CHEMICAL ENGINEERING TRANSACTIONS
VOL. 74, 2019
A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.cetjournal.it
Guest Editors: Sauro Pierucci, Jiří Jaromír Klemeš, Laura Piazza

ISBN 978-88-95608-71-6; ISSN 2283-9216
DOI: 10.3303/CET1974176

Head Loss Accessories Calculation through CFD. Does It Worth the Effort?

Catalina Valencia Peroni

Full time professor, Process and Energy Department, Universidad Nacional de Colombia- Sede Medellin
Cr 80 N 65-223 Facultad de Minas Bl M3-109
cavalenciapa@unal.edu.co

For several years, head loss calculation of simple piping accessories like tees and elbows was performed through empirical coefficients of well-known equations like Hooper 2 K equation (1981) or Darby's 3 k equation developed later. As more powerful computers are available nowadays it is reasonable to think that more accurate results could be obtained by using physical phenomena equations like Navier-Stokes. However, Navier-Stokes is difficult to solve in an analytical way for geometries more complex than a sphere or an infinite plane. Time demanding numerical calculation of those equations is needed for simple geometries like elbows and tees.
Computational fluid dynamics (CFD) solves numerically Navier-Stokes. Software that implements CFD has usually a high step in the learning curve, even more for a pregrad chemical engineering student that is not familiar with 3D Cad modelling.
In this work two different commercial software results i.e. Fluent Ansys and Solid Works Flow Simulation are compared with traditional Hooper's 2K calculation for head loss on elbows and tees through 117 CFD simulations of chemical, biological and mechanical engineering students.
CFD results follow the experimental data contained in Hooper paper but are numerically different of those obtained through Hooper equation. Some possible explanation for this is discussed here. Also, CFD software learning curve for the students categories is analyzed concluding that for a fresh chemical engineer CFD is preferable for complex accessories where no experimental data is available although if you can get over the first step on the learning curve you become the owner of a powerful tool.

1. Introduction

Since 1981 Hooper equation for head loses calculation has become popular. The availability of calculators has increased the engineer predilection for an empirical equation calculation over a diagram or experimental curve.
The experimental solution that is, using a real pump and the piping accessory on a piping system is both time and cost demanding.
Hooper equation is transcribed below. Correlation coefficients are presented also in Table 1.

$$K = \frac{K_1}{N_{Re}} + K_\infty \left(1 + \frac{1}{ID}\right)$$

Where

$K_1 = K$ for the fitting at $N_{Re} = 1$

$K_\infty = K$ for a large fitting at $N_{Re} = \infty$

ID is the internal diameter of attached pipe in inches.

Paper Received: 30 May 2018; Revised: 5 October 2018; Accepted: 9 February 2019

Please cite this article as: Valencia Peroni C., 2019, Head Loss Accessories Calculation Through CFD ¿does It Worth the Effort?, Chemical Engineering Transactions, 74, 1051-1056 DOI:10.3303/CET1974176

On the opposite for complex geometries not listed on Hooper table one should either built the experimental set-up or nowadays solve numerically Navier-Stokes equations with the help of computers. Is on the engineer decision the accuracy needed on the results . Computer time demand is higher for more accurate results. Some methodologies for CFD implementation have been proposed to reduce computer time or complexity and to increase results accuracy.

Table 1: Hooper 1981 K values for different accessories

		Fitting type			K1	Kinfinite
Elbows	90°	Standard (R/D=1), screwed			800	0.4
		Standard (R/D=1), flanged, welded			800	0.25
		Long-radius (R/D=1.5), all types			800	0.2
		Mitered	1 Weld (90° angle)		1000	1.15
		Elbows	2 Weld (45° angle)		800	0.35
		R/D=1.5	3 Weld (30° angle)		800	0.3
			4 Weld (22.5° angle)		800	0.27
			5 Weld (18° angle)		800	0.25
	45°	Standard (R/D=1), all types			500	0.2
		Long-radius (R/D=1.5), all types			500	0.15
		Mitered, 1 Weld (45° angle)			500	0.25
		Mitered, 2 Weld (22.5° angle)			500	0.15
	180°	Standard (R/D=1), screwed			1000	0.6
		Standard (R/D=1), flanged, welded			1000	0.35
		Long-radius (R/D=1.5), all types			1000	0.3
Tees	Used as Elbow	Standard, screwed			500	0.7
		Long-radius, screwed			800	0.4
		Standard, flanged or welded			800	0.8
		Stub-in-type brachn			1000	1
	Run-through tee	Screwed			200	0.1
		Flanged or welded			150	0.5
		Stub-in-type branch			100	0
Valves	Gate, ball, plug	Full line size, beta=1.0			300	0.1
		Reduced trim, beta=0.9			500	0.15
		Reduced trim, beta=0.8			1000	0.25
	Globe Estándar			All	1500	4
	Globe angle or Y-type			values for	1000	2
	Diaphragm, dam type			100%	1000	2
	Butterfly			Open	800	0.25
	Check	Lift			2000	10
		Swing			1500	1.5
		Tilting-disk			1000	0.5

Moreover, CFD software has a high step in the learning curve due to the complexity of solving numerically Navier-Stokes partial equations for a given complex geometry. i.e. a plane or a car, or for a chemical engineer, a piping system or a chemical reactor.
In this work fluid dynamics curse of Universidad Nacional de Colombia is used as a test bench to analyse whether numerical analysis of simple geometries like pipe accessories worth the effort of learning how to do it. Students who are newly introduced to CFD are presented with a homework of head loses calculation. Two different commercial software used for CFD calculations are also compared. The results obtained are compared against Hooper method.

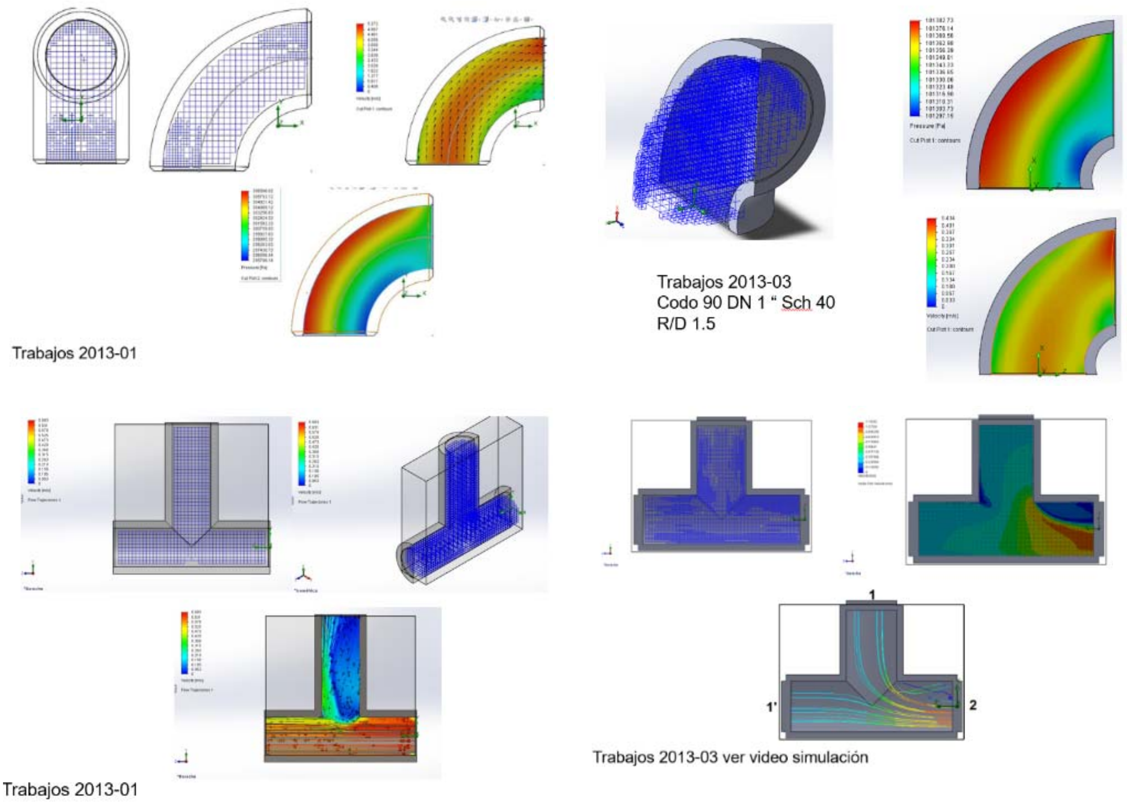

Figure 1: CFD velocity and pressure fields results of different fluid dynamics under graduate students

2. Methodology

As described by Valencia Peroni 2018, Navier-stokes equations are presented in an attractive way to third year fluid dynamics students in a two hours lesson. CFD is used to catch the attention of students and to introduce a modern way of solving Navier-Stokes equations. For chemical, mechanical and biological engineering students of Universidad Nacional de Colombia-Sede Medellin, Navier -Stokes is first introduced in the transport phenomena course. Appealing aeronautical applications are used. i.e new RedBull design of F1 car or Nasa drawings of the CFD results for the space shuttle.
CFD three main steps are described to students, that is, mesh generation, selection of boundary conditions and briefly, some methodologies used to solve the discretized Navier-Stoles are explained. Some CFD software is also described. About a third part of the students of the fluid dynamics course are mechanical engineering students, who are familiar with Solid Works (a 3D developer software). Solid Works implements CFD on a toolbox. ANSYS license is also available at the university and was also used in the course.
To obtain the results presented in the following section a group homework is proposed to the students. It includes the selection of a given pipe accessory (elbow), simulation with CFD software to find head looses and the comparison with the traditional Hooper 2k method.

Multidisciplinary groups are encouraged and some previous years results are also given. For the ANSYS problems the mesh is also provided. A graduate student is also available for questions regarding the use of the software.
Each semester 10 to 16 groups of three students each solve each one a different CFD problem. Either Reynolds number, size or type of accessory changes. This methodology was used from 2013 to 2017 in mixed groups of 50 students from biological, mechanical and chemical engineering on the fluid dynamics course.
Homework is rated according to the analysis of the results.

3. Results

Some mesh and CFD results of different courses are presented in Figure 1. Beautiful pictures or velocity and pressure fields are easily obtained by the students. For accuracy comparison against Hooper method is presented in the following figures. Please notice that sometimes Hooper table does not include some accessories coefficients like for instance when two different pipes are converging into a tee accessory.
For numerical comparison, each group result (a point) was plotted against Hooper curve.
On the next Figure 2 internal diameter plot against head loss coefficient (K) for elbows from 2 to 8 inches are presented. Different Reynolds numbers were used for calculations as neither Hooper article or experimental points specify this value. Please note that Hopper paper is quite old and not available in a digital format. Besides, CFD simulations require more information about the system than the data found on Hopper paper.

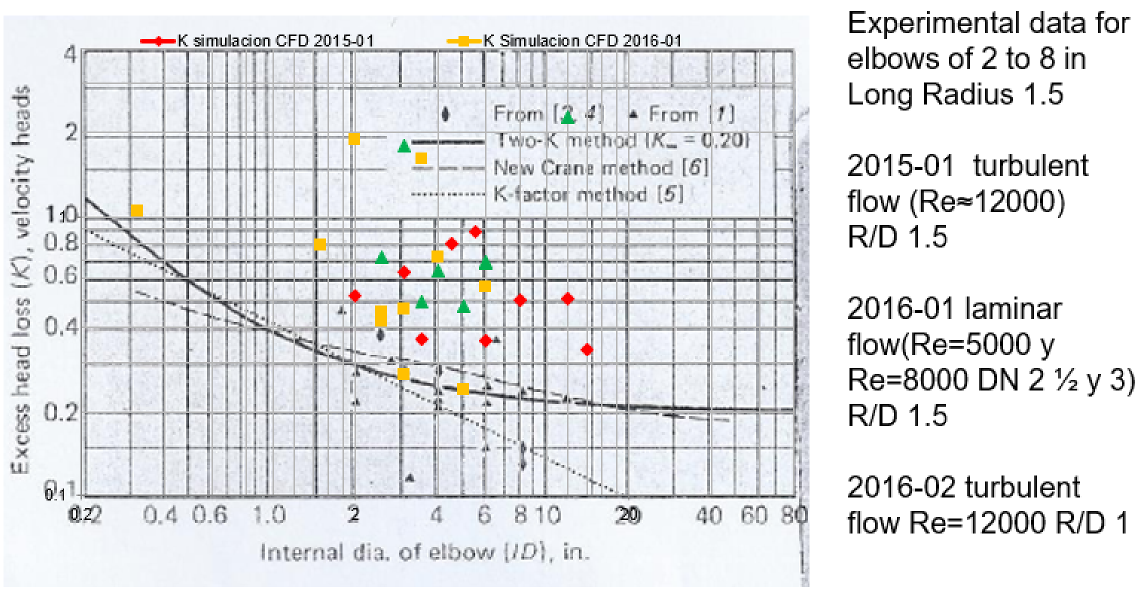

Figure 2: Internal diameter plot against head loss coefficient (K) . Hooper plot is used as a template for students results.

One can see from this figure that always CFD predicted K value is higher than Hooper predicted K.
Although CFD data seems sparse, it was found that CFD predicted values are more similar to experimental values. Little effort was made to perform this calculations. Students take just few weeks to obtain this results. Maybe more accuracy could be obtained if some CFD specialized software is employed, other than Solid Works.
Figure 3 presents also the results obtained with Fluent ANSYS simulator. Elbow radius (R/D) changes also. Again K values are overestimated by the CFD software. Please noted that ANSYS simulation results are less sparse.
Finally, in Figure 4 Reynolds number is plotted against K coefficient. As one can see from this picture CFD prediction follows the 2K Hooper prediction for welded elbows.

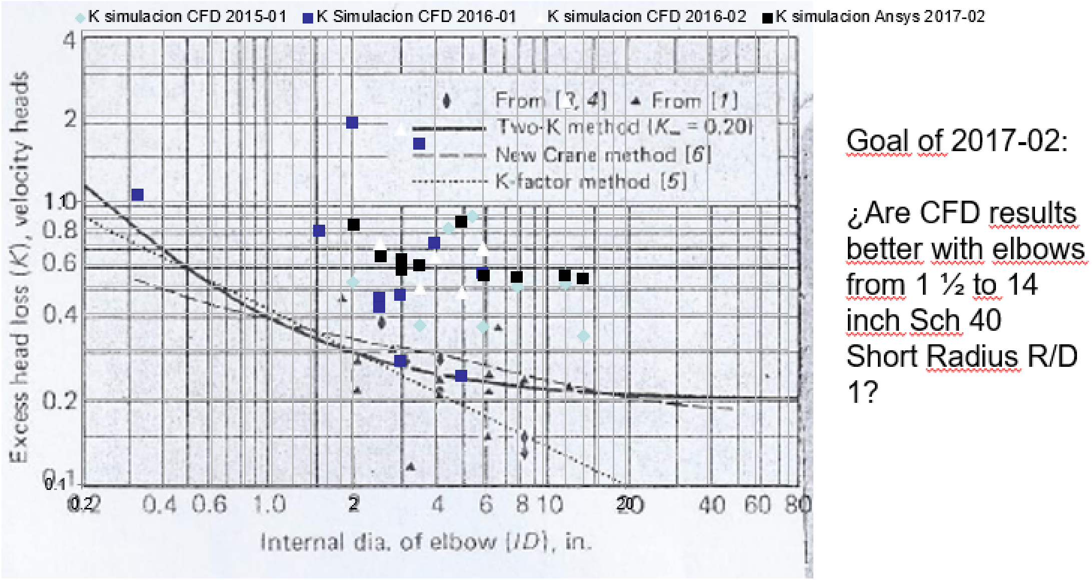

Figure 3: Internal diameter plot against head loss coefficient (K) . Hooper plot is used as a template for students results.

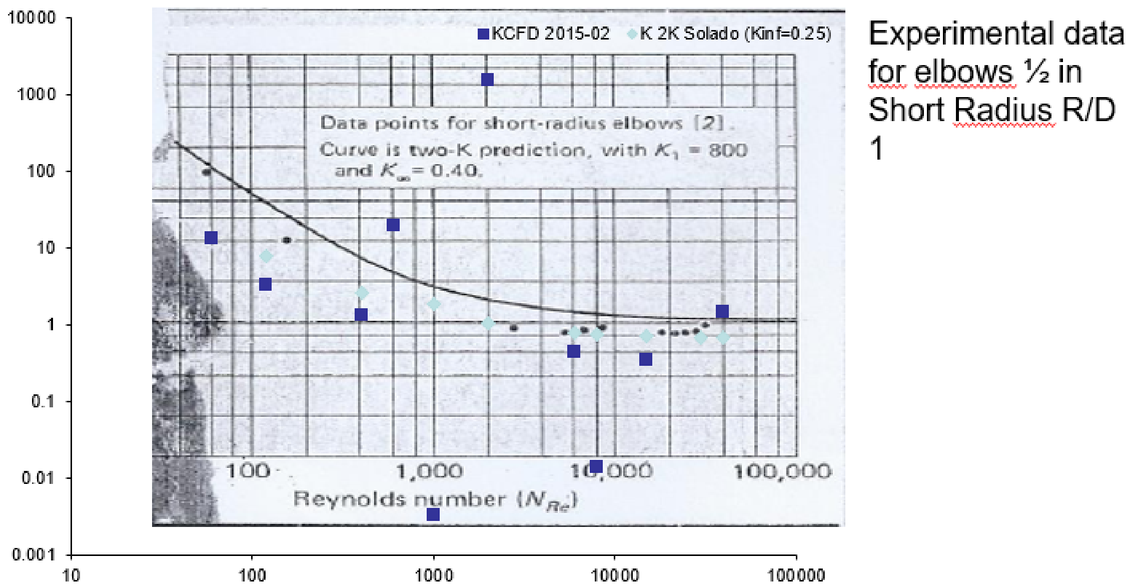

Figure 4: Reynolds number against head loss coefficient K.

As shown in the Figure 4, CFD results follow the experimental data contained in Hooper paper but are numerically different of those obtained through Hooper equation. Some possible explanation for this could be the reliability of the Hooper measure instrumentation (experimental data was taken in 1940) and, on the other hand, the accuracy of CFD calculations because Solid Works software is not intended for CFD.

Regarding the CFD software learning curve on students, two different CFD software were used. Solid Works, that was familiar to a third part of the students and is more user friendly, and ANSYS fluent package that for the feasibility of the homework mesh needed to be supplied to students. Also, a graduate student was permanently available for questions regarding the software. During the five-year period where the homework was proposed to students, a month was necessary for them to get familiar with the variables and graphic environment. 3D cad design should be provided for chemical engineering students. In conclusion CFD calculation effort is worth if one is familiar with 3D cad design or head loose is needed for a complex geometry and experimental set up is difficult to build.

4. Conclusions

CFD calculation is useful for engineers although its head loose results are numerically different from experimental data results, as presented in Hooper paper. Engineer should evaluate in advance whether the effort of building a 3D design and long calculation time is needed.
In conclusion CFD calculation effort is worth if one is familiar with 3D cad design or head loose is needed for a complex geometry and experimental set up is difficult to build. However, Hooper correlation is easier to calculate for simpler geometries.

References

Hooper W.B., 1981, The Two-K Method Predicts Head Losses in Pipe Fittings, Chemical Engineering (New York). 88. 96, 98-100.

Valencia Peroni C. 2018, Introducing Navier Stokes to chemical engineering fluid dynamics students, XXIX Oral presentation in the Interamerican Congress of Chemical Engineering Incorporating the 68th Canadian Chemical Engineering Conference, 28-31 October, Toronto Canada.