01/19    (123)

# Numerical Analysis of Heat Transfer in a Pipe Using Twisted Tape Inserts

**R.Santhaseelan[1], Dr. C. Mathalai Sundaram[2], T. Sudarsanan[3]**

[1] *Assistant Professor, Department of mechanical Engineering, Nadar Saraswathi College of Engineering & Technology, Theni*

[2] *Principal, Nadar Saraswathi College of Engineering & Technology, Theni*

[3] *Assistant Professor, Department of mechanical Engineering, Nadar Saraswathi College of Engineering & Technology,Theni*

**Abstract -** This paper reports numerical examinations of warmth move qualities in swirling stream conditions utilizing CFD simulation. A commercial CFD package, STAR CCM+, was utilized in this examination. 3D models for circular tube fitted with twisted tape inserts were created for the simulation. The swirling stream was presented by utilizing wound tape set inside the circular pipe. The outcomes got from the twisted tape insert are compared with those without turned tape. The information acquired from the CFD reproduction was confirmed with inlet and outlet temperature distinction and warmth exchange attributes. The outcomes show that there was a significant increase in heat transfer coefficient and Reynolds number in the pipe fitted with twisted tape.

**Keywords -** *swirling* flow, twisted tape, circular pipe, heat transfer coefficient

## I.    INTRODUCTION

### 1.1 Overview of CFD

Computational fluid elements, typically condensed as CFD, are a part of fluid mechanics that utilizes numerical techniques and calculations to take care of and break down issues that include fluid streams. PCs are utilized to play out the estimations required to recreate the collaboration of fluids and gases with surfaces characterized by limit conditions. With rapid supercomputers, better arrangements can be accomplished. Continuous research yields programming that enhances the exactness and speed of complex reproduction situations, for example, transonic or fierce streams. Starting trial approval of such programming is performed utilizing a breeze burrow with the last approval coming in full-scale testing..

### 1.2 Finite volume method

The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows (like combustion).

In the finite volume method, the governing partial differential equations (typically the Navier - Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes. This discretisation