Computational Fluid Dynamics Study of Flow within a Double Gap Cylinder

by

Leroy Wong Tze-Shyuen
15086

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

MAY 2014

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Computational Fluid Dynamics Study of Flow within a Double Gap Cylinder

by

Leroy Wong Tze-Shyuen
15086

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

in partial fulfilment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved by,

____

(Dr. Tuan Muhammad Yusoff Shah b. Tuan Ya)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK.

May 2014

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own expect as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

(LEROY WONG TZE-SHYUEN)

ACKNOWLEDGEMENT

The author would like to acknowledge the opportunity and support given by the Mechanical Engineering Department of Universiti Teknologi PETRONAS, especially in providing the facilities needed for the simulation project for the duration of two semesters. My utmost gratitude to Dr Tuan Yusoff for his supervision in my final year project entitled “Computational Fluid Dynamics Study of Flow within a Double Gap Cylinder”. His assistance and guidance throughout the process of the project, particularly on ANSYS Fluent have indeed helped in the completion of the project. And not forgetting Mr Fakrul Radzi who was constantly helping with the input on the application of double gap cylinder in rheometers. Thank you too to the other students in the mechanical engineering department who have been with me throughout the simulation process for the final year project. Though your names might not be mentioned here, gratitude to those who have directly or indirectly helped me through the ups and downs, during the execution of the project. This project would not have been completed if not for their help and support.

ABSTRACT

Drag reduction is a field that has been focused on these days, especially in certain fields like the oil and gas industry. Drag reduction agents are used to assist in drag reduction. To analyse the performance of drag reduction agents, rheometers with the double gap cylinder designs are preferred. However, the usage of double gap cylinder in assessing drag reduction has been slightly inaccurate. This is due to a secondary flow formed in the instrument during the assessment, which cause overflow of the fluid when the rotor is rotated up to a certain speed. This secondary flow is closely related to Taylor-Couette flow. Thus, the objective of this study is to investigate the flow behaviour of the fluid in the double concentric cylinder under specific parameters chosen. In order to do so, computational fluid dynamics would be used to the study the flow. Simulation is done using ANSYS Fluent. The geometry model is constructed and exported into Fluent, prior to the appropriate solution setups. Results were analysed though contour of velocity and graphs as well. It is found out that the instabilities starts to form in the region of angular velocities of 13-14 rad/s. Furthermore, the instabilities tend to grow with higher angular velocity applied. Results also shown that there is presence of spillage of water out from the geometry. Therefore, the instabilities in the flow that causes the secondary flow in the double gap cylinder are simulated. Nevertheless, it is recommended that simulation setup is further refined together with other parameters to be assessed further.

Table of Content

LIST OF FIGURES

LIST OF TABLE

CHAPTER 1
INTRODUCTION

Fluids can be transported or transferred using pipes or pipelines. One of the main industries which utilize pipelines is the field of oil and gas whereby pipelines are used to transport crude oil, a liquid of high viscosity and also a non-Newtonian liquid. Often, in these pipelines, turbulence would occur due to the friction between the oil with the inner surface of the pipelines, which would result in the lower rate of transfer of the fluid. The usual methodology to solve such issues would be to improve the mechanical components of the system, such as upgrading of the mechanical pumps, changing of pipes, and so on, which would incur higher cost for the whole process of transportation of the fluid.

With the development in research of drag reduction, polymers functioning as drag reducing agents (DRA) are used to help reduce turbulent friction in pipes. This could reduce the loss in energy in transferring crude oil (the fluid), without the need to improve or to change the mechanical components of the system altogether. In addition to that, certain additives are also used, together with DRA to further improve the effect of DRA on the fluid, which means that the performance of the DRA could be improved.

1.1 PROJECT BACKGROUND

In order to evaluate or measure the performance of any drag reducing agents (DRA) especially with the presence of additives, many methods have been done before indeed, including assessing the friction factor of fluid and polymer solution within the same Reynolds number, calculating the pressure drop between fluids added with DRA and without DRA, and usage of rheometer to assess the performance of DRA. Of all types or geometries of rheometer, including parallel plate and cone and plate; double concentric cylinder is chosen to be the preferred geometry or method (Malik et al., 2010). A rheometer assesses the performance by measuring the torque required to rotate the cylinder up till a required rotational speed with the presence of the sample liquid. This method is known to be much more cost effective as the samples are needed in much smaller amount, compared to the other methodologies, such as flow loop (Henaut et al, 2009). With the significantly lesser amount of sample needed, the usage of rheometer would definitely save a lot of time in assessing the performance of DRA.

1.2 PROBLEM STATEMENT

As stated, drag reduction can be assessed using a double gap cylinder which is also known as double concentric cylinder (Henaut et al, 2009). This is done by measuring the torque required to rotate the spindle of the rotator, while the stator remains stationary. During the assessment of the performance of the drag reduction, the speed or velocity of the rotor would be increased gradually. As the rate of rotation or angular velocity increases, the nature of the fluid in the double concentric cylinder changes. And at certain point, the flow of the fluid becomes unstable and tends to overflow, out of the double concentric cylinder. This is also called as the secondary flow, during the assessment of the drag reduction. Consequently, the instability of the flow in the sample of the liquid causes inaccurate measurement of the torque which is used to assess the drag reduction, hence, inaccurate experimental results. This particular phenomenon is simulated computationally, to investigate the cause of the overflow issue as well as to study the behaviour of the flow in a double gap cylinder.

Figure 1.2: Sample of secondary flow in the double gap cylinder

1.3 OBJECTIVE OF STUDY

The objectives of this study are to:

- Simulate the flow of liquid in a double gap cylinder using a CFD software and to study the behaviour of the flow of liquid through the simulation
- Study the effects of different parameters (different angular speed) involved in a double gap cylinder through simulation

1.4 SCOPE OF STUDY

This study focuses mainly on the rheology, particularly on the instrument rheometer. This study is on the geometry of double gap cylinder for a rheometer. As for the CFD study, the focus is on the behaviour of the flow of liquid in the double concentric cylinder, according to various speeds applied on the rotor. A specific type of flow is also being studied on, named the Taylor-Couette flow.

CHAPTER 2
LITERATURE REVIEW

2.1 DRAG REDUCTION

Significant amount of pressure is often lost in the fluid flows in pipeline, causing loss in energy. This scenario is caused by the friction in between the fluid and the inner walls of the pipeline, which results in the produced drag in the fluid flow. This would be a costly problem for industries, like oil and gas, as there are hundreds of thousands of kilometres of pipelines involved. Various methods have been initiated to solve this problem. Some has resorted to improving the mechanical systems of the pipeline by installing higher rated pumps, changing pipes, etc. Drag reduction method has been introduced, whereby small quantities of additives are added to reduce the turbulent friction factor of the fluid flow (Mowla and Naderi, 2005). In fact, studies on drag reduction have started all the way back from 1948, (Toms, 1948). Toms researched on the effect of certain polymers added into pipe flow in minimising friction and the drop in pressure. The end results here has led to the drag reduction effect and also sometimes referred to as Tom's effect. Percentage of drag reducing can be defined as below:

$$\%\mathrm{DR} = \frac{\Delta \mathrm{p_f} - \Delta \mathrm{p_{fdra}}}{\Delta \mathrm{p_f}} \times 100$$

where $\Delta \mathrm{p_f}$ is the pressure drop without drag reducing agent and $\Delta \mathrm{p_{fdra}}$ is the pressure drop with drag reducing agent.

2.2 DRAG REDUCING AGENT

To induce drag reduction, drag reducing agents (DRA) are used. Currently, there are three groups of DRAs, namely polymers, surfactants and fibers (Mowla and Naderi, 2005). According to Mowla and Naderi, the function of surfactants is to reduce the surface tension of the fluid whereas function of fibers is to reduce drag by orientating themselves in line with the flow direction. Polymers are widely used now, such as poly(ethylene oxide), polyacrylamide and Xanthan gum. Pereira et al. (2013) have research on these polymers which are categorised into flexible and rigid polymers. In their findings, Reynolds number played a minor role for Xanthan gum (rigid) but a major role for poly(ethylene oxide and polyacrylamide in terms of efficiency of the DRA. In a separate finding, drag reduction of about 48% can be achieved in fluid with injection of polymer (Sarkhi and Hanratty, 2000). Sarkhi and Hanratty also mentioned that the effectiveness of drag reduction is dependent on the concentration of polymer in the solution. Temperature has been listed as one of the contributing factors too, to the effectiveness of drag reduction as suggested by Choi et al. (2000) and Karami and Mowla (2013). Numerous case studies have been done on the effectiveness of drag reducing agents, including seawater injection system, wet gas pipeline and fluid motion in pipeline, (Al-Anazi et al., 2006; Chen et al., 2000; Strelnikova et al., 2013). These papers suggest that drag reducing agents acts for a certain period of time and temperature before becoming ineffective. It is further shown by Choi et al. (2000), whereby polymers were found to be experiencing degradation according to an exponential function with respect to time; shown below:

$$\frac{\mathrm{DR}(t)}{\mathrm{DR}(0)} = \exp(-\frac{t}{\lambda_s})$$

where DR(t) and DR(0) are the percentage of drag reduction at times t and t=0, λ_s is and adjustable parameter and $1/\lambda_s$ is a parameter that measures the rate of drag-reduction activity loss or in this case, the rate of degradation. Therefore, this explains the studies done to improve the performance of drag reducing agents, such as by adding additives or others, (Chen and Jaafar, 2012).

2.3 RHEOLOGY

Rheology is the study of deformation and the flow of matter. This study is mainly on flow of matters that are of non-Newtonian fluids, fluids which their viscosity depends on shear rate, unlike Newtonian fluids. Rheometry refers to the techniques or methods involved in the study of rheology. In rheometry, properties of fluid such as steady shear viscosity, viscoelasticity, etc are assessed. Rheometer is the most common instrument used in rheometry such as rheometry of liquid metal systems (Malik et al., 2010), rheometry of cement suspensions (Ferrara et al., 2012) and rheology of sludge (Ratkovich et al. 2012). Most of these researchers have accompanied their experimental results with findings from modelling and simulations. This is done to validate their finding by computational methods.

2.4 RHEOMETER

To measure the effectiveness of drag reduction agents, industries have tried by directly injecting into the pipelines, like Chevron, at their own mobile field in Gulf of Mexico (Chen et al. 2000). Researchers have tried to measure drag reduction using flow loop. Al-Sarkhi et al. (2001) used did so to study on drag reduction in multiphase flow. Mowla and Naderi (2006) have used flow loop to study drag reduction for a two-phase flow of crude oil and pipeline. Karami and Mowla (2013) suggested a model of flow loop in predicting drag reduction in crude oil pipelines. Currently, usage of rheometer in the study of drag reduction has been implemented. This has been done by Henaut et al. (2009) in selecting the most suitable DRA. One of the more recent researches was done by Pereira et al. (2013), using a double gap device, which is related quite closely to rheometer. Chen and Jaffar (2012) too used rheometer to investigate the effects of additives on drag reduction. Usage of rheometer has found to be advantageous compared to flow loop. It requires only a small amount of sample and shorter duration of time needed to obtain the test results. This incurs lesser cost.

Figure 2.3: Rheometer used by Chen and Jaafar

2.5 DOUBLE CONCENTRIC CYLINDER

In the market, three different geometries available for the use of rheometer are, parallel plates, cone and plate and double concentric cylinder. To investigate on drag reduction on crude oil, double concentric cylinder (DCC) is preferred. The concept of usage of DCC is that it can measure the viscosity of the sample fluid, by determining the torque required to rotate the axle of the rheometer device to a specific angular velocity. DCC has been widely used to measure viscosity of fluids including in the food industry; (James et al., 2004), in a project to investigate the feasibility of heating or cooling foodstuffs in a shell-and-tube heat exchanger. DCC is also used to investigate on the degradation of the long chain polymers used as DRA, especially on polyacrylamide and poly (ethylene oxide), (Pereira and Soares, 2012). In their study, Pereira and Soares have utilized the Taylor-Couette geometry as Taylor-Couette flow effect was predicted.

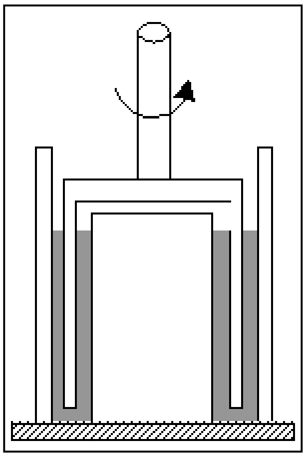


Figure 2.5: Cross-sectional view of a DCC

2.6 TAYLOR-COUETTE FLOW

The Taylor-Couette flow is the phenomenon that happens in the fluid in between two concentric cylinders which are relative in motion, rotationally. In specific, the inner cylinder would be rotating while the outer cylinder is kept fixed. This flow was first researched by G.I. Taylor (1923), whose work is related to another flow, named Couette flow by M. Couette (1890).

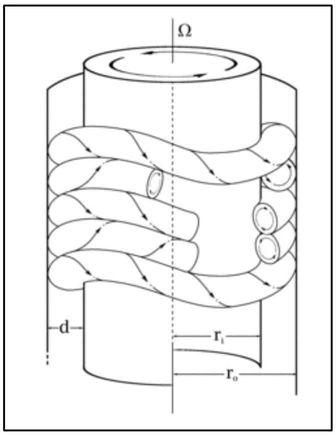


Figure 2.6 (a): Taylor-Couette flow

Taylor was first to be able to relate on the stability of the Couette flow as the rotational speed increases. And, Couette added that the torque, required to rotate the cylinder increases linearly with the rotational speed, but the torque would increase drastically up to a certain point, or known as the critical rotational speed. This is due to the transition of laminar flow to a turbulent flow. At this point, flow instability is produced and hence, vortices are produced. These vortices are called as Taylor vortices. If the rotational speed is increased progressively, turbulence patterns could be observed. To characterize the critical condition for instability, Taylor proposed a parameter, Taylor number; shown below:

$$T = \mathrm{Re}^2\,(h/R_0),$$

where Re is the Reynolds number, related to the gap width (h) and the rotational speed of the inner cylinder; and R_0 is the mean radius of both the inner and outer cylinders (for geometries of concentric cylinders).

From previous analysis, the critical value of the Taylor number for instability is 1708. Studies have shown that the mechanism of the instability in Taylor-Couette flow is related to energy gradient concept (Dou et al., 2008). In fact, many other studies have been done on Taylor-Couette flow by modelling and numerical investigations (Nemri et al., 2012; Kadar and Balan, 2012). However, despite extensive studies, the issue with Taylor-Couette flow has not been resolve completely, especially in DCC. Often, those who have studied drag reduction using

DCC have encountered this problem. This have been specifically mentioned by Chen and Jaafar (2012) in the study of effect of additives on drag reducing agent, as secondary flow was clearly seen in their research using a double gap cylinder.

As for the case of double gap cylinder, there is a unique scenario where two separate flows could be seen for the two separate gaps. Referring to the diagram below of a double concentric cylinder with its geometry,

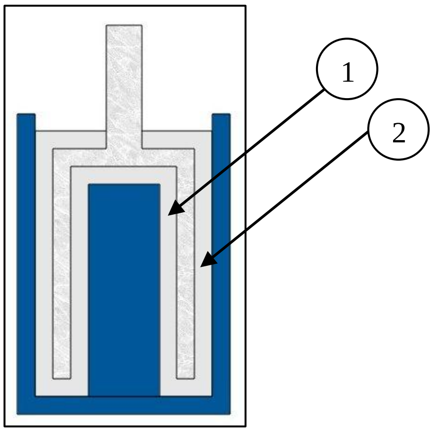


Figure 2.6 (b): Cross-sectional view of a double gap cylinder with labelled gaps

As stated before, (referring to the literature studies), the phenomenon of the Taylor-Couette flow would occur in concentric cylinders when the inner cylinder is rotated with a specific velocity or speed, while the other cylinder is kept stationary (at rest).

However, for a double concentric cylinder, two separate scenarios can be seen. There are two gaps involved in the DCC, which can be referred as the inner gap and the outer gap.

1. For the inner gap (referring to the diagram of the DCC, labelled '1'), the boundaries are the stationary inner wall of the stator as the inner boundary and the rotating wall of the rotor as the outer boundary; whereas
2. For the outer gap (again referring to the diagram of the DCC, labelled '2'), the boundaries are the rotating wall of the rotor as the inner boundary and the stationary outer wall of the stator as the outer boundary.

Definitely, the fluid in the outer gap is experiencing Taylor-Couette flow, as the scenario tallies with the requirement of the Taylor-Couette flow to occur, which is the inner boundary to rotate while the outer boundary to remain stationary. Whereas, for the inner gap, where the fluid is having the opposite conditions experienced by the fluid in the outer gap; the fluid is just experiencing Couette flow. Couette flow is a flow whereby which is normally experienced by fluid in parallel plates where one surface is kept stationary or at rest, whilst the other is moving with a specified velocity.

2.7 COMPUTATIONAL FLUID DYNAMICS (CFD)

It is suggested to model DCC and to simulate the flow of the fluid in DCC while the rotor is being rotated, to get a better understanding of the flow within the geometry. This is to be done to study on the behaviour of the flow of the fluid. Generally, the concepts behind CFD are; finite difference method (FDM), finite element method (FEM), and finite volume method (FVM). FDM is based on the application of polynomials and expansion series to represent the differential equations. FDM is the oldest method and is obsolete in the market today (Jeong and Seong, 2013). In FEM, regions are divided into smaller regions to obtain a piecewise approximation of the solution whereas; in FVM, partial differential equations are to be solved and the conserved variables are to be averaged its values (Jeong and Seong, 2013). Usually, CFD is used to compliment experimental results and has been used to model and simulate various flows of various fluids including microfluidic applications (Glatzel et al., 2008), fluid particles in turbulent flows (Andersson and Helmi, 2013), fluid in rotating wall vessel bioreactors (Cinbiz et al., 2010), simulation of power law and yield stress fluid flow (Wang et al., 2011). Comparatively, in terms of meshing, FVM is better than FEM as FVM is able to work with any types of mesh and need not consider quality and number of mesh elements (Jeong and Seong, 2013). There are also some challenges faced in CFD, such as unstructured mesh generation which is faced especially for machines with lower specifications, (Ito, 2012).

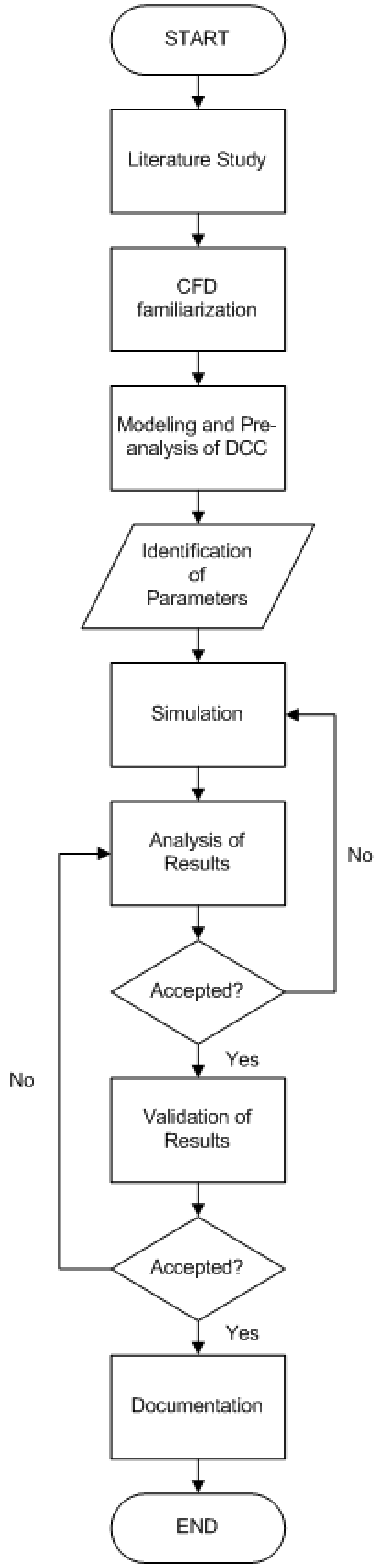
CHAPTER 3
METHODOLGY

For this project, past research papers on drag reduction and on the methods of assessing it are studied. Greater focus was given to the usage of rheometer and rheology, particularly on the geometry of double concentric cylinder (DCC). Hence, the behaviour of the flow of the fluid in DCC is also looked into, which is the main issue as stated in the problem statement. This phenomenon is closely related to a phenomenon known as Taylor-Couette flow and also another phenomenon known as Couette flow, as describe in the literature review. With that, simulation is done to study the flow in the DCC.

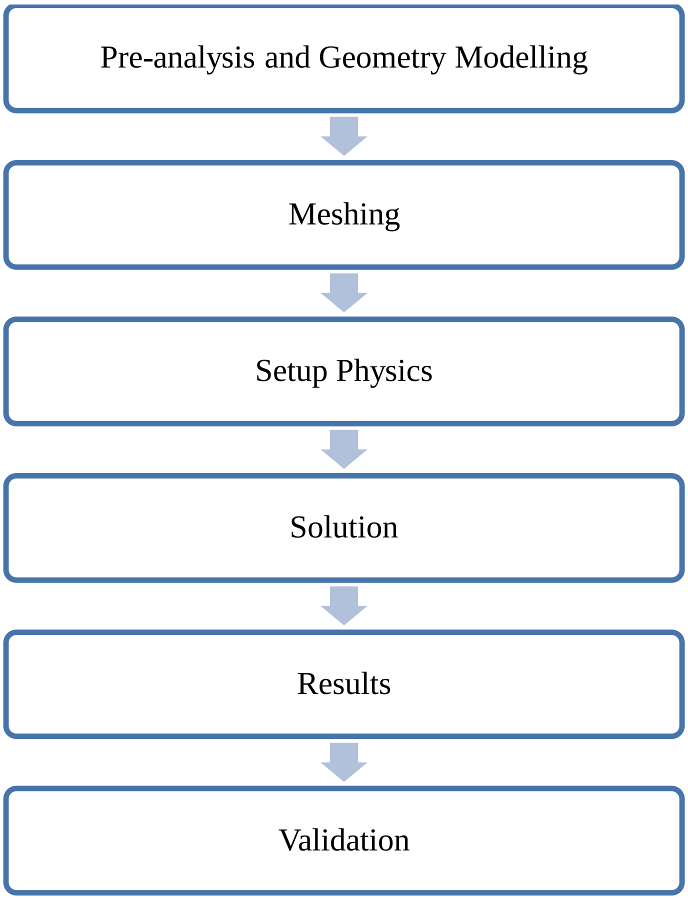
To analyse this, CFD analysis was done to study on the behaviour of the flow of the liquid in the DCC. Different parameters were set, for which the flow of the fluid is to be assessed; but for this study only one is being focused on which is the varying of angular speed/velocity of the spindle/rotor.

ANSYS Fluent is the software used for this simulation project, while the modelling of the DCC would be done using Autodesk Inventor. Finite volume method (FVM) as adopted by Fluent is a good solution as the focus of this study is on fluid (which is the main objective of this project).

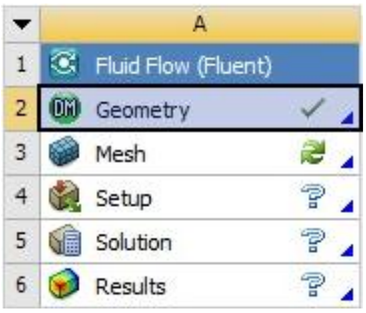
3.1 Process Flowchart of CFD study of DCC



3.2 Work Process Flow of Simulation in ANSYS FLUENT (using Ansys Workbench)



3.2.1 Pre-analysis and Geometry Modeling



This is the very first step of the simulation. The geometry of the double gap cylinder is constructed according to the exact dimensions of the geometry used in the laboratory. These exact dimensions were obtained from the software bundled with the rheometer, to help with the assessment of viscosity of fluids.

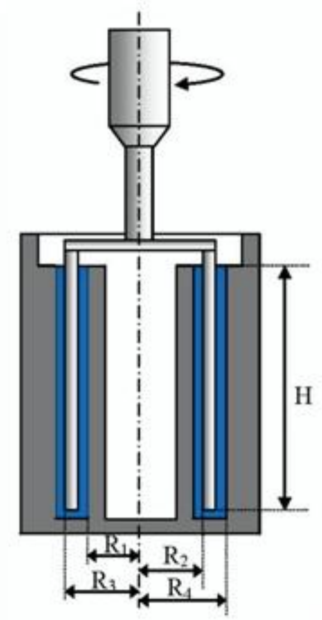


Figure 3.2.1 (a): Cross-sectional view of a DCC
with its important dimensions, shown below

Table 3.2.1: Important Dimensions of the DCC

	Dimension (mm)
R_1	15.14
R_2	16.00
R_3	17.48
R_4	18.51
H	53.00

The identical geometry is drawn using Autodesk Inventor in 3D format, and then exported to a component in ANSYS, called the Design Modeler.

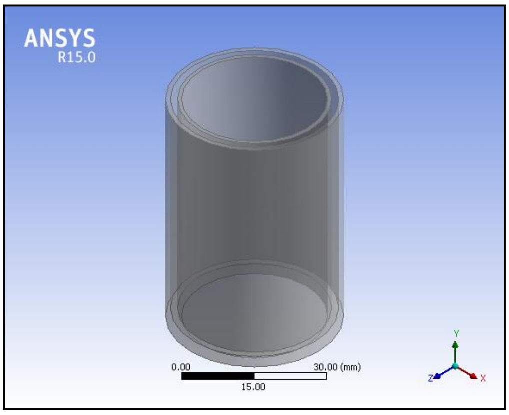
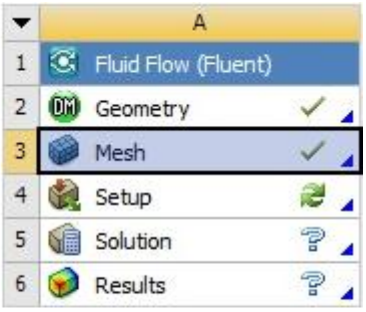


Figure 3.2.1 (b): Exported geometry into the Design Modeler

Prior to the construction of the geometry, pre-analysis has been done to determine the points of interest accurately. These points of interests refer to the important walls or surfaces to be as referral point, to be rotated or to be stationary. As for this simulation project, the identified points of interest are the axis of rotation, rotating walls (rotor) and stationary walls (stator). Next, the constructed geometry is then transferred into ANSYS Fluent for meshing purposes before the simulation process.

3.2.2 Meshing



Meshing is needed to be done after constructing the geometry. It is an important component as it determines the accuracy of the results obtained. The criteria for a good mesh includes size of mesh, uniformity of mesh arrangment, shapes of mesh used, etc. For this project, the minimum size of meshing used is 0.8mm as to accommodate for the duration of time given ensuring that simulation can be done in time for the project study, giving just enough details and not affecting the accuracy of the end results.

Sizing	
Use Advanced Size Fu...	On: Fixed
Relevance Center	Fine
Initial Size Seed	Full Assembly
Smoothing	High
Transition	Slow
Min Size	0.80 mm
Max Face Size	0.80 mm
Max Size	0.80 mm
Growth Rate	Default (1.20)
Minimum Edge Length	95.1270 mm

Figure 3.2.2 (a): Sizing of mesh used

If more detailed results are to be obtained with given extra time, smaller size of mesh with refinement should be done. This would give a more refined and detailed simulation results.

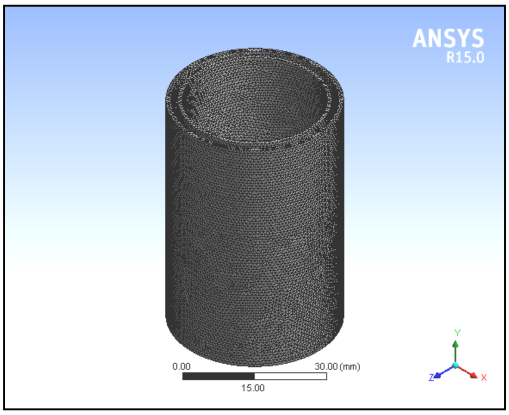


Figure 3.2.2 (b): Meshed Geometry

Then, the walls of the geometry of the fluid are to be named for easier simulation setup later on. The most important walls to be labelled are the cylindrical walls i.e. R1, R2, R3 and R4 and also the surface of the fluid in contact with bottom of the rotor.

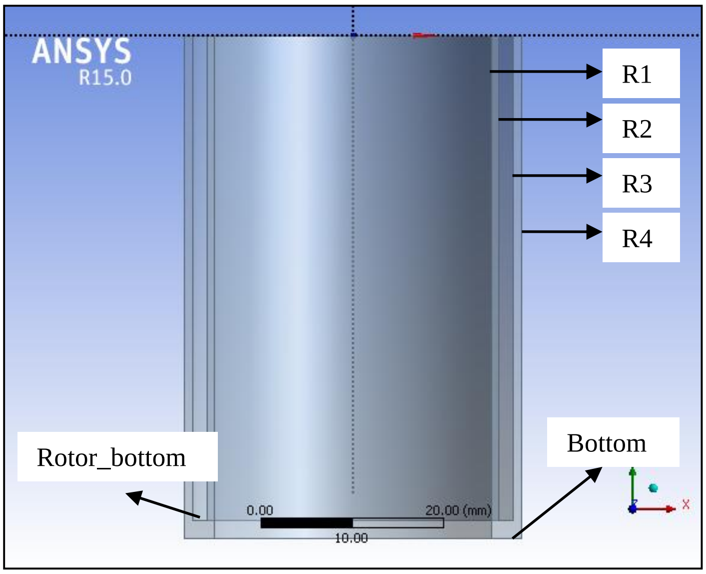
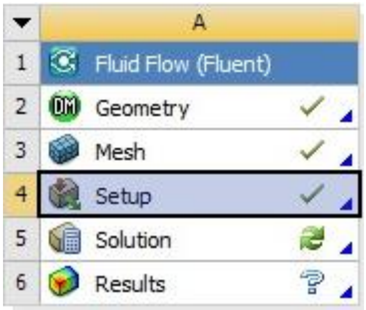


Figure 3.2.2 (c): Important walls to be labelled

3.2.3 Setup Physics



Upon completion of the meshing, the geometry model is then run in ANSYS Fluent 15.0. For this stage, specific settings were set-up for the model, such as general, model, materials, cell zone conditions, boudary conditions, etc. For the general setup, the following setup were chosen:

- Solver type: Pressure-based; as pressure is a true unknown in all fluid flow computations, regardless of whether the flow os compressible or not. Hence, pressure-based algorithms are valid for all cases. Gravity-based is usually used for high speed of higher than 0.7 Mach number
- Velocity formation: Absolute; as simulations are to be done based on fixed reference frames
- Time: Transient; as for the rotational process in a rheometer, it does not follow the steady-state scheme
- Gravitational acceleration of 9.81 m/s^2

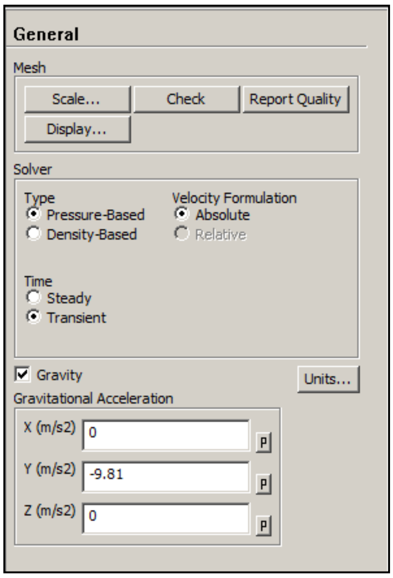


Figure 3.2.3 (a): General setup configured

Next, for models, viscous model is selected with the option of k-epsilon (2eqn) being chosen, with a realizable model and enhanced wall treatment. These settings were used as explained in most tutorials and examples on rotational geometries in Fluent. Enhanced wall treatment was used to obtain better results of fluid flow especially nearer to the walls of rotation.

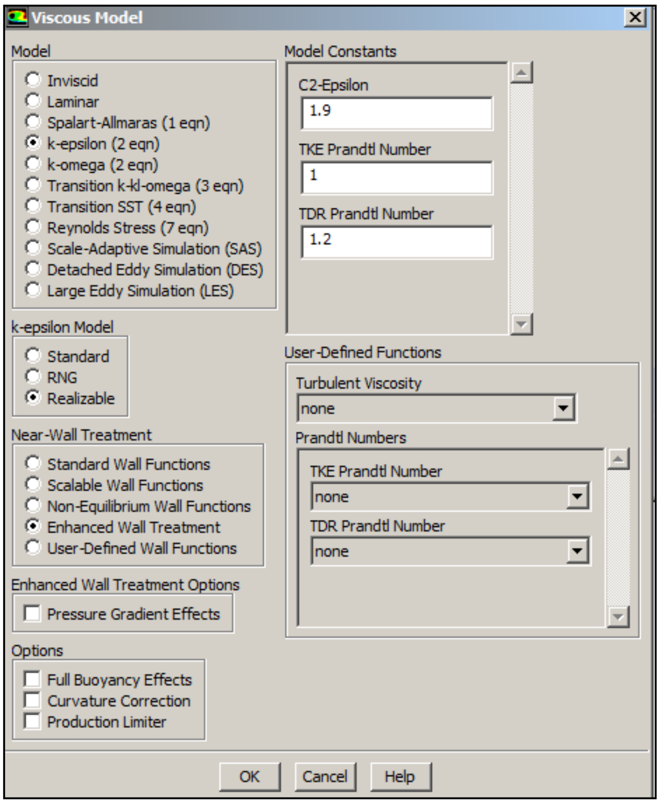


Figure 3.2.3 (b): Viscous model setup

As for the fluid to be simulated, water is selected, a material selected for preliminary results. This is done by adding the water in the materials setup, specifically in the fluent database materials, whereby the properties of are linked to each material.

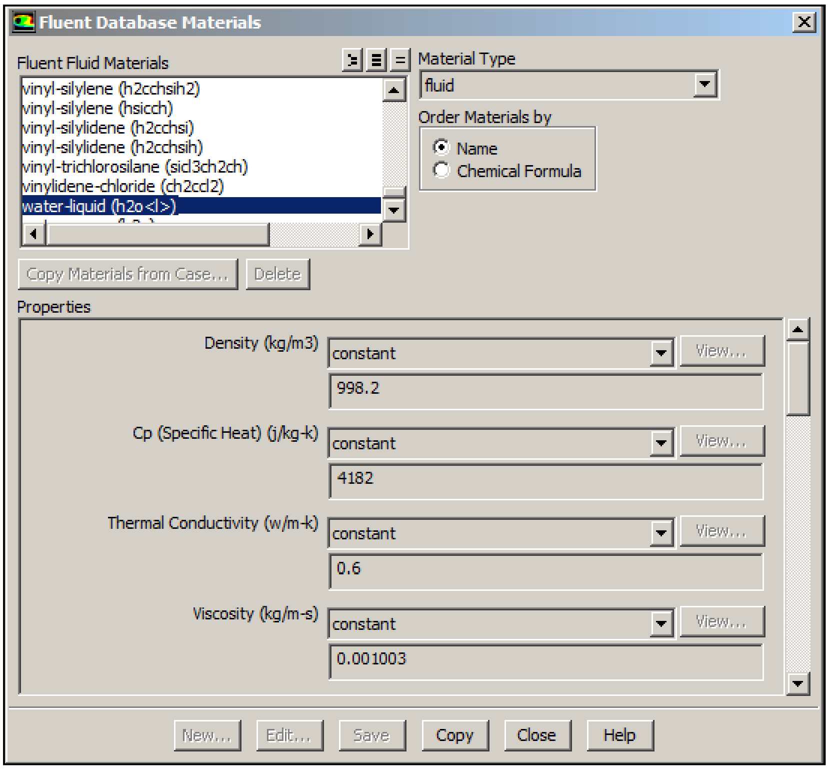


Figure 3.2.3 (c): Fluent database material

The following step would be to configure the settings for cell zone and boundary conditions. Water is selected as the material to be used under the cell zone conditions.

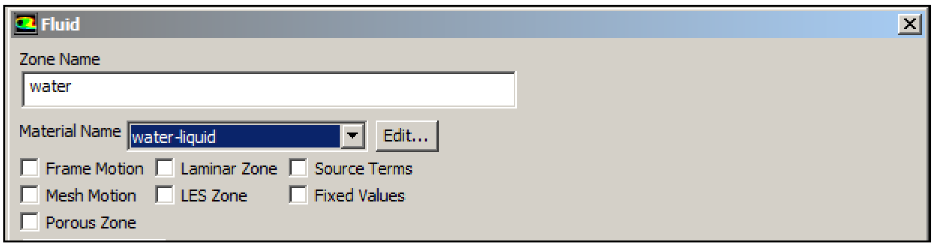


Figure 3.2.3 (d): Cell zone conditions

Under the boundary conditions, for the double gap cylinder, three walls (R2, R3 and rotor_bottom) are to be rotated with the specified angular velocity ranging in between 0 to 300 rad/s (range of angular velocity given by the manufacturer of the rheometer), with no slip condition.

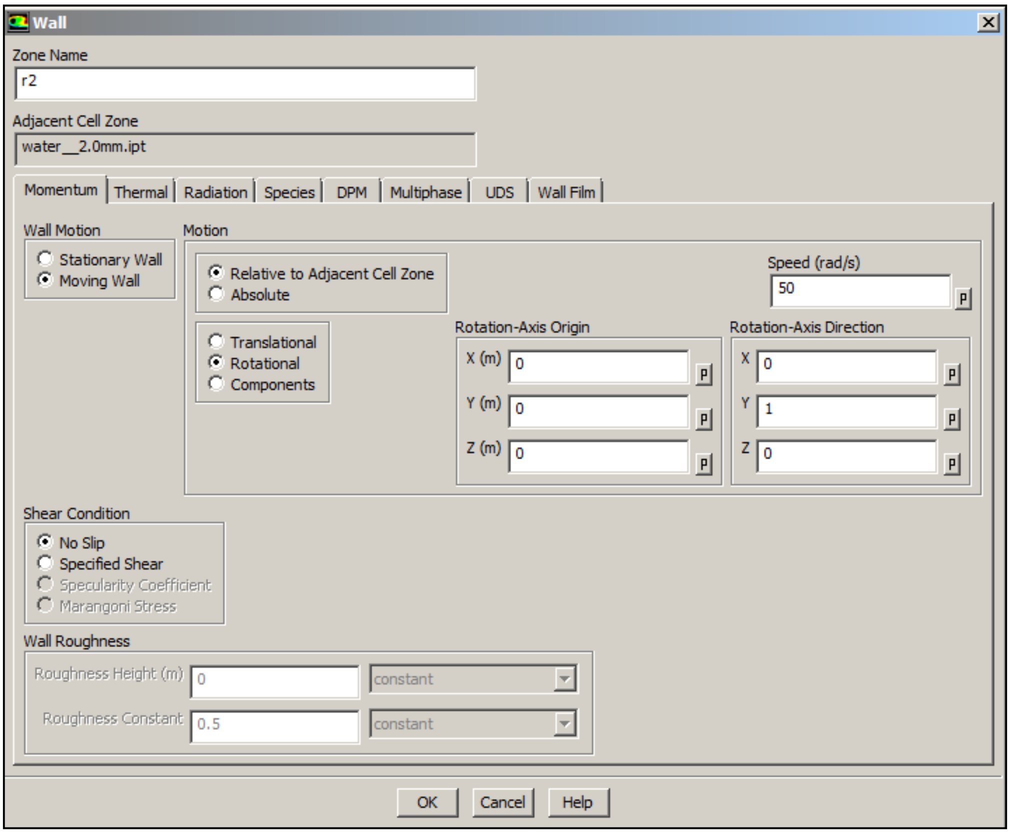


Figure 3.2.3 (e): Boundary conditions of wall R2

Next, dynamic mesh is setup for a few surfaces/walls of the fluid which are in contact with the stator of the double gap cylinder (namely R1, R4 and the bottom, referring to figure 3.2.2(c)). These surfaces/walls are set to be stationary.

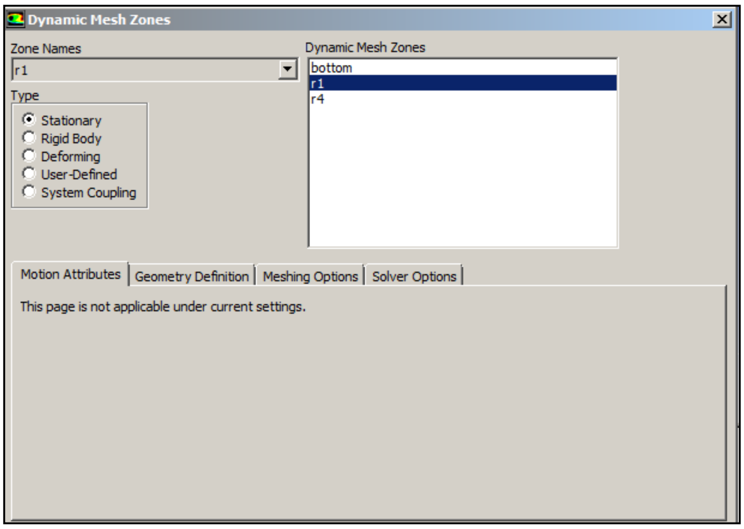
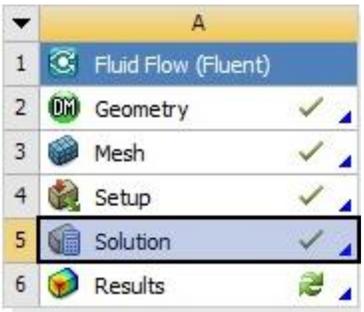


Figure 3.2.3 (f): Stationary wall setup in Dynamic Mesh

3.2.4 Solution



Upon setting up the solution setup, the model is not ready to be solved. The solution method selected is "SIMPLE" which stands for Semi-Implicit Method for Pressure Linked Equations. "SIMPLE" is chosen as it is a practical procedure used to solve Navier-Stokes equations. Second Order Implicit option for transient formulation is used. This is a common and advised practise to achieve better accuracy in the results of the simulation.

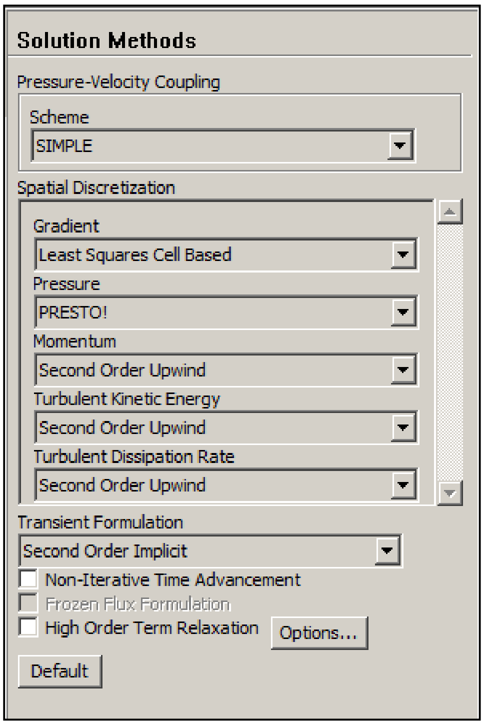


Figure 3.2.4 (a): Solution Methods

The time step used is 0.01s and the number of iterations vary for each case as to get the solution to converge. Initially, 100 iterations were set. To obtain improved results, it is advisable to increase the number of iterations and reduce the time step taken.

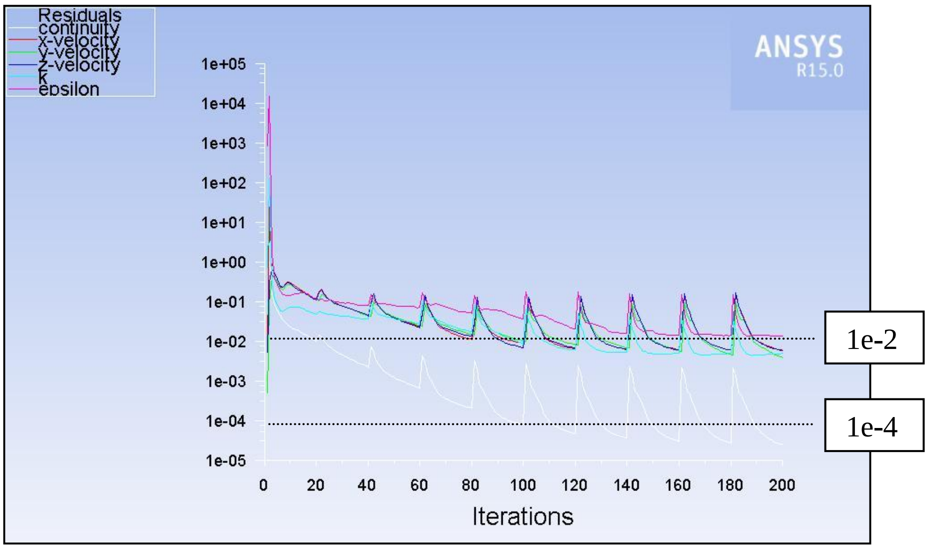


Figure 3.2.4 (b): Example of residual plot

As for the convergence criteria, the selected criteria are as shown as the figure below:

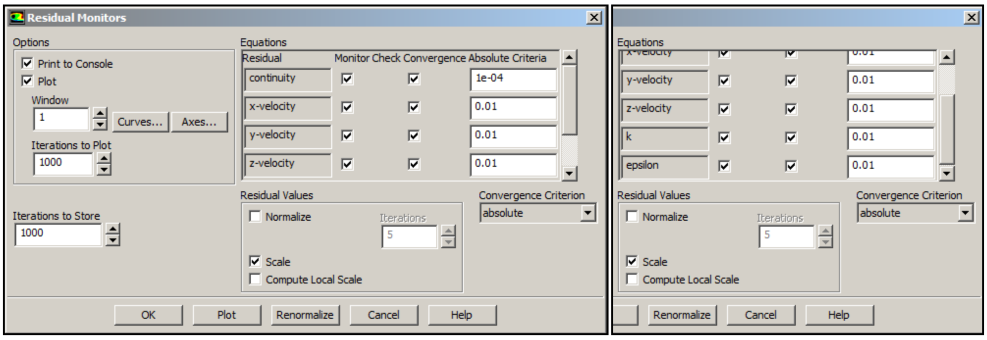
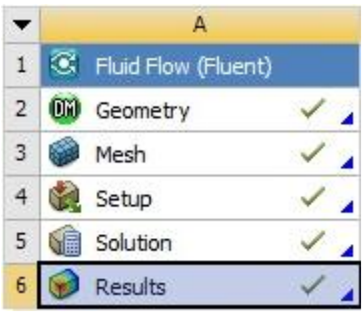


Figure 3.2.4 (c): Selected convergence criteria

3.2.5 Results



As for results, the usual component used is CFD-Post which is bundled up together in the ANSYS software package. Here, contours, vectors, streamlines, etc can be produced to show clearly the results of simulation for further analysis. To simplify the results, it can be opted to show contours or vectors only on a surface, eg, a plane, rather than the whole voloume of the geometry. For this simulation, contour is chosen to display the results, which would be shown on a selected plane (bisecting plane). Besides, charts or graphs can be drawn too, to display results in graphical formats.

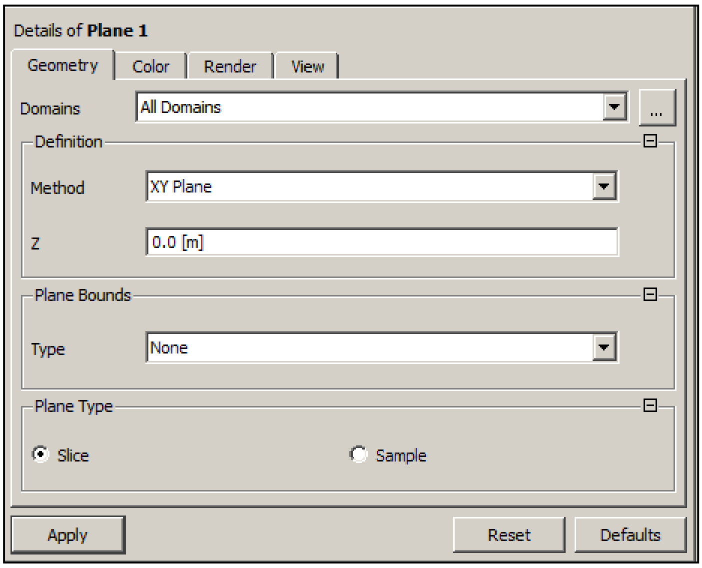


Figure 3.2.5 (a): Configuration of the bisecting plane

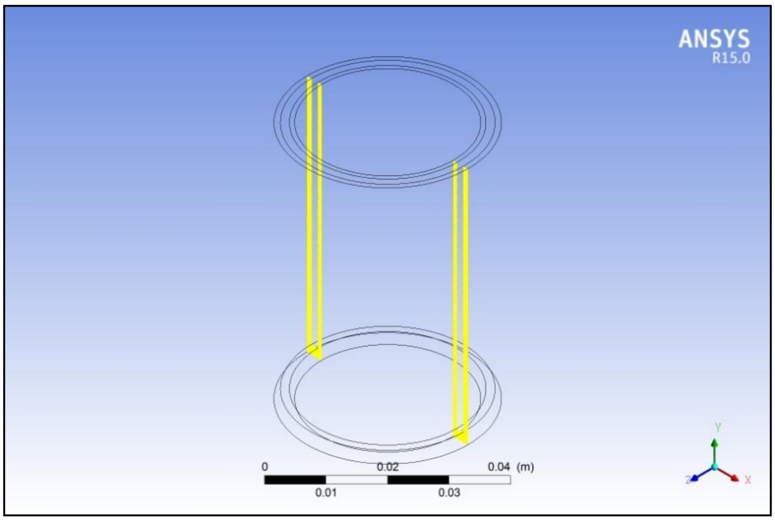


Figure 3.2.5 (b): Bisecting planes in CFD-Post

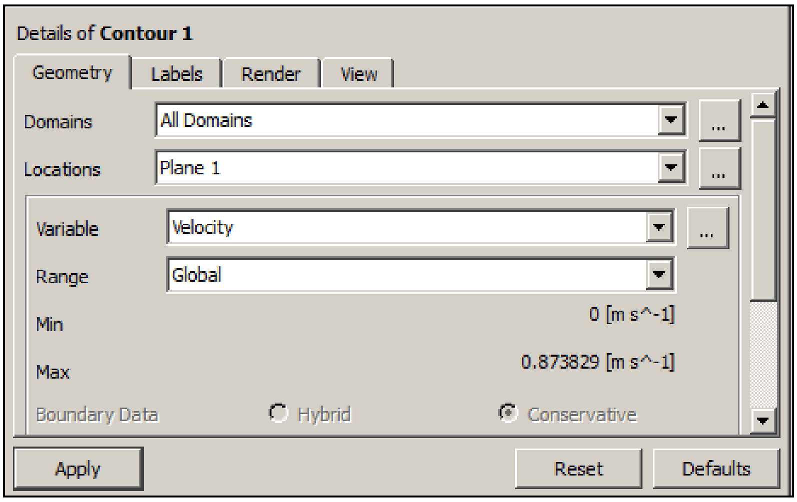


Figure 3.2.5 (c): Configuration of contour

3.2.6 Validation

Based on readings on previous papers, it was predicted that there would be disturbance in the flow of liquid in the geometry of double gap cylinder, which means that there would be irregularities in the velocity of fluid flow in the double gap cylinder. Based on previous experiments as well, there is the formation of secondary flow, when there is an overflow issue out from the geometry of the double gap cylinder. Hence, there should be liquid splashing out from the geometry model. It is predicted that the irregularities in the fluid flow which would be seen in the velocity of the fluid causes the overflow, whereby collisions between fluid particles of various speeds happens. Therefore, these are the results which are of interest for this simulation project.

3.3 Additional setup for second simulation

The second simulation is done to simulate the interaction between the surfaces of the top of the liquid in the double gap cylinder with the surrounding air. The aim is to visualise the spillage or secondary flow of the liquid.

3.3.1 Geometric Modeling

The model is remodeled together with the air section above the double gap cylinder. The whole model is constructed as a single geometry as the multiphase model is to be selected later in the solution setup.

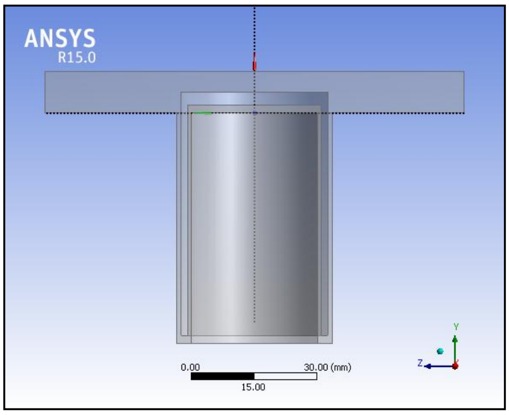


Figure 3.3.1: Cross-sectional view of the double gap cylinder with the air section of top

3.3.2 Meshing

The similar setup for meshing is conducted except for the labelling of several significant surfaces in addtion, which are the rotor sufaces in contact with the air (top and bottom).

3.3.3 Setup physics

Now that the air section is modeled, the multiphase model is to be selected in the models section. Volume of Fluid (VOF) is opted with the Implicit Body Force turned on.

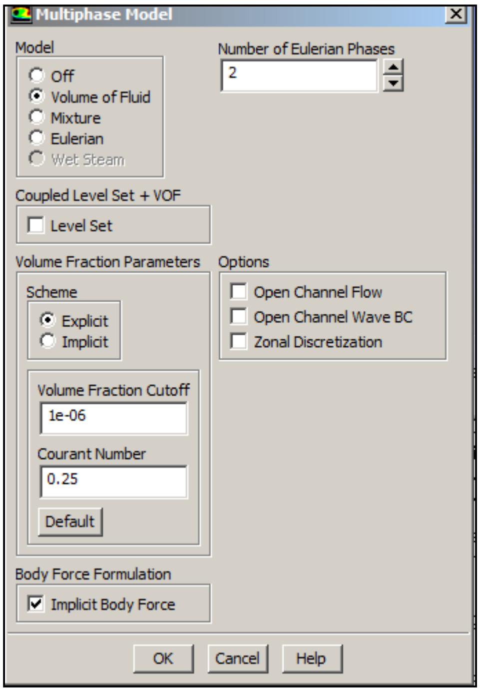


Figure 3.3.3 (a): Multiphase Model

As for Boundary Conditions, additional walls are to be rotated, namely the surfaces/walls of the rotor in-contact with the air (top and bottom).

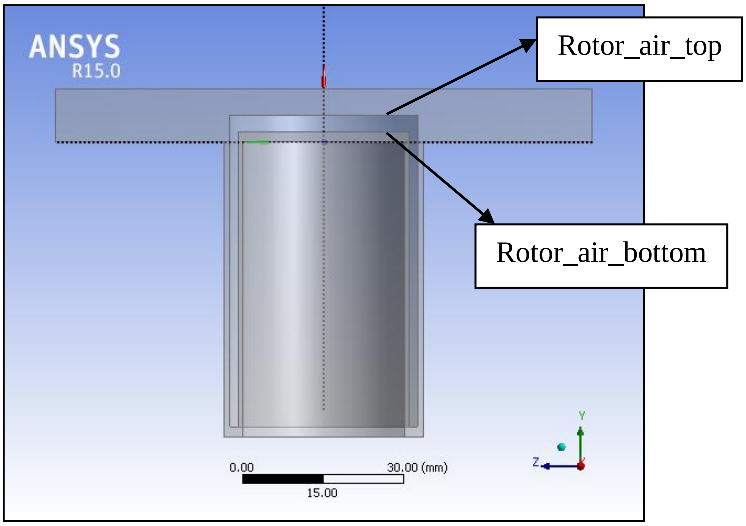


Figure 3.3.3 (b): Labelled walls

Then, solution initialization is conducted with the option of the air Volume Fraction to be '1'. This is done to ensure that the whole 3D model is filled with 100% of air.

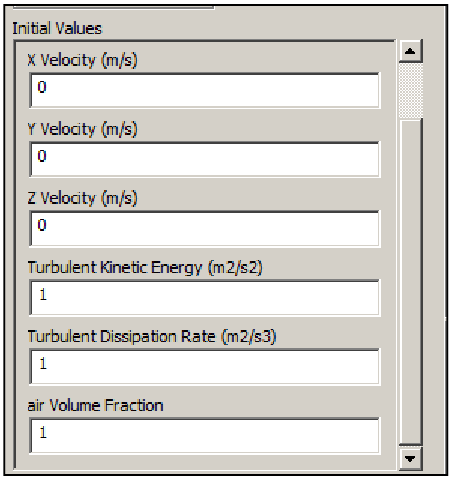


Figure 3.3.3 (c): Initialization of solid model

To set the lower section of the geometry to be filled with water, 'patching' is done. 'Patching' is a method to set an initialization criteria that the volume of a certain region to be filled with only one material/fluid before further simulation process.

Prior to that, region adaptation is conducted to mark the section which is to be filled with water. Exact coordinates are needed to ensure that the marking is accurate.

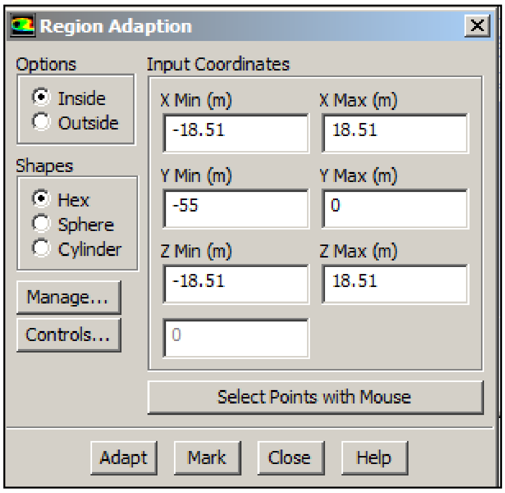


Figure 3.3.3 (d): Marking in Region Adaption

Next, to patch, the value is assigned to '0' for air for the marked section. This would ensure that the the air 'material' is to be of zero volume for the marked section and hence, to be filled with water.

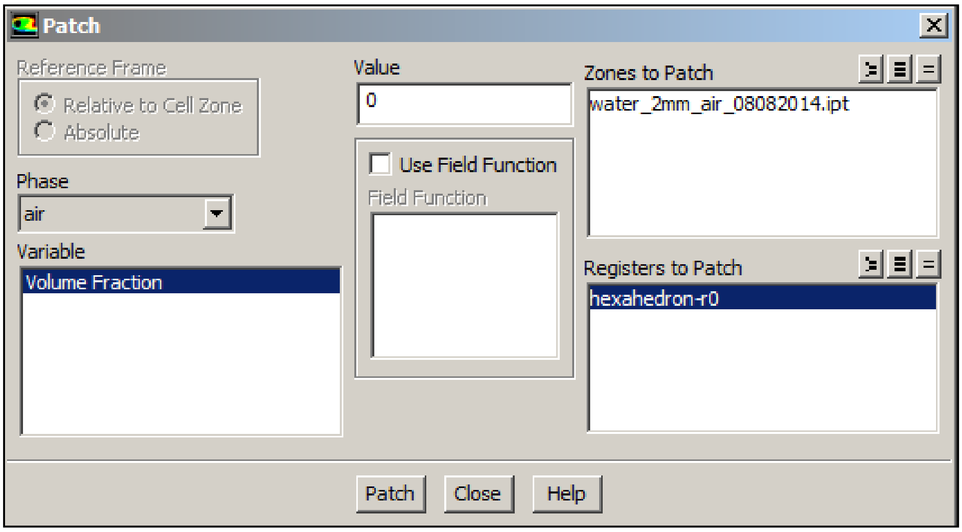


Figure 3.3.3 (e): Patching marked section

3.3.4 Solution

Basically, the setup would be the same as the previous simulation.

3.4 Software

ANSYS Fluent 15.0 is the computational fluid dynamics (CFD) software used to simulate the fluid flow in this project. It has the all the capability starting from modeling to meshing to solution and to present results for the simulation.

3.5 Gantt chart for Final Year Project

	SEMESTER 1														SEMESTER 2														
Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28	29
Selecting topic	■	■																											
Defining scope and research on background of study		■	■																										
Literature study - Drag reduction and drag reducing agent - Rheology and rheometer - Double Concentric Cylinder - Taylor-Couette Flow - CFD			■	■	■																								
Extended Proposal						▲																							
Familiarisation - Rheometer - ANSYS Fluent							■	■	■	■	■	■																	
Proposal Defence								▲																					
Develop ANSYS Fluent methodology												■	■																
Interim Report														▲															
Geometry Modelling of DCC and meshing of DCC															■	■	■												
Simulation of fluid flow																	■	■	■	■									
Analysis of results of simulation																			■	■									
Progress Report																					▲								
Improvement of simulation - Setting different parameters																				■	■								
Further analysis of simulation																				■	■	■	■						
Concluding findings from simulation																						■	■	■					
Pre-SEDEX																								▲					
Final Report																									▲				
Viva																											▲		
Dissertation																													▲

▲ - *Key milestone*

Figure 3.5: Gantt Chart

CHAPTER 4
RESULTS AND DISCUSSION

4.1 SIMULATION ON VARYING THE ANGULAR SPEED

The first simulation was done with the focus on varying the angular speed of the rotor (in this project; it would be the surfaces/walls in contact with the rotor. The overflow issue that occur in the double gap cylinder is closely related to the Taylor-Couette flow. Hence, to portray the influence of Taylor-Couette flow in the flow of liquid in the double gap cylinder, the velocity of the flow of the water in the geometry is assessed. Therefore, contour of velocity is the preferred graphical representation for this.

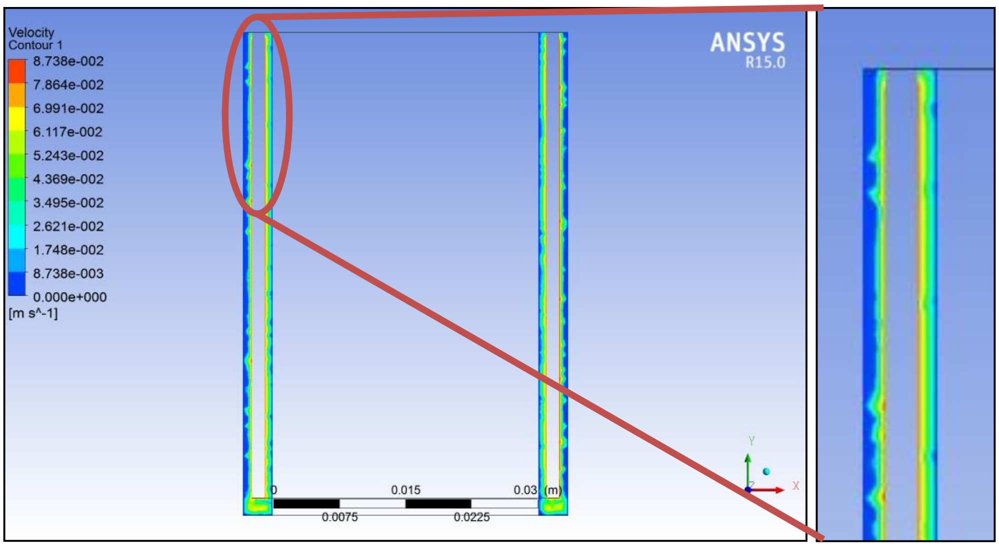
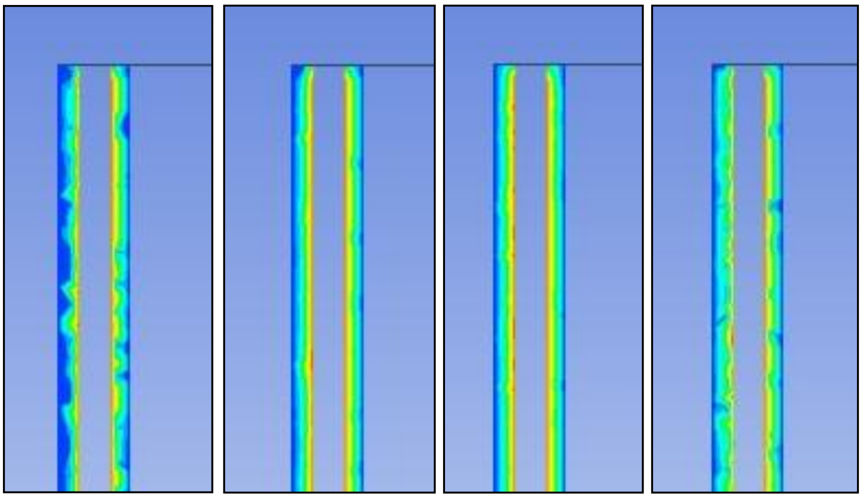


Figure 4.2 (a): Contour of velocity of water at angular velocity of 5 rad/s with its magnified contour

The first angular velocity assessed is as low as 5 rad/s and the contour of velocity is shown above. According to the contour, majority of the volume water on the outer gap is experiencing low velocity or even zero velocity (indicated by the blue region). This is probably due to the low angular velocity applied on the surfaces/walls. It shows that some of water is having zero nett velocity i.e. not moving at all. Nevertheless, there are still streams of water to be moving with velocity, in between the region of stagnant water sections.

To analyse further, the value of angular velocity is increased and the contour of velocity is analysed again. For convenience sake, the angular velocities assessed are in the region of around 13 rad/s. This is in accordance with the study by Chen and Jaafar (2012) that the Taylor instabilities start to form at/after an angular velocity of 13 rad/s. Thus, the angular velocities of 11 rad/s till 14 rad/s are analysed.



11rad/s 12 rad/s 13 rad/s 14 rad/s

Figure 4.2 (b): Magnified contours of water at specified angular velocities

From the figure above, a comparison of the velocity contour can be done. As the angular velocity increases, the blue regions tend to reduce in area, which indicates that less water is being stationary and more volume of water to be moving with velocity. The volume of water to gain rotational speed (resulting from the rotation of the rotor walls) is gradually increasing only till the angular velocity of 13 rad/s. It can be conveyed here that the water (liquid) in the double gap cylinder are moving in a manner such that the vertical layer of water nearest to rotor wall to be rotating the fastest, followed by the second vertical layer with the second highest speed and so on; till the vertical layer furthest from the rotor (nearest to the stator) to be travelling with the lowest speed. This can only be seen for the contour of velocity at 13 rad/s.

However, at angular velocity of 14 rad/s, irregularities in the velocity contour can be observed. The water (liquid) in the double gap cylinder is no longer moving with angular speed according to vertical layers (classified by the relative distance from the rotor). Possible cause for the irregularities in the contour velocity for 14 rad/s is the formation of Taylor instabilities. This is according to an explanation that; vortices known as Taylor vortices are formed. It happens during the transformation of laminar flow to turbulent flow (Dou et al., 2008).

These irregularities in the velocity of fluid flow can be seen clearer at higher angular velocity applied. For discussion, 200 rad/s is selected, quite a high angular velocity within the range of operation of the rheometer (0 to 300 rad/s).

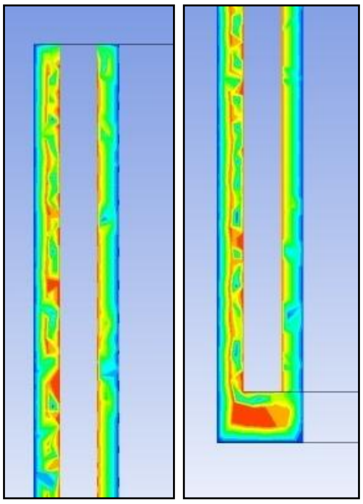


Figure 4.2 (c): Contour of Velocity of the top section (left) and at the bottom (right) for 200 rad/s

Form the figure above, the disturbance or irregularities in the velocity has become much more significant. Patches of red region (higher velocity) is seen not only at the surface of the rotor, but at random locations of the water (liquid). As a result, uniform layers of water which travels at different velocities can no longer be identified. Apart from that, at the bottom of the double gap cylinder, there is a large region of high velocity flow of liquid, shown as the large red and orange patch. The existence of this region with higher velocity further predicts the occurrence of instabilities in the flow of the liquid.

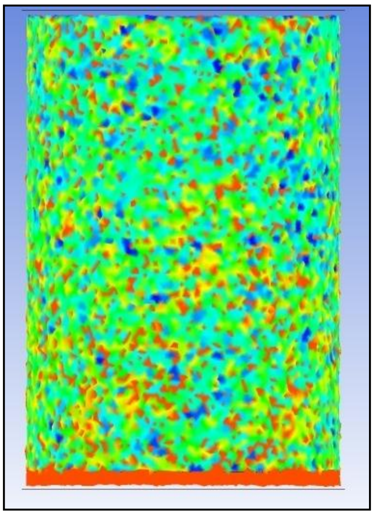


Figure 4.2 (d): 2D view of the isosurface of the velocity

The isosurface shown is drawn according to various velocities in the double gap cylinder. This highlights the irregular layers of water (liquid) which moves or rotates with their different velocities respectively. Here too, the high velocity region formed at the bottom of the geometry could be seen. This probably happened due to the rotation of the surface of the bottom section of the rotor.

Another method to analyse the instabilities formed in the geometry during rotation is through graphs of velocities. For analysis purposes, parallel lines were formed, starting with a distance of 17.55 mm from the axis of rotation, and subsequently with gaps of 0.1 mm; hence, 17.65mm, 17.75 mm, 17.85 mm and 17.95 mm respectively.

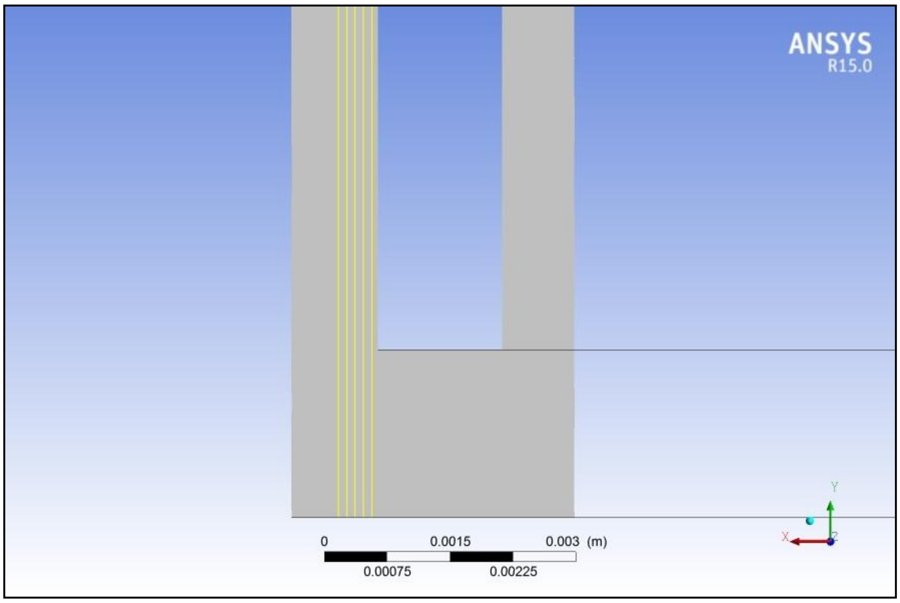


Figure 4.2 (e): Parallel lines drawn on plane;
from right to left: 17.55mm, 17.65mm, 17.75mm, 17.85mm, 17.95mm

Clearly, the focus of this particular analysis is on the outer gap of the double gap cylinder, where there are more significant instabilities, as well as the criteria for Taylor-Couette flow is matched.

Ideally, the velocity nearest to the rotor is the highest and the slowest is to be at the surface of the stator (furthest away from the rotor). However, based on previous results on the contours of velocity, it is known that the velocity at certain surfaces, regardless of the distance from the rotor; can achieve high velocities, as high as the velocities of the fluid nearest to the rotor. The graph below would show the relation in a clearer manner.

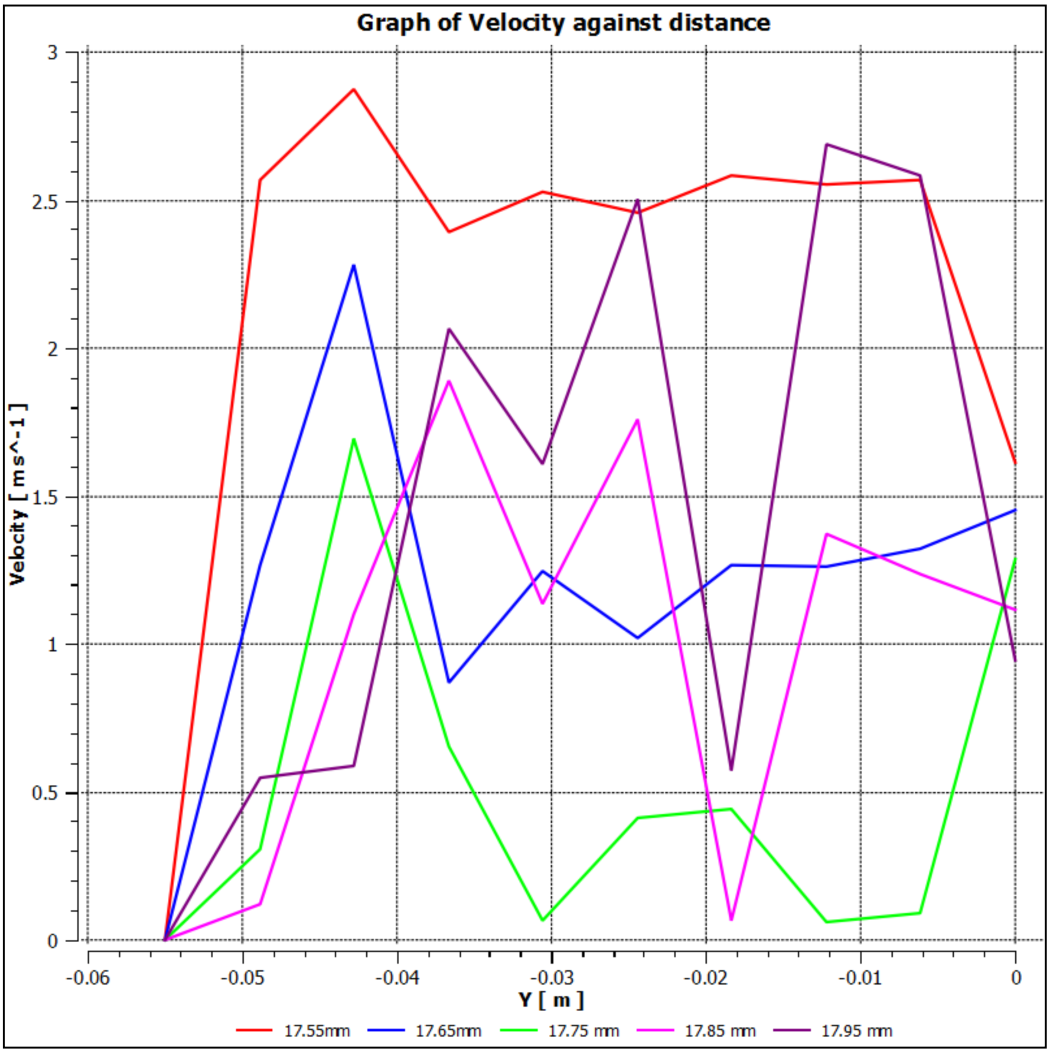


Figure 4.2 (f): Graph of Velocity against distance

For this graph, the x-axis shows the height of the water in the geometry, equivalent to 55 mm. From the figure, all the graphs starts at zero velocity right at the surface of the bottom of the geometry, but ends with various velocities at the surface of the top of the geometry. Different velocities at the surface of the water could contribute towards the spillage issue as the liquid particles travelling with different velocity could experience frequent collisions within the geometry; hence causing splashes.

The main observation from the graph is that there is overlapping of velocities for different vertical surfaces of different distance away from the axis of rotation. For instance, graph for 'surface 17.95 mm' shows the most irregular velocity and could be as high velocity as for 'surface 17.55 mm' at certain vertical distance.

Besides assessing velocity, another physical quality, turbulence kinetic energy can also show the instabilities formed in the flow of water in the geometry.

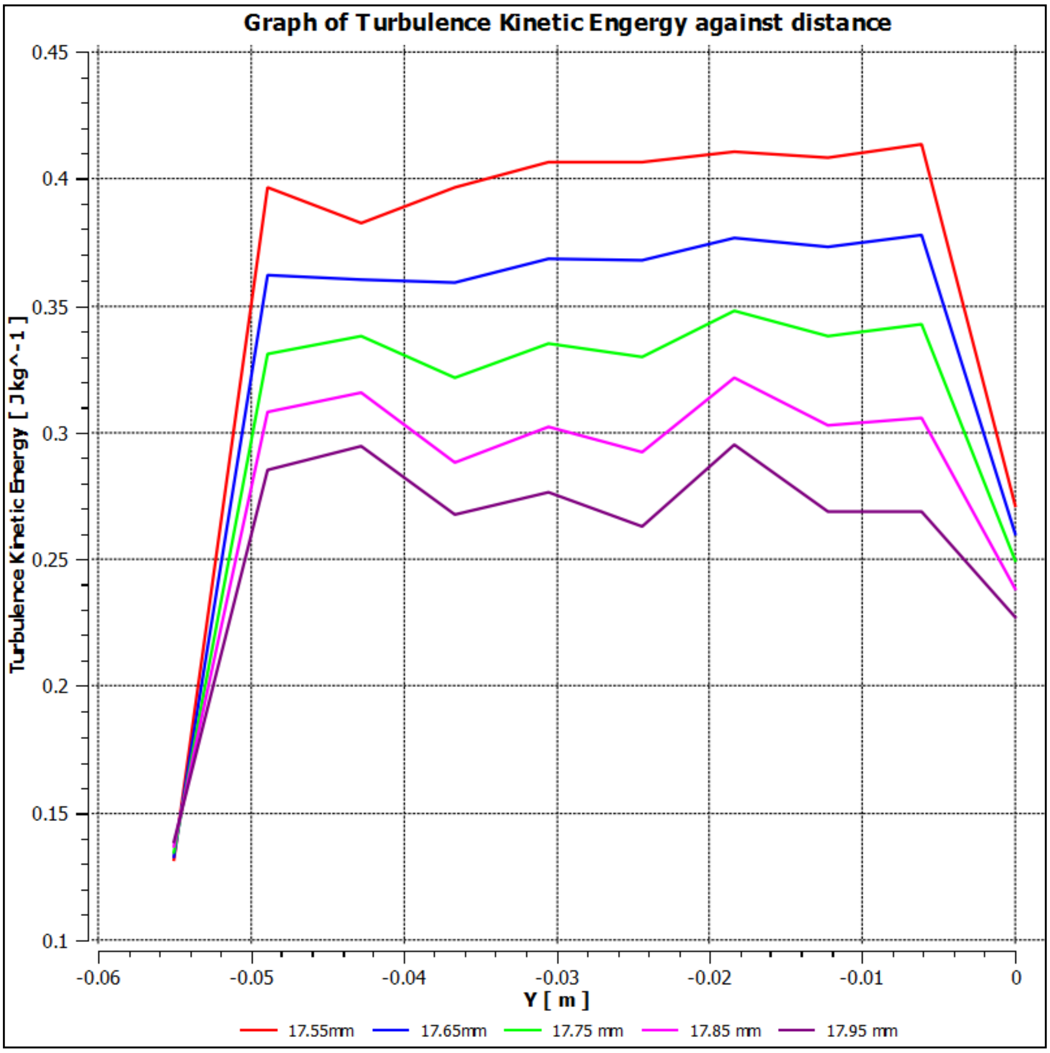


Figure 4.2 (g): Graph of Turbulence Kinetic Energy against distance

For the graph as shown in the figure above, similar setup for the positioning of the surfaces of different distance from the centre is configured. Obviously, the surface nearest to the rotor, i.e. 17.55 mm portrays the highest turbulence kinetic energy (red graph) and on the other opposite, the furthest away from the centre, 17.95 mm portrays the lowest turbulence kinetic energy (purple graph). However, the stability of the turbulence kinetic energy tends to reduce as it moves further away from the centre. Based on the graph for surface 17.55 mm, the graph is almost constant, approximately in between height of 5 mm till 50 mm. Almost similar consistency can be seen for the blue line. But, the same consistency is not valid for the next subsequent graphs as instability is shown. It can be inferred that instabilities occur more frequent at surfaces further away from the centre of rotation.

For this simulation setup, the interaction between the surface of the water at the top of geometry with the particles of air in contact could not be assessed. This requires the geometry of air to be modelled together as well. The geometry of the air need not be of any exact dimension or with any specific boundary conditions. The next part of results is focused on the spillage of water out from the double gap cylinder.

4.2 SIMULATION ON SECONDARY FLOW

For the second simulation, the main objective is to show the presence of the secondary flow in the double gap cylinder. In a paper, “Effects of Additives on the performance of drag reduction agents”, by Chen and Jaafar (2012), it was mentioned that there was spillage of sample during the assessment of drag reduction at a certain angular speed. This overflow issue or rather secondary flow is caused by the disturbance of flow within the double gap cylinder. This phenomenon is also supported by Pereira and Soares (2012) and Pereira et al. (2013).

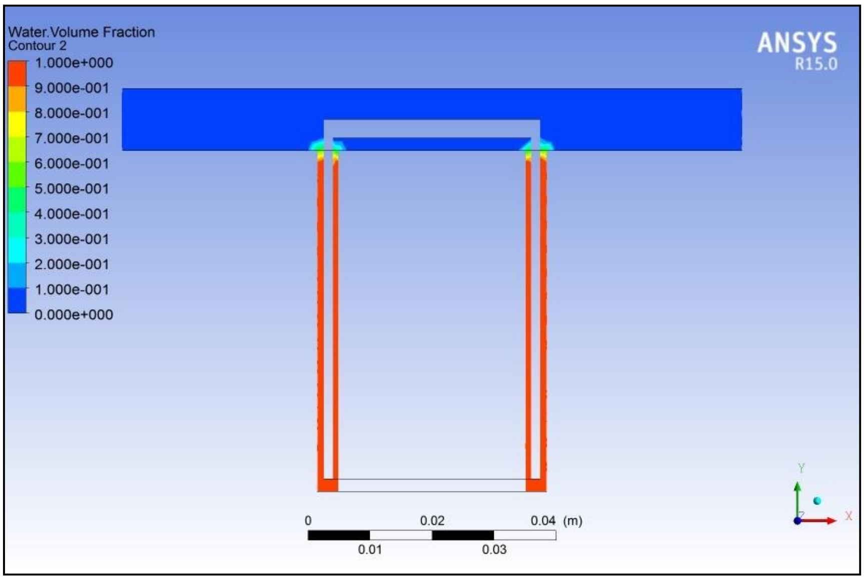


Figure 4.3 (a): Contour of water volume fraction with an angular velocity on the rotor

In ANSYS Fluent, this (figure above) is managed to be seen as shown above, for an angular velocity of 15 rad/s, which corresponds to the findings by Chen and Jaafar (2012) that secondary flow initiates at angular velocity of 13 rad/s. The contour of water volume fraction above clearly shows that the there is presence of water into the supposed-to-be air section; during rotation of the rotor (in this case, it would be the walls or surfaces in contact with the rotor). It is indicated that the water is actually splashing out from the surface of the water in the double gap cylinder. The scene can be seen clearer in the following figure:

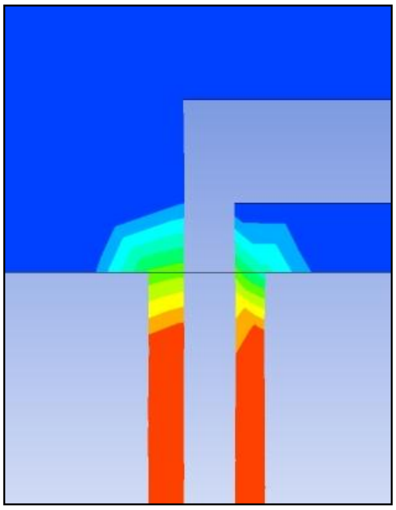


Figure 4.3 (b): Magnified view of water splashing

This simulation actually tallies with the statement that the liquid in the double gap cylinder is overflowing out of the stator during the assessment of viscosity of the liquid for the purpose of assessment of drag reduction. To further show the splashing of water on the surface, an isosurface can be used, as below:

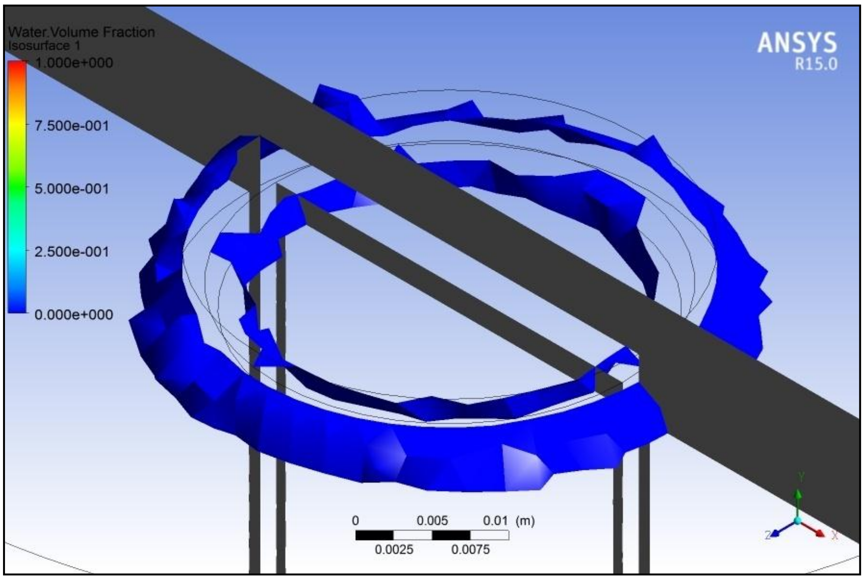


Figure 4.3 (c): Isosurface of the top surface of water (liquid)

Ideally, for an accurate measurement of viscosity of fluid in a double gap cylinder, the flow of water should not affect the air section, hence not resulting in a secondary flow. The ideal condition; which is without the presence of splashing, is shown in the following contour, with zero angular velocity on all surfaces.

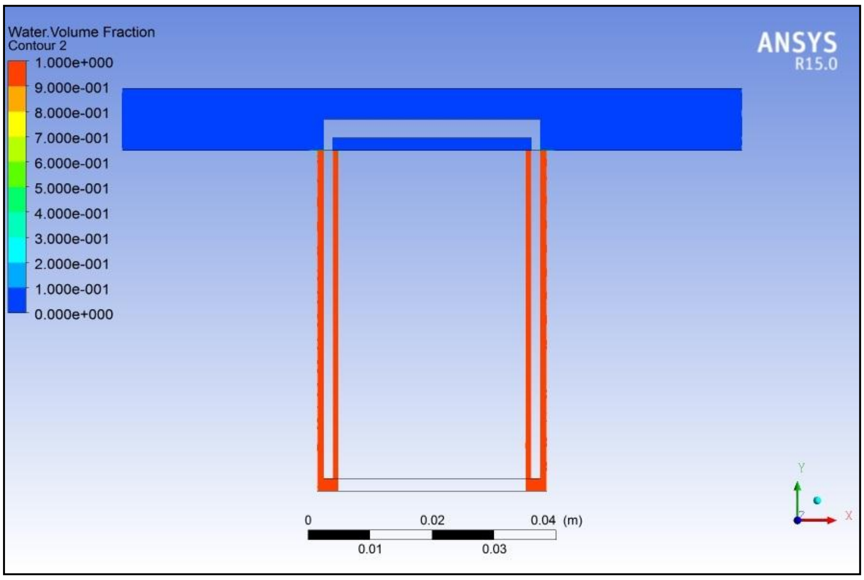


Figure 4.3 (d): Contour of water volume section with zero angular velocity

However, there are indeed some shortcomings for this simulation. Firstly, this setup of simulation does not display the speed of both water and air separately. It is only able to display the velocity of the fluid (water and air) as a whole. It is definitely important to display the velocities of water and air separately to show the predicted irregularities in the speed of the flow of water in the double gap cylinder.

Another shortcoming of this simulation is that it could not show the total volume of water which displaces the air in the air section. This would also prove the presence of water in the air section due to secondary flow issues.

Finally, there is also an error in this simulation; whereby the displayed water volume fraction which shows the overflow issue can actually be seen for angular velocity as low as 1 rad/s. This is actually rather inaccurate as it was reported only at or more than 13 rad/s, Taylor instabilities would occur (Chen and Jaafar, 2012). This could be due to incomplete defined functions for the input of the simulation setup.

CHAPTER 5
CONCLUSION AND RECOMMENDATIONS

A rheometer with double gap cylinder geometry is an important instrument or tool used to assess performance in drag reduction as it saves a lot on the quantity of the sample and also time for assessment. It is indeed an important task to investigate on the behaviour of the flow in a double gap cylinder or also known as double concentric cylinder. The scenario of the secondary flow has to be simulated to help with further studies to come up with a solution to reduce the effect of this instability. Hence, it is hope that the inaccuracy caused could be minimised.

Through the CFD study of the flow of fluid in double gap cylinder and utilizing the ANSYS Fluent software, the flow of the liquid is simulated successfully. Various input and setup are done in accordance to specific research papers and also the exact AR-G2 rheometer in the laboratory. For this simulation project, all the objectives were met as the behaviour flow in the double gap cylinder is studied and simulated in ANSYS Fluent, with varying angular velocities as the input variables.

Based on the results obtained through the simulation, the irregularities in the velocity of the fluid flowing in the double gap cylinder are demonstrated clearly using contours and isosurfaces. Results show that the instabilities formed at speed in the region of 13-14 rad/s, which tallies with the claim by Chen and Jaafar (2012). The non-uniform shapes formed in the contours and isosurfaces of flow at higher velocities definitely shows the higher tendency of instabilities formed in the liquid during rotation of the rotor. This claim is further strengthened through the graphs of velocities against vertical distances, the velocities at different surfaces of the water in the outer gap is shown. The graphs display inconsistent velocities of water regardless of the distance of water from the axis of rotation.

Furthermore, the splashing or spillage of liquid out from the geometry of double gap cylinder was managed to be simulated. This is clearly shown by the contour of water volume fraction.

Unfortunately, the irregularities in the velocity of the water and the formation of the secondary flow couldn’t be linked in one simulation. It could be due to incomplete input solution in the solution setup or other factors.

Thus, it is suggested that further simulations are to be done with more input solution setup; perhaps user defined functions (UDF) to be included to further refine the simulation results. More refined results can also be obtained through smaller mesh sizes and further refined meshing. With this, more detailed results would be achieved. Next, other suggested parameters should be assessed as well, such as the height of the gap (vertical length between the rotor and stator – in between the outer and inner gaps) and width of the outer and inner gaps.

REFERENCES

Al-Anazi, H. A., Al-Faifi, M. G., Tulbah, F., & Gillespie, J. (2006). Evaluation of Drag Reducing Agent (DRA) for Seawater Injection System: Lab and Field Cases. *SPE Asia Pacific Oil & Gas Conference and Exhibition*.

Al-Sarkhi, A., & Hanratty, T. J. (2001). Effect of drag-reducing polymers on annular gas-liquid flow in a horizontal pipe. *International Journal of Multiphase Flow*, *27*, 1151-1162.

Andersson, R., & Helmi, A. (2013). Computational fluid dynamics simulation of fluid particle fragmentation in turbulent flows. *Appl. Math. Modelling*. doi:10.1016/j.apm.2014.01.005

Barenghi, C. F., & Jones, C. A. (1989). Modulate Taylor-Couette Flow. *J. Fluid Mech.*, *208*, 127-160.

Chen, M. H., & Jaafar, A. J. (2012). Effects of Additives on the performance of drag reduction agents.

Chen, H. J., Kouba, G. E., Fouchi, M. S., Fu, B., & Rey, D. G. (2000). Field application of a drag reducing agent to increase gas production. *NACE International*.

Choi, H. J., Kim, C. A., J. S., & Jhon, M. S. (2000). An exponential decay function for polymer degradation in turbulent drag reduction. *Polymer Degradation and Stability*, *69*, 341-346.

Cinbiz, M. N., Tigh, R. S., Beskardes, I. G., Gymysderelioglu, M., & Colak, U. (2010). Computational fluid dynamics modeling of momentum transport in rotating wall perfused bioreactor for cartilage tissue engineering. *Journal of Biotechnology*, *150*, 389-395. doi:10.1016/j.jbiotec.2010.09.950

Dou, H., Khoo, B. C., & Yeo, K. S. (2008). Instability of Taylor-CouetteFlow between Concentric Rotating Cylinders. *Inter. J. of Thermal Science*, *47*(11), 1422-1435.

Ferrara, L., Cremonesi, M., Tregger, N., Frangi, A., & Shah, S. P. (2012). On the identification of rheological properties of cement suspensions: Rheometry, Computational Fluid Dynamics modeling andfield test measurements.

Cement and Concrete Research, *42*, 1134-1146. doi:10.1016/j.cemconres.2012.05.007

Gils, D. P. M. V., Huisman, S. G., Grossmann, S., Sun, C., & Lohse, D. (2012). Optimal Taylor-Couette Turbulence. *J. Fluid Mech*, *706*, 18-149.

Glatzel, T., Litterst, C., Cupelli, C., Lindermann, T., Moosmann, C., Niekrawietz, R., & Koltay, P. (2008). Computational fluid dynamics (CFD) software tools for microfluidic applications ? A case study. *Computers & Fluids*, *37*, 218-235. doi:10.1016/j.compfluid.2007.07.014

Henaut, I., Darbouret, M., Palermo, T., Glenat, P., & Hurtevent, C. (2009). Experimental methodology to evaluate DRA: Effect of water content and waxes on their efficiency. *SPE International Symposium on Oilfield Chemistry.*

Ito, Y. (2012). Challenges in unstructured mesh generation for practical and efficient computational fluid dynamics simulations. *Computers & Fluids*, *85*, 47-52. doi:10.1016/j.compfluid.2012.09.031

James, P. W., Jones, T. E., & Hughes, J. P. (2004). The determination of apparent viscosity using a wide gap, double concentric cylinder. *Journal of Non-Newtonian Fluid Mechanics*, *124*, 33-41. doi:10.1016/j.jnnfm.2004.07.010

Jeong, W., & Seong, J. (2013). Comparison of effects on technical variances of computationalfluid dynamics (CFD) software based onfinite element andfinite volume methods. *International Journal of Mechanical Sciences*, *78*, 19-26. doi:10.1016/j.ijmecsci.2013.10.017

Kadar, R., & Balan, C. (2012). Transient dynamics of the wavy regime in Taylor? Couette geometry. *European Journal of Mechanics B/Fluids*, *31*, 158-167. doi:10.1016/j.euromechflu.2011.07.003

Karami, H. R., & Mowla, D. (2013). A general model for predicting drag reduction in crude oil pipelines. *Journal of Petroleum Science and Engineering*, *111*, 78-86. doi:10.1016/j.petrol.2013.08.04

Lueptow, R. (2009). Taylor-Couette flow. Scholarpedia, 4(11):6389., revision #91854

Malik, M. M., Jeyakumar, M., Hamed, M. S., Walker, M. J., & Shankar, S. (2010). Rotational rheometry of liquid metal systems: Measurement geometry selection and flow curve analysis. *Journal of Non-Newtonian Fluid Mechanics*, *165*, 733-742. doi:10.1016/j.jnnfm.2010.03.009

Mowla, D., & Naderi, A. (2006). Experimental study of drag reduction by a polymeric additive in slug two-phase flow of crude oil and air in horizontal pipes. *Chemical Engineering Science*, *61*, 1549-1554. doi:10.1016/j.ces.2005.09.006

Nemri, M., Climent, E., Charton, S., Lanoe, J., & Ode, D. (2012). Experimental and numerical investigation on mixing and axial dispersion in Taylor?Couette flow patterns. *Chemical Engineering Research and Design*. doi:10.1016/j.cherd.2012.11.010

Pereira, A. S., & Soares, E. J. (2012). Polymer degradation of dilute solutions in turbulent drag reducing flows in a cylindrical double gap rheometer device. *Journal of Non-Newtonian Fluid Mechanics*, *179-180*, 9-22. doi:org/10.1016/j.jnnfm.2012.05.001

Pereira, A. S., Andrade, R. M., & Soares, E. J. (2013). Drag reduction induced by flexible and rigid molecules in a turbulent flow into a rotating cylindrical double gap device: Comparison between Poly (ethylene oxide), Polyacrylamide, and Xanthan Gum. *Journal of Non-Newtonian Fluid Mechanics*, *202*, 72-87. doi:10.1016/j.jnnfm.2013.09.008

Petera, J. & Nassehi, V. (1995). Use of the finite element modelling technique for the improvement of viscometry results obtained by cone-and-plate rheometers. *Journal of Non-Newtonian Fluid Mechanics*, *58*, 1-24.

Ratkovich, N., Horn, W., Helmus, F. P., Rosenberger, S., Naessens, W., Nopens, I., & Bentzen, T. R. (2013). Activated sludge rheology: A critical review on data collection and modelling. *Water Research*, *47*, 463-482. doi:10.1016/j.watres.2012.11.02

Strelnikova, S., & Michkova, D. (2013). Mathematical modeling of fluid motion in pipelines using drag reducing agents. *PSIG Annual Meeting*.

Wang, W., Kee, D. D., & Khismatullin, D. K. (2011). Numerical simulation of power law and yield stress fluid flows in double concentric cylinder with slotted rotor and vane geometries. *Journal of Non-Newtonian Fluid Mechanics*, *166*, 734-744. doi:10.1016/j.jnnfm.2011.04.001