Balancing the flow – Optimisation of the Malad IPS Screen Chamber using CFD

John Chesterton
Environmental Engineering Services
Mott MacDonald Ltd
United Kingdom

Sarah Jones
Water UK Central
Mott MacDonald Ltd
United Kingdom

ABSTRACT

The Mumbai Sewage Disposal Project will provide a much needed, healthier and improved environment for the people of Mumbai by increasing the quality and reliability of wastewater collection, treatment and disposal, whilst minimizing the impact of wastewater on the natural environment of the city.

As part of this project, major tunnelling works and the construction and refurbishment of new pumping stations is planned. This will include a new influent pumping station in the Mumbai suburb of Malad incorporating a screen chamber and deep wet well which is currently undergoing tender design.

The screen chamber is 25 m deep formed as a circular shaft containing four screens in separate channels. The chamber is fed by two sewer tunnels, 3.2 m and 2.2 m in diameter collectively delivering peak flows of 18.3 m^3/s which occur during the monsoon season. Concern was raised during design that the tunnel configuration and size of the circular chamber would result in unbalanced flow across the four screens.

A CFD model was built using Flow3D and confirmed that this was indeed the case. The CFD model was then used to establish if the balance of flow required could be met with baffling or if realignment of the tunnels or a costly increase in the size of the chamber was required. This paper discusses the modelling process, the arrangements evaluated and the final configuration of the chamber and screens.

CFD was perceived to add value at this early design stage as it gave greater confidence to the designers that the final arrangement determined would be adequate, avoiding the need for conservatism in the design and reducing the cost.

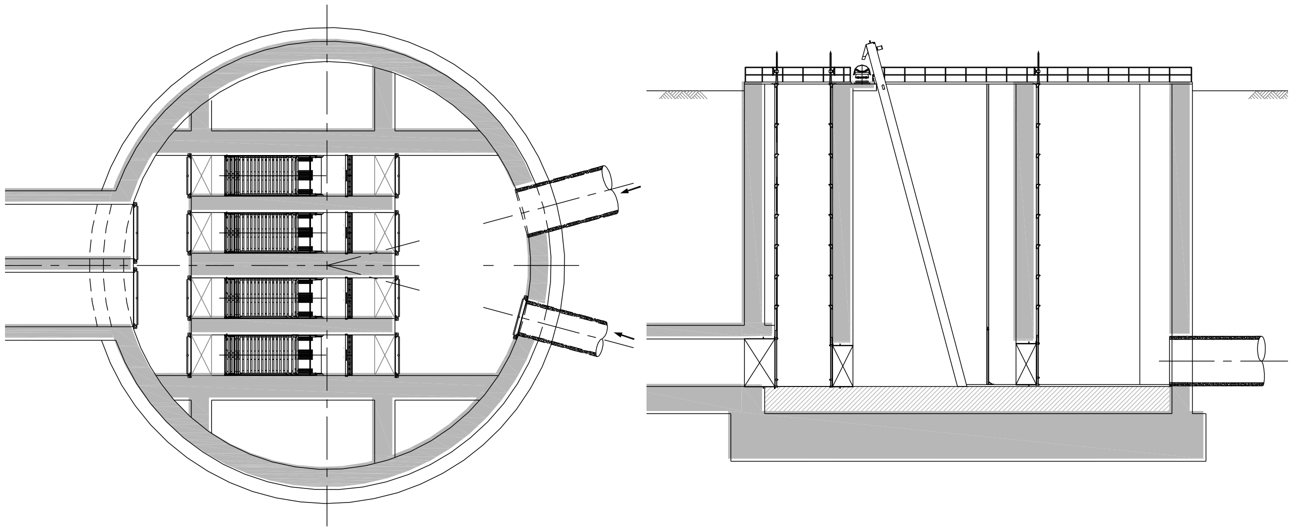
1. INTRODUCTION

As part of the Mumbai Sewage Disposal project, major tunnelling works and the construction and refurbishment of new pumping stations are planned. This will include a new influent pumping station in the Mumbai suburb of Malad, incorporating a screen chamber and deep wet well which, at the time of writing, is currently undergoing tender design.

1.1. Malad IPS Screen Chamber

The screen chamber is 25 m deep and formed as a 30 m circular shaft. As seen in Figure 1, the chamber contains four automatically cleaned bar screens in separate channels 3 m wide by 15 m long. The chamber will be fed by two sewer tunnels, an existing 3.2 m diameter and proposed 2.2 m diameter tunnel, collectively delivering peak flows of 18.3 m^3/s which occur during the monsoon season. Flow then passes through a dual 3.5 m x 4 m square connecting tunnel to the pumping station wet well.

The long screen channels are needed to contain the inclined screens and cleaning equipment along with upstream and downstream closure penstocks. Currently, 10 m is provided between the incoming tunnels and channels. 5 m has been provided between the channels and connection tunnels.



Plan *Section*

Figure 1: Malad IPS Screen Chamber

1.2. Design risk

Concern was raised during tender design that the tunnel configuration and size of the circular chamber would result in unbalanced flow across the four screens. As both tunnels focus flows toward the center of the shaft they were expected to over-load the central screen channels. Higher than allowable velocities through the central screens could make cleaning difficult and result in blockage or washout of solids. Lower velocities in adjacent screens would allow grit to deposit in these areas.

The solutions proposed to remediate any flow imbalance were the provision of flow control baffles and, if required, an increased screen shaft diameter. Hence, the key question was whether a 30 m diameter shaft was sufficient to contain the flow control measures required to provide balanced flows to the screen channels. While the addition of flow control measures was not expected to have a material effect on the tender price, increasing the shaft diameter would be a significant cost increase for the client.

Given the anticipated flow balance and solutions proposed, the risk that the existing design was insufficient could be mitigated in a number of ways; through the implementation of a more conservative design such as a pre-emptive increase in the shaft diameter or through more thorough investigation of the hydraulic conditions though model studies. In this case, numerical modelling studies were preferred so timescales and costs could be kept to a minimum. While physical model studies have traditionally answered these questions, they can be expensive to procure and may involve lengthy lead times. Mott MacDonald instead utilized their in house numerical modelling capability to evaluate the problem.

This paper discusses the modelling process, the arrangements evaluated and the final recommended configuration of the chamber and screens.

1.3. Model System

The numerical modelling was undertaken using the FLOW-3D software package developed by Flow Science Inc. FLOW-3D solves the Reynolds Averaged Navier-Stokes (RANS) system of equations in three dimensions to simulate the flow of fluid. Solver and interface software versions **10.1.0.27 win64 2012** and **10.1.0.22** were used respectively.

Flow3D uses a rectangular structured grid coupled with a fractional area/volume (FAVOR) method for modelling the fluid volume and capturing complex geometries. Multiple mesh blocks are utilised when a change in the resolution of the mesh is required.

For this model study a one-fluid solver was used which employs a proprietary volume-of-fluid (VOF) method to track the free surface. This approach is one of the defining features of the software and provides three important functions for free surface flow: accurate location and orientation of free surfaces within computational cells, accurate tracking of free surface motion through cells, and an accurate boundary condition applied at the free surface interface. The volume of fluid within the model is monitored to ensure that the calculations were converging correctly and to sufficient accuracy.

1.4. Scope and study criteria

The scope of the modelling was defined in terms of both the number of flow simulations and criteria to measure against. The model was to be run for maximum, minimum and an intermediate flow with both inflows operating and with only a single inflow operating as summarised in Table 1.1.

The model geometry would include;

1. modelling of a 30 m diameter shaft with no baffling
2. modelling of a 30 m diameter shaft with a baffling arrangement determined during modelling

Table 1.1: Modelling Flows and Levels

	3.2 m Tunnel		2.2 m Tunnel		Wet Well
Flow Case	Flow (m^3/s)	Flow Depth (m)	Flow (m^3/s)	Flow Depth (m)	Level (m)
Pass Forward Flow (PFF)	13.681	2.144	4.606	1.386	9.82
Peak Dry Weather Flow (PDWF)	8.727	1.607	2.928	1.047	9.22
Average Dry Weather Flow (ADWF)	6.713	1.382	2.257	0.905	8.62
Pass Forward Flow (PFF)	13.681	2.144	0.0	-	9.52
Peak Dry Weather Flow (PDWF)	8.727	1.607	0.0	-	9.19
Average Dry Weather Flow (ADWF)	6.713	1.382	0.0	-	8.32

The performance criteria which need to be met in order to ensure that the screens are not damaged or preferentially blocked are outlined in Table 1.2 below.

Table 1.2: Performance Criteria

Parameter	Criteria
Flow	+/- 33% of mean flow
Mean Velocity	< 1.2 m/s

For the performance criteria, mean flow is defined as the total flow divided by the number of screen channels (4). Mean Velocity is the average velocity across the screen area over the simulation time.

2. MODEL STEUP

In order to accurately model the flows within the screen chamber, the modelled area included the chamber, the upstream tunnels and a portion of the downstream pumping station wet well. The geometries were developed using Autodesk's AutoCAD and were exported as STL files for import to FLOW-3D.

The following figure (Figure 2.1) shows model geometry and the definition of flow domains using blocks of computational mesh. The computational mesh in FLOW-3D is orthogonal and blocks of mesh are applied to the model to define flow area. Multiple connected mesh blocks are utilised to more efficiently define flow areas and to provide changes in mesh resolution within the model. The maximum ratio used between adjacent mesh blocks when changing mesh resolution has been 2.

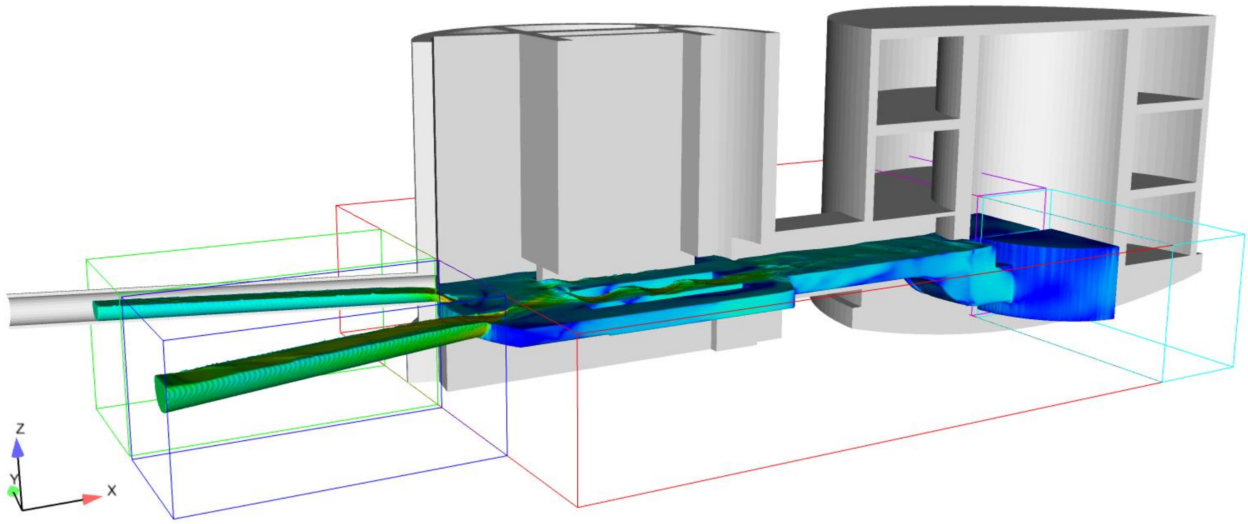


Figure 2.1: Model Setup showing geometry sectioned, computational mesh blocks and flows

2.1. Upstream extents

Flow enters the area of interest via the existing and proposed sewer tunnels, 3.2 m and 2.2 m in diameter respectively. According to the calculations undertaken during tender design, these tunnels will be flowing at normal depth and partially full some distance upstream of the screen chamber. Depending on the water levels within the chamber the depth of water in the pipe was expected to be gradually increasing or decreasing as it approached the chamber.

To capture the extents of this gradually varied profile, the model was extended 26 m upstream of the tunnel connections to the shaft. The flows were then applied to the model via a flow boundary condition that included flow rate, direction and fluid level in the upstream tunnel.

2.2. Downstream extents

To capture the effect of the chamber and connection tunnels to the pumping station wet well, the effects of the wet well were modelled in part. The model was extended to the centre of the wet well and flow was allowed to exit over the full depth at this boundary. The boundary was defined as a fixed velocity boundary, which was calibrated to give the appropriate level at the flow rate simulated. This allowed the volume of fluid within the model to stabilise at the required inflow and downstream levels.

2.3. Model representation of screens

The screens were represented within the model as inclined 2D baffles with a defined porosity and head loss relationship. Given the screen dimensions, the screen porosity for entry into the model was taken to be 0.8065.The head loss across the screens was estimated as a function of the screen geometry using Kirschmer's formula[1] as follows.

[1] Novak P., Moffat A.I.B., Nalluri C., Narayanan R., Hydraulic Structures - Fourth Edition (2007)

Head Loss, $\Delta h = \beta \left(\frac{s}{b}\right)\left(\frac{s}{b}\right)^{4/3} \sin\delta \frac{v^2}{2g}$ (1)

Where:	Bar Width	(s)	0.012	m
	Bar Length	(L)	0.060	m
	Bar Spacing	(b)	0.050	m
	Screen Angle	(δ)	75.00	deg
	Form Factor	(β)	2.42	-
	Gravity	(g)	9.81	m/s^2
	Fluid Density	(ρ)	1000	kg/m^3

Flow3D calculates head loss across a porous baffle using the following formula. This was converted to a head loss and the linear and quadratic constants were modified so as to achieve agreement with the Kirschmer formula.

Pressure loss across baffle in Pascals:

$$\Delta p = \rho \cdot (\mathrm{KBAF1} \cdot u + 0.5 \cdot \mathrm{KBAF2} \cdot u\,|u|) \quad (2)$$

Where:	Screen or baffle internal velocity	(u)	(calculated)	m/s
	F3D Constant1	(KBAF1)	0.002	m/s
	F3D Constant2	(KBAF2)	0.225	m/s^2

Linear and quadratic head loss coefficients selected gave a head loss across the screen of 26 mm at the maximum criterion velocity of 1.2 m/s.

Partially blocked screens were not included as clean screens were considered to give the most conservative results with respect to flow imbalance.

2.4. Model Physics and Sensitivity Analysis

The software includes many optional models that add to or modify the basic Navier-Stokes equations. Additional models that are used frequently in hydraulics include options for describing the effects of turbulence, surface tension, air entrainment and cavitation etc. The screen chamber was modelled using a single fluid with free surface interface tracking.

All modelling undertaken was transient, allowing the performance to be examined over time. This was important to understanding the behaviour of unsteady flows such as those that may occur around the baffling arrangements.

In order to ensure that the correct input parameters were chosen, sensitivity analyses were conducted on the computational mesh resolution, model roughness and turbulent mixing length. As the model was intended to evaluate flow balance across the four screen channels, this flow distribution was evaluated in the sensitivity studies as defined by the Symmetry Coefficient derived below.

For sensitivity and for evaluation of baffle options, only the pass forward flow was investigated. Following a selection of the most suitable baffle option, the model was run for all three cases, comparing the existing geometry with that recommended.

2.4.1. Symmetry Coefficient

To evaluate the flow distribution between the channels a symmetry coefficient was used to represent flow balance.

The symmetry of flow through the four screen channels was calculated using the following relationship to express the symmetry of the flow as a number from 0 to 1. 1 being 100% symmetry or equal flow through all channels, and 0 being 0% symmetry or all flow through a single bay.

$$\text{Symmetry Coefficient , S } = 1-\left(\sum\left|\frac{x_i}{\sum x_i}-\frac{1}{n}\right|\times\frac{n}{2n-2}\right) \tag{3}$$

Where: x = Flow in bay
i = Number of bays

The coefficient is similar to an evaluation of the difference between the maximum and minimum flows and is independent of the total flow. This coefficient was used as the key metric against which sensitivity analysis was done and was also useful when comparing final options and runs.

2.4.2. Mesh Resolution

The flow area in a CFD model is subdivided into smaller control volumes with a computational mesh. Flow calculations are done at the resolution of this mesh so it was important that the elements are small enough to capture the hydraulics that may be occurring. Elements that are too large mean that the geometry is not effectively captured and critical hydraulics may be averaged out. Models with elements that are very small may take an excessive amount of time to solve with little increase in accuracy.

The mesh element sizes investigated were 400 mm, 300 mm, 200 mm, 150 mm and 100 mm. Below a 200 mm resolution, the model was not found to be significantly sensitive to mesh size hence the final mesh resolution selected for the model was 100 mm, which was considered to adequately capture both the geometry and hydraulics.

2.4.3. Model Roughness

Model roughness is input as a sand equivalent roughness height as for pipe friction calculations when calculating a Darcy-Weisbach friction factor. A roughness height of 0.003 m was used as may be expected for reasonably well formed concrete. The model symmetry coefficient was not found to be influenced significantly by roughness heights between the range of 0.001 m and 0.01 m with a 1000% increase in roughness resulting in an 8 % increase in the symmetry coefficient. Hence, the 0.003 m roughness height was maintained.

2.4.4. Turbulence Model

The sensitivity of the model to the turbulence model and to turbulence parameters such as turbulent mixing length was investigated in order to ensure that the turbulence in the model had been accurately modelled. The model showed some sensitivity to these parameters and the turbulence model selected was the RNG turbulence model with a dynamically calculated maximum turbulent mixing length.

2.5. Simulations

Following set-up and sensitivity, the base case and optioneering was undertaken followed by final runs for both tunnels operating and a single tunnel only (3.2 m diameter). The final run list is outlined below in Table 2.1. The simulation names will be used in the following sections.

Table 2.1: Simulation Run List

OPTIONEERING				
Simulation ID	**Geometry**	**Flow**	**Number of Tunnels**	**Simulation Name**
S01	Base	PFF	Both (3.2 m and 2.2 m Dia.)	S01.Base.PFF
S02	Option1	PFF	Both (3.2 m and 2.2 m Dia.)	S02.Option1.PFF
S03	Option2	PFF	Both (3.2 m and 2.2 m Dia.)	S03.Option2.PFF
S04	Option3	PFF	Both (3.2 m and 2.2 m Dia.)	S04.Option3.PFF
S05	Option4	PFF	Both (3.2 m and 2.2 m Dia.)	S05.Option4.PFF
S06	Option5	PFF	Both (3.2 m and 2.2 m Dia.)	S06.Option5.PFF
S07	Option6	PFF	Both (3.2 m and 2.2 m Dia.)	S07.Option6.PFF
FINAL RUNS				
S09 (As per S01)	Base	PFF	Both (3.2 m and 2.2 m Dia.)	S08.Base.PFF
S09	Base	PDWF	Both (3.2 m and 2.2 m Dia.)	S09.Base.PDWF
S10	Base	ADWF	Both (3.2 m and 2.2 m Dia.)	S10.Base.ADWF
S12 (As per S04)	Option3	PFF	Both (3.2 m and 2.2 m Dia.)	S11.Option3.PFF
S12	Option3	PDWF	Both (3.2 m and 2.2 m Dia.)	S12.Option3.PDWF
S13	Option3	ADWF	Both (3.2 m and 2.2 m Dia.)	S13.Option3.ADWF

3. OPTIONEERING

A number of geometries were developed for optioneering. These are listed and described in this section as Options 1 through 6.

Simulations S01 through S07 were run for the screen chamber without baffles and for the six baffle arrangements operating under the Pass Forward Flow (PFF) as detailed below. The following sections and illustrate and discuss the flow performance for each option.

3.1. Base Case

3.1.1. S01.Base.PFF - Base (un-baffled) case Results

In the existing design, the area upstream of the screen channels was shown to be ineffective at distributing the flow evenly. Flows from the 3.2 m tunnel entered the screen chamber as a jet that was maintained through to the screen channels, impacting to the left of the dividing wall between Channel 2 and 3. Channel 3 took the majority of the flow followed by Channel 2. The 2.2 m tunnel plunged into the chamber and spread before entering the channels but was predominantly directed toward Channel 2. Channel 1 took a smaller proportion of the flow while Channel 4 was completely excluded and showed reverse flow.

The performance of the screen chamber without flow controls or increased sizing confirmed that concerns raised during design were real and that some modifications were required.

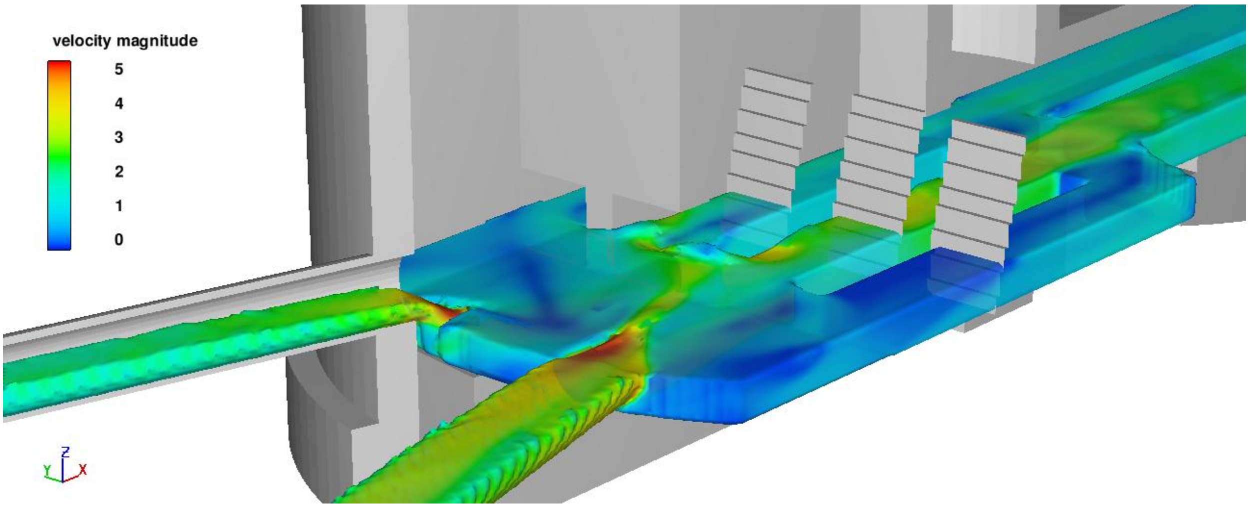


Figure 3.1: S01.Base.PFF - 3D View – Surface velocity (m/s)

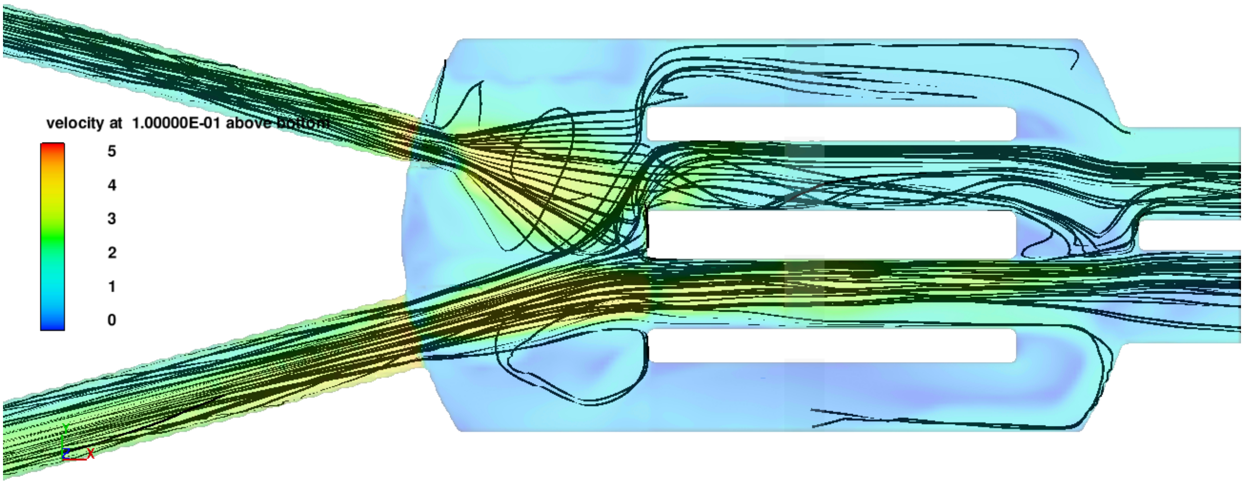


Figure 3.2: S01.Base.PFF – Plan View – Streamlines

3.2. Baffled Option 1 – Double aligned baffles

3.2.1. Geometry

The first baffle option selected was a single baffle 'blocking' each tunnel as shown in Figure 3.3 below. Baffles were selected to be equal in width to the diameter of the tunnel, located one diameter from the exit and aligned perpendicular to it. The baffles were 1m thick and extended to tunnel mid height.

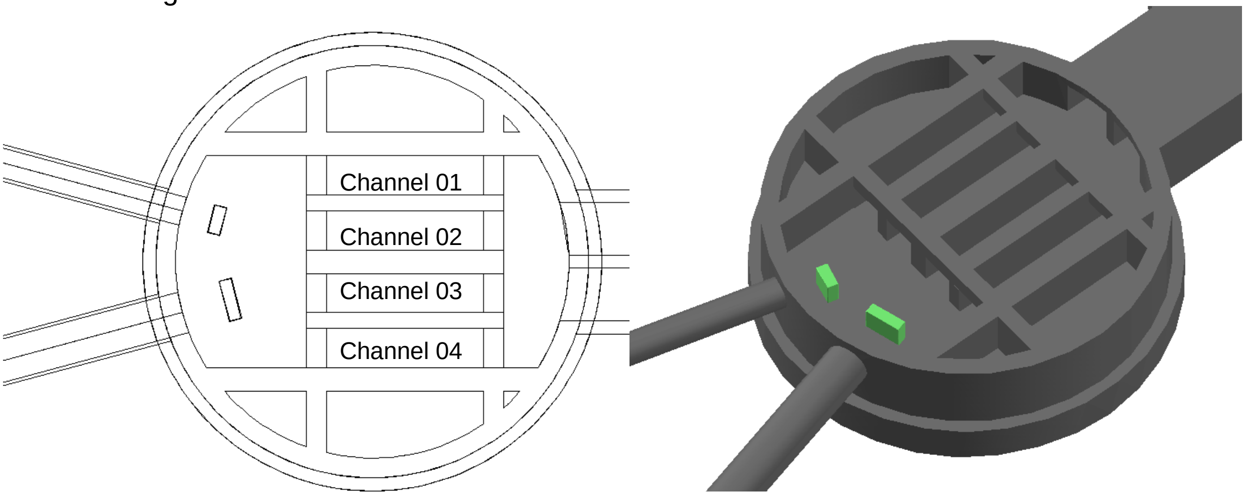


Figure 3.3: Baffled Option 1 – Double aligned baffles

3.2.2. S02.Option1.PFF - Results

The inclusion of two baffles was trialled with the aim of breaking up the jets formed by the tunnels and reducing the flow into Channel 3. These were only partially successful and resulted in almost complete blockage of Channel 3 with the jet simply split and continuing to Channels 2 and 4. The majority of the flow from the 2.2 m tunnel was pushed toward Channel 1. The baffles were overtopped but these flows were small.

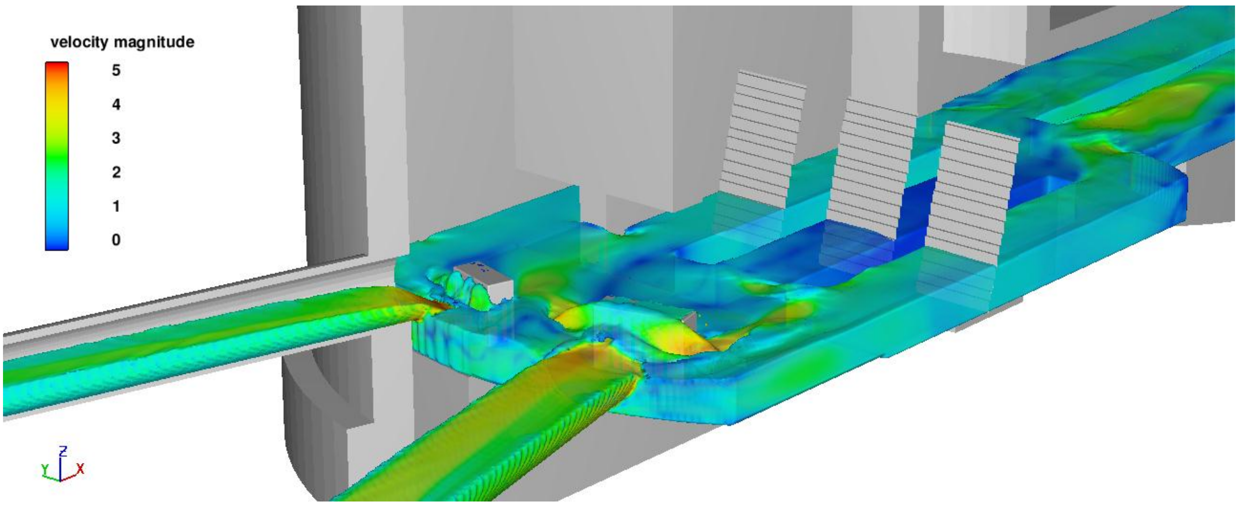


Figure 3.4: S02.Option1.PFF – 3D View - Surface Velocity (m/s)

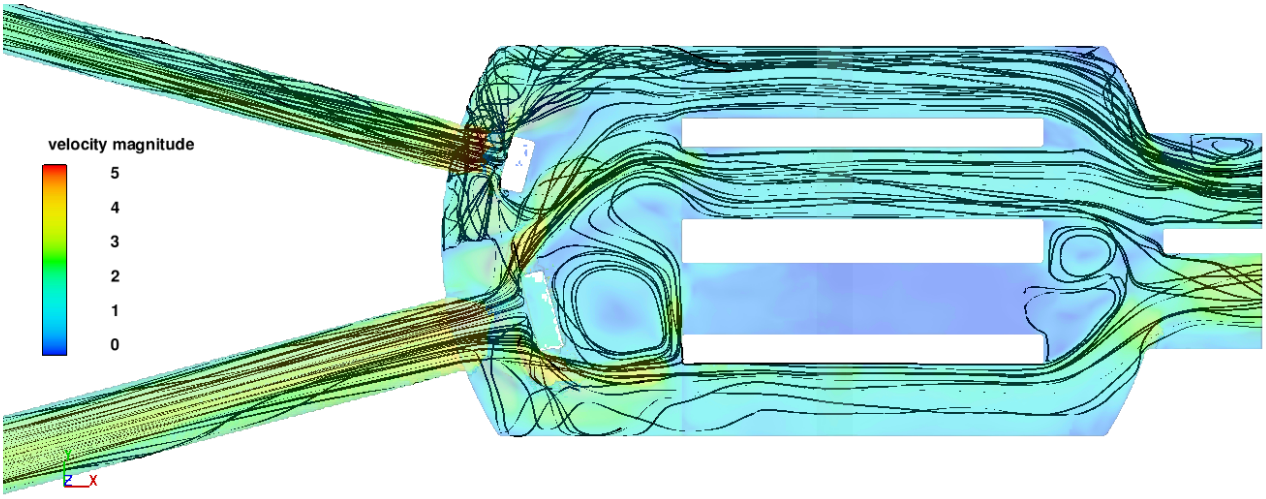


Figure 3.5: S02.Option1.PFF – Plan View – Streamlines

3.3. Baffled Option 2 – Baffle Row

3.3.1. Geometry

A single row of baffles was placed midway between the tunnel outlets and screen channel penstocks as shown in Figure 3.6 below. The baffles were all of equal size: 2 m width by 2.5 m high by 1 m thick. The spacing provided between baffles was 1 m.

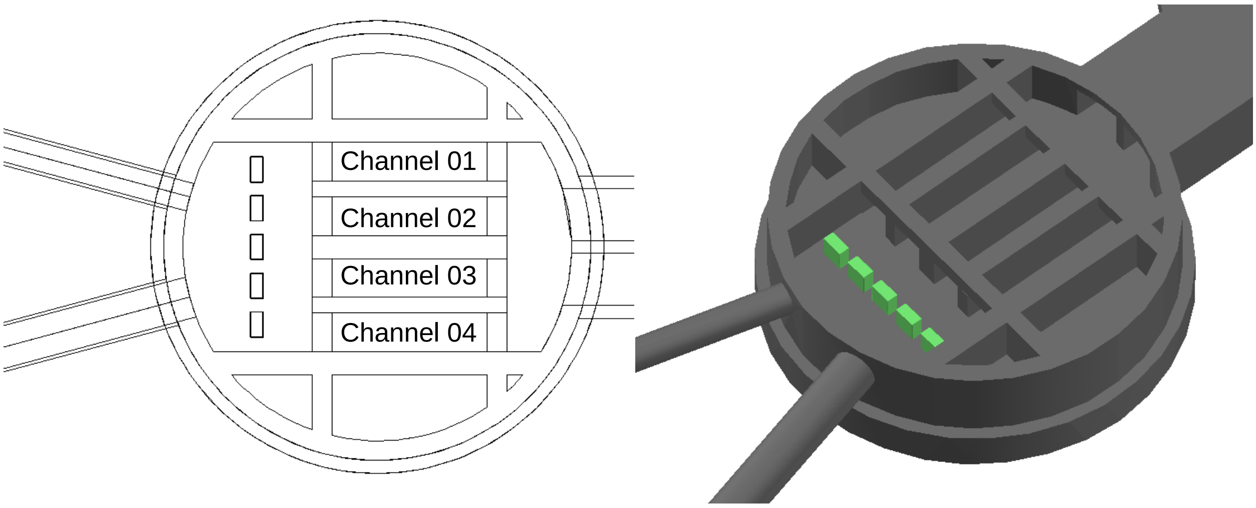


Figure 3.6: Baffled Option 2 – Baffle Row

3.3.2. S03.Option2.PFF - Results

A baffle row was implemented using equal baffle sizes across the chamber. This showed an improvement in the flow distribution in the channels although some bias was seen toward Channels 3 and 4. Flows observed to be overtopping the baffles were small and did not significantly affect the channel distribution.

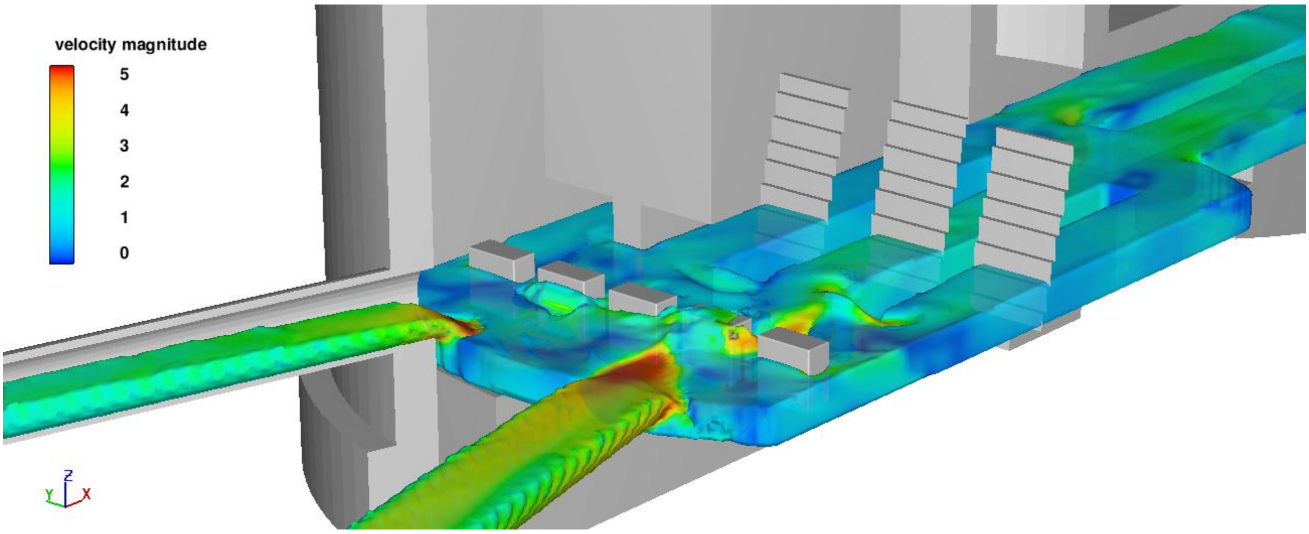


Figure 3.7: S03.Option2.PFF – 3D View - Surface Velocity (m/s)

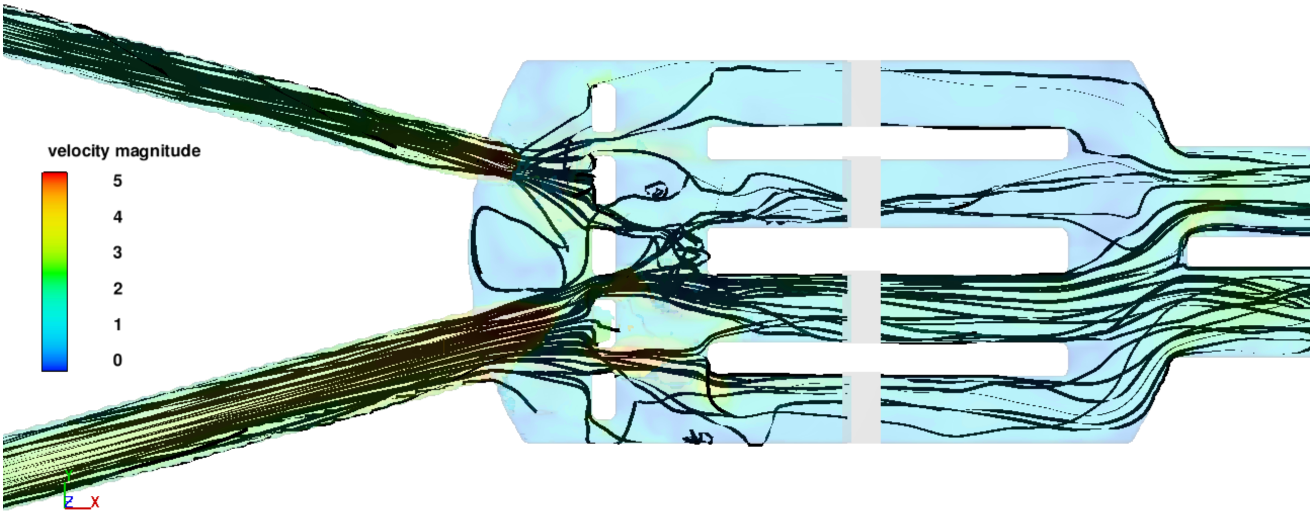


Figure 3.8: S03.Option2.PFF – Plan View – Streamlines

3.4. Baffled Option 3 – Baffle Row Adjusted

3.4.1. Geometry

Following baffle options 1 and 2, the single row of baffles was modified, increasing the width of the baffle opposite the 3.2 m tunnel symmetrically by 0.5 m as shown in Figure 3.9. This option was developed in order to reduce biased flows occurring in channel 3 and 4. The location of other baffles was not changed which reduced the spacing between the enlarged baffle and its adjacent baffles to 0.75 m.

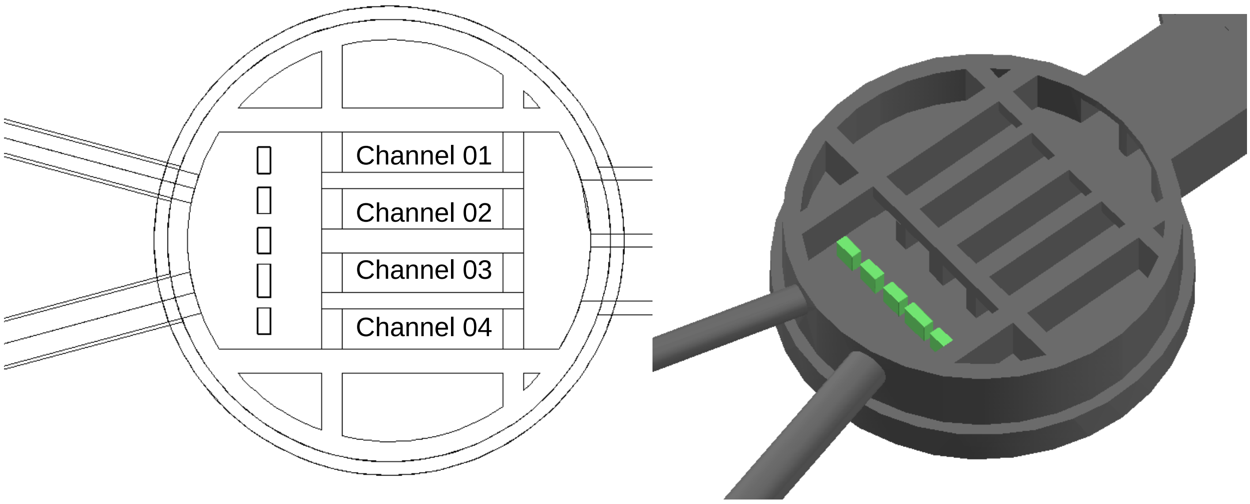


Figure 3.9: Baffled Option 3 – Baffle Row Adjusted

3.4.2. S04.Option3.PFF – Results

The baffle row trialled in Option 2 was modified to prevent the bias seen in the previous run. The jet emerging from the 3.2 m dia. tunnel impacts the 4th baffle (from left to right) and this was widened to reduce the gaps on either side and reduce the excess flow into Channels 3 and 4.

These modifications were successful and showed a more balanced flow with Channel 3 at 20.3% below mean flow and Channel 4 only 16.5% above.

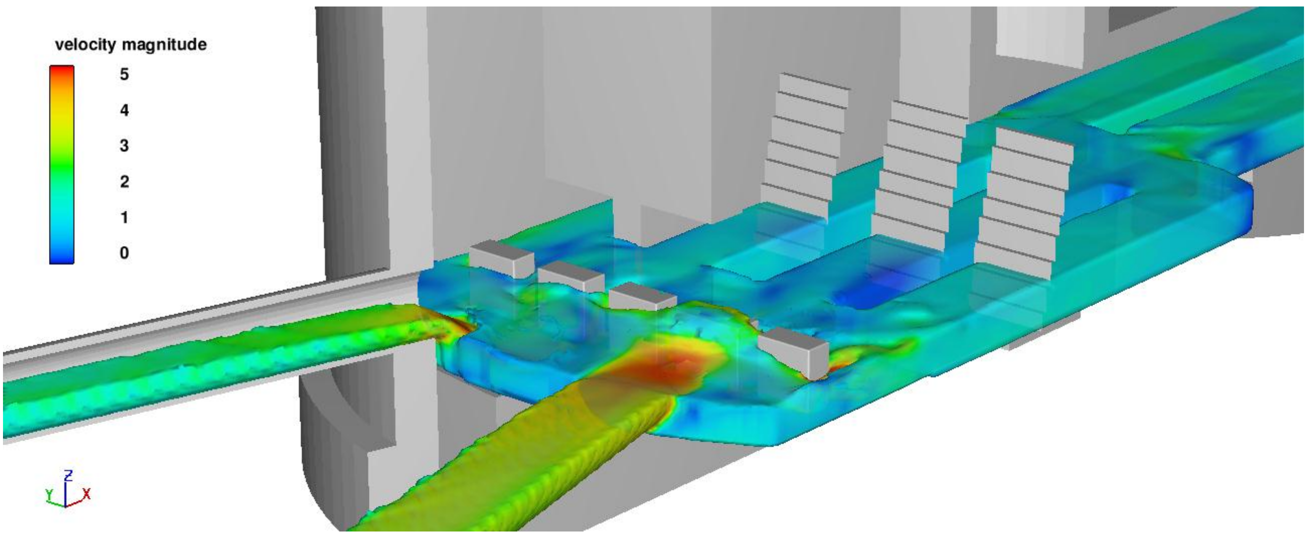


Figure 3.10: S04.Option3.PFF - 3D View - Surface Velocity (m/s)

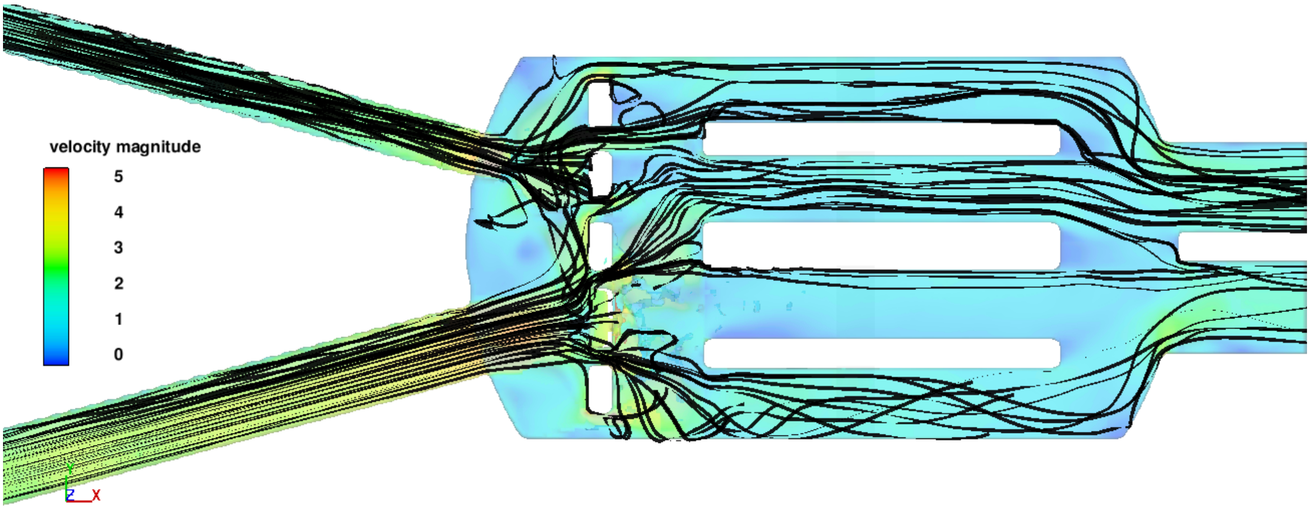


Figure 3.11: S04.Option3.PFF - Streamlines

3.5. Baffled Option 4 – Double Baffle Row

3.5.1. Geometry

An alternative baffle arrangement which was developed to disperse the high velocity flows originating from the larger 3.2m diameter inlet was the double row of baffles shown in Figure 3.12 below. The single line of baffles previously modelled (Baffle Option 2) was moved upstream by 1 m and a second row added as shown in Figure 3.12 offset by 1 m. All baffles were equally sized at 2.5m x 2m by 1m.

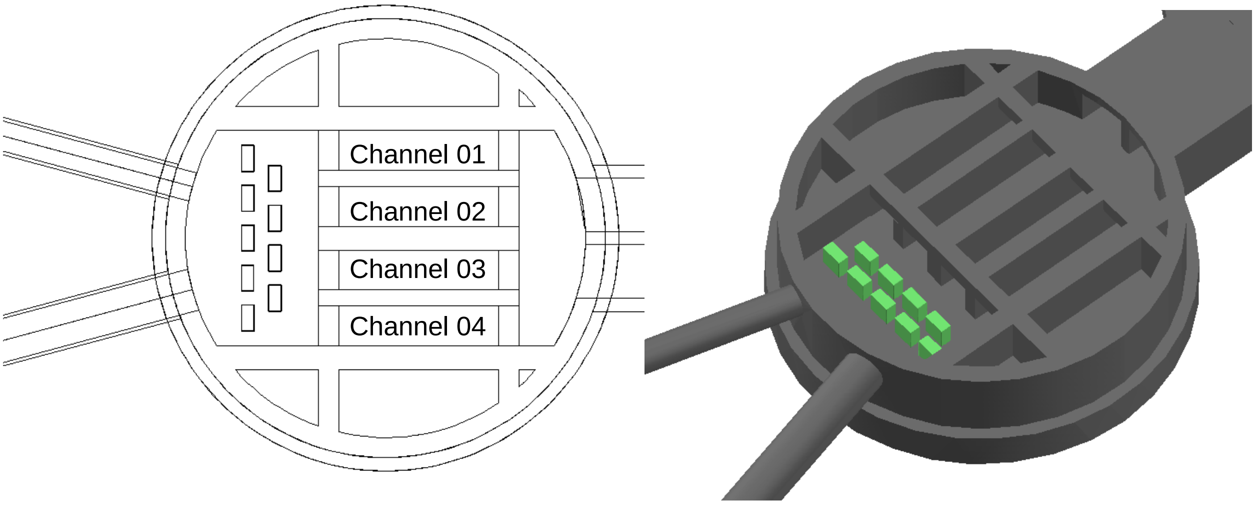


Figure 3.12: Baffled Option 4 – Double Baffle Row

3.5.2. S05.Option4.PFF– Results

A double baffle row was trialled as an alternative means of breaking up the jet formed by the 3.2 m diameter tunnel. This proved partially successful and resulted in reduced flows in the central channels and increased flows in the outer channels. This may be due to the downstream baffle row not extending to the wall on either side. The addition of two extra baffles (not modelled) may improve or correct this.

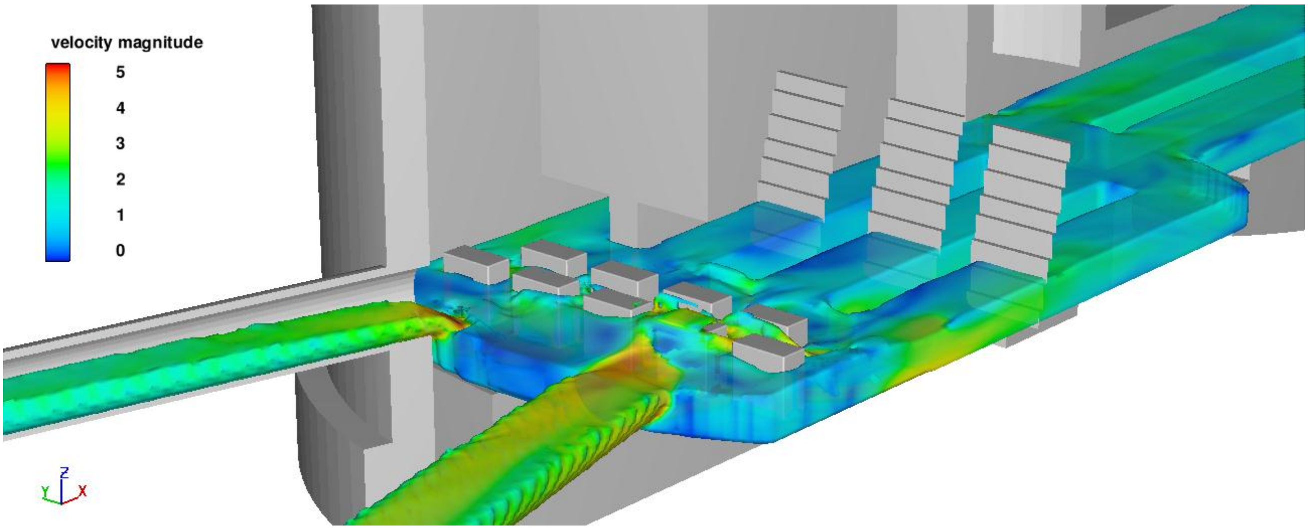


Figure 3.13: S05.Option4.PFF - 3D View - Surface Velocity (m/s)the flow distribution.

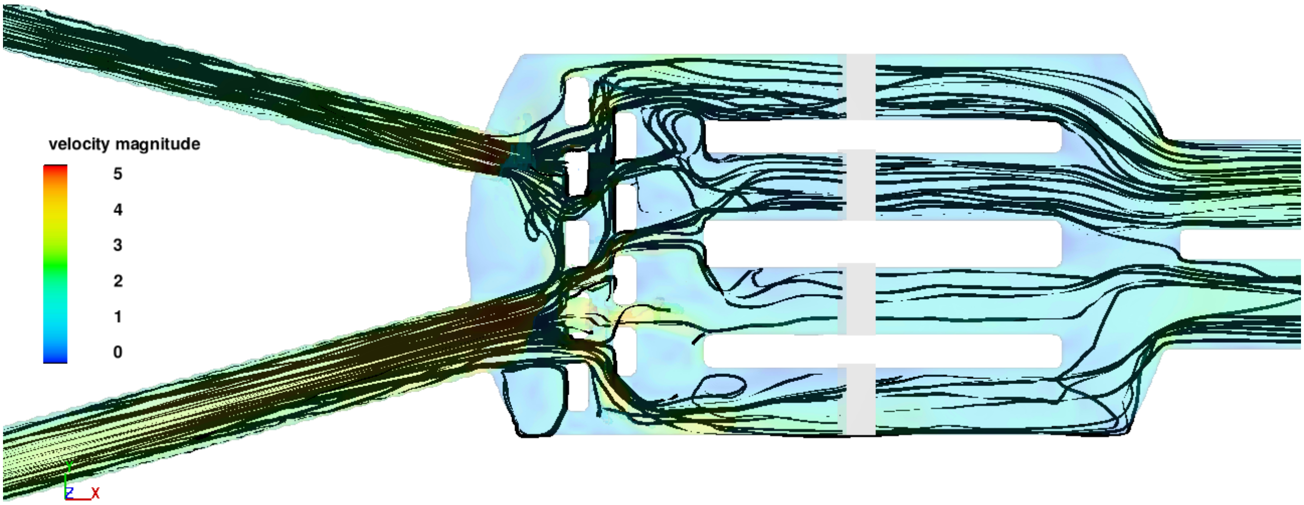


Figure 3.14: S05.Option4.PFF – Plan View – Streamlines

3.6. Baffled Option 5 – Downstream Slots

3.6.1. Geometry

A slotted weir was trialled in order to introduce a significant head loss downstream of the screens to help balance the flow independent of the shaft size. The baffles forming the slot were 0.5 m thick and protruded into the screen channel by 1.05 m to leave a 900 mm slot. Benching was applied at the base, both upstream and down, at 45 degrees to prevent sediment deposition as shown in Figure 3.15 below.

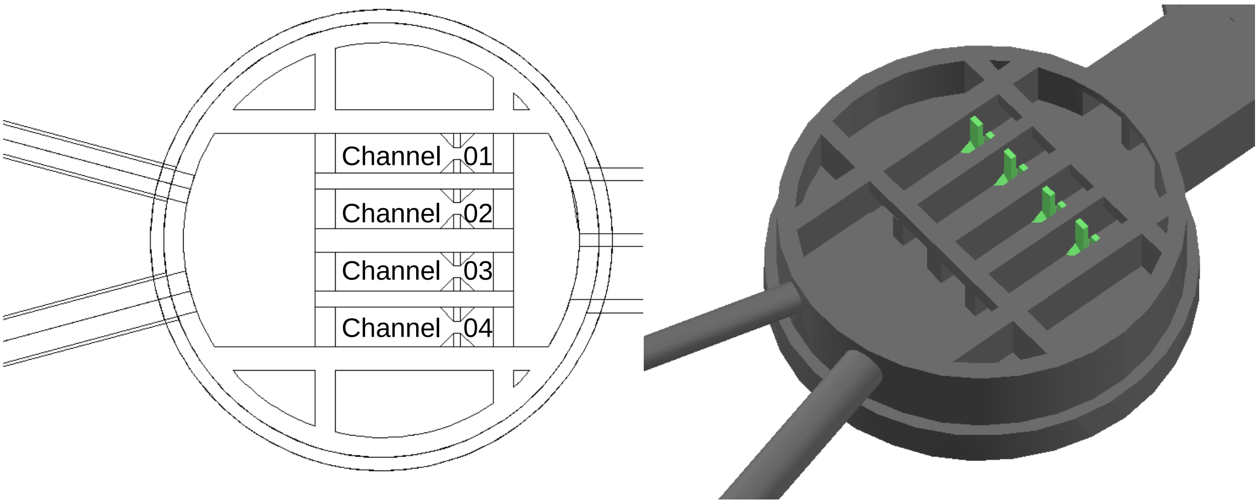


Figure 3.15: Baffled Option 5 – Downstream Slots

3.6.2. S05.Option4.PFF - Results

An alternative method for improving flow distribution was also posited and trialled. This involved the inclusion of head loss elements downstream of the screens in the attempt to provide an even flow distribution independent of the screen chamber size.

The head loss elements trialled were slot weirs which showed some success but would need an additional constriction to perform as required. Channel 3 continued to take significantly more flow than the mean (36.8%).

To provide this head loss, the flow depths in the upstream portion of the chamber were increased and affected the depths in the 3.2 m diameter tunnel.

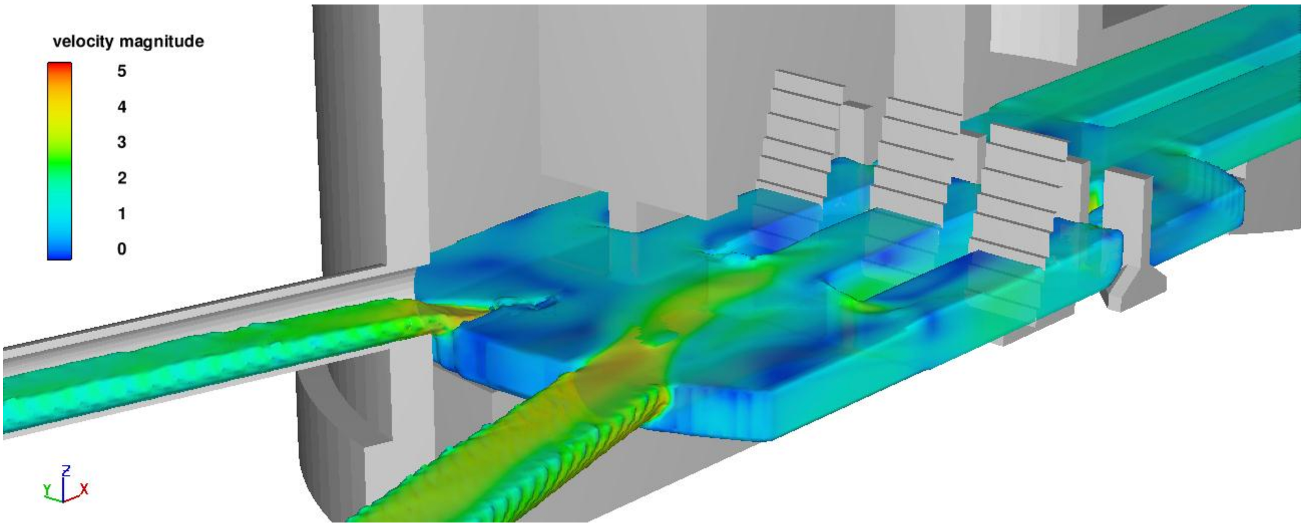


Figure 3.16: S06.Option5.PFF - 3D View - Surface Velocity (m/s)

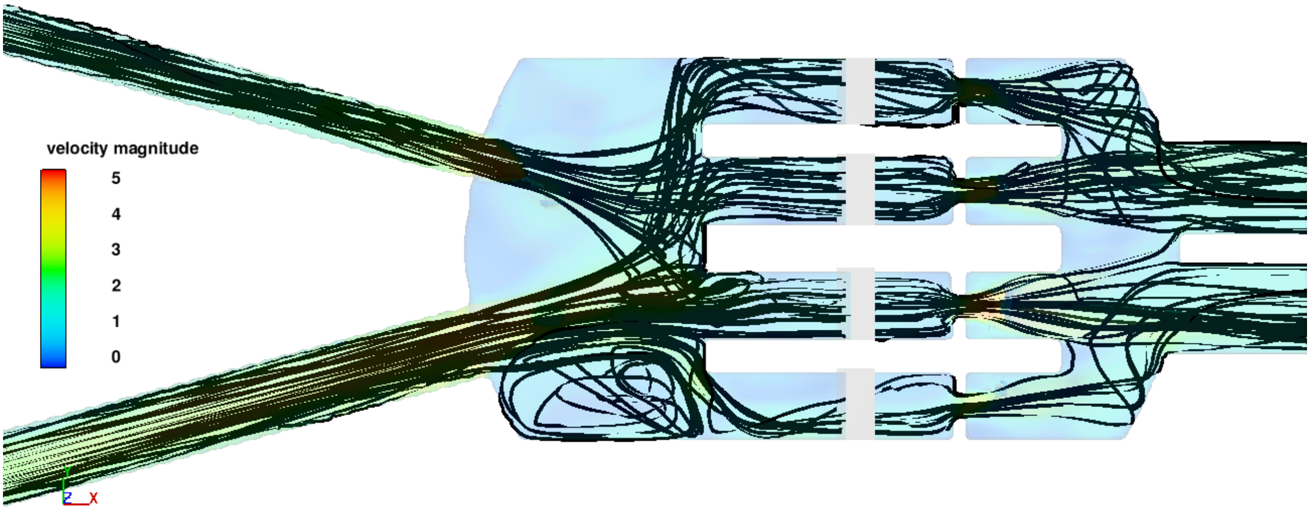


Figure 3.17: S06.Option5.PFF – Plan View - Streamlines

3.7. Baffled Option 6 – Baffle Row and Downstream Slots

3.7.1. Geometry

Following the partial success of baffle option 5, it was clear that further head losses would be required in order to prevent the fourth channel in particular from carrying biased flows and so an option combining Options 3 and 5 was developed as shown in Figure 3.18.

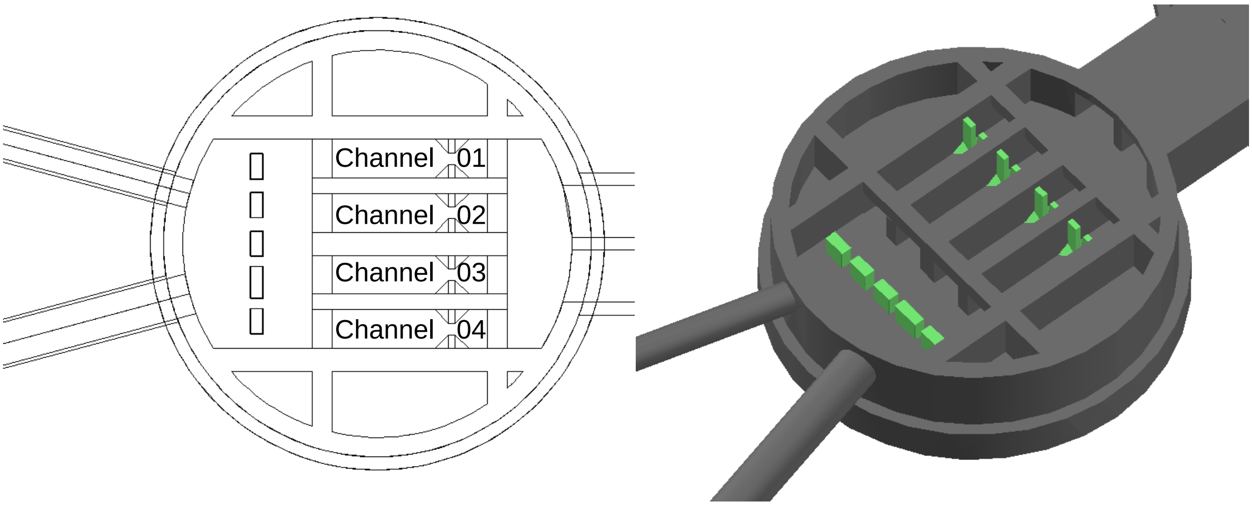


Figure 3.18: Baffled Option 6 – Baffle Row and Downstream Slots

3.7.2. S07.Option6.PFF - Results

Option 6 involved the combination of Options 3 and 5 and resulted in the most balanced flow of all options trialled. Flow again backed up into the larger tunnel which was considered not to be acceptable.

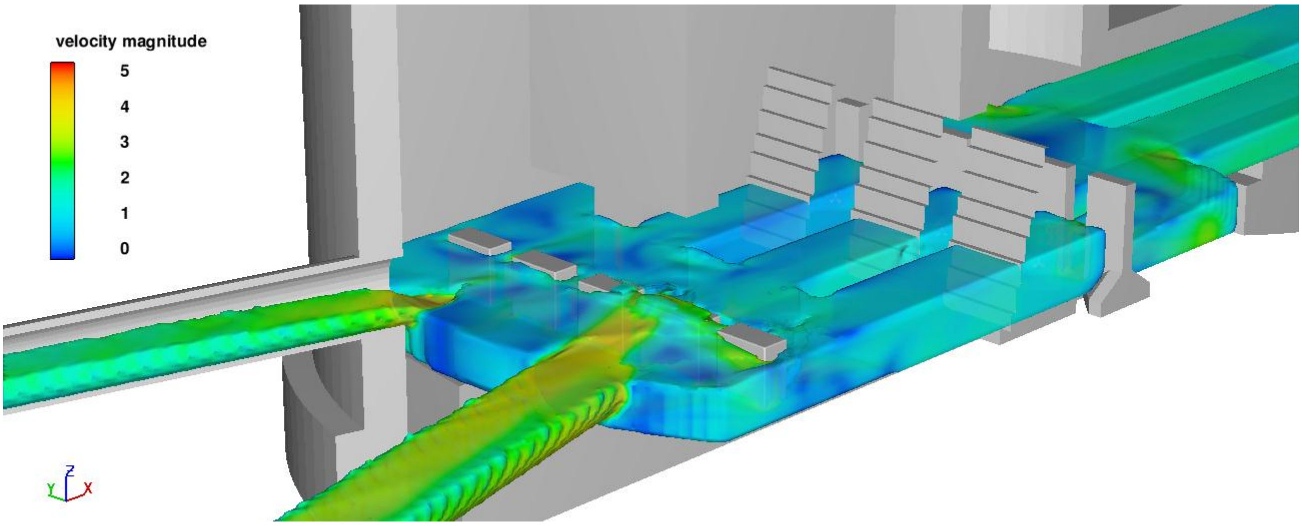


Figure 3.19: S07.Option6.PFF - 3D View - Surface Velocity (m/s)

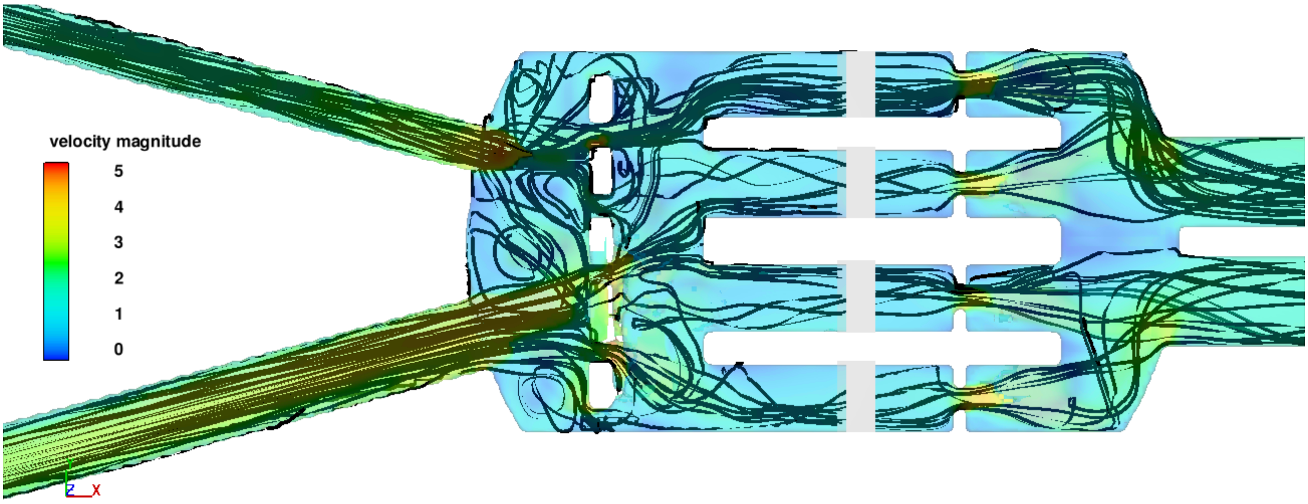


Figure 3.20: S07.Option6.PFF – Plan View - Streamlines

3.8. Optioneering summary

The flows, velocities and depths are shown summarised in the following figures at PFF for all geometries optioneering geometries.

Flows under the original geometry showed heavy bias toward the third screen channel resulting in heavily unbalanced flow and over 1.5 m^3/s of reverse flow through the fourth channel. The double aligned baffle simply blocked flow access to the third channel. Options 2 through 6 showed more promise as described previously.

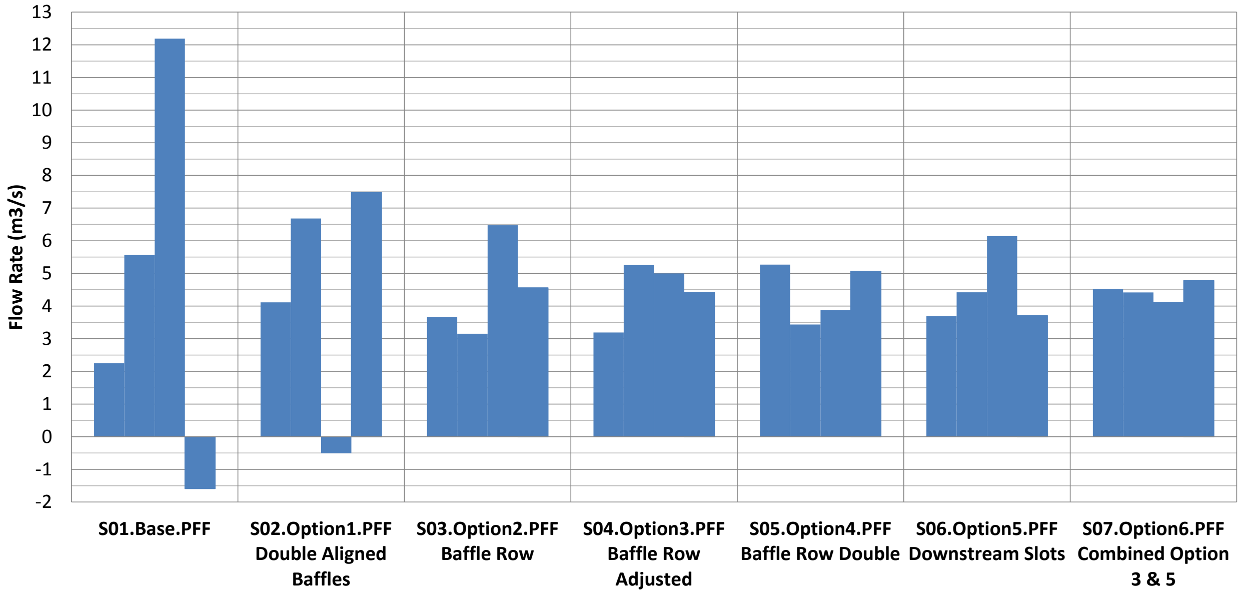


Figure 3.21: PFF – Mean Flow in Channels

The percentage difference from the mean channel flow presented in Figure 3.22 below, shows that options 3, 4 and 6 meet the flow difference criteria of <33%. Option 6 achieves the most even flow distributions and a maximum deviation of only 7.5% but results in flow back up.

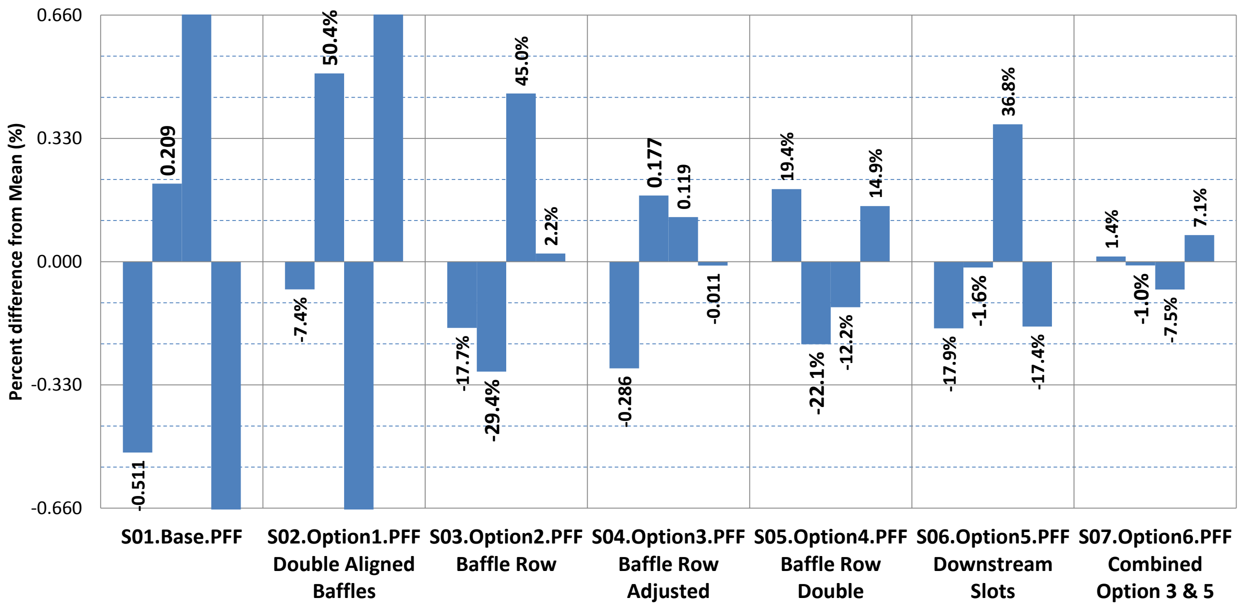


Figure 3.22: PFF – Percent Difference from Mean Flow in Channels

Velocities in the channels compared against the maximum velocity criteria of 1.2 m/s are presented in Figure 3.23, show that Options 3, 5 and 6 are acceptable with Option 4 only marginally above the acceptance criterion. The velocities reported are mean velocities over 20 seconds of simulation time.

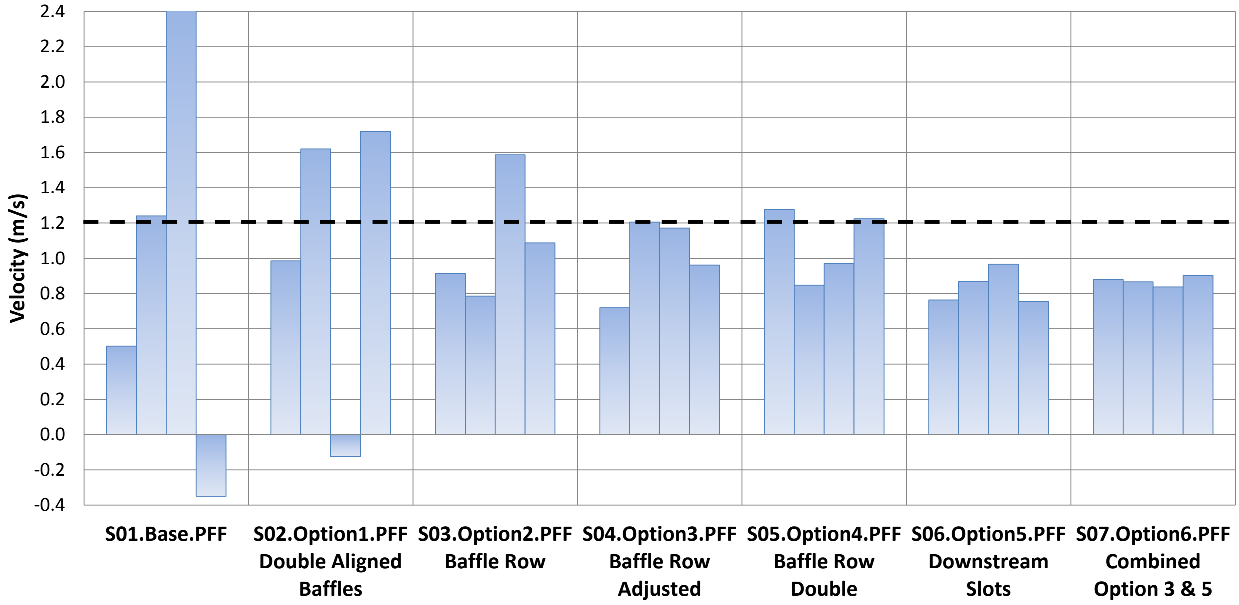


Figure 3.23: PFF – Channel Mean Velocities

The mean flow depths in the channels are largely uniform and show that options 5 and 6 involving the downstream flow constriction raise the water levels in the system by 0.5m on average. This causes backing up of flow in the inlets with an increase in flow depth of 0.2m in the 3.2m diameter inlet between Option 2 and Option 6.

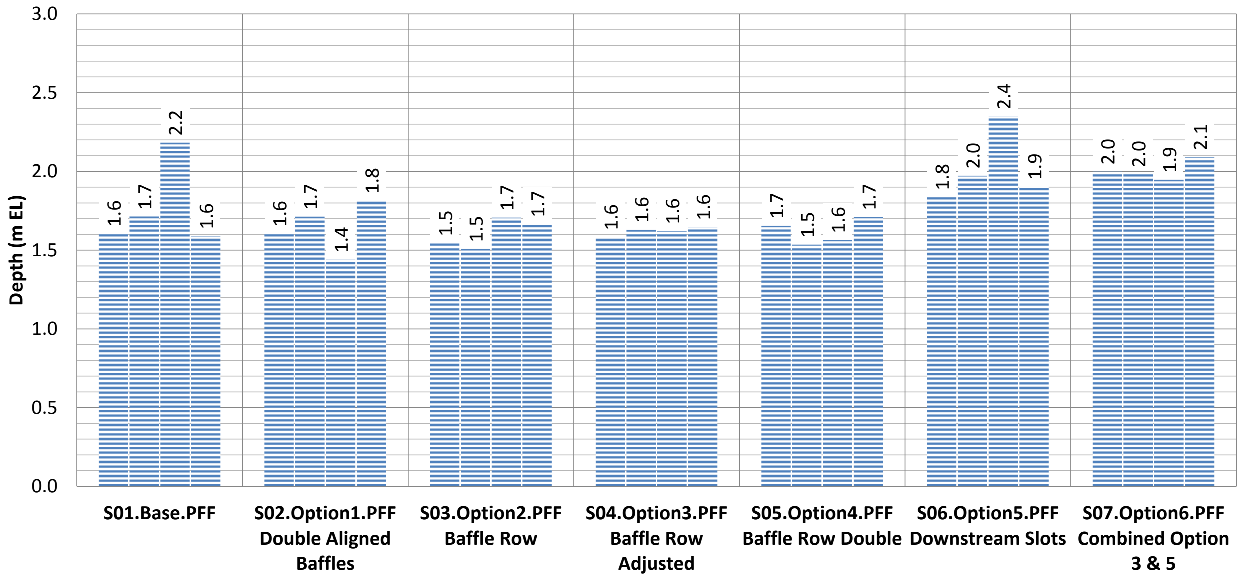


Figure 3.24: PFF – Mean Flow Depth in Channels

3.8.1. Symmetry Coefficient

The description of the symmetry coefficient is presented in Section 2.4.1 and ranges from 0 to 1 where a symmetry coefficient value of 1 indicates that each of the channels are carrying equal flow. As can be seen from

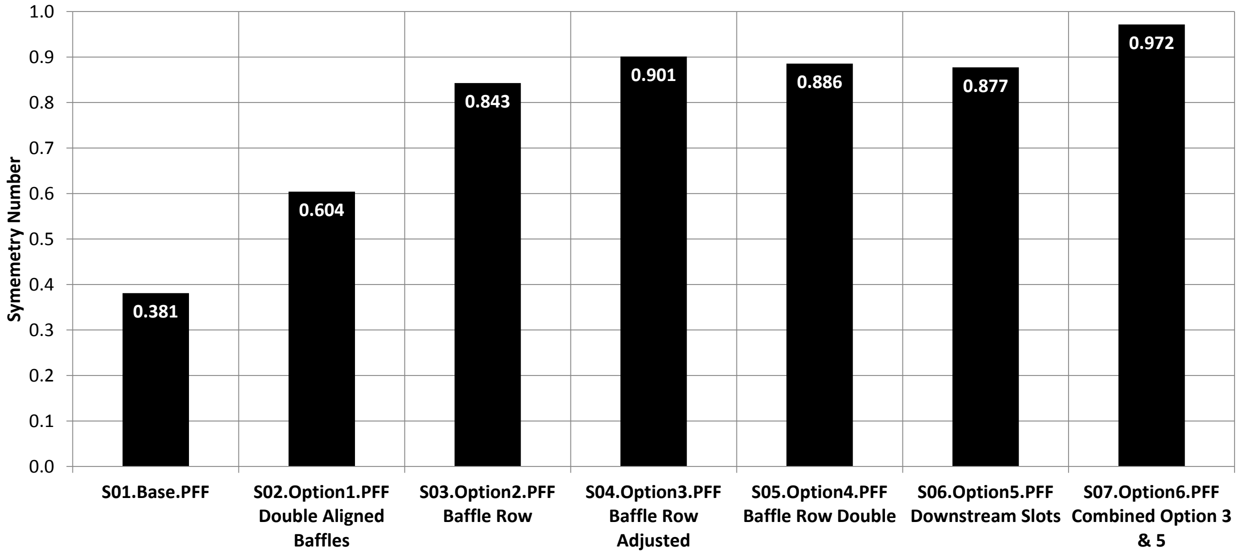


Figure 3.25 below, Option 6 achieves the most symmetrical flows followed by Option 3 and Option 4 which achieving the next highest symmetry coefficients and also meeting the acceptance criteria of no greater than +/- 33% of the average flow through any channel.

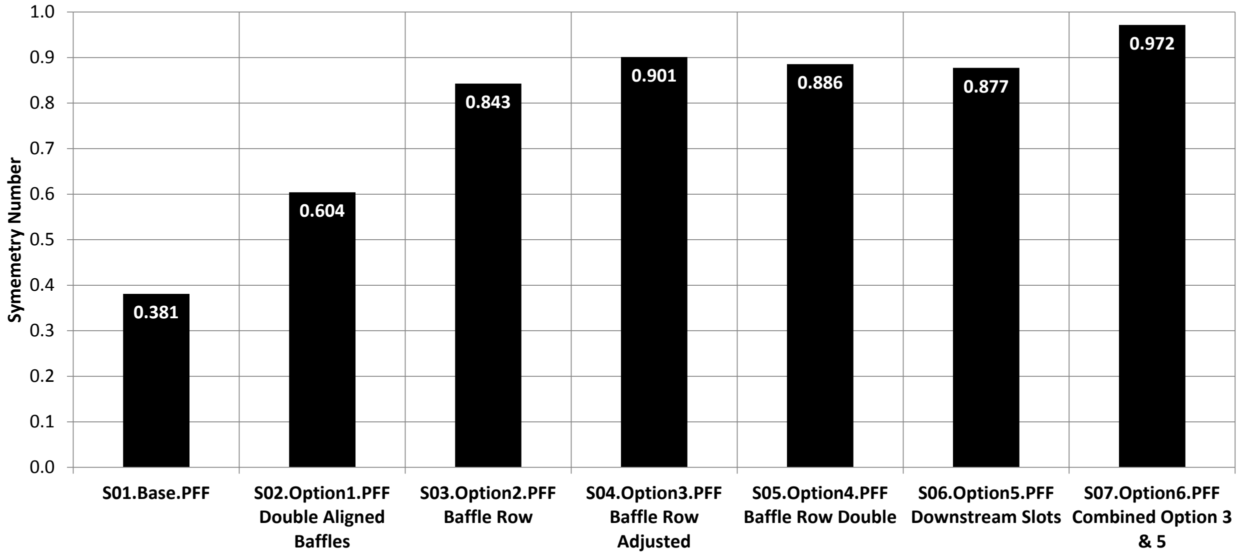


Figure 3.25: PFF – Flow Symmetry Coefficient per geometry option

3.9. Recommended Options

Table 3.1 summarises the acceptability against the velocity and flow criteria showing that Option 3, Option 4 and Option 6 provide the required flow balance however the velocities in Option 4 surpass the maximum allowable velocity.

As mentioned previously, option 6 achieves the highest degree of flow and velocity symmetry in the channels and so is the superior option for efficiency alone. However, as option 6 increases the flow depths through the system and is simply the addition of downstream slots to the already acceptable option 3, Option 3 was recommended as the preferred configuration for the screen shaft diameter of 30 m.

Given flow controls were able to meet the acceptance criteria, an increase in the shaft diameter was not considered necessary and the 30 m shaft diameter was adopted as adequate.

Table 3.1: Summary of acceptance criteria

Criteria	**Base Case** Un-baffled	**Option 1** Double aligned baffles	**Option 2** Baffle Row	**Option 3** Baffle Row Adjusted	**Option 4** Baffle Row Double	**Option 5** Down-stream Slots	**Option 6** Com-bined Option 3 & 5
Flow < +/- 33%	N	N	N	Y	Y	N	Y
Mean Velocity < 1.2 m/s	N	N	N	Y	N	Y	Y
ACCEPTABLE?	N	N	N	Y	N	N	Y

The dimensions of the recommended option are shown in Figure 3.26.

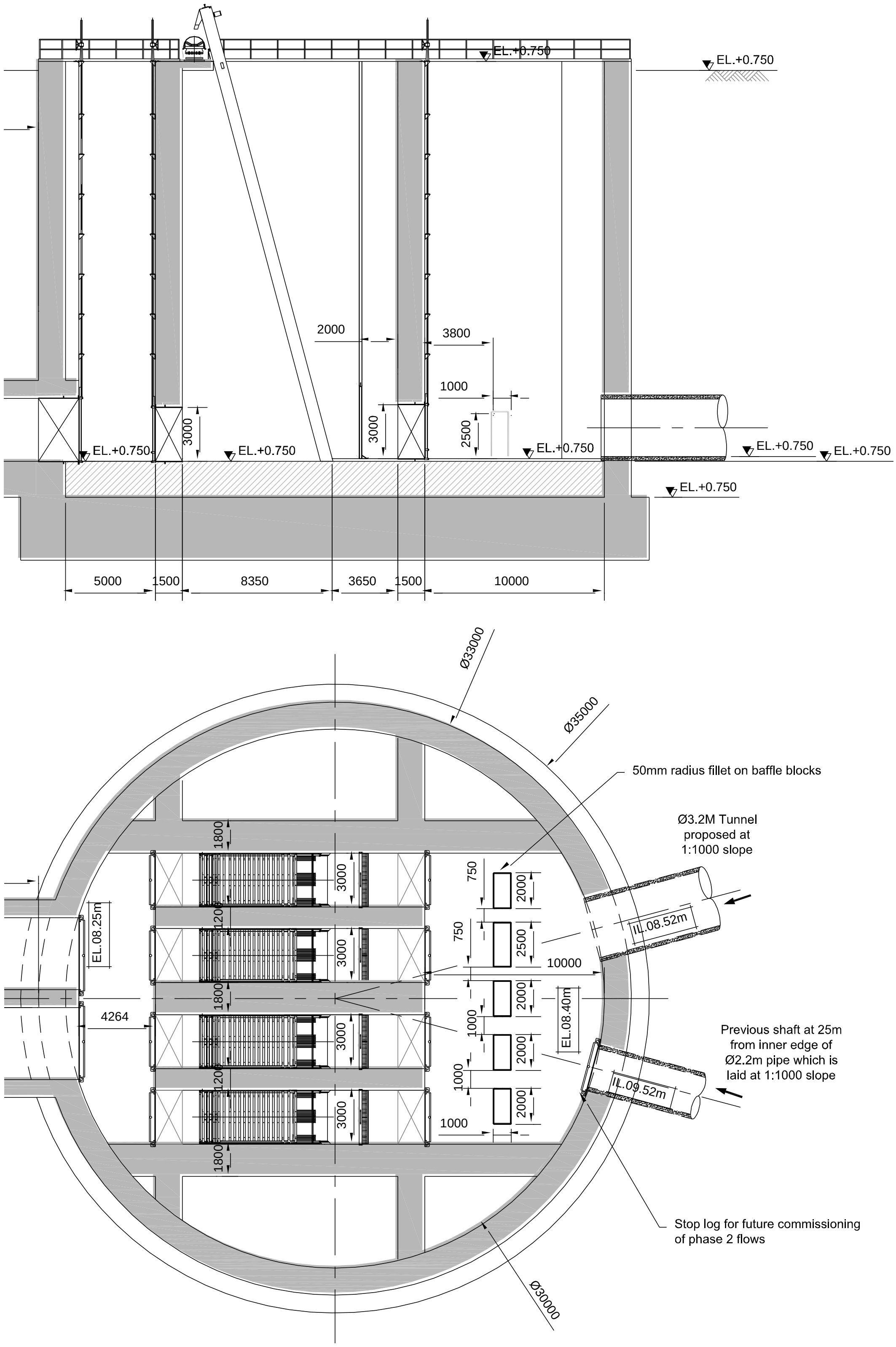


Figure 3.26: Option 3 Detailed Geometry – Plan and Elevation

4. PERFORMANCE OF RECOMMENDED OPTION

Following review and adoption of the 30 m diameter shaft with an adjusted baffle row to establish a reasonable flow balance (Option 3), the geometry was run for all flow cases with both the base case geometry and that proposed. Two full sets of runs were undertaken for two tunnels operating and for only the 3.2 m diameter tunnel operational.

4.1. Flows and symmetry

Channel flows for the base case and Option 3 baffled simulations are summarised in Figure 4.1. Base case runs are shown on the left and Option 3 performance on the right. These runs confirmed that at lower flows, the screen chamber continued to perform well, improving as flows were reduced. Flow symmetry was shown to be high (>90%) for all Option 3 cases, indicating that performance criteria would be met.

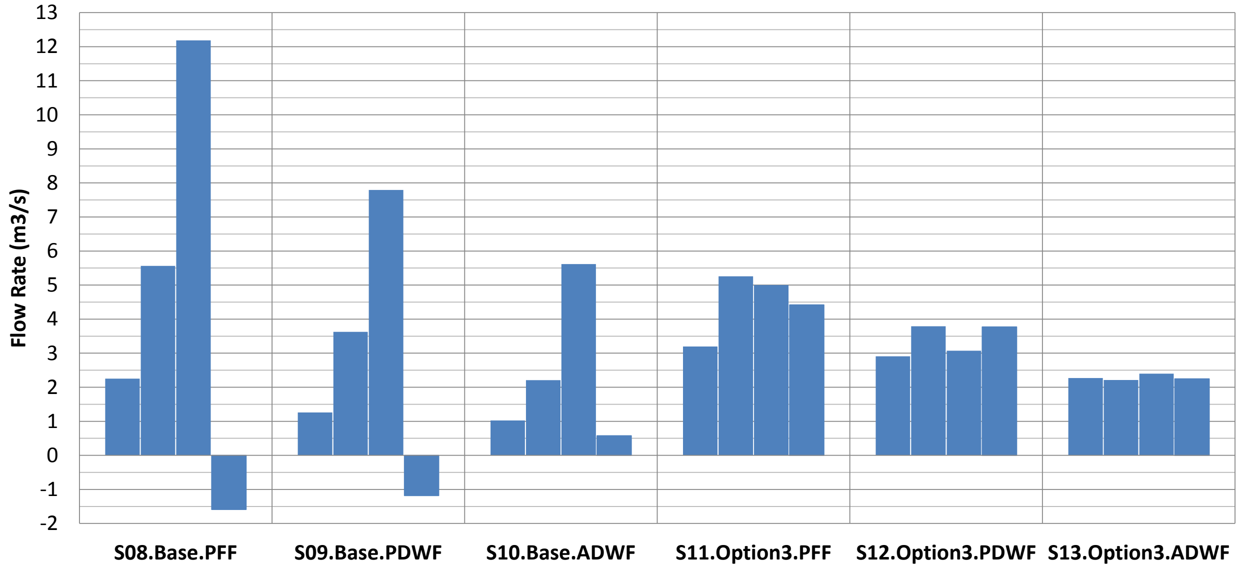


Figure 4.1: Base and Final Case Mean Channel Flows

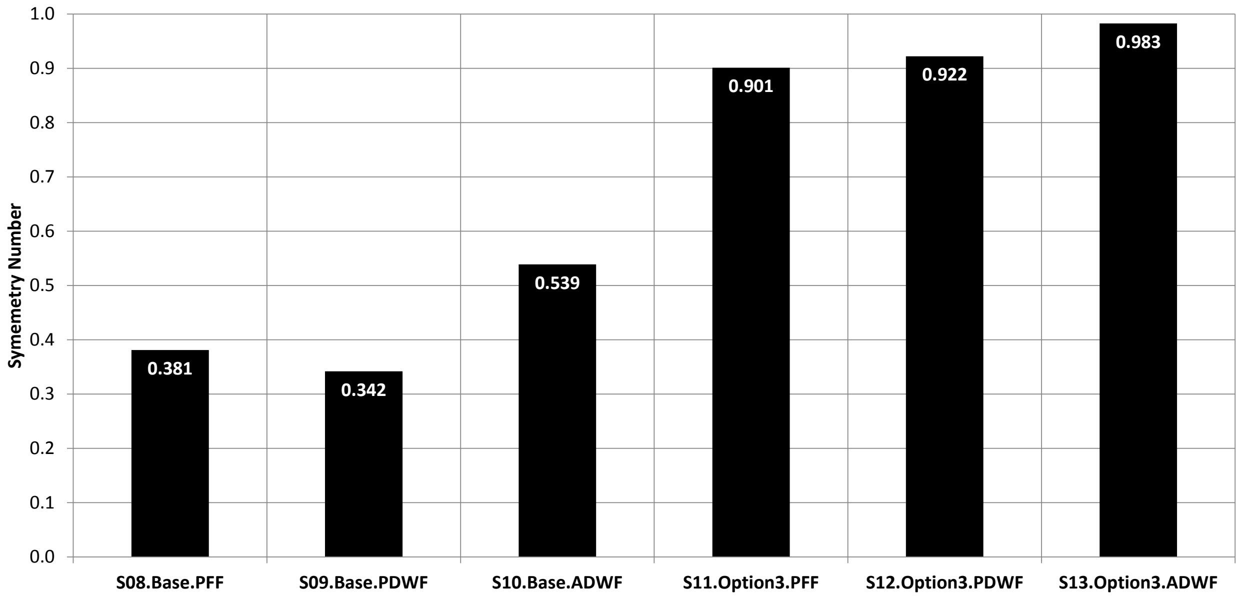


Figure 4.2: Base and Final Case Mean Symmetry Coefficients

4.2. Performance criteria

As can be seen in Figure 4.3, the maximum deviation from mean flow for the Option 3 baffled case is 28.6% and occurs in Channel 1 for the PFF as shown during optioneering. By the time the flows drop to the annual dry weather flows, the maximum deviation from the mean is only 5.1% and in all Option 3 flow scenarios the flow balance is acceptable. Without flow controls, flows are unbalanced across the range of flows modelled.

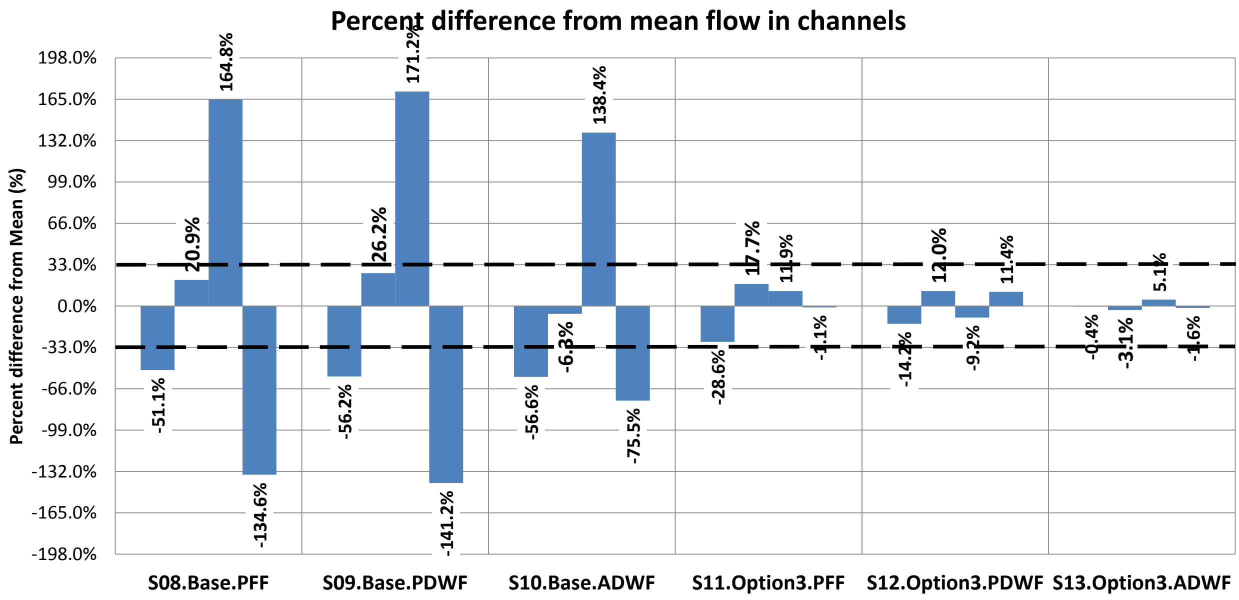


Figure 4.3: Final Case Deviation from Mean Flow

The maximum channel velocity at the pass forward flow and the final baffle case is at the maximum allowable value of 1.2 m/s, as can be seen from Figure 4.4 below. The baffled results are compared to the base case results for all flow cases and show marked improvement and acceptability against the criterion.

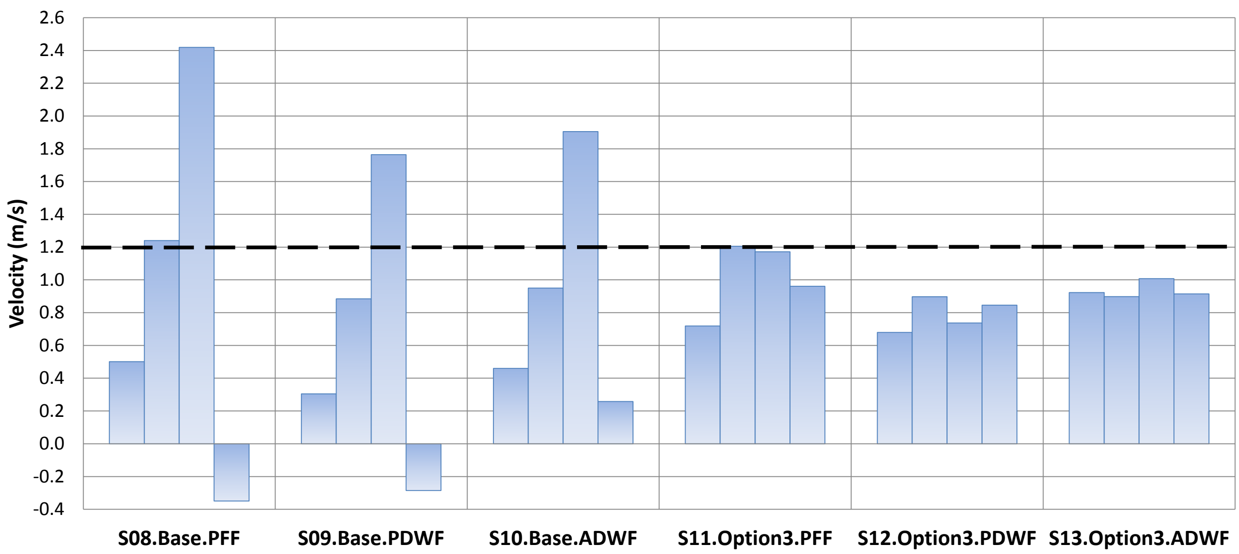


Figure 4.4: Base and Final Case Mean Channel Velocities

5. CONCLUSIONS

Without flow controls, numerical modelling showed the existing shaft diameter suffered un-balanced flow and unacceptable velocities across the four screen channels.

The modelling undertaken as part of this model study identified a single row baffle configuration that will provide balanced flows within a 30 m diameter screen chamber across the Pass Forward Flow, Peak Dry Weather Flow and Average Dry Weather Flow.

5.1. Choice of solution

The final baffle case involved a single row of five baffles. The first three and the last one in the line were 2.5 m high by 2 m wide by 1 m deep baffles and the fourth was adjusted to be an extra 0.5 m wider in order to prevent preferential flows occurring in the third channel (opposite the 3.2 m diameter tunnel exit).

From this baffle arrangement, it was possible to achieve a maximum difference in channel flows of 28.6% and a maximum channel velocity of 1.2m/s for the pass forward flow, and so to meet the performance criteria specified. Whilst one of the investigated options proved to be more efficient in terms of ensuring even channel flows, with a maximum deviation from the mean of only 7.5% at the pass forward flow, this would require an additional row of flow constrictors behind the screens in the channels and so was not deemed to be economical. The recommended baffle geometry (Option 3) is presented in Figure 3.26.

5.2. Mitigation of design risk though CFD modelling

CFD modelling undertaken at an early design stage proved useful in demonstrating the problems that were expected and the design changes necessary to remedy them. The study was successful in decreasing the design risk by increasing designer confidence in the system and modifications proposed by showing that the current shaft dimensions were adequate.

As shown through this modelling, CFD is proving increasingly useful when utilised in the early stages of design. If used correctly, early modelling can reduce design and program risk minimising any unforeseen issues or problems during detailed design and physical modelling ultimately reducing the costs for contractors and end clients.

5.3. Future Work

Following tender design it is expected that further modelling work would be undertaken during detailed design. This will likely include more detailed CFD analysis of the screen chamber and wet well and physical modelling of combined system prior to construction.

REFERENCES

Novak P., Moffat A.I.B., Nalluri C., Narayanan, R., Hydraulic Structures - Forth Edition (2007)