Investigation of the Flow Field inside a Drainage System: Gully – Pipe - Manhole

Md Nazmul Azim Beg[1,2], Rita F. Carvalho[1], Jorge Leandro[1], Pedro Lopes[1] and Lincoln Cartaxo[1]

[1]MARE - Marine and Environmental Research Centre
Department of Civil Engineering, University of Coimbra,
Coimbra, Portugal

[2]Early Stage Researcher, Marie Curie Actions ITN (QUICS)

6th IAHR International Junior Researcher and Engineer Workshop on Hydraulic Structures, Lübeck/Germany

May 30th to June 1st 2016

www.quics.eu

Introduction

One of the busiest city in Dhaka, due to 46mm of rainfall in one and a half hour; on afternoon of September 1, 2015.
Photo Credit: The Daily Star on September 2, 2015.

Pluvial flooding at City centre of Coimbra on May 2006
Photo Source: http://www.raingain.eu/en/actualite/flood-solutions-north-south-europe

Chertsey, UK on February 11, 2014
Photo source: The Guardian on 11 February, 2014

Introduction

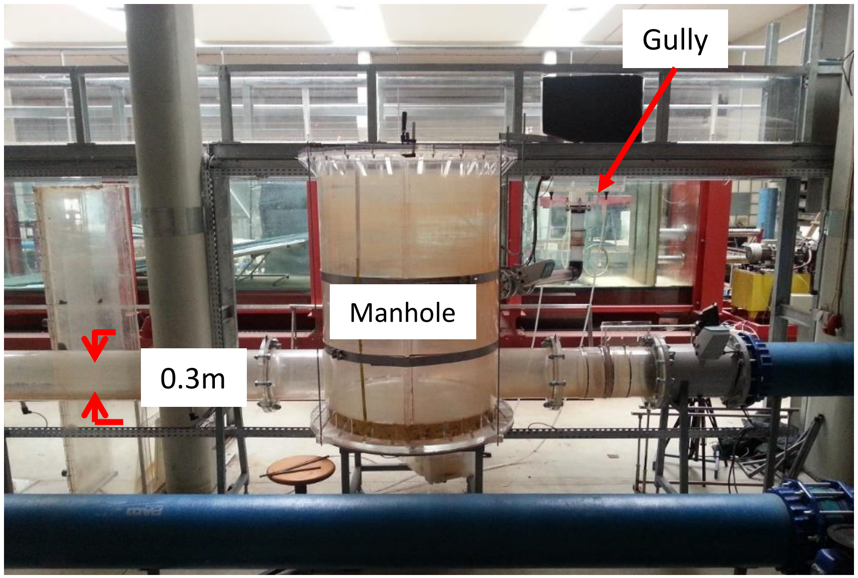
- Flooding is one the of biggest threats for a busy urban city
- The urban drainage system is responsible for safe routing of flood water; hence an efficient drainage is mandatory
- Drainage system efficiency is dependent on the individual efficiency of each element
- Gully and Manhole are two common element of an urban drainage system
- Flow analysis inside these structures can lead to a better understanding of the efficiency of a drainage system

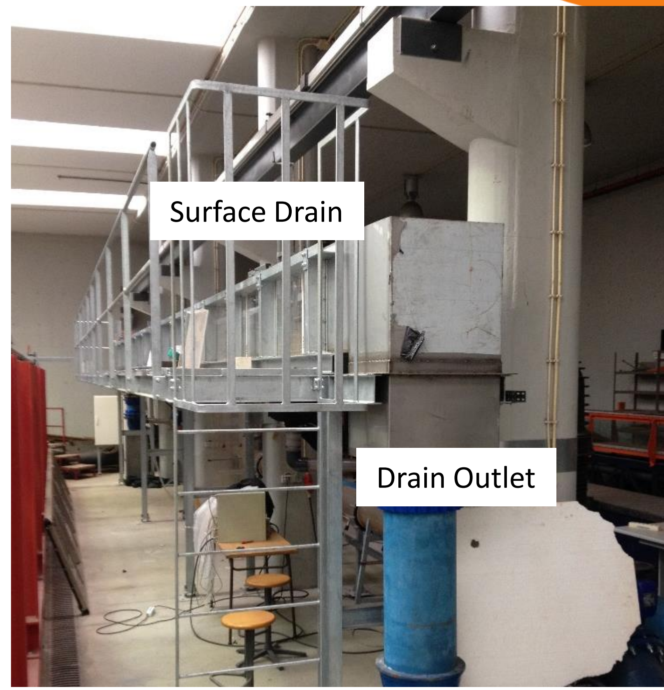
Objective

- To validate CFD model with experimental measurement at the laboratory
- To analyse the different flow behaviour inside a gully-manhole drainage system

Methodology

Physical Model set up



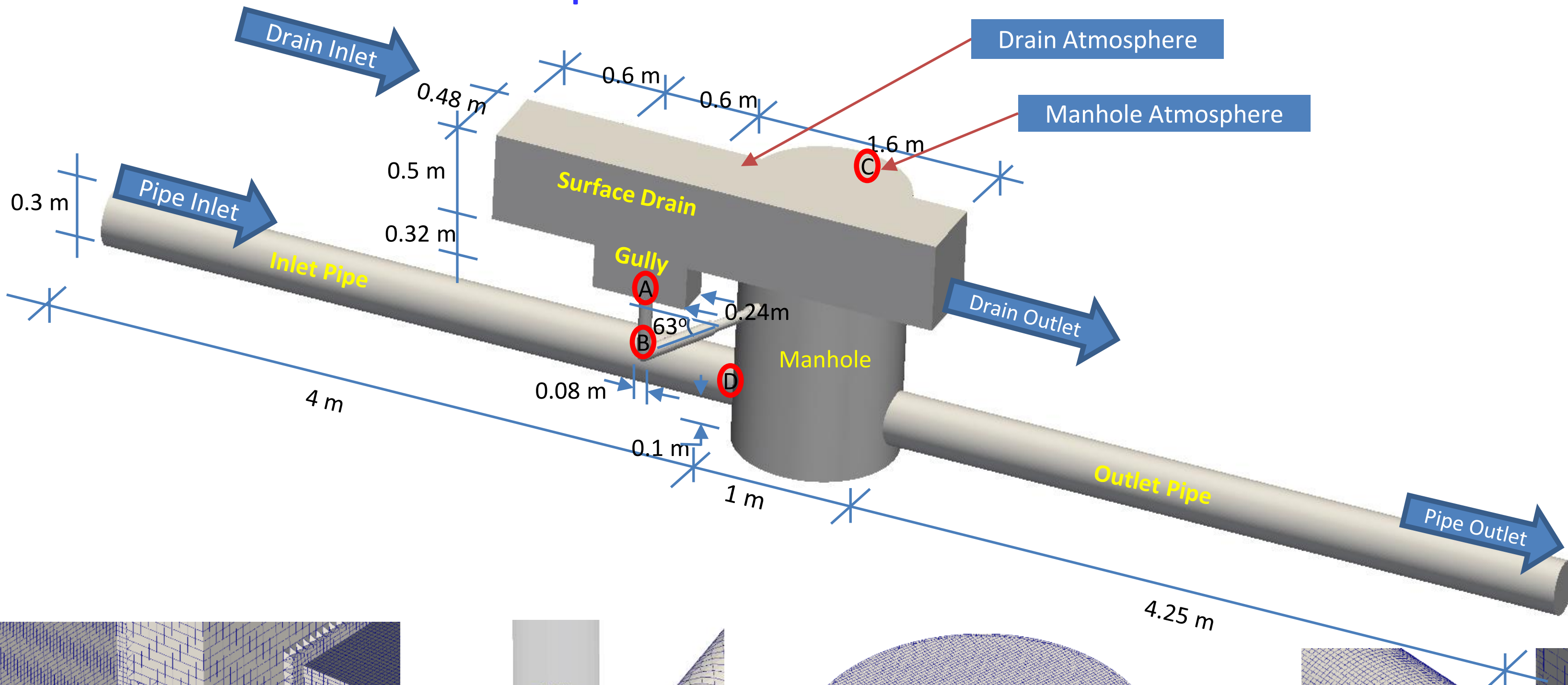


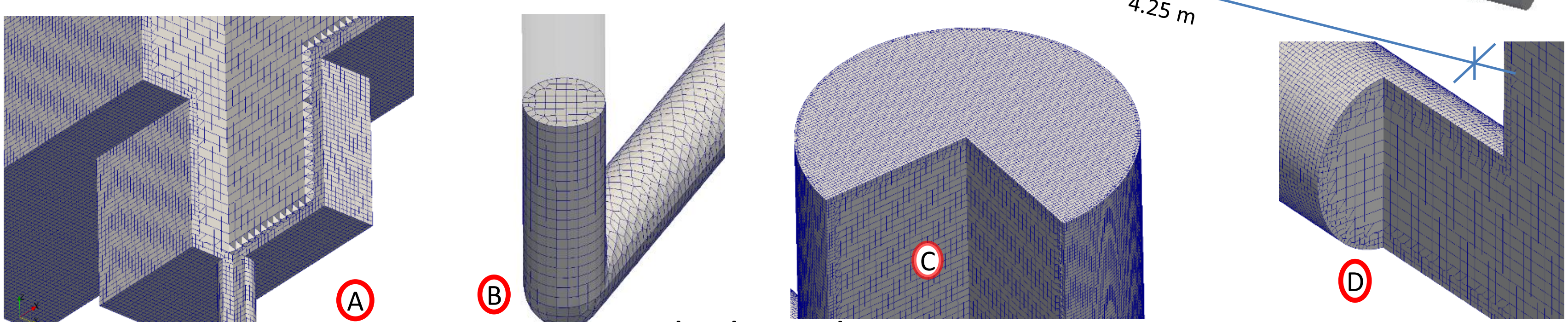
The physical model facility is installed at the Department of Civil Engineering, University of Coimbra.

- 1m diameter manholes
- Connected by a Ø300 sewer pipe
- 0.5m wide and 1% slopped surface channel
- 0.6 × 0.24 × 0.32 [m] (L × W × D) gully

Methodology

Numerical Model set up





- *cfMesh*
- Mesh size 2cm
- 1 cm at the boundaries
- 821,500 computational with 1.01 million nodes

Field Data collection

Methodology

OpenFOAM simulation

- OpenFOAM v. 2.3.0
- *interFOAM* solver: considering isothermal, incompressible and immiscible two-phase flow (air and water for this case)
- Mass and Momentum conservation

$$\frac{\partial \rho}{\partial t} + \nabla . (\rho \boldsymbol{u}) = 0$$

$$\frac{\partial \rho \boldsymbol{u}}{\partial t} + \nabla . (\rho \boldsymbol{u}\boldsymbol{u}) = -\nabla p^{*} + \nabla . \tau + \boldsymbol{g} . \boldsymbol{x} \nabla \rho + \boldsymbol{f_{\sigma}}$$

- Uses Volume of Fluid (VOF) method (Hirt and Nichols 1981) to track the free surface or interface location
- RAS k-ε turbulent model was used
- PISO algorithm is used

Methodology

Tests performed

- Numerical model: combination of two different experimental studies:
    1. only the manhole with inlet and outlet pipe were used; a flow of 43.7 l/s was applied through the manhole inlet.
    2. flow through the drain and gully was observed; 19.8 l/s flow was measured at the upstream of the drain inlet

- Two different Numerical simulations are tested

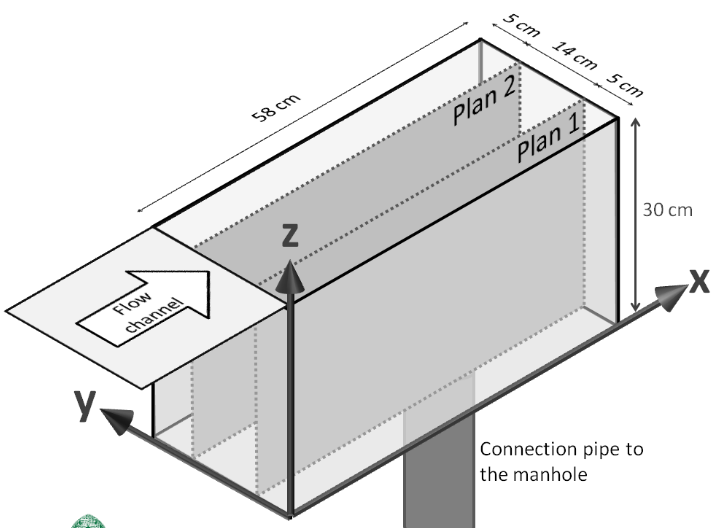
	Drain inlet Q (l/s)	Manhole inlet Q (l/s)	Manhole surcharge level (m)	Remarks
Simulation 1	19.8	43.7	0.67	Experimental case scenario
Simulation 2	19.8	43.7	1.29	Additional scenario

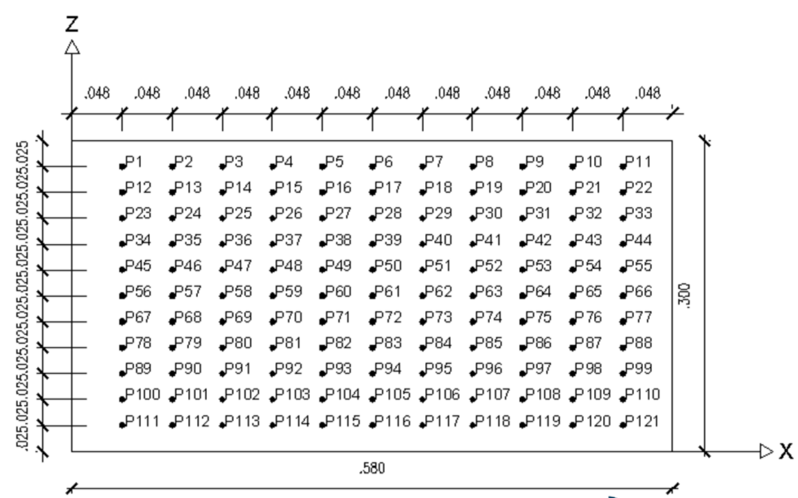
- 40seconds of run to reach steady state condition
- Each steady state simulation took 138hrs using 16 processors

Results

Comparison with experimental tests performed

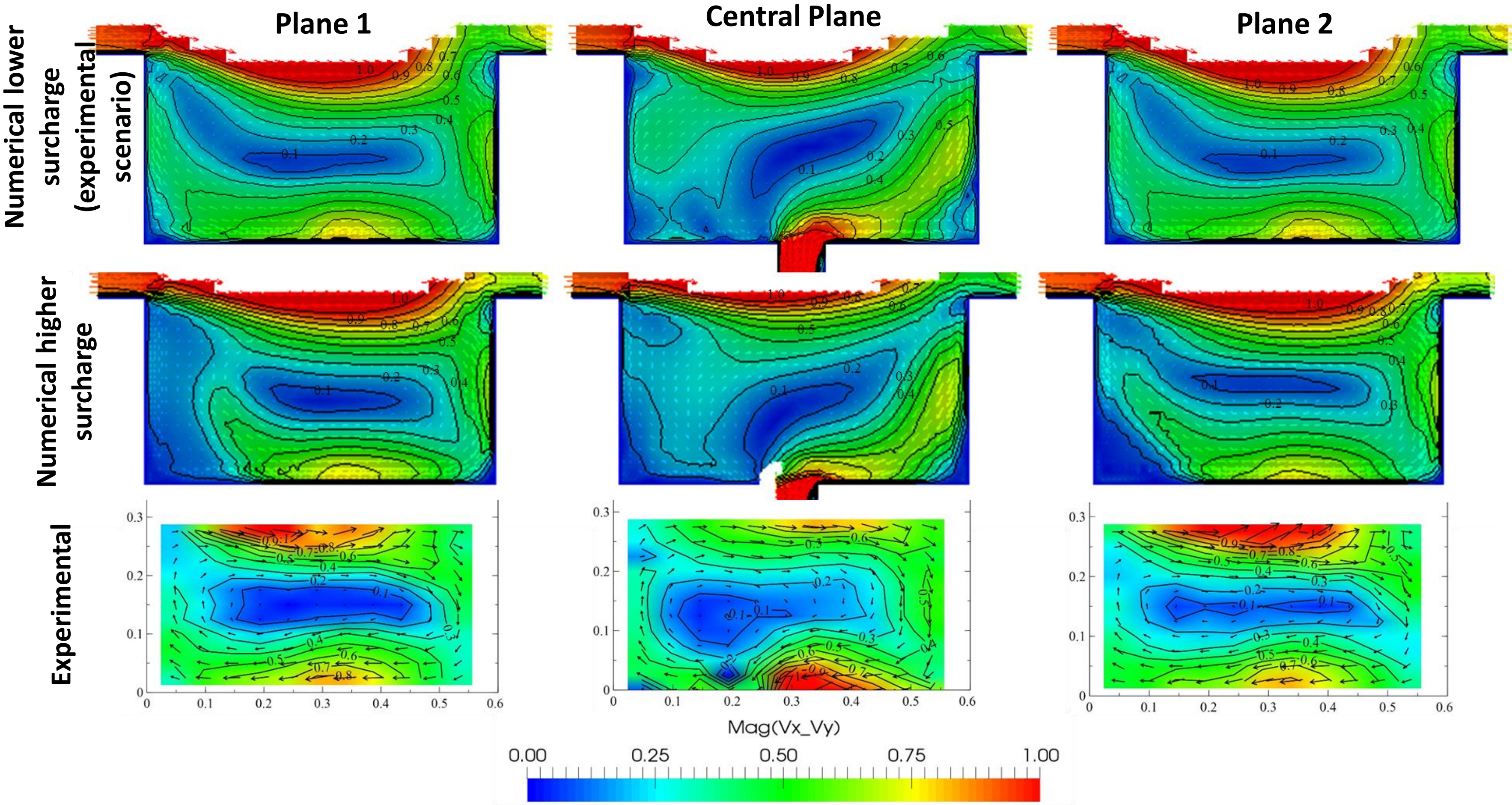
- During the experimental study, velocity at the gully was observed at three plane using Nortec Vectrino acoustic velocimeter
- The first and the third plane are at 5 cm distance from the longitudinal walls of the gully
- The second plane is the central plane
- Each plane contained 121 velocity measurements





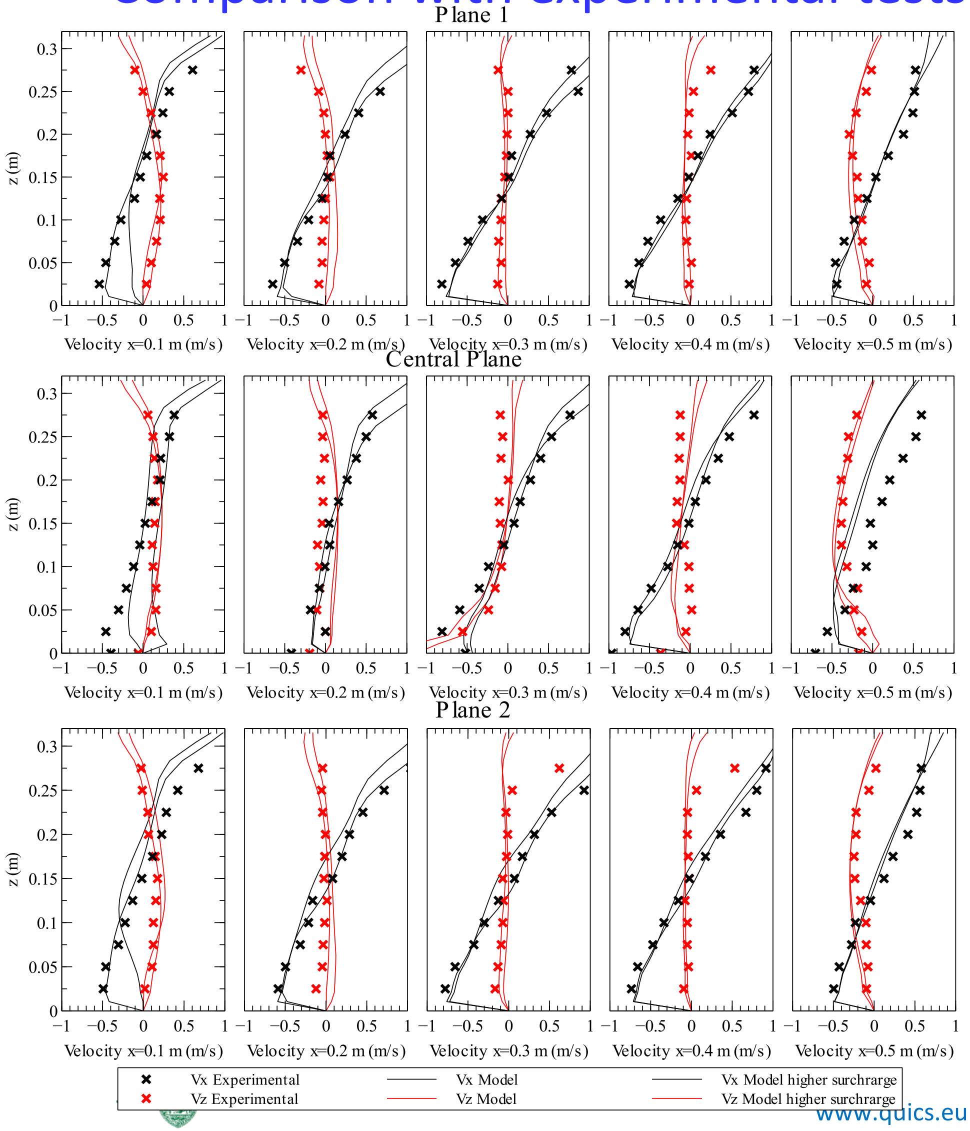
Results

Comparison with experimental tests performed



Results

Comparison with experimental tests performed



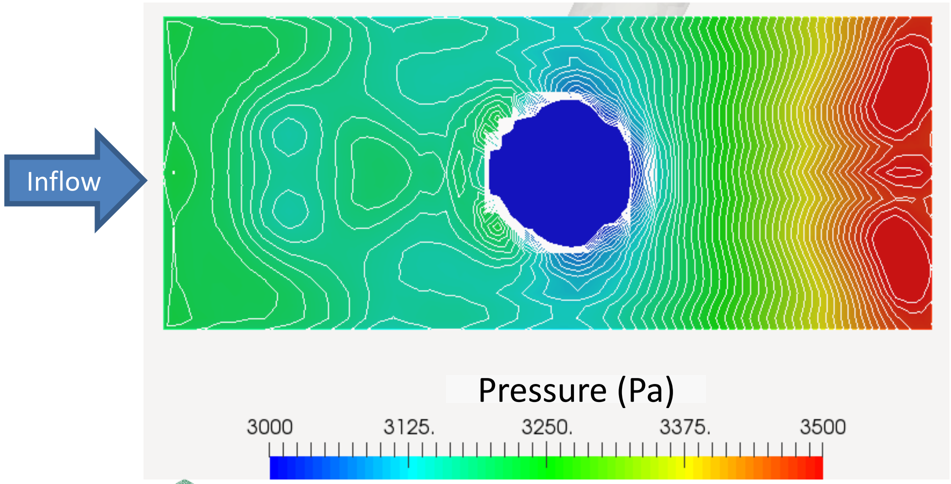
		BIAS					
		X=0.1m	X=0.2m	X=0.3m	X=0.4m	X=0.5m	Avg.
P 1	Vx	0.060	0.014	-0.078	-0.068	-0.007	**-0.016**
P C		-0.223	-0.034	-0.024	-0.009	0.186	**-0.021**
P 2		0.096	0.023	-0.010	-0.016	0.028	**0.024**
Avg.		-0.023	0.001	-0.037	-0.031	0.069	**-0.004**
P 1	Vz	0.004	-0.031	0.009	0.072	0.073	**0.025**
P C		-0.020	-0.141	0.015	-0.020	0.021	**-0.029**
P 2		-0.029	-0.021	0.069	0.079	0.089	**0.037**
Avg.		**-0.015**	**-0.064**	**0.031**	**0.044**	**0.061**	**0.011**

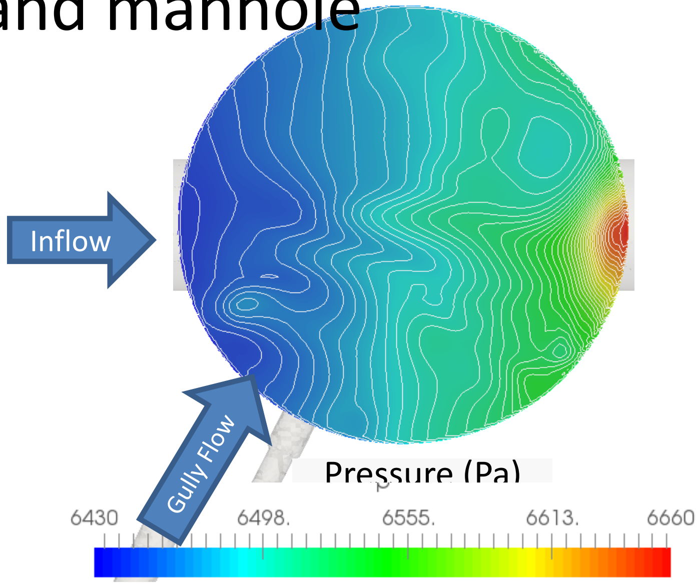
		r					
		X=0.1m	X=0.2m	X=0.3m	X=0.4m	X=0.5m	Avg.
P 1	Vx	0.993	0.985	0.988	0.996	0.981	**0.988**
P C		0.817	0.964	0.974	0.998	0.931	**0.937**
P 2		0.994	0.993	0.996	0.992	0.985	**0.992**
Avg.		0.935	0.981	0.986	0.995	0.966	**0.972**
P 1	Vz	0.932	0.834	0.728	0.891	0.845	**0.846**
P C		0.600	-0.221	0.917	-0.731	0.806	**0.274**
P 2		0.920	0.233	0.129	0.845	0.840	**0.593**
Avg.		**0.817**	**0.282**	**0.591**	**0.335**	**0.830**	**0.571**

Results

Pressure at the bottom

- The pressure at both the gully and manhole bottom are not uniform
- Higher pressure near the drain outlet and lower pressure at the inlet
- Difference between the max and min pressure is in the range of 300Pa and 200Pa at gully and manhole bottom respectively

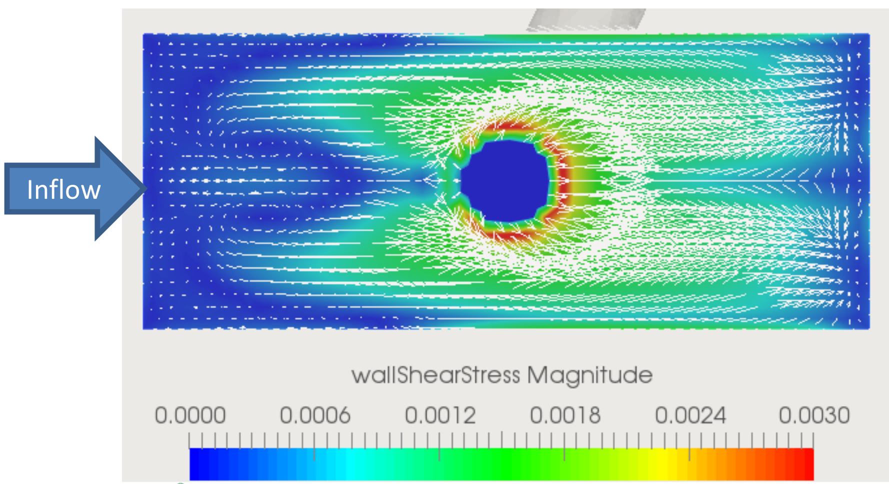


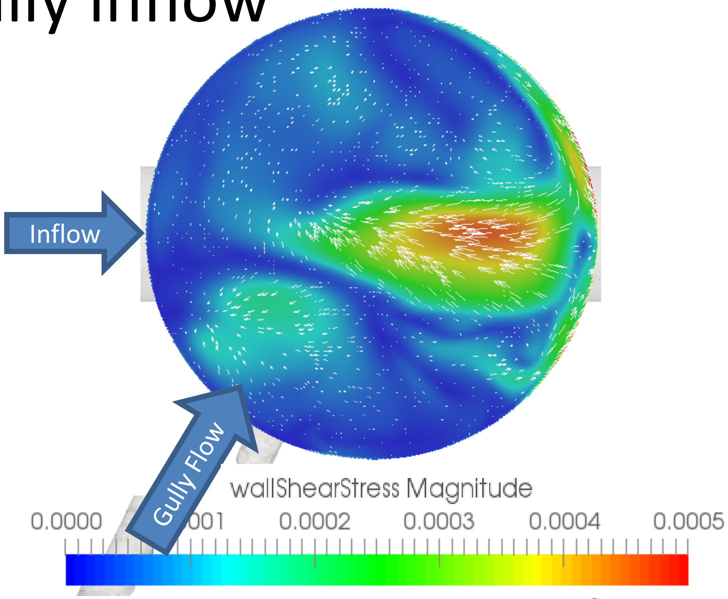


Results

Wall shear stress at the bottom

- Like pressure map, the wall shear stress is not uniform
- The shear stress direction is opposite to the flow
- For gully, higher shear stress near the gully outlet
- For manhole, higher shear stress near the central axis
- The shear stress pattern is asymmetric for the manhole bottom, probably a result of gully inflow



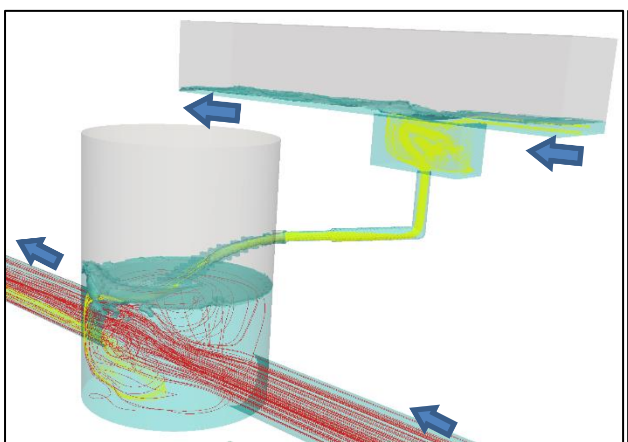


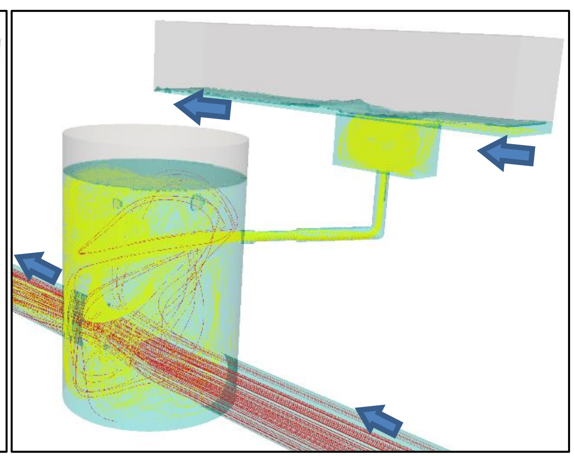
Results

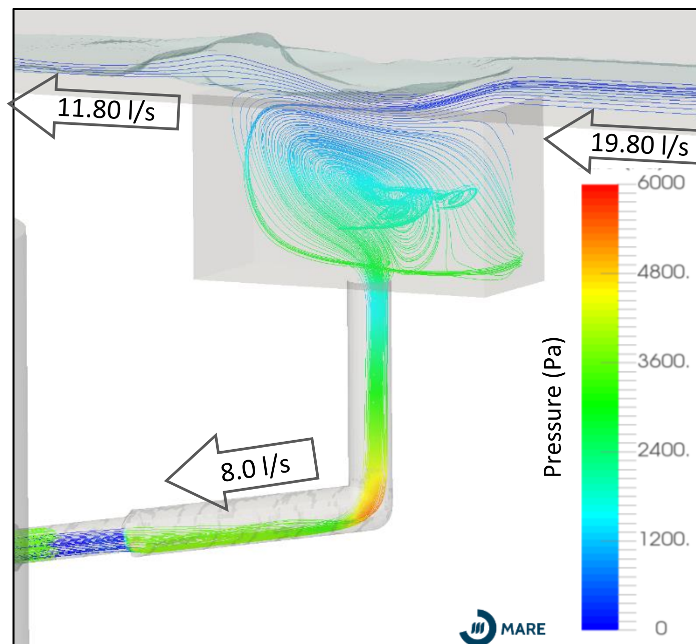
Streamline

- Flows coming from gully and manhole inlet becomes well mixed inside manhole
- Surcharge level has influence in the vortex formation
- Fraction of the flow from drain inlet goes inside the gully and later comes out to the drain
- The gully outlet flow occupies partial area of the pipe

Flow distribution	Drain Inlet	Drain Outlet	Gully Pipe	Manhole Pipe Inlet	Manhole Pipe Outlet
Flow (l/s)	19.80	11.80	8.0	43.70	51.70







Conclusion

- The work presented shows the first step numerical assessment of flow behaviour inside a gully-manhole drainage system
- OpenFOAM® v. 2.3.0 with solver interFOAM was used with RANS k-ε turbulence model
- Numerical model shows good agreement with measured velocity at the gully
- Flow streamline show different characteristics with change in surcharge level in the manhole.

Future Work

- The model will be validated with flow measurement inside the manhole
- The work will be further developed to better understand the particulate transport phenomena inside the drainage system

Thank you for your attention

Nazmul Azim Beg
Email: mnabeg@uc.pt

Partners and Acknowledgements

This project has received funding from the European Union's Seventh Framework Programme for research, technological development and demonstration under grant agreement no 607000.

First author would also like to thank FCT (Portuguese Foundation for Science and Technology) for their support through the Project UID/MAR/04292/2013 financed by MEC (Portuguese Ministry of Education and Science) and FSE (European Social Fund), under the program POCH (Human Capital Operational Programme).

www.quics.eu