Alice Mythen 170154415
*MEng Aerospace Engineering*

UNIVERSITY OF SHEFFIELD

# MEC320 COMPUTATIONAL FLUID DYNAMICS

# FLOW OVER A BACKWARD FACING STEP
# ASSIGNMENT 1: MATHEMATICAL MODEL

Spring 2020

## 1 ABSTRACT

The purpose of this report was to develop an understanding of the physics of a flow over a backwards facing step and select a turbulence model to solve the closure problem of the RANS equation in order to best model the flow of the problem. This was done by studying a paper written by Driver and Seegmiller on an experiment they conducted in 1985 and following this modules lectures. Assumptions were made about the flow to simplify the RANS equation used to model the flow and different turbulence models were compared in order to find the best one for this situation.

## TABLE OF CONTENTS

1. Abstract……..……...……………………………………………………………1
2. Physics of the Problem ..………..……………………………………………….1
   2.1 *Physics of the problem*………......………………………………………….……1
   2.2 *Flow assumptions*……………………………………………………………….2
3. Mathematical Model……………………………………………………………...3
4. Turbulence Behaviour……………………………………………………………5
5. References……………….……………………….………………………………7

## 2 PHYSICS OF THE PROBLEM
### 2.1 Physics of the problem

Driver and Seegmillers experiment consists of air entering a wind tunnel at atmospheric total pressure and temperature with an inlet velocity, $U_{in}$, of $44.2ms^{-1}$. As the fluid flows down the wind tunnel the boundary layers on the walls continue to build up resulting in the flow to be fully developed and turbulent before arriving at the step. In the experiment grit sandpaper was placed 1.0m upstream of the step to ensure fully developed turbulent flow. In order to ensure this in the ANSYS analysis, the upstream channel length will be extended.

As the flow passes over the step, it is subjected to a sudden increase of cross-sectional area [1] causing the boundary layer to separate, forming a separated free shear-layer. This free shear-layer bends towards and eventually impinges upon the tunnel wall [2], as shown in figure 2.1, at the point of reattachment. Between this point and the step, there is a recirculation region, which features low pressure and is enclosed by the separated free shear-layer. As the enclosed flow bends towards the reattachment point, the adverse pressure gradient at the step wall redirects part of the flow towards the step creating a primary recirculation region [3]. A secondary recirculation region is formed at the lower corner of the step (as shown in figure 2.1), which is counter-rotating with respect to the primary one.

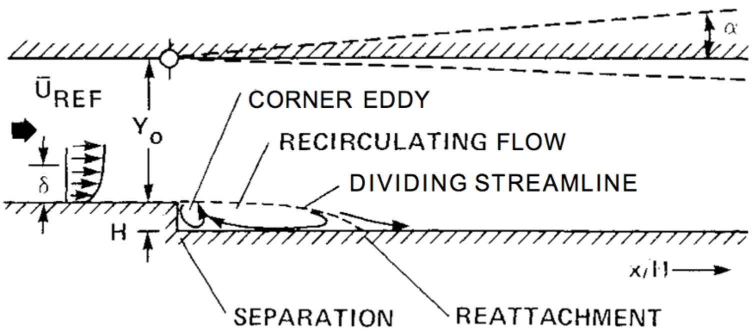


**Figure 2.1** *Driver and Seegmillers backward-facing step-flow experimental geometry*

## 2.2 Flow assumptions

In order to simplify the problem mathematical description, several physical assumptions have been identified and justified below.

1. **2D Flow**
   In the experiment conducted by Driver and Seegmiller, a wind tunnel with a large aspect ratio was used in order to minimise three-dimensional effects in the separated region [4]. Also as stated in the paper 'Two-dimensionality of the mean flow was validated'. This therefore means when modelling the flow mathematically any terms in the z-direction can be neglected.

2. **Incompressible**
   Flows are usually treated as being incompressible when the Mach number is less than 0.3 [5]. As the Mach number is 0.128 (less than 0.3) the flow can be treated as incompressible and therefore the fluids density is taken to be constant [6]. This justifies the neglection of any terms involving the change in density in the mathematical model.

3. **Fluid is air / constant viscosity**
   Although not specified, as the experiment is using a wind tunnel, we can assume the fluid being used is air. As air is considered to be a Newtonian fluid, which is defined as one with a constant viscosity, this simplification can be made. This also suggest there is zero velocity at all walls.

4. **Isothermal**
   As stated in the paper, the fluid entered the wind tunnel at atmospheric total pressure and temperature. Throughout the paper there is no mention of a change in temperature, therefore the assumption can be made that no heat exchange or work has been done, meaning the energy equation can be neglected.

5. **Turbulent flow**
   A Reynolds number of 5000 was chosen to ensure the boundary layer would be fully turbulent before passing over the step, as flow with Reynolds number greater than 4000 is classified as turbulent [7].

6. **No body forces**
   There are no magnetic or centrifugal forces mentioned in the paper, therefore can be assumed to not be present. As the fluid is air, the density is very small meaning the effect gravity has on it is so minimal it can be neglected. This means there are no body forces present and therefore this term can be removed from the mathematical model.

7. **Steady State**
   The inlet velocity is given with no mention of acceleration; therefore, it can be assumed the velocity is constant meaning the flow is steady state. This simplifies the mathematical model as the local acceleration term can be removed.

## 3 MATHEMATICAL MODEL

The Navier -Stokes Equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. They consist of a time-dependent continuity equation for conservation of mass, thee time-dependent conservation of momentum equations and a time-dependent conservation of energy equation [8]. Due to assumption 4 in section 4.2, the energy equation (equation 5.1) can be neglected in this problem as there is no heat exchange (Q) in the system or work done (W) on the fluid.

$$\Delta U = Q - W$$

**Equation 5.1**

In theory the Navier-Stokes equations could be solved for a given flow problem using methods from calculus. However, in practice they are too difficult to solve analytically, therefore in most cases, the equations have to be solved numerically, using high speed computers to perform Computational Fluid Dynamics (CFD). For turbulent flows, like in this problem, Navier-Stokes Equations can only be solved numerically for low Reynold flows in simple geometries. A solution to this problem was proposed by Osborne Reynolds to use time-averaged equations of motion for fluid flow – Reynolds Averaged Navier-Stokes equations (RANS). In these equations, the solution variables in the instantaneous Navier-Stokes equations are decomposed into the mean and fluctuating components [9].

In order to arrive at the RANS continuity equation, the Navier-Stokes continuity equation must be identified first. The continuity equation is also referred to as the conservation of mass, as the mass flow rate entering a fluid element is equal to the mass flow rate leaving it. This can be seen in equation 5.2 and rearranged into equation 5.3 (the continuity equation).

$$\left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z}\right] dxdydz = -\frac{\partial \rho}{\partial t} dxdydz$$

**Equation 5.2**

$$\frac{\partial \rho}{\partial t} + \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z}\right] = 0$$

**Equation 5.3**

To get from equation 5.3 to the RANS continuity equation, several steps of derivation must be followed. First the instantaneous density ($\rho$) and velocity (u, v, w) quantities are decomposed into their mean ($\bar{\rho}$, $\bar{u}$, $\bar{v}$, $\bar{w}$) and fluctuating ($\rho'$, $u'$, $v'$, $w'$) components e.g. $u = \bar{u} + u'$. Next the whole equation is time averaged and the rules of time averaging are followed to reduce the equation. Finally, the equation is put into tensor form and equation 5.4 is produced.

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad where\ i = 1, 2, 3$$

**Equation 5.4**

Following assumptions 1, 2 and 7 this equation can be simplified to equation 5.5. This is because; the assumption of 2D flow gets rid of any terms in the z-direction, the assumption of incompressible flow means there's no change in density with respect to x, y, z, and the assumption of steady state removes the term involving time.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

**Equation 5.5**

The RANS momentum equation is found using the same method as the continuity equation, starting with the Navier-Stoke momentum equations and arriving at equation 5.6

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial U_i}{\partial x_j} - \rho \overline{u_i' u_j'}\right) + \rho F_i \qquad where\ i, j = 1, 2, 3$$

**Equation 5.6**

[1] [2] [3] [4] [5] [6]

The left-hand side of this equation represents the mass times total acceleration of the fluid, while the right-hand side represents the total force acting on the fluid. Therefore, the total equation describes Newtons second law of motion. As term 1 of the equation represents the transient/unsteady term (acceleration with respect to time) it can be ignored due to the assumption that the flow is steady state. Term 6 can also be disregarded, as explained in section 4.2 regarded assumption 6.

The second term in the equation is the convection term (acceleration with respect to space coordinates) which describes the transport of fluid from one location to another. The third term is a source/sink term and represents the pressure gradient, which, provides energy for the flow. The transport of fluid due to shearing and volume deformation can be described by the fourth term; the diffusion term. And the fifth term represents the Reynolds stresses, which are the mean forces imposed on the mean flow by turbulent fluctuations [10]. This term introduced more unknowns into the equation therefore leading to the closure problem. In order to solve this problem a turbulence model must be selected to close the RANS equation. The selection process of this turbulence model is gone through in section 6.

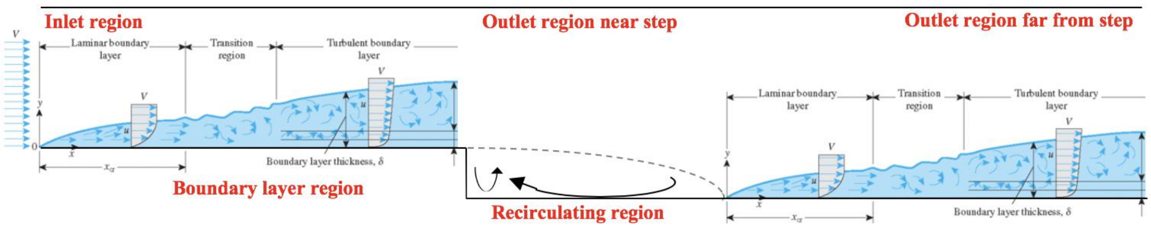


**Figure 3.1** *Geometrical domain of the problem split into regions*

By dividing the problem geometrical domain into regions, the mathematical terms most relevant to capture the flow physics in that region can be identified. As shown in figure 3.1, the geometrical domain has been split into the inlet region, boundary layer region, outlet region near the step, outlet region far from the step and recirculating region. Both the inlet region and outlet region far from the step contain mainly freestream flow, therefore the convection term is most relevant in these regions as it describes the transport for fluid from one location (the inlet) to another (the outlet).

Due to the sudden change in cross sectional area after the step, there is a substantial pressure gradient induced. Therefore, the term most relevant to the outlet region near the step is the third term; the source/sink. As boundary layers are caused by diffusion of momentum, the diffusion term is most relevant here. This term is also dominant in the recirculation region, as eddies are turbulent, and turbulence is caused by diffusion in the flow.

If, like in the Driver and Seegmiller experiment, the different levels of deflection of the upper wall were to be analysed, the diffusion term would become even more relevant as turbulence would increase. Also due to the greater increase in cross sectional area, a greater pressure gradient would occur therefore the sink/source term would also become more relevant.

## 4 TURBULENCE MODEL

As mentioned in section 3, the fifth term in the RANS momentum equation (5.6) represents the Reynolds stresses and leads to the closure problem. Therefore, a suitable turbulence model must be selected to close the RANS model and approximate/model this Reynolds stress. Among the hundreds of turbulence models available, there is no single one which will predict the physics of a problem best for all situations. Each model has different strengths and limitations and it is important to understand the physics of the flow of the specific problem in order to find which model will be best suited for that flow. It is also dependent on the computational resources available and justifiable for the problem.

The types of turbulence models commonly used to close the RANS model can be classified into Eddy Viscosity Models (EVM) and Reynold Stress Turbulence Models (RSM). The RSM is more elaborate than EVMs, as it solves transport a total of six equations for the

Reynold stresses and the dissipation rate [9]. Due to its complexity, RSM is more computationally intensive and difficult to converge than EVM, which use an eddy (or turbulent) viscosity, $\mu_T$, to model Reynolds stresses [11]. This is a reasonable hypothesis for simple turbulent shear flows and therefore EVM has been chosen over RSM for this problem.

Next, the type of EVM must be decided as each turbulence model is different in the way they calculate $\mu_T$. When determining what is a good model it is important to look at 5 main qualities; accuracy, simplicity, robustness (numerically), breadth of applications and the economics to run it. Some of the simplest and easiest models to implement are zero equation models. However, they are completely incapable of describing flows involving separation or recirculation, which are key parts of this flow problem, therefore are disregarded for this situation.

One-equation models are generally economical and accurate for attached wall-bounded slows and slows with mild separation and recirculation. The Spalart-Allmaras model has been shown to give good results for boundary layers subject to adverse pressure gradients, similar to what occurs in the flow problem however, it appears to produce relatively large errors for some free shear flows and is therefore ruled out.

The standard, RNG and realizable k-$\varepsilon$ models, are all two-equation models which solve transport equations for the turbulent kinetic energy (k) and its dissipation rate ($\varepsilon$). All models assume turbulent flow, and therefore is appropriate for this problem. While the realizable model provides the best performance of all the k-$\varepsilon$ model versions for several validations of separated flows, the $\varepsilon$ equation for all three models contains a term which cannot be calculated at the wall. Therefore, wall functions must be used.

The k-$\omega$ family of turbulence models avoid the use of wall functions as they can be integrated to the wall due to not containing terms which are undefined at the wall. Therefore, no near-wall treatment will be needed. The standard k-$\omega$ model, although not being as popular as the k-$\varepsilon$, does have several advantages, for example it's better for flows with adverse pressure gradients and separation, like the flow problem. Also, it is numerically very stable and tends to produce converged solutions more rapidly than the k-$\varepsilon$ models.

A combination of the k-$\omega$ model and the k-$\varepsilon$ model is the SST (Shear Stress Transport) model. It blends the robust and accurate formulation of the k-$\omega$ model in the near-wall region with the freestream independence of the k-$\varepsilon$ model in the far field by converting the k-$\varepsilon$ model into the k-$\omega$ formulation [9]. The SST is designed to avoid the freestream sensitivity of the standard k-$\omega$ model making it more accurate and reliable for a wider class of flows. This is why the SST model is one of the most widely used models for aerodynamic flow and why it has been selected as the turbulence model used to close the RANS equations for this flow problem.

## 5 REFERENCES


[1] COMSOL, *Turbulent Flow over a Backward-Facing Step*, Available at: https://uk.comsol.com/model/turbulent-flow-over-a-backward-facing-step-228 [Reviewed March 2020].

[2] Eaton, J. K. and Johnson, J. P., *"A Review of Research on Subsonic Turbulent Flow Reattachment"* AIAA Journal, 1981 [Reviewed March 2020]

[3] Alberto Felipe Ruis Vidales, *Air-Wake Flow Dynamics on a Simplified Frigate Shape*, MSc Thesis, December 14, 2016. [Reviewed March 2020]

[4] David M. Driver and H. Lee Seegmiller, *Features of a Reattaching Turbulent Shear Layer in Divergent Channel Flow*, AIAA Journal, 1985 [Reviewed March 2020]

[5] Wikipedia, *Compressible flow*, Available at: https://en.wikipedia.org/wiki/Compressible_flow [Reviewed March 2020]

[6] Chegg Study, *Incompressible and Compressible Flow*, Available at: https://www.chegg.com/homework-help/definitions/incompressible-and-compressible-flow-5 [Reviewed March 2020]

[7] Aerodynamics for students, *Classification of Flows, Laminar and Turbulent Flows*, Aerospace, Mechanical & Mechatronic Eng, University of Sydney, 2005. Available at: http://www-mdp.eng.cam.ac.uk/web/library/enginfo/aerothermal_dvd_only/aero/fprops/pipeflow/node8.html [Reviewed March 2020]

[8 National Aeronautics and Space Administration, *Navier-Stokes Equations*, Nancy Hall, 2015. Available at: https://www.grc.nasa.gov/www/k-12/airplane/nseqs.html [Reviewed March 2020]

[9] ANSYS, *ANSYS Fluent Theory Guide*, November 2013. [Reviewed March 2020]

[10] Meteorology Glossary, *Reynolds Stresses*, 20 February 2012. Available at: http://glossary.ametsoc.org/wiki/Reynolds_stresses [Reviewed March 2020]

[11] ANSYS, *Modelling Turbulent Flows – Introductory FLUENT Training*, December 2006. Available at: http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf [Reviewed March 2020]