

CFD Analysis of Heat Transfer Enhancement in a Channel Pipe Using Anchor-Shaped Inserts

Guillaume VALLENET^{a,b}, Teerapat Thungthong^a, Weerachai Chaiworapuek^a, Jay-Tawee Pukrushpan^{a,*}

^aDepartment of Mechanical Engineering, Faculty of Engineering, Kasetsart University, Bangkok 10900, Thailand

^bDepartment of Mechanical Engineering, Faculty of Engineering, Ecam La Salle, Lyon 69005, France

*Corresponding author: fengjwp@ku.ac.th

Abstract

Improving convective heat transfer in circular channels is important for the design of efficient thermal systems such as heat exchangers and cooling channels. In this study, a numerical investigation is performed to evaluate the thermo-hydraulic behavior of a circular pipe equipped with anchor-shaped inserts. In addition to the conventional straight anchor insert, two modified configurations with twist angles of 30° and 45° are analyzed in order to promote stronger flow structures and enhanced fluid mixing. The geometries are modeled in ANSYS Fluent under steady-state conditions with a constant wall heat flux. The k- ω SST turbulence model is used to resolve near-wall thermal gradients. Simulations are performed at Reynolds numbers of 3000 and 10000 to evaluate the influence of flow conditions on heat transfer and pressure losses. The results show that twisted anchor inserts significantly enhance fluid mixing and heat transfer compared with the straight configuration, with the 30° twisted insert providing the best thermo-hydraulic balance.

Keywords: Heat transfer enhancement, Anchor-shaped, Circular channel

Introduction

Enhancing heat transfer in internal flows is important for improving the efficiency of thermal systems such as heat exchangers and cooling channels. In smooth pipes, the formation of a thermal boundary layer limits convective heat-transfer performance. Passive enhancement techniques, such as internal inserts, are commonly used to disturb the flow and promote fluid mixing without additional energy input (Versteeg and Malalasekera, 2007; Manglik and Bergles, 1993). Anchor-shaped inserts can generate vortices and secondary flows that disrupt the thermal boundary layer, but straight inserts may also increase pressure losses (Patankar, 1980). In this study, twisted anchor-shaped inserts with rotation angles of 30° and 45° are numerically investigated and compared with a smooth pipe and a straight anchor insert to evaluate their effects on heat transfer and flow resistance.

Geometry and Numerical Setup

Geometry

- Pipe diameter: 60 mm
- Pipe length: 1850 mm
- 30 anchor elements
- Element spacing: 60 mm

Insert Configurations

- Smooth pipe
- Straight anchor insert
- 30° twisted anchor insert
- 45° twisted anchor insert

CFD Setup

- Software: ANSYS Fluent
- Steady-state simulations
- k- ω SST turbulence model
- Constant wall heat flux boundary condition

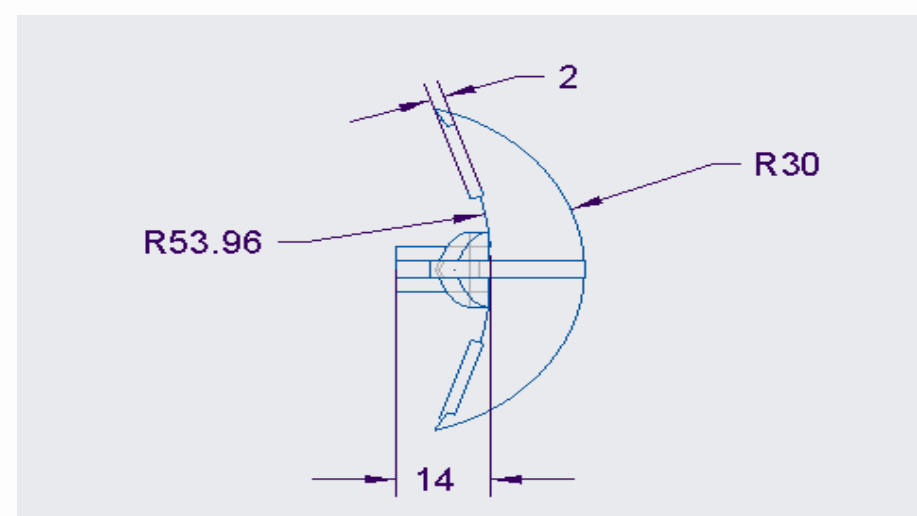


Fig. 1. Top view of an anchor-shaped component

Performance Parameters

Reynolds Number

$$Re = \frac{\rho U D}{\mu}$$

Defines the flow regime inside the pipe.

Nusselt Number

$$Nu = \frac{h D}{k}$$

Represents the convective heat transfer performance.

Friction Factor

$$f = \frac{\Delta P D}{2 \rho U^2 L}$$

Indicates the pressure loss caused by the inserts.

Thermal Enhancement Factor

$$TEF = \frac{Nu/Nu_s}{(f/f_s)^{1/3}}$$

Evaluate the overall thermo-hydraulic efficiency relative to the smooth pipe.

Acknowledgements

The author gratefully acknowledges the financial support provided by La Région Auvergne-Rhône-Alpes through a regional scholarship program, which made this research work possible.

Results

Flow Structure

- The smooth pipe shows a relatively uniform velocity distribution with limited mixing inside the flow.
- The introduction of insert geometries disturbs the flow field, generating secondary flows and vortices.
- These vortical structures promote stronger fluid mixing between the core region and the pipe wall.
- The twisted inserts (30° and 45°) produce more pronounced swirl motion compared to the straight anchor insert.
- This enhanced mixing mechanism contributes to improved thermal boundary layer disruption, which leads to higher heat transfer performance.

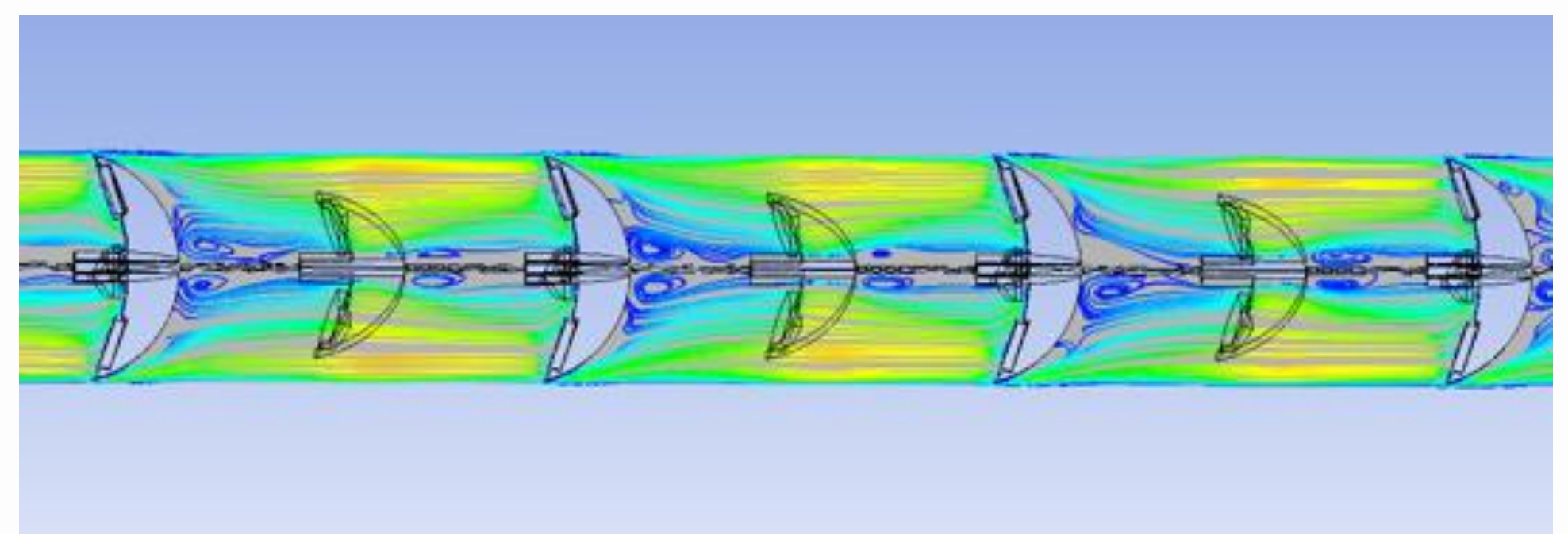


Fig. 2. Velocity streamlines in the axial plane for 45° twisted anchor insert.

Heat Transfer

The intensified mixing produced by the twisted inserts significantly enhances convective heat transfer. The Nusselt number increases by approximately 41–42% compared with the smooth pipe, demonstrating the effectiveness of twisted geometries for heat-transfer augmentation.

Effect of Reynolds Number

- Increasing the Reynolds number from 3000 to 10000 significantly enhances the heat transfer performance in all configurations.
 - Higher Reynolds numbers promote stronger fluid mixing and increased turbulence, which improves convective heat transfer at the pipe wall.
 - As a result, the average Nusselt number increases for all cases, indicating a higher heat transfer rate.
 - This effect is particularly noticeable for the insert configurations, where the generated vortices further intensify the mixing process.
- Overall, the results confirm that heat transfer enhancement becomes more pronounced at higher Reynolds numbers.

Conclusion

- Insert geometries significantly modify the flow structure and enhance fluid mixing inside the pipe.
- Compared to the smooth pipe, the inserts improve heat transfer by disrupting the thermal boundary layer.
- Twisted inserts generate stronger swirl flow and provide higher heat transfer performance.
- Increasing the Reynolds number further enhances the overall thermal performance.

References

- Manglik R. M. and Bergles A. E., 1993, *Heat transfer and pressure drop correlations for twisted-tape inserts in isothermal tubes: Part I—laminar flows*, Journal of Heat Transfer, vol. 115, no. 4, pp. 881–889.
- Patankar S. V., 1980, *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation, New York.
- Spiegelman M., *Conservation equations*, course lecture notes on continuum mechanics and fluid dynamics, unpublished.
- Versteeg H. K. and Malalasekera W., 2007, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed., Pearson Education, Harlow.
- Xie Y., Zhang Y., Wang H., and Liu Z., 2018, *Numerical study on flow structure and heat transfer in a channel with internal vortex-generating structures*, Applied Thermal Engineering, vol. 136, pp. 502–514.