International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878, Volume-8 Issue-4, November 2019

Optimization of Flexible Hose for Reducing Flow Maldistribution in Manifolds

Mohit Kumar, V K Bajpai

***Abstract:** Coolant Distribution Systems (CDS) are required to be optimally designed to ensure predefined mass flow rate in all parallel channels between exit and collecting manifolds, while maintaining low-pressure drop across them. Even small change in the pressure drop at component level will result in maldistribution of flow rate and increase in overall pressure drop of CDS. Numerical and experimental study had been carried out in the work proposed for change in pressure drop due to deformation of end connector's bend cross section, in a flexible hose. The methodology for optimum modelling of the problem on CFD tool using sub-structuring method is also suggested. In sub-structuring method a part of complete hose (henceforth referred as sub-structured model), has been used instead of complete model of flexible hose. Results of sub-structuring model were compared with that of a complete model of a flexible hose. Numerical values obtained from simulation were validated with experimental results. Optimum bend cross section for avoiding maldistribution in parallel channels and increase in pressure drop of CDS were calculated. Substantial reduction in computation cost was achieved with negligible loss of accuracy in pressure drop values.*

***Keywords:** CFD, Maldistribution, Pressure Drop, Turbulence.*

I. INTRODUCTION

Distribution manifolds find application in coolant distribution systems (CDS), heat exchangers etc. Coolant distribution systems are typically complex networks with several channels in series/parallel as per requirement of systems. Each channel consists of several components for the purpose of distribution, heat exchange, sensing parameters and controlling devices to regulate parameters like mass flow rate, pressure etc. If such systems are designed for low pressure operation, slight rise in pressure drop becomes an appreciable percentage of overall pressure drop in the system. Shape and size of cross-section has effect on pressure drop of various components of system apart from other variables like length, surface roughness etc. Change in pressure drop of any component of CDS may require additional tuning with control valve, thus resulting in rise of overall pressure drop of system.

Flexible hoses (Fig.1) form an important part of any CDS. These hoses are composed of rubber tube with metallic 'end connectors' on each end. 'End connectors' may be straight, bend at 45° or 90° depending on system requirement. During fabrication of flexible hoses, it is not possible to maintain the end connector's cross-section as circular throughout. It generally deforms to an elliptical shape at bend location. The proposed work studies the effect of this deformation on pressure drop of a flexible hose. Numerical simulation and experimental validation had been carried out in this regard. Effect of deformation has been evaluated for different Reynolds number ($\boldsymbol{R_e}$). Values of $\boldsymbol{R_e}$ at which pressure drop is evaluated are those values, which are typically used in industry for a selected size of flexible hose. Literature review carried out has revealed that the effect of variation (due to manufacturing tolerances) in shape of cross-section of end connector of flexible hose in parallel channels, on fluid parameters of a CDS is not available in open literature to the best of knowledge of the authors. However, research had been done in related areas as summarized below.

Reference [1]-[4] reported the effect of geometry of exit-port cross-section on variation in mass flow rate effusion from distribution manifold. Reference [5] suggested concept of second header configuration for improving the uniformity of flow distribution. Reference [6] presented numerical study for optimal placement of guide vanes in combining headers to achieve low-pressure drop and uniformity of flow across channels. Reference [7]-[9] reported the parameters influencing velocity distribution and static pressure in distribution header and connecting tubes. Reference [10]-[11] reported experimental and numerical simulations results for flow distribution in parallel tubes between headers, subjected to various operating conditions. Reference [12],[13] presented new empirical relationship for pressure drop calculation of flow separation of around sharp 90° pipe bends. Reference [14]-[18] reported fluid properties in vicinity of bend. Reference [19]-[21] proposed empirical equation for flow parameters of non-circular duct. The effect of different internal waviness in the vicinity of a 90° bend for different curvature radius had been studied previously [22]. Empirical correlation has been proposed by the authors for flow characteristics of an internally wavy bend. The simulated flow patterns and heat transfer in duct with a wavy-wall had been reported [23]. The effect of a mitre bend on the pressure drop in a microchannel had been studied [24].

Revised Manuscript Received on November 25, 2019.

* Correspondence Author
Mohit Kumar*, Research Scholar, NIT Kurukshetra, Haryana, 136119, India. Email: mkabcd@gmail.com
Prof. V K Bajpai, NIT Kurukshetra, Haryana, 136119, India. . Email: vkbajpai@yahoo.com

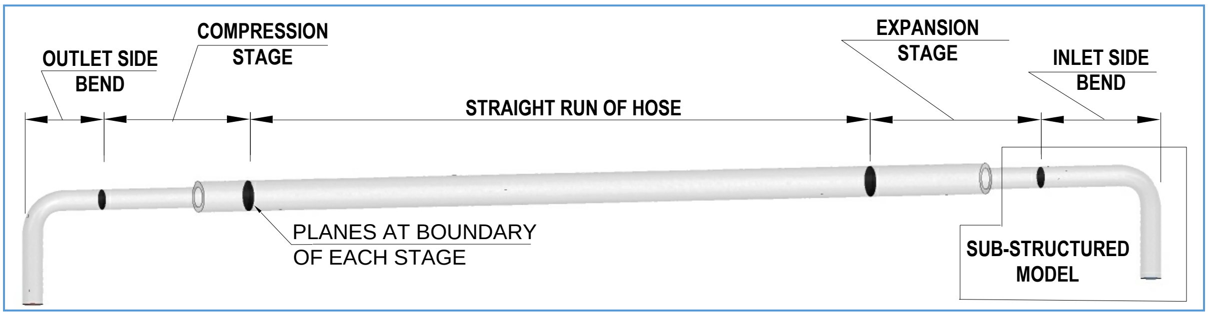


Fig. 1.Inside fluid volume of flexible hose

Authors had reported that at low $\boldsymbol{R_e}$ pressure drop is a function of $\boldsymbol{R_e}$ alone. However, at higher values of $\boldsymbol{R_e}$ pressure drop becomes a function of $\boldsymbol{R_e}$ and hydraulic diameter. Further, beyond a certain threshold values of $\boldsymbol{R_e}$ pressure drop becomes constant because vortices get saturated. The effect of aspect ratio on flow pattern, pressure drop and void-fraction in a rectangular microchannel had been reported for adiabatic two-phase flow [25]. Authors had reported that void-fraction is more dependent on aspect ratio than hydraulic diameter. Both aspect ratio and hydraulic diameter impacts flow characteristic and hence both need to be evaluated independently.

In all the above reported research works, effect of geometry of manifold, and other parameters of CDS on flow maldistribution in parallel channels and pressure drop of CDS had been evaluated. The effect of bends on fluid properties at bend location or near its vicinity had also been reported. However, the effect of deformation of bend of end connector of flexible hose has never been reported. In flexible hose with 90^{o} end connector on each end, fluid changes direction twice. There is expansion followed by compression between two bends because of different diameters of end connectors and hose (Fig. 1). Hence three effects viz. change of direction, expansion and compression are to be evaluated simultaneously. The proposed research work evaluates effect of end connector deformation, for different $\boldsymbol{R_e}$, on pressure drop of flexible hose. Effects of intermediate expansion and compression between bends of two end connectors were taken into consideration together with deformation of bend cross section. The fluid properties, fluid phase and other variables were kept constant.

Analysis of pressure drop was carried out by utilizing CFD simulation. Proposed work also suggests method of decreasing computation cost of CFD simulation by utilizing sub-structuring method with negligible loss of accuracy.

II. NUMERICAL SIMULATION

A. Modeling of end connector of flexible hose

During manufacturing process cross section of end connector of flexible hose is deformed to elliptical instead of circular [26], [27]. Degree of deformation varies each time due to variation in manufacturing process. Optimum value of this deformation is required to be calculated to decide manufacturing tolerances. This is because too tight tolerance means high manufacturing cost. It is worth mentioning here that, due to improper control of manufacturing process or faulty raw material etc., different irregular shapes are possible. End connectors with irregular shapes are considered as defective. These types of defective items can be easily identified by visual inspection and are rejected. Also extreme values of major and minor diameter of ellipse are not practical. The proposed work does not evaluate these cases of extreme aspect ratio experimentally. However, simulation results have been presented.

B. Assumptions

i. Both the connectors of the flexible hose lie in the same plane.
ii. Connectors at both ends of the flexible hose are taken as identical in simulation to reduce number of iterations runs.
iii. Diameter (6.5mm) of a commercially available standard flexible hose for a 3/8” size of British Standard Pipe(BSP) with its standard connector was used.
iv. Cross section of end connector is elliptical at bend location instead of circular. The perimeter of ellipse and original circular pipe (which was bent to create end connector) are same {Eqn (1)}. Since perimeter is held constant, minor axis radius ‘**a**’ becomes a dependent variable. Therefore, major axis radius ‘**b**’ of ellipse alone is sufficient to measure of degree of deformation, in place of ellipticity.

$$\text{Perimeter} = 2\,\pi\,\sqrt{\{(a^2+b^2)/2\}} = 2\,\pi\,R \qquad (1)$$

Where, R = radius and
$2\,\pi\,R$ is circumference of original circular pipe

C. CFD Turbulence Simulation Model

In proposed work, *k-ω-SST (Shear stress transport) model* has been used [28]. *k-ω-SST r*equires of low value of Y+ (< 1) thus requiring finer mess compared to popular k-ε model. This limitation increases the computation cost slightly but, because of versatile nature of *k-ω-SST* for reliably dealing with large variations of flow conditions, this model had been used. SST model is available in most commercial CFD software tools. Therefore, Ansys Fluent, Release 19.0 has been used instead of solving this model numerically.

D. Numerical simulation methodology

In the proposed work, sub-structure method has been used to reduce the complexity in CFD simulation. Validation of sub-structuring scheme has been done by evaluating the complete hose (Fig.1) and comparing its difference with that of sub-structured model

(Fig.1). For simulating the hose, the fluid volume inside of the hose (Fig.1) has been extracted.

Ansys Fluent has been used as solver, and CFD Post as post-processor. Planes were created at different locations (Fig.1) in post processor and 'Area weighted pressure' was measured at these plane locations. Pressure drop was calculated at various stages (inlet and outlet bends, expansion stage, compression stage and straight run) of the flexible hose (Fig.1). The difference in pressure drop of a particular stage was calculated by taking difference of average pressure at planes on either end of corresponding stage. Major and minor diameters of ellipse were tweaked across simulation iterations and different input velocities were studied for each profile to study the impact of different R_e. Pressure drop for circular cross section corresponds to case where major and minor radius of ellipse are equal. As is evident from Fig.1, outlet side bend is having larger length upstream of bend and smaller length downstream, when compared with inlet side bend. Length of connector upstream of bend does not have any appreciable impact on the pressure drop. This is a known fact from literature. For downstream side, smaller length condition of connector need not be modeled separately. This condition was simulated by placing additional plane at equivalent distance in CFD post processor, while simulating sub-structured model. Difference of pressure drop between inlet plane and this plane gives pressure drop for reduced length.

III. EXPERIMENTATION

Test jig (Fig. 2) has been used for experimental verification of numerical model. Test jig setup consists of fluid reservoir, motor with pump, flow rate sensor, pressure sensors and control valves. The flow rate can be changed within range 0-30 lpm (litre per minute) on a 3/8-inch BSP hose. Flow rate in flexible hose can be varied by adjusting ratio of flow rate between main loop and bi-pass loop. Pressure drop of flexible hose can be calculated as difference of reading of pressure sensors P1 and P2.

Experimental Setup

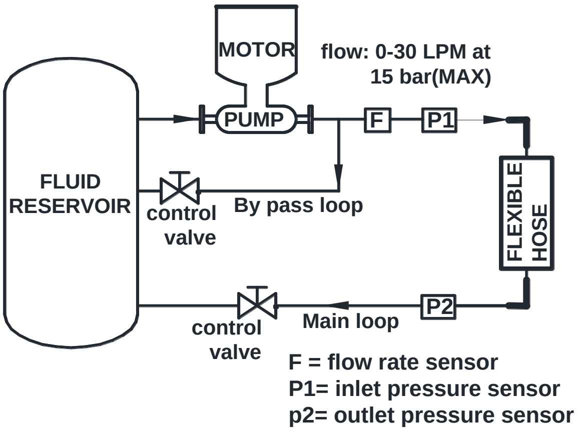


Fig. 2. Test setup for experimental validation

IV. RESULTS AND DISCUSSION

Pressure drop of bend was calculated for both model of complete hose and its sub-structured model (Fig.1), with the help of Eq. (2) and Eq. (3) respectively.

$$\Delta P_{o,c} = \Delta P_{b1,c} + \Delta P_{ep} + \Delta P_{st} + \Delta P_{cm} + \Delta P_{b2,c} \qquad (2)$$

$$\Delta P_{o,s} = \Delta P_{b1,s} + \Delta P_{ep} + \Delta P_{st} + \Delta P_{cm} + \Delta P_{b2,s} \qquad (3)$$

Where,

$\Delta P_{o,c}$	Overall pressure drop of flexible hose, simulated with its complete model on CFD tool
$\Delta P_{b1,c}$	Pressure drop of inlet side bend, simulated with complete model of flexible hose
ΔP_{ep}	Pressure drop of expansion stage of flexible hose
ΔP_{st}	Pressure drop of straight run of flexible hose
ΔP_{cm}	Pressure drop of compression stage of flexible hose
$\Delta P_{b2,c}$	Pressure drop of outlet side bend simulated with complete model of flexible hose
$\Delta P_{o,s}$	Overall pressure drop of flexible hose, simulated with its sub-structured model on CFD tool
$\Delta P_{b1,s}$	Pressure drop of inlet side bend, simulated with sub-structured model of flexible hose
$\Delta P_{b2,s}$	Pressure drop of outlet side bend simulated with sub-structured model of flexible hose

A. Numerical simulation results

Pressure drop v/s major axis radius of end connector of flexible hose, for different R_e has been plotted in Fig. 3. Fig. 4 compares the simulated pressure drop obtained with sub-structured model and model of complete hose (Fig.1), for bend stages (inlet side and outlet side). Magnitude difference between the two was observed to be at second decimal place or lesser. Therefore, it was concluded that the sub-structured model could be used to simulate the changes in the pressure drop with a good accuracy level.

The pressure drop results of complete hose (Fig.5), shows that the pressure drop in stages other than bend stage (viz. expansion stage, compression stage, straight length) of hose (Fig.1) is constant, for all dimensional deviations in the bend cross section. However, increase in pressure drop was observed when R_e was increased. Straight lines (Fig. 4) for all values of R_e, clearly depicts that deformation of bend cross section is not having any appreciable impact on pressure drop of stages other than bend stage. Effect of local turbulence from the previous stage diminishes before the start of next stage, since length of the stage is sufficient to damp this turbulence. However, for a case when the length is not sufficient this may not be the case. Care must be taken so that, turbulence effect of preceding stage is not carried to next stage. This can be easily observed by studying the eddy viscosity plot in software, which can show the distribution of turbulence. It can thus be concluded that Eq. (2) and Eq. (3) holds good, provided superposition of turbulence of two stages do not take place. For proposed case, this assumption holds good.

Optimum value of tolerances on bending parameter to be used while fabrications of end connector, to sustain pressure drop value of flexible hose below targets, are available from graph.

B. *The observations based on numerical simulation are summarized as below:*

1. When R_e is low, change in values of pressure drop due to incorporation of effect of deformation of cross section of end connector of flexible hose is insignificant. Modeling of end connector cross section as circular is sufficient for these cases.
2. For higher values of R_e, there is significant rise in pressure drop values predicted by deformed cross section, compared to that predicted by ideal (circular) cross section of end connectors of flexible hose.
3. Pressure drop increases linearly until threshold value (b<3.8) after this it rises sharply (Fig. 4) with degree of ovalisation (b). There is nearly 10 percent increase in pressure drop at threshold point, for all R_e.
4. Fig. 4 and Fig. 5 confirms that, sub-structured model (Fig.1) may predict pressure drop within accuracy level suitable for all practical purpose in industry. This is because turbulence of bend is not carried to next stage and they remain constant

C. *Experimental validation*

Comparison of numerically calculated results with experimental values of pressure drop of entire hose has been carried out. Overall pressure drop has been calculated by Eq. (2) & Eq. (3) and are tabulated in Table I and plotted in Fig. 6 for following hose.

For flexible hose with elliptical cross-section of bends having Major axis radius, b for

i. Inlet side bend = 3.8 mm
ii. Outlet side bend = 3.8 mm
ΔP_{meas} = experimentally measured value of pressure drop.

Analysis of Table I and Fig. 6 reveals that, percent error with respect to experimentally measured values has reduced to single digit value, when ovalisation of cross section of end connector is considered in simulation. This error was typically, in double digit (14.3% to 18.3%) when cross section of end connector was taken as circular in simulation. CFD simulation accuracy had been kept to second decimal place only, to optimize computation cost. Overall pressure drop for low values of ovalisation (in terms of b) and R_e are also of same order. Therefore, this error gets reflected in CFD simulation results for these cases. However, it is insignificant for practical engineering systems, due to low absolute value.

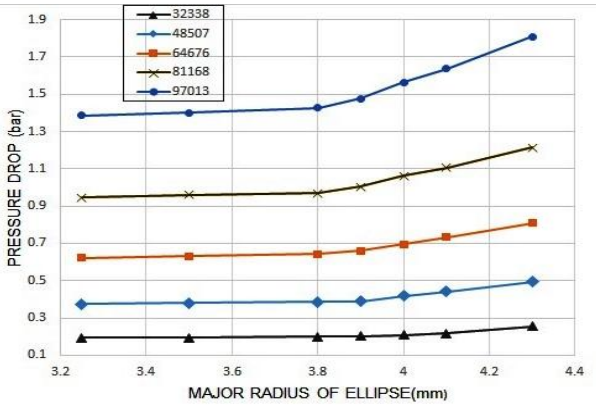
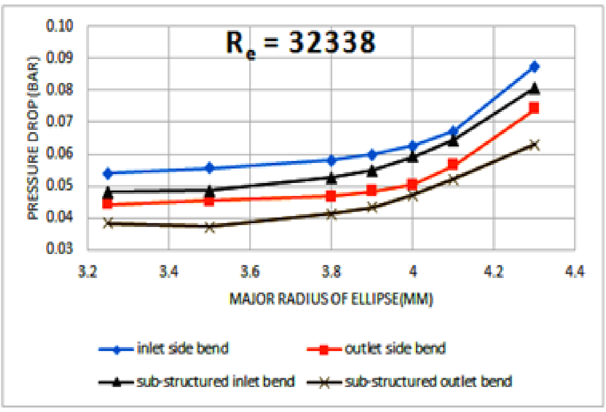


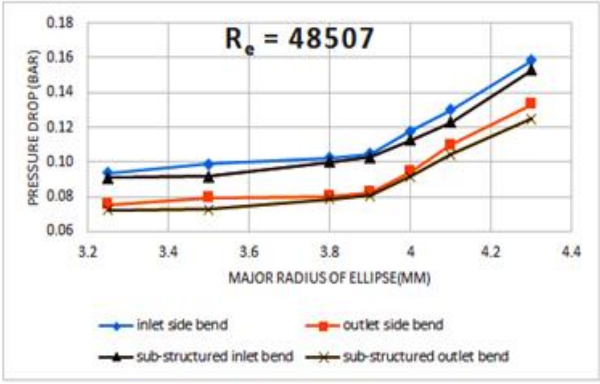
Fig. 3. Pressure drop for flexible hose, for different R_e

Table I and Fig. 6 also reconfirm the fact that, accuracy levels achieved with sub-structured method of simulation are very close to those of full model of flexible hose. Hence, significant amount of computation cost can be saved by sub-structuring method.

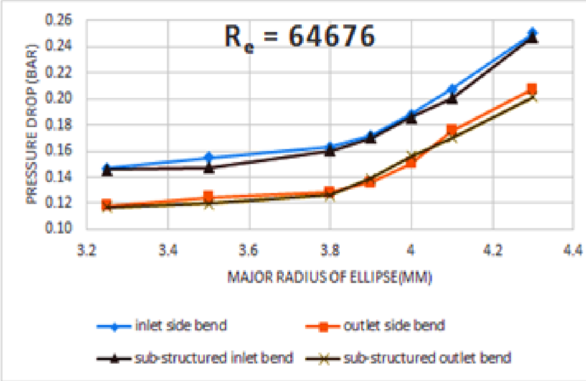
Optimal values of manufacturing tolerances on bending process of end connectors of flexible hoses can be derived by proposed scheme.



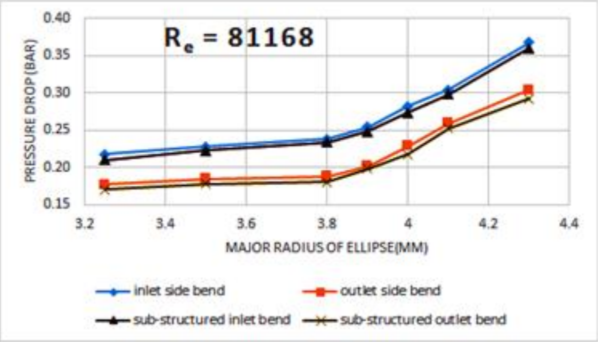
(a) $R_e = 32338$



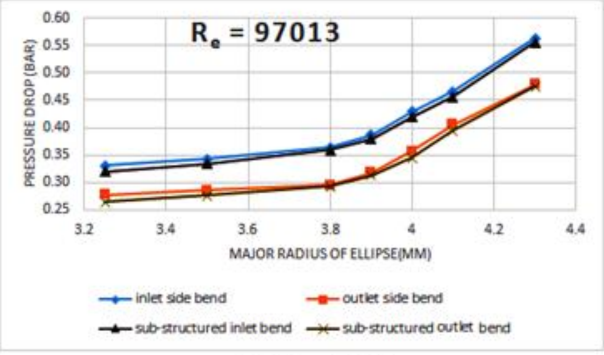
(b) $R_e = 48507$



(c) $R_e = 64676$

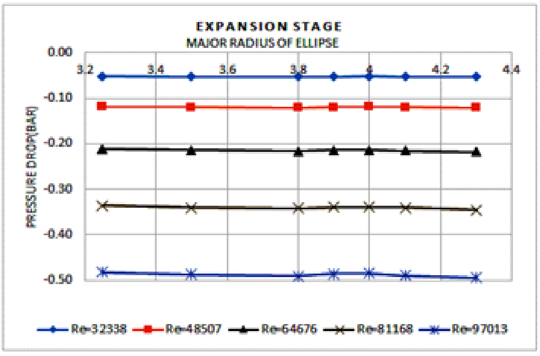


(d) $R_e = 81168$

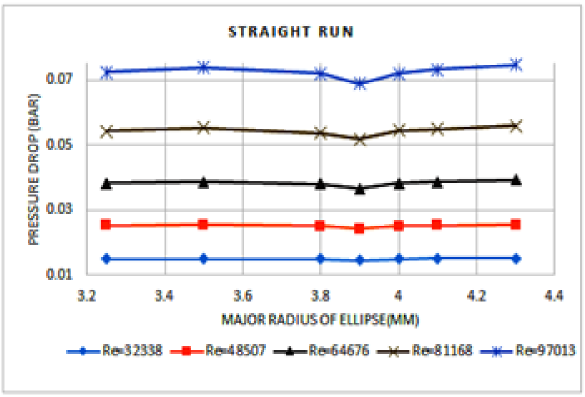


(e) $R_e = 97013$

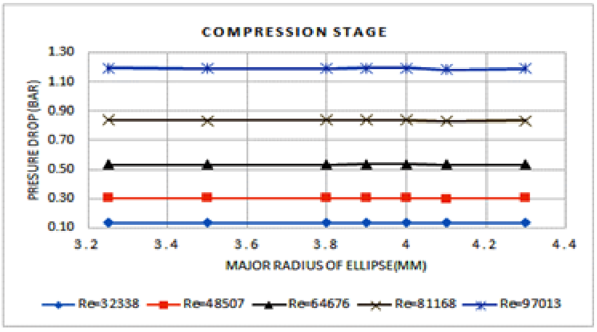
Fig. 4. Pressure drop values for inlet and outlet bend stages against major axis radii, for different R_e



(a) Expansion stage



(b) Straight run for hose



(c) Compression stage

Fig. 5. Pressure drop values for other stages of hose against major axis radii for different R_e

Table I Comparison of experimentally measured and simulated values

R_e	ΔP_{meas} (bar)	ΔP_{cir} (bar)	$\Delta P_{o,c}$ (bar)	$\Delta P_{o,s}$ (bar)	E_{rr} in ΔP_{cir}(%)	Err in $\Delta P_{o,c}$(%)	Err in $\Delta P_{o,s}$(%)
32338	0.21	0.18	0.20	0.19	14.3	4.8	9.5
48507	0.42	0.36	0.39	0.38	14.3	7.1	9.5
64676	0.73	0.62	0.68	0.67	15.1	6.8	8.2
81168	1.09	0.92	1.02	0.99	15.6	6.4	9.2
97013	1.64	1.34	1.50	1.48	18.3	8.5	9.8

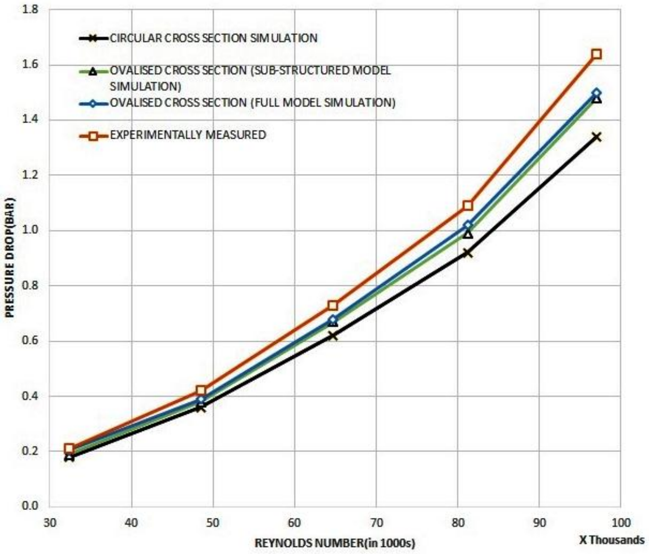


Fig. 6. Comparison of experimentally measured and simulated values

V. CONCLUSION

Pressure drop values were calculated using CFD analysis for complete model of flexible hose and with sub-structure method for its end connectors. Results were compared with experimental values and following conclusions are drawn:

a. When proposed model with deformed end connector of flexible hose is used in simulation, reduction in error of calculated results is observed w.r.t. experimentally measured values. Error in simulated results with proposed model is reduced to less than half for most of cases, when compared with results of ideal end connector's geometry.

b. Increase in accuracy in terms of percentage is observed for all cases.

This accuracy is more significant at high R_e, where proposed model is highly recommended. In these cases, accuracy improved is high for nominal increase of computation cost, especially if proposed sub-structured method is utilized.

c. At low values of R_e for all practical engineering applications extra computation cost involved is not justified.
d. Pressure drop increases with increase in ovalisation (in terms of b) of end connector's cross-section. Pressure drop increase is non-linear and is negligible till threshold value of b (≈ 3.8 mm). After this threshold, pressure drop rises sharply.
e. For range of R_e under consideration i.e 32000 to 100000, change in pressure drop is a function of deformation (ovalisation) of end connector's cross section at bend location.
f. Increase in error (for pressure drop) when using sub-structured model of flexible hose instead of its complete model is negligible for practical engineering application. Hence, sub-structure method is proposed to reduce computation cost.

Improvement in accuracy of pressure drop values of flexible hose was observed when deformation of its end connector was included in CFD simulation. Precise calculation of pressure drop of elements in CDS results in more predictable flow rate in system. Reduction in requirement of flow control valve tuning is observed, which in turn results in reduction of overall pressure drop in CDS.

REFERENCES

1. Chen, A. W., & Sparrow, E. M. (2009). Effect of exit-port geometry on the performance of a flow distribution manifold. Applied Thermal Engineering, 29(13), 2689-2692.
2. Tong, J. C., Sparrow, E. M., & Abraham, J. P. (2009). Geometric strategies for attainment of identical outflows through all of the exit ports of a distribution manifold in a manifold system. Applied thermal engineering, 29(17-18), 3552-3560.
3. Zhou, J., Sun, Z., Ding, M., Bian, H., Zhang, N., & Meng, Z. (2017). CFD simulation for flow distribution in manifolds of central-type compact parallel flow heat exchangers. Applied Thermal Engineering, 126, 670-677.
4. Yang, H., Wang, Y., Ren, M., & Yang, X. (2017). Effect of the rectangular exit-port geometry of a distribution manifold on the flow performance. Applied Thermal Engineering, 117, 481-486.
5. Jiao, A., Zhang, R., & Jeong, S. (2003). Experimental investigation of header configuration on flow maldistribution in plate-fin heat exchanger. Applied Thermal Engineering, 23(10), 1235-1246.
6. Kim, S., Kim, K., & Son, C. (2016). Optimum arrangements of guide vanes in a combining header and its effect on the performance of a tubular heat exchanger. Applied Thermal Engineering, 103, 1145-1155.
7. Miao, Z., & Xu, T. (2006). Single phase flow characteristics in the headers and connecting tube of parallel tube platen systems. Applied thermal engineering, 26(4), 396-402.
8. Camilleri, R., Howey, D. A., & McCulloch, M. D. (2015). Predicting the flow distribution in compact parallel flow heat exchangers. Applied Thermal Engineering, 90, 551-558.
9. Wang, J., & Wang, H. (2015). Discrete method for design of flow distribution in manifolds. Applied Thermal Engineering, 89, 927-945.
10. Wang, C. C., Yang, K. S., Tsai, J. S., & Chen, Y. (2011). Characteristics of flow distribution in compact parallel flow heat exchangers, part I: Typical inlet header. Applied Thermal Engineering, 31(16), 3226-3234.
11. Wang, C. C., Yang, K. S., Tsai, J. S., & Chen, Y. (2011). Characteristics of flow distribution in compact parallel flow heat exchangers, part II: modified inlet header. Applied Thermal Engineering, 31(16), 3235-3242.
12. Crawford, N. M., Cunningham, G., & Spence, S. W. T. (2007). An experimental investigation into the pressure drop for turbulent flow in 90 elbow bends. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, *221*(2), 77-88.
13. Crawford, N. M., Cunningham, G., & Spedding, P. L. (2003). Prediction of pressure drop for turbulent fluid flow in 90 bends. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, *217*(3), 153-155.
14. Daneshfaraz, R., Rezazadehjoudi, A., & Abraham, J. (2018). Numerical investigation on the effect of sudden contraction on flow behavior in a 90-degree bend. *KSCE Journal of Civil Engineering*, *22*(2), 603-612.
15. Shusser, M., Ramus, A., & Gendelman, O. (2016). Flow in a Curved Pipe With a Sudden Expansion. *Journal of Fluids Engineering*, *138*(2), 021203.
16. Gorman, J. M., Sparrow, E. M., Smith, C. J., Ghosh, A., Abraham, J. P., Daneshfaraz, R., & Rezazadeh Joudi, A. (2018). In-bend pressure drop and post-bend heat transfer for a bend with a partial blockage at its inlet. *Numerical Heat Transfer, Part A: Applications*, *73*(11), 743-767.
17. Hellström, L. H., Zlatinov, M. B., Cao, G., & Smits, A. J. (2013). Turbulent pipe flow downstream of a $90 {{}^\circ} $ bend. *Journal of Fluid Mechanics*, *735*.
18. Kim, J., Yadav, M., & Kim, S. (2014). Characteristics of secondary flow induced by 90-degree elbow in turbulent–pipe flow. *Engineering Applications of Computational-Fluid Mechanics*, *8*(2), 229-239.
19. Muzychka, Y. S., & Yovanovich, M. (1998, June). Modeling friction factors in non-circular ducts for developing laminar flow. In *2nd AIAA, Theoretical Fluid Mechanics Meeting* (p. 2492).
20. Muzychka, Y. S., & Yovanovich, M. M. (2009). Pressure drop in laminar developing flow in noncircular ducts: A scaling and modeling approach. *Journal of Fluids Engineering*, *131*(11), 111105.
21. Shah R.K.(1978). A correlation for laminar hydrodynamic entry length solutions for circular and noncircular ducts. *Journal of Fluids Engineering*, 1978;100(2), 177-179.
22. Benbella S., Al-Shannag M., & Al-Anber Z.A. (2009) Gas–liquid pressure drop in vertical internally wavy 90 bend. *Experimental Thermal and Fluid Science*,2009; 33(2), 340-347.
23. Ramgadia A.G., & Saha A.K.(2013). Three-dimensional numerical study of turbulent flow and heat transfer in a wavy-walled duct. *International Journal of Heat and Mass Transfer*,2013; 67, 98-117.
24. Xiong R., & Chung J.N.(2008). Effects of miter bend on pressure drop and flow structure in micro-fluidic channels. International Journal of Heat and Mass Transfer, 2008; 51(11-12), 2914-2924.
25. Choi C.W., Yu.D.I., & Kim M.H.(2011). Adiabatic two-phase flow in rectangular microchannels with different aspect ratios: Part I–Flow pattern, pressure drop and void fraction. International Journal of Heat and Mass Transfer,2011; 54(1-3), 616-624.
26. Poonaya S., Teeboonma U., & Thinvongpituk C.(2009). Plastic collapse analysis of thin-walled circular tubes subjected to bending. *Thin-walled structures*,2009; 47(6-7), 637-645.
27. Poonaya S., Thinvongpituk C., & Teeboonma U.(2007). An analysis of collapse mechanism of thin-walled circular tubes subjected to bending. *International Journal of Aerospace and Mechanical Engineering*, 2007.
28. Menter F.R. (1994) Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, 1994; 32(8), 1598-1605.

AUTHORS PROFILE

Mohit Kumar is currently a Research scholar in N.I.T. Kurukshetra, Haryana, India. He is working as Manager in R&D department of BEL, Ghaziabad, A Govt of India, Ministry of Defence Entp. He had passed his M.Tech degree in Mechanical Engg from NIT Kurukshetra, Haryana. His areas of interest are:

Current interest: Design and thermal analysis for air as well as liquid cooled electronics enclosures for Radars and Satellite communication equipment.

Research Activities: Liquid cooling for large size antennas for Radar applications. Thermal management of Satellite communication equipment.

Future plans: Wind load analysis for reflector type antenna systems and composite Radomes, using coupled analysis of CFD and FEA. Conversion of liquid cooled systems to air cooled systems.

Dr VK Bajpai is currently working as Professor in National Institute of Technology, Kurukshetra, Haryana, 136119 India. He is Ph.D In Thermal Engineering. His Area of Interest are:

Current interest: Renewable energy resources, Heat Transfer, Refrigeration and air conditioning, Thermodynamics, IC Engines and gas turbines, Fluid Engineering, Steam and Power generations.

Research Activities: Use of solar energy in HVAC is being explored. Regeneration of Desiccant wheel by solar energy and optimization of its different parameters.

Future plans: Planning to have a project on solar energy from MNRE and planning to develop Centre of excellence in Renewable Energy Sources in the Institute.