

Numerical Flow Simulation using Star CCM+

Upendra Rajak, Dr. Vishnu Prasad, Dr. Ruchi Khare

Department of Civil Engineering, M.A. National Institute of Technology, Bhopal, MP, India

*E-mail: upendrarajak86@gmail.com

Phone No: +91 9753509775

ABSTRACT

The accuracy of numerical simulation of any physical process depends on the technique adopted for the simulation and the software used. Many software i.e., ANSYS, PHONICS, NUMECA, STAR-CCM+, etc. have been developed to solve the governing equations in discretizing the flow domain. Each software has its own capabilities and limitations. In the present work, STAR-CCM+ has been used for numerical simulation in draft tube of hydraulic turbine and result are presented in graphical and tabular form.

Keyword: Elbow draft tube, numerical flow simulation, efficiency, head loss.

Nomenclature

γ	specific weight of water(N/m ³)
P03	total pressure inlet (kPa)
P05	total pressure outlet (kPa)
V03	inlet velocity (m/s)
V05	outlet velocity (m/s)
HLD	head loss (m)
HRLD	total relative loss (%)
HRD	head recovery (m)
η_D	draft tube efficiency(%)
g	acceleration due to gravity (m/s ²)
D	inlet diameter (m)
H	height of draft tube (m)
L	length of draft tube (m)

INTRODUCTION

The design of any physical flow system is based on some assumption for simplicity but the flow behavior is to be checked before making the actual system[2].The computational fluid dynamics is efficient and cost effective tool for flow simulation in comparison to experimental testing approach which is time consuming and expensive. With the advancement of computational techniques, and also computer speed, the researchers are seeking answer to the complexity in flow by Simulating the flow by approximate solution of governing equation in specified domain.

In 2004, CD-adapco introduces a new product, STAR-CCM+, “CCM” stands for Computational Continuum Mechanics from STAR-CD. STAR-CCM+ is fully integrated tool allows user to build geometry as well as provides engineering physics simulation inside a single integrated package[5, 8]. There is inbuilt geometry modeler within the STAR-CCM + environment, as well as it has facility to import external CAD models and modifying them before running the simulation. In STAR-CCM+ analysis of the flow behavior is possible while the simulation is running [6,7]. It can be applied for the solution of Aerospace, Automotive, Biomedical, Chemical, Power Generation. It can also be used for the numerical flow simulation of various hydraulic machine, pumps and turbines etc.

One of the important components of reaction turbine, from the aspect of efficiency is draft tube, which connects runner exit to tail race and converts major part of kinetic energy coming out from runner into pressure energy and therefore has direct impact on the total efficiency of hydroelectric power plant [2, 4].The present work deals with the implicit unsteady flow simulation of elbow draft tube of a reaction turbine using Star CCM+. The analysis is carried out for velocity and pressure variation with different Courant numbers.

GEOMETRIC MODELING

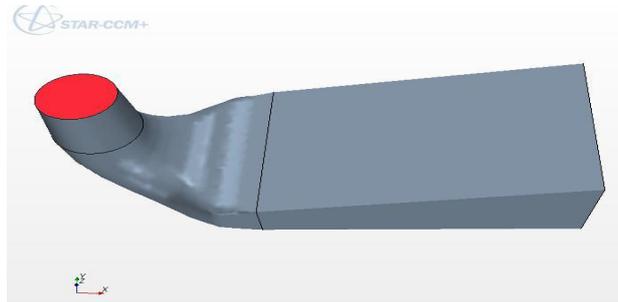


Fig.1 MODEL OF DRAFT TUBE

The fig. 1 shows the geometric model of elbow draft tube of length (L) 11.0 m, inlet diameter (D) 1.726 m and height (H) 2.542 m.

MESHING

The size and shape of element and density of meshing affects the accuracy of numerical simulation. The mesh continua has been used for selection of mesh model and to define the base size of mesh.



Fig.3.MESH MODEL OF DRAFT TUBE

Fig.2 shows the tetrahedral volume mesh of draft tube generated in STAR-CCM+. Different types of mesh models are available in STAR-CCM+ to carry the meshes on surface inside flow domain.

PARAMETER SPECIFICATIONS

The following properties parameters are to be define for simulation:

Material

The material property like density, viscosity etc. are to be defined.

Implicit Unsteady model

Segregated fluid energy and segregated models are only applied for implicit unsteady flow[1]. This model is alternative to explicit unsteady. The choice between models is based on the time scales of the phenomena of interest at each time-step. Implicit unsteady model uses implicit solver, its function is to control the update at each physical time for the calculation. It also controls the time-step size.

Region & boundary condition

Volume domains in space that are completely surrounded by boundaries is termed as regions. The inlet velocity in three directions are defined as inlet boundary condition. The value of velocity are given in table.

TABLE NO.1

	X(m/s)	Y(m/s)	Z(m/s)
Inlet velocity	-0.0422	0.0148	-7.69
Inlet reference pressure			1 (atm.)
Outlet relative pressure			0 (atm.)

The static pressure equal to 1atm has been taken as outlet boundary conditions.

Derived parts

A derived part is created using one or more other parts as parent parts, or inputs, and it is dependent on these parent parts. If any of the parent parts change, all parts depending on those parent parts are automatically recalculated and redisplayed to reflect the change [6].

Editing and addition can be done in derived parts to examine the solution, derived parts get its data from the input parent data. Here in the present example streamline and iso- surface is used as derived parts.

Turbulence Models

The turbulence are used to capture the turbulence scale to predict the turbulence behavior. The selection of model depends on nature of boundaries of and Reynolds number[7].

The segregated flow implicit solver, K-Epsilon Turbulence are defined as the solving parameters in the present example.

TERMINATION OF SIMULATION

The computations for simulation are to be stopped after it meets some specific requirements like number of iteration, value of residuals etc.

In this criteria it specifies how long the solution should run and under what conditions it should stop iterating and/or marching in time. Each enabled criterion is evaluated at the completion of every iteration and time step and a check is used to determine if the requirement of all the criteria are met to stop the solver.

Maximum physical time

Maximum physical time step define the total time step in analysis. In this present work 8 step has been taken as the maximum physical time.

Maximum step

Maximum number of iteration in steady-solver and unsteady-solver analysis can be set as an stopping criteria to stop the simulation.

FORMULE USED FOR CALCULATION

The following formula used for calculation of head loss, head recovery, efficiency, and relative loss[3]:

Head loss in draft tube

$$H_{LD} = \frac{(P_{03} - P_{05})}{\gamma}$$

Head recovery in draft tube

$$H_{RD} = \frac{(V_{03}^2 - V_{05}^2)}{2g} - H_{LD}$$

draft tube Efficiency of the

$$\eta_D = \frac{2gH_{RD}}{V_{03}^2} \times 100$$

Relative loss

$$H_{RLD} = \frac{2gH_{LD}}{V_{03}^2} \times 100$$

RESULT AND DISCUSSIONS

The pressure and velocity distributions in the flow domain are obtained from numerical simulation. These are used for computation of head loss and efficiency of draft tube.

The viscous flow unsteady analysis has been carried out in draft tube of length 11.0 m, height 2.542 m and diameter 1.726 m and result are presented in tabular and graphical form. The boundary conditions are kept same for time steps. It is seen that the velocity at the centre of the outlet is more than the velocity at the periphery of the outlet. The values of velocity and pressure were calculated using the report on mass flow average basis as a field function.

TABLE NO.2
Average value of Pressure and Velocity.

Time step	Inlet		Middle section		Outlet	
	Velocity (m/s)	Pressure (kPa)	Velocity (m/s)	Pressure (kPa)	Velocity (m/s)	Pressure (kPa)
0.2	7.7	4.67	3.306	2.391	1.684	1.525
0.25	7.7	6.49	3.563	3.322	1.444	1.084
0.5	7.7	8.43	4.356	5.434	1.833	1.951
1.0	7.7	8.10	4.331	5.164	1.879	2.029
1.5	7.7	9.22	4.266	6.288	2.341	3.112
2.0	7.7	9.38	4.242	6.429	2.528	3.663
2.5	7.7	8.54	4.328	5.468	2.066	2.459
3.0	7.7	9.2	4.238	6.243	2.537	3.653

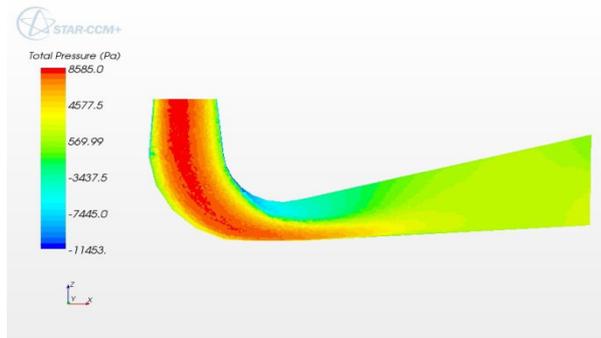


Fig.13. (a): Pressure contour at 0.2 sec time step

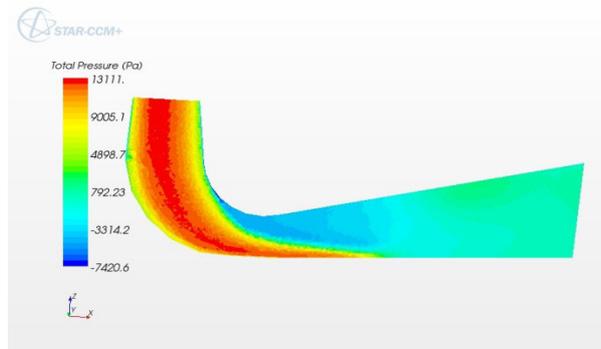


Fig.13. (b) Pressure contour at 1.5 sec time step

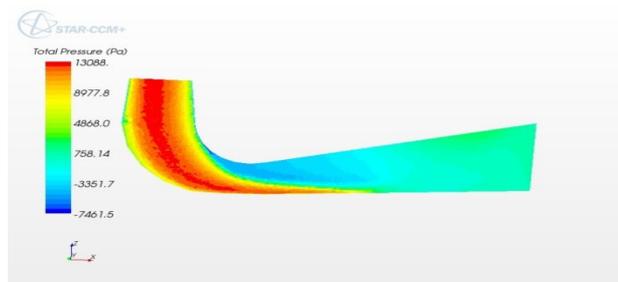


Fig.13. (c): Pressure contour at 3.0 sec time step

The Fig.13 a, b, and c shows the variation of total pressure distribution on longitudinal plane created from inlet to outlet in draft tube at different time steps of 0.2 sec, 1.5 sec, and 3.0 sec respectively. It is seen that in fig. 13(a), the maximum and minimum pressure values are 8.5kPa and -1.4kPa. The pressure variation in fig. 13(b) is between 13.11KPa and -7.4KPa while in fig. 13(c), the variation is between 13.80KPa and -7.46KPa. This indicates that at small time step, the pressure variation is less and become nearly constant at higher time steps.

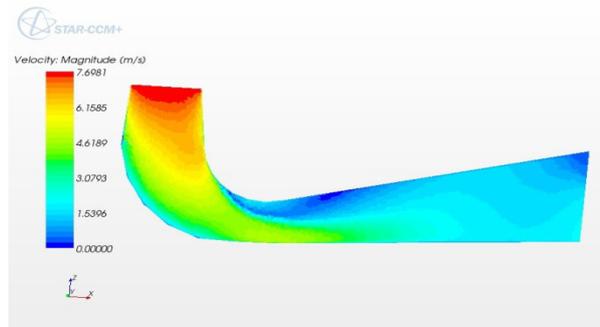


Fig.14. (a):Velocity contour at 0.2 sec time step

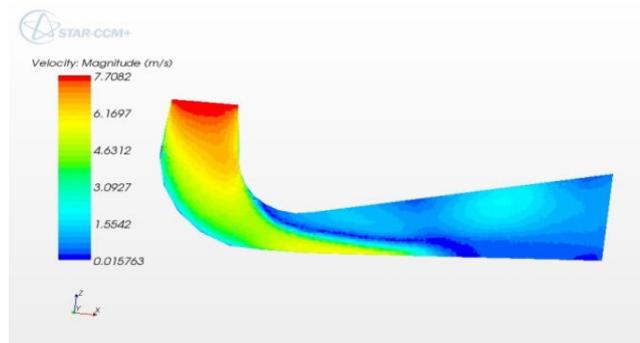


Fig.14. (b): Velocity contour at 1.5 sec time step

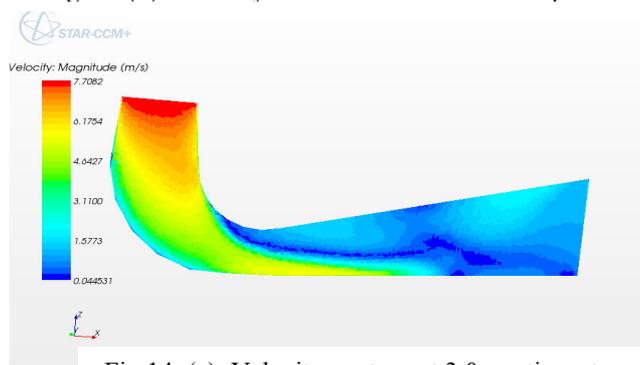


Fig.14. (c): Velocity contour at 3.0 sec time step

The variations of velocity for different time steps of 0.2 sec, 1.5 sec, and 3.0 sec are shown in fig. 14(a), (b) and (c) respectively on the same longitudinal plane. It is observed that velocity decreases from inlet to outlet in all cases due to enlargement of flow area. The velocity distribution is more uniform at small time step. It is more at outlet at where pressure is low.

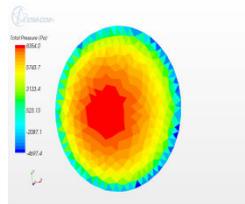


Fig.15(a)

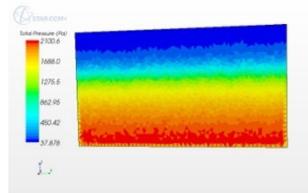


Fig.15(b)

Fig.15 (a, b) pressure contour at inlet and outlet (0.2 sec) time step.

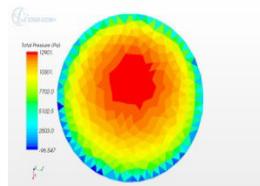


Fig.(c)

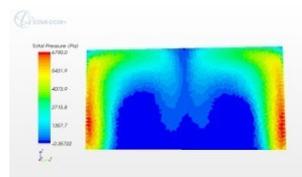


Fig.(d)

Fig.15 (c, d) pressure contour at inlet and outlet (1.5 sec) time step.

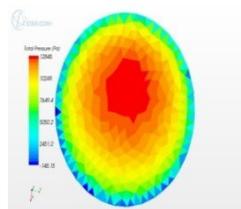


Fig.(e)

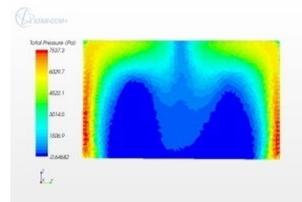


Fig.(f)

Fig.15 (e, f) pressure contour at inlet and outlet (3.0 sec) time step.

Figures 15(a) to (f) show the pressure contours at inlet and out for different time steps. At inlet, pressure is maximum at centre and decreases towards periphery for all time steps. The pressure distribution at outlet for time step of 0.2 sec indicates that pressure decreases gradually from bottom to top. As the time step is increased, pressure distribution become becomes very uneven and gives min pressure at bottom and maximum at sides of outlet section,

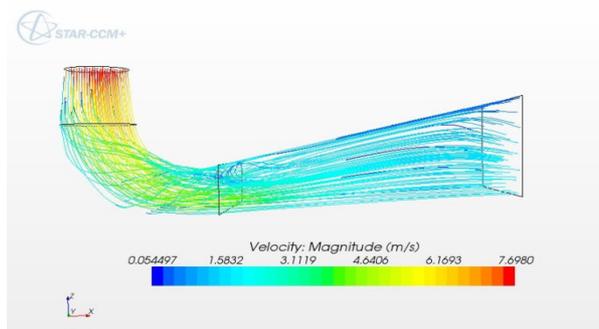


Fig.16. (a): velocity stream line at 0.2 sec time step

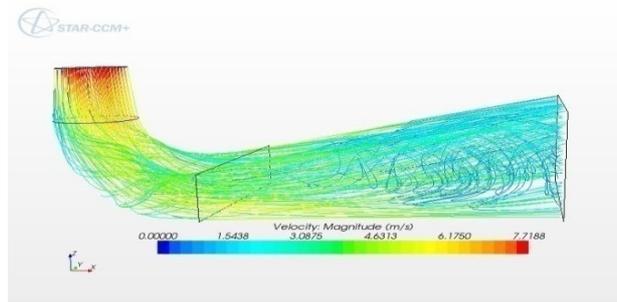


Fig.16. (b): velocity stream line at 1.5 sec time step

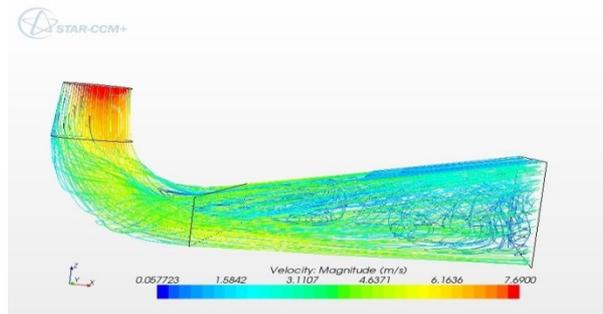


Fig.16. (c): velocity stream line at 3.0 sec time step

The streamline of flow are shown in Fig.16 (a) to (c) at different time steps of 0.2 sec, 1.5 sec, and 3.0 sec respectively. The above figure shows the maximum swirl at 3.0 sec as compared to 0.2 sec and 1.5 sec. It is seen that stream lines are nearly parallel at 0.2 sec time steps and as the time step is increased, flow becomes very turbulent with eddies as given in fig. 16 (b) and (c).

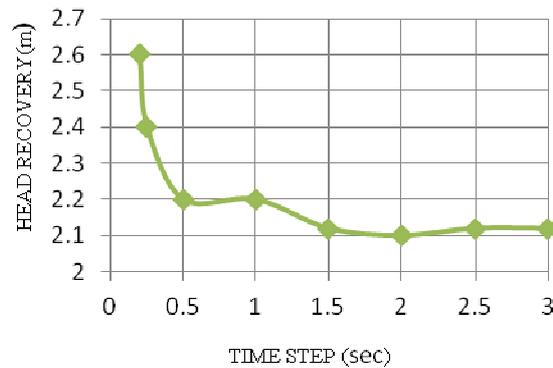


Fig.17. (a): Draft tube head recovery with time step.

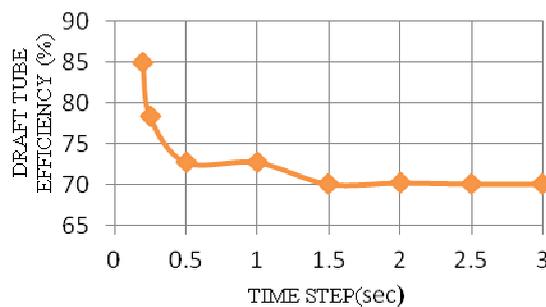


Fig.17. (b): Draft tube efficiency with time step.

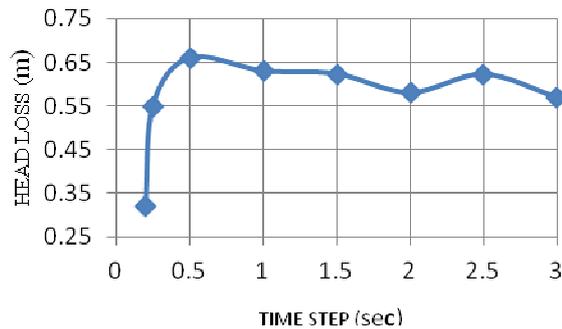


Fig.17. (c): Draft tube head loss with time step

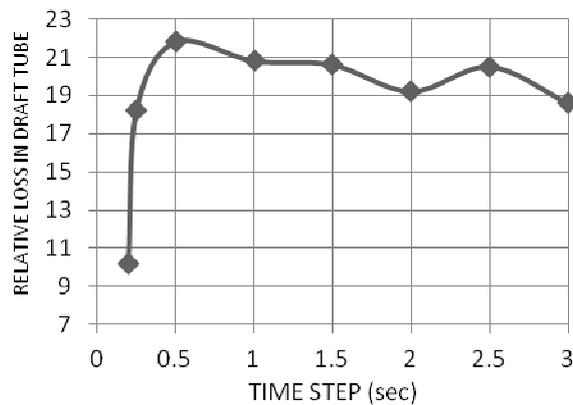


Fig.17. (d): Draft tube relative loss with time step

The head or energy recovery in draft tube and its efficiency decreases with increase in time steps initially and becomes constant after time step of 1.5 sec in fig. 17(a) and (b). The head loss and relative head loss in fig. 17(c) and (d) has reverse pattern to that of head recovery and efficiency with time step.

CONCLUSIONS

It is seen from the results of numerical simulation in draft tube that time step has significant effect on pressure and velocity distribution and also on the hydraulic performance of draft tube. The variation of pressure and velocity from inlet to outlet are similar to theoretical energy change with flow area. STAR-CCM+ may be very useful tool for performance of hydraulic systems and design optimization.

REFERENCE

1. ARPE Jorge, "Pressure Wall Measurement in The Whole Draft Tube Steady and Unsteady Analysis", Proceeding of the XXI IAHR Symposium on Hydraulic Machine and Systems September 9-12, 2002, Lausanne.
2. Khare Ruchi, Prasad Vishnu, Mittal Sushil Kumar, "Effect of Runner Solidity on Performance of Elbow Draft Tube", Energy Procedia 14(2012), pp 2054-2059.
3. M.F Gubin, "Draft Tubes of Hydraulic Station", Amerind Publishing Company Pvt. Ltd, 1973 .
4. Prasad Vishnu, Khare Ruchi, Abhas Chincholikar, "Hydraulic Performance of Elbow Draft Tube for Different Geometric Configurations Using CFD", IGHEN-2010, Oct.21-23, 2010, AHEC, IIT Roorkee, India.
5. Used Software STAR-CCM+(6.06.016).
6. STAR-CCM+(6.06.016) user manual.
7. <https://cd-adapco.secure.force.com>
8. <http://en.wikipedia.org/wiki/CD-adapcom>
9. <http://www.cdapco.com/products/starccmplus>

This academic article was published by The International Institute for Science, Technology and Education (IISTE). The IISTE is a pioneer in the Open Access Publishing service based in the U.S. and Europe. The aim of the institute is Accelerating Global Knowledge Sharing.

More information about the publisher can be found in the IISTE's homepage:

<http://www.iiste.org>

CALL FOR PAPERS

The IISTE is currently hosting more than 30 peer-reviewed academic journals and collaborating with academic institutions around the world. There's no deadline for submission. **Prospective authors of IISTE journals can find the submission instruction on the following page:** <http://www.iiste.org/Journals/>

The IISTE editorial team promises to review and publish all the qualified submissions in a **fast** manner. All the journals articles are available online to the readers all over the world without financial, legal, or technical barriers other than those inseparable from gaining access to the internet itself. Printed version of the journals is also available upon request of readers and authors.

IISTE Knowledge Sharing Partners

EBSCO, Index Copernicus, Ulrich's Periodicals Directory, JournalTOCS, PKP Open Archives Harvester, Bielefeld Academic Search Engine, Elektronische Zeitschriftenbibliothek EZB, Open J-Gate, OCLC WorldCat, Universe Digital Library, NewJour, Google Scholar

