, ...)
% appropriately as a boundary condition.
%
% Specify type of streamwise periodictiy (default=NONE, PRESSURE_DROP, MASSFLOW)
KIND_STREAMWISE_PERIODIC= NONE
%
% Delta P [Pa] value that drives the flow as a source term in the momentum equations.
% Defaults to 1.0.
STREAMWISE_PERIODIC_PRESSURE_DROP= 0
%
```

```
% ---------------------- DYNAMIC MESH DEFINITION -----------------------------%
%
SURFACE_MOVEMENT= MOVING_WALL
MARKER_MOVING= ( INNER_CIRCLE )
SURFACE_ROTATION_RATE= 0 0 100
%
% -------------------- BOUNDARY CONDITION DEFINITION --------------------------%
%
% Navier-Stokes (no-slip), constant heat flux wall  marker(s) (NONE = no marker)
% Format: ( marker name, constant heat flux (J/m^2), ... )
MARKER_HEATFLUX= ( INNER_CIRCLE, 0.0, OUTER_CIRCLE, 0.0 )
%
% Periodic boundary marker(s) (NONE = no marker)
% Format: ( periodic marker, donor marker, rotation_center_x, rotation_center_y,
% rotation_center_z, rotation_angle_x-axis, rotation_angle_y-axis,
% rotation_angle_z-axis, translation_x, translation_y, translation_z, ... )
MARKER_PERIODIC= NONE
INC_OUTLET_TYPE= PRESSURE_OUTLET
MARKER_OUTLET = (FARFIELD, 0)
%
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%% BLOCK 3 - TIME-STEPPER AND SOLUTION METHODS
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%
% -------------------- TIME-STEPPER & LINEAR SOLVER DEFINITION -----------------%
%
% Time discretization (RUNGE-KUTTA_EXPLICIT, EULER_IMPLICIT, EULER_EXPLICIT)
```

TIME_DISCRE_FLOW= EULER_IMPLICIT

%

% Linear solver or smoother for implicit formulations:

% BCGSTAB, FGMRES, RESTARTED_FGMRES, CONJUGATE_GRADIENT (self-adjoint problems only), SMOOTHER.

LINEAR_SOLVER= FGMRES

%

% Preconditioner of the Krylov linear solver or type of smoother (ILU, LU_SGS, LINELET, JACOBI)

LINEAR_SOLVER_PREC= ILU

%

% Linear solver ILU preconditioner fill-in level (0 by default)

LINEAR_SOLVER_ILU_FILL_IN= 0

%

% Minimum error of the linear solver for implicit formulations

LINEAR_SOLVER_ERROR= 1E-3

%

% Max number of iterations of the linear solver for the implicit formulation

LINEAR_SOLVER_ITER= 20

%

% Restart frequency for RESTARTED_FGMRES

LINEAR_SOLVER_RESTART_FREQUENCY= 10

%

% Relaxation factor for smoother-type solvers (LINEAR_SOLVER= SMOOTHER)

LINEAR_SOLVER_SMOOTHER_RELAXATION= 1.0

%

% ------------------------------- SOLVER CONTROL ------------------------------%

%

% Maximum number of inner iterations

```
INNER_ITER= 250

%

% Convergence field

CONV_FIELD= REL_RMS_PRESSURE

%

% Min value of the residual (log10 of the residual)

CONV_RESIDUAL_MINVAL= -10

%

% ------------- COMMON PARAMETERS DEFINING THE NUMERICAL METHOD ---------------%

%

% CFL number (initial value for the adaptive CFL number)

CFL_NUMBER= 1000.0

%

% Adaptive CFL number (NO, YES)

CFL_ADAPT= YES

%

% Parameters of the adaptive CFL number (factor-down, factor-up, CFL min value,

%                                        CFL max value, acceptable linear solver convergence)

% Local CFL increases by factor-up until max if the solution rate of change is not limited,

% and acceptable linear convergence is achieved. It is reduced if rate is limited, or if there

% is not enough linear convergence, or if the nonlinear residuals are stagnant and oscillatory.

% It is reset back to min when linear solvers diverge, or if nonlinear residuals increase too much.

CFL_ADAPT_PARAM= ( 0.1, 2.0, 10.0, 1e10, 0.001 )

%

%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%

%% BLOCK 4 - MONITORING AND OUTPUT DEFINITION

%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
```

```
%

% ----------------------- SURFACES IDENTIFICATION ---------------------------%

%

% Marker(s) of the surface in the surface flow solution file

MARKER_PLOTTING = ( INNER_CIRCLE )

%

% Marker(s) of the surface where the non-dimensional coefficients are evaluated.

MARKER_MONITORING = ( INNER_CIRCLE )

%

% ------------------------ SCREEN/HISTORY VOLUME OUTPUT ------------------------%

% Output the performance summary to the console at the end of SU2_CFD

WRT_PERFORMANCE= YES

%

% Screen output fields (use 'SU2_CFD -d <config_file>' to view list of available fields)

SCREEN_OUTPUT= (INNER_ITER, RMS_PRESSURE, RMS_VELOCITY-X, RMS_VELOCITY-Y,
REL_RMS_PRESSURE, REL_RMS_VELOCITY-X, REL_RMS_VELOCITY-Y)

% Writing frequency for screen output

SCREEN_WRT_FREQ_INNER= 1

%

% History output groups (use 'SU2_CFD -d <config_file>' to view list of available fields)

HISTORY_OUTPUT= (INNER_ITER, RMS_RES, REL_RMS_RES)

% Writing frequency for history output

HISTORY_WRT_FREQ_INNER= 1

%

% Volume output fields/groups (use 'SU2_CFD -d <config_file>' to view list of available fields)

VOLUME_OUTPUT= (SOLUTION, PRIMITIVE)

%%

%
```

% ------------------------ INPUT/OUTPUT FILE INFORMATION --------------------------%

%

% Mesh input file

MESH_FILENAME=CylinderMesh.su2

%

% Mesh input file format (SU2, CGNS)

MESH_FORMAT= SU2

%

% Output file for restarting flow

RESTART_FILENAME= restart_flowCouette.dat

%

% Input file for restarting flow when RESTART_SOL= YES

SOLUTION_FILENAME= solution_flowCouette.dat

%

% Output tabular file format (TECPLOT, CSV) for convergence history - choose CSV for ParaView

TABULAR_FORMAT= CSV

% Output file convergence history (w/o extension)

CONV_FILENAME= historyCouette

%

% Files to output

% Possible formats : (TECPLOT_ASCII, TECPLOT, SURFACE_TECPLOT_ASCII,

% SURFACE_TECPLOT, CSV, SURFACE_CSV, PARAVIEW_ASCII, PARAVIEW_LEGACY, SURFACE_PARAVIEW_ASCII,

% SURFACE_PARAVIEW_LEGACY, PARAVIEW, SURFACE_PARAVIEW, RESTART_ASCII, RESTART, CGNS, SURFACE_CGNS, STL_ASCII, STL_BINARY)

% default : (RESTART, PARAVIEW, SURFACE_PARAVIEW)

OUTPUT_FILES= (RESTART, PARAVIEW, SURFACE_PARAVIEW)

% list of writing frequencies corresponding to the list in OUTPUT_FILES

OUTPUT_WRT_FREQ= (250, 250, 250)

%

% Output file flow variables (w/o extension)

VOLUME_FILENAME= flowCouette

%

% Output file surface flow coefficient (w/o extension)

SURFACE_FILENAME= surface_flowCouette

%

Boundary Condition 2 Configuration File

% ------------- DIRECT, ADJOINT, AND LINEARIZED PROBLEM DEFINITION ------------%

%

% Solver type (EULER, NAVIER_STOKES, RANS,

%   INC_EULER, INC_NAVIER_STOKES, INC_RANS,

%   NEMO_EULER, NEMO_NAVIER_STOKES,

%   FEM_EULER, FEM_NAVIER_STOKES, FEM_RANS, FEM_LES,

%   HEAT_EQUATION_FVM, ELASTICITY)

SOLVER= INC_NAVIER_STOKES

%

RESTART_SOL= NO

%

SYSTEM_MEASUREMENTS= SI

%

%~~~~~~%

%% BLOCK 1 - PHYSICAL MODEL

%~~~~~~%

%

% --------------- INCOMPRESSIBLE FLOW CONDITION DEFINITION -----------------%

%

% Density model within the incompressible flow solver.

% Options are CONSTANT (default), BOUSSINESQ, or VARIABLE. If VARIABLE,

% an appropriate fluid model must be selected.

INC_DENSITY_MODEL= CONSTANT

%

% Solve the energy equation in the incompressible flow solver

INC_ENERGY_EQUATION = NO

%

% Initial density for incompressible flows

% (1.2886 kg/m^3 by default (air), 998.2 Kg/m^3 (water))

INC_DENSITY_INIT= 1.2886

%

% Initial velocity for incompressible flows (1.0,0,0 m/s by default)

INC_VELOCITY_INIT= (0.0, 0.0, 0.0)

%

% Non-dimensionalization scheme for incompressible flows. Options are

% INITIAL_VALUES (default), REFERENCE_VALUES, or DIMENSIONAL.

% INC_*_REF values are ignored unless REFERENCE_VALUES is chosen.

INC_NONDIM= DIMENSIONAL

%

% List of inlet types for incompressible flows. List length must

% match number of inlet markers. Options: VELOCITY_INLET, PRESSURE_INLET.

%INC_INLET_TYPE= NONE

%

% Fluid model (STANDARD_AIR, IDEAL_GAS, VW_GAS, PR_GAS,

% CONSTANT_DENSITY, INC_IDEAL_GAS, INC_IDEAL_GAS_POLY, MUTATIONPP, SU2_NONEQ, FLUID_MIXTURE, COOLPROP)

```
FLUID_MODEL= CONSTANT_DENSITY
%
% Viscosity model (SUTHERLAND, CONSTANT_VISCOSITY, POLYNOMIAL_VISCOSITY).
VISCOSITY_MODEL=CONSTANT_VISCOSITY
%
MU_CONSTANT= 1.716E-5
%
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%% BLOCK 2 - SPATIAL DISCRETISATION
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%
% ------------------------ FLUX DISCRETISATIONN ---------------------------%
% Convective numerical method (JST, JST_KE, JST_MAT, LAX-FRIEDRICH, CUSP, ROE, AUSM,
%                         AUSMPLUSUP, AUSMPLUSUP2, AUSMPLUSM, HLLC, TURKEL_PREC,
%                         SW, MSW, FDS, SLAU, SLAU2, L2ROE, LMROE)
CONV_NUM_METHOD_FLOW= FDS
%
% Monotonic Upwind Scheme for Conservation Laws (TVD) in the flow equations.
%         Required for 2nd order upwind schemes (NO, YES)
MUSCL_FLOW= YES
%
% Slope limiter (NONE, VENKATAKRISHNAN, VENKATAKRISHNAN_WANG,
%             BARTH_JESPERSEN, VAN_ALBADA_EDGE)
SLOPE_LIMITER_FLOW= NONE
%
% Numerical method for spatial gradients (GREEN_GAUSS, WEIGHTED_LEAST_SQUARES)
```

```
NUM_METHOD_GRAD= WEIGHTED_LEAST_SQUARES
%
% --------------------- STREAMWISE PERIODICITY DEFINITION ---------------------%
%
% Generally for streamwise periodictiy one has to set MARKER_PERIODIC= (<inlet>, <outlet>, ...)
% appropriately as a boundary condition.
%
% Specify type of streamwise periodictiy (default=NONE, PRESSURE_DROP, MASSFLOW)
KIND_STREAMWISE_PERIODIC= NONE
%
% Delta P [Pa] value that drives the flow as a source term in the momentum equations.
% Defaults to 1.0.
STREAMWISE_PERIODIC_PRESSURE_DROP= 0
% ---------------------- DYNAMIC MESH DEFINITION ----------------------------%
%
SURFACE_MOVEMENT= MOVING_WALL
MARKER_MOVING= ( OUTER_CIRCLE )
SURFACE_TRANSLATION_RATE= 0.0 0.0 0.1
%
% -------------------- BOUNDARY CONDITION DEFINITION -------------------------%
%
% Navier-Stokes (no-slip), constant heat flux wall  marker(s) (NONE = no marker)
% Format: ( marker name, constant heat flux (J/m^2), ... )
MARKER_HEATFLUX= ( INNER_CIRCLE, 0.0, OUTER_CIRCLE, 0.0 )
%
% Periodic boundary marker(s) (NONE = no marker)
% Format: ( periodic marker, donor marker, rotation_center_x, rotation_center_y,
```

% rotation_center_z, rotation_angle_x-axis, rotation_angle_y-axis,

% rotation_angle_z-axis, translation_x, translation_y, translation_z, ...)

MARKER_PERIODIC= NONE

INC_OUTLET_TYPE = PRESSURE_OUTLET

MARKER_OUTLET = (FARFIELD)

%

%~~~~~~%

%% BLOCK 3 - TIME-STEPPER AND SOLUTION METHODS

%~~~~~~%

%

% --------------------- TIME-STEPPER & LINEAR SOLVER DEFINITION -----------------%

%

% Time discretization (RUNGE-KUTTA_EXPLICIT, EULER_IMPLICIT, EULER_EXPLICIT)

TIME_DISCRE_FLOW= EULER_IMPLICIT

%

% Linear solver or smoother for implicit formulations:

% BCGSTAB, FGMRES, RESTARTED_FGMRES, CONJUGATE_GRADIENT (self-adjoint problems only), SMOOTHER.

LINEAR_SOLVER= FGMRES

%

% Preconditioner of the Krylov linear solver or type of smoother (ILU, LU_SGS, LINELET, JACOBI)

LINEAR_SOLVER_PREC= ILU

%

% Linear solver ILU preconditioner fill-in level (0 by default)

LINEAR_SOLVER_ILU_FILL_IN= 0

%

% Minimum error of the linear solver for implicit formulations

LINEAR_SOLVER_ERROR= 1E-3

%

% Max number of iterations of the linear solver for the implicit formulation

LINEAR_SOLVER_ITER= 20

%

% Restart frequency for RESTARTED_FGMRES

LINEAR_SOLVER_RESTART_FREQUENCY= 10

%

% Relaxation factor for smoother-type solvers (LINEAR_SOLVER= SMOOTHER)

LINEAR_SOLVER_SMOOTHER_RELAXATION= 1.0

%

% ------------------------------- SOLVER CONTROL ------------------------------%

%

% Maximum number of inner iterations

INNER_ITER= 300

%

% Convergence field

CONV_FIELD= REL_RMS_PRESSURE

%

% Min value of the residual (log10 of the residual)

CONV_RESIDUAL_MINVAL= -10

%

% ------------- COMMON PARAMETERS DEFINING THE NUMERICAL METHOD ---------------%

%

% CFL number (initial value for the adaptive CFL number)

CFL_NUMBER= 1000.0

```
%
% Adaptive CFL number (NO, YES)
CFL_ADAPT= YES
%
% Parameters of the adaptive CFL number (factor-down, factor-up, CFL min value,
%                                        CFL max value, acceptable linear solver convergence)
% Local CFL increases by factor-up until max if the solution rate of change is not limited,
% and acceptable linear convergence is achieved. It is reduced if rate is limited, or if there
% is not enough linear convergence, or if the nonlinear residuals are stagnant and oscillatory.
% It is reset back to min when linear solvers diverge, or if nonlinear residuals increase too much.
CFL_ADAPT_PARAM= ( 0.1, 2.0, 10.0, 1e10, 0.001 )
%
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%% BLOCK 4 - MONITORING AND OUTPUT DEFINITION
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%
% ------------------------ SURFACES IDENTIFICATION ----------------------------%
%
% Marker(s) of the surface in the surface flow solution file
MARKER_PLOTTING = ( OUTER_CIRCLE )
%
% Marker(s) of the surface where the non-dimensional coefficients are evaluated.
MARKER_MONITORING = ( OUTER_CIRCLE )
%
% ------------------------- SCREEN/HISTORY VOLUME OUTPUT --------------------------%
% Output the performance summary to the console at the end of SU2_CFD
WRT_PERFORMANCE= YES
```

%

% Screen output fields (use 'SU2_CFD -d <config_file>' to view list of available fields)

SCREEN_OUTPUT= (INNER_ITER, RMS_PRESSURE, RMS_VELOCITY-X, RMS_VELOCITY-Y, REL_RMS_PRESSURE, REL_RMS_VELOCITY-X, REL_RMS_VELOCITY-Y)

% Writing frequency for screen output

SCREEN_WRT_FREQ_INNER= 1

%

% History output groups (use 'SU2_CFD -d <config_file>' to view list of available fields)

HISTORY_OUTPUT= (INNER_ITER, RMS_RES, REL_RMS_RES)

% Writing frequency for history output

HISTORY_WRT_FREQ_INNER= 1

%

% Volume output fields/groups (use 'SU2_CFD -d <config_file>' to view list of available fields)

VOLUME_OUTPUT= (SOLUTION, PRIMITIVE)

%%

%

% ------------------------ INPUT/OUTPUT FILE INFORMATION --------------------------%

%

% Mesh input file

MESH_FILENAME=CylinderMesh.su2

%

% Mesh input file format (SU2, CGNS)

MESH_FORMAT= SU2

%

% Output file for restarting flow

RESTART_FILENAME= restart_flowCouette.dat

%

% Input file for restarting flow when RESTART_SOL= YES

SOLUTION_FILENAME= solution_flowCouette.dat

%

% Output tabular file format (TECPLOT, CSV) for convergence history - choose CSV for ParaView

TABULAR_FORMAT= CSV

% Output file convergence history (w/o extension)

CONV_FILENAME= historyCouette

%

% Files to output

% Possible formats : (TECPLOT_ASCII, TECPLOT, SURFACE_TECPLOT_ASCII,

% SURFACE_TECPLOT, CSV, SURFACE_CSV, PARAVIEW_ASCII, PARAVIEW_LEGACY, SURFACE_PARAVIEW_ASCII,

% SURFACE_PARAVIEW_LEGACY, PARAVIEW, SURFACE_PARAVIEW, RESTART_ASCII, RESTART, CGNS, SURFACE_CGNS, STL_ASCII, STL_BINARY)

% default : (RESTART, PARAVIEW, SURFACE_PARAVIEW)

OUTPUT_FILES= (RESTART, PARAVIEW, SURFACE_PARAVIEW)

% list of writing frequencies corresponding to the list in OUTPUT_FILES

OUTPUT_WRT_FREQ= (250, 100, 250)

%

% Output file flow variables (w/o extension)

VOLUME_FILENAME= flowCouette

%

% Output file surface flow coefficient (w/o extension)

SURFACE_FILENAME= surface_flowCouette

%

Boundary Condition 3 Configuration File

% ------------- DIRECT, ADJOINT, AND LINEARIZED PROBLEM DEFINITION ------------%

%

% Solver type (EULER, NAVIER_STOKES, RANS,

```
%           INC_EULER, INC_NAVIER_STOKES, INC_RANS,
%           NEMO_EULER, NEMO_NAVIER_STOKES,
%           FEM_EULER, FEM_NAVIER_STOKES, FEM_RANS, FEM_LES,
%           HEAT_EQUATION_FVM, ELASTICITY)
SOLVER= INC_NAVIER_STOKES
%
RESTART_SOL= NO
%
SYSTEM_MEASUREMENTS= SI
%
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%% BLOCK 1 - PHYSICAL MODEL
%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
%
% ---------------- INCOMPRESSIBLE FLOW CONDITION DEFINITION ------------------%
%
% Density model within the incompressible flow solver.
% Options are CONSTANT (default), BOUSSINESQ, or VARIABLE. If VARIABLE,
% an appropriate fluid model must be selected.
INC_DENSITY_MODEL= CONSTANT
%
% Solve the energy equation in the incompressible flow solver
INC_ENERGY_EQUATION = NO
%
% Initial density for incompressible flows
% (1.2886 kg/m^3 by default (air), 998.2 Kg/m^3 (water))
INC_DENSITY_INIT= 1.2886
```

%

% Initial velocity for incompressible flows (1.0,0,0 m/s by default)

INC_VELOCITY_INIT= (0.0, 0.0, 0.0)

%

% Non-dimensionalization scheme for incompressible flows. Options are

% INITIAL_VALUES (default), REFERENCE_VALUES, or DIMENSIONAL.

% INC_*_REF values are ignored unless REFERENCE_VALUES is chosen.

INC_NONDIM= DIMENSIONAL

%

% List of inlet types for incompressible flows. List length must

% match number of inlet markers. Options: VELOCITY_INLET, PRESSURE_INLET.

%INC_INLET_TYPE= NONE

%

% Fluid model (STANDARD_AIR, IDEAL_GAS, VW_GAS, PR_GAS,

% CONSTANT_DENSITY, INC_IDEAL_GAS, INC_IDEAL_GAS_POLY, MUTATIONPP, SU2_NONEQ, FLUID_MIXTURE, COOLPROP)

FLUID_MODEL= CONSTANT_DENSITY

%

% Viscosity model (SUTHERLAND, CONSTANT_VISCOSITY, POLYNOMIAL_VISCOSITY).

VISCOSITY_MODEL=CONSTANT_VISCOSITY

%

MU_CONSTANT= 1.716E-5

%

%~~~~~~%

%% BLOCK 2 - SPATIAL DISCRETISATION

%~~~~~~%

%

```
% ------------------------ FLUX DISCRETISATIONN ---------------------------%
% Convective numerical method (JST, JST_KE, JST_MAT, LAX-FRIEDRICH, CUSP, ROE, AUSM,
%                               AUSMPLUSUP, AUSMPLUSUP2, AUSMPLUSM, HLLC, TURKEL_PREC,
%                               SW, MSW, FDS, SLAU, SLAU2, L2ROE, LMROE)
CONV_NUM_METHOD_FLOW= FDS
%
% Monotonic Upwind Scheme for Conservation Laws (TVD) in the flow equations.
%           Required for 2nd order upwind schemes (NO, YES)
MUSCL_FLOW= YES
%
% Slope limiter (NONE, VENKATAKRISHNAN, VENKATAKRISHNAN_WANG,
%                BARTH_JESPERSEN, VAN_ALBADA_EDGE)
SLOPE_LIMITER_FLOW= NONE
%
% Numerical method for spatial gradients (GREEN_GAUSS, WEIGHTED_LEAST_SQUARES)
NUM_METHOD_GRAD= WEIGHTED_LEAST_SQUARES
%
% --------------------- STREAMWISE PERIODICITY DEFINITION ---------------------%
%
% Generally for streamwise periodictiy one has to set MARKER_PERIODIC= (<inlet>, <outlet>, ...)
% appropriately as a boundary condition.
%
% Specify type of streamwise periodictiy (default=NONE, PRESSURE_DROP, MASSFLOW)
KIND_STREAMWISE_PERIODIC= NONE
%
% Delta P [Pa] value that drives the flow as a source term in the momentum equations.
% Defaults to 1.0.
```

```
STREAMWISE_PERIODIC_PRESSURE_DROP= 0

% ----------------------- DYNAMIC MESH DEFINITION -----------------------------%

%

SURFACE_MOVEMENT= (MOVING_WALL, MOVING_WALL)

MARKER_MOVING= ( INNER_CIRCLE, OUTER_CIRCLE )

SURFACE_TRANSLATION_RATE= 0 0 0 0.0 0.0 0.1

SURFACE_ROTATION_RATE = 0 0 100 0 0 0

%

% -------------------- BOUNDARY CONDITION DEFINITION --------------------------%

%

% Navier-Stokes (no-slip), constant heat flux wall  marker(s) (NONE = no marker)

% Format: ( marker name, constant heat flux (J/m^2), ... )

MARKER_HEATFLUX= ( INNER_CIRCLE, 0.0, OUTER_CIRCLE, 0.0 )

%

% Periodic boundary marker(s) (NONE = no marker)

% Format: ( periodic marker, donor marker, rotation_center_x, rotation_center_y,

% rotation_center_z, rotation_angle_x-axis, rotation_angle_y-axis,

% rotation_angle_z-axis, translation_x, translation_y, translation_z, ... )

MARKER_PERIODIC=  NONE


INC_OUTLET_TYPE = PRESSURE_OUTLET

MARKER_OUTLET = (FARFIELD)


%

%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%

%% BLOCK 3 - TIME-STEPPER AND SOLUTION METHODS

%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%
```

%

% -------------------- TIME-STEPPER & LINEAR SOLVER DEFINITION -----------------%

%

% Time discretization (RUNGE-KUTTA_EXPLICIT, EULER_IMPLICIT, EULER_EXPLICIT)

TIME_DISCRE_FLOW= EULER_IMPLICIT

%

% Linear solver or smoother for implicit formulations:

% BCGSTAB, FGMRES, RESTARTED_FGMRES, CONJUGATE_GRADIENT (self-adjoint problems only), SMOOTHER.

LINEAR_SOLVER= FGMRES

%

% Preconditioner of the Krylov linear solver or type of smoother (ILU, LU_SGS, LINELET, JACOBI)

LINEAR_SOLVER_PREC= ILU

%

% Linear solver ILU preconditioner fill-in level (0 by default)

LINEAR_SOLVER_ILU_FILL_IN= 0

%

% Minimum error of the linear solver for implicit formulations

LINEAR_SOLVER_ERROR= 1E-3

%

% Max number of iterations of the linear solver for the implicit formulation

LINEAR_SOLVER_ITER= 20

%

% Restart frequency for RESTARTED_FGMRES

LINEAR_SOLVER_RESTART_FREQUENCY= 10

%

% Relaxation factor for smoother-type solvers (LINEAR_SOLVER= SMOOTHER)

LINEAR_SOLVER_SMOOTHER_RELAXATION= 1.0

```
%
% ------------------------------ SOLVER CONTROL -----------------------------%
%
% Maximum number of inner iterations
INNER_ITER= 500
%
% Convergence field
CONV_FIELD= REL_RMS_PRESSURE
%
% Min value of the residual (log10 of the residual)
CONV_RESIDUAL_MINVAL= -10
%
% ------------ COMMON PARAMETERS DEFINING THE NUMERICAL METHOD --------------%
%
% CFL number (initial value for the adaptive CFL number)
CFL_NUMBER= 1000.0
%
% Adaptive CFL number (NO, YES)
CFL_ADAPT= YES
%
% Parameters of the adaptive CFL number (factor-down, factor-up, CFL min value,
%                                        CFL max value, acceptable linear solver convergence)
% Local CFL increases by factor-up until max if the solution rate of change is not limited,
% and acceptable linear convergence is achieved. It is reduced if rate is limited, or if there
% is not enough linear convergence, or if the nonlinear residuals are stagnant and oscillatory.
% It is reset back to min when linear solvers diverge, or if nonlinear residuals increase too much.
CFL_ADAPT_PARAM= ( 0.1, 2.0, 10.0, 1e10, 0.001 )
```

```
%

%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%

%% BLOCK 4 - MONITORING AND OUTPUT DEFINITION

%~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~%

%

% ------------------------ SURFACES IDENTIFICATION ----------------------------%

%

% Marker(s) of the surface in the surface flow solution file

MARKER_PLOTTING = ( OUTER_CIRCLE )

%

% Marker(s) of the surface where the non-dimensional coefficients are evaluated.

MARKER_MONITORING = ( OUTER_CIRCLE )

%

% ------------------------- SCREEN/HISTORY VOLUME OUTPUT --------------------------%

% Output the performance summary to the console at the end of SU2_CFD

WRT_PERFORMANCE= YES

%

% Screen output fields (use 'SU2_CFD -d <config_file>' to view list of available fields)

SCREEN_OUTPUT= (INNER_ITER, RMS_PRESSURE, RMS_VELOCITY-X, RMS_VELOCITY-Y,
REL_RMS_PRESSURE, REL_RMS_VELOCITY-X, REL_RMS_VELOCITY-Y)

% Writing frequency for screen output

SCREEN_WRT_FREQ_INNER= 1

%

% History output groups (use 'SU2_CFD -d <config_file>' to view list of available fields)

HISTORY_OUTPUT= (INNER_ITER, RMS_RES, REL_RMS_RES)

% Writing frequency for history output

HISTORY_WRT_FREQ_INNER= 1

%
```

% Volume output fields/groups (use 'SU2_CFD -d <config_file>' to view list of available fields)

VOLUME_OUTPUT= (SOLUTION, PRIMITIVE)

%%

%

% ------------------------ INPUT/OUTPUT FILE INFORMATION --------------------------%

%

% Mesh input file

MESH_FILENAME=CylinderMesh.su2

%

% Mesh input file format (SU2, CGNS)

MESH_FORMAT= SU2

%

% Output file for restarting flow

RESTART_FILENAME= restart_flowCouette.dat

%

% Input file for restarting flow when RESTART_SOL= YES

SOLUTION_FILENAME= solution_flowCouette.dat

%

% Output tabular file format (TECPLOT, CSV) for convergence history - choose CSV for ParaView

TABULAR_FORMAT= CSV

% Output file convergence history (w/o extension)

CONV_FILENAME= historyCouette

%

% Files to output

% Possible formats : (TECPLOT_ASCII, TECPLOT, SURFACE_TECPLOT_ASCII,

% SURFACE_TECPLOT, CSV, SURFACE_CSV, PARAVIEW_ASCII, PARAVIEW_LEGACY, SURFACE_PARAVIEW_ASCII,

% SURFACE_PARAVIEW_LEGACY, PARAVIEW, SURFACE_PARAVIEW, RESTART_ASCII, RESTART, CGNS, SURFACE_CGNS, STL_ASCII, STL_BINARY)

% default : (RESTART, PARAVIEW, SURFACE_PARAVIEW)

OUTPUT_FILES= (RESTART, PARAVIEW, SURFACE_PARAVIEW)

% list of writing frequencies corresponding to the list in OUTPUT_FILES

OUTPUT_WRT_FREQ= (250, 100, 250)

%

% Output file flow variables (w/o extension)

VOLUME_FILENAME= flowCouette

%

% Output file surface flow coefficient (w/o extension)

SURFACE_FILENAME= surface_flowCouette

%