Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Effect of Stairs Fixed in the Rectangular Channel on the Non–Newtonian Fluid Flow; Computational Analysis

Nisar Ahmed

M.Phil., Department of Mathematics, Shah Abdul Latif University Khairpur Mir's nisarahmedsoomro385@gmail.com

Ihsanullah

M.Phil., Department of Mathematics, Shah Abdul Latif University Khairpur Mir's seelroihsanullah@gmail.com

Mansoor Ali

M.Phil., Department of Mathematics, Shah Abdul Latif University Khairpur Mir's mansoorsangi422@gmail.com

Afshan Parveen Dehraj

Assistant Professor, GRA-GDC Kandiaro. afshankb02@gmail.com

Abdul Samad Shaikh

M.Phil. Scholar, Department of Mathematics, Shah Abdul Latif University Khairpur Mir's samadshykh78@gmail.com

Dr. Hisam Uddin Shaikh

Professor, Department of Mathematics, Shah Abdul Latif University Khairpur Mir's

ABSTRACT

The steps hindrance flow is widely utilized in the enormous different areas of biosciences and hydraulic engineering the hindrances came along with the advent the various diseases in the blood flows of the human organs in the body due to the stenosis, heavy stresses and increased the fat and many more. And likewise, the obstruction of the deep leap which it takes in the water canals and most river actions. As such, selected a double step flows fixed in the rectangular channel with the analysis of the various type of behave of fluids particularly the change in the rate of viscosity of the fluids. To analyze the fluids, the mathematical equations of fluid motion were chosen as momentum equation where both fluid parameters were that of pressure and velocity and combined with the continuity equation to demonstrate

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

the continuity transported the fluid flow characteristics of the two step flows fixed to the lower wall of the channel. Owing to highly complicated flow issues, utilized the utilization of the numerical computations through the assistance of the finite element process borrowed on the computational platform COMSOL Multiphysics. The problem is first resolved in this platform with the necessary length of the body including the addition of the vertices, edges and faces. Following that to do the numerical calculations, generated the corresponding mesh of the geometry. The fluid components will be viewable through the streamline laminar flow patterns with the various investigations of the Reynolds numbers and will alter the power law indices rate due to the various fluid behave. As the phenomena of the advent different eddies were seen in the channel steps, and the index rate was increased and decreased, eddies slowly grew and moved and suddenly increased eddies, and moved towards the outlet of the channel.

Keywords: Non-Newtonian Fluid, Rectangular Channel, Double Step Geometry, CFD, Finite Element Method, Reynolds Number, Power Law Model, Flow Recirculation

Introduction

The research of stirrer fixed in the downstream of the body are discovered in the science of the hydraulic engineering particularly, stepped spill ways singly and doubled and beyond stirrer fixed in the downstream of the channel. The energy generations in the area of the hydraulic engineering are carried out with the help of the stirrers, which raise the velocity of the water and form the hydraulic jumps in the river and canals (Amin, 2025). The water is mainly transported to dam in the downstream by the use of mostly the spillways and in the transferring of the water to the dams, generates the high-level energy (kinetic). The energy will also destroy the spillways; in case the energy is unable to reduce after any predetermined discharge flow rates. Using the spillways, the different flow patterns are noted such as nappe, conversion and may be skimming fluid flows (Kisi, et al., 2008; Khosrow, et al., 2016). Following, reviewed the massive literature on the liquid and wind flows stirred in stirrer attached in the downstream and upstream of the bodies of the spillways. Here the research problem is selected to study the liquid flows of the non-

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Newtonian fluids using downstream in the rectangular channel fitted with double stirrer. Different flow phenomena are also calculated here such as streamline flow patterns, vortex intensity and vortex measurements as length which can be either primary and secondary vortex length. The computations of the velocity in 2D by using momentum equations with continuity equation will use the finite element tool and the result will be compared with other predicted and experimental results provided in literature.

MOTIVATION

The CFD packages are strongly helpful to control the heavy number of iterations on the numerical solutions of the mathematical complex equations especially, partial differential equations. The CFD provides the quick, accurate and converge result of complex mathematical problems and provides the most accurate mesh structure of the simple and complex geometries of the solution of PDE's. The PDE's are largely applicable in the transport of fluid flow phenomena in the various fields of hydraulic engineering to design the model of spillways flows and stepped weirs problems (Imtiaz, et al., 2025; Amin, et al., 2024; Imtiaz, Malik & Khan, 2024). Stepped weirs are found in the water structures through the dams and river training works and etc. also spillways phenomena are used in the environment sciences and hydraulic sciences for the water transport phenomena to create the kinetic energy and observed hydraulic jumps in the river dynamics.

OBJECTIVES

1. To get the knowledge of the CFD studies with the treatment of the mathematical equations and understand the research problem in the field of CFD.
2. To validate the CFD solution of the mathematical equations especially Navier – Stokes equations with findings the literature about the research problem.
3. To elaborate the further approximate solution obtained through CFD studies and developed the flow phenomena. Also, the flow phenomena will provide the benefits for the improvement of mathematics scholar and also beneficial for the hydraulic and environment sciences

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

LITERATURE REVIEW

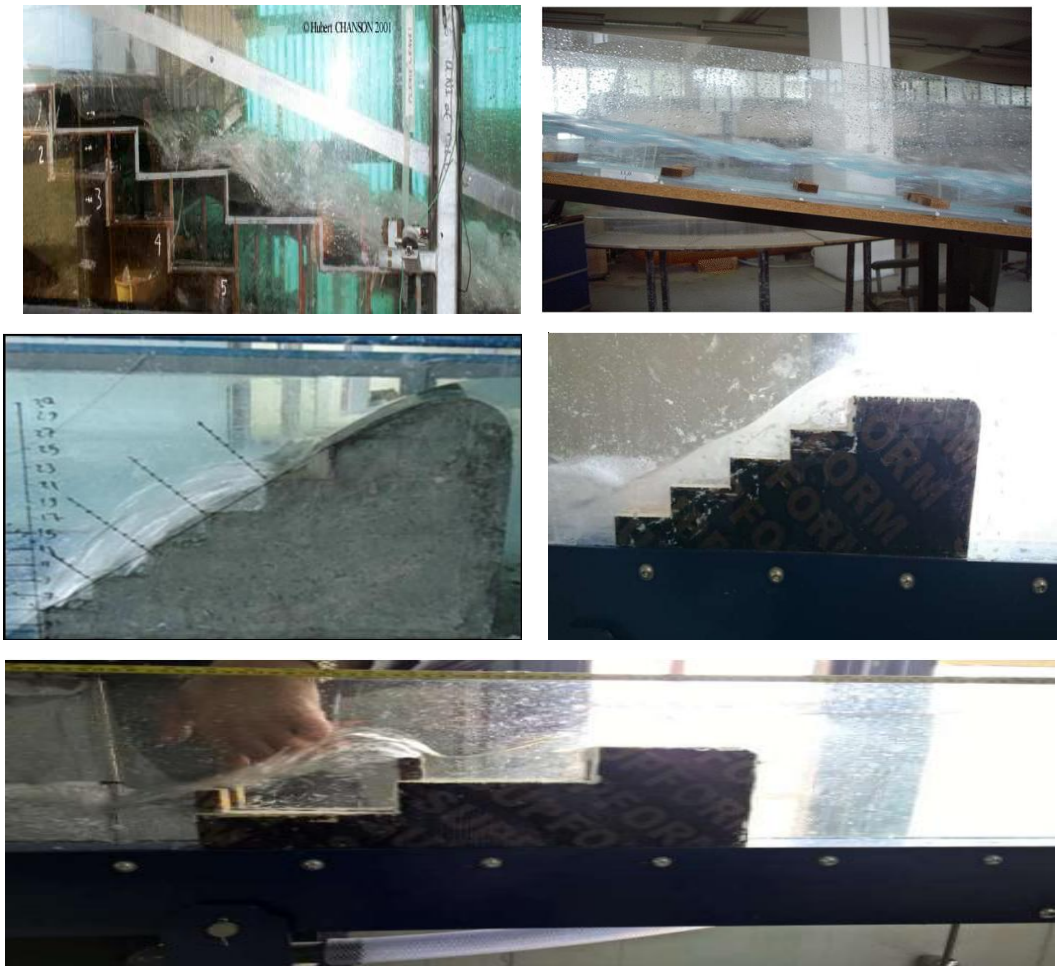


Figure – 2.1: water flows through spillways.

Introduction

Water flow around structures like steps, baffles and barriers is an issue in many areas of engineering, especially hydraulic engineering to design canal and river flows. The steps existing have a great impact upon the flow behavior producing hydraulic jumps and increasing the energy dissipation. They are applicable not only to open-channel hydraulics but also to biomechanics, as obstructions in arteries (e.g. stenosis) influence the dynamics of blood flow and cause changes in the stresses. A considerable amount of research has been done on single and multiple step configurations in channels and ducts, and the implications on the flow structure, turbulence and energy dissipation.

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Literature Background

Hassan et al. (2014) examined various step geometries in stepped spillway and discovered that non-horizontal steps (inclined and combined steps) dissipate 1.6 to 18.6 more energy than horizontal steps, combined steps being the most effective. In ANSYS, Rad and Teimouri (2010) applied the finite volume technique to determine the energy dissipation in stepped channels. They found that the greater the number of steps and the steeper spillway slope the greater the energy dissipation whereas the higher the discharge the lesser the energy dissipation. Shawkyward et al. (2018) used the RANS model with FLOW-3D and found out that the step-to-channel height ratio is the most beneficial at 0.25, which gives the maximum energy dissipation. Afshar and Hoseini (2013) performed numerical and experimental investigation of broad-crested weirs in RNG, standard and LES turbulence models. They established that lower water surface profiles are more predictable by the use of the RNG model.

Lucio et al. (2015) studied the flow of skimming at stepped spillways with FLOW-3D with RNG and LES. They recorded minor differences between the numerical and experimental values and this proved the applicability of CFD methods (Amin, Said & Butt, 2025; Shah, Amin & Khan, 2025; Dash, et al., 2025). Haque et al. (2007) investigated flow reattachment in 3D and 2D backward facing step channels (to different turbulence models). They noted that there were some disparities on the reattachment length because of the variation in mass flow and advanced models like DES and RANS-LES were suggested to provide accurate vortex prediction.

Salmasi et al. (2010) experimentally observed the energy dissipation in gabion-step channels with different porosity and slopes. They attained a prediction accuracy of 85% with respect to the energy dissipation using a decision tree method. Khalaf et al. (2014) carried out experiments on semi-circular crest spillway and found flow regimes, such as jet and skimming flows. They discovered that energy dissipation also increases in combination with Froude number and water depth.

Subsequent work on turbulence models revealed that RANS-based models can be restricted to the prediction of unsteady flows, and advanced models like BSL, v 2 -f, SST and SAS can make predictions of vortices and velocity fields, with SAS

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

giving a more accurate representation of vortex structures. Salmasi et al. (2008) conducted ANSYS-Fluent modelling of stepped spillways, and found that there was good consistency with experimental values (within 3%), which confirms the usefulness of CFD in predicting shear stress, pressure, and energy dissipations.

ANALYSIS OF THE FLUID FLOW MATHEMATICAL EQUATIONS WITH THE NUMERICAL COMPUTATIONS

INTRODUCTION

This chapter studied the understanding of mathematical theory of the partial differential equations and focused complete boundary value problem. The boundary value problem is known as non –linear problems that not easily compute their analytical solution for the reason that couple of unknown parameters and hybrid system of equations. Conversely, preferred the most appropriate numerical approaches and also familiar that approximate solution occurred some limited error because of the large computations in the iterative procedure (Amin, 2025; Soma, et al., 2025; Amin, Daudpota & Khan, 2025). Therefore, for the controlling the large computations and decreased error, preferred here the computational based plate form called as COMSOL Multi Physics. Here employed fluid motion equation related modelling via COMSOL that adopt mathematical related optimization and employed mathematical iterative treatment to compute the maximum level of accuracy through the procedure adopted by Computational tool. This technique FEM is used for the default setting or fitting the personnel setting in the mathematical treatment and maximize convergence for different type of problems applicable in vast number of fields especially for the fluid and heat transported in the mechanical problems termed as Navier Stokes equations. Furthermore, the computational tool is also visualizing the differential and non-differential optimal problems that quickly and easily adopted the built in numerical technique Newton's method and then preferred the required setting to plot the physical unknowns selected from the flow chart of the COMSOL Multiphysics.

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

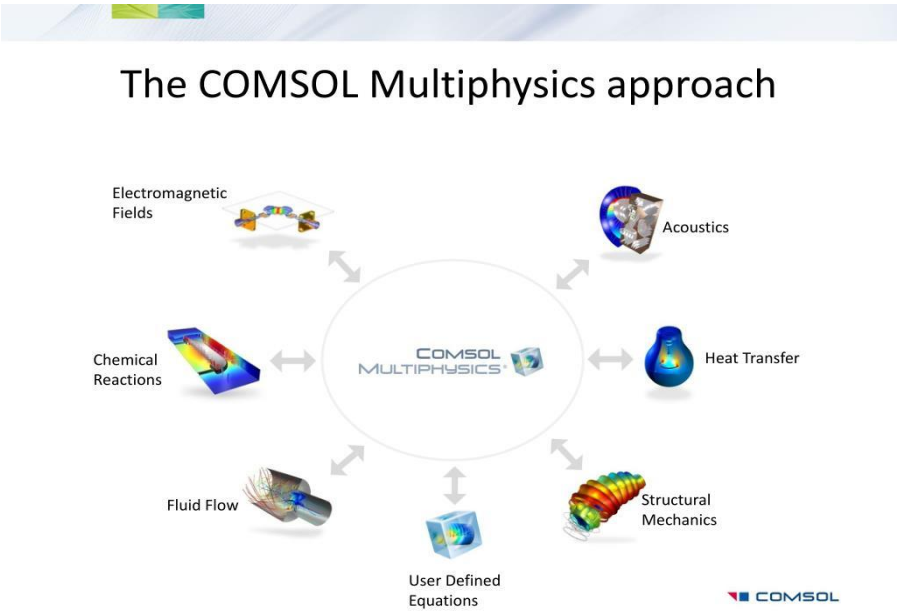


Figure – 3.1: Flow chart of the COMSOL Multiphysics

This tool has vast material and number of equations fixed in the default settings in the menu bar and equations are separated in the fluid flow menu bat that divided into the various number of laws such as Power law model and bird chereu model and distributed towards number of numerical schemes such as PARADISO, PISO and etc.

MATHEMATICAL MODEL IN THE PDE'S

The mathematical model is focused on the partial differential equation in mathematics and same known as fluid motion equations and also type of Navier – Stokes equations. Here studied the incompressible fluids equation knowns as momentum equation that is define from the Newton second law of motion and needs the continuum approach for the numerical modelling so imposed the continuity equation.

EQUATION OF CONTINUITY

This equation is well known model to describe the continuum attitude towards the fluid moved in a closed body figures such as channels, ducts and many more geometries. This model is also called as first order partial differential equations or known as divergence free vector. It is focused on the liquid flow moved only and not for gases that are stated as;

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

For Incompressible fluids.

$$\nabla . \boldsymbol{\eta} = 0 \qquad (3\text{–}1)$$

Though, η designated as solenoidal vector field in two – dimension as η = (u, v)

EQUATION FOR MOMENTUM TRANSPORT

This equation is designated as also conservation law of momentum that is employed in the theory of fluid motion from one place to another place through the closed domain in the two – dimension Cartesian coordinates form. In this model of equation, wide variety of terms involved that focused on the separate flow phenomena such as time independent for the velocity field. It shows the how much fluid flow moved due to time changes but here described the time fixed as zero because of steady state position of the fluid transports. Similarity, other various terms involved as convection, diffusion and pressure gradient with external forces. This model is also derived from the most important law of physics that are very familiar in the theory of physical concepts of the mass and fluid flows as Newton's second law of motion as force is directly proportional to the acceleration of the body. The vector form of this model in the form of equation stated as;

$$\underbrace{\frac{\partial \boldsymbol{\eta}}{\partial t} + u(\nabla\, \boldsymbol{\eta})}_{\text{These terms are called as time derivative terms or material derivatives}} = \underbrace{\nabla P}_{\text{Pressure gradient}} + \underbrace{\frac{1}{\mathrm{Re}}(\nabla^2 \boldsymbol{\eta})}_{\text{Diffusion terms or acceleration of the body}} + \underbrace{g}_{\text{External forces}} \qquad (3\text{–}2)$$

The equation (3 – 2) is highly important in the theory of fluid motion and it must be limited on the choice of the research problem. The choice of the problem is two dimension and for used here the liquid flow transported whereas density of the fluids remained constant. It described as

$$\frac{\partial \eta_1^*}{\partial t} + \left(\eta_1^* \frac{\partial \eta_1^*}{\partial x^*} + \eta_2^* \frac{\partial \eta_1^*}{\partial y^*} \right) = -\frac{\partial P^*}{\partial x^*} + \frac{1}{\mathrm{Re}} \left[\frac{\partial}{\partial x^*} \left(\frac{\partial \eta_1^*}{\partial x^*} \right) + \frac{\partial}{\partial y^*} \left(\frac{\partial \eta_1^*}{\partial y^*} \right) \right] \qquad (3\text{–}3a)$$

$$\frac{\partial \eta_2^*}{\partial t} + \left(\eta_1^* \frac{\partial \eta_2^*}{\partial x^*} + \eta_2^* \frac{\partial \eta_2^*}{\partial y^*} \right) = -\frac{\partial P^*}{\partial y^*} + \frac{1}{\mathrm{Re}} \left[\frac{\partial}{\partial x^*} \left(\frac{\partial \eta_2^*}{\partial x^*} \right) + \frac{\partial}{\partial y^*} \left(\frac{\partial \eta_2^*}{\partial y^*} \right) \right] \qquad (3\text{–}3b)$$

Though, re-illustrated as the value of the fluid resistance or fluid inertia that

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189
http://amresearchreview.com/index.php/Journal/about

described as

$$\mathrm{Re} = \frac{\rho \times u_{avg} \times L}{\mu}$$

Though, all fluid parameters are listed in this thesis in the list of figures.

INITIAL AND BOUNDARY CONDITIONS

The conditions are most appropriate for the complete solution of the velocity field and pressure because of the needs in the numerical solution. The Dirchlet boundary conditions are preferred for the stationary walls in the geometry and known as no – slip boundary conditions for velocity field. The choice of the problem is 2^{nd} order that classified as parabolic so fixed constant parabolic velocity profile at the inlet of the geometry with assume maximum pressure. On the same, fixed zero pressure at the outlet of the geometry whereas converge the solution of velocity field.

$$u(y) = 6 \times U_m \times \frac{y}{L}\left(1 - \frac{y}{L}\right), \mathrm{P} = \mathrm{P}_{\max} \qquad (3\text{–}4)$$

PREDICTABLE COMPUTATIONS VIA COMSOL MULTIPHYSICS PACKAGE

BACKGROUND

The numerical techniques are highly necessary to compute the complex and higher order solution with couple of unknowns. The vast variety of methods employed in the applied sciences and based upon the number of iterative procedures in the numerical tools. For the quick convergence and most accurate solution, necessary to design the algorithm for the computational language or needs the computational codes defaulted by the programming or most appropriate computational tool for the approximate solution and moved towards the fluid dynamical problems. Most appropriate numerical method as finite element tool preferred here through the CFD tool known as COMSOL Multiphysics. This tool provides the plate form where pre-process the problem, chosen the solver settings and further applied in the fluid motion problem to visualise the flow structure through the bodies of interest.

FINITE ELEMENT METHOD

This technique is used for the splitting the whole domain into the further piece wise discretization of each unknown such as velocity field, pressure and further vorticity.

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

This tool is used through the selection of triangular elements by the developing the mesh phenomena. This discretization tool originated for the computations of real-life problems relate to the solid mechanics but with the increasing time, FEM found vast application mostly in every science not limited but highly useful in the engineering and industrial simple and complex problems and it combined into the single field as CFD. Also known as nowadays a universal computational tool to compute vast complex equations numerically. The FEM have different dimension and various approaches are used such as Galerkin pattern in the mesh and also Petrov – Galerkin pattern. The some shapes of the FEM are described as;

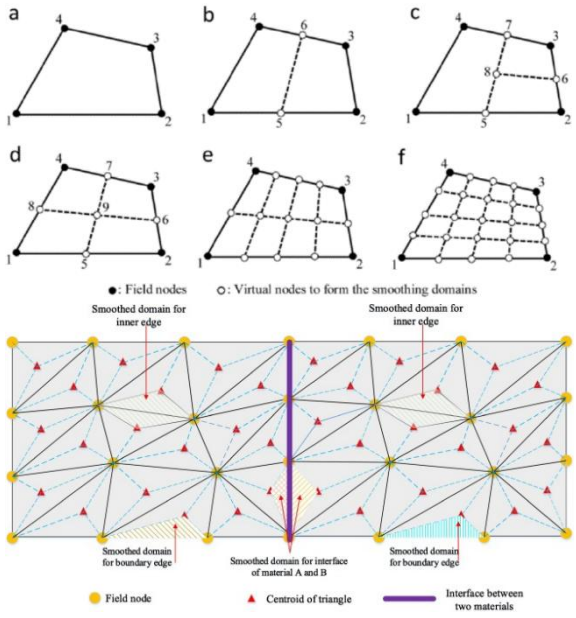
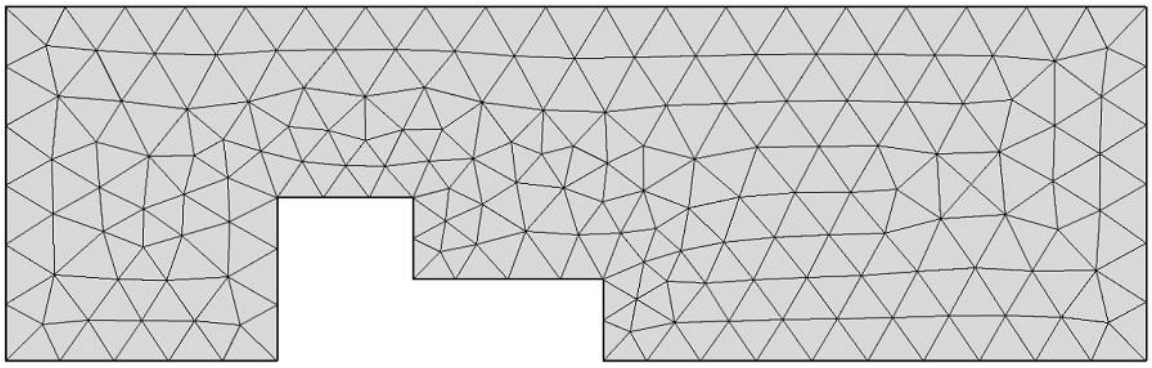


Figure – 3.2: Shapes of the finite element method



Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

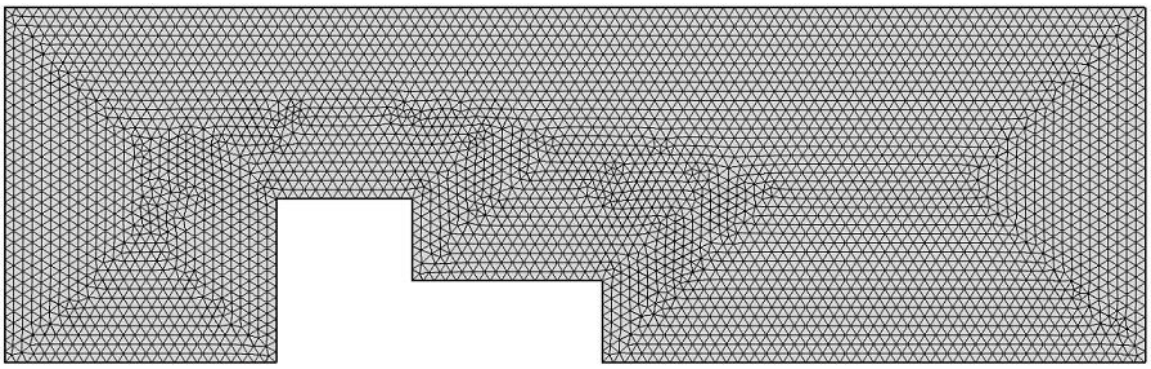


Figure – 3.3: combined structured of FEM styles.

In the time of 1950's, variety of application extended and vast applicable by through the FEM and designed by the describing a set of spatial piece wise domain known as finite element. That make up the whole geometry of choice. The behavior of individual element is examined relates to the limited set of modes known as shape function. Mostly, shape functions are taken as low order polynomials and preferred here required degree of freedom through the choice of degree of freedom via defined the displacements either rotations at particular points known as nodes and collection of all nodes in Cartesian form is called as mesh or grid. This procedure is highly complex and too difficult so that choice is to prefer the CFD plate form or it is called COMSOL Multiphysics and many more available in literature like ANSYS, FlexPDE and Ideas and etc.

NUMERICAL PROCEDURE ADOPTED THROUGH COMSOL BY THE SELECTION OF LSGFEM

The numerical computations are carried out here by selection of finite element discretization with couple of Galerkin structured shape of elements through the mesh and also implemented the least square residuals. This computational procedure is only applicable for the linear system of equations. The model of equation is higher order so not possible to solve directly through this technique. Alternatively, adopted the mathematical way out to diminish the higher order model of equations towards the single order system called as vorticity. Such as;

$$\nabla . \eta = 0 \qquad (3-5)$$

$$\frac{\partial \eta}{\partial t} + u(\nabla \eta) = \nabla P + \frac{1}{\mathrm{Re}}(\nabla^2 \eta) + g \qquad (3\text{–}6)$$

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Here needs the split the above system $(3-6)$ towards first order so substitute the vorticity defined as under;

$$\boldsymbol{w} = \nabla \times \eta \qquad (3-7)$$

Substitute the assumption as $(3-7)$ towards the $(3-6)$ and $(3-5)$ will becomes as;

$$\nabla . \boldsymbol{\eta} = 0 \qquad (3-8a)$$

$$\boldsymbol{\eta}.\nabla \boldsymbol{\eta} + \frac{1}{\mathrm{Re}} \nabla \times \boldsymbol{w} + \nabla p = 0 \qquad (3-8b)$$

$$w - \nabla \times \boldsymbol{\eta} = 0 \qquad (3-8c)$$

Now needs to describe the system on the requirement of the domain;

$$\frac{\partial \eta_1}{\partial x} + \frac{\partial \eta_2}{\partial y} = 0 \qquad (3-9a)$$

$$\eta_1 \frac{\partial \eta_1}{\partial x} + \eta_2 \frac{\partial \eta_2}{\partial y} + \frac{\partial P}{\partial x} + \frac{1}{\mathrm{Re}} \frac{\partial w}{\partial y} = 0 \qquad (3-9b)$$

$$\eta_1 \frac{\partial \eta_2}{\partial x} + \eta_2 \frac{\partial \eta_2}{\partial y} + \frac{\partial P}{\partial y} - \frac{1}{\mathrm{Re}} \frac{\partial \eta_2}{\partial x} = 0 \qquad (3-$$

$$9c)\, w + \frac{\partial \eta_1}{\partial y} - \frac{\partial \eta_2}{\partial x} = 0 \qquad (3-9d)$$

The system $(3-9)$ is system of couple PDE's so needs here to discretize each components of fluid features. For the discretize required to convert the system into the matrix form that is stated as;

$$C_1 = \begin{pmatrix} 1 & 0 & 0 & 0 \\ \eta_1 & 0 & 1 & 0 \\ 0 & \eta_1 & 0 & -\frac{1}{\mathrm{Re}} \\ 0 & -1 & 0 & 0 \end{pmatrix}, \qquad C_2 = \begin{pmatrix} 0 & 1 & 0 & 0 \\ \eta_2 & 0 & 0 & \frac{1}{\mathrm{Re}} \\ 0 & \eta_2 & 1 & 0 \\ 1 & 0 & 0 & 0 \end{pmatrix}$$

$$\mathrm{C} = \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix}, \qquad f = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}, \qquad \boldsymbol{\eta} = \begin{pmatrix} \eta_1 \\ \eta_2 \\ p \\ w \end{pmatrix}$$

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Here needs to find stiffness matrix through the process of Galerkin FEM tool with hybrid least square described as;

$$L\Psi_j = \Psi_{j,x} C_1 + \Psi_{j,y} C_2 + \psi_j \Psi \qquad (3-10a)$$

$$K_e = \int_{\Pi_d} <L\Psi_1, L\Psi_2, L\Psi_3, ..., L\Psi_{Ne}>^T < L\Psi_1, L\Psi_2, L\Psi_3, ..., L\Psi_{Ne} > d\Pi \quad (3-10b)$$

Though, the process adopted by (3 – 10) with discretization procedure to develop the complete stiffness matrix as K. After that, employed matrix system as KU=f to transport the system of equations into the discrete variable equations for separately of each component as velocity field, pressure and vorticity. In tool COMSOL, defaulted by the iterative approach Newton's method for further solutions on required level of accuracy.

NEWTON RAPHSON METHOD

This method is most vital in the theory of the numerical analysis and highly used in the solution of the non-linear discreet variable equations either for single variable or more than single variables. The convergence criteria is double other than direct and other indirect methods. This method is initially designed by the most renowned persons known as Burden fairs [2005]. The general format of the newton's method for the single variable is stated as;

$$\eta_{i+1} = \eta_i - \frac{f(\eta_i)}{f'(\eta_i)} \qquad i = 0, 1, 2, 3, ... \qquad (3-11)$$

The above form (3 – 11) may be extended for the more than single variables simultaneously. That needs the transform into the determinant in the real analysis concept through the Jacobean determinants as follows;

$$\eta_{i+1} = \eta_i - J(\eta_i)^{-1} f(\eta_i) \qquad i = 0, 1, 2, 3, ... \qquad (3-12)$$

Though, $J(\eta_i)^{-1}$ is stated as inverse matrix of all differential coefficients in two – dimensions such as x and y and also known as Jacobean matrix as prescribed below;

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

$$J(\eta)^{-1} = \begin{bmatrix} \frac{\partial h_1}{\partial x_1}(x) & \frac{\partial h_1}{\partial x_2}(x) \ldots\ldots \frac{\partial h_1}{\partial x_n}(x) \\ \frac{\partial h_2}{\partial x_1}(x) & \frac{\partial h_2}{\partial x_2}(x) \ldots\ldots \frac{\partial h_2}{\partial x_n}(x) \\ . & . \quad \ldots\ldots \quad . \\ . & . \quad \ldots\ldots \quad . \\ . & . \quad \ldots\ldots \quad . \\ \frac{\partial h_n}{\partial x_1}(x) & \frac{\partial h_n}{\partial x_2}(x) \ldots\ldots \frac{\partial h_n}{\partial x_n}(x) \end{bmatrix}$$

The system (3 – 12) needs the guessing on any assumptions for the iterative solutions of each component so chosen here (0, 0, 0, 0) because of the quick convergence and saved the labor work of computations and fixed the termination level as 10-05.

Effect of Fixed Stairs in a Rectangular Channel on Non-Newtonian Fluid Flow

Introduction

The study of stepped geometries in rectangular channels is important in hydraulic engineering, particularly for irrigation systems, spillways, and flow energy dissipation. With the advancement of computational power, Computational Fluid Dynamics (CFD) has become a reliable alternative to experimental methods, reducing cost, time, and labor while providing detailed flow visualization. CFD is also widely applied in biomedical fields, such as modeling blood flow in arteries and analyzing diseases like stenosis (Al-Husseini, 2015; Morovati et al., 2016). Al-Husseini (2015) experimentally investigated flow behavior over sloped and stepped spillways to analyze energy dissipation. Various flow regimes, including nappe and skimming flows, were observed. The study concluded that stepped spillways significantly enhance energy dissipation compared to flat slopes, particularly when the number of steps is optimized.

Similarly, Morovati et al. (2016) used CFD (Flow-3D with AutoCAD) to model turbulent flow over spillways using the Reynolds-Navier-Stokes equations with continuity. Their findings showed that increasing the number of steps reduces flow velocity and enhances energy dissipation, while fewer steps result in higher kinetic energy. Vortex formation was also observed at different step locations. The present study focuses on analyzing non-Newtonian fluid flow in a rectangular

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189
http://amresearchreview.com/index.php/Journal/about

channel with fixed double steps (stirrers/spillways). The governing equations are based on momentum conservation and continuity for incompressible flow. A numerical approach using COMSOL Multiphysics is employed, utilizing Galerkin discretization with least-squares stabilization. Key flow characteristics, such as eddy length, eddy intensity, and resistance effects, are evaluated along with statistical and empirical flow measures.

Problem Definition and Boundary Conditions

The computational domain consists of a rectangular channel with two fixed steps of varying lengths. Appropriate boundary conditions are applied at the inlet, outlet, and walls. A parabolic velocity profile is imposed at the inlet to represent fully developed flow, ensuring realistic simulation of fluid behavior within the channel.

$$u = f(y) = 10 \times U_{max} \times y(1 - y)$$

For the Mesh structure, the detail of the elements at boundaries, inner and edge elements and vertex elements are given in a table:

Description	Value
Minimum element quality	0.7474
Average element quality	0.9897
Triangular elements	4862
Edge elements	262
Vertex elements	10

Description	Value
Maximum element size	0.0065
Minimum element size	1.3E-5
Curvature factor	0.2
Predefined size	Extremely fine

OUTCOMES OF AN APPROXIMATE RESULTS

The outcomes of the fluid dynamics are discussed here and based upon the completely approximate results of the velocity field and investigated the change in the rate of viscosity moved towards the rectangular channel with imposed the double step obstacles. The power law model is adopted for the achieving the rate of

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

viscosity of the different type of fluids called as non-Newtonian fluids such as honey, blood, greywater and many more. The outcomes relate on the various value of the fluid inertia with change in the value of the viscosity (0.90 and 1.1) that determined behave of thinning and thickening fluids. Picture – A .1(a) illustrated the graphical structure of the complete domain, where illustrated the no – slip boundary conditions on the solid walls because of the no – activity of the fluids at the boundaries. Also, imposed the initial condition as parabolic velocity and no pressure imposed at the outlet of the domain because of the free fluids after the outlet. Picture – A.2 illustrated the grid style computed through FEM procedure and examined the three-grid style as weaker (lowest accuracy), finer (more accurate) and finally preferred the extremely fine mesh (most accurate) of the rectangular channel imposed double step obstacles. Similarly, further outcomes are distributed towards the various values of the viscosity of the fluids as change in the fluid inertia and change in the power law index rate described below;

Influence of fluid inertia on the flow features with change in the rate of viscosity (index rate 0.90)

Picture – A.3 (a and b) illustrated the streamline laminar flow patterns of the shear thickening fluids at power law index rate streamline flow patterns of the velocity at different values of the fluid inertia. Picture – A.3 (a) illustrated the streamlines of the velocity at lowest value of the fluid inertia (01 to 600) with fixed index rate (0.90) and is not appeared the vortices at the turning of the pace of the obstacles imposed in the rectangular channel because of the lowest value of the fluid inertia (01). Further, the very too tinny in size vortices appeared in the pace of the obstacles at the value of the fluid inertia (=100). Slowly, the vortices in sizes are appeared with growing structure at the turning of the pace of the obstacles because of the increase the value of the fluid inertia (=300) and lastly the primary vortex appeared in the turning of the 1st pace of the channel and filled the whole region of the pave at the value of the fluid inertia. But the secondary vortex is increased very slowly in the turning of the pace in the channel. Picture–A.3 (b) illustrated more streamline of the velocity that showed the flow phenomena through the appearance of the vortices and streamlines of the velocity. After the value of the fluid inertia (=300) the primary vortices are

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

merged into the secondary vortex appeared in the 2nd turning of the pace of the channel and slowly increased and moved towards outlet of the channel at the value of the fluid inertia (=700). Due to lowest thin viscosity of the fluids, the flow is quickly moved towards outflow of the channel and quickly secondary vortex filled the whole region of the turning of the pace in the channel.

Influence of Fluid Inertia on Flow Features (Power Law Index = 1.1)

Figures A.4 (a, b) illustrates streamline patterns for shear-thickening fluid (power law index = 1.1) under varying fluid inertia. At low inertia the formation of a vortex is not detected at either of the turning sections of the channel. And as the inertia is further increased (Re = 100 a small vortex starts developing at the turnover area. As the inertia increases, the primary vortex slowly extends in the first turning area with the secondary vortex becoming more noticeable in the second turning because there is more space to move through. The main vortex takes up the entire first turning region at increased inertia (Re = 600) but the secondary vortex still increases in the downstream part.

And at very high inertia (to Re = 1300) the primary vortex is entirely filled in the turning area and finally converges at the secondary vortex. The secondary vortex, then, grows gradually and moves towards the outlet; but does not fill up the whole of the downstream area. These findings suggest that augmenting the inertia of fluids increases vortex many folds in shear-thickening fluids.

Effect of Viscosity rate (n = 0.9, 1.1) on Flow Characteristics at Constant fluid Inertia

This section looks at how the viscosity variation (power law index 0.9 and 1.1) affects the flow behavior at constant Reynolds number. When the inertia is very low (Re = 1), there is no vortex formation in either fluid type, and the flow is smooth at both turning points.

Formation of vortex commences at high inertia (Re = 400). In case of shear-thinning fluid (n = 0.9), the main vortex initially develops slowly, but then rapidly in the first turning region and generally joins the secondary vortex. Conversely, in the case of a shear-thickening fluid (n = 1.1), the vortex growth is relatively slower. The larger the inertia (Re = 600) the more important the effect of viscosity becomes. The

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

main vortex rapidly fills in the first turning region and combines with the secondary one before the shear-thinning fluid. Secondly, the secondary vortex increases at a greater rate and travels towards the outlet and is bigger in shear-thinning behavior because it has less effective viscosity.

Velocity magnitude (Figures 4.6 a, b) and recirculation flow rate numerical analysis (Figure 4.6 a, b, n.d., p. 46) depict nonlinear enhancement trends whereby more recirculation is found in shear-thinning fluid (n = 0.9) than in shear-thickening fluid (n = 1.1). Figure 4.7 continues to affirm the idea that eddy structures enhance as the fluid inertia increases, particularly in shear-thinning fluids where the primary and secondary vortices grow swiftly. These observations conform to streamline configurations. The maximum and minimum values of recirculation rates and eddy development at various levels of inertia are summarized in table 4.8 and table 4.9. Moreover, least squares analysis is used to formulate empirical relationships that can be used to measure the behavior of flow through various fluids.

Power law index rate (n = 0.90)

$$X = -3.43429e-06\,\mathrm{Re} + 2.212e-03$$

Power law index rate (n = 1.1)

$$X = 2.973e-05\,\mathrm{Re} + 3.1897e-02$$

Though, re-illustrated the fluid inertia in the flow features.

SUMMARY

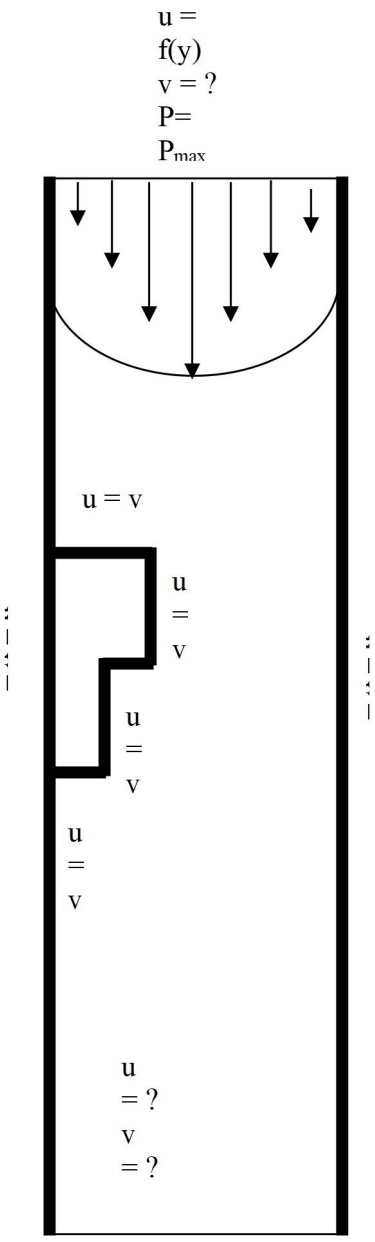
Newton's iterative tool is employed for the estimating the fluid parameters through the most well-known mathematical equations that are highly applicable in every field of the physics, engineering and also in medical sciences. The equations are called as momentum equation and continuity equations for the two – dimension and incompressible fluids with different type of fluids known as non – Newtonian physically. Numerical methodology is based upon the different approaches, initially, discretize the differential coefficient through Galekrin finite element process and obtained the stiffness matrix (K) and further applied the direct numerical solver PARADISO to convert the matrices into the non–linear equations and then adopted the Newton's approach for the final estimating the fluid parameters and this complete process is added with the choice of the CFD tool COMSOL MultiPhysics.

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

These fluid parameters (velocity and pressure) are used in the fluid flow phenomena to visualize the behave of various fluids through channel with number of stairs. Different flow phenomena as streamlines, eddies occurrence in the corner of the stairs and eddies intensity computed at various tests of the Reynolds number. The iterative results as streamline patterns are fully compared with the other flow patterns of the other scholars contributed in the CFD field

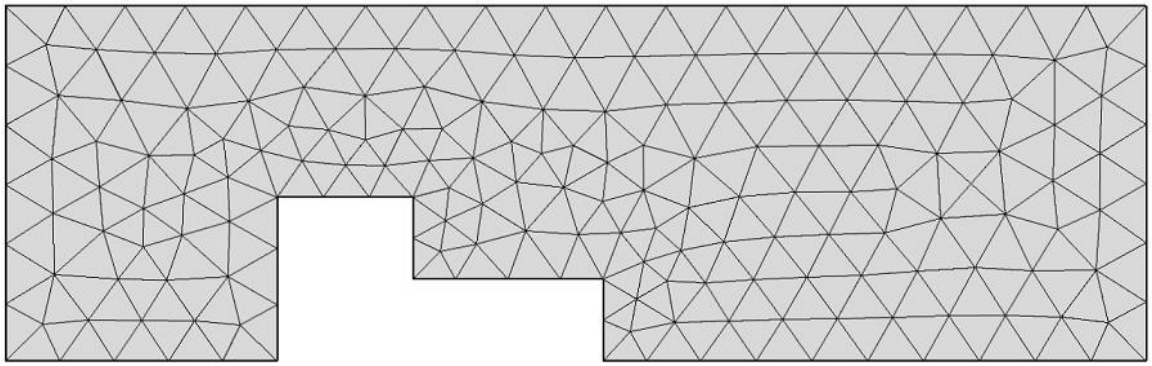


Picture –A.1: Schematic diagram of the two–dimension rectangular channel fixed double stairs

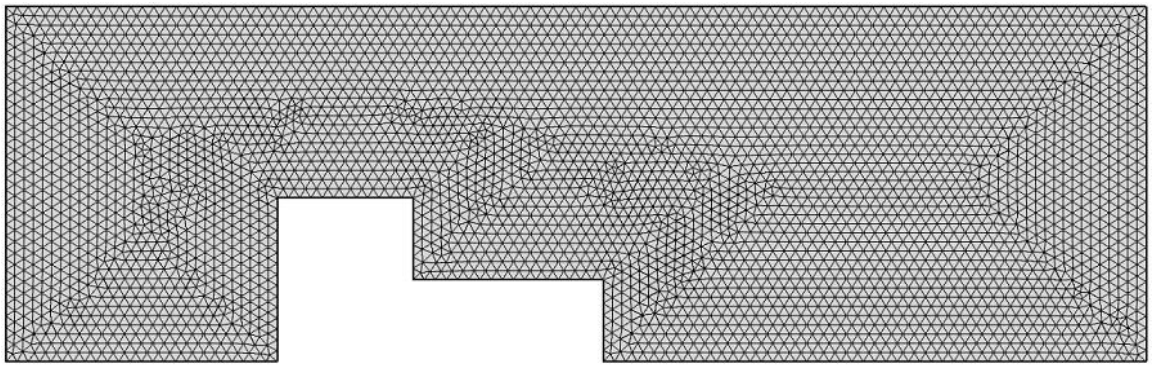
Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

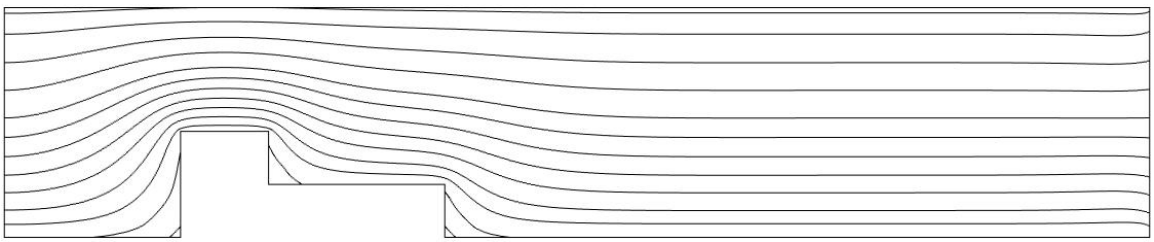


(a)

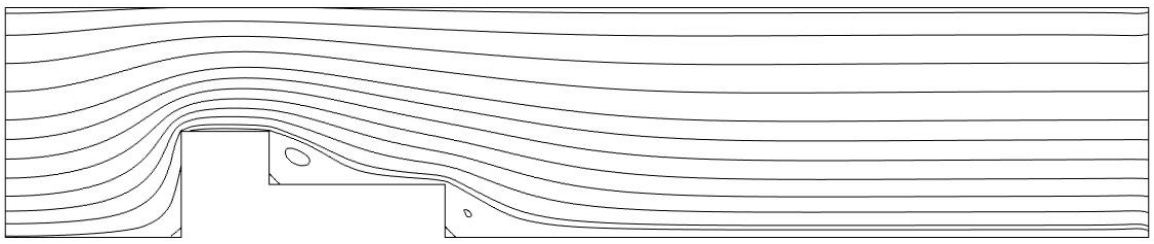


(b)

Picture –A.2: Finer and Extremely fine mesh of the two–dimension rectangular channel fixed double stairs.



Re = 01

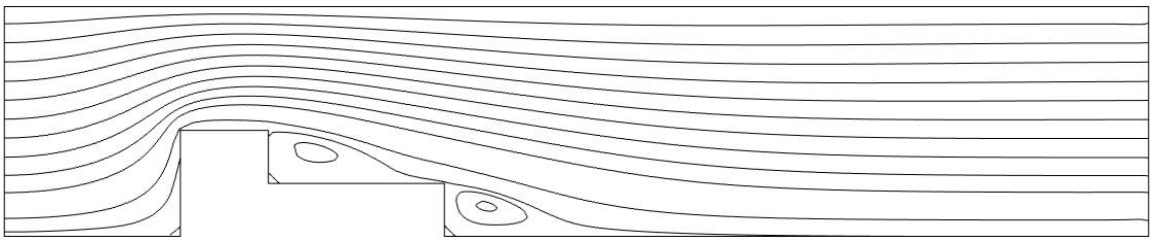


Re = 100

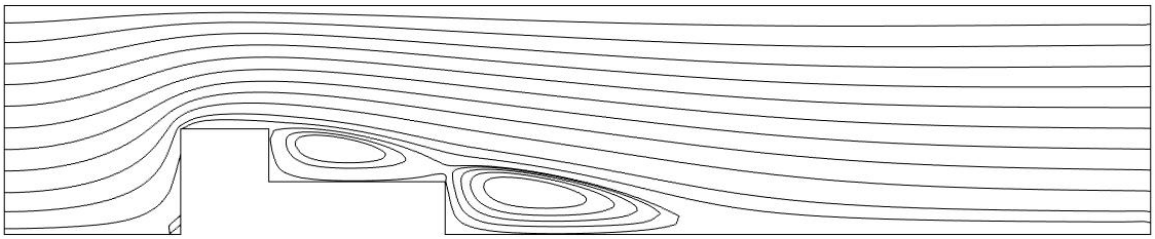
Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

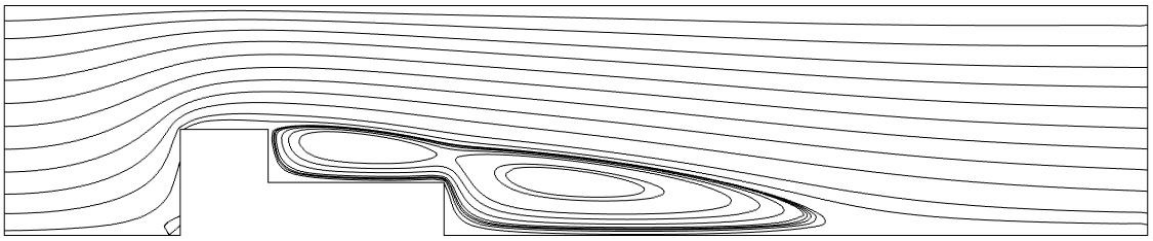


Re = 200

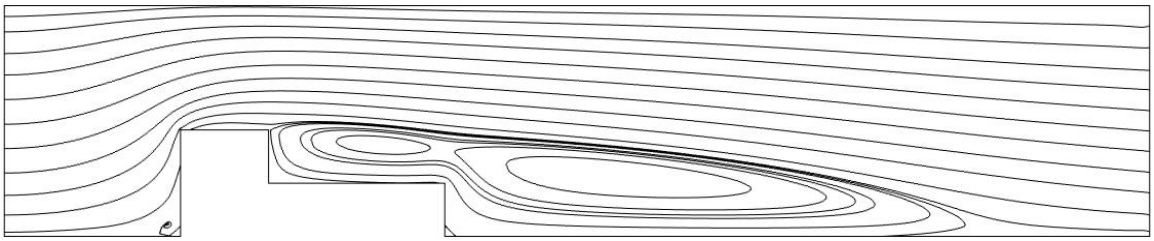


Re = 300

Picture –A.3(a): Streamline Contours of Velocity of the two–dimension rectangular channel fixed double stairs (1 ≤ Re ≤ 300) on fixed power law index rate (n) = 0.9



Re = 400

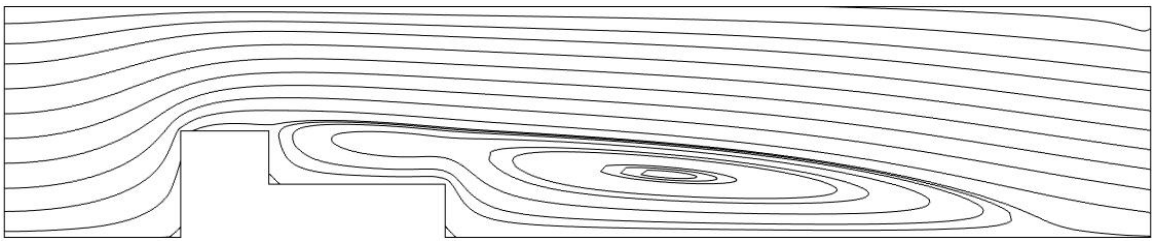


Re = 500

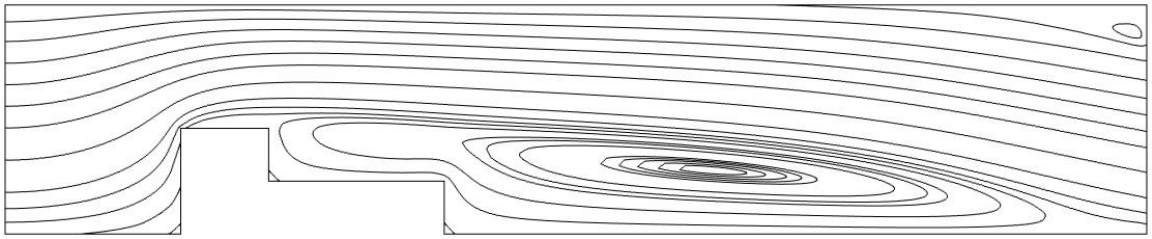
Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

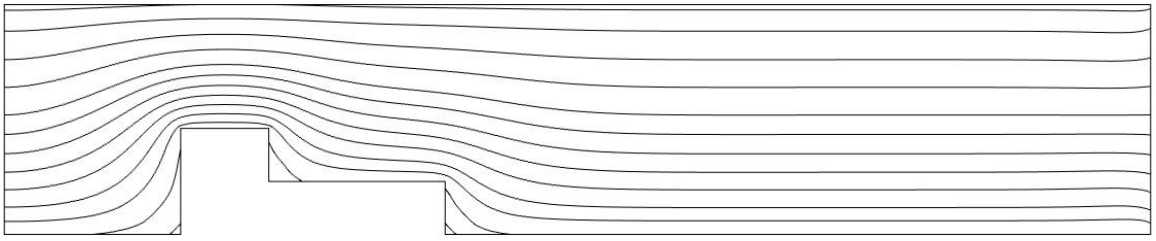


Re = 600

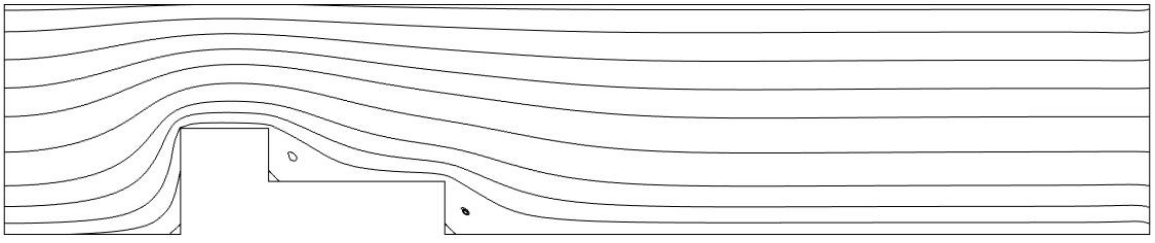


Re = 700

Picture –A.3(b): Streamline Contours of Velocity of the two–dimension rectangular channel fixed double stairs (400 ≤ Re ≤ 700) on fixed power law index rate (n) = 0.9



Re = 01

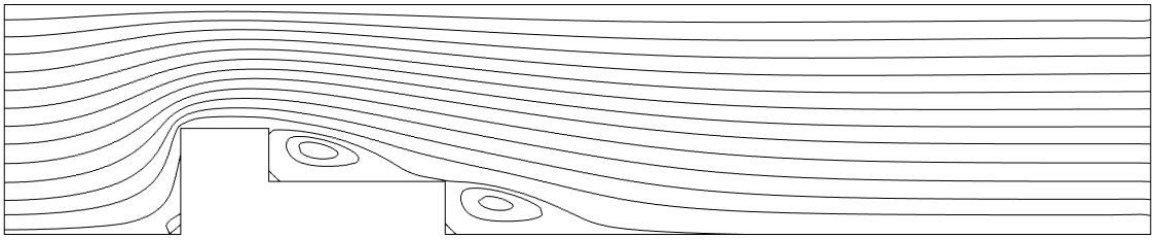


Re = 100

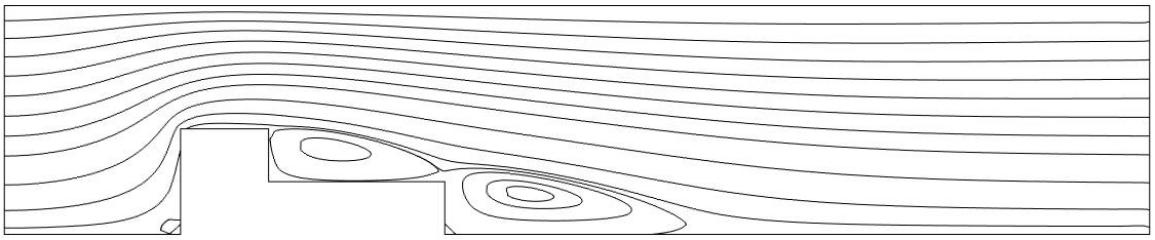
Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

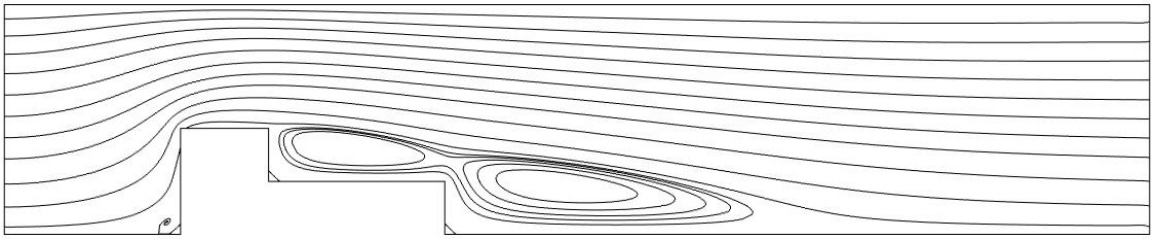


Re = 400

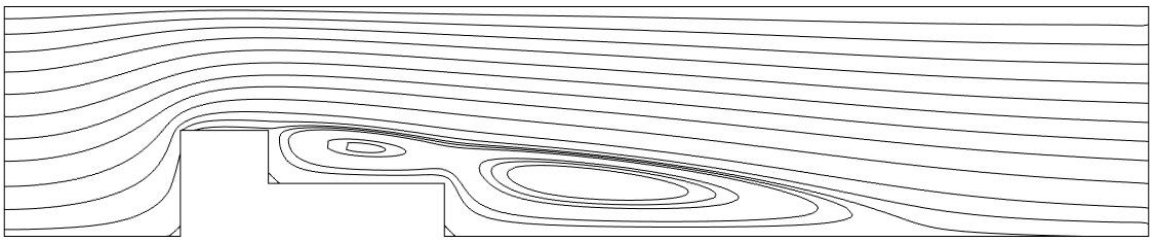


Re = 600

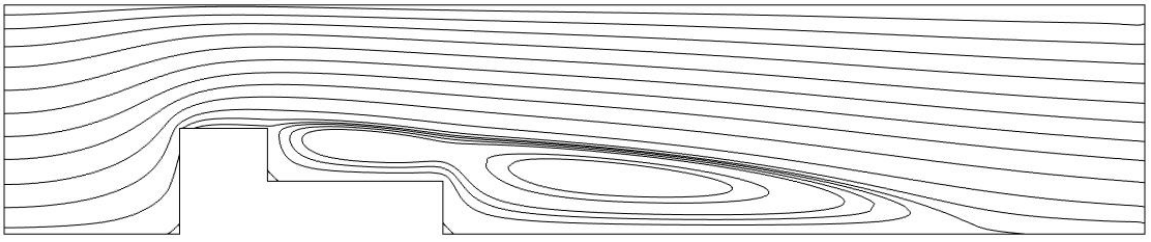
Picture –A.4(a): Streamline Contours of Velocity of the two–dimension rectangular channel fixed double stairs (01 ≤ Re ≤ 600) on fixed power law index rate (n) = 1.1



800



1000

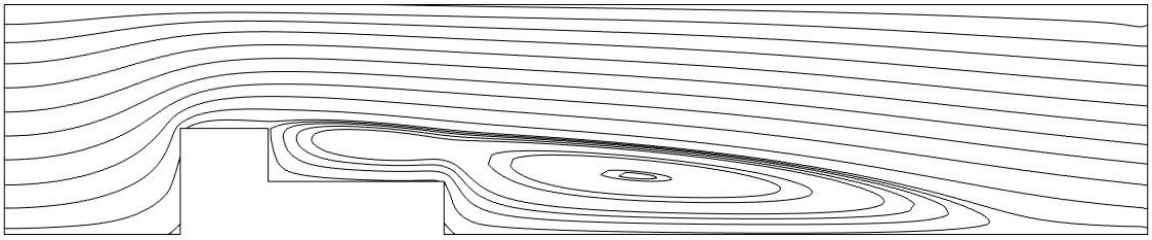


1150

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

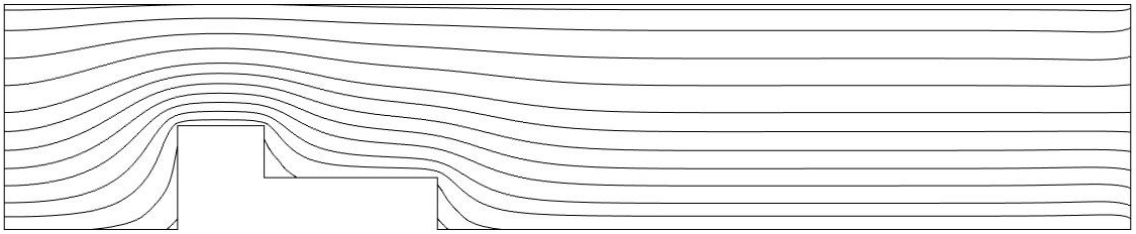
Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

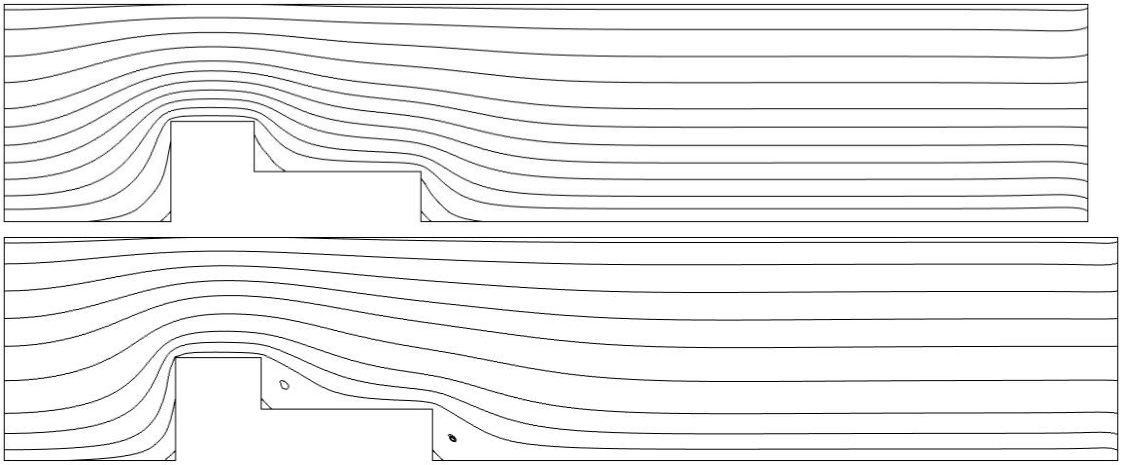


1300

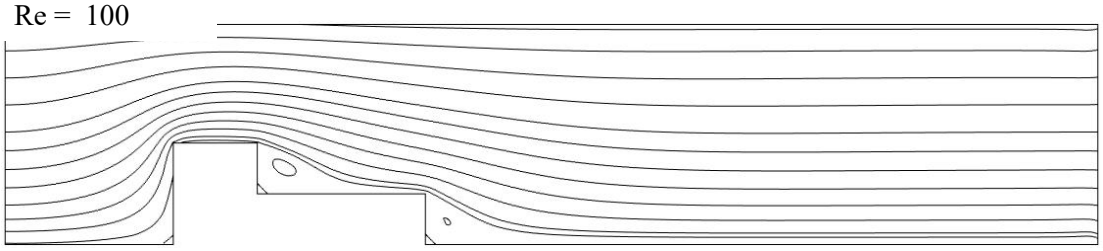
Picture –A.4(b): Streamline Contours of Velocity of the two–dimension rectangular channel fixed double stairs (800 ≤ Re ≤ 1300) on fixed power law index rate (n) = 1.1



Re = 01



Re = 100

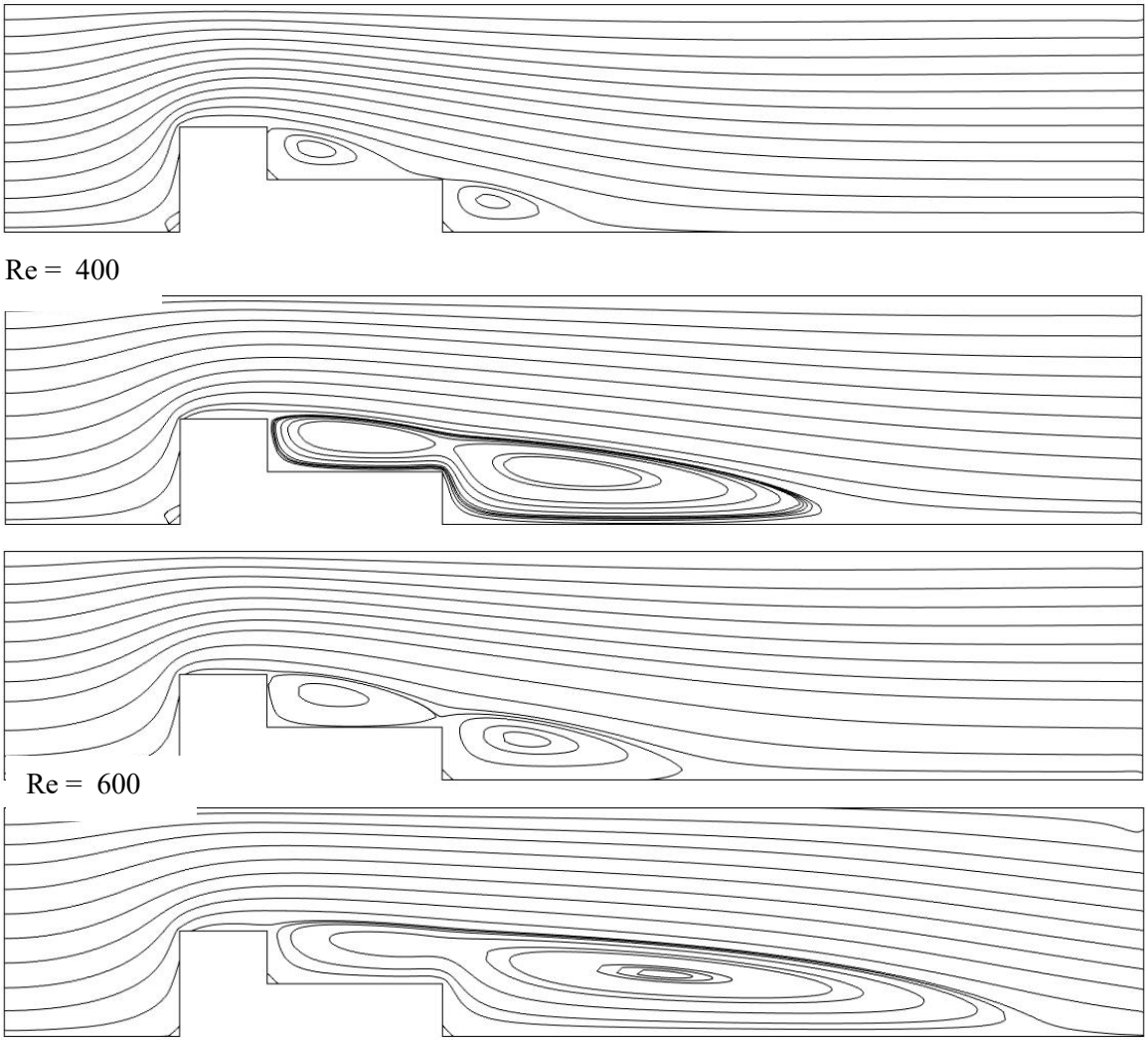


Picture –A.5(a): Streamline Contours of Velocity of the two–dimension rectangular Channel fixed double stairs on fixed Reynolds number with different power law Indices (n) = 0.9 and 1.1

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

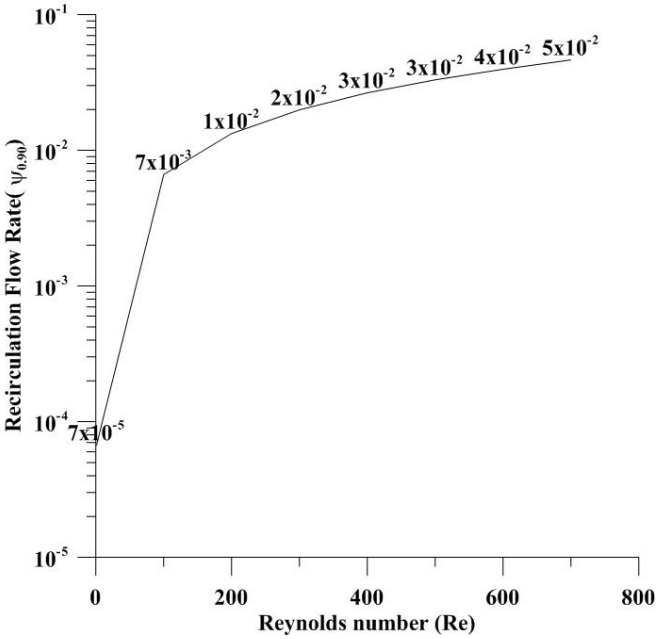


Picture –4.5(b): Streamline Contours of Velocity of the two–dimension rectangular Channel fixed double stairs on fixed Reynolds number with different power law Indices (n) = 0.9 and 1.1

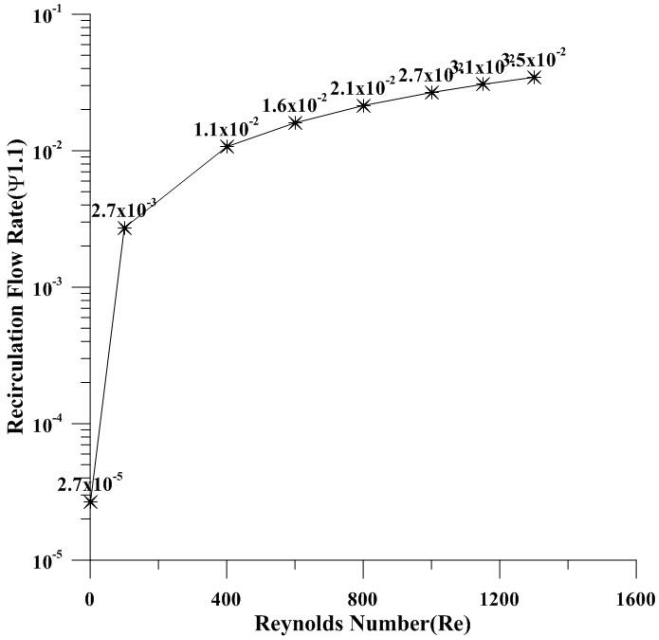
Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about



(a)



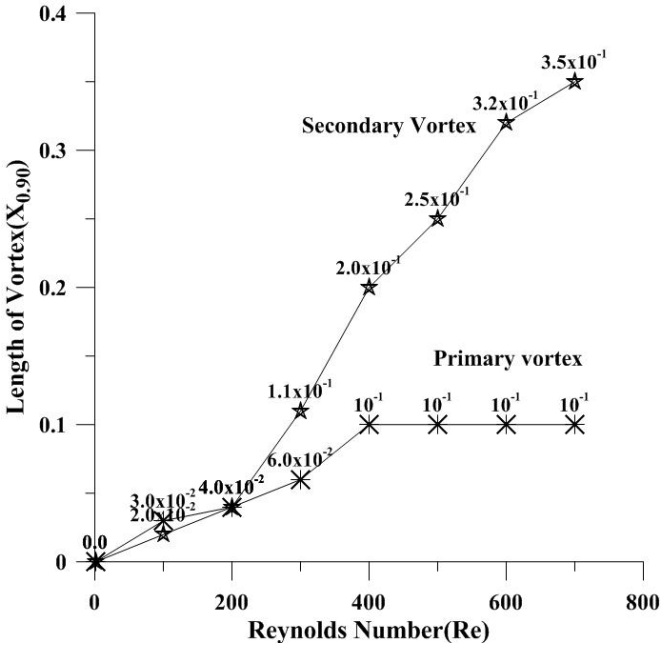
(b)

Picture –A.6(a and b): Recirculation flow rate (Ψ) at various Reynolds numbers (Re) for the two–dimension rectangular Channel fixed double stairs

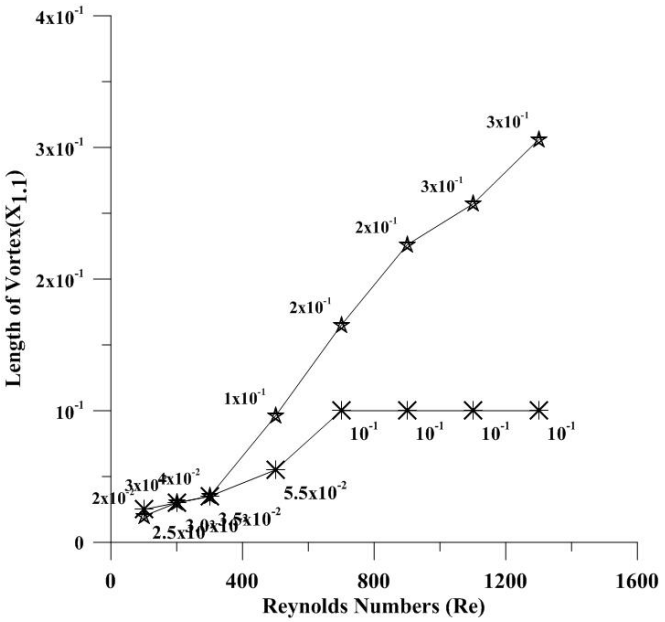
Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about



(a)



.(b)

Picture –A.7(a and b): Vortex length (X) at various Reynolds number (Re) for the two–dimension rectangular Channel fixed double stairs

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Recirculation Flow rate(Ψ)

Reyn Number (Re)	Power law index rate (1.1)		Reyn Number (Re)	Power law index rate (α = 0.90)	
	($\Psi_{1\ min}$)	($\Psi_{1\ max}$)		($\Psi_{1\ min}$)	($\Psi_{1\ max}$)
01	6.78e-11	2.68e-5	01	1.70e-10	6.72e-5
100	5.48e-9	2.72e-3	100	7.69e-9	6.64e-3
400	3.33e-8	0.0107e-2	200	1.70e-8	0.0133e-2
600	5.25e-8	0.016e-2	300	3.27e-8	0.0199e-2
800	6.82e-8	0.0214e-2	400	5.40e-8	0.0266e-2
1000	6.76e-8	0.0267e-2	500	5.28e-8	0.0331e-2
1150	6.72e-8	0.0306e-2	600	5.20e-8	0.0396e-2
1300	6.65e-8	0.0345e-2	700	5.13e-8	0.0464e-2

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Table–A.8: ***Minimum and Maximum values of Recirculation flow rate of non-Newtonian fluids at different Reynolds Number the two–dimension rectangular Channel fixed double stair.***

Vortex Length (X)					
Reynolds Number (Re)	**Power law index rate (1.1)**		**Reynolds Number (Re)**	**Power law index rate (α = 0.90)**	
	Primary	**Secondary**		**Primary**	**Secondary**
01	0.00	0.00	01	0.00	0.00
100	0.02	0.020	100	0.03	0.02
400	0.07	0.06	200	0.05	0.06
600	0.10	0.15	300	0.05	0.15
800	0.10	0.20	400	0.05	0.25
1000	0.10	0.25	500	0.05	0.35
1150	0.10	0.30	600	0.05	0.55
1300	0.10	0.35	700	0.05	0.15

Table–A.9: ***Vortex Length (X) of non-Newtonian fluids at different Reynolds Number the two–dimension rectangular Channel fixed double stairs***

CONCLUSION

Fully computed the steady state solution of governing equations followed by the non-Newtonian fluids at power law indices rate (0.90 and 1.1). The steady state solution is based upon the finite element numerical discretization with hybrid scheme as least square residuals. The final solution is obtained through the iterative technique known as Newton's method for all parameters as u and v, p and w. The

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

whole methodology is taken from the CFD package COMSOL Multiphysics and also, the package is benefitted in the research study as for the designing a research domain, fixed boundary conditions and setup the physics about the research domain. Similarly, at the same plate form, applied the Galerkin finite element way to sketch the grid system with number of triangular elements preferred. The various grid styles are tested as weakest, coarser, finer and extra fine grid style. Finally, applied the predicted procedure through the same package to compute the all physical unknown imposed in the governing equations.

Initially, observed the shear thinning fluids features in the channel with double step at different values of the fluid inertia. In these streamline flow patterns, observed various eddies at different locations of the steps. Due to thin fluids, large eddies observed very quickly and moved towards outflow of the channel at the value of the fluid inertia (=700).

Alternatively, when the thick fluids transported in the channel then the behave is altered and clearly visualized the behave of eddy appearance in the step of the channel. Very slow eddy appeared at the steps of the channel and too slowly increased in the size of the eddies. Finally occupied the region of the channel at the value of the fluid inertia (1400). The same enhancement also observed in the plots of the recirculation flow rate and computation of the eddies length. The predicted results are fully compared the flow pattern with the other flow patterns of different numerical result.

HIGHER STUDY EXTENSION

The double step hindrance can be extended more and also changed the location of the walls. The flow may be laminar towards turbulent. The fluids may be extended to analyze behave of other power law indices rates. Double step may be changed as rectangular into the circular and semicircular steps.

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

BIBLOGRAPHY

Afshar, M. H., & Hoseini, S. M. (2013). *Experimental and numerical study of flow over broad-crested weirs using turbulence models.* Flow Measurement and Instrumentation, 30, 50–58.

Al-Husseini, T. R. (2015). Flow characteristics over stepped spillways. *Journal of Hydraulic Engineering*, *141*(6), 04015010. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001005.

Al-Shukur, A. H. K., Al-Khalaf, S. K. H. and Al-sharifi, I. M. A., (2014), "Flow Characteristics and Energy Dissipation Losses in Different Configurations of Steps of Stepped Spillway", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 3, Issue 1.

Amin, F. (2025). Binary Flaw Detection: A Security Analysis Paper. *International Conference on Artificial Intelligence and Cybersecurity.*

Amin, F., (2025). A Scalable Framework for Interpretable Binary Vulnerability Analysis Using Data Dependency Modeling. *In Proceedings of the International Conference on Artificial Intelligence and Cybersecurity (ICAIC 2025)*, 249 – 255.

Amin, F., But, M. A., Amin, I., & Khan, A. (2024). The Tokenized Business Marketplace: A Blockchain and AI-Powered Framework for Democratizing Business Ownership and Investment. *International Journal of Business and Management Sciences*, *5*(4), 318-328.

Amin, F., Daudpota, N., & Khan, A. (2025). A Complete Penetration Testing Framework: Simulating Attacks and Evaluating Post-Exploitation Techniques with Kali Linux and Metasploit. *Spectrum of Engineering Sciences*, 386-407.

Amin, F., Said, I., & Butt, M. A. (2025). AI-Based Cybersecurity Solutions: Securing Information and Privacy in the Evolving Digital Age. *Journal of Engineering and Computational Intelligence Review*, *3*(2), 142-158.

Bird, R. B., Stewart, W. E., & Lightfoot, E. N. (2007). *Transport phenomena* (2nd ed.). John Wiley & Sons.

Burden, R. L., & Faires, J. D. (2005). *Numerical analysis* (8th ed.). Brooks/Cole.

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Cengel, Y. A., & Cimbala, J. M. (2014). *Fluid mechanics: Fundamentals and applications* (3rd ed.). McGraw-Hill Education.

Chanson, H. (2001). *The hydraulics of stepped chutes and spillways*. A. A. Balkema.

Chapra, S. C., & Canale, R. P. (2015). *Numerical methods for engineers* (7th ed.). McGraw-Hill.

Chaudhry, S. R. Anwar-ul-Haque, Fareed Ahmad, and Yamada, S., (2007), "Assessment of Turbulence Models for Turbulent Flow over Backward Facing Step," Proceedings of the World Congress on Engineering Vol II WCE, London, U.K.

Chhabra, R. P., & Richardson, J. F. (2011). *Non-Newtonian flow and applied rheology: Engineering applications* (2nd ed.). Butterworth-Heinemann.

COMSOL AB. (2020). *COMSOL Multiphysics reference manual* (Version 5.5). COMSOL.

Crest," International Journal of Civil & Environmental Engineering IJCEE-IJENS Vol: 14 No: 02. doi:10.3906/tar-1011-1506.

Dash, A., Amin, F., Sahoo, S. K., & Mishra, S. K. (2025). Secure comparative evaluation of Alzheimer MRI classification models using blockchain. In *2025 13th International Conference on Intelligent Systems and Embedded Design (ISED)* (pp. 905-911). IEEE.

Donea, J., & Huerta, A. (2003). *Finite element methods for flow problems*. John Wiley & Sons.

Felder, S., & Chanson, H. (2013). Energy dissipation, flow resistance and gas-liquid interfacial area in skimming flows on stepped spillways. *Environmental Fluid Mechanics*, *13*(6), 503–525. https://doi.org/10.1007/s10652-013-9298-0.

Ferziger, J. H., & Perić, M. (2002). *Computational methods for fluid dynamics* (3rd ed.). Springer.

Fox, R. W., McDonald, A. T., & Pritchard, P. J. (2015). *Introduction to fluid mechanics* (9th ed.). John Wiley & Sons.

Goncalves Matos, J.D.S. and Meireles, I.O.D.C. (2015), "Numerical modeling of skimming flow over stepped spillways: application on small embankment

Online ISSN 3007-3197
Print ISSN 3007-3189

dams," Institute Superior Tecnico, Lisboa, Portugal.

Haque, M. M., *et al.* (2007). *Flow reattachment in backward-facing step channels: A CFD study.* International Journal of Computational Fluid Dynamics, 21(5), 213–225.

Hassan, M., *et al.* (2014). *Energy dissipation analysis of different stepped spillway geometries.* Journal of Hydraulic Engineering, 140(5), 1–10.

Hossein, A., and Sayed Hosseini, H., (2013), " Experimental and 3-D Numerical Simulation of Flow over a Rectangular Broad-Crested Weir," International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-2.

Hughes, T. J. R. (2000). *The finite element method: Linear static and dynamic finite element analysis.* Dover Publications.

Imtiaz, U., Ahmad, B., Sajid, M. H., Abbas, Q., Qureshi, M. A., Rasheed, S., & Khan, A. (2025). An Integrated Machine Learning Framework for Structural Health Monitoring of Bridges: A Case Study on Soan Bridge. *The Asian Bulletin of Big Data Management*, *5*(2), 194-207.

Imtiaz, U., Malik, S., & Khan, A. (2024). Blockchain-Driven Cybersecurity Framework for Smart Homes: Integrating IoT and Machine Learning for Secure Automation. *The Asian Bulletin of Big Data Management*, *4*(4), 570-583.

Khalaf, R., *et al.* (2014). *Experimental investigation of flow regimes over semi-circular crest spillways.* Journal of Hydraulic Engineering, 140(7), 04014030.

Khalaf, R.M., Irzooki, R.H. and Saleh, J. S. S., (2014), "Flow Characteristics and Energy Dissipation Over Traditional and Stepped Spillway with Semicircular.

Khosro, M. Eghbalzadeh, A. and Soori, S., (2016), "Numerical Study of Energy Dissipation of Pooled Stepped Spillways", Civil Engineering Journal Vol. 2, No. 5.

Khosrow, P., *et al.* (2016). *Flow characteristics and energy dissipation in stepped spillways.* Water Resources Management, 30(12), 4353–4367.

Kisi, O., *et al.* (2008). *Modeling of flow over stepped spillways using soft computing techniques.* Journal of Hydraulic Research, 46(4), 508–515.

Annual Methodological Archive Research Review
http://amresearchreview.com/index.php/Journal/about
Volume 4, Issue 3 (2026)

Online ISSN 3007-3197
Print ISSN 3007-3189

http://amresearchreview.com/index.php/Journal/about

Lucio, G., *et al.* (2015). *Numerical modeling of skimming flow over stepped spillways using FLOW-3D.* Journal of Hydraulic Research, *53*(2), 249–258.

Morovati, K., Barati, R., & Azimi, A. H. (2016). Numerical simulation of flow over stepped spillways using Flow-3D. *Water Science and Engineering*, *9*(2), 123–130. https://doi.org/10.1016/j.wse.2016.06.003.

Munson, B. R., Young, D. F., Okiishi, T. H., & Huebsch, W. W. (2013). *Fundamentals of fluid mechanics* (7th ed.). John Wiley & Sons.

Quarteroni, A., & Valli, A. (2008). *Numerical approximation of partial differential equations*. Springer.

Rad, I.N., (2010), "An Investigation of Flow Energy Dissipation in Simple.

Rad, M., & Teimouri, M. (2010). *Numerical investigation of energy dissipation in stepped channels using finite volume method.* Applied Mathematical Modelling, 34(9), 2561–2572.

Reddy, J. N. (2006). *An introduction to the finite element method* (3rd ed.). McGraw-Hill.

Salmasi, f, and samadi, a., (2018), "experimental and numerical simulation of flow over stepped spillways", department of science and water engineering, faculty of agriculture university of Tabriz Tabriz Iran.

SALMASI, F. Sattari, M.T. and Pal, M., (2010), "Application of data mining on evaluation of energy dissipation over low gabion-stepped weir," Turk,J Agric For 36 (2012) 95-106 © TÜBİTAK.

Shah, S. M. H., Amin, F., & Khan, A. (2025). Cyber-Resilient Mobile Edge Computing: A Deep Neural Approach for Secure and Efficient Task Offloading. *The Asian Bulletin of Big Data Management*, *5*(1), 200-215.

Soma, R., Sahoo, S. K., Amin, F., & Mishra, S. K. (2025). A Federated Learning Framework for Multi-Parameter Optimization in Edge Computing. In *2025 13th International Conference on Intelligent Systems and Embedded Design (ISED)* (pp. 1-6). IEEE.