University of Exeter coursework header sheet

ECMM102 Group Project (Meng) (A, TRM1+2 2016/7)

009412

1027443

630022828

Coursework: Group report and presentation
Submission Deadline: Thu 4th May 2017 12:00
Personal tutor: Dr Hao Qin

Marker name:	G_Tabor
Word count:	5957

First marker's comments	Indicative mark
Second marker's comments	Second mark
Moderator's comments	Agreed mark

G2 Report

An investigation into the Optimisation of a Draft Tube

Joseph Gowans

2017

4th year MEng Group Project

I certify that all material in this thesis that is not my own work has been identified and that no material has been included for which a degree has previously been conferred on me.

Signed...

College of Engineering, Mathematics, and Physical Sciences
University of Exeter

G2 Report

ECMM102

Title: An investigation into the Optimisation of a Draft Tube

Word count: 5957

Number of pages: 20

Date of submission: Wednesday, 03 May 2017

Student Name: Joseph Gowans

Programme: MEng Mechanical Engineering

Student number: 63002828

Candidate number: 009412

Supervisor: Dr. Gavin Tabor

Table of Contents

1 Group Objectives and Deliverables

The primary objective of the project was to optimise the geometry of a hydraulic draft tube. Optimisation of draft tubes translates to the maximisation of pressure recovery between draft tube inlet and outlet. This objective was achieved through the use of Computational Fluid Dynamics (CFD), experimentation and optimisation.

The function of the draft tube is to decrease pressure at the outlet of the turbine, resulting in a greater fluid head across the turbine and as such increasing the efficiency of the turbine. This is achieved by altering the geometry of the draft tube to reduce losses within the draft tube itself.

Installation of draft tubes incurs significant civil cost, which subsequently causes them to remain in operation for a significant lifespan. Therefore, minor gains in operational efficiency can have a large impact on the energy production, which lends itself well to a shape optimisation process. Shape optimisation using machine learning allows for the discovery of non-obvious solutions as it eliminates design bias based on intuition [1].

Flow regimes in draft tubes encounter flow physics such as separation, shear layers and vortex transport amongst others [2]. This makes numerical simulation of the draft tube challenging and computationally expensive, especially in the context of an optimisation problem. Traditionally, evolutionary algorithms have been used to optimise fluid dynamic problems, however to reduce computational expense, Bayesian optimisation is proposed, as it requires fewer objective function evaluations to reach an optimal solution. Much of the previous draft tube parameterisation relied on a series of cross sections to represent the geometry, however in this project a novel Catmull-Clark spline representation is proposed.

Accurate modelling of draft tube flow is of paramount importance within optimisation because of the potential for inaccurate solutions to mislead the optimiser, resulting in a false optimum. The accuracy of the simulation is reliant on how close the initial and boundary conditions are to that of the experiment and real world application as well as the quality of the mesh generated.

To be able to confirm the accuracy of CFD models there is a reliance on validating the simulations against experimental data. This required manufacturing of models able to match the geometry being simulated and was achieved through 3D printing which allowed for a range of abstract geometries to be manufactured quickly and cost effectively. Modelling and experimentation was performed on a 1:220 scale model, using real time data recording.

1.1 Group Objectives

Literature reviews along with visitation to a hydroelectric power plant were undertaken to help determine the group objectives that would be used throughout the project. They are as follows:

I. Calculate the most likely flow rates through the draft tube to give the optimisation most applicability to an industrial application.
II. Understand the flow within a draft tube to create realistic CFD models.
III. Collect experimental data of the flow through a draft tube for validation of CFD models.
IV. Develop robust and accurate meshing tools for a range of geometries.
V. Investigate the most applicable optimisation techniques for the use in this CFD optimisation case.
VI. Undertake an optimisation of the draft tube to determine a near optimum geometry.

1.2 Group Deliverables

I. Realisation of the flow conditions available within the experimental equipment, providing initial and boundary conditions for the CFD.
II. A method for manufacturing high curvature geometries able to tolerate the stresses imparted during experimentation.
III. Experimental flow measurements that will be utilised to validate the CFD models used in optimisation.
IV. A method to determine the most desirable turbulence model and initial conditions for the simulation based on mesh quality.
V. A range of methods for automatically generating high quality structured meshes and robust hybrid meshes capable of accurate flow simulation.
VI. Optimisation tools for optimisation of the draft tube.

2 Work Programme – Individual Work Packages

The project was categorised into three sub-groups; experimental, CFD and optimisation which were integrated together to achieve individual and interconnected objectives that contribute to the overall project aim. This enabled the CFD team to develop and then identify an appropriate automated modelling strategy, which the optimisation team utilised for function evaluation. Experimental data was obtained for validation of numerical models.

2.1 Experimental

2.1.1 Experimentation on Scale Models

This work package is focused on performing experiments on physical scale models of select draft tubes being simulated to provide the CFD team with useful validation data.

2.1.1.1 Objectives

I. Design an experiment to measure the pressure drop between select cross-sections including the inlet and outlet, along with flow rate and velocity profiles.
II. Explore and produce physical versions of the models used in CFD simulations which can withstand stresses during experimentation and enable flow measurements.
III. Measure and provide an estimate for the model surface roughness.

2.1.1.2 Deliverables

I. Four scale models which can be used for experimentation.
II. An experimental setup which can be used to measure pressure drop between select cross-sections of the model, including inlet and outlet, along with inlet flow rate and velocity profile at the outlet.
III. A range of the surface roughness values for the scale models.
IV. Pressure drop and pressure recovery values between the inlet and outlet for all the scale models.

2.1.2 Design and Prototyping of a Custom Data Collection System

The focus of this work package is to provide a system for measuring and recording the performance of the scale model draft tubes in the fluids lab. The expectation for this work package is that the data collection system will be electronic.

2.1.2.1 Objectives

I. Explore methods of recording data in real time using appropriate sensors based on accuracy and budget.
II. Develop a system for taking the physical output signals from the various sensors chosen and converting them to human interpretable data.
III. Employ the realised system to collect data from experiments on model draft tubes.

2.1.2.2 Deliverables

I. A system comprising of sensors and appropriate signal conditioning which stores and displays data.
II. An estimate of the error within the data collection system itself.

2.2 CFD

2.2.1 Investigation of Turbulence Model Performance of Automated Meshes Generated using cfMesh

The objective of this work package is to provide an automatic meshing tool, to facilitate the optimisation of draft tube geometry, valid for a range of turbulence models. The meshing tool was required to be robust to high curvature geometry changes and able to provide elements with high quality, whilst minimising computational expense. cfMesh is an open source mesh generation library, extending the OpenFOAM framework, which is known to be highly robust. cfMesh provided the basis for meshing within this work package, forming a novel boundary layer meshing technique. In order to validate the computational modelling realised within the project a mesh sensitivity and turbulence model study was achieved.

2.2.1.1 Objectives

I. Develop a technique for automatic mesh generation.
II. Provide a work flow that can be implemented into the optimisation.
III. Understand the complex internal flow to form the basis of turbulence modelling.
IV. Validate the numerical modelling.

2.2.1.2 Deliverables

I. Geometry modification through boundary layer meshing techniques using cfMesh.
II. Turbulence model study and mesh refinement study.
III. Validation of simulations for different model geometries.
IV. Development of swirling inlet conditions for use in future work.

2.2.2 Development of an Automatic Numerical Draft Tube Modelling Technique

This individual work package aimed to develop a mesh distortion method compatible for automating design optimisation. The automatic meshing technique will contribute to a diverse design process of the CFD sub-group. Investigation into the sensitivity of inlet velocity conditions to the simulation of draft tube flow is also included within this individual work package.

2.2.2.1 Objectives

I. Develop an alternative structured mesh approach suitable for automated meshing.
II. Investigate the sensitivity of computational draft tube models to a variety of empirical velocity profiles of turbulent flow prescribed at the draft tube inlet.
III. Establish a verified inlet velocity profile correlating to the experimental set up which can be applied to the draft tube models.
IV. Determine appropriate limitation boundaries to the optimisation search space.

2.2.2.2 Deliverables

I. Meshing methodology of combined, fixed and variable meshed geometries compatible for automating design optimisation.
II. Sensitivity study on the effects of prescribed inlet velocity conditions to the pressure recovery of a draft tube.
III. A conservative bounding box based on numerical model limitations.

2.2.3 Structured and Unstructured Automatic Meshing using Pointwise

The overall objective of this work package is to create a meshing tool to allow the various geometries to be meshed robustly and automatically with high quality elements within an optimiser. This was achieved using Python along with the recent developments in scripting within the commercial meshing software Pointwise. The controlling functions within elliptic partial differential equations were studied to improve the quality of meshes generated with the transfinite interpolation method. A framework for combining two separate meshing tools was also developed, allowing a less robust, high quality meshing tool to be combined with a more robust tool with less focus on mesh quality.

2.2.3.1 Objectives

I. Develop an understanding of programming within the object-oriented language Python.
II. Create a structured meshing tool with a focus on high quality elements.
III. Interface and automate meshing tools within an optimiser.
IV. Develop methods to prevent poor quality meshes misleading the optimiser by producing inaccurate results.

2.2.3.2 Deliverables

I. Structured meshing tool which produces high quality meshes for non-extreme curvature geometries.

II. Robust unstructured meshing tool.
III. A method for monitoring mesh quality within the optimiser and penalising low quality meshes.
IV. Framework for combining high quality and robust meshing tools.

2.3 Optimisation

2.3.1 Implementation of Efficient Global Optimisation for Optimising a CFD Modelled Draft Tube

The aim of this work package is to implement a machine learning tool that works well in automated design optimisation with CFD. The focus was Efficient Global Optimisation (EGO), a Bayesian optimisation algorithm using a DACE model as the surrogate model and maximum expected improvement as the acquisition function [3], [4]. Built on previous work of optimising Pitz Daily CFD tutorial with genetic algorithms (GA) [5], the EGO optimiser programmed was first implemented on Pitz Daily case before modified to optimise the draft tube, and its performance on Pitz Daily was compared with a GA.

2.3.1.1 Objectives

I. Implement EGO on simple test functions.
II. Develop an interface between the optimiser and CFD to allow automation.
III. Investigate the performance of EGO in optimising the CFD problem over GAs.
IV. Optimise the hydraulic draft tube using the EGO with the CFD meshes prepared by the CFD team.

2.3.1.2 Deliverables

I. A program that runs the initial samples of the EGO and GA in parallel.
II. A program that converts a CFD case into arguments required by EGO.
III. An analysis of the performance of EGO on Pitz Daily compared with GA.
IV. Optimised draft tube designs with one to four control points.

2.3.2 Surface Representation and Parameterisation for Optimisation

To optimise the draft tube, or any problem, the problem must be parameterised in such a manner that the optimiser, in this case the EGO algorithm, may have a set of parameters to alter, allowing exploration of the problem space leading to the detection of the optimal solution. The objective of this work package is to develop an approach to representing the geometry of the draft tube, in such a way that the geometry can be optimised for an objective, by parameterising the mesh. One such way this could be achieved is by having a low-resolution simplistic control mesh which is subsequently refined to the surface mesh used in the CFD. For this project it is proposed to evaluate and work with the Catmull-Clark splines.

2.3.2.1 Objectives

I. Develop an understanding of the fundamentals of digital polygon mesh representation.
II. Understand the operation and theory behind the Catmull-Clark division algorithm.
III. Use the knowledge from the above to implement an approach to creating a surface mesh from a set of parameters that can be varied from the optimiser.

2.3.2.2 Deliverables

I. A Python implementation of a suitable data structure for storing polygon (both quads and triangles) meshes, allowing both manipulation and traversal.
II. Using said implementation, a utility for mesh refinement, using a low-resolution mesh to generate a high-resolution surface mesh (as an STL) to pass to OpenFOAM during an optimisation.
III. A Python tool for creating a fluids problem mesh (e.g. draft tube) using common parameterised components (e.g. inlets and outlets). This generates a polygon mesh which can be refined using the above.

2.3.3 Reducing the Number of CFD Calls During Optimisation

The main objective of this work package is focussed on developing a novel method that reduces the number of function calls to the CFD and reduces the time taken for the optimisation to take place. Following the implementation of the methods a simple CFD case was investigated to demonstrate the validity of the method and implementation. Further to this, the adjoint solver was compared to another implementation that computes the adjoint solution simultaneously with the CFD solution to ensure validity and correct implementation of the adjoint solver.

2.3.3.1 Objectives

I. Develop an understanding of how to increase efficiency in optimisation.
II. Implement an adjoint solver that calculates the adjoint solution from a converged CFD solution.
III. Extract the surface sensitivities from the CFD and adjoint solution.
IV. Calculate the gradient of the objective function for use in Bayesian methods.
V. Validation of the adjoint solver, surface sensitivities and gradient calculation.

2.3.3.2 Deliverables

I. An implementation of an adjoint solver that is applicable to a range of geometries.
II. An implementation of a surface sensitivity calculator which calculates the sensitivity for a given surface for a range of geometries.
III. A program that calculates the convex hull of the spline.
IV. A program that calculates the influence of altering spline control points on the surface points.
V. An implementation that extracts the gradient of the objective function with respect to the spline control points.
VI. A workflow that is ready to be integrated with the Bayesian optimiser to reduce the number of function calls that the optimiser makes.

3 Methodology

In the following section, there will be a brief overview of how the work packages interact – a visual summary is displayed in figure 1. The project was based on optimising the draft tube geometry from the Turbine-99 case study since it had been extensively covered in literature [6]. Models of this draft tube and developments of its geometry were created at 1:220 scale, limited by the print space of the 3D printer.

Only the diffuser of the draft tube was optimised as this reduced the complexity of the optimisation, which was prioritised. This can also be said for the inlet conditions, which were simplified and did not contain swirl. The inlet and outlet cross sections were fixed, to mimic industrial constraints. This also reduced complexity.

The single objective function is to maximize the average pressure recovery factor C_p which is calculated using equation 1.

$$C_p = \frac{p_{out} - p_{in}}{\frac{1}{2}\rho U_{in}^2} \tag{1}$$

Where p_{out}, p_{in} are the outlet and inlet pressure respectively, ρ is the fluid density and U_{in} is the inlet velocity.

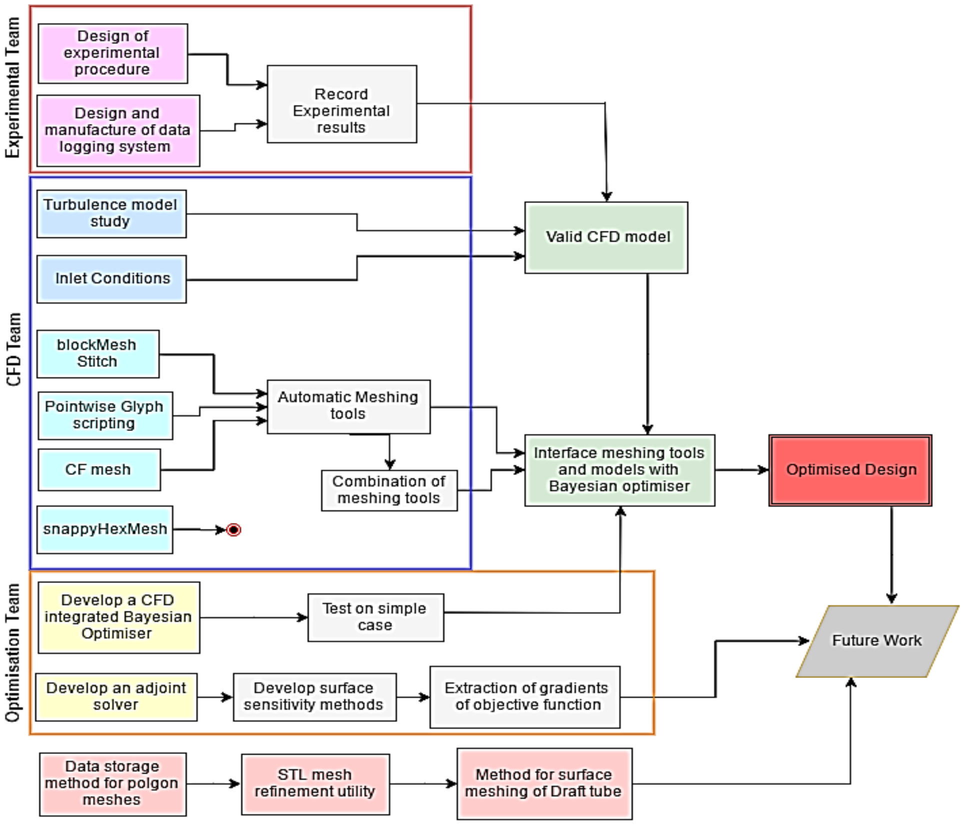

Figure 1: Flow diagram representing the project integration.

3.1 Experimental Models

The models were 3D printed using the Formlabs 2 stereolithographic printer and made from the associated standard clear resin. The scale models were formed from two individually printed sections that connected via a flange, and consisted of an elbow and diffuser, the latter being exchanged to recreate the different models. Other features included pressure tappings within the model, incorporated at specific cross-sections, notably the inlet and outlet. The model was positioned using two manufactured stands.

3.2 Experiment Instruments and Measurements

The models were submerged and connected to the Armfield F1-10 Hydraulics Bench. Pressure drop was measured between the inlet and outlet cross-sections of the model using a manometer board. Flow rate data at the inlet and flow velocity data at the model outlet were provided using a turbine flow meter and flow sensing probe respectively. The data from these were recorded by the data collection system. However due to time constraints the velocity measurements were not performed. Model surface roughness was also investigated using a non-contact scanner (Talyscan 150) to provide further information about the model properties which may be incorporated into the CFD simulations at a later date.

3.3 Computational Fluid Dynamics

3.3.1.1 Meshing Techniques

The study by [5] influenced the initial direction of the project. The snappyHexMesh utility was used within their optimisation of the Pitz Daily case to provide an automatic meshing technique. snappyHexMesh was unable to generate a constant boundary layer mesh which, as a result, reduced the mesh quality below the project requirements, therefore alternative meshing techniques, Pointwise, cfMesh and a combined meshing technique stitching the blockMesh and Pointwise meshes were proposed.

Structured meshing techniques used in Pointwise and blockMesh are advantageous in that they are less computationally expensive and can reduce numerical errors if good quality cells are present. cfMesh was also opted for because it is an OpenFOAM extension that produces a robust hybrid mesh, using an inside-out cut cell meshing technique to generate boundary layer meshes automatically. Pointwise is a commercial meshing software that uses a bottom-up meshing technique that was scripted to generate meshes automatically. The OpenFOAM utility blockMesh was used in conjunction with a Pointwise mesh of the draft tube heel to automatically generate meshes.

3.3.1.2 Mesh Quality

In general, mesh quality can impact CFD solution accuracy, therefore it is paramount that the mesh quality for all models within the optimiser remains below an acceptable level. As is alluded to in [7], inaccurate solutions can mislead the optimiser, possibly producing falsely high values, causing the optimiser to exploit regions of the design space that are non-optimal. Using tools within Pointwise the mesh quality for each geometry can be reported and thresholds can be put in place to inhibit inaccurate solutions from poor quality meshes influencing the optimiser.

3.3.1.3 Alternative Models for Validation Purposes

Flow characteristics of two models were used for validation, one being the original draft tube geometry and a selected geometry, Model 2, that represented an exaggerated exploration of the search space. The test on Model 2 aimed to provide validation of a ‘worst-case’ scenario, that involved high-curvature geometry, known to be challenging to mesh.

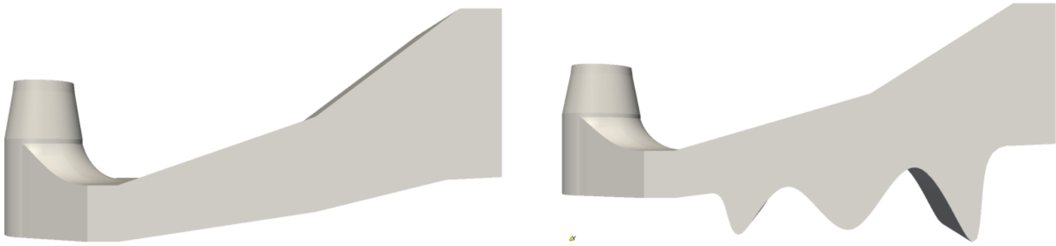

Figure 2: Experimental draft tube models. Original geometry (left) and Model 2 (right).

3.3.1.4 Inlet Conditions

An overriding theme in the literature documenting the computational modelling of hydraulic draft tubes is the sensitivity of results to prescribed inlet velocity boundary conditions [8]. A CFD predicted inlet velocity profile is proposed to be superimposed onto the computational model and the results proposed to be compared against experimental data.

3.3.1.5 Turbulence Modelling

In order to validate the standard and Model 2 geometries generated for the project, a study on turbulence models is proposed to find which model provides the best result of pressure recovery when compared to experiment. The models proposed are the k-ε, RNG k-ε, k-ω, k-ω SST turbulence models.

3.4 Optimisation

3.4.1 Efficient Global Optimisation

The optimiser is composed of a surrogate model which approximates the objective function and an acquisition function which helps balance the exploration of highly uncertain areas and exploitation of current optimum area [3], [4]. The draft tube design is proposed to be altered by changing the bottom spline generated after performing Catmull-Clark subdivision [9] five times. The surrogate model is constructed using Gaussian process module after initial evaluations. The $(11 \times no.of\ dimensions - 1)$ initial samples [4], each represents a design which was generated using a Latin hypercube sampling method. A further 100 function evaluations were run, with each sequential search point being selected by maximising the expected improvement of the model, with the use of a genetic algorithm. The surrogate model is then refitted after each evaluation.

3.4.2 *Surface Representation and Parameterisation for Optimisation*

The optimisation component of this project was based upon the previously undertaken Pitz Daily case where a genetic optimiser generated a two-dimensional spline which was then used to cut predefined geometry. A draft tube is more complex requiring a more comprehensive means of geometry modification. To achieve this, the structure was broken down into separate sections, with a small number of parameters, connected to form a low-resolution control mesh which was then refined.

3.4.3 Reducing the Number of CFD Calls During Optimisation

The Bayesian method is clearly the most appropriate optimiser. There is a need to reduce the total number of function calls the optimiser makes to reduce the total time taken. It is shown in [10] that Bayesian optimiser with the introduction of gradient information is beneficial.

Various methods are possible but the method chosen was the adjoint method as this only requires one CFD function call, which has already been undertaken, and one adjoint solution call calculated using equation 1.

$$\boldsymbol{\lambda}^T\left(\frac{\partial \boldsymbol{R}}{\partial \boldsymbol{Q}}\right) = -\left(\frac{\partial J}{\partial \boldsymbol{Q}}\right) \quad (1), \qquad \frac{\partial L}{\partial \beta} = -A\upsilon(\boldsymbol{n}\cdot\nabla)\boldsymbol{u}_t\cdot(\boldsymbol{n}\cdot\nabla)\boldsymbol{v}_t \quad (2), \qquad \frac{\partial L}{\partial \boldsymbol{x}} = \frac{\partial L}{\partial \boldsymbol{\beta}}\frac{\partial \boldsymbol{\beta}}{\partial \boldsymbol{x}} \quad (3)$$

Where $\boldsymbol{\lambda} = (\boldsymbol{u}, q)^T$ are the adjoint variables, $\boldsymbol{R}$ are the Navier-Stokes equations, $\boldsymbol{Q} = (\boldsymbol{v}, p)$ are the converged flow variables, and J is the objective function. To be able to calculate the gradient surface sensitivities are required to be calculated achieved utilising equation 2. Where $L = J + \int_\Omega (\boldsymbol{u}, q)\boldsymbol{R}d\Omega$ is the augmented objective function, which is valid as $\boldsymbol{R} = \boldsymbol{0}$ for a converged CFD solution, over the domain Ω, β is a location on the surface and $\boldsymbol{n}$ and t are the surface normal and tangential directions respectively. From this, the gradient of the objective function can be calculated using equation 3 for $\boldsymbol{x}$, the spline control points.

4 Data Analysis

4.1 Validation of Computational Models

Due to the size of the scaled model and the flow rate generated during the experiments, the boundary layer thickness was a large percentage of the internal flow area, and as such producing meshes with correct grid requirements, whilst capturing the flow physics was challenging.

A comparison between experimental and CFD was only achieved with cfMesh as the experimental data was not provided until the last week of the project, therefore it was not feasible to do such a study on the other meshing techniques. This may have impacts on the accuracy of the CFD results which later may impact the accuracy of the optimiser as poor CFD results can lead to inaccurate optimisation and a false optimum for the problem. Also, it should be noted that the experiment values for C_p in table 1 do not take into account any flow rate errors and assume a constant flow rate of $0.5Ls^{-1}$.

In order to validate the CFD simulations, a number of turbulence models, mesh densities and inlet conditions were investigated. Table 1 shows cfMesh and Pointwise have a good agreement with the experimental mean for Model 1. cfMesh was run using a k-ω model following the results of a turbulence study. Pointwise and blockMesh were run using k-ε due to the delayed experimental results. Possible reasons for the lack of agreement of the blockMesh simulation solution was due to non-orthogonal cells within the mesh that OpenFOAM is known to be particularly sensitive to. The flow solution is shown in figure 3.

Analysis of Model 2 simulations highlight the robustness of the cfMesh tool and the accuracy of the k-ω model. The inaccuracy in Pointwise and blockMesh show the inability of structured meshes to deal with high curvature geometries. Therefore, for high control number splines where more extreme geometries are more likely, a combination of structured and unstructured meshing was used. The results from the inlet condition study showed the sensitivity of the solution to these inlet profiles, however the study was inconclusive as to which inlet profile best represents the experimental conditions. The inlet sensitivity study was able to show that the ability to optimise the geometry is unaffected.

Table 1: Showing the results from the different mesh generation methods against experimental data.

	Experimental Maximum	Experimental Minimum	Experimental Mean	blockMesh	Pointwise	cfMesh
Model 1 C_p	0.646	0.570	0.608	0.665	0.617	0.612
Model 2 C_p	0.599	0.493	0.546	0.642	0.448	0.563

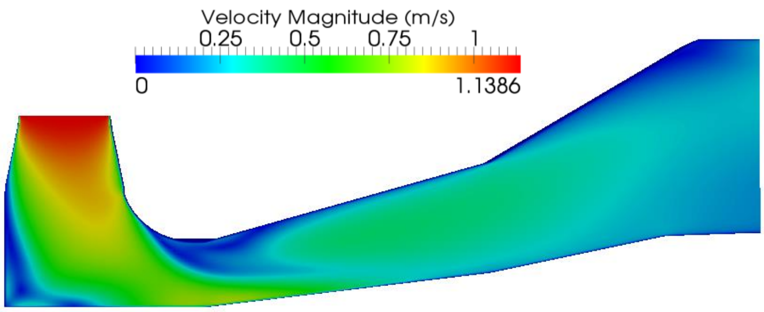

Figure 3: Velocity distribution of original geometry, Model 1, down the centreline.

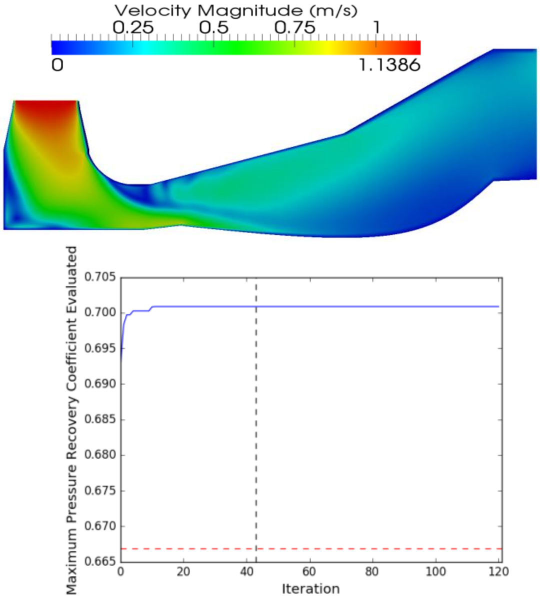

Figure 4: Top: Velocity distribution of an optimised geometry with one control point. Bottom: Optimiser performance.

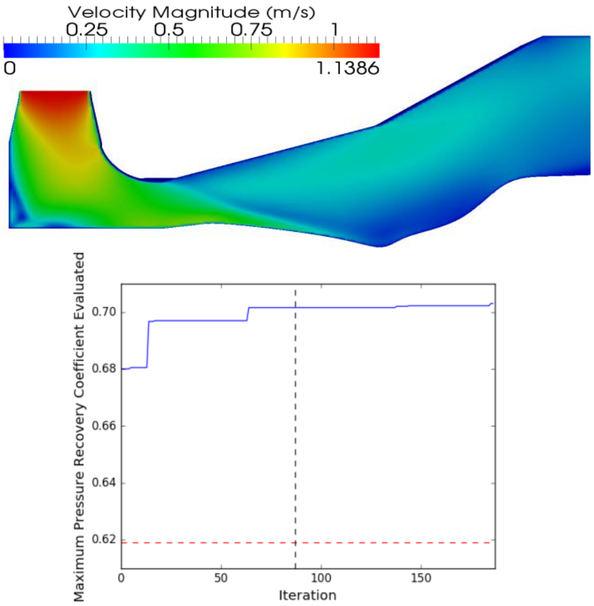

Figure 5: Top: Velocity distribution of an optimised geometry with four control points. Bottom: Optimiser performance.

4.2 Optimised Results

Figures 4 and 5 show velocity distributions and optimiser performance for control points of one and four. A range of the number of control points were also investigated, however these are the extremes. For lower number of control points, a structured meshing strategy is appropriate. However, as the number of control points increased, more extreme geometries could be produced, a structured mesh is not achievable. A hybrid meshing technique and a combination of structured and unstructured meshing techniques were adopted.

There is a consistent attempt to expand the diffuser section of the draft tube, therefore reducing the velocity of the flow resulting in an increase in pressure and a larger difference in pressure between the inlet and outlet. This is logical as an optimum draft tube design exhibits a negative pressure gradient moving counter to the direction of flow.

When comparing the plots of optimiser performance, figures 4 and 5 show that with a lower number of control points the optimiser approaches upon a near optimal solution within a very small number of iterations. Correlation appears to be good between the number of design variables and the improved pressure recovery, this is expected as the design space is increased [7]. It is known that as the number of design variables increases the number of iterations required to reach convergence also increases. In this case the number of evaluations beyond the initial sample remained constant, therefore for higher numbers of control points one cannot conclude that the optimisation has converged. This is evident in figure 5 where small changes in pressure recovery were found late in the optimisation.

The four control point optimum solution was tested experimentally. The simulated results showed an over-prediction of the improvement in pressure recovery when compared to experimental test. The optimiser predicted a 14.5% improvement; the experimental result only experienced a 7.74% improvement. Despite the potential modelling inaccuracy, the optimisation process was able to generate an improved and possibly optimum design. This is compared to [11] who optimised the entire geometry resulting an improvement of 10.1%.

4.3 Surface Representation and Parameterisation for Optimisation

Though not used in the collection of the main results for this project, a utility has been created to process a low-resolution control mesh which can be efficiently optimised, into a high-resolution surface, by using the Catmull-Clark algorithm to recursively subdivide a mesh to approximate a smooth surface [9]. This was extended to allow for variably sharp edges, which remain after subdivision allowing for simple transitions between square and smooth sections in the geometry, figure 6 [12].

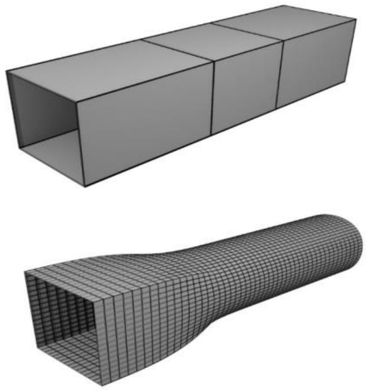

Figure 6: Pipe geometries meshed using the Catmull-Clark subdivision implementation showing the presence of sharp corners.

4.4 Reducing the Number of CFD Calls During Optimisation

The solution of the adjoint solver was compared to one that has been pre-written. This pre-written code solves the adjoint flow simultaneously as the CFD solution is computed. The solvers gave similar results for the solution of the adjoint equations and lead to the conclusion that the implementation of the solver is correct. It is difficult to validate the results as the adjoint variables have no physical meaning along with no experimental data to compare against.

The surface sensitivities do show physical meaning, which represent which direction the surface should move. The results generated for a flow over a bump in a channel show that the surface should be moved to remove the bump and from this it appears that the surface sensitivity has been implemented correctly and gives reasonable results.

There is no experimental data to compare the generated results of the gradient information to. The gradient information is compared to other methods for generating the gradient information from the surface sensitivities. The gradient information generated are similar to other methods.

The implementation of these methods took a little while longer than anticipated and as such mitigation procedures were undertaken. The majority of time was taken in understanding the mathematics there was little time to be able to integrate it into the Bayesian optimiser and was not completed.

4.5 Project Limitations

The error in the experimental data made it difficult to determine the true value of pressure recovery. Therefore, conclusions have to be based on assumptions and errors in the experimental results. This could have had a knock-on effect on the final optimum. However, the models were able to capture the general fluid trends with the geometry changes, which has been proven to be sufficient to produce a near-optimum geometry [13].

It can be debated whether changing meshing techniques within the optimiser is fully valid as it has been shown that the solution is sensitive to the meshing tool. Therefore, work has to be done to reduce this sensitivity before it can be concluded as valid.

In addition, only the diffuser section of the draft tube was optimised, where it is hypothesised that the largest pressure recovery is experienced in the inlet cone and elbow [14]. The diffuser optimisation was also limited by the constrained outlet cross-section meaning that the full potential improvement may have not been realised.

5 Conclusion, Deduction and Final Design

Three separated automatic meshing techniques were developed to produce high quality and robust meshes for a range of geometries. An efficient global optimiser was integrated with CFD to evaluate the objective function. The optimiser performed well for simple test cases as well as the more complex draft tube case. The optimiser over predicted the pressure recovery by around 7%. Two-dimensional Catmull-Clark spline parameterisation was utilised within optimiser which is a novel technique within draft tube optimisation. Having only optimised the diffuser section, with the outlet cross-section constrained, an improvement of approximately 7% was seen, which is very encouraging.

A framework was also outlined for future work with a more advanced parameterisation technique enabling three dimensional surfaces using Catmull-Clark method. An adjoint solver with surface sensitivity and gradient calculator was also developed to be used with future studies of this project. The gradient information generated can be used to potentially reduce the run time of the optimiser by decreasing the number of CFD evaluations.

5.1 Future Work

Using the three-dimensional parameterisation, the entire draft tube geometry could be optimised. This would require more design variables and hence more iterations are expected for the optimiser to converge, therefore the gradient method would be highly beneficial in reducing the computational expense. With these considerations in mind, a greater improvement would be expected.

6 Budget, Sustainability and Risk Assessment

6.1 Budget

The budget total (£680) and was subdivided into a mechanical budget (£265), an electronics budget (£215) and an emergency budget (£200). The spending of mechanical budget went towards producing the 3D models and the materials used in the experimental setup and the electronics budget was spent on the components required to produce the data collection system. The emergency budget was a select amount of money which could cover the costs of expensive items such as models or sensors in the case where they were unusable due to damage.

6.2 Sustainability

The experimental design was influenced by the need for sustainable measures. The models were designed for repeated used, enabling potential use in further experiments. The modular feature of the models allows for exchangeable elbow or diffuser sections, reducing 3D printing resources and spending to create new models.

Use of the hydraulics bench was limited to 10 minutes of operation per test run to reduce electricity consumption. Consumption of water has also been minimised since the bench is a closed system.

The implications of our work add to the improvement of draft tube design, leading to improvements in energy extraction from water flow and therefore greater electricity generation, making hydropower a more competitive option to alternative harmful CO_2 emitting sources.

Table 2: Risk Assessment Table, L=likelihood, C= consequence, I=importance

Risk Item	Cause	Effect	L (/10)	C (/10)	I (/10)	Action to Minimise Risk
Project Risks						
Illness	Various	Work package delay.	8	5	7	Account for potential minor illness in the project schedule, in the case of major illness apply for mitigation.
Budget overspending	Budget not up to date, poor budget plan, not sticking to budget plan.	Limits experiment progress.	2	9	10	Keep record of invoices, audit budget, keep an emergency budget to mitigate for risk of damaged items, refer to budget before spending.
Failure of model, experiment instruments or electronics	Instrument instructions not followed, model mishandling/ design, electronics exposure to water.	Delays to experimental results.	5	9	10	Follow instrument instructions. Design model with safety factor, hold with both hands, place away from edges. Ensure protective box is splash proof before circuit is situated inside. Situate electronics as far a distance as reasonable from water which could come in contact.
Loss or corruption of data	Misplaced storage device, computer software issues	Work package delay.	4	8	10	Save multiple backups on more than one storage device or file hosting service.
Health and Safety Risks						
Ingestion of contaminants	Water from hydraulics bench, 3D printer resin.	Work package delay.	3	9	9	Thoroughly wash hands after contact with water or printer resin, wear gloves when handling under-cured model.
Computer display	Excessive viewing of screens.	Eye strain.	9	2	3	Take regular breaks from screen viewing.
Injury from workshop tools	Loose clothing, member of body not a safe distance from cutting/ rotating implement. Lack of PPE	Minor to severe bodily damage.	4	10	10	Wear appropriate PPE, maintain safe distance when operating machinery, ask for assistance from technician if unsure of safe operation of machinery.

7 References

[1] J. McNabb, et al, “CFD Based Draft Tube Hydraulic Design Optimization,” in *IOP Conference Series: Earth and Environmental Science*, 2014.

[2] G. Vunjak-Novakovic, et al, “Flow regimes and liquid mixing in a draft tube gas-liquid-solid fluidized bed,” *Chemical Engineering Science,* vol. 47, no. 13-14, pp. 3451-3458, 1992.

[3] B. Shahriari, et al, “Taking the Human Out of the Loop: A Review of Bayesian Optimization,” *Proceedings of the IEEE,* vol. 104, no. 1, pp. 148 - 175, January 2016.

[4] D. R. Jones, M. Schonlau and W. J. Welch, “Efficient Global Optimization of Expensive Black-Box Functions,” *Journal of Global Optimization,* vol. 13, no. 4, pp. 455-492, 1998.

[5] S. Daniels, et al, “A review of shape distortion methods available in the OpenFOAM framework for automated design optimisation,” Exeter.

[6] M. J. Cervantes, U. Andersson, and H. M. Lövgren, “Turbine-99 unsteady simulations – Validation,” *IOP Conference Series: Earth and Environmental Science,* vol. 12, no. 1, pp. 1-10, 210.

[7] J. Vassberg and A. Jameson, “Influence of Shape Parameterization on Aerodynamic Shape Optimization,” Brussels, 2014.

[8] S. Galvan, F. Guibault and M. Reggio, “Optimisation of the inlet velocity profile of Turbine-99,” in *Proceedings of the third IAHR/EROCFTAC workshop on draft tube flows, Turbine-99 III,* Porjus, 2005.

[9] E. Catmull and J. Clark, “Recursively generated B-spline surfaces on arbitrary topological meshes,” *Computer-Aided Design,* vol. 10, no. 6, pp. 350-355, 1978.

[10] C. E. Rasmussen and C. K. Williams, Gaussian Precesses for Machine Learning, Massachusetts: MIT Press, 2006.

[11] R. Eisinger, “Automatic Shape Optimisation of Hydro Turbine Components Based on CFD,” in *Institute of Fluid Mechanics and Hydraulic Machinery,* Stuttgart, Germany, 2001.

[12] H. Hoppe, et al, “Piecewise smooth surface reconstruction,” in *Proceedings of the 21st annual conference on Computer graphics and interactive techniques,* 1994.

[13] B. Marjavaara, “Parameterisation and flow design optimisation of hydraulic turbine draft tubes,” *Licentiate dissertation,* vol. 32, no. 5, 2004.

[14] M. Cervantes and F. Engstrom, “Eddy Viscosity Turbulence Models and Steady Draft Tube Simulations,” in *Proceedings of the third IAHR/EROCFTAC workshop on draft tube flow,* Porjus, 2005.