Cambridge Waste Water Treatment Plant Relocation Project
Anglian Water Services Limited

Appendix 20.7 Outfall CFD Report

[redacted]

Application Document Reference: 5.4.20.7
PINS Project Reference: WW010003
APFP Regulation No. 5(2)a

Revision No. 01
April 2023

Issue 01

CFD MODELLING OF OUTFALL

Cambridge WWTP Relocation project

Project no. 4020267

Prepared for:

Anglian Water

August 2022

binnies

Enhancing lives, communities and the environment

Table of contents

Details of document preparation and issue:

Version no.	Prepared	Checked	Reviewed	Approved	Issue date	Issue status
P01	J Powell	A Harriott	D Winzer	James Powell	16 August 2022	Draft

Project no. 4020267 Client's reference no. 00001-100006-ZZZZZZ-ZZZ-RPT-Y-3112

File name: CambridgeOutfallCFD_4020267_Report_03.docx

1. Introduction

1.1 Background

Anglian Water has proposed relocating its Cambridge Waste Water Treatment Plant to enable the regeneration of North East Cambridge. The relocation will provide upgraded treatment and enable new homes and commercial space to be constructed. The project is known as the Cambridge Waste Water Treatment Plant Relocation Project (CWWTPRP).

The project involves the provisions of a new waste water treatment plant (WWTP), an extension of the existing Riverside Sewer Tunnel to convey flows to the new WWTP and infrastructure to convey the treated effluent from the new WWTP to a new outfall on to the River Cam.

The design has progressed to a stage that modelling of the proposed discharge of final effluent and storm flows to the River Cam is required for optimising the design of the outfall. Binnies has been assigned a task covering:

- Stage 1: River modelling of the River Cam using an existing 1D-2D hydraulic model of the River Cam. This is to assess fluvial flood levels throughout the River Cam and the relative impact of the new outfall compared to existing conditions.
- Stage 2: River and outfall modelling using a new local hydrodynamic model of the River Cam in the vicinity of the new outfall (in 2D or 3D). This is to assess velocities and mixing of the effluent as it enters the River Cam.
- Stage 3: Outfall modelling using Computational Fluid Dynamics (CFD). This is to inform the design of the outfall, for example to prevent scour of the riverbed and opposite bank.

This report presents the CFD modelling work undertaken on this task (Stage 3 above).

1.2 Scope

The scope provided for this work in the Project Brief is [Ref 1] is copied below:

Use of CFD (Computational Fluid Dynamics) model, during design definition, to carry out a 3D numerical simulation of the outfall and interface with the River.

- *To optimise and refine the outfall arrangement to maximise energy dissipation and to minimise the impact of the outfall on river users and the natural environment.*
- *To demonstrate the outfall design provides adequate mixing of the effluent entering the river.*

Whilst not specifically part of the initial scope, consideration is also required of flow velocities and disturbances (e.g. waves) that may influence:

- River craft movements, including vulnerable craft (such as potentially unstable rowing boats and canoes)
- The stability of the riverbed (scour and erosion)
- The riverbank system (including existing engineered protection measures and the natural bank systems)

The work presented in this report gives predicted velocities and flow patterns that can be used to inform decisions about the above. The outfall arrangement is bespoke and the CFD model is aiming, at this initial design stage, to refine the outline design arrangement. The work has tried to develop designs that optimise the dispersion of flow, thereby minimising peak velocities and hydraulic disturbances within the river. However, the acceptability with respect to craft and stability of the river banks will need to be confirmed by others with specific experience in these fields prior to accepting a final design.

The scope did not define any specification for the CFD modelling method such as software, model extents or mesh resolution.

2. Model extents and layouts

2.1 Extents

The approximate extents of the model are overlaid on a Google Earth satellite image in Figure 1. The upstream boundary for the River Cam is located just downstream of the A14 road bridge and the model extends to 100m downstream of the new outfall. The riverbed was digitised from a triangulated surface fitted to bathymetric survey points. The banks of the river were trimmed in the model with a vertical wall to "clean" irregularities in the bathymetry data caused by vegetation or debris close to the river banks.

The west side of the river, opposite the outfall, features a tow path and has bank protection. The east side of the river, in the near vicinity of the outfall, will be protected by sheet piling (or equivalent) but is to retain the existing natural riverbank system further downstream. The river banks at the A14 bridge crossing, immediately upstream of the model, are protected with sheet piling with a concrete capping beam (extending up to a level of approximately 4.2m AOD).

The model has been developed to consider various geometrical arrangements of the outfall and different rates of flow where the water level remains within the river channel. It should be noted that:

- The outfall is in a flood plain and is intended to continue operating when the river banks are overtopped by occasional high water levels (approximately 4.2 to 4.3m AOD). A recent flood report [Ref 2] includes a diagram (Figure 4.6) demonstrating that the river will remain within channel for rates of flows up to and exceeding a 1 in 2 year return period, but that overtopping will likely occur for less frequent events approaching the 1 in 10 year level.
- The top of the outfall structure is intended to be approximately flush with the existing river bank. As a result, should the river level rise above the bank, then the outfall would not impede the flood water during this higher return event.

Figure 1 Approximate model extents overlaid on Google Earth satellite image

2.2 Outfall selection

An optioneering exercise considered various alternative outfall arrangements (see below). However, during the selection process an arrangement similar to the existing outfall (see Appendix C for details) was selected and the CFD studies presented in this report are based on that arrangement.

Optioneering exercise

The optioneering exercise considered the use of a USBR Type VI impact basin outfall arrangement, with an energy dissipation baffle and basin. An example of a USBR type VI outfall is shown in the CIRIA guide (in Section 12.5.4 of Ref 3). This option was eventually ruled out as:

— The height of the structure required to accommodate the overflow arrangement (above the baffle) was significantly elevated above the existing river bank.

— The arrangement was only suitable for the FE compartment; as the intermittent flows from the storm compartment may be subject to sediment and debris build-up from the river.

2.3 Initial layout

The layout for the initial model (model 100) is shown in Figure 2. This layout was digitised from dimensioned sketches provided by the design team [Ref 4] which are reproduced in Appendix A.

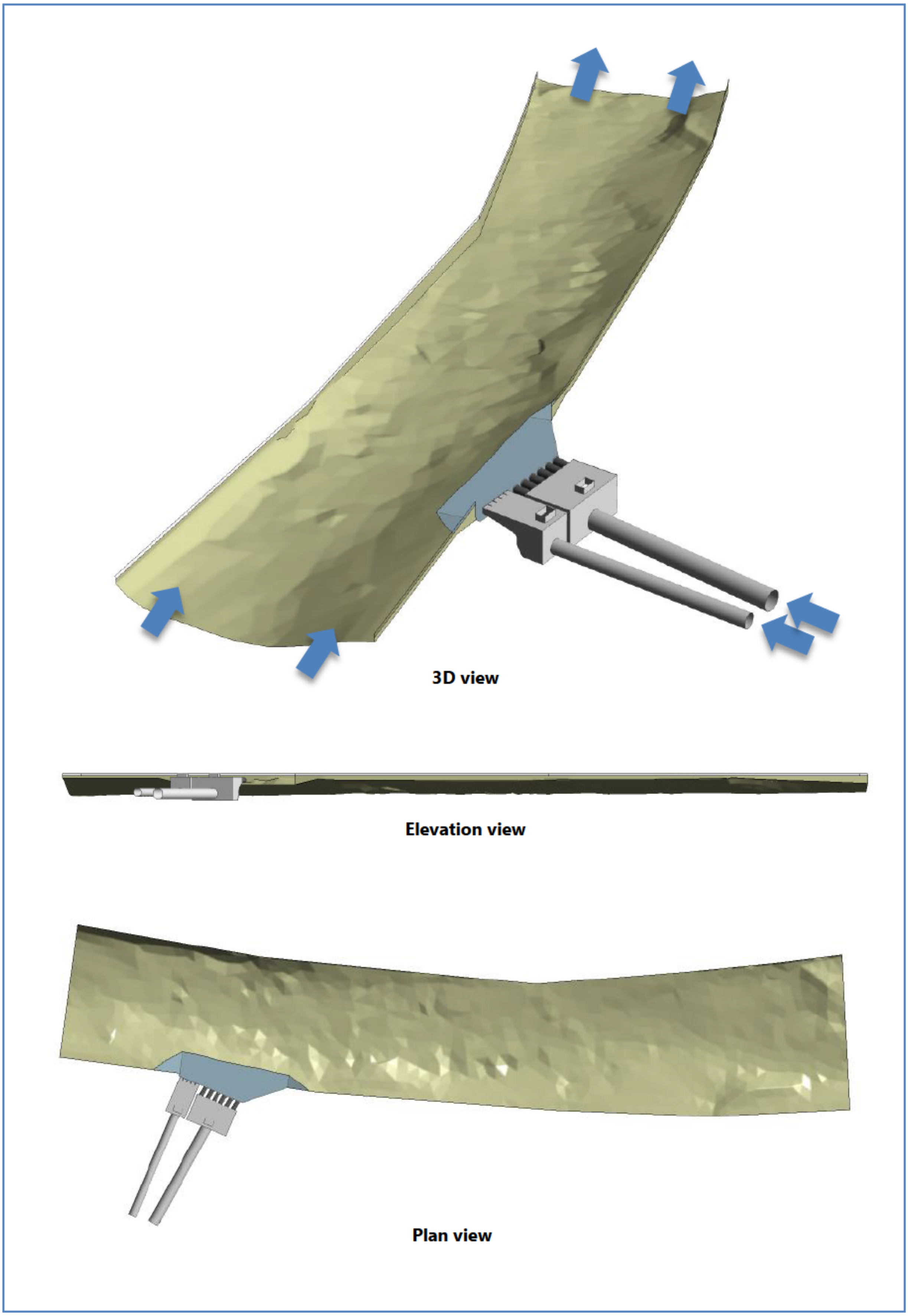


Figure 2 Initial design (Layout 100)

Outlet details

The FE outfall consists of five square (0.6m × 0.6m) openings with vanes directing the flow downstream in the river. This is based on the design of the existing outfall located upstream of the river which is understood to have performed well.

Although the outfall should be permanently submerged, the openings are of sufficient size that there could be a risk of unauthorised entry or ingress by large debris. **These risks should be recorded on the Risk Register** and addressed as the design develops. For the initial model, vertical bars have been placed at 150mm spacing to protect the outlets, but alternative or additional protective measures such as flap gates could also be considered. The outfall is also to include individual stop-logs frames for each of the channels, but these have not been included in the model.

The storm outfall operates intermittently with zero flow passing through it most of the time. This increases the risk of unauthorised entry and gives the potential for sediment or debris to accumulate. It was therefore concluded that the storm outlets require sealed back flow prevention. It is proposed to use Tideflex Checkmate valves to provide that seal. These valves consist of a rubber flap retained within a rubber cylinder that is sleeved within the pipe (Figure 3). The benefit of using Checkmate valves for this application are:

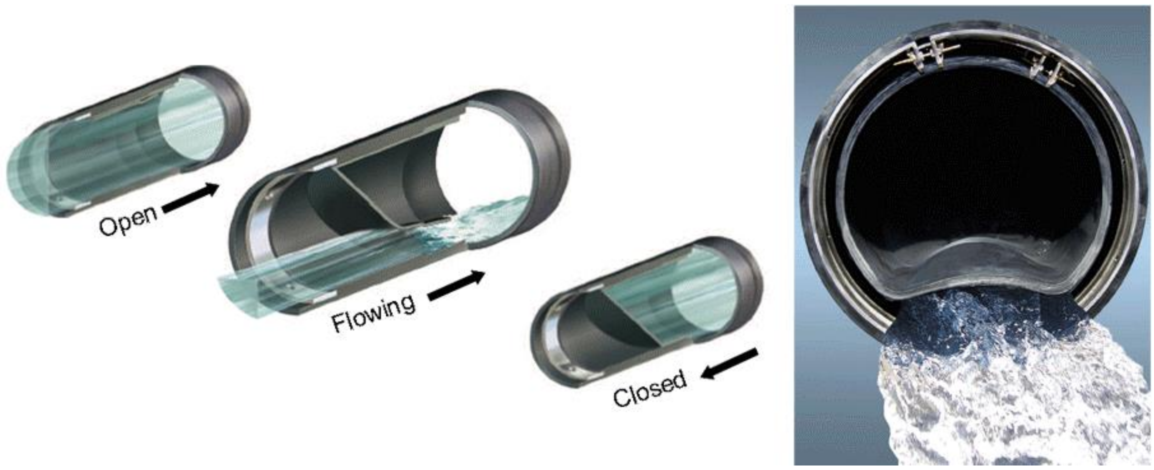
— They provide a full seal against backflow preventing ingress of sediment or debris.

— They provide a seal against odour.

— They prevent unauthorised entry

— Low maintenance.

Modelling a flexible rubber flap is not possible with conventional CFD as the shape of the surface and the opening area vary with flow. This would require a highly complex Fluid-Structural-Interaction (FSI) model which is not viable for this study. The valve has therefore been simplified in the model with the flap defined as a rigid inclined plate (Figure 4). Unfortunately, information is not available for the opening area of the flap vs flow and so we have arbitrarily set the position of the flaps so that it is 70% open (height above invert).



from: https://www.althon.co.uk/products/tideflex-checkmate-inline-valve/detail/

Figure 3 Manufacturer images of Tideflex Checkmate valve

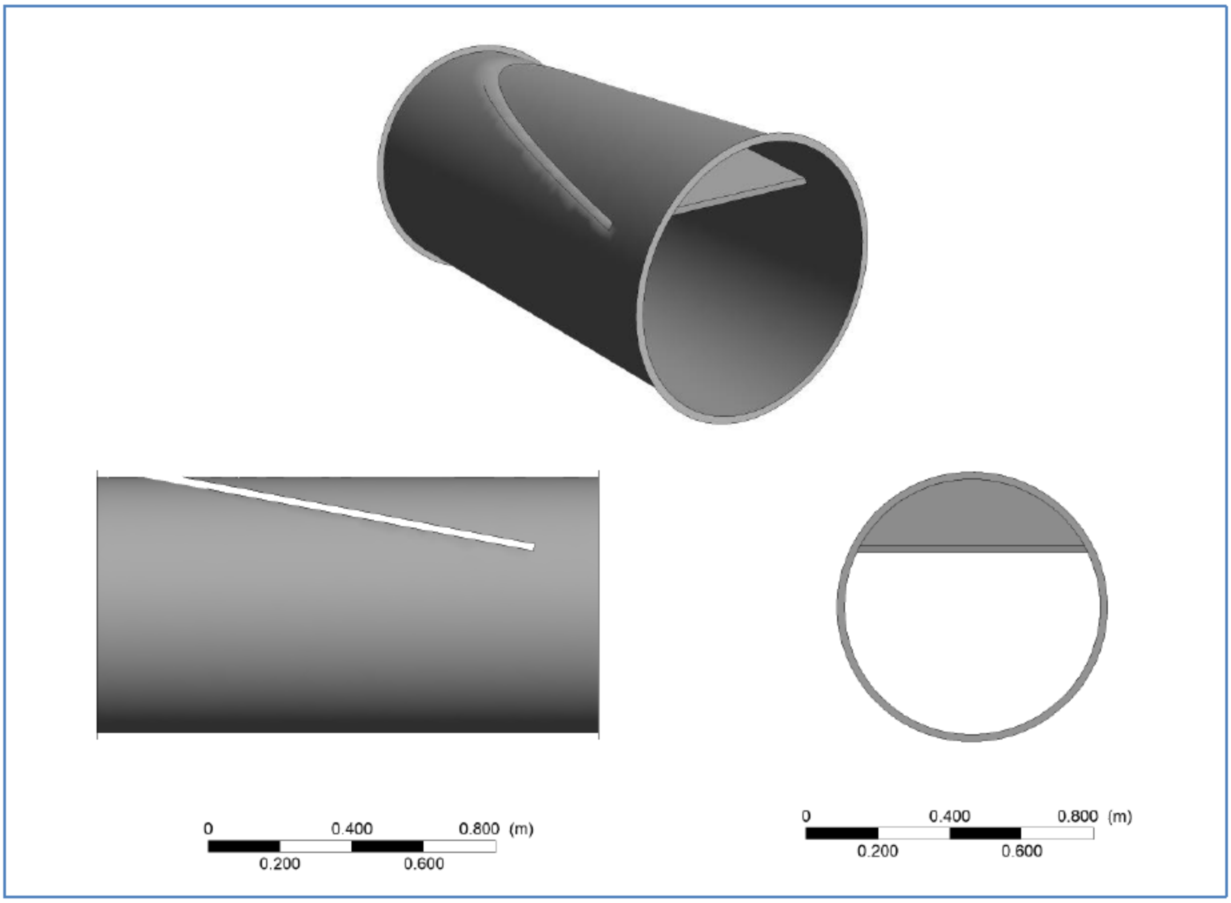


Figure 4 Tideflex Checkmate valve as modelled

2.4 Alternative layouts tested

The initial layout and three potential refinements were tested as listed in Table 1 with details of the modifications shown in Figure 5.

Table 1 Layouts

Layout	Description
100	• Initial design concept.
200	• Reduced flare on downstream side of outfall bay.
300	• Removed flare on outfall bay. • Realigned outfall ports to be perpendicular to outfall chamber.
400	• Removed outfall bay. • Chamber aligned parallel to river wall.

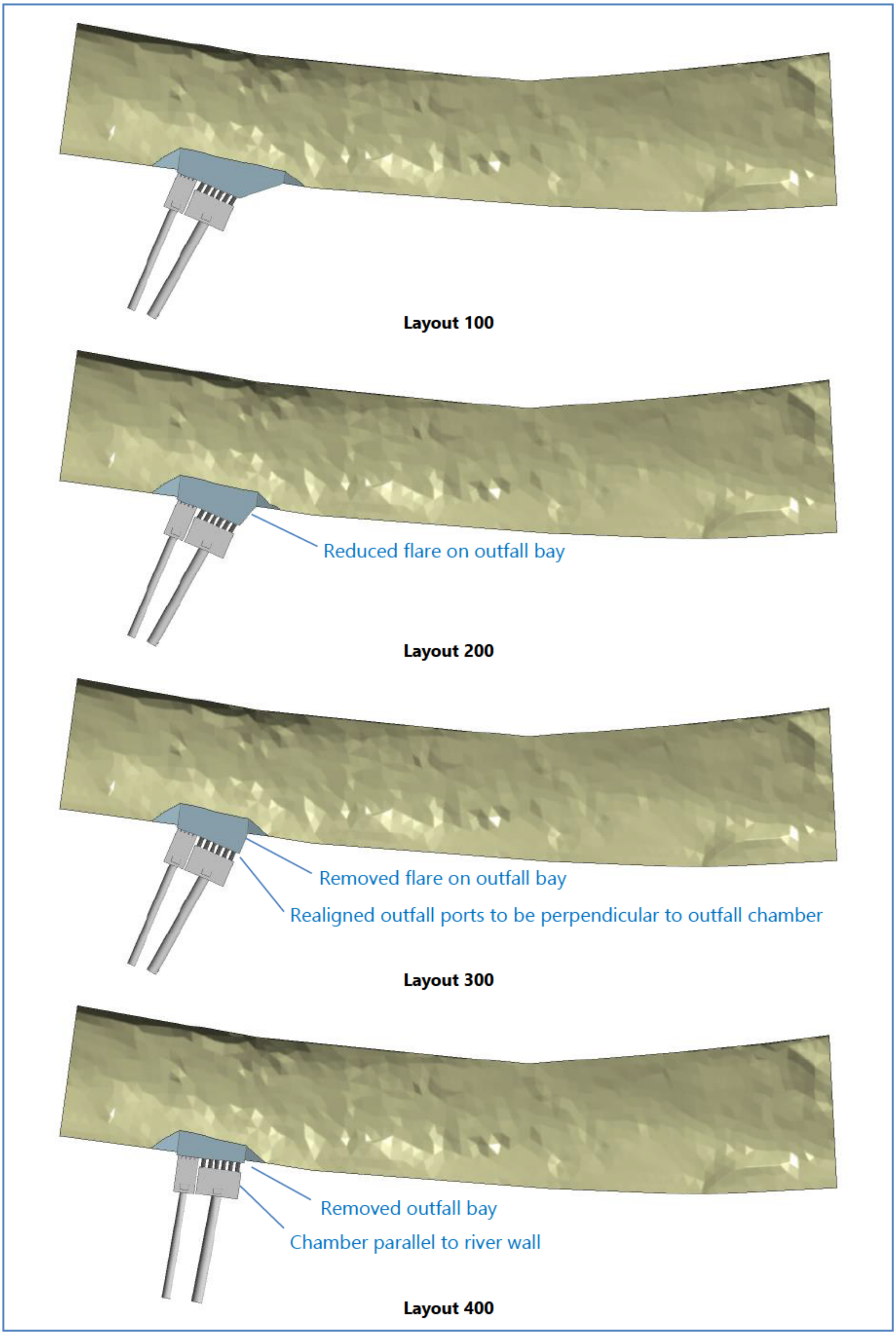


Figure 5 Details of modified layouts

3. Modelling approach and setup

3.1 General modelling details

CFD is a computer modelling method which simulates three-dimensional fluid flows using the finite volume method. The problem is discretised into numerous small elements, each possessing an algebraic approximation for the continuity of mass, momentum and energy, which are then solved simultaneously. The smaller the elements, the more accurate the simulation but computing power will restrict the maximum number of elements that it is viable to incorporate in a model. The models built for this project contained up to 7 million elements.

The modelling was conducted using Ansys CFX2021 R1 with the geometry and mesh created using Ansys Design Modeler and Workbench. The models were solved using 32 × parallel processing on a cluster of multi-core workstations (Intel Xeon W-2145 3.7GHz processors). This gives a twenty-fold increase in solver speed compared to a single core and enables more detailed models to be solved than would otherwise be viable.

In order to predict the water surface, the model solves the flow of both the water and the air above. For this study, a homogeneous multi-phase model has been used which solves a single velocity field shared by both the water and air phase. This simplification is applicable for water bodies in which there is a clear separation between the air and water phase. It will not give good resolution of plunging flow where there is strong mixing of the phases. As the outfall was drowned in all the simulated scenarios, this method was considered appropriate for this study.

Temperature affects have not been simulated as representative boundary conditions for temperature are unknown. Temperature differentials between the river and the outfall discharge could have some influence on the flow path as could convection currents caused by heating of the water surface by the sun. However, the significance of temperature is probably secondary to that of the magnitude of the river and outfall flows.

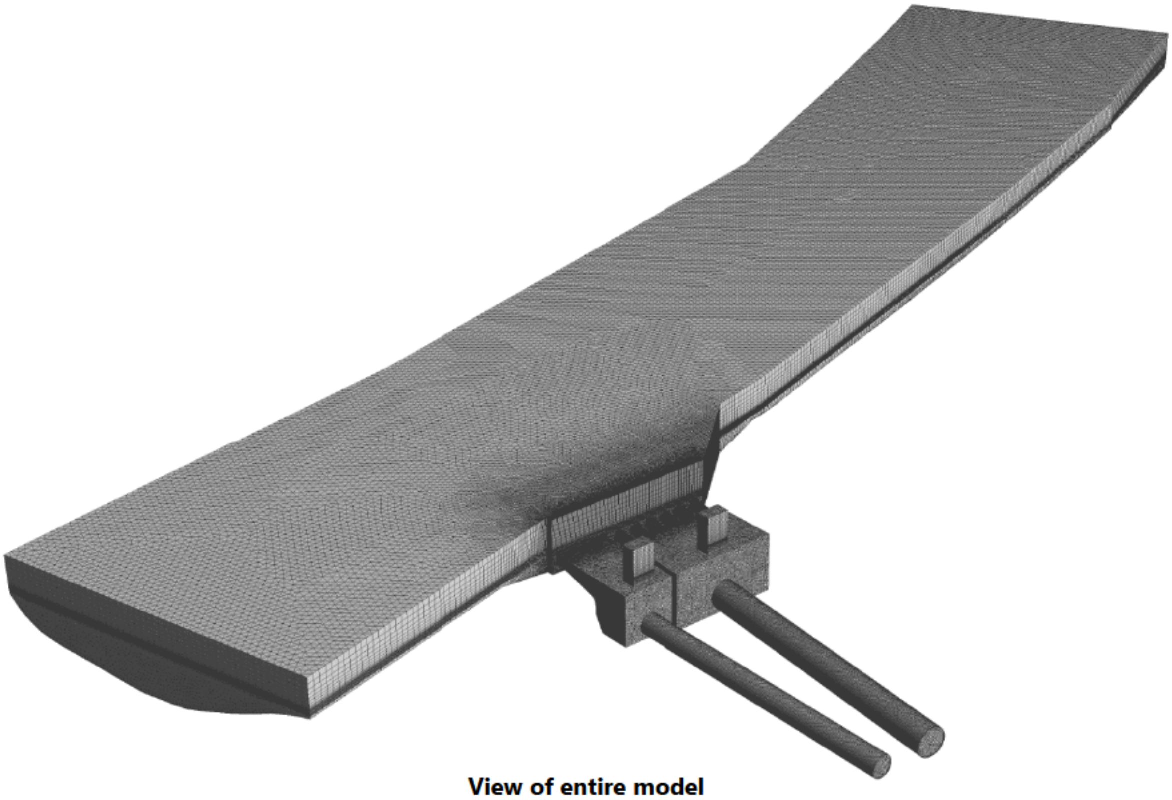
Further details of model setup are given in Appendix B.

3.2 Mesh setup

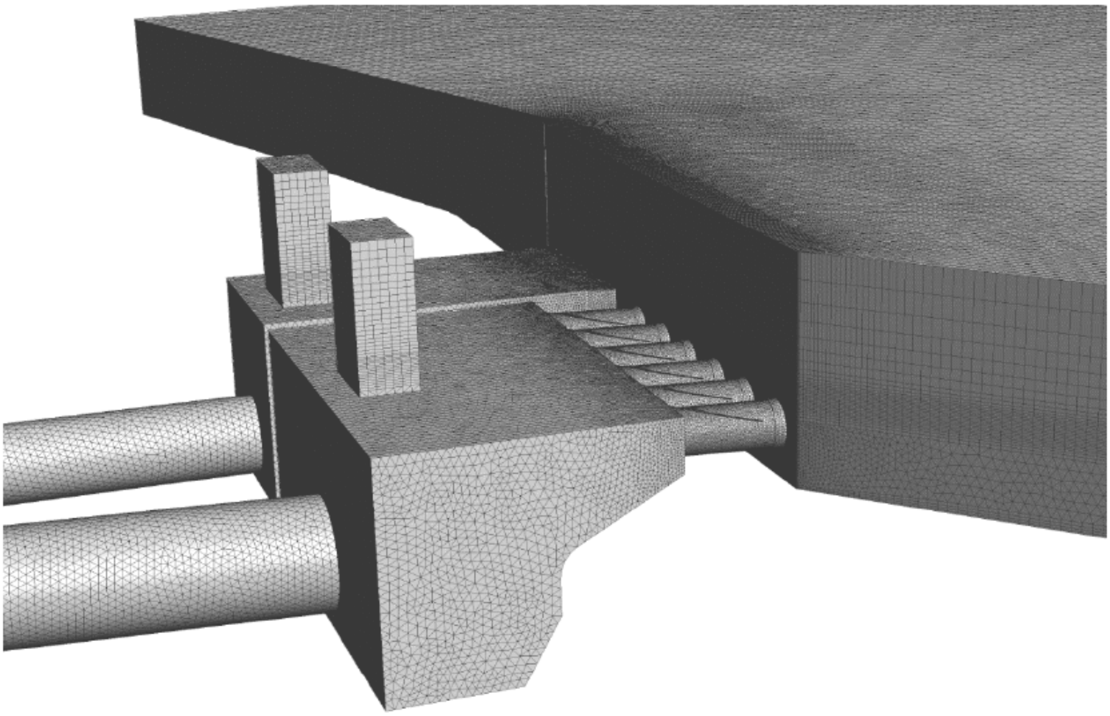
Models such as these with a large volume of retained water take a large number of computational iterations to fully converge on a stable solution, particularly for low flow cases. In order to reduce the overall model run time, two different meshes were used for each simulation. A relatively course mesh was used to get a good initial solution with a finer mesh used to complete each simulation.

The initial course mesh contained approximately 3 million elements and the fine mesh contained approximately 7 million elements. A typical fine mesh is shown in Figure 6. The mesh is predominantly tetrahedral to fit the complex detail of the outfall and the irregular riverbed. In the vicinity of the free surface, a swept mesh has been used to create thin horizontal layers which better resolve the water surface. In addition, thin inflation layers have been used on the walls of the outfall structure and the riverbed to give improved accuracy of wall boundary effects. General sizing for the refined mesh is as follows:

- Default for Air space: 1000mm
- Default for Water body: 250mm
- Outfall forebay: 150mm
- Chambers: 125mm
- Directly at outfall: 50mm
- Tideflex valves / outfall vanes: 35mm
- Water surface: 30mm layers



View of entire model



Zoomed in view of outfall

Figure 6 Typical model Mesh

3.3 General boundary conditions

The layout of the model is shown in Figure 7 with the model boundaries highlighted in different colours. Details about the setup of each boundary are given below:

- Inlets (**green**): The upstream boundaries for the river and outfall pipes were defined as fixed flow inlets, giving a uniform velocity across each boundary.
- Outlet (**Red**): The downstream river boundary was defined as an opening with a fixed hydrostatic pressure, allowing for flow to both enter and leave the model to maintain this level.
- Vents (**Blue**): Vents are included above the river inlet boundary and at the hatches in the outfall chamber. These are zero pressure openings that allow air to freely enter or exit the model domain. Without vents, the headspace of air in the model would pressurise if the water level rises or fall below atmospheric pressure if the water level falls giving unrealistic results.
- Top (Transparent): The top surface of the model above the river was defined as a free slip wall. This is computationally more stable than a vent opening across the entire top surface.
- Walls (**Brown**): The river channel and outfall structure were defined as no slip walls, with the following roughness values:
  - River channel: 20mm
  - Riprap: 10mm
  - Concrete walls: 0.6mm
  - Tideflex valve walls: 0.06mm

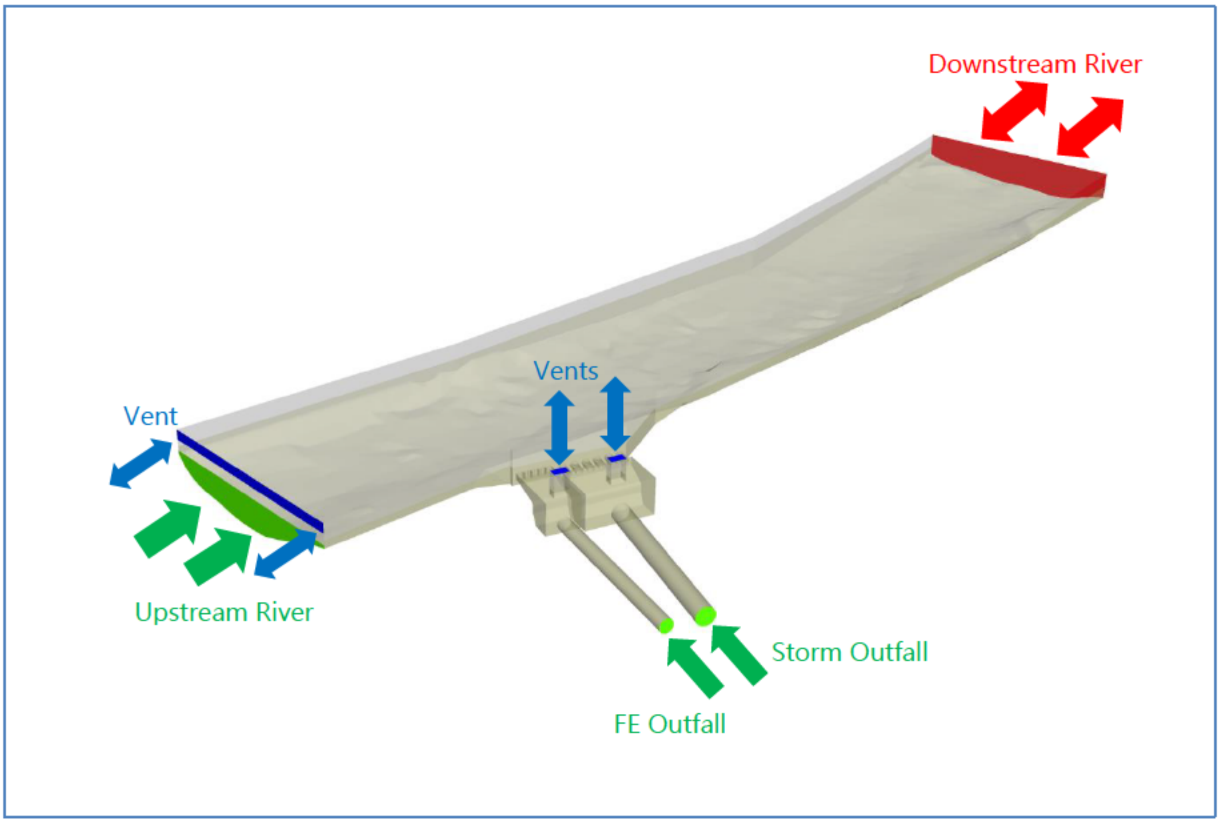


Figure 7 Model boundaries

3.4 Flow cases

The models in this study were simulated for three flow cases, as shown in Table 2:

Table 2 Flow cases

Case	Description	River inflow (m^3/s)	FE inflow (m^3/s)	Storm inflow (m^3/s)	Downstream river level (mAOD)
A	• Max FE • Max storm • 1:2yr river flow	21.8	2.0	5.0	3.965
B	• Max FE • Max storm • 50% exceedance river flow	2.43	2.0	5.0	3.840
C	• Max FE • Zero storm • 50% exceedance river flow	2.43	2.0	0.0	3.840

4. Results

In the following sections, the results are presented by flow case to allow direct comparison between the different layouts tested.

The results, for each of the flow cases (and layout), include the following outputs from the CFD model:

— Water surface near the outfall (Figures 8, 14 and 20)

— Velocity at the outfall ports (Figures 9, 15 and 21)

— Velocity at the water surface (Figures 10, 16 and 22)

— Velocity near the river floor (Figures 11, 17 and 23)

— Streamlines in the River (Figures 12, 18 and 24)

— Sections across the river (Figures 13, 19 and 25)

4.1 Flow case A

Flow Case A (Table 3) is an infrequent, extreme case since the storm outfall is expected to operate approximately once every 10 years [Ref 5].

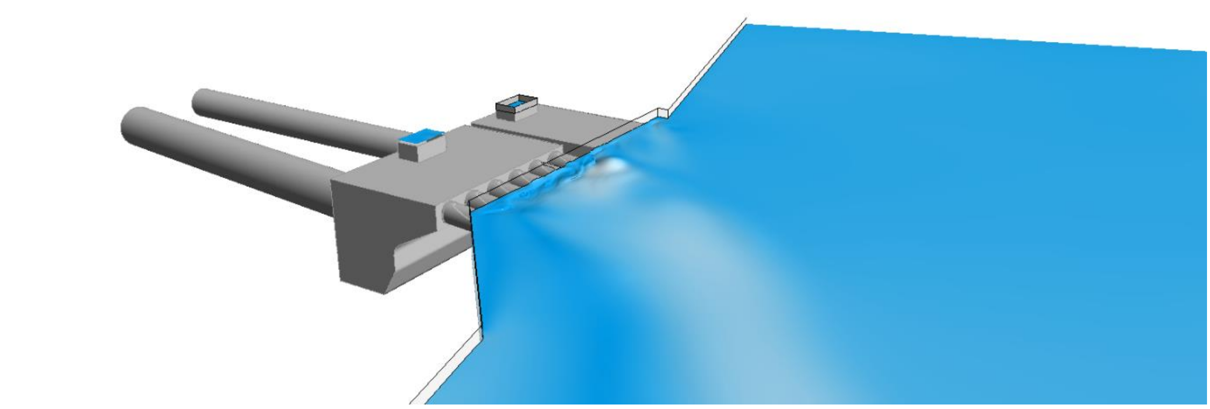
Table 3. Flow case A

Case	Description	River inflow (m^3/s)	FE inflow (m^3/s)	Storm inflow (m^3/s)	Downstream river level (mAOD)
A	• Max FE • Max storm • 1:2yr river flow	21.8	2.0	5.0	3.965

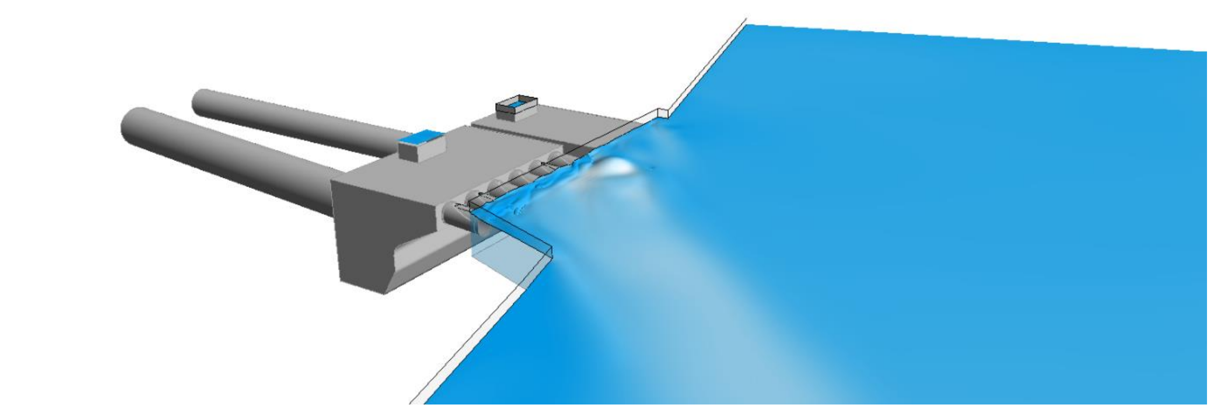
The water surface in the vicinity of the outfall is shown in Figure 8. Some disturbance is predicted immediately downstream of the storm outlets, but it is relatively modest with a maximum upwelling at the water surface of less than 200mm. The upwelling is reduced for layout 04, which we attribute to the outfalls being closer to the deeper river channel rather than discharging into a shallow forebay.

The velocity directly at each outlet port is shown in Figure 9. The FE flow is reasonably balanced between each of the FE outlet ports, and likewise the storm flow is reasonably balanced between the storm outlet ports. However, the peak velocities at the storm outlets is significantly higher at the storm outlets than at the FE outlets, the implications of this are discussed further in Section 5.

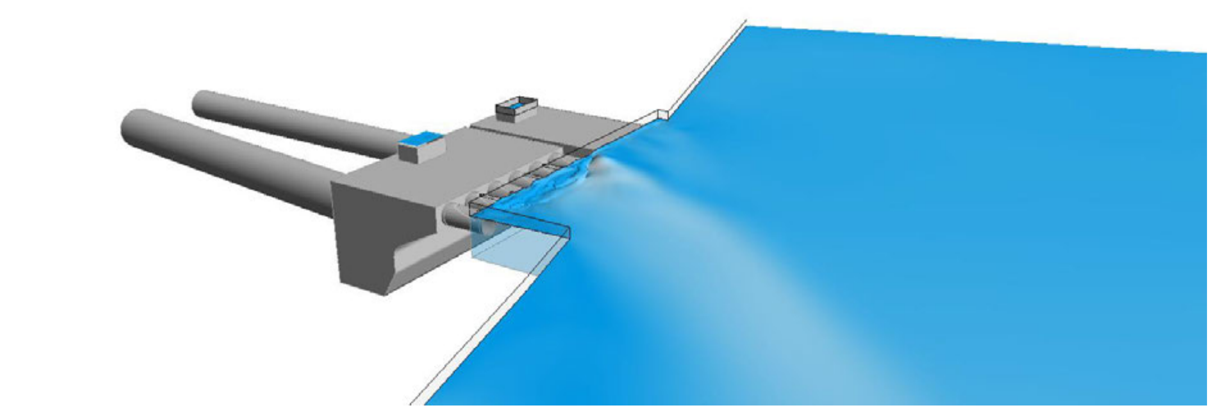
The velocity and flow path are shown by Figure 10 through Figure 13. In all cases, the outfall jet gets turned by the river flow and does not impact directly on the far bank. The flow pattern and magnitude of velocity is broadly similar for each layout, although there is a larger recirculating zone downstream of the outfall for Layout 400. The plume from the outfall has a wavy alignment as it passes downstream indicating that it is not steady and the jet will oscillate from side to side.



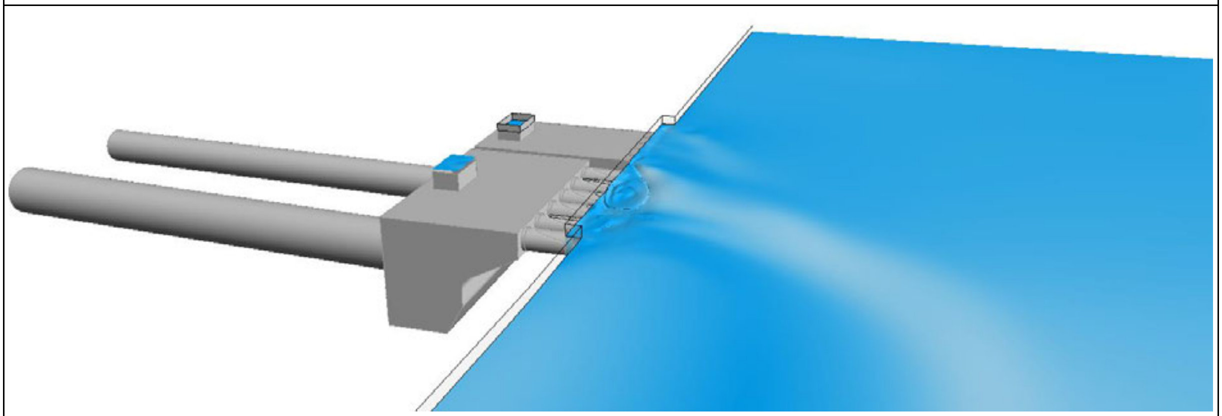
Layout 100



Layout 200



Layout 300



Layout 400

Figure 8 Water surface near outfall – Flow case A

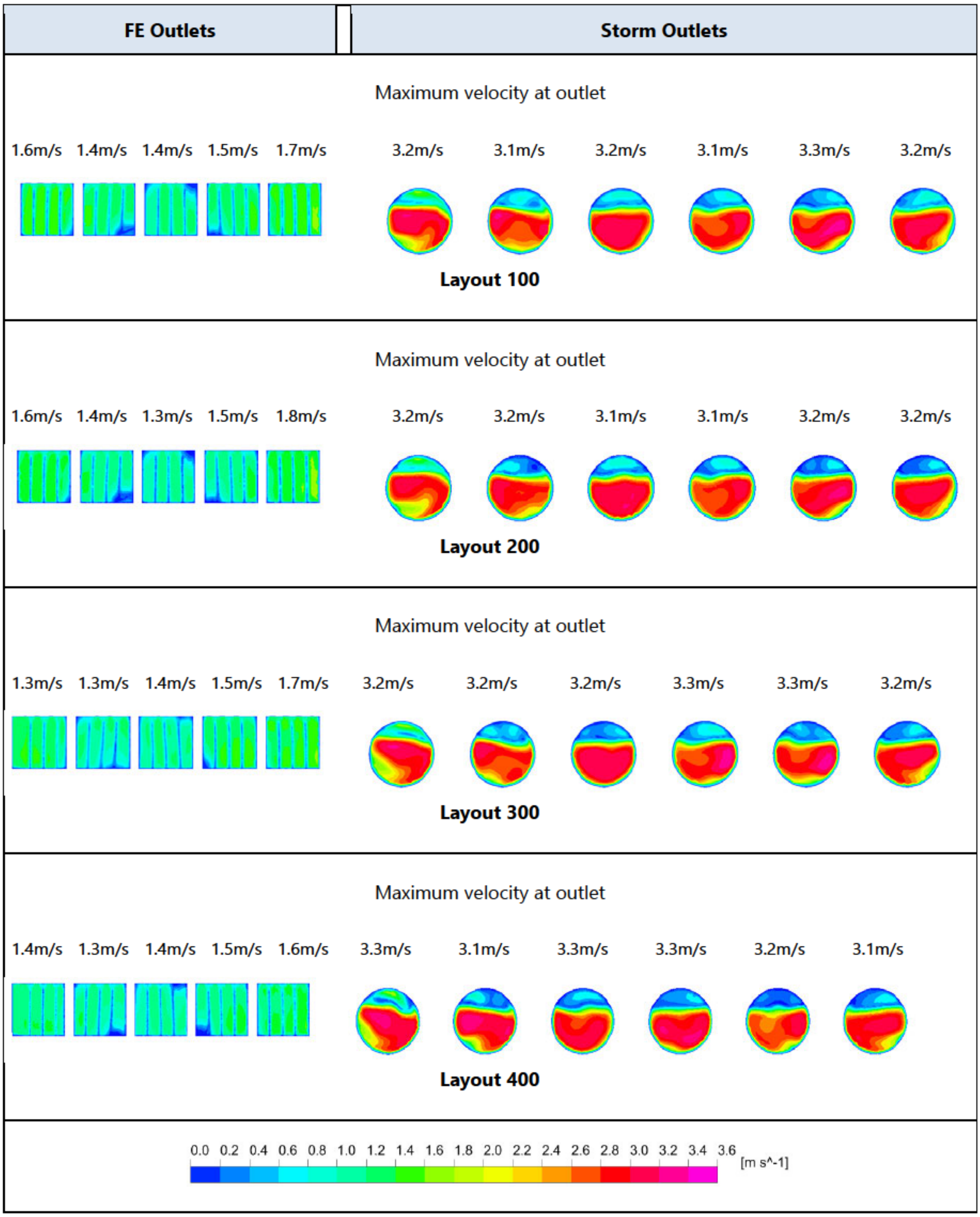


Figure 9 Velocity at outfall ports – Flow case A

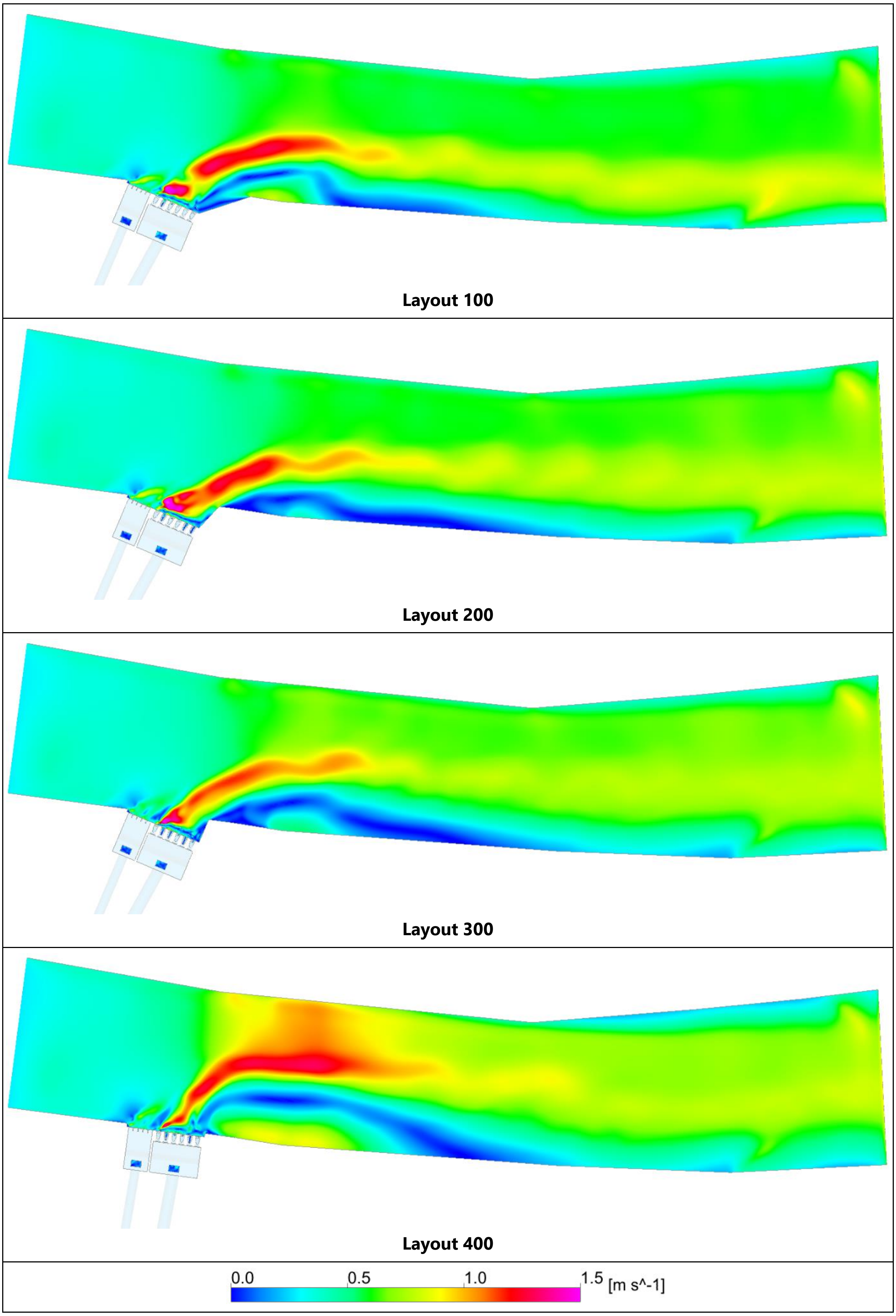


Figure 10 Velocity at water surface – Flow case A

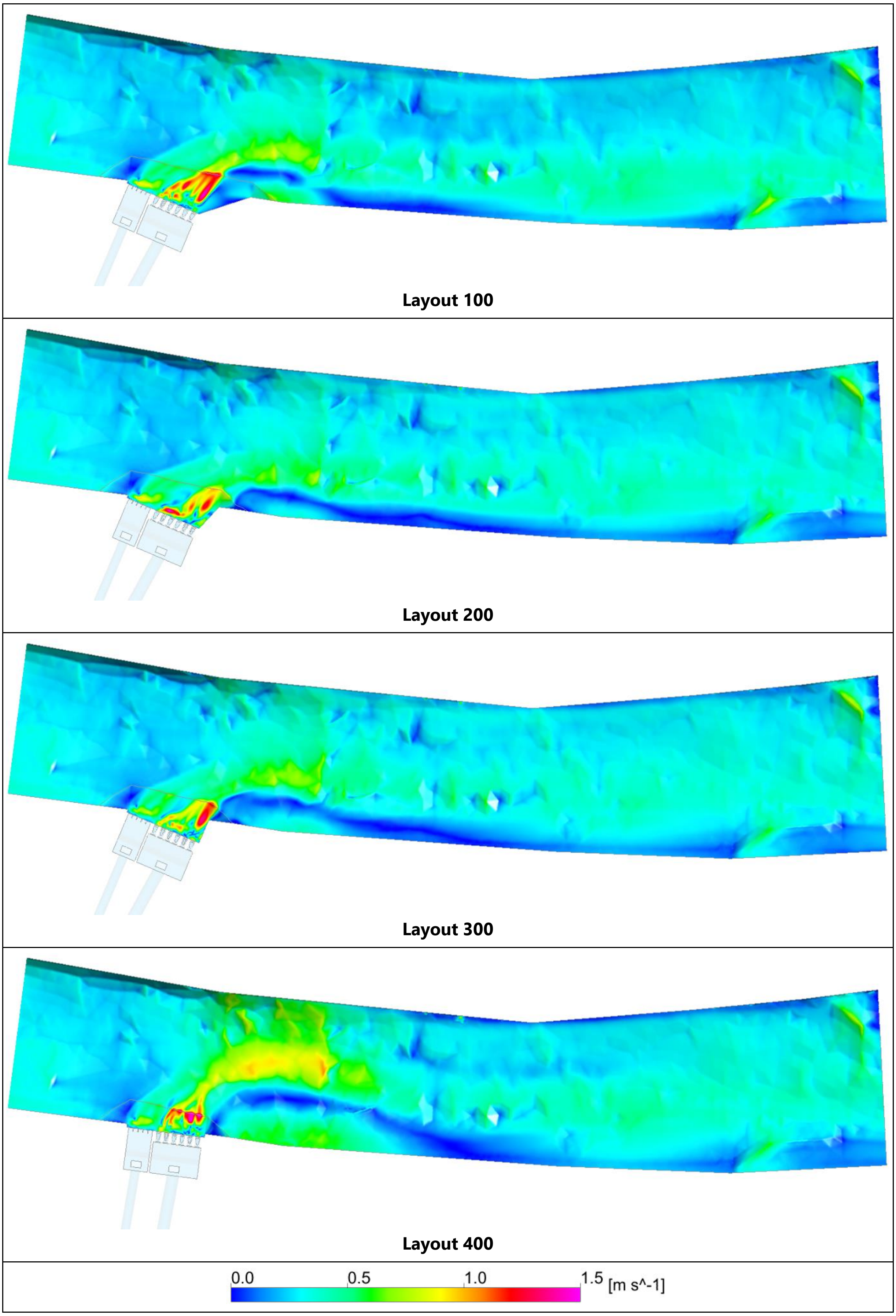


Figure 11 Velocity near floor – Flow case A

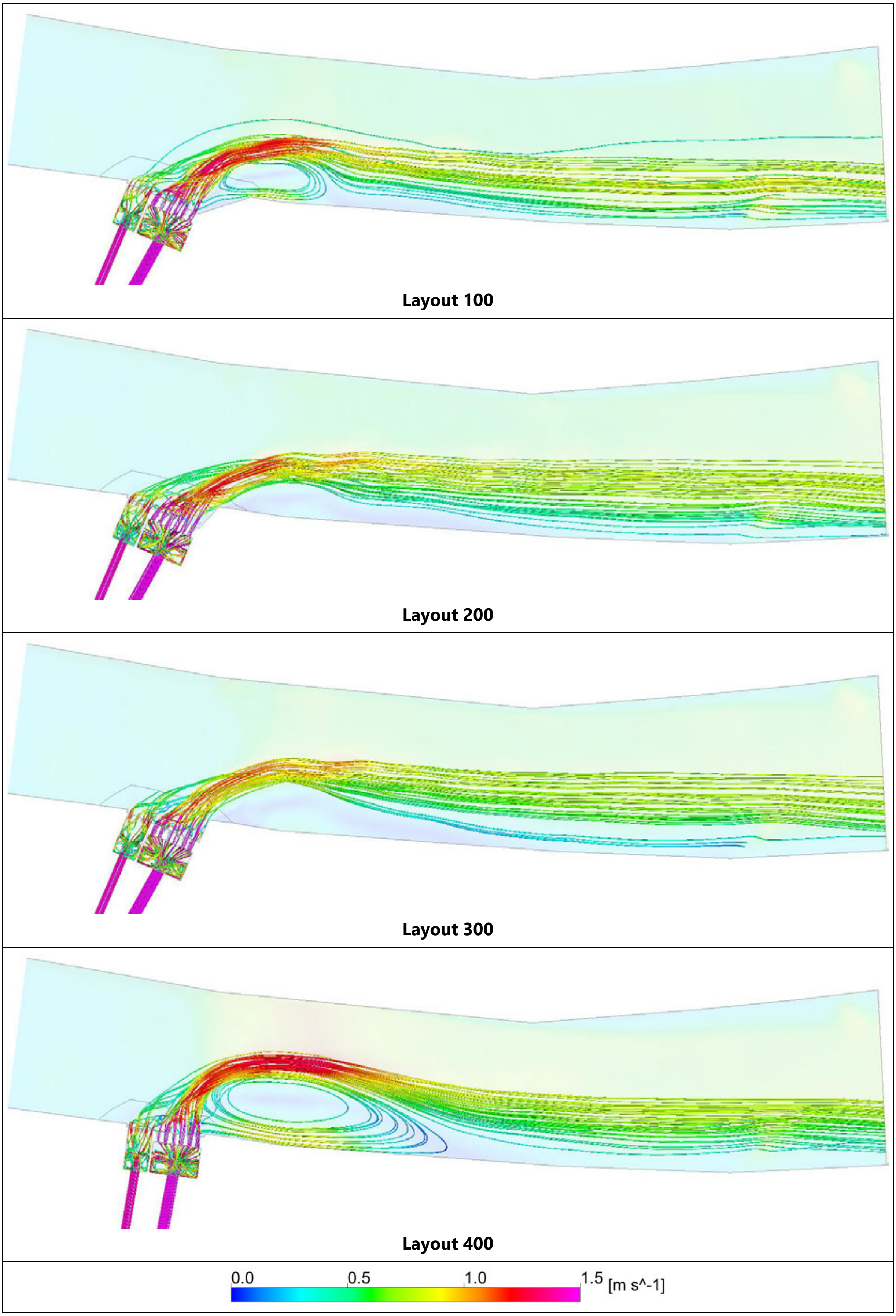


Figure 12 Streamlines – Flow case A

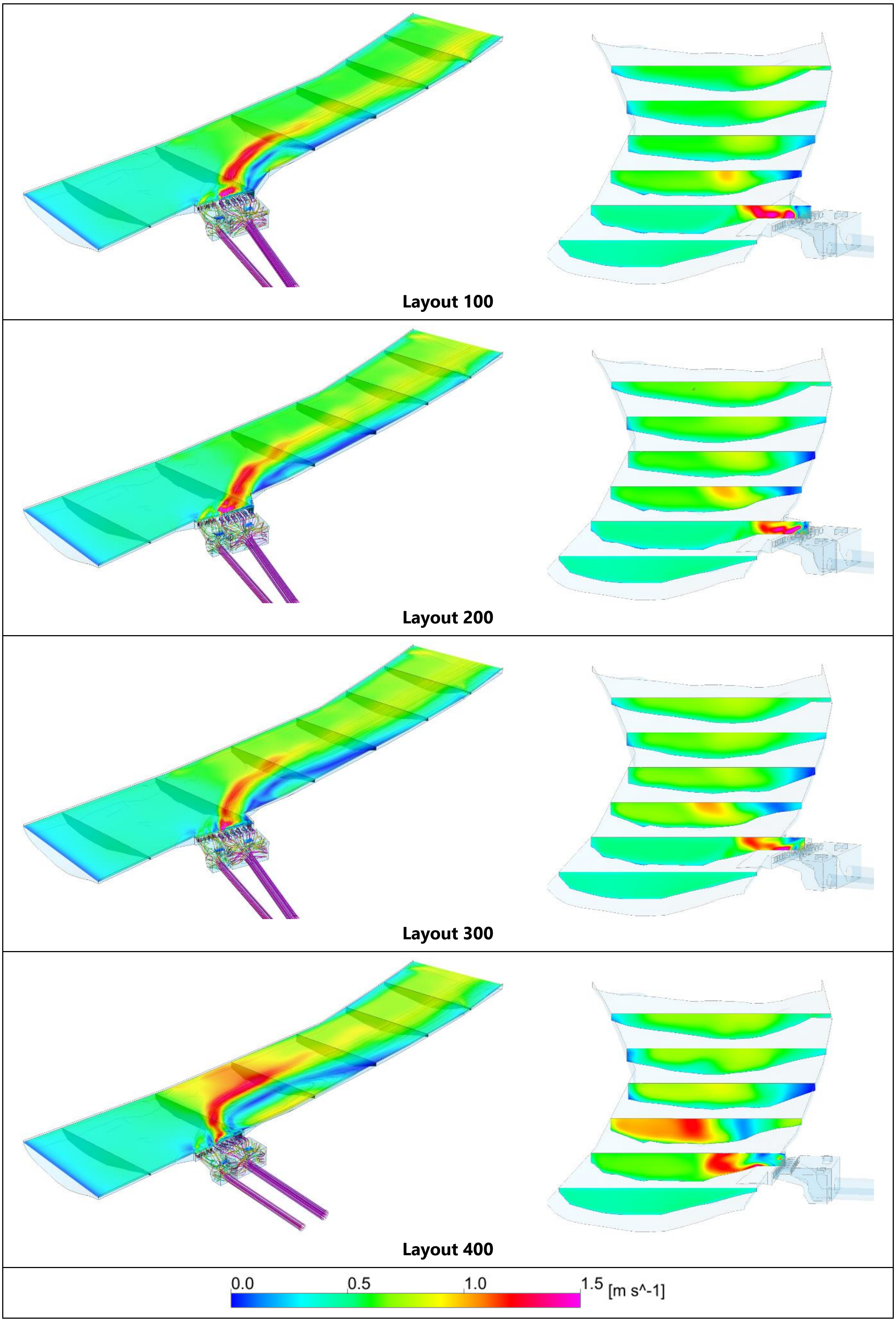


Figure 13 Sections – Flow case A

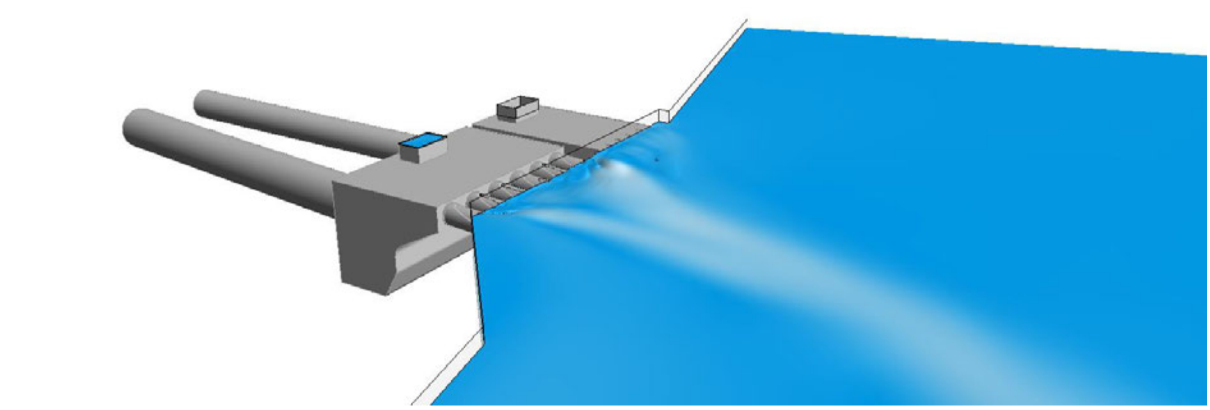
4.2 Flow case B

Flow Case B (Table 4) has the same outfall flow as case A, but in conjunction with a lower river flow. The river flow will tend to turn the outfall jet and help re-align it along the river channel. This case with maximum outfall flow, but a more typical river flow is therefore expected to give worse case conditions in terms of the outfall jet impacting on the far bank and causing erosion.

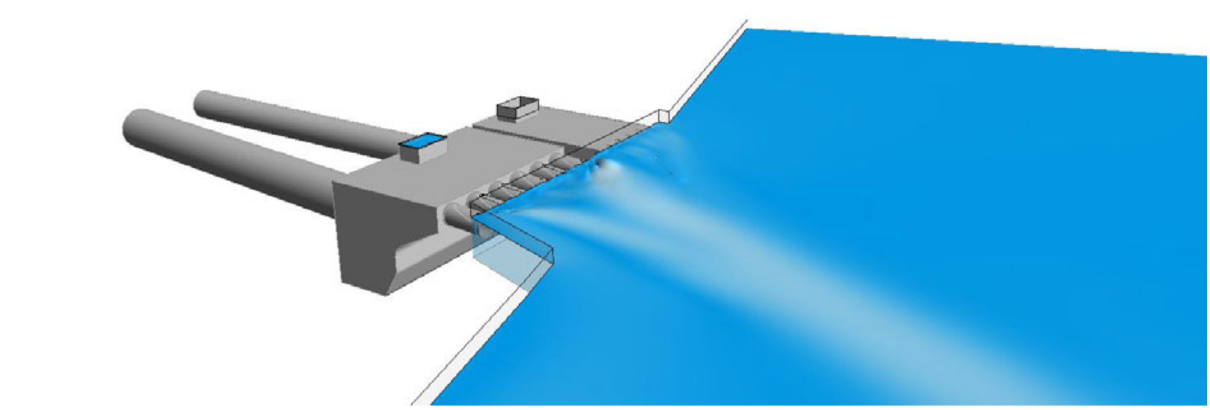
Table 4. Flow case B

Case	Description	River inflow (m^3/s)	FE inflow (m^3/s)	Storm inflow (m^3/s)	Downstream river level (mAOD)
B	• Max FE • Max storm • 50% exceedance river flow	2.43	2.0	5.0	3.840

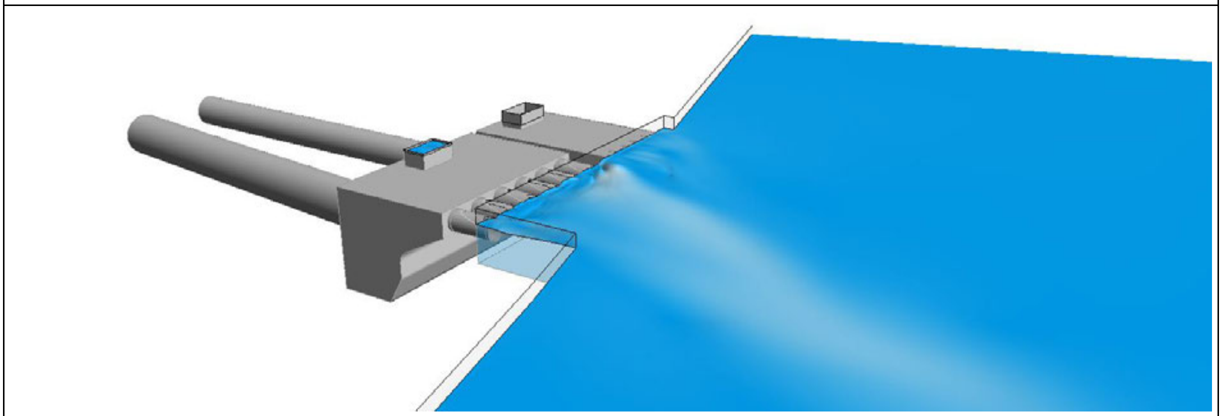
The results are shown in Figure 14 to Figure 19. The extent of disturbance at the outfall and the peak velocities are very similar to flow case A. However, the outfall jet now passes across the full width of the river, giving increased velocities along the far bank. This is particularly notable for layout 400 in which the outfall jet impacts the far bank. Nevertheless, if this scenario occurs only once every 10 years [Ref 5] then the cumulative erosion should be low.



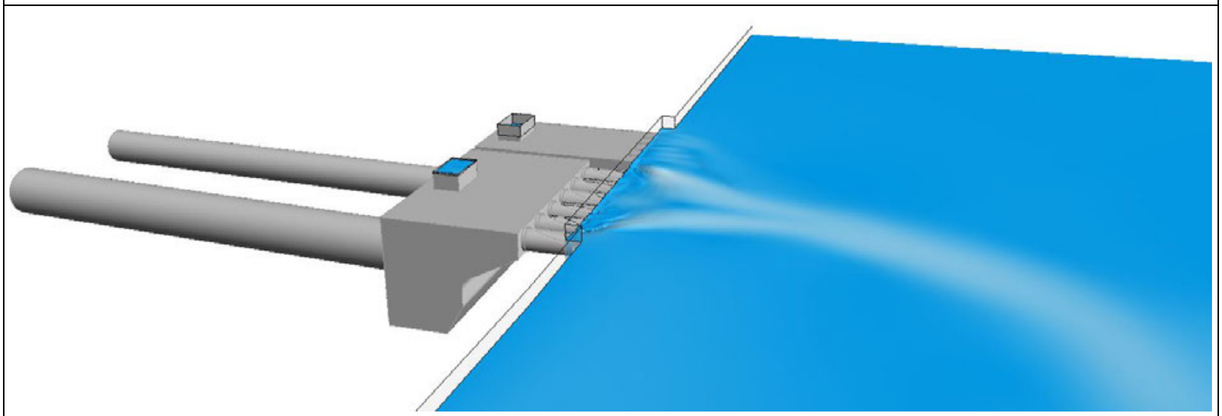
Layout 100



Layout 200



Layout 300



Layout 400

Figure 14 Water surface near outfall – Flow case B

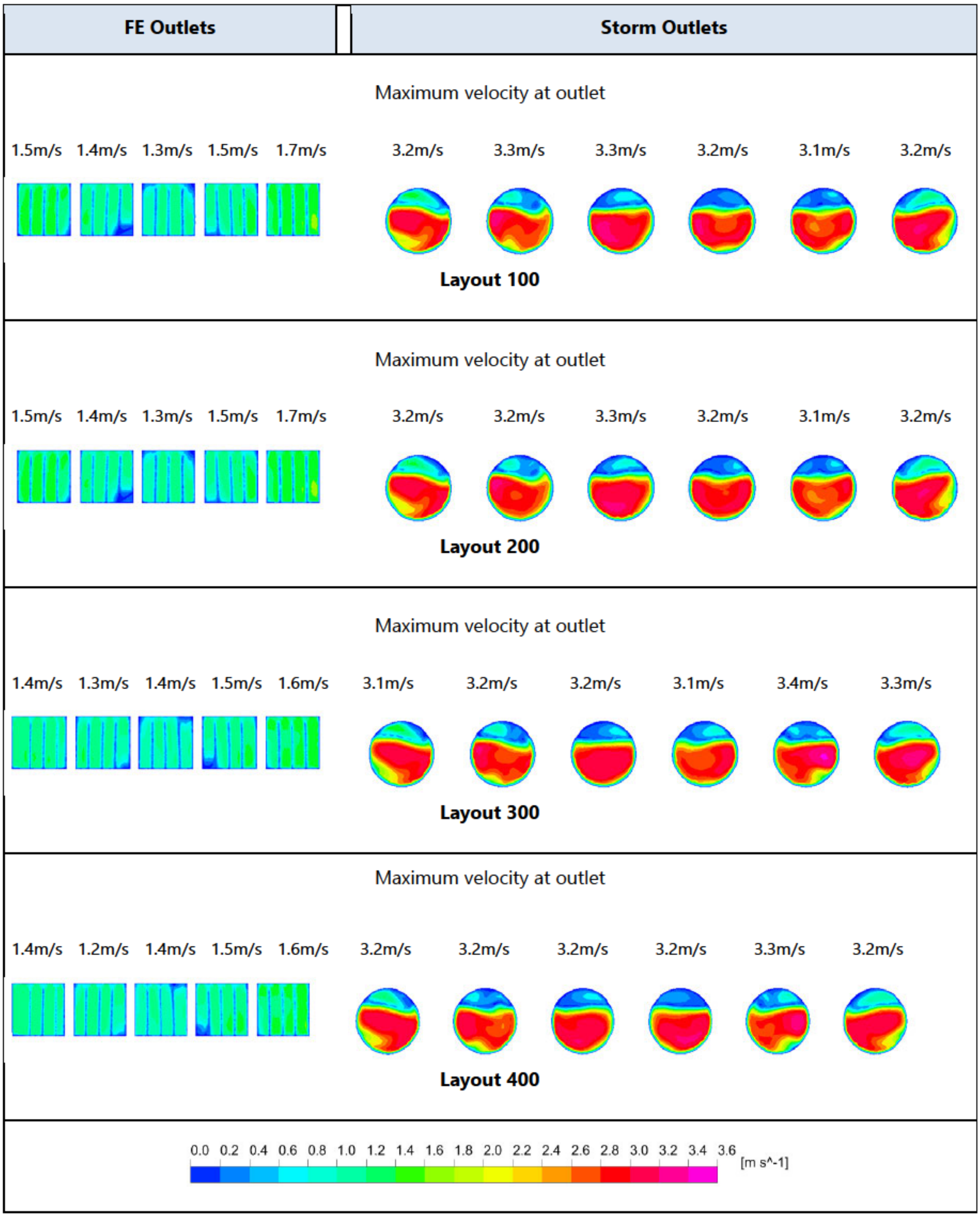


Figure 15 Velocity at outfall ports – Flow case B

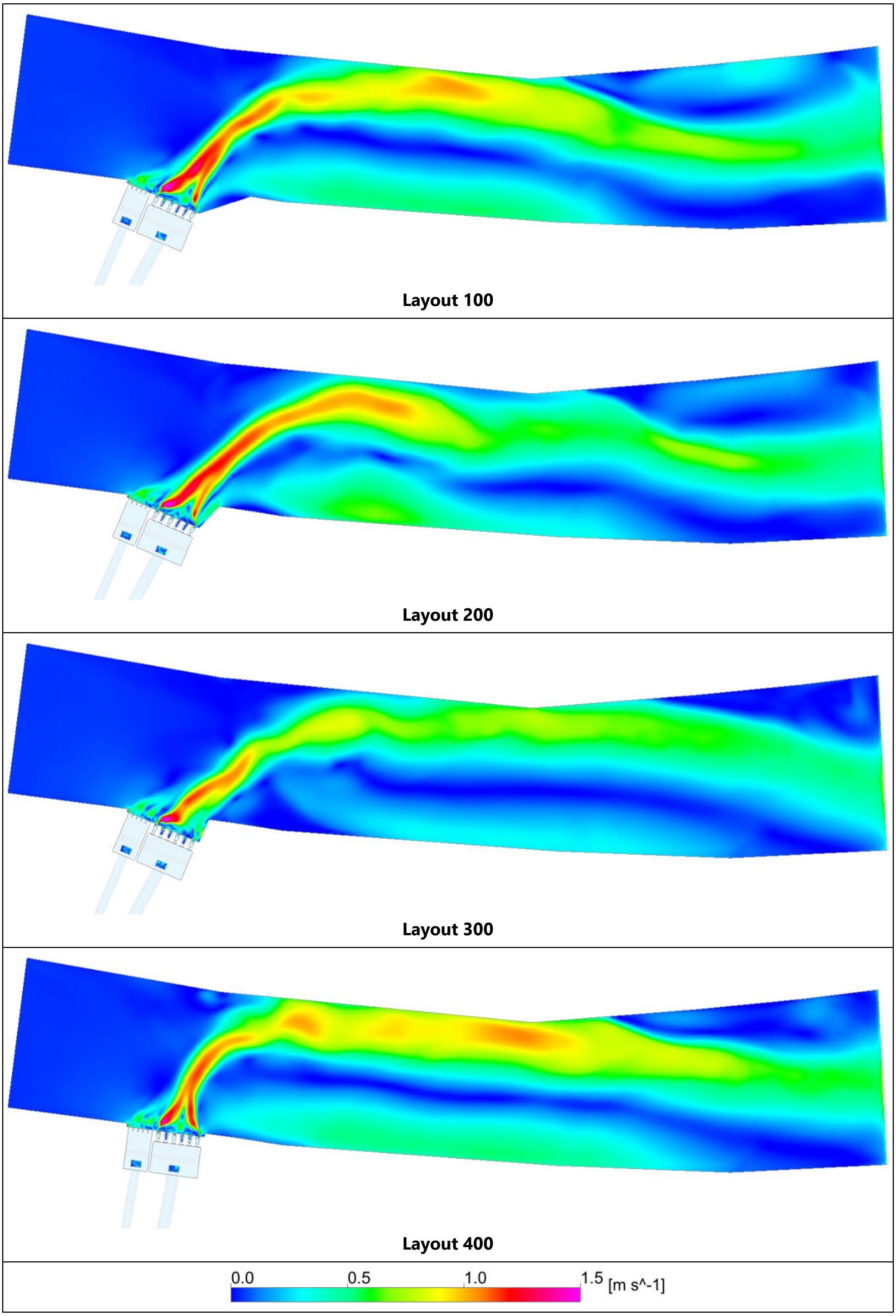


Figure 16 Velocity at water surface – Flow case B

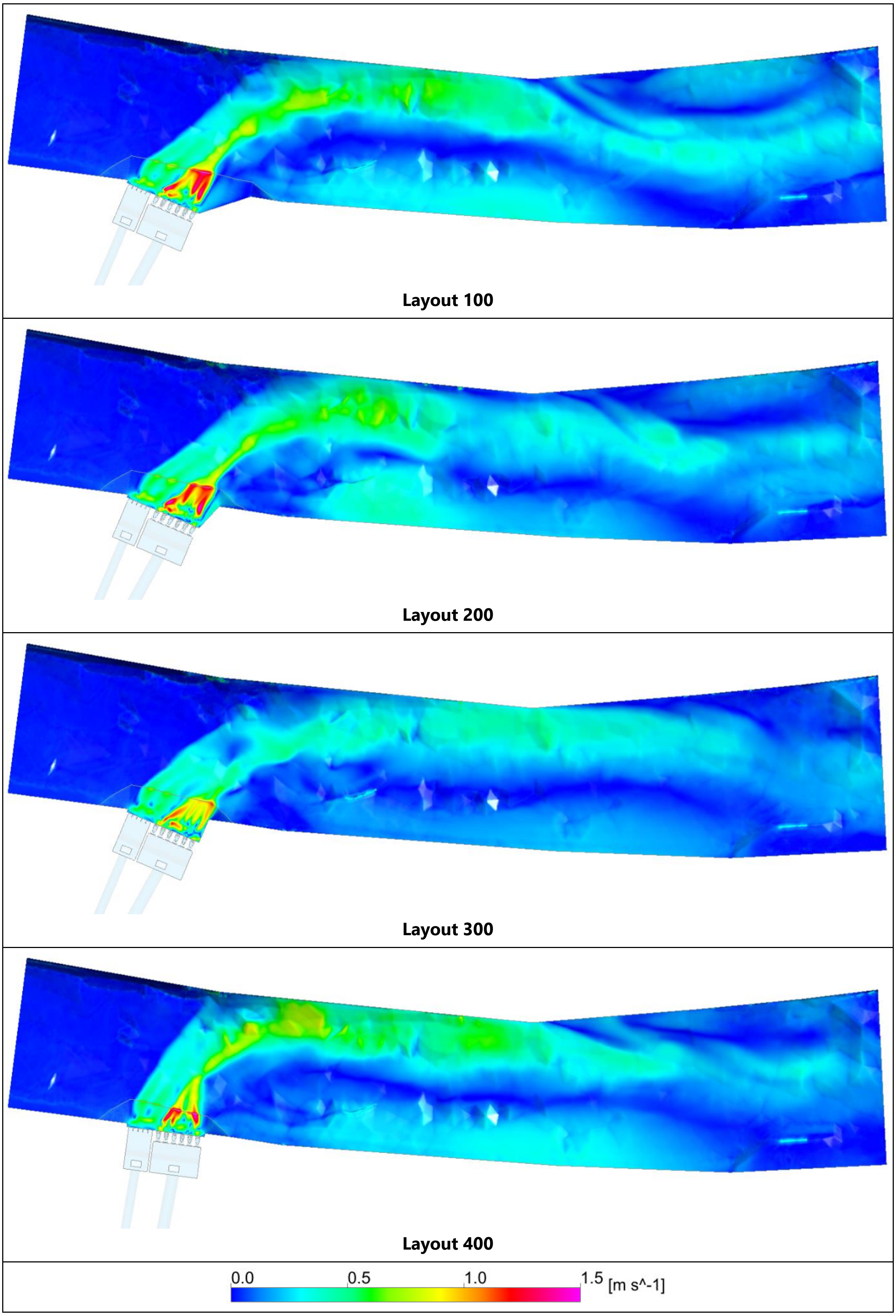


Figure 17 Velocity near floor – Flow case B

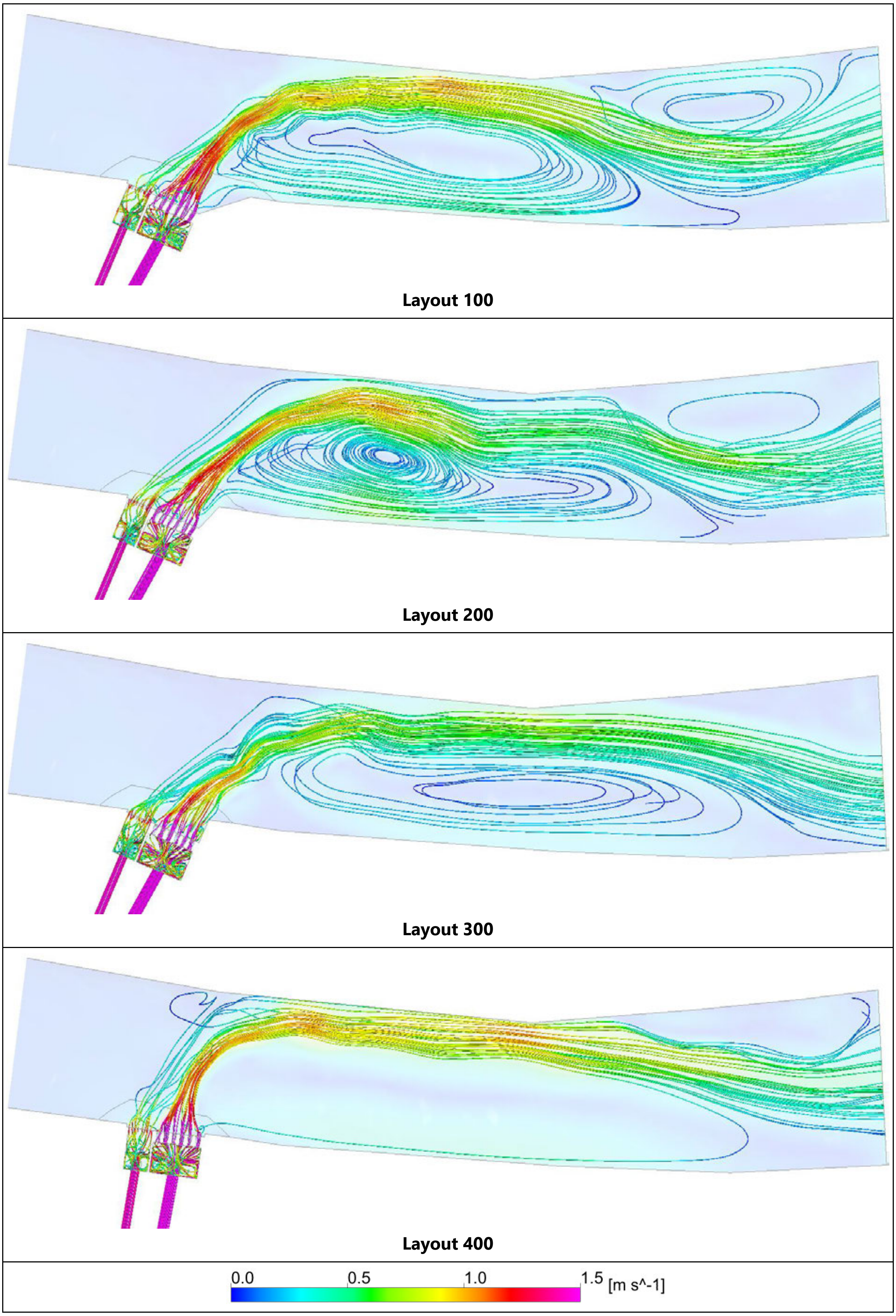


Figure 18 Streamlines – Flow case B

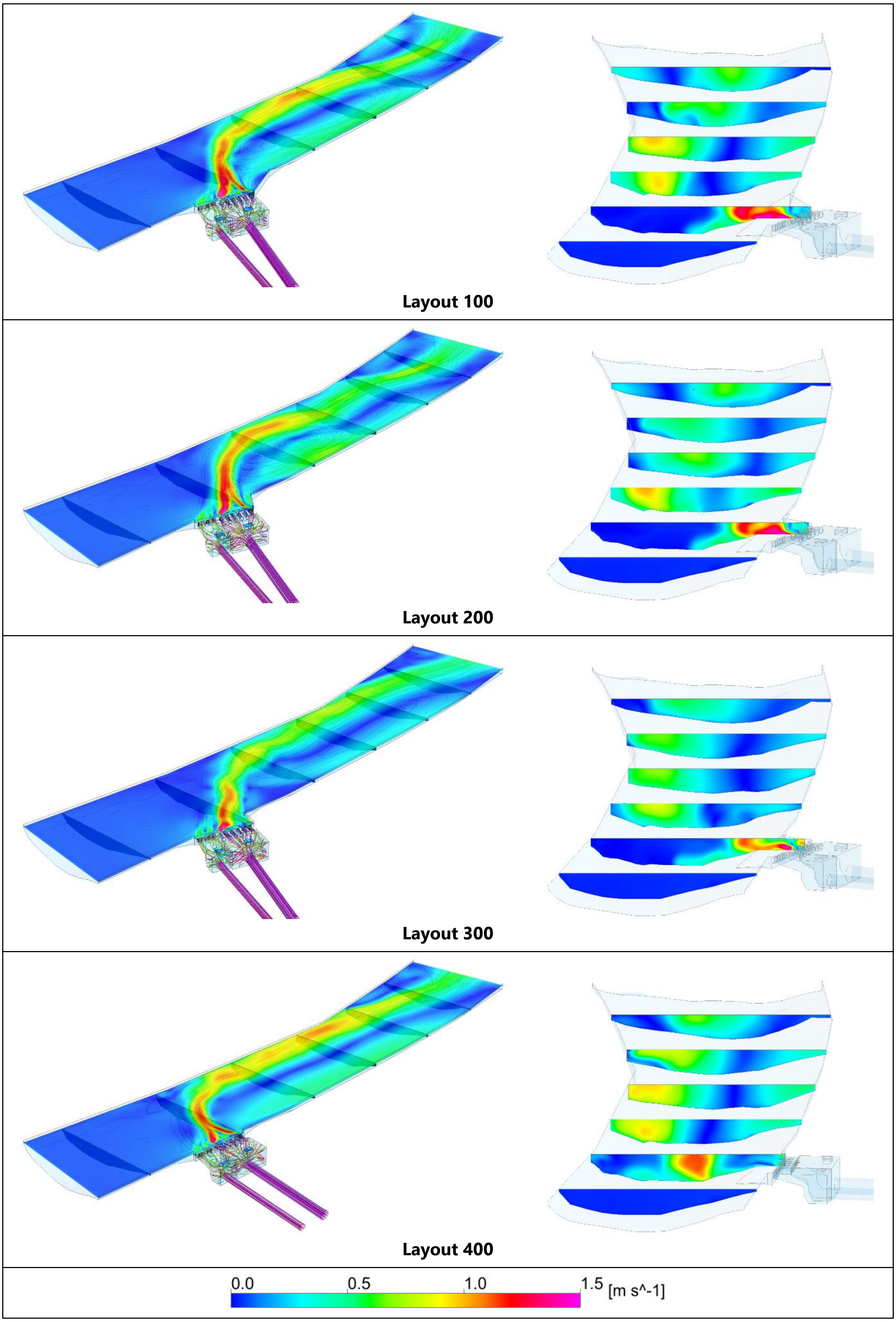


Figure 19 Sections – Flow case B

4.3 Flow case C

Flow Case C (Table 5) is a more typical case with the FE outlet operating but with no storm flow.

Table 5. Flow case C

Case	Description	River inflow (m^3/s)	FE inflow (m^3/s)	Storm inflow (m^3/s)	Downstream river level (mAOD)
C	• Max FE • Zero storm • 50% exceedance river flow	2.43	2.0	0.0	3.840

The results are shown in Figure 20 to Figure 25. The extent of disturbance at the outfall is minimal with no significant upwelling at the water surface. Wind or passing craft are likely to cause greater surface disturbance.

Due to the low river velocities, the outfall jet still passes across the full width of the river, giving increased velocities along the far bank. However, the magnitude of these velocities is relatively low at less than 0.5m/s.

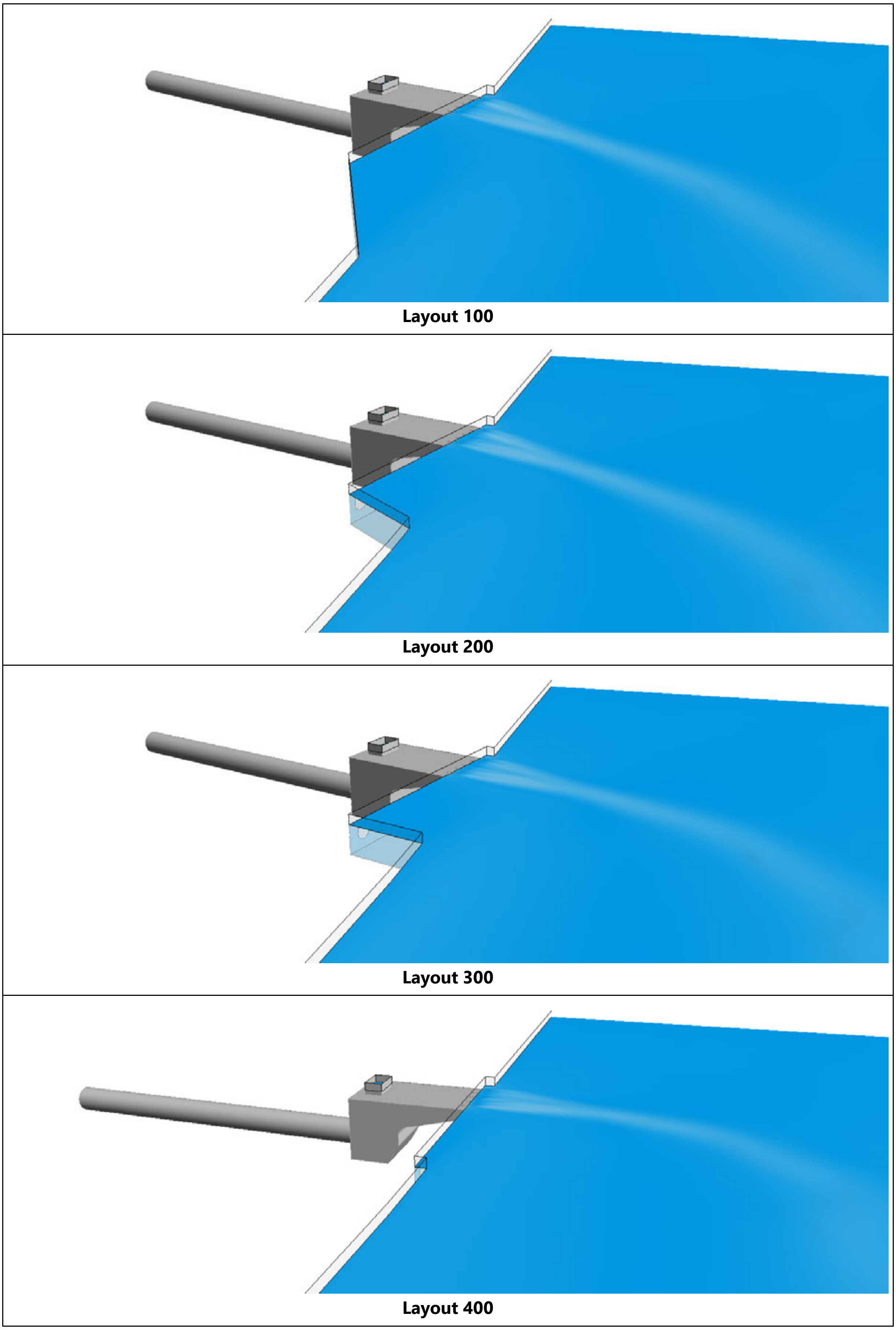
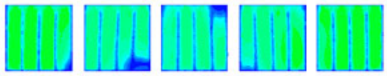


Figure 20 Water surface near outfall – Flow case C

FE Outlets	Storm Outlets (offline)

Maximum velocity at outlet

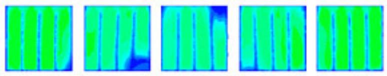
1.6m/s 1.4m/s 1.3m/s 1.5m/s 1.6m/s



Layout 100

Maximum velocity at outlet

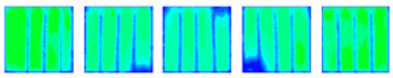
1.6m/s 1.4m/s 1.3m/s 1.5m/s 1.5m/s



Layout 200

Maximum velocity at outlet

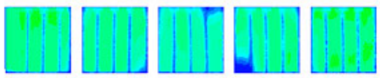
1.4m/s 1.3m/s 1.5m/s 1.4m/s 1.5m/s



Layout 300

Maximum velocity at outlet

1.4m/s 1.3m/s 1.4m/s 1.4m/s 1.5m/s



Layout 400

Figure 21 Velocity at outfall ports – Flow case C

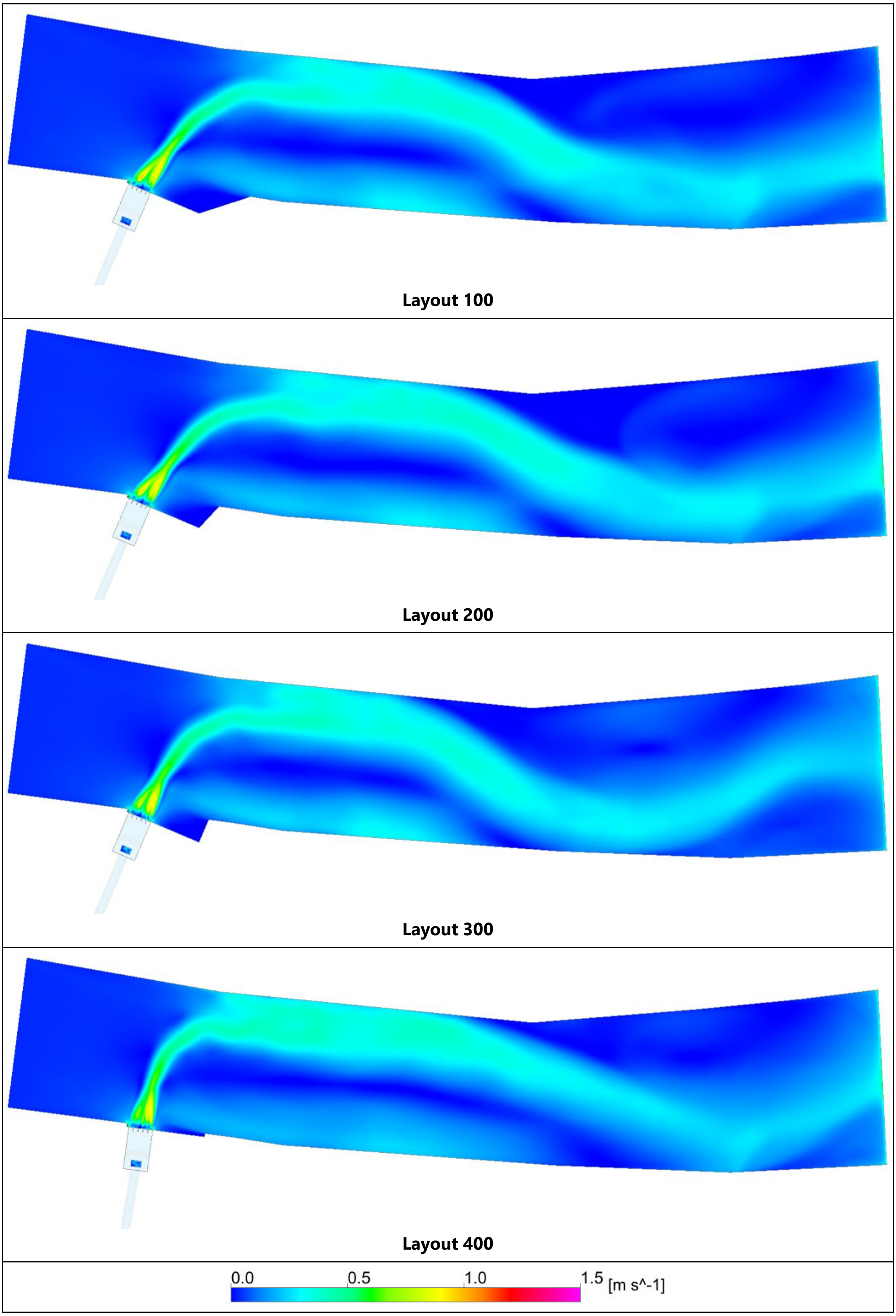


Figure 22 Velocity at water surface – Flow case C

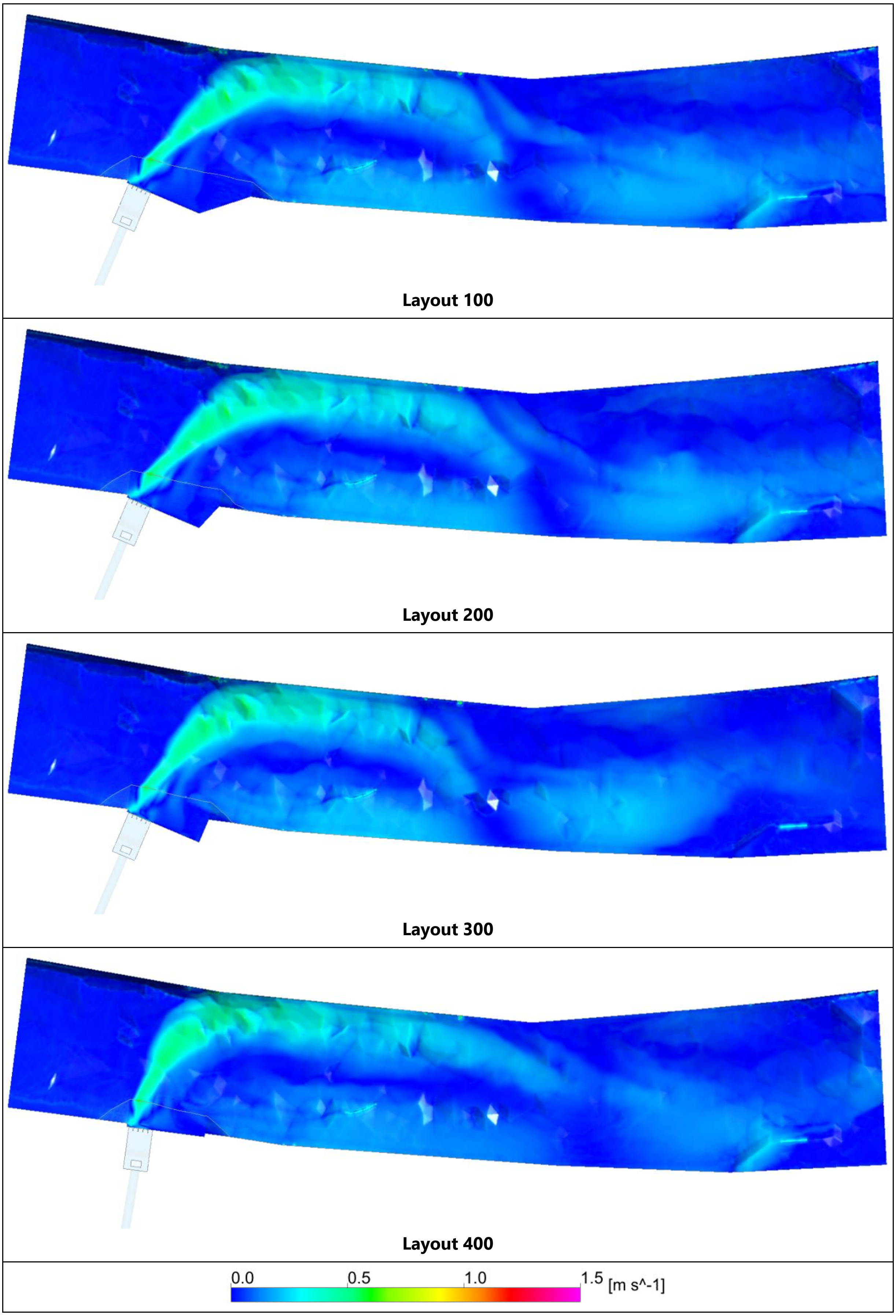


Figure 23 Velocity near floor – Flow case C

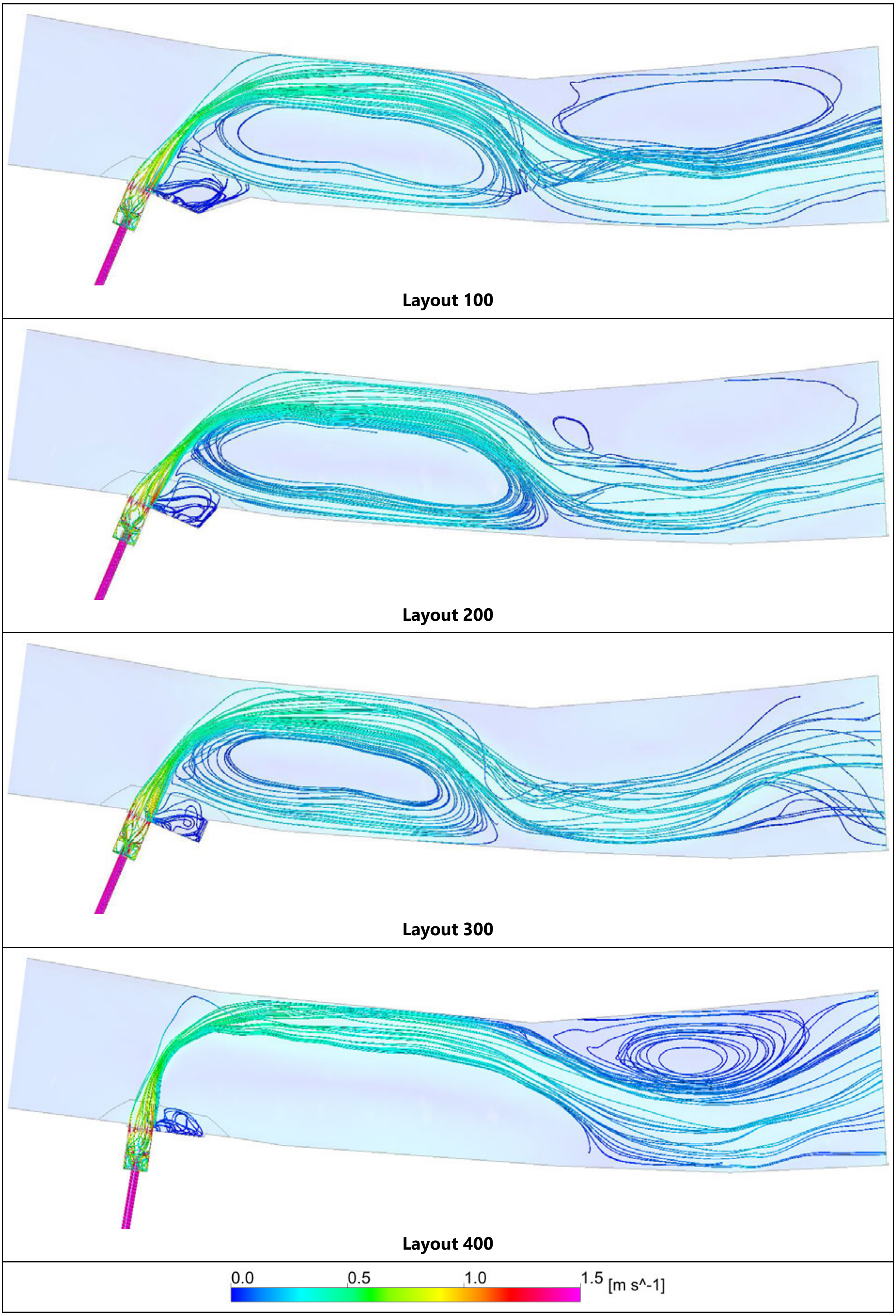


Figure 24 Streamlines – Flow case C

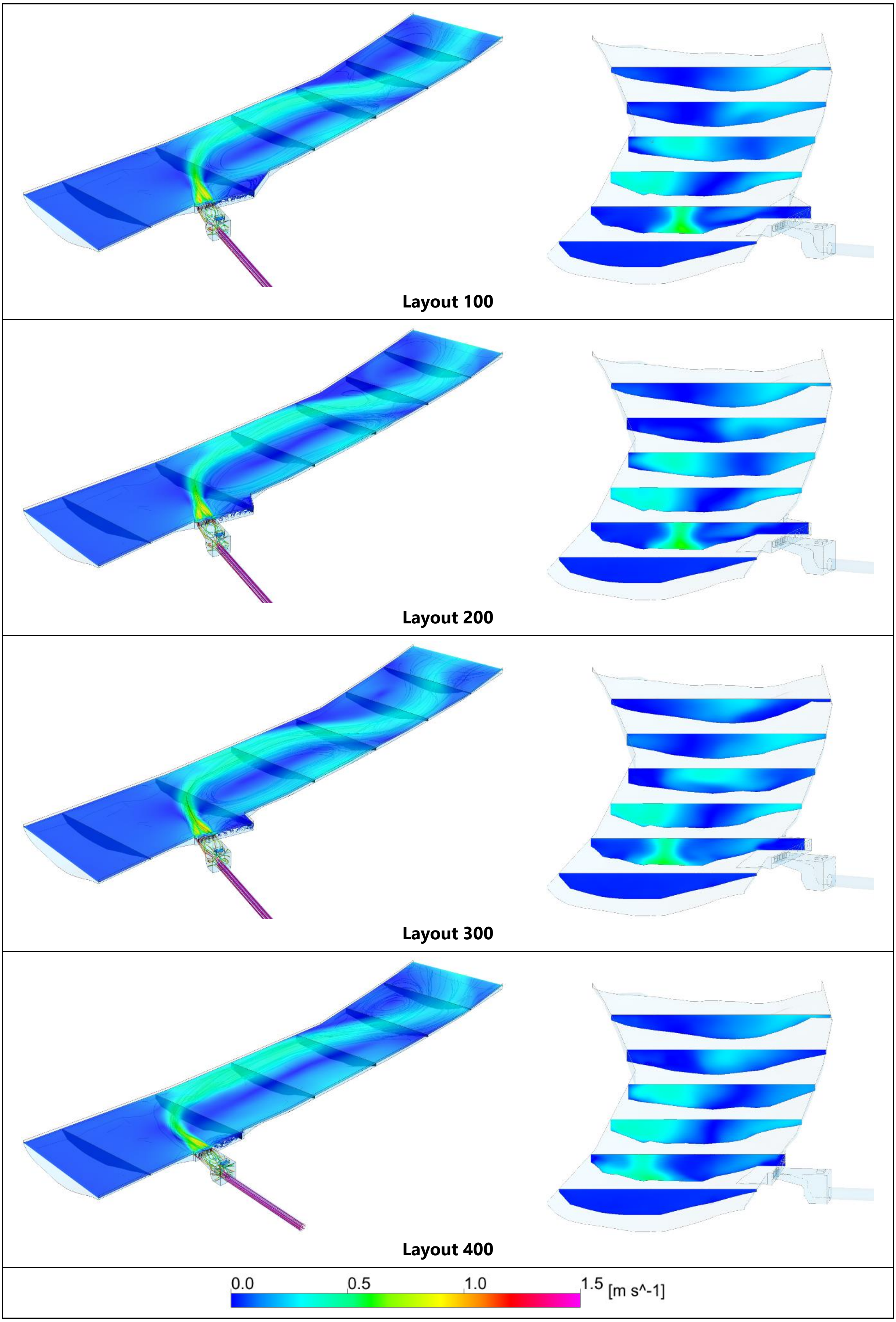


Figure 25 Sections – Flow case C

5. Discussion

The first three layouts tested (Layout 100, 200 and 300) give broadly similar performance in terms of velocities and flow patterns for each flow case. Layout 200 appears to give the best dispersion of the flow from the outfall, but the benefit over Layouts 100 and 300 is small and subjective. The last layout tested (Layout 400) with the outfalls directly on the river wall is less effective as the outfall flow passes across the river and impacts the far bank giving an increased risk of erosion.

The water level at the outfall location is maintained by the weir at Baits Bite Lock (approximately 500m downstream). This managed water level results in average velocities within the river well below 0.1m/s under typical conditions. Even with the 1:2 year river flow, the average velocity remains below 0.5m/s. Hence, any currents from the outfall are likely to be perceptible to lightweight passing craft such as kayaks, canoes, stand-up paddle boards or rowing boats.

The storm outfall is only expected to operate once every 10 years [Ref 5], hence flow case A and B (with the storm outfall operating at maximum flow) are infrequent cases. Flow case C represents a more typical condition that could occur for extended periods.

Under Flow case C, there is no discernible disturbance of the water surface adjacent to the outfall. The peak velocities at the water surface are 0.9m/s and remain above 0.5m/s until approximately mid-way across the river. Velocities of this magnitude would be perceptible to lightweight craft but are not abnormal.

With the storm outlets operating (Flow case A or B) there is a small disturbance of the water surface near the outfall, but the upwelling at the water surface is less than 200mm. The peak velocity at the water surface is 1.6m/s and a current exceeding 1m/s spreads across most of the width of the river. This strength of this current would have an impact on lightweight craft. Although the operation of the storm outfall is infrequent it is foreseeable that small recreational craft could be using the river under these conditions and so the acceptability and safety implications should be reviewed. **This should be added to the Risk Register**.

Options for reducing the velocities under storm conditions that could be investigated further are:

- Increasing the depth of the outfall bay
- Increasing the submergence of the outfall ports
- Increase the port area for the storm outfalls.

It should be noted that increasing the diameter of the Checkmate valves may not be effective for increasing the port area. At the storm flow of 5.0m^3/s, giving 0.83m^3/s per valve, the 750mm checkmate valves will be operating close to their “snap open” flow. If the valves are increased in size, the “snap open” flow will increase, and it becomes likely that not all of the valves will open. It is not possible to be specific about this as the manufacturer’s information on the snap open flow is only indicative. We therefore suggest reverting to an outlet details similar to the FE outlets, but with the addition of a mid-weight (e.g. timber) flap gate.

CIRIA C786 Section 12.5 [Ref 3] has been used as a reference to provide general design guidance for the outfall arrangement and protection requirements. However, the bespoke nature of the outfall and the local site constraints have limited the use of the guide and the design process is therefore focussed on the output of the CFD model. In terms of velocity the likely impact on the river users and natural environment is summarised as follows:

Impact on River Users:

— The typical condition with just the FE compartment operating (Flow Case C) gives no discernible disturbance of the water surface, but results in velocities of between 1.0 and 0.5m/s extending to mid width of the river. These currents could have some influence on small craft.

— The flows from the storm outfall are focussed by the non-return outfall ports resulting in significantly higher velocities when these are operating. Consideration of alternatives (such as rectangular flap gates with larger openings) is recommended to identify an option that would give lower velocities.

Scour of the riverbed:

— Except in the immediate vicinity of the outfall (where scour protection is intended) the velocities at the riverbed (as shown in Figures 11, 17 and 23) are well below 1.0m/s and excessive scour is not expected; this applies to all three flow cases.

Damage to the riverbanks:

— The flows from the FE outfall compartment indicate velocities of approximately 0.5m/s or less in the vicinity of the riverbanks, and this is considered to present a low risk to both the protected banks and natural (vegetated) riverbanks.

— The flows from the Storm outfall compartment indicate concentrated and focussed streamlines extending across the river, in particular where the flows in the river are slack (as for Flow Case B). Further modelling, considering alternative outfall ports (possibly using flap gates with a larger area), is advised to mitigate this effect.

6. Recommendations

Further development is required to reduce the peak velocities from the storm compartment. The tide flex valves tested for this study give a relatively focused flow stream and alternative options should be tested that give better dispersal of flow whilst maintain protection against the risk of unauthorised entry or ingress by large debris. Upsizing the TideFlex valves is not recommended as the “snap open” flow will increase resulting in some valves remaining shut.

Layout 200 is the preferred arrangement for developing further as it gave the best performance of the four layouts tested. Dimensions for this layout are given in Appendix D.

To reduce the velocities from the outfall it is recommended that further CFD modelling be carried out to determine if the following refinements provide benefits:

— Increasing the depth of the outfall bay

— Increasing the submergence of the outfall ports

— Increase the port area for the storm outfalls (including the evaluation of flap gates with larger openings).

As the existing outfall has performed adequately it is recommended that this outfall also be modelled using CFD to provide a direct comparative reference for the new outfall.

7. References

[Ref 1] *TASK BRIEF No. 8. River Modelling.V3 Scope Change 2.xlsx* - Email 02 March 2022

[Ref 2] *Cambridge WWTP River Modelling by Binnies, Final Version P02, 22 April 2022*

[Ref 3] *CIRIA C786: Culvert, Screen and Outfall Manual, revised edition February 2020*

[Ref 4] *Modified Outfall Sketch (Rev B -draft at 12 Apr 2022).pdf*. Received by email from David Winzer, 12 April 2022.

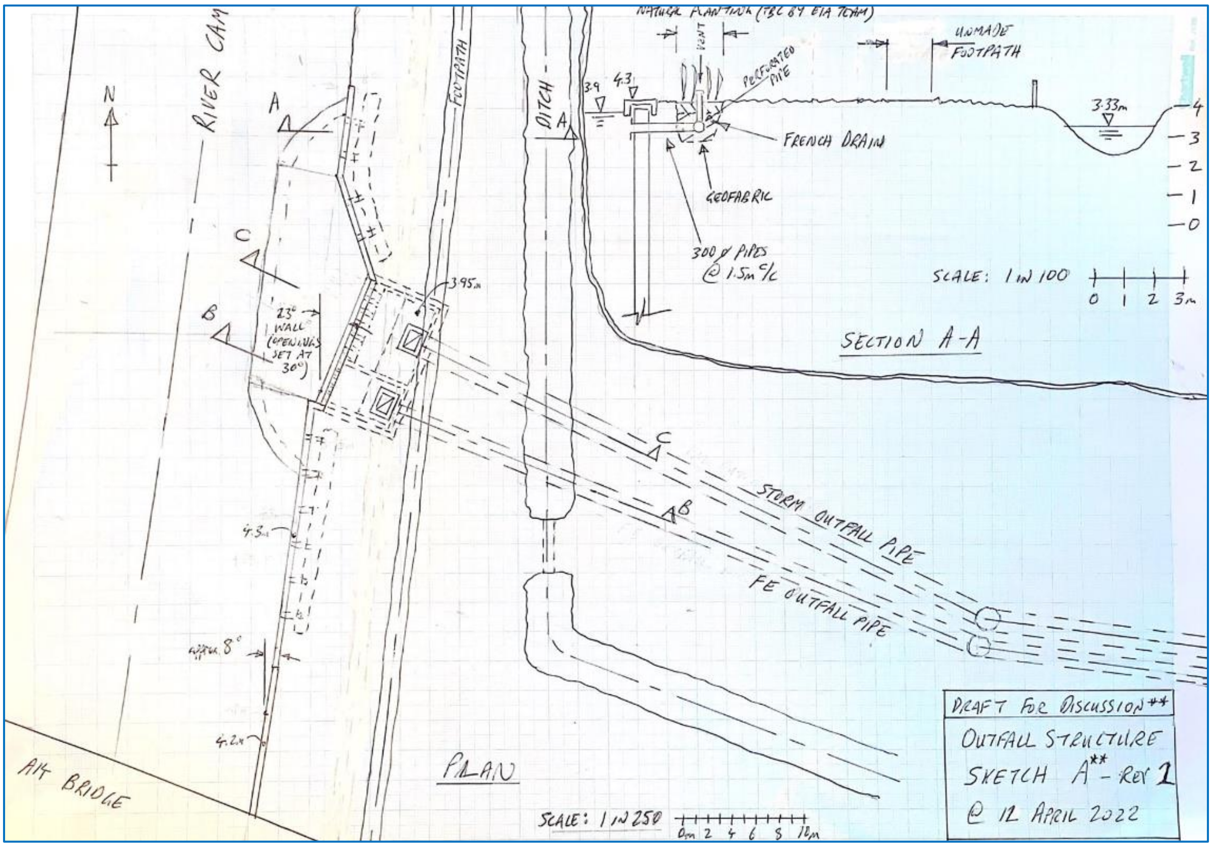
[Ref 5] *Cambridge Wastewater Treatment Plant Relocation Project - Network Modelling Report - Spills to the watercourse*, Binnies, 9 Feb 2022

[Ref 6] *Drawing of Existing Effluent Outfall for the Milton Works (ref Drawing number S/212/13/5, Rev C) dated September 1976.*

APPENDICES

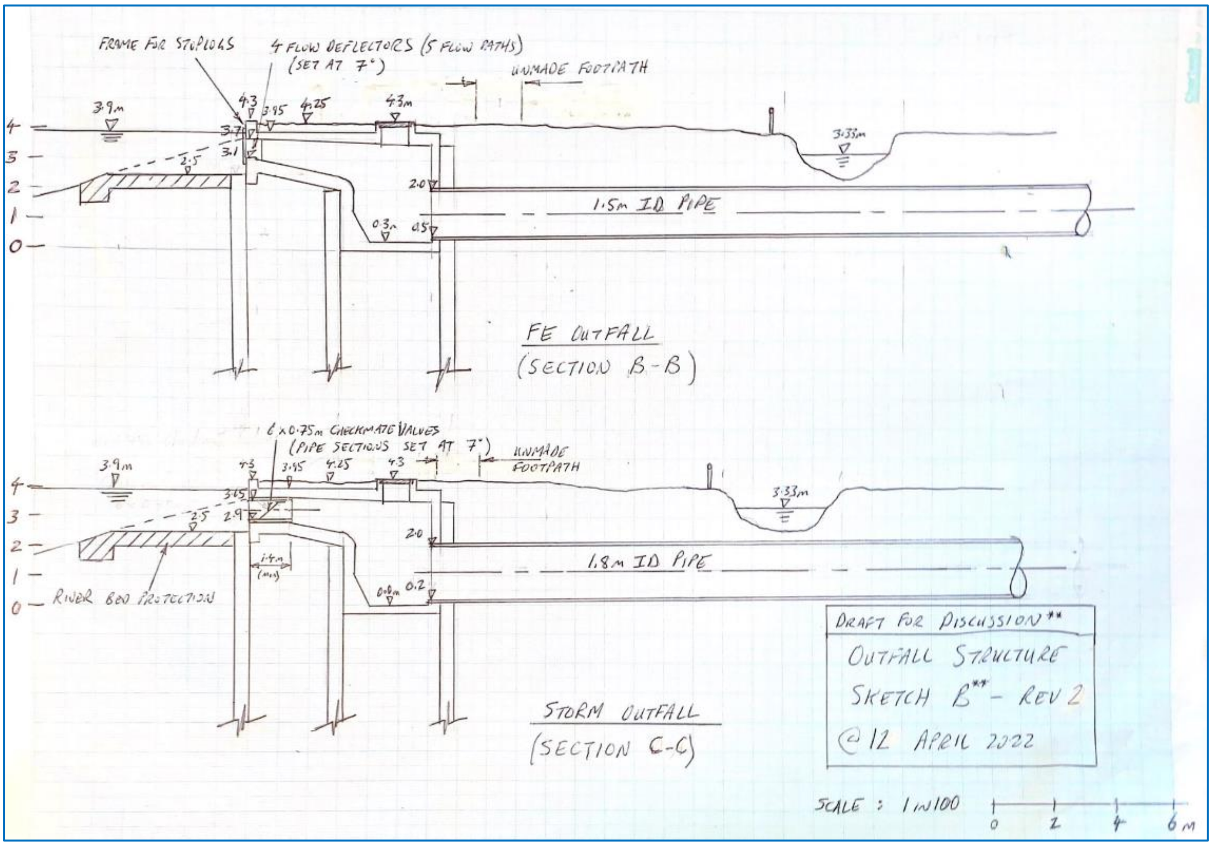
Appendix A: Sketches of initial layout

The following sketches were provided by the design team as the basis for the initial model layout [Ref 4].



Sheet 1 from Modified Outfall Sketch (Rev B -draft at 12 Apr 2022) pdf by David Winzer

Figure 26 Sketch plan for initial model layout



Sheet 2 from Modified Outfall Sketch (Rev B -draft at 12 Apr 2022) pdf by David Winzer

Figure 27 Sketch sections for initial model layout

Appendix B: Model settings

Details of key model settings and parameters are listed below

Software:

- Ansys CFX 2021 R1

Simulation type:

- Steady-state
- Physical timescale = 0.1s

Fluid models:

- Homogeneous multi-phase.

Water phase:

- Density = 997kg/m^3
- Dynamic viscosity = 8.899×10^{-4} kg/(m s)

Air phase:

- Density 1.185kg/m^3
- Dynamic viscosity = 1.831×10^{-5} kg/(m s)

Free surface model

- Standard
- Interface compression = 2

Turbulence model

- Homogeneous
- Shear Stress Transport (SST). This is an enhancement of the k-omega model which is recommended by CFX for general purpose modelling. The model has an automatic wall function that is suitable for a wide range of wall mesh scales (both low or high Y+).

Interphase transfer

- Mixture model
- Length scale 2mm

Appendix C: Existing outfall arrangement at Milton

The following details were extracted from the existing outfall drawing [Ref 6].

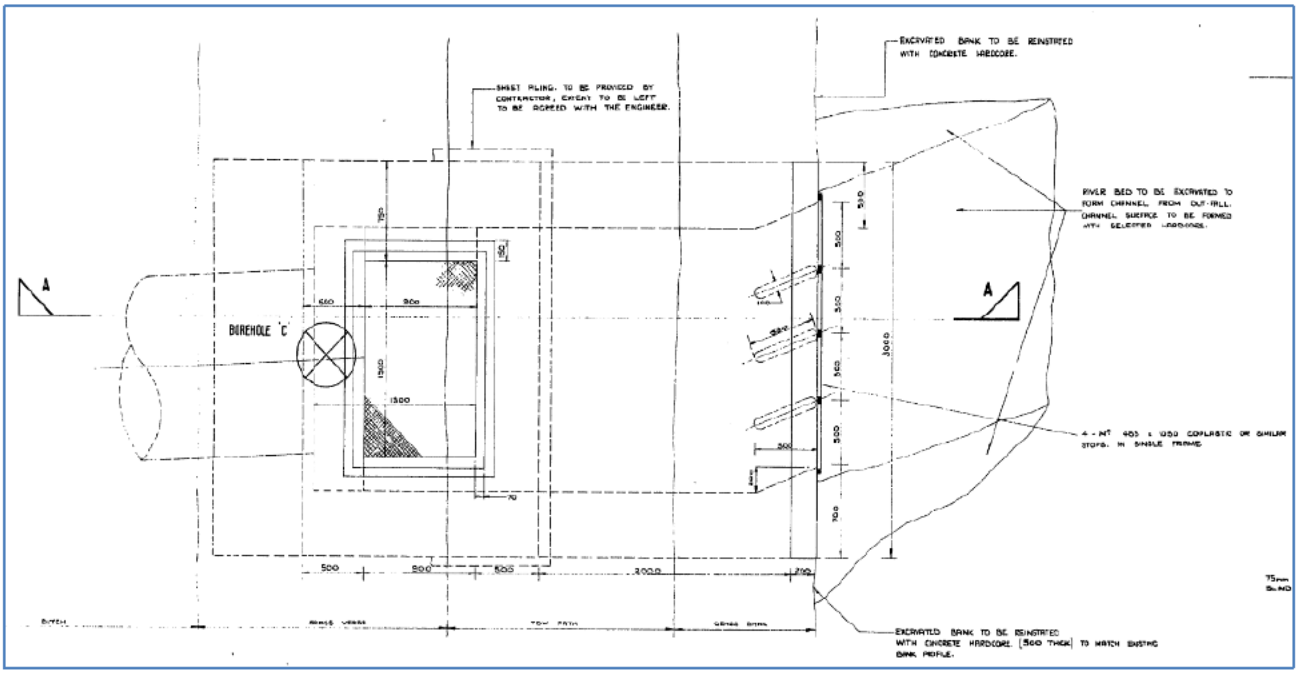


Figure 28 Plan View of the Existing Outfall

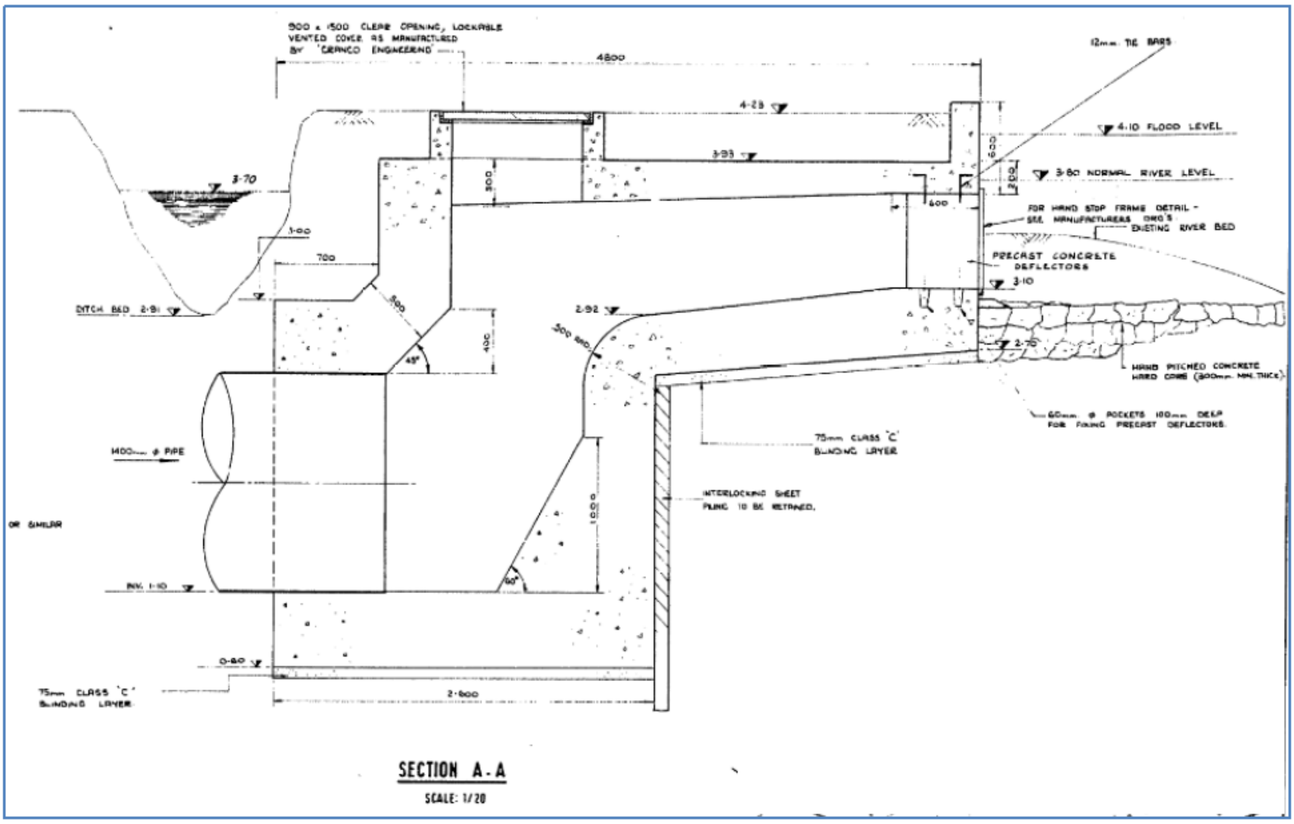


Figure 29 Section A-A through Existing Outfall

Appendix D: Chamber dimensions for Layout 200

Layout 200 is the preferred arrangement for developing further as it gave the best performance of the four layouts tested. Chamber dimensions for the other layouts are similar other than the angle of the ports and Tideflex valves.

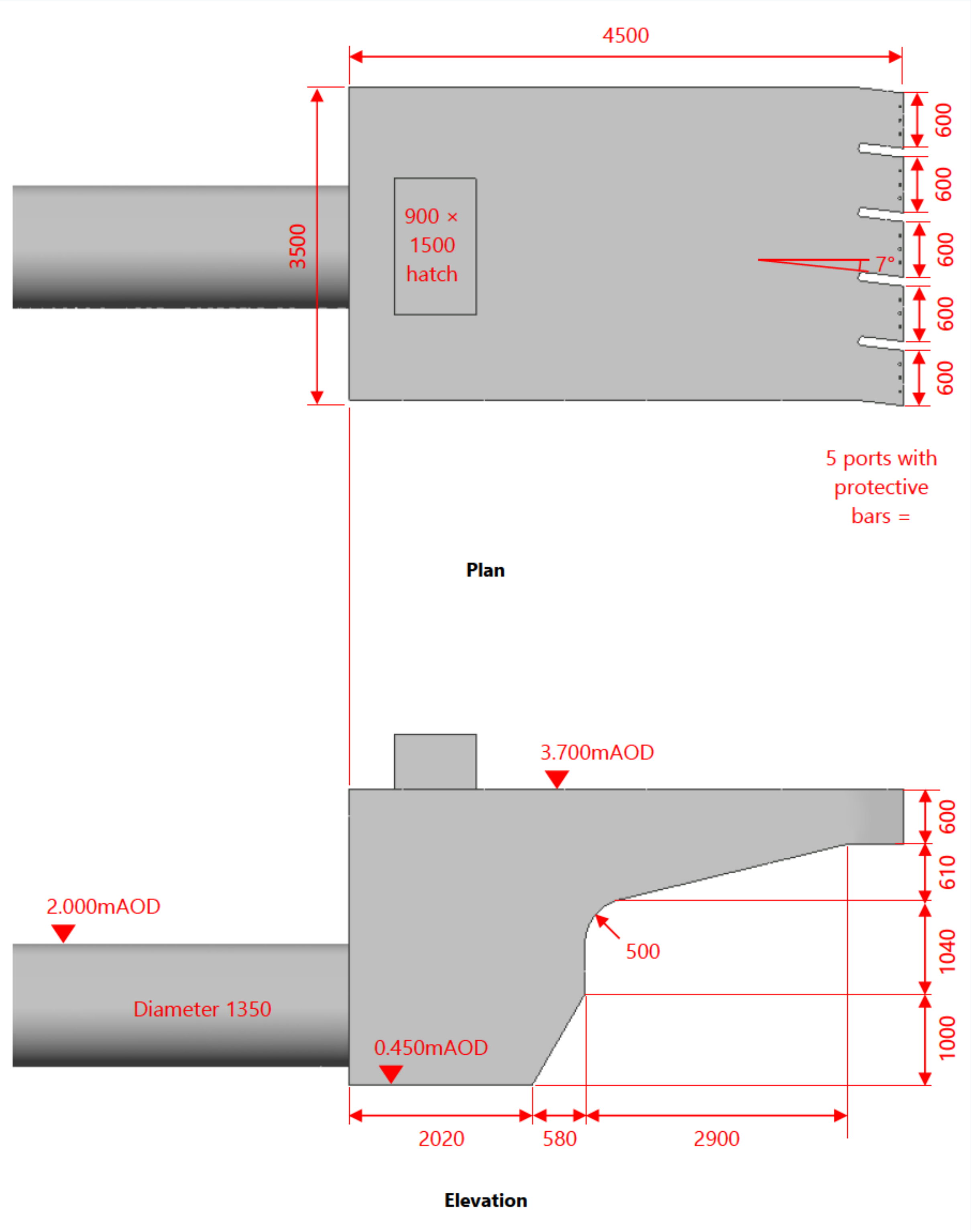


Figure 30 Dimension of FE chamber as modelled

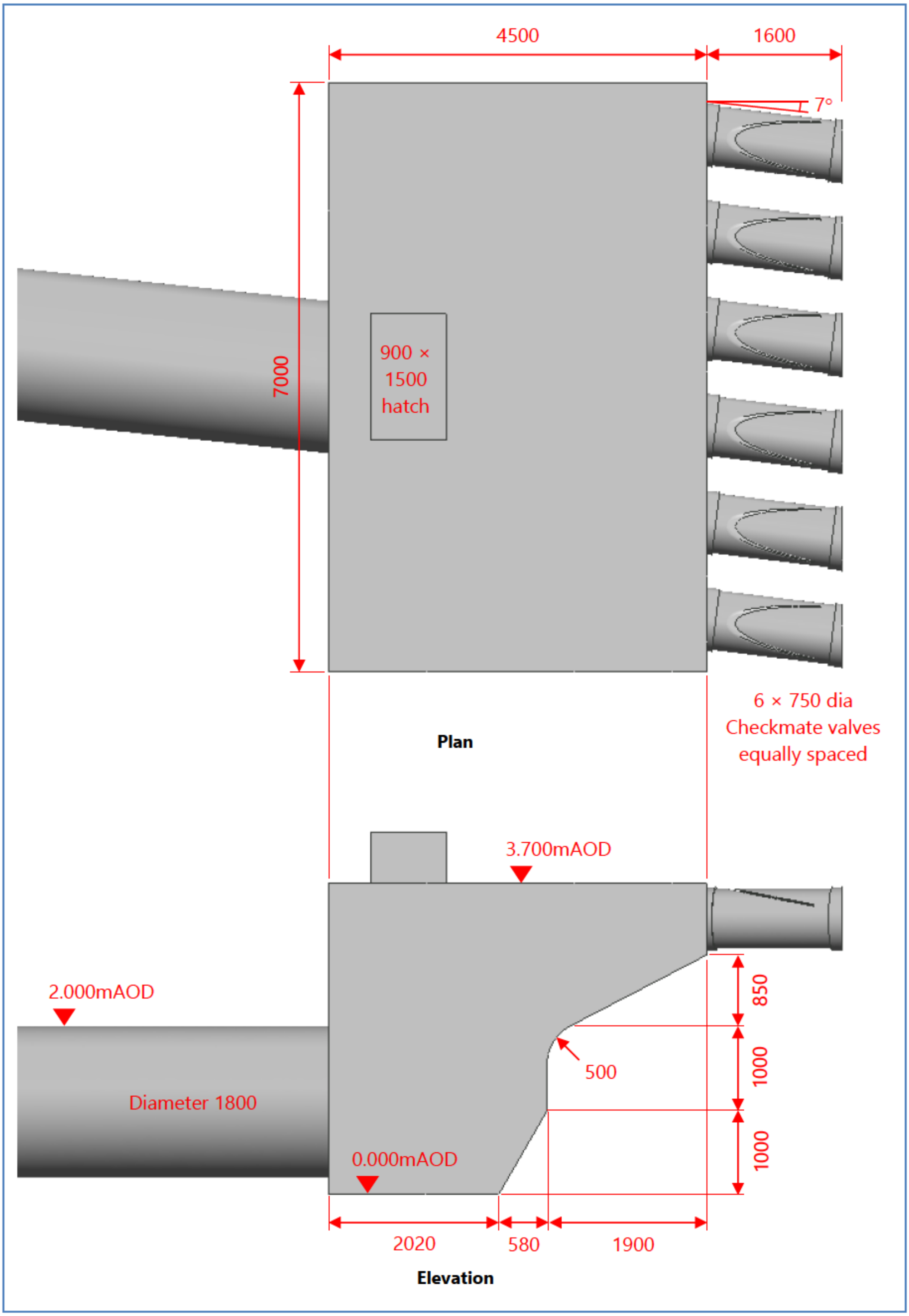


Figure 31 Dimension of storm chamber as modelled

Get in touch

You can contact us by:

Emailing at **info@cwwtpr.com**

Calling our Freephone information line on **0808 196 1661**

Writing to us at **Freepost: CWWTPR**

Visiting our website at [redacted]

You can view all our DCO application documents and updates on the application on The Planning Inspectorate website:

https://infrastructure.planninginspectorate.gov.uk/projects/eastern/cambridge-waste-water-treatment-plant-relocation/