THE UNIVERSITY OF WESTERN ONTARIO
DEPARTMENT OF CIVIL AND
ENVIRONMENTAL ENGINEERING

Water Resources Research Report

General Methodology for Developing a CFD Model for Studying Spillway Hydraulics using ANSYS Fluent

By:
R Arunkumar
and
Slobodan P. Simonovic

Report No: 098
Date: October 2017

ISSN: (print) 1913-3200; (online) 1913-3219
ISBN: (print) 978-0-7714-3148-7; (online) 978-0-7714-3149-4

General Methodology for Developing a CFD Model for Studying Spillway Hydraulics using ANSYS Fluent

R. Arunkumar

&

Slobodan P. Simonovic

Department of Civil and Environmental Engineering

The University of Western Ontario

London - N6A 5B9, Ontario, Canada

Executive Summary

The advancement of computing facilities has led to the development of advanced software packages and tools for solving various practical engineering problems. One such advancement is the development of various computational fluid dynamic (CFD) software with different numerical solver methods. These computational methods are identified as suitable tools for solving various engineering problems. They also have various advantages over the traditional physical modeling. One such CFD software tool is ANSYS Fluent. In this report, the general guide and practical steps for developing a full 3D CFD spillway model using ANSYS Fluent have been presented.

Table of Contents

List of Figures

1. Introduction

The spillway is an important hydraulic structure of a dam, which facilitates the safe passage of flow from the upstream reservoir to the downstream. A hydraulically efficient and structurally strong spillway is very important for the dam safety, and protection of the life and property at the downstream. Many hydraulic models have been extensively developed to study, visualize and understand the hydraulic behavior of flow over the spillway. Various hydraulic design aspects such as discharge capacity, velocity, pressure and water surface profiles are considered to study the spillway hydraulics. The hydraulic behavior of flow over spillway can be studied through physical or numerical modeling. Physical modeling of spillways is expensive, cumbersome and time consuming (Savage and Johnson, 2001). To overcome the limitations of the physical modeling, numerical models including three-dimensional (3D) computational fluid dynamics (CFD) tools have been used in recent years due to the advancement in computing technology and numerical methods.

A diverse variety of problems can be studied using CFD modeling. It is widely used in fluid mechanics, aerodynamics, multiphase, and free-surface flow studies, etc. In dam engineering, CFD modeling have been widely used to study the hydraulic performance of various types of spillways, eg. ogee spillway (Savage and Johnson, 2001), circular spillway (Rahimzadeh *et al.*, 2012), stepped spillway (Chanel and Doering, 2008; Chinnarasri *et al.*, 2014; Dursun and Ozturk, 2016), and also for labyrinth weirs (Savage *et al.*, 2016), rectangular channels (Mohsin and Kaushal, 2016) and many more. Aydin and Ozturk (2009) reported that the CFD models are more flexible and require less time, money, and effort than physical hydraulic models. The other advantage of CFD modeling is that the scale effects of physical modeling can also be eliminated through the real dimensions of the prototype developed using in the CFD model (Bhajantri *et al.*, 2006). All these studies reported that the CFD is a reliable method for assessing the hydraulic characteristics of a spillway. However, these studies also cautioned that CFD cannot be a complete replacement for physical modeling, but it can definitely be used as a supplementary tool for the spillway design process (Chanel and Doering, 2008). There are many CFD software packages available, both including open source and preoperatory software. OPENFOAM is an open sources CFD software package (OpenFOAM, 2017). The other proprietary software packages include ANSYS Fluent, FLOW-3D, and others. This report focuses on the methodology of developing a 3D CFD spillway model using the ANSYS Fluent (ANSYS, 2016).

The report is organized in the following manner. The general introduction to ANSYS is given in Section 2. The four major steps involved in developing a CFD model is discussed in Section 2.1. The step by step procedure in developing an ANSYS Workbench Project is discussed in Section 2.2. The process of creating the geometry of the structure is given in Section 2.3. The steps involved in creating mesh is discussed in Section 2.4. The full model development using ANSYS Fluent is discussed in detail in Section 2.5. The analysis of results is given in Section 2.6.

2. ANSYS Fluent

Computational fluid dynamic software package ANSYS Fluent (ANSYS, 2016) is extensively used to develop 2-D and 3-D models for the fluid flow simulation. It is a state of the art computer program for modelling fluid flow in complex geometries. All functions required to compute a solution and display the results are available in ANSYS Fluent through an interactive menu driven interface. The software package includes ANSYS Fluent solver, pre-processor tool for geometry modelling and mesh developer. The ANSYS Fluent solver has capability of modelling incompressible, two phase, and free surface turbulent flows in unsteady mode and therefore it was found suitable for modelling flow over spillway aerator. ANSYS Fluent's parallel solver allows for computation by using multiple processes on the same computer. Parallel ANSYS Fluent splits up the grid and data into multiple partitions, then assigns each grid partition to a different computer process. Considering all these features ANSYS Fluent was selected for the spillway simulation modeling in the present study. The various steps in problem formulation, obtaining the solution and analyzing the results are described in the following text.

2.1. General Methodology for Developing a CFD Model

There are four basic primary steps involved in developing a 3D CFD model as shown in Figure 1. The first step is the creation of the structure geometry. ANSYS Fluent have its own geometry builder. Otherwise, the structure geometry can be created in any computer aided design (CAD) software and then imported into the ANSYS Fluent module. The second step is creating the mesh. Meshing is the process of describing the structure using mesh of different shapes and sizes, as cubes, prism, tetrahedral, hexacore, or hybrid volumes (ANSYS, 2013). For each of these mesh units, the hydraulic particulars are computed using the numerical method in the ANSYS Fluent. The developed mesh is then imported into the ANSYS Fluent and the solution is obtained using different solvers depending on the type of analysis. Finally, the results are analyzed using a separate graphical analysis tool in ANSYS. Alternatively, the results can also be analyzed within ANSYS Fluent which offers some limited functionalities.

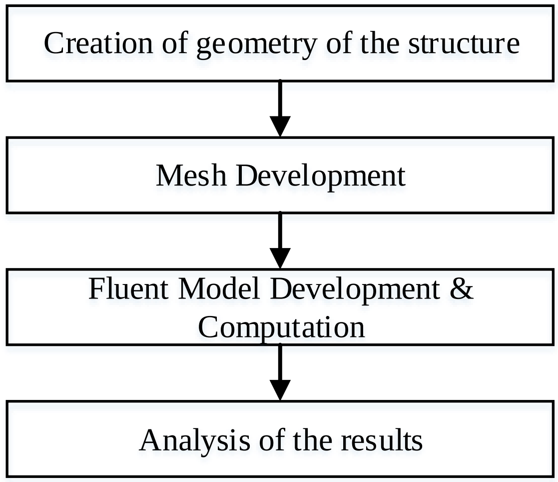

Figure 1. General methodology

The detailed procedure involved in developing an ANSYS Fluent workbench model is explained below.

2.2. Developing an ANSYS Workbench Project

ANSYS Workbench combines access to ANSYS applications with utilities that manage the product workflow. The ANSYS tools that can be accessed from the workbench are ANSYS DesignModeler for creating geometry, Meshing tool for generating mesh, Fluent or CFX tool for setting up and solving fluid dynamics analyses and ANSYS CFD-Post tool for postprocessing the results. The home screen of workbench is shown in Figure 2. From the ANSYS workbench, the 'Fluid Flow (ANSYS Fluent)' has to be drag and dropped into the project window to initiate the modelling process. Alternatively, one can select each tool (Geometry, Mesh, Fluent, Results) individually and create a project. The project may be saved with a user defined name in the File menu.

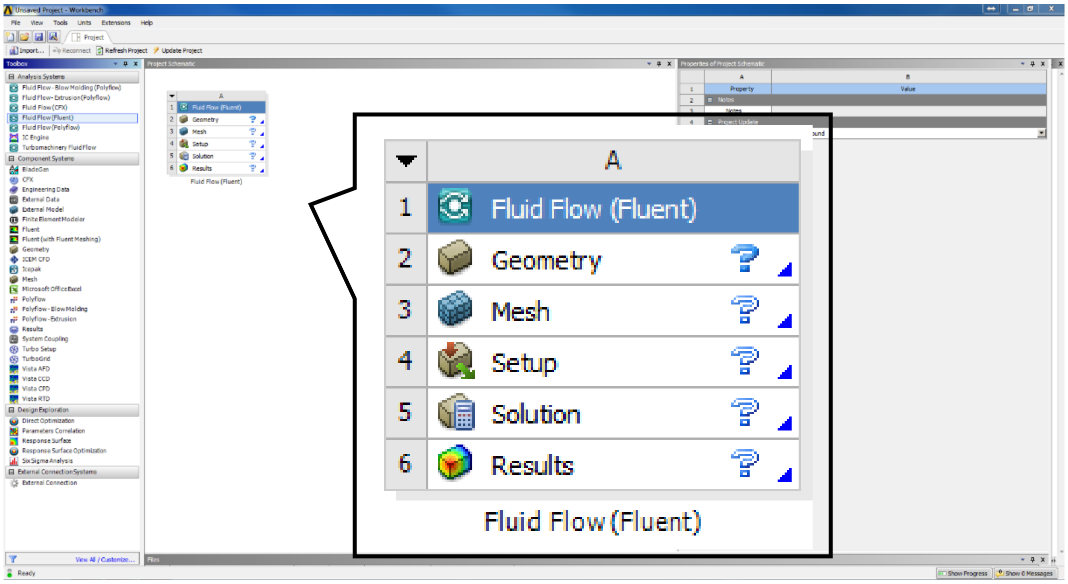

Figure 2 ANSYS workbench and its various tools

The complete ANSYS toolbox is shown in Figure 3.

2.3. Creating / Importing Geometry

The geometry of the structure can be imported from any external design software like AutoCAD or simply created with ANSYS-Design Modeler. By right clicking on 'Geometry' in the ANSYS workbench project, the ANSYS-Design Modeler can be initiated, as shown in Figure 4. The interface of ANSYS Design Modeler is shown in Figure 5. Generation of geometry can be carried out by creating points, edges, faces and volumes of the geometry in three dimensions. After creating/importing the geometry of the structure into the Design Modeler, the fluid flow regions (liquid) and solids regions (solid) of the structure have to be defined. The geometry needs to be divided into separate regions in order to apply constraints for the resulting mesh. Boundary regions are need to be specified for finer or denser meshing.

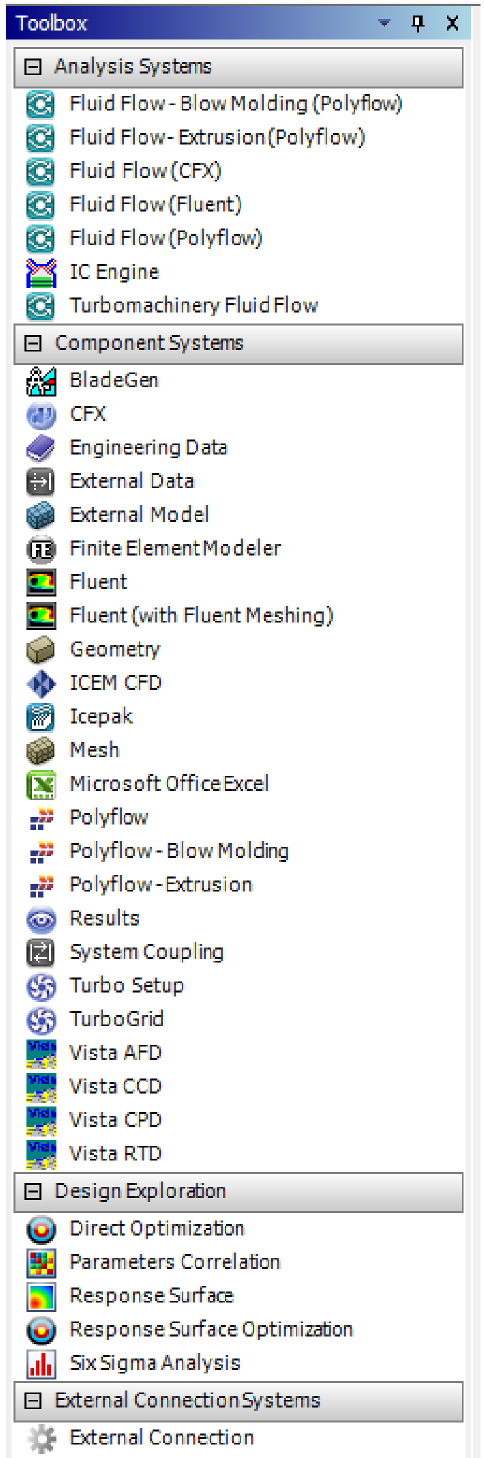

Figure 3. ANSYS toolbox

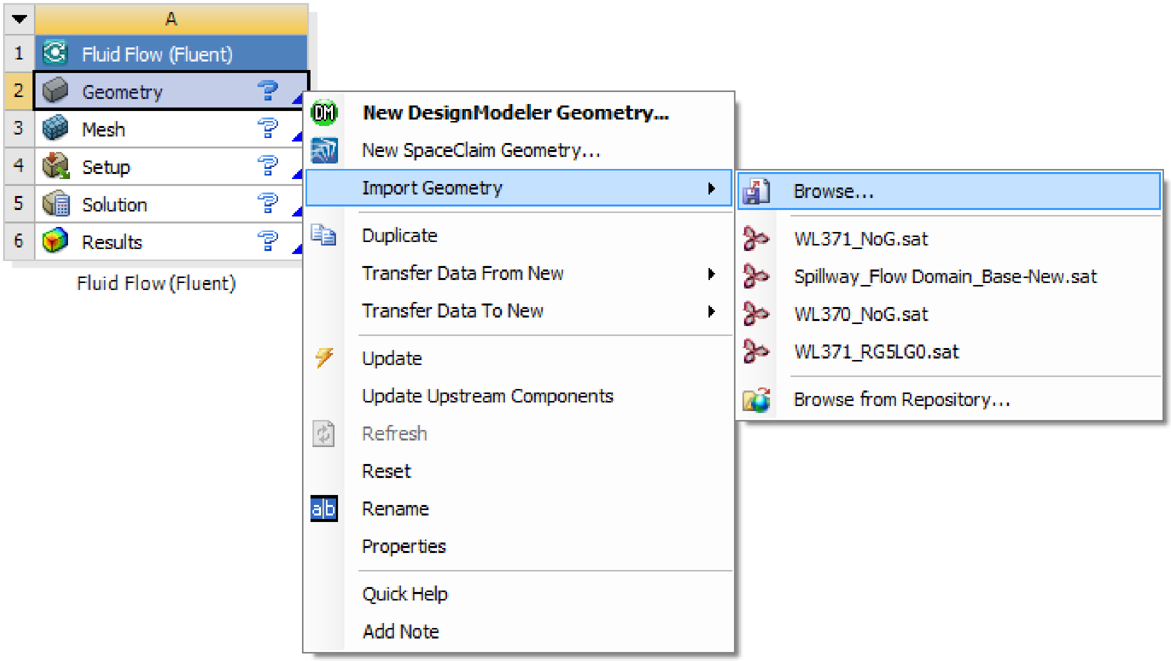

Figure 4. Invoking ANSYS-DesignModeler

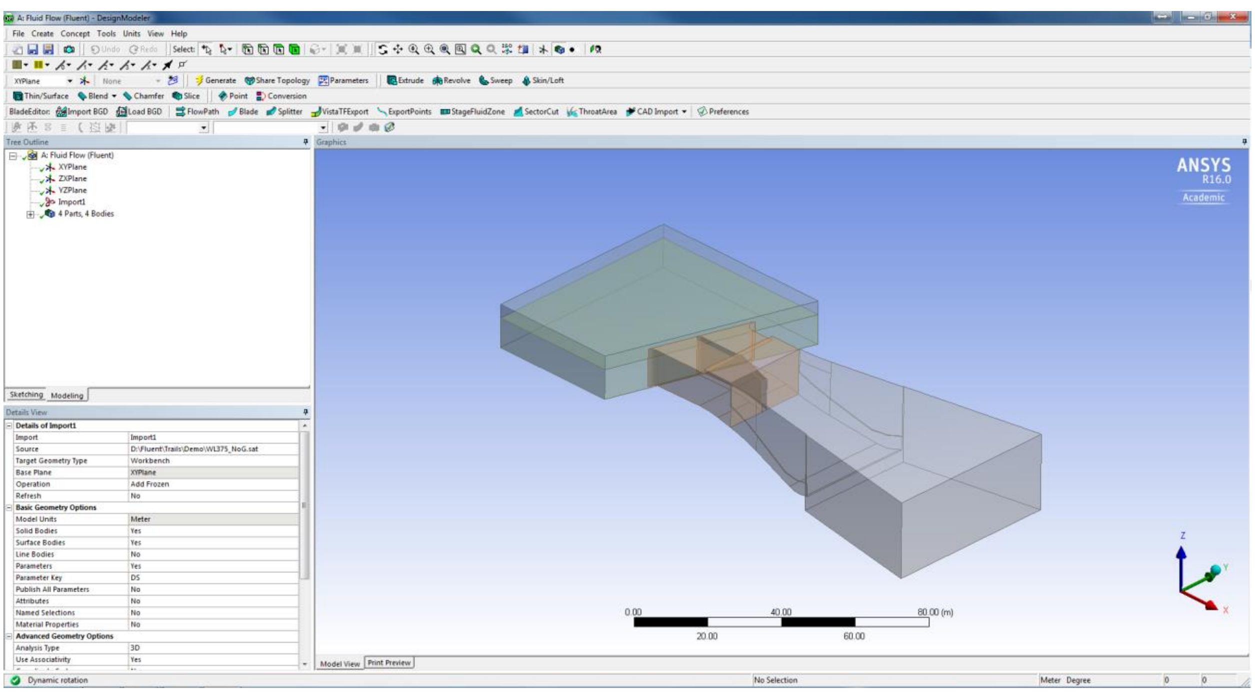

Figure 5. ANSYS Design Modeller

When the procedure is correctly done, the workbench tool box will show a green tick mark next to the 'Geometry' as shown in Figure 6. Whenever each step is completed successfully, the green tick mark will appear on the right side. The process which requires an update or has any error will be highlighted by different symbols.

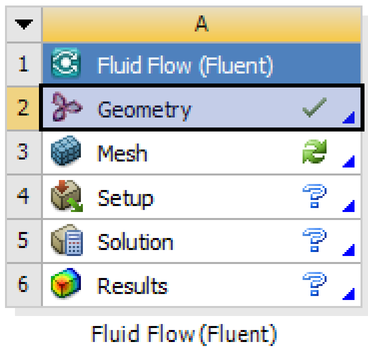

Figure 6. Workflow update - I

2.4. Mesh Generation

The Meshing tool can be invoked by right clicking on 'Mesh' at the workbench project and selecting 'Edit', similar to ANSYS-Design Modeler. The home interface of ANSYS Meshing tool is shown in Figure 7. For creating the mesh, the 'Objects' have to be identified. An object is generally a set of face zones and edge zones. Objects are generally closed solid volumes or closed fluid volumes. Different types of mesh can be created in ANSYS Meshing tool through 'Insert' option as shown in Figure 8. The mesh settings can be adjusted according to the model requirement as seen in Figure 9. After creating the mesh, the proceeds with assigning the 'nameports' for different parts of the model structure. This is helpful for defining the boundary conditions. The 'nameports' can be created by right clicking on the region and selecting 'Create Named Selection' as shown in Figure 10.

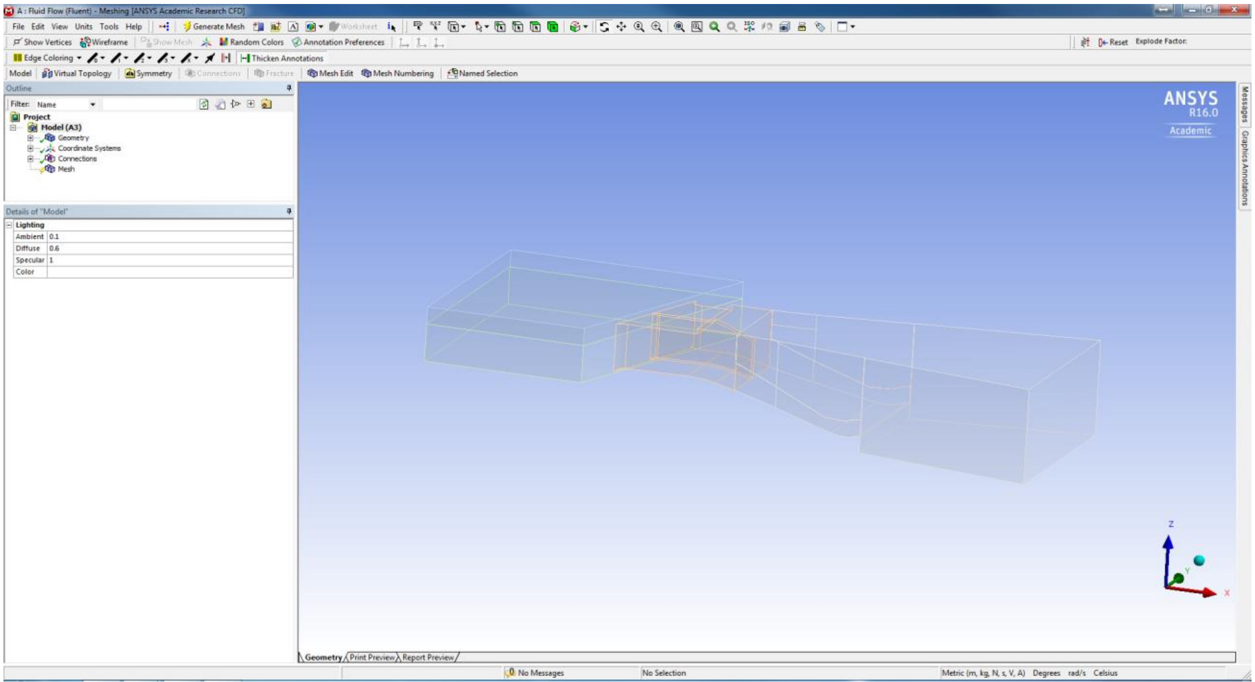

Figure 7. Interface of ANSYS Meshing tool

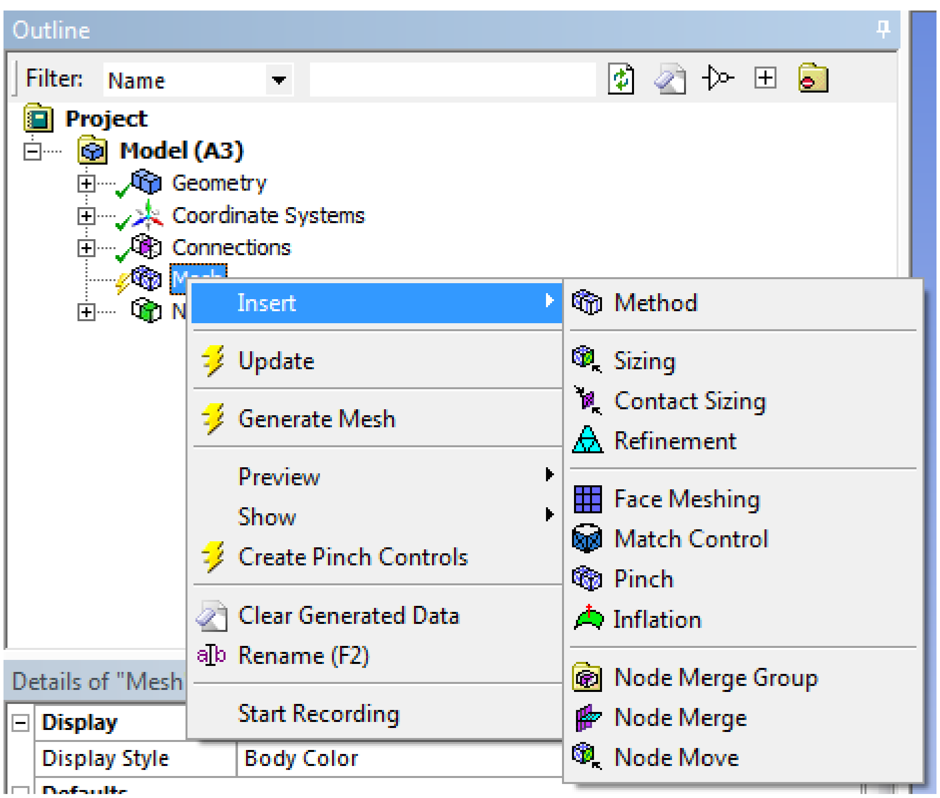

Figure 8. ANSYS Meshing tool

Details of "Mesh"

Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	75
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Medium
Smoothing	Medium
Curvature Normal Angle	Default (12.3750 °)
Min Size	Default (4.4895e-002 m)
Max Size	Default (5.74660 m)
Growth Rate	Default (1.13440)
Minimum Edge Length	8.9757e-006 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	20
Growth Rate	1.2
View Advanced Options	No
Assembly Meshing	
Method	CutCell
Feature Capture	Program Controlled
Tessellation Refinement	Program Controlled
Intersection Feature Creation	Program Controlled
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Statistics	
Nodes	
Elements	
Mesh Metric	Skewness
Min	0.
Max	0.
Average	0.
Standard Deviation	0.

Figure 9. Various mesh settings

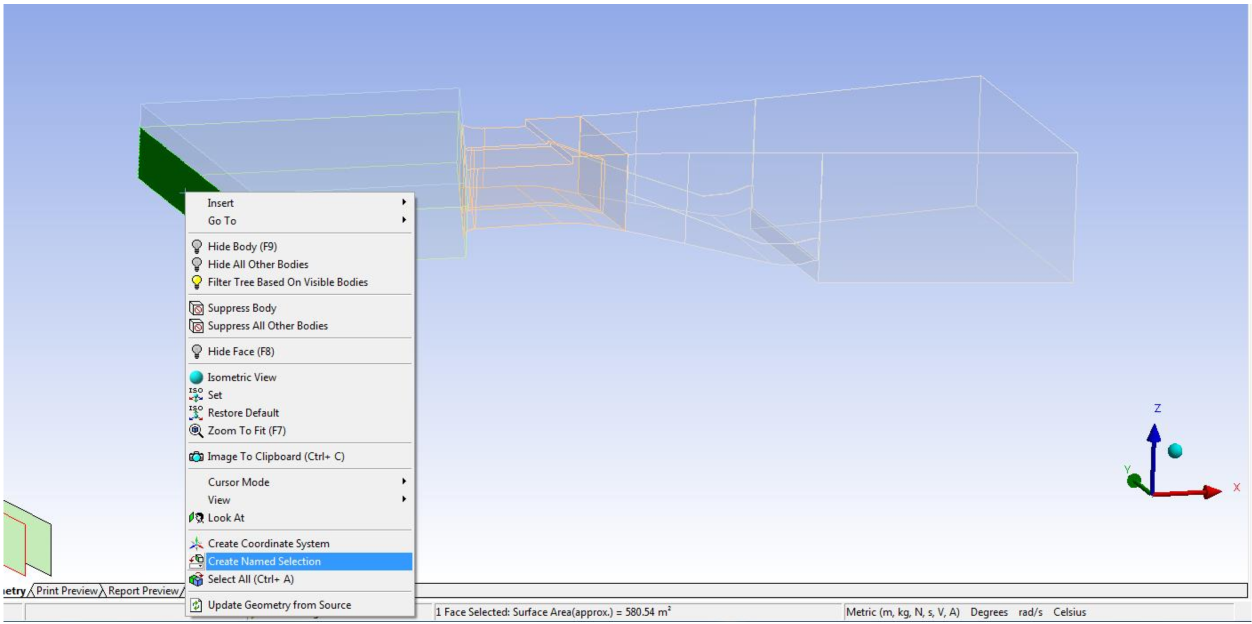

Figure 10. Creating 'nameports'

Figure 11 shows the sample of various regions that have been defined using 'nameports'. The boundary conditions have to be specified in the ANSYS Fluent for all these regions.

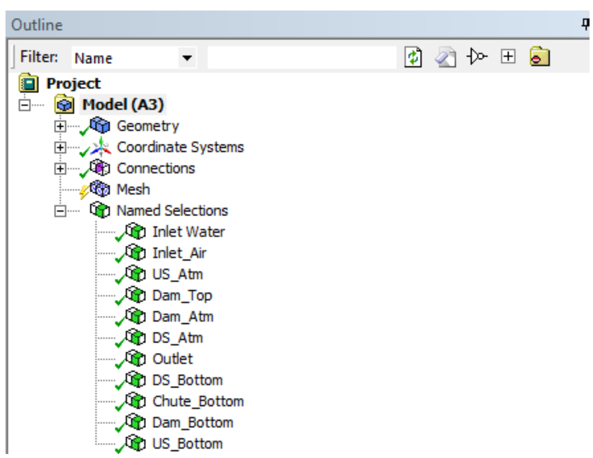

Figure 11. View of sample 'nameports' in ANSYS Meshing tool

Finally, the mesh can be generated by selecting 'Generate Mesh'. The status of mesh generation process is displayed as shown in Figure 12.

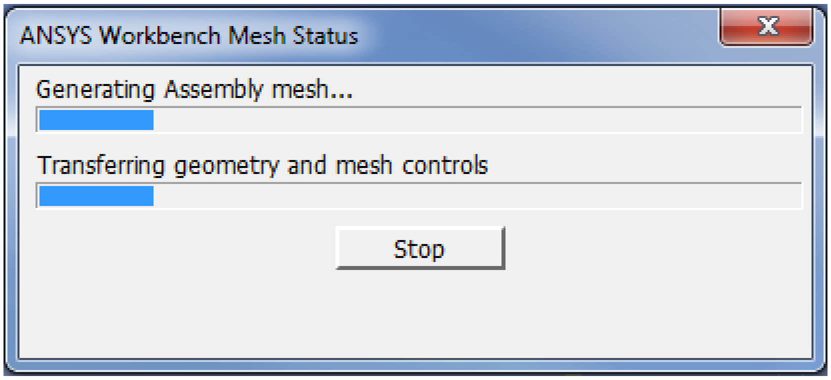

Figure 12. Progress of mesh creation

A fully developed mesh is shown in Figure 13.

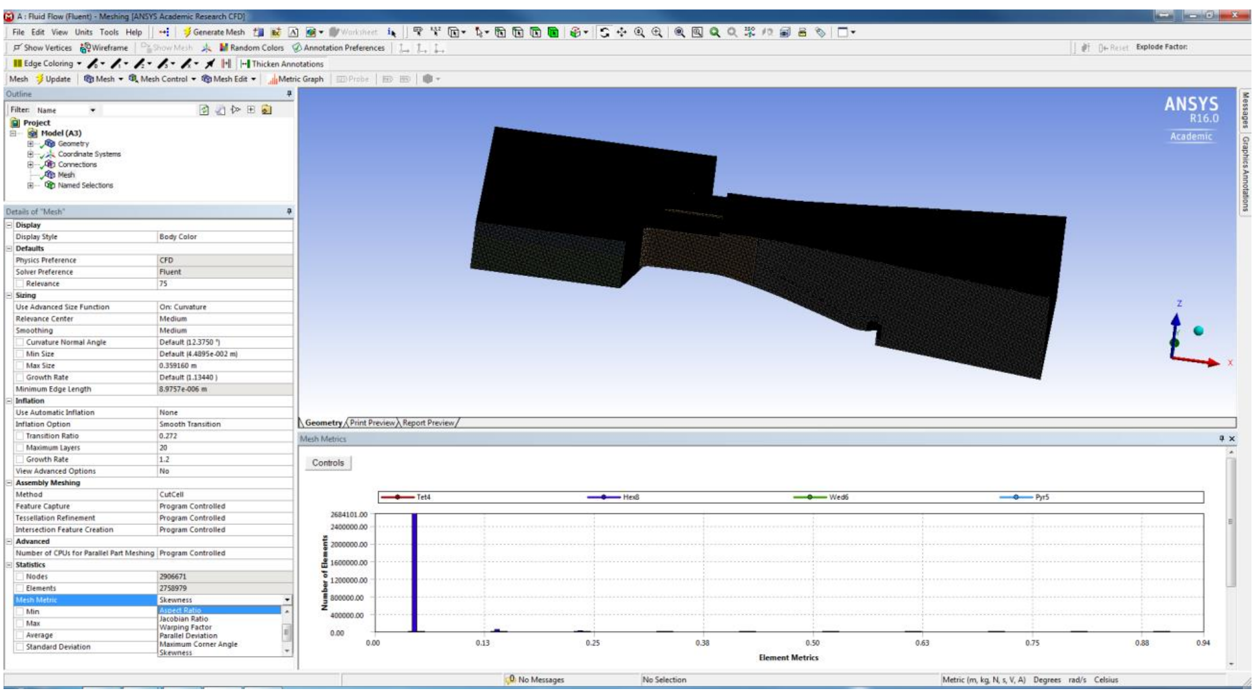

Figure 13. A fully developed mesh using ANSYS-Meshing tool

The developed mesh has to pass certain criteria to be successfully solved in ANSYS Fluent. There are various mesh metrics to check the quality of the mesh. For example, the maximum skewness index should be less than 0.98 as shown in Figure 14.

Statistics	
Nodes	2906671
Elements	2758979
Mesh Metric	Skewness
Min	1.3057e-010
Max	0.9448
Average	6.5991e-003
Standard Deviation	4.5228e-002

Figure 14. Mesh quality matrices

The mesh quality parameters can also be viewed as graphs, as shown in Figure 15.

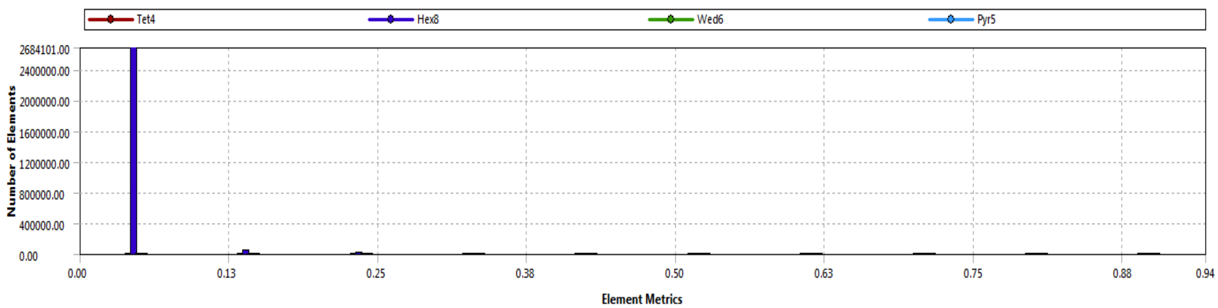

Figure 15. Number of mesh elements

When the mesh is successfully developed, the ANSYS workbench is updated with the green tick mark as shown in Figure 16.

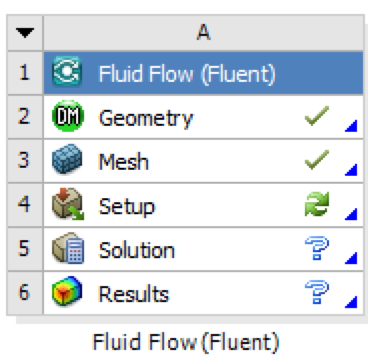

Figure 16. Workflow update - II

2.5. Model Development in ANSYS Fluent

After the mesh generation, the geometry should be exported as a mesh file for use in ANSYS Fluent. The ANSYS Fluent can be launched from the workbench either by double clicking or right clicking and selecting ‘Edit’ in the ‘Setup’ menu. The ANSYS Fluent launch screen is shown in Figure 17(a). While launching the ANSYS Fluent, the user has to select optional modelling aspects like 2D or 3D, Double Precision or Meshing modes, Serial or Parallel processing, etc. as shown in Figure 17(b).

(a) **(b)**

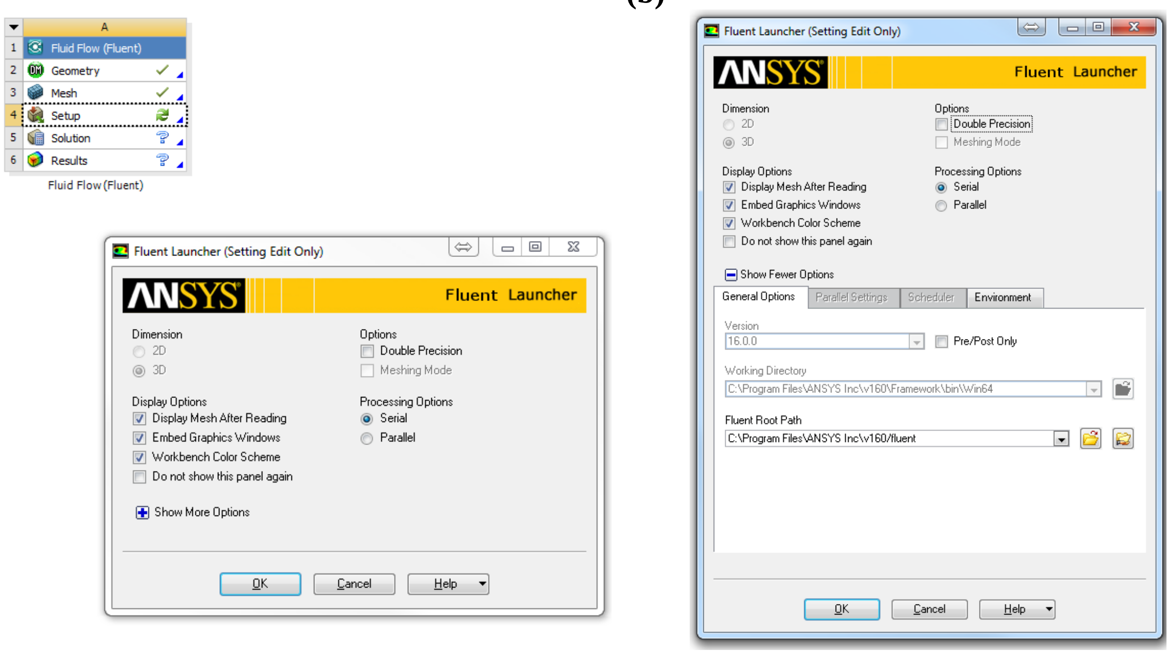

Figure 17. ANSYS Fluent launch screen

The main ANSYS Fluent interface is shown in Figure 18.

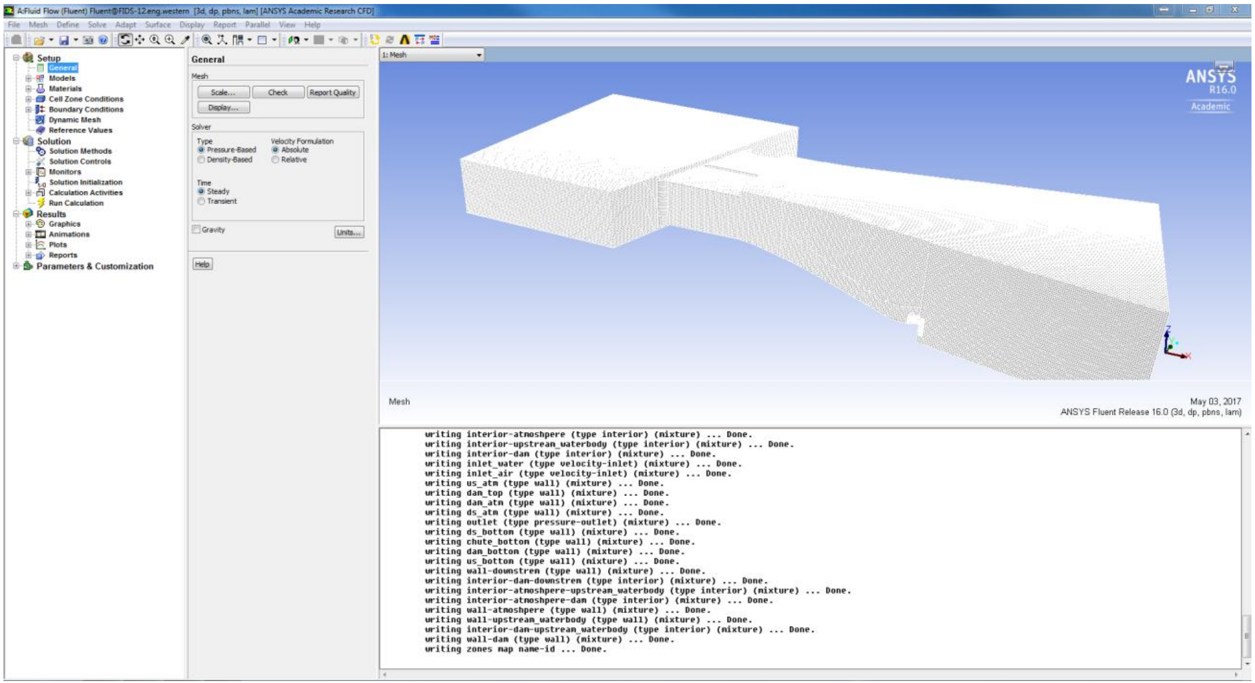

Figure 18. ANSYS Fluent main interface

The steps involved in setting up the ANSYS Fluent model are shown in Figure 19.

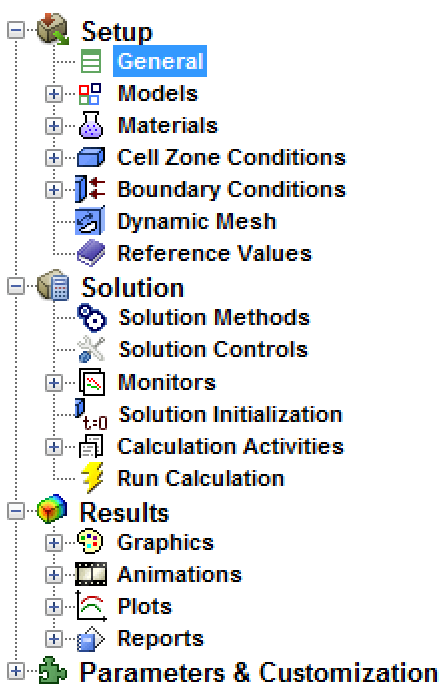

Figure 19. Steps involved in setting up the ANSYS Fluent model

Each step is explained in detail in the further sections.

2.5.1 General Setting

The options for general model setting in ANSYS Fluent is shown in Figure 20. First, developed mesh has to be checked in ANSYS Fluent for its quality. Any irregular shapes or sizes have to be rectified before developing a full model. Only when the parameters such aspect ratio, skewness index, etc. are within the specified range, the model development in ANSYS Fluent can proceed. The checks are performed by selecting 'Check' and 'Report Quality' from the menu screen shown in Figure 20. The 'Time' option is used to select either steady or unsteady state.

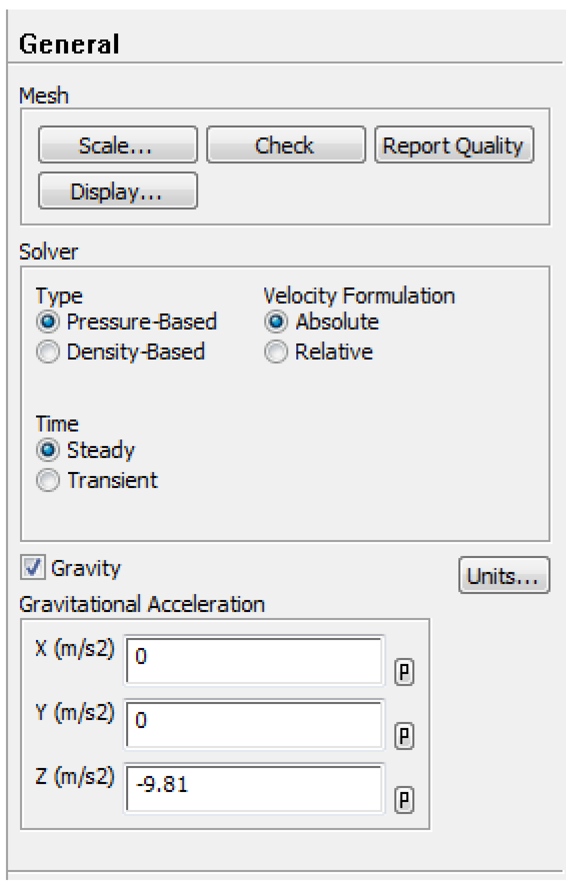

Figure 20. General model setting in the ANSYS Fluent tool

2.5.2 Model Selection

There are various models available within the ANSYS Fluent. A specific model has to be selected for the particular problem at hand. For our spillway example, the flow over a spillway involves two phases, both water and air. Hence, the Multiphase → Volume of Fluid model (VOF) has been selected as shown in Figure 21. It is a multi-phase model best suited for the two phase air-water flow (ANSYS, 2013). In VOF, each phase is considered as the volume fractions (α_1, α_2) and the momentum, heat and mass exchanges are calculated between the phases.

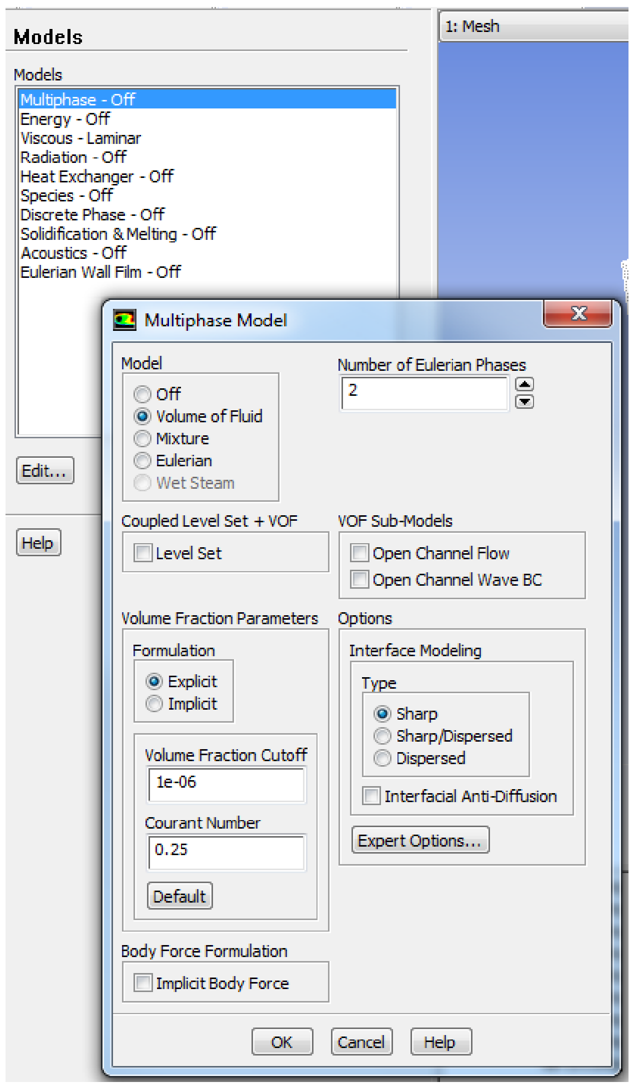

Figure 21 Selection of VOF model

An appropriate model has to be selected to consider the turbulent flow over the spillway. There are various turbulence models available in ANSYS Fluent as shown in Figure 22. In this study, realizable k-ε model (k-epsilon model) has been selected. The realizable k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε).

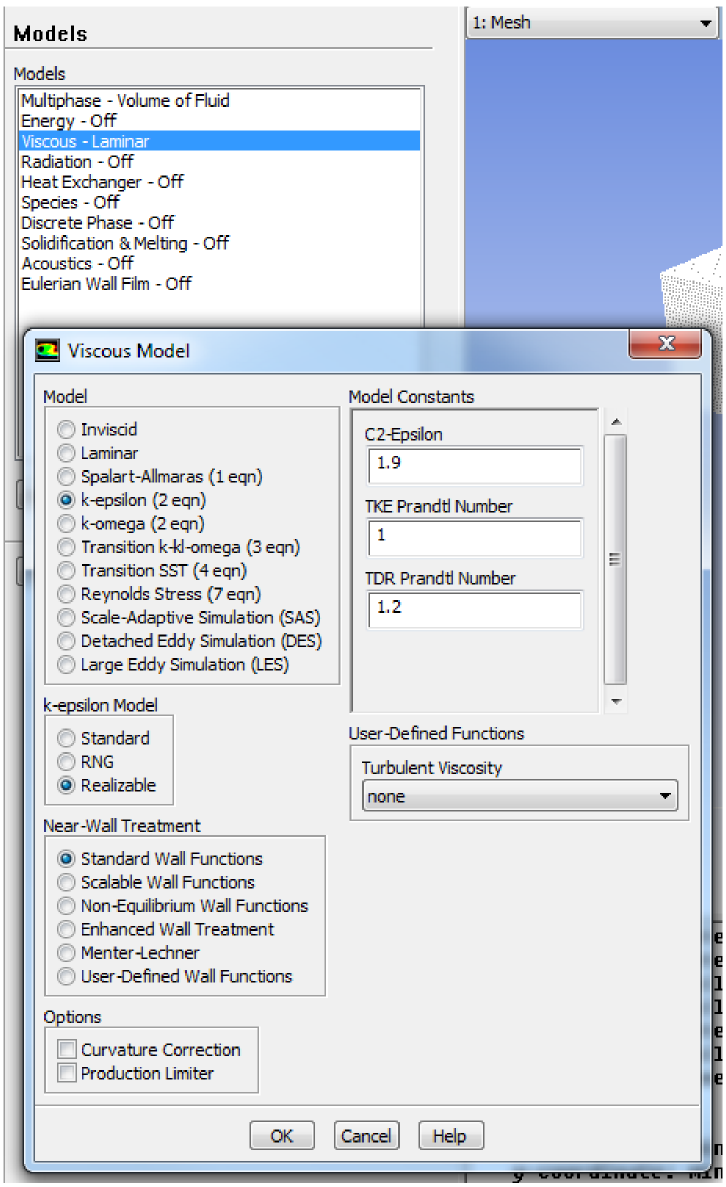

Figure 22. Selecting the turbulence model

2.5.3 Materials

After selecting the appropriate model for the problem, the materials involved in problem have to be defined. ANSYS Fluent has its own database of different "materials" with their properties. Users can also manually input the material with its properties, if it is not available in the database. The selection of materials window is shown in Figure 23. As already mentioned, the spillway is modelled as a multi-phase model and involves both air and water. Hence, air and water materials are selected from the ANSYS Fluent database.

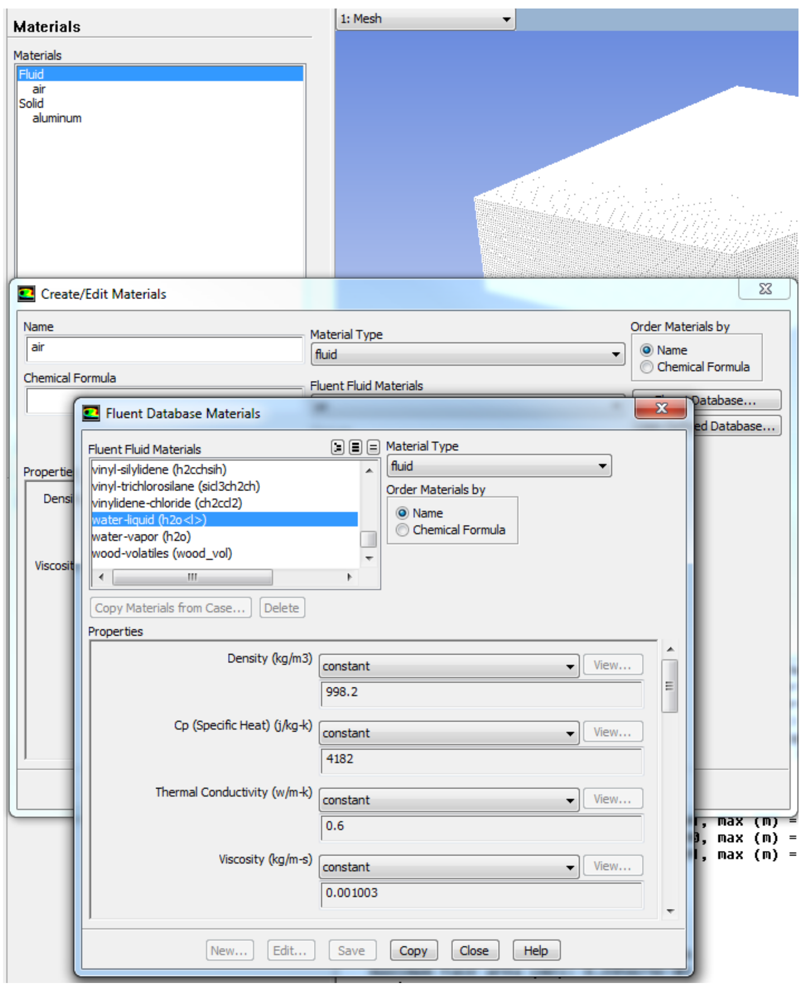

Figure 23. Selection of materials

Then, the primary and secondary phases should be defined. Open channel systems involve the flowing fluid (water) and the fluid above it (air). Usually in VOF, air is defined as the primary phase and water as the secondary phase. This is necessary as the flow over spillway is considered as a free surface two phase flow. Then the phase interactions should be defined as shown in Figure 24.

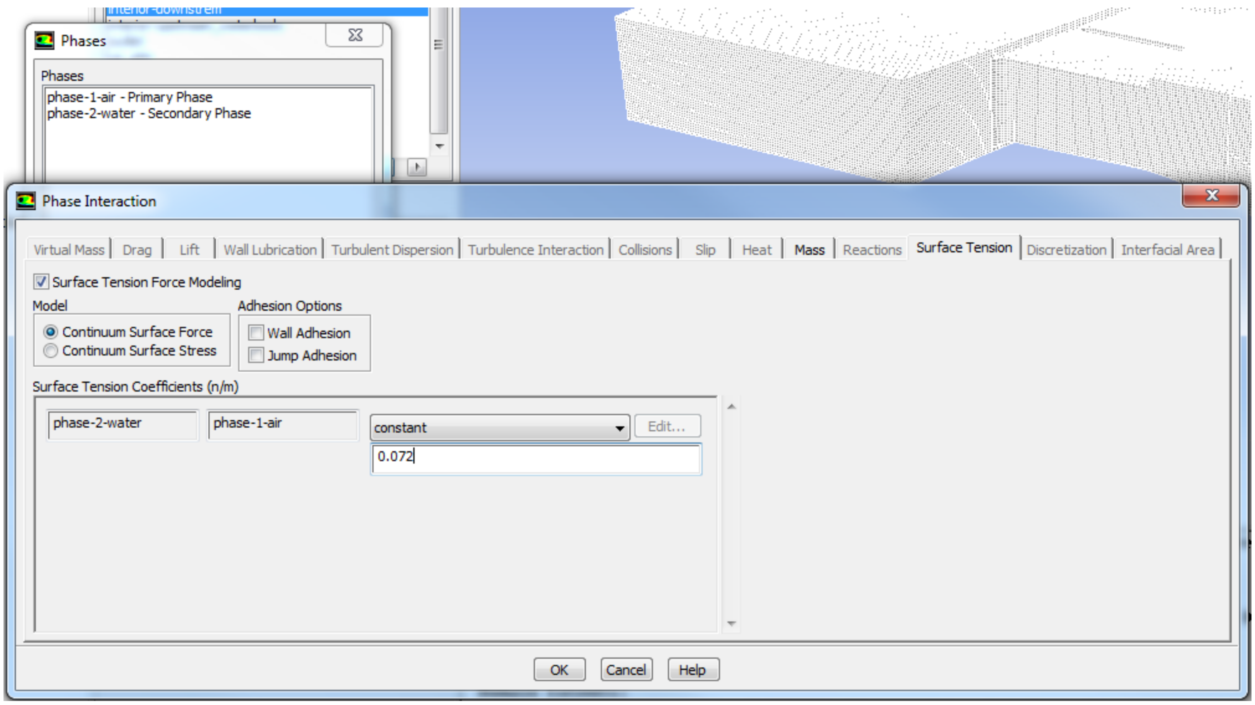

Figure 24. Defining phases and their interactions

2.5.4 Cell Zone Condition

In the Cell Zone Condition window, the model operating conditions have to be defined. The pressure is normally set as atmospheric pressure (101325 Pascal) for the free open flow channels. Reference pressure location and gravity need to be defined as shown in Figure 25. The reference pressure location is the cell whose pressure value is used to adjust the gauge pressure field for incompressible flow that do not involve any pressure boundaries. The gravity option is set on in Z direction at -9.81 m/s^2, which is necessary as the flow over spillway is gravity driven. The negative sign indicates that the gravity is downwards in the Z direction. The variable density parameter is set as operating density for the phase air at 1.225 kg/m^3.

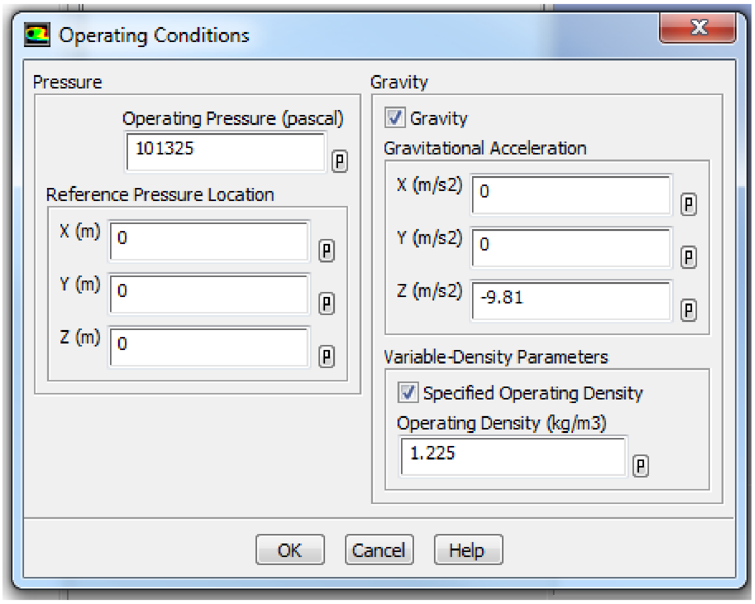

Figure 25. Setting up operating conditions

2.5.5 Boundary Conditions

The most important aspect of setting up the simulation is the definition of the boundary conditions. There are various boundary conditions available in ANSYS Fluent. The boundary conditions for each face/zone have to be selected according to the problem as shown in Figure 26. Basically, the spillway hydraulics have three types of boundary zones viz. pressure inlet, wall and pressure outlet. The upstream end of the reservoir is the flow inlet. The velocity of flow is known, then velocity inlet can be given. For most of the spillway hydraulics problem, the flow velocity is unknown. Hence, the upstream inlet is defined as a pressure inlet (shown in Figure 27) to maintain a reservoir water level. The turbulence specification is given as 'intensity and viscosity ratio', that requires input of turbulence parameters. When a flow enters the domain at the inlet, ANSYS Fluent requires specification of transported turbulence quantities. Turbulence intensity is the ratio of the root mean square of the velocity fluctuations to the mean flow velocity. A turbulence intensity of 1% is considered as low and 10% as high. Turbulent viscosity ratio is directly proportional to the Reynolds number. The surface of the spillway and chute are treated as wall (shown in Figure 28). The default roughness constant of 0.5 is selected, so that it reproduces Nikuradse's resistance data for pipes with uniform sand grain roughness (ANSYS, 2013). The roughness coefficient can be varied to study the effect of roughness and thereby turbulence on the spillway flow and head loss. The downstream boundary was set as pressure outlet. The backflow conditions have to be specified to avoid the reverse flow direction at the pressure outlet boundary during the solution process.

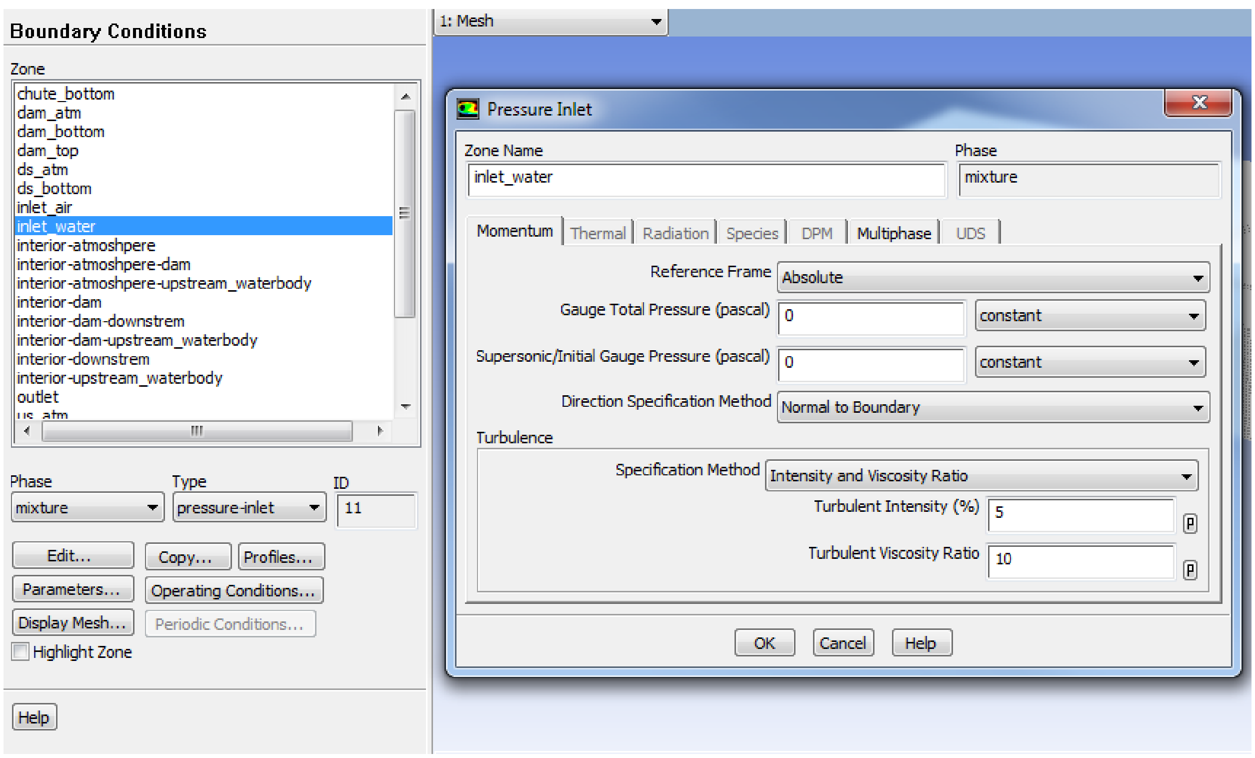

Figure 26 Defining boundary conditions in ANSYS Fluent

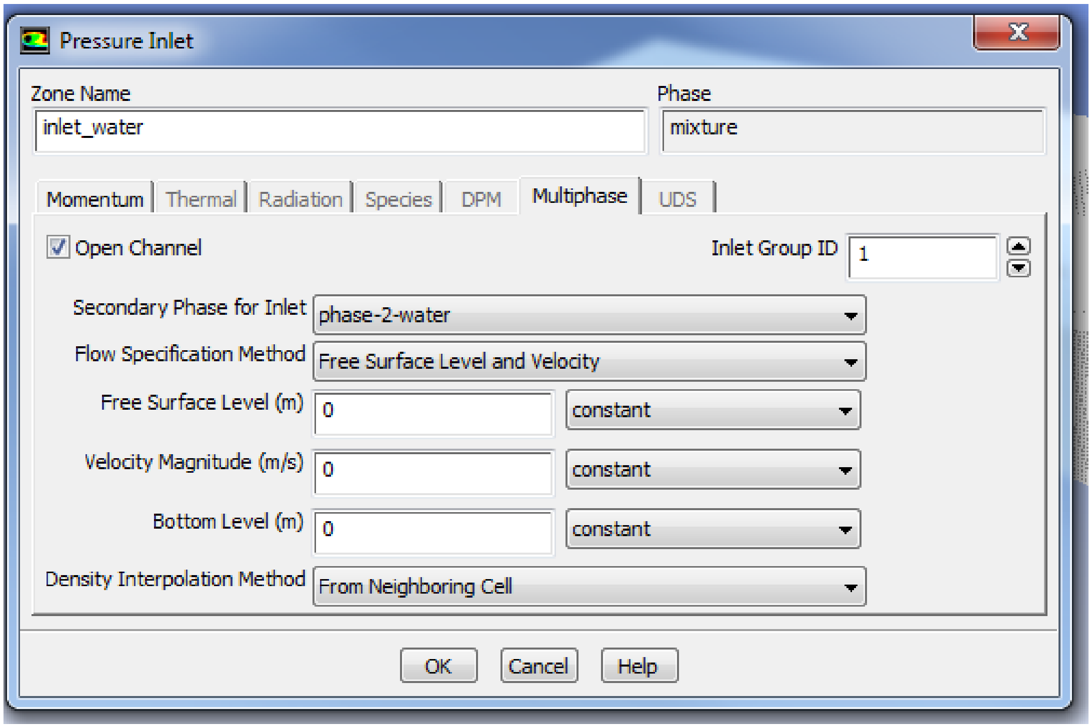

Figure 27. Pressure inlet boundary conditions

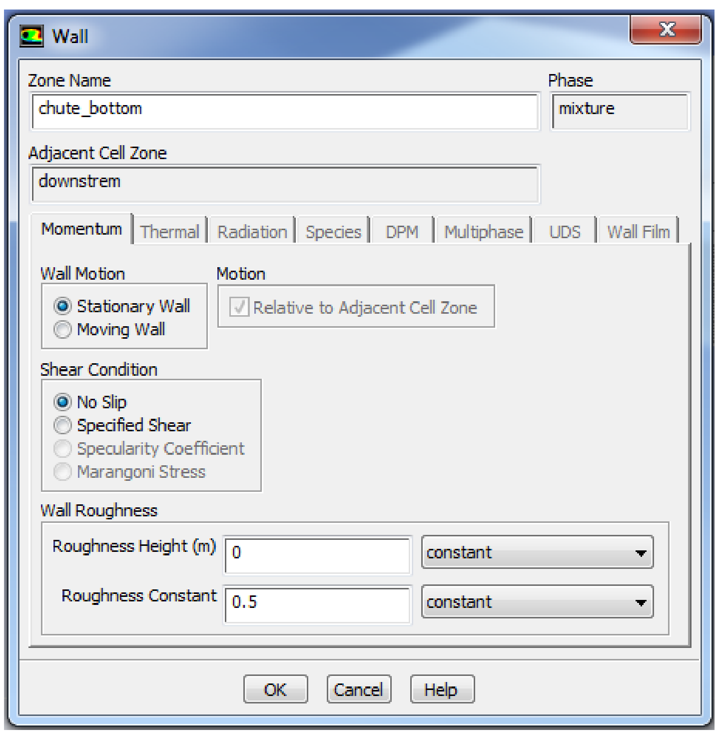

Figure 28. Wall boundary conditions

2.5.6 Solution Methods

There are solution methods available in ANSYS Fluent to solve the model as shown in Figure 29.

The solution methods should be selected according to the problem type.

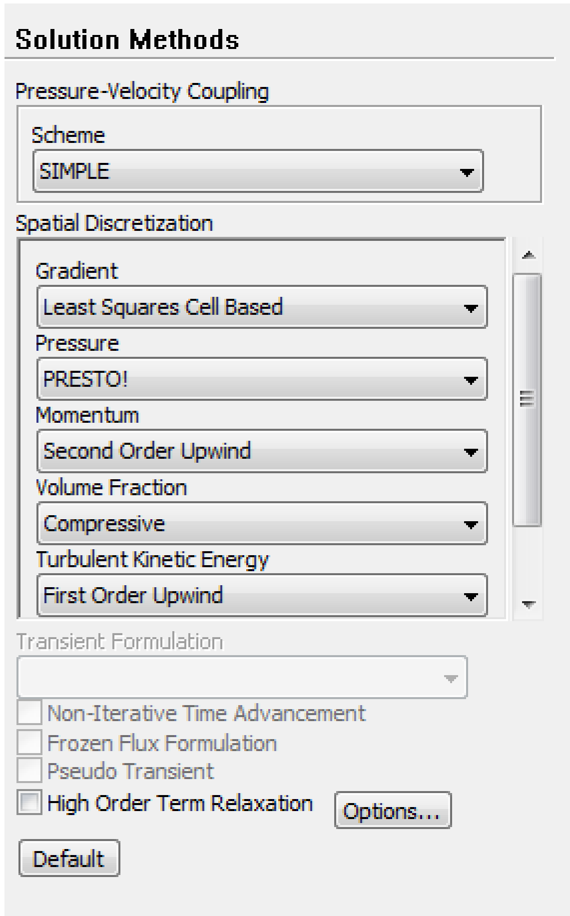

Figure 29. Different solution methods

2.5.7 Solution Control

Sometimes, certain parameter values need to be relaxed for the successful simulation. Those parameter settings can be adjusted as shown in Figure 30.

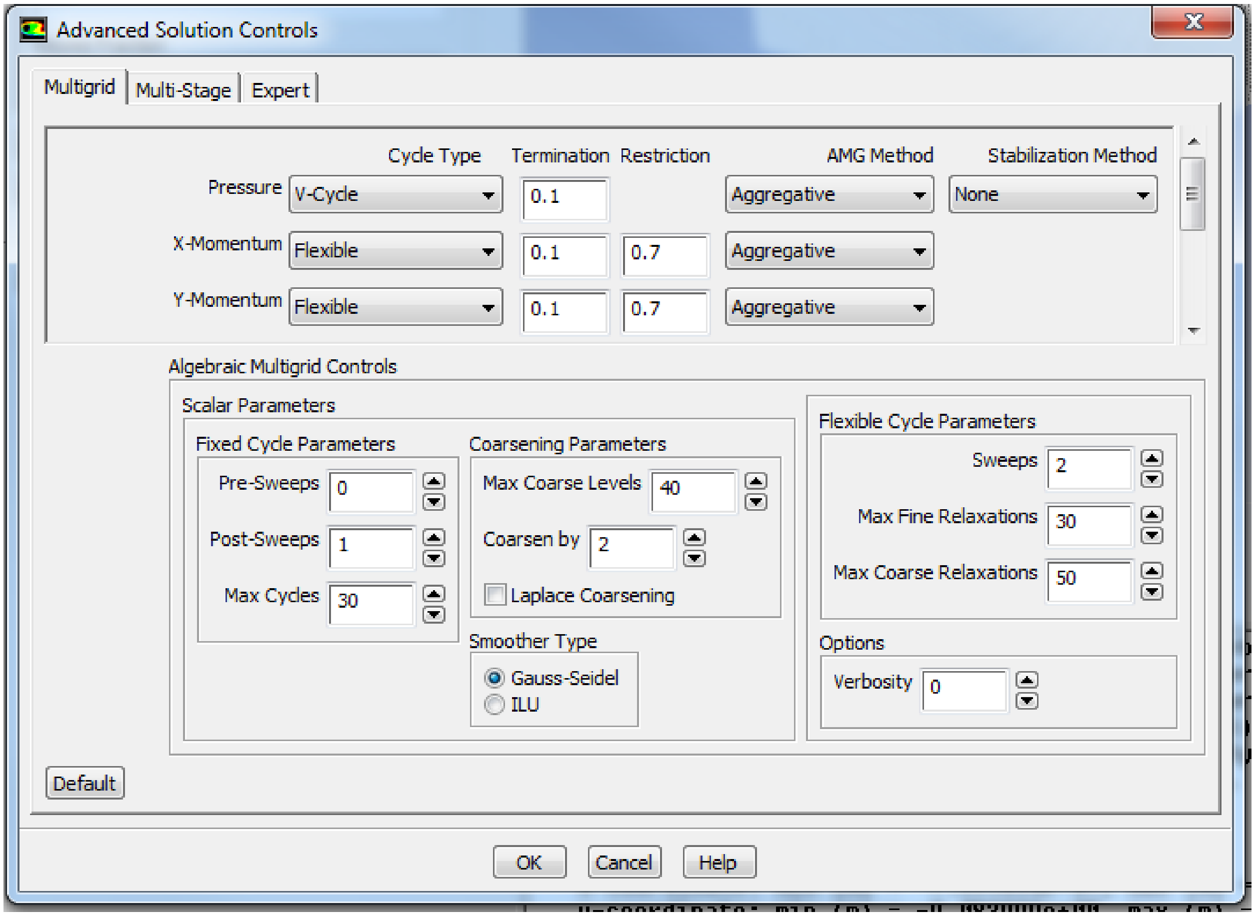

Figure 30. Setting various simulation parameters

2.5.8 Monitors

During the simulation process, users can monitor and visualize certain parameters which are significant for the simulation. Those parameters can be set in monitor control window as shown in Figure 31. The values of the defined parameters will be displayed as graphs in the monitor during the simulation process.

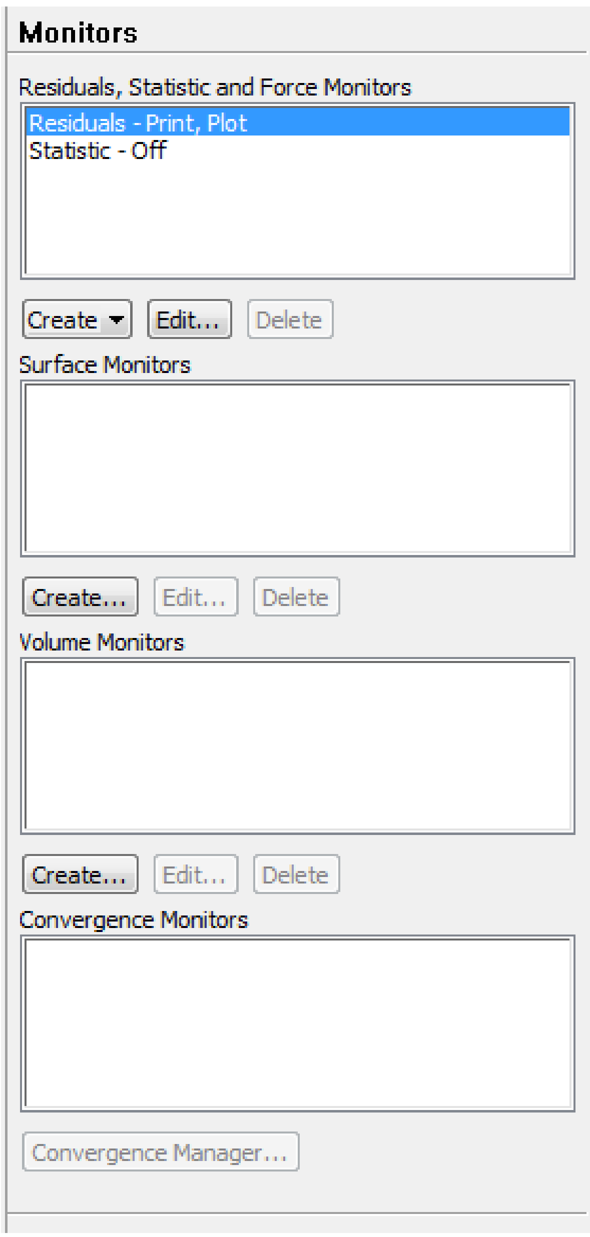

Figure 31. Monitor control

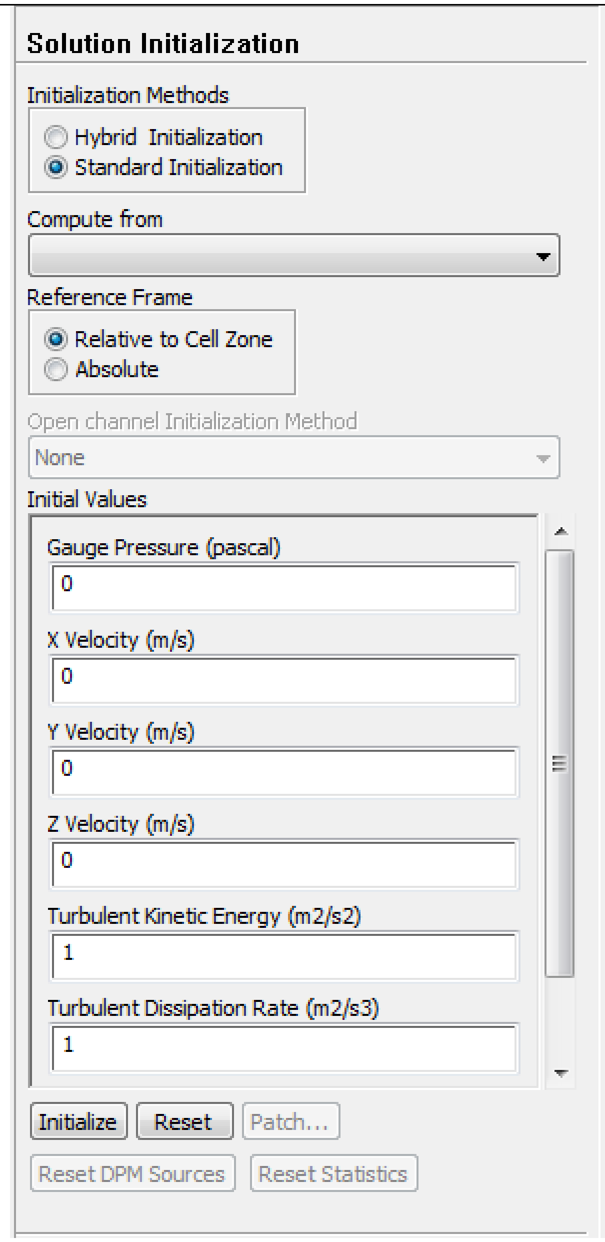

Figure 32. Solution initialization

2.5.9 Solution Initialization

Before starting the CFD simulation, ANSYS Fluent needs the initial value for the solution field.

The solution initialization can be done by two ways as shown in Figure 32. First, the initialization is done for the entire flow field. Then, specific regions or zones are created and patch values or functions for selected flow variables are provided for selected cell zones. In the example simulation, solution is initialized with patch values for the water zone in the reservoir up to a selected reservoir water level. Specific regions are created with the same functions that are used to mark cells for adaptation as shown in Figure 33 and Figure 34.

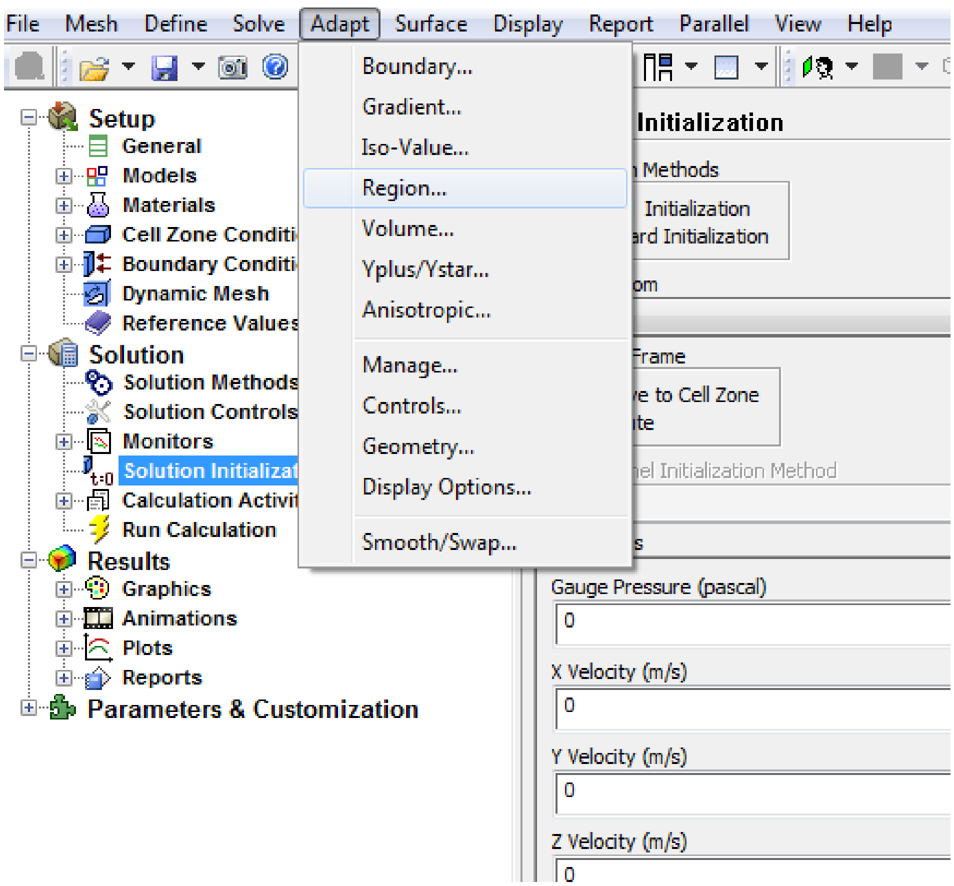

Figure 33. Creating specific cell zones and cell adaptation characteristics

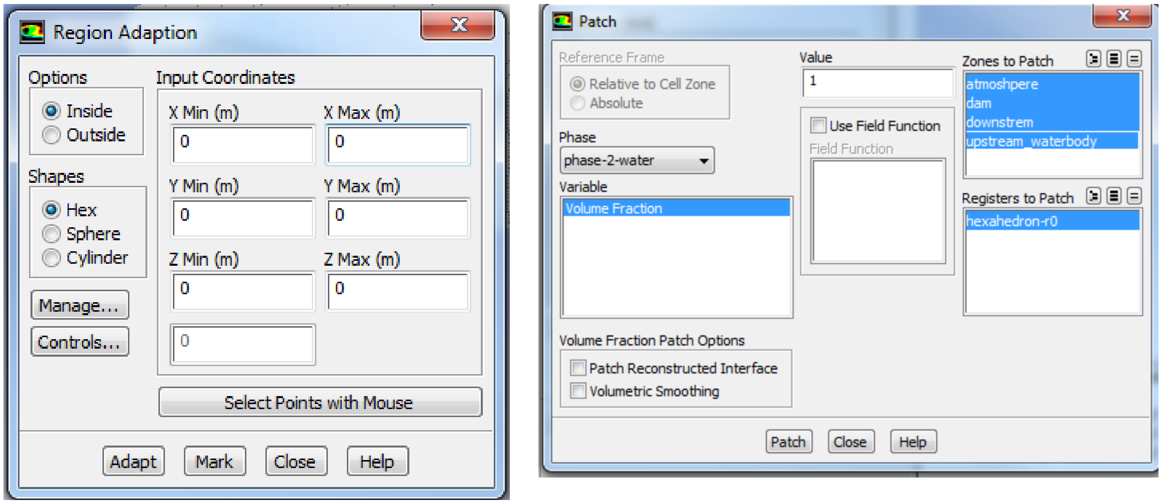

Figure 34. Region adaptation and patching

2.5.10 Run Calculation

Finally, the number of iterations and reporting intervals should be set before starting the calculations as shown Figure 35.

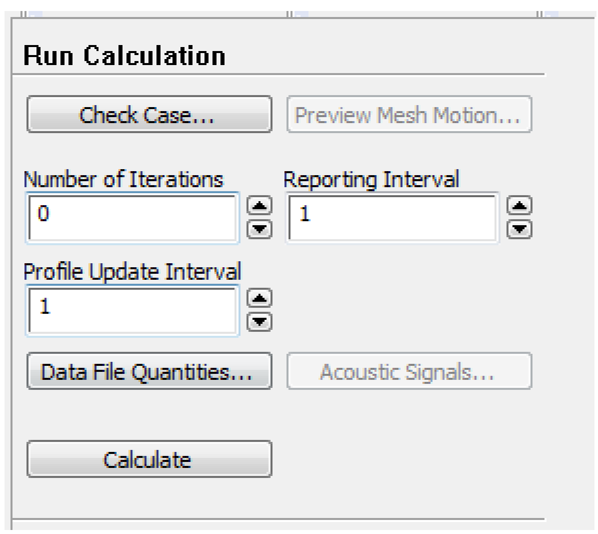

Figure 35 Run calculation

2.6. Results and Post Processing

ANSYS has separate tool for analyzing and post processing the results. User can also do limited processing within the ANSYS Fluent itself. Using the post processing tool, iso-surfaces of grid are created studying the flow features such as velocity, pressure and air profiles in longitudinal direction as well as across the depth. Contours and XY Plots of different quantities such as pressure, velocity and phases can also be generated.

3. Summary

In this report, the detailed guide for developing, simulating and analyzing a CFD model using ANSYS-Fluent software package is discussed. A simple case study of the dam spillway is used to illustrate the process of model development.

Acknowledgments

This work is jointly funded by the National Sciences and Engineering Research Council and BC Hydro. The authors would like to thank BC Hydro for their input to the presented work.

References

ANSYS. (2013). *ANSYS Fluent Theory Guide*. Canonsburg, PA.

ANSYS. (2016). *ANSYS® Academic Research, Release 16.0, ANSYS Fluent in ANSYS Workbench User's Guide*. ANSYS, Inc., Canonsburg, PA.

Aydin, M. C., and Ozturk, M. (2009). "Verification and validation of a computational fluid dynamics (CFD) model for air entrainment at spillway aerators." *Canadian Journal of Civil Engineering*, 36(5), 826–836.

Bhajantri, M. R., Eldho, T. I., and Deolalikar, P. B. (2006). "Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model." *Sadhana*, 31(6), 743–754.

Chanel, P. G., and Doering, J. C. (2008). "Assessment of spillway modeling using computational fluid dynamics." *Canadian Journal of Civil Engineering*, 35(12), 1481–1485.

Chinnarasri, C., Kositgittiwong, D., and Julien, P. Y. (2014). "Model of flow over spillways by computational fluid dynamics.” *Proceedings of the Institution of Civil Engineers - Water Management*, 167(3), 164–175.

Dursun, O. F., and Ozturk, M. (2016). “Determination of flow characteristics of stepped spillways.” *Proceedings of the Institution of Civil Engineers - Water Management*, 169(1), 30–42.

Mohsin, M., and Kaushal, D. R. (2016). “Experimental and CFD Analyses Using Two-Dimensional and Three-Dimensional Models for Invert Traps in Open Rectangular Sewer Channels.” *Journal of Irrigation and Drainage Engineering*, 4016087.

OpenFOAM. (2017). *OpenFOAM: The Open Source CFD Toolbox*.

Rahimzadeh, H., Maghsoodi, R., Sarkardeh, H., and Tavakkol, S. (2012). “Simulating flow over circular spillways by using different turbulence models.” *Engineering Applications of Computational Fluid Mechanics*, 6(1), 100–109.

Savage, B. M., Crookston, B. M., and Paxson, G. S. (2016). “Physical and Numerical Modeling of Large Headwater Ratios for a 15° Labyrinth Spillway.” *Journal of Hydraulic Engineering*, 138(July), 4016046.

Savage, B. M., and Johnson, M. C. (2001). “Flow over Ogee Spillway: Physical and Numerical Model Case Study.” *Journal of Hydraulic Engineering*, 127(8), 640–649.

Appendix A: List of Previous Reports in the Series

ISSN: (Print) 1913-3200; (online) 1913-3219

In addition to 78 previous reports (No. 01 – No. 78) prior to 2012

Samiran Das and Slobodan P. Simonovic (2012). Assessment of Uncertainty in Flood Flows under Climate Change. Water Resources Research Report no. 079, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 67 pages. ISBN: (print) 978-0-7714-2960-6; (online) 978-0-7714-2961-3.

Rubaiya Sarwar, Sarah E. Irwin, Leanna King and Slobodan P. Simonovic (2012). Assessment of Climatic Vulnerability in the Upper Thames River basin: Downscaling with SDSM. Water Resources Research Report no. 080, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 65 pages. ISBN: (print) 978-0-7714-2962-0; (online) 978-0-7714-2963-7.

Sarah E. Irwin, Rubaiya Sarwar, Leanna King and Slobodan P. Simonovic (2012). Assessment of Climatic Vulnerability in the Upper Thames River basin: Downscaling with LARS-WG. Water Resources Research Report no. 081, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 80 pages. ISBN: (print) 978-0-7714-2964-4; (online) 978-0-7714-2965-1.

Samiran Das and Slobodan P. Simonovic (2012). Guidelines for Flood Frequency Estimation under Climate Change. Water Resources Research Report no. 082, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 44 pages. ISBN: (print) 978-0-7714-2973-6; (online) 978-0-7714-2974-3.

Angela Peck and Slobodan P. Simonovic (2013). Coastal Cities at Risk (CCaR): Generic System Dynamics Simulation Models for Use with City Resilience Simulator. Water Resources Research Report no. 083, Facility for Intelligent Decision Support, Department of Civil and Environmental

Engineering, London, Ontario, Canada, 55 pages. ISBN: (print) 978-0-7714-3024-4; (online) 978-0-7714-3025-1.

Roshan Srivastav and Slobodan P. Simonovic (2014). Generic Framework for Computation of Spatial Dynamic Resilience. Water Resources Research Report no. 085, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 81 pages. ISBN: (print) 978-0-7714-3067-1; (online) 978-0-7714-3068-8.

Angela Peck and Slobodan P. Simonovic (2014). Coupling System Dynamics with Geographic Information Systems: CCaR Project Report. Water Resources Research Report no. 086, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 60 pages. ISBN: (print) 978-0-7714-3069-5; (online) 978-0-7714-3070-1.

Sarah Irwin, Roshan Srivastav and Slobodan P. Simonovic (2014). Instruction for Watershed Delineation in an ArcGIS Environment for Regionalization Studies.Water Resources Research Report no. 087, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 45 pages. ISBN: (print) 978-0-7714-3071-8; (online) 978-0-7714-3072-5.

Andre Schardong, Roshan K. Srivastav and Slobodan P. Simonovic (2014). Computerized Tool for the Development of Intensity-Duration-Frequency Curves under a Changing Climate: Users Manual v.1. Water Resources Research Report no. 088, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 68 pages. ISBN: (print) 978-0-7714-3085-5; (online) 978-0-7714-3086-2.

Roshan K. Srivastav, Andre Schardong and Slobodan P. Simonovic (2014). Computerized Tool for the Development of Intensity-Duration-Frequency Curves under a Changing Climate: Technical Manual v.1. Water Resources Research Report no. 089, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 62 pages. ISBN: (print) 978-0-7714-3087-9; (online) 978-0-7714-3088-6.

Roshan K. Srivastav and Slobodan P. Simonovic (2014). Simulation of Dynamic Resilience: A Railway Case Study. Water Resources Research Report no. 090, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 91 pages. ISBN: (print) 978-0-7714-3089-3; (online) 978-0-7714-3090-9.

Nick Agam and Slobodan P. Simonovic (2015). Development of Inundation Maps for the Vancouver Coastline Incorporating the Effects of Sea Level Rise and Extreme Events. Water Resources Research Report no. 091, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 107 pages. ISBN: (print) 978-0-7714-3092-3; (online) 978-0-7714-3094-7.

Sarah Irwin, Roshan K. Srivastav and Slobodan P. Simonovic (2015). Instructions for Operating the Proposed Regionalization Tool "Cluster-FCM" Using Fuzzy C-Means Clustering and L-Moment Statistics. Water Resources Research Report no. 092, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 54 pages. ISBN: (print) 978-0-7714-3101-2; (online) 978-0-7714-3102-9.

Bogdan Pavlovic and Slobodan P. Simonovic (2016). Automated Control Flaw Generation Procedure: Cheakamus Dam Case Study. Water Resources Research Report no. 093, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 78 pages. ISBN: (print) 978-0-7714-3113-5; (online) 978-0-7714-3114-2.

Sarah Irwin, Slobodan P. Simonovic and Niru Nirupama (2016). Introduction to ResilSIM: A Decision Support Tool for Estimating Disaster Resilience to Hydro-Meteorological Events. Water Resources Research Report no. 094, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 66 pages. ISBN: (print) 978-0-7714-3115-9; (online) 978-0-7714-3116-6.

Tommy Kokas, Slobodan P. Simonovic (2016). Flood Risk Management in Canadian Urban Environments: A Comprehensive Framework for Water Resources Modeling and Decision-

Making. Water Resources Research Report no. 095. Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 66 pages. ISBN: (print) 978-0-7714-3117-3; (online) 978-0-7714-3118-0.

Jingjing Kong and Slobodan P. Simonovic (2016). Interdependent Infrastructure Network Resilience Model with Joint Restoration Strategy. Water Resources Research Report no. 096, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 83 pages. ISBN: (print) 978-0-7714-3132-6; (online) 978-0-7714-3133-3.

Sohom Mandal, Patrick A. Breach and Slobodan P. Simonovic (2017). Tools for Downscaling Climate Variables: A Technical Manual. Water Resources Research Report no. 097, Facility for Intelligent Decision Support, Department of Civil and Environmental Engineering, London, Ontario, Canada, 95 pages. ISBN: (print) 978-0-7714-3135-7; (online) 978-0-7714-3136-4.