Proceedings of the Korean Nuclear Society Autumn Meeting
Yongpyong, Korea October 2003

Assessments of Crud Deposition Effect on Flow Field in Converging Flow Hole

KangHoon Kim, EungJun Park, HongJu Kim, SunTack Hwang, KyeongLak Jeon

KEPCO Nuclear Fuel Co., Ltd.
493 DeogJin-Dong YuSong-Gu, DaeJon, Korea

Abstract

Flow geometry with chamfered or converged at inlet and/or outlet is widely used in flow engineering not only for flow measurement through pressure drop (nozzle-type flow meters such as flow nozzle and Venturi) but also for the holed plate such as top/bottom nozzle of nuclear fuel assembly. Generally, there is benefit on pressure drop concerns with flow hole chamfered or converged than square edged. The deposition on the inside surface as well as its own nature will affect pressure drop characteristics of flow holes. The assessments for two kinds of mechanisms for the effect of deposition, geometrical distortion and perturbation of flow structure, were performed in this study for single Venturi type flow hole geometry. The result from the flow equation and the CFD analysis for geometrical distortion is well matched at throat region and frontier region of diffuser, but it is not for thick deposition at later region of diffuser. In the throat region, the pressure drop due to perturbation is higher than that due to geometrical distortion for thin deposition, but it is not for thick deposition. In the diffuser region, the pressure drop due to perturbation is much higher than that due to geometrical distortion regardless of thickness of deposition. Earlier flow separation leads to increase drag and prevents pressure recovering on diffuser region. This is the main reason of increasing pressure loss through flow element. The total effect is thought to be a superposition of two different mechanisms, even it is not arithmetical sum of the results obtained in this study. By summary, the deposition of flow nozzle chamfered/converged leads to increase pressure drop not only for inlet to throat region but also for whole flow element for a same flow rate. The relative

contribution of two different mechanism, geometrical distortion and flow structural perturbation, is 50 to 50 for the pressure drop at throat region with thin deposition. But the effect of flow structural perturbation is dominant factor for the pressure drop through flow element. For the obstruction flow instrument, increased pressure loss at throat region due to deposition may lead to higher indicated flow than actual for a flow measurement if this effect is not corrected. Loss of recovery of pressure drop at the diffuser region due to flow separation leads to adverse effect on main advantage of introducing chamfered flow holes.

1. Introduction

Flow geometry with chamfered or converged at inlet and/or outlet is widely used in flow engineering not only for flow measurement through pressure drop (nozzle-type flow meters such as flow nozzle and Venturi, so called Bernoulli flow meter) but also for the holed plate such as top/bottom nozzle of nuclear fuel assembly. Generally, there is benefit on pressure drop concerns with flow holes chamfered at inlet and/or outlet than non-chamfered holes with square edged at inlet and/or outlet as given in Figure 1 and Figure 2. Figure 1 is for loss coefficient of inlet region with various hole shapes and Figure 2 is for non-recoverable head loss in Bernoulli obstruction meters, respectively. The deposition on the inside surface as well as its own nature will affect pressure drop characteristics of flow hole, inferring from Figure 2.

The deposition inside surface of flow nozzle, so called fouling, is occurred due to precipitation of super-saturated solution. It is thought to be arising primary from the presence of metalic oxides/hydroxides or non-metalic compounds on the flowing fluid. The preferred location of this solid-layer deposition is converging section of flow element. The erosion of flowing surface and flow accelerated corrosion (FAC) may give additional source of fouling. The example of deposition on flowing surface is given in Figure 3.

It is thought that there are two kinds of mechanisms for the effect of deposition to increase pressure drop. The one is the effect of geometrical distortion and the other is the effect of the perturbation of flow structure. The former results the reduction of flow area due to thickness of deposition. The reduction of flow area on throat region may lead to change of characteristic coefficient, discharge coefficient: Cd, of flow element. The later results from the perturbation of boundary layer due to increase in roughness. In thermodynamic aspects, it leads to increase entropy, irreversible nature. The effect of first mechanism can be easily evaluated based on the relationship between pressure drop and flow rate for a region of interest. For the effect of the second mechanism, even though it was thought to have similar or greater contribution to whole effect, any assessment is not tried yet.

In this study, the effects of the geometrical distortion and the flow structural perturbation due to deposit on the surface of nozzle were investigated for all range of flow element with the computational fluid dynamic code. With the result, the adverse effect of deposition will be estimated for actual application.

2. Assessment based on Flow Equation

The relationship between flow rate and pressure drop for a nozzle type flow element can be derived by the Bernoulli theorem and the continuity equation. The relationship includes parameter related to its geometric characteristics and the pressure drop due to flow resistance as follows [1].

$$FWMF \propto d^2 \times \frac{1}{\sqrt{1-(d/D)^4}} \times C_d \times Y \times F_a \times \sqrt{\frac{FWF}{FWSV}} \quad (1)$$

where

FWMF	=	*Flow rate*
FWF	=	*Iindicated pressure drop*
FWSV	=	*Specific volume of fluid*
d	=	*Throat diameter of the flow element*
D	=	*Diameter at the inlet pressure tap*
C_d	=	*Discharge coefficient*
Y	=	*Expansion factor (= 1 for water)*
F_a	=	*Thermal expansion factor (CA + CB * FWT)*
FWT	=	*Fluid temperature*
CA,CB	=	*Constants*

Per reference 2, the effects of deposition on flow rate and/or pressure drop can be evaluated as decrease in flow area. Based on above relationship, the pressure drop (FWF) change due to deposition on constant flow rate (FWMF) condition can be derived;

$$\frac{(FWF)_{affected}}{(FWF)_{initial}} = \frac{(d^2 \cdot C_d)^2_{initial}}{(d^2 \cdot C_d)^2_{affected}} \times \frac{[1-\beta^4]_{affected}}{[1-\beta^4]_{initial}} \quad (2)$$

The flowrate (FWMF) change with constant pressure drop (FWF) due to deposition can be derived;

$$\frac{(FWMF)_{affected}}{(FWMF)_{initial}} = \frac{(d^2 \cdot C_d)_{affected}}{(d^2 \cdot C_d)_{initial}} \times \frac{\sqrt{[1-\beta^4]_{initial}}}{\sqrt{[1-\beta^4]_{affected}}} \qquad (3)$$

Figure 4 shows the results with above relationship for a Venturi type with assumption of constant discharge coefficient (Cd) for the limiting condition, deposition on throat region only. The effects of deposition are increased as its thickness is increased. The deposition thickness of 1% of throat radius lead to 4% increase in pressure drop with same flow rate, or 2% decrease in flowrate with same pressure drop. The difference between result from constant Cd and that from the Cd with function of d/D and/or Reynolds number is negligible (Reference 2).

3. Numerical Assessment

3.1 Motivation

The effect of increased roughness on friction loss can be quantified with Moody diagram for circular pipe [1]. For a complete turbulence zone, the friction loss coefficient is a function of relative roughness only. Per the Figure 2, even though the inlet geometry is the same for both flow nozzle and Venturi, there is a large difference in head loss between them. The difference is existed even for between Venturies with different diffuser cone angle. Those implied that there is something related downstream flow structure as well as geometry itself.

3.2 CFD Model

The proposed CFD model is typical Venturi flow meter. Whole flow element of Venturi is consisted of inlet pipe, converging section, throat region and diffuser plate. If the diffuser plate is removed, then it is for flow nozzle. The results without removal of diffuser, it is for chamfered flow hole at inlet and outlet or Venturi flow meter.

Following assumptions were applied to setup the model.

- steady-state, incompressible turbulent flow
- 2-dimensional axi-symmetric
- fully developed turbulent velocity distribution at inlet

- inlet Reynolds number of $1.5X10^7$

As a closer relationship, standard k-e and the wall function are applied.

Uniform deposition only in throat region for both the flow direction and the azimuthal direction is assumed. For geometrical distortion model (thickness model), deposition acts like additional thickness of inner surface of throat region and lead to decrease in flow area. For perturbation model of flow structure (roughness model), deposition acts like purely rough component of inner surface of throat region while RMS flow area is retained.

3.3 Benchmarking the CFD Model

Before the assessment, simple benchmarking tests have been performed for the circular pipe. As a baseline, this is to check general results of FLUENT [3] with respect to two different mechanisms of deposition on flow.

The first test is for the relationship of friction loss coefficient and Reynolds number for smooth pipe. Prandtl's equation is selected as a reference.

$$\frac{1}{f^{1/2}} = 2 \cdot \log\left(\mathrm{Re} \cdot f^{1/2}\right) - 0.8 \qquad (4)$$

Figure 5 shows the results from both methods. Even though there is a small difference in absolute value, the trend is well matched.

The second test is for the effect of surface roughness. For fully developed flow, the relationship between friction loss coefficient and relative roughness at the same Reynolds number is

$$\frac{1}{f^{1/2}} = -2 \cdot \log\left(\frac{\varepsilon/d}{3.7}\right) \qquad (5)$$

The predicted friction loss coefficient is increase as relative roughness increase for both Eq. (5) and FLUENT as shown in Figure 6.

Based on above tests, it is concluded that the model and Fluent is expected to work properly for this assessment of two different mechanisms separately. The flow field result within the smooth Venturi is given in Figure 7.

3.4 Results from Thickness Model (Geometrical Distortion)

As described on section 3.2, deposition is assumed that acts like additional thickness of inner surface of throat region and lead to reduction of flow area. Pressure behavior through flow element for various thickness of deposition is given in Figure 8. As expected the pressure loss has its maximum at throat region and recovered at diffuser region. The unrecoverable loss, defined as the difference between inlet region and region beyond the later of diffuser is almost same for all cases while there is apparent difference in pressure loss at throat region. The amount of pressure loss at throat region is increased with thickness of deposition on that region. There is no difference in pressure loss at the frontier region of diffuser for all thickness of deposition with respect to the case of no deposition, smooth.

For the flow velocity, its gradient at throat region is increased with thicker deposition as shown in Figure 9 for relative thickness of 0.0001, 0006, 0.001, 0.003 and 0.006 (from top to bottom). The distribution for far downstream of throat is similar for all thickness of deposition.

With specified inlet condition, the Reynolds number of throat region is higher than that of inlet region. Similar way, Reynolds number for narrower throat is higher than that for wider one. This is easily verified using the definition of Reynolds number and continuity. For the condition of same pressure and temperature, the ratio of Reynolds number is;

$$\mathrm{Re}_R = \frac{\mathrm{Re}_2}{\mathrm{Re}_1} = \frac{V_2\, D_2}{V_1\, D_1} \quad \leftarrow \quad \left(V_1 \cdot D_1^2 = V_2 \cdot D_2^2\right)$$
$$= {D_1}/{D_2} > 1.0 \quad (\because D_1 > D_2\,)$$

Even though the higher Reynolds number leads to the less friction factor as Figure 5 of benchmarking, the pressure loss is increased by the following relationships.

$$\Delta P \sim \left(K + f \cdot \frac{L}{D}\right) \times \frac{V^2}{2g_c}$$
$$\Delta P_R = \frac{\Delta P_2}{\Delta P_1} \sim \left({V_2}/{V_1}\right)^2 = \left({D_1^2}/{D_2^2}\right)^2 > {D_1}/{D_2} > 1.0$$

Thus, the pressure drop for narrower throat (i.e., case with thicker deposition) is higher with power of 4 of the flowing diameter ratio.

3.5 Results from Roughness Model (Perturbation of Flow Structure)

Deposition in this model acts like purely rough component of inner surface of throat region with retaining RMS flow area. Pressure behavior through flow element for various roughness is given in Figure 10. The unrecoverable loss and the pressure loss in throat region are increased as roughness increase. The difference is more apparent for later part of diffuser. The pressure loss at the frontier region of diffuser is deviated with respect to the case of smooth.

For the flow velocity, its gradient is increased with roughness increasing not only for throat region but also for frontier region of diffuser as shown in Figure 11 for relative roughness of 0.0001, 0006, 0.001, 0.003 and 0.006 (from top to bottom). The distribution for far downstream of throat is deviated as roughness increase. The blue zone, zone with zero flow velocity, moves forward as increased roughness. This mechanism of flow separation leads to increased pressure loss through flow element with respect to smooth case.

4. Evaluation of Results

The result from flow equation and CFD for geometrical distortion is well matched at throat region (as given in Figure 12) and frontier region of diffuser (as given in Figure 13), but it is not for thick deposition at later region of diffuser (Figure 14). Figures 12, 13 and 14 also give comparison of calculated results between thickness model (geometrical distortion) and roughness model (perturbation of flow structure). In the throat region, the pressure drop due to perturbation is higher than that due to geometrical distortion for thin deposition, but it is not for thick deposition. In the diffuser region, the pressure drop due to perturbation is much higher than that due to geometrical distortion regardless of thickness of deposition. Earlier flow separation increases drag and prevents pressure recovering on diffuser region. This is the main reason of increasing pressure loss through flow element. The total effect is thought to be a superposition of two different mechanisms, even it is not arithmetical summation of the results obtained in this study.

By summary, the deposition of flow nozzle chamfered at inlet and/or outlet lead to increase pressure drop not only for inlet to throat region but also for whole flow element for a same flowrate. The relative contribution of two different mechanism, geometrical distortion and flow structural perturbation, is 50 to 50 for the pressure drop at throat region with thin deposition. But the effect of flow structural perturbation is dominant factor for the pressure drop through flow element.

5. Application

Even though flow hole geometry considered in this study is not exactly the same for actual design, the results are bounding case to maximize its effect on pressure drop. Thus the results of this study can be applied to the following estimations.

- The advantage of pressure drop with chamfered flow holes is evaluated by 10% to 25% of inlet region or, 20% to 50% of component depending on type of chamfer. The design with larger advantage will have larger adverse effect due to deposition. The maximum adverse effect for most favorable design is less than 25% for the component, and leads to less than 2% increase in fuel assembly pressure drop with respect to that with clean unaffected holes.

- The indicated flowrate or pressure drop of obstruction flow meter will have bias with deposition. Deposition of 0.5% of throat radius may lead to upto 4% increase in pressure drop indication at the same flowrate, or 2% decrease in flowrate at the same pressure drop indicated. For this condition, the performance capability may be monitored falsely and the effect can be worse as more material deposited.

6. References

[1] White, F.M, “Fluid Mechanics, 2nd ed.,” McGRAW-HILL, 1986.
[2] KangHoon Kim, et.al., “Study on the Effects of the Deposition to the Venturi Flow Measurements,” Proc. KNS Conference, Spring 2002.
[3] FLUENT User’s Manual, Fluent Inc., 2001

Acknowledgement

This research is a part of “Collaboration on the Advanced 17x17 Next Generation Fuel Assembly Development Program” sponsored by a grant of MOST. We would like to appreciate those related MOST personnel and “NGF Team”. Authors also give thanks to Y.N. Kim for his advices.

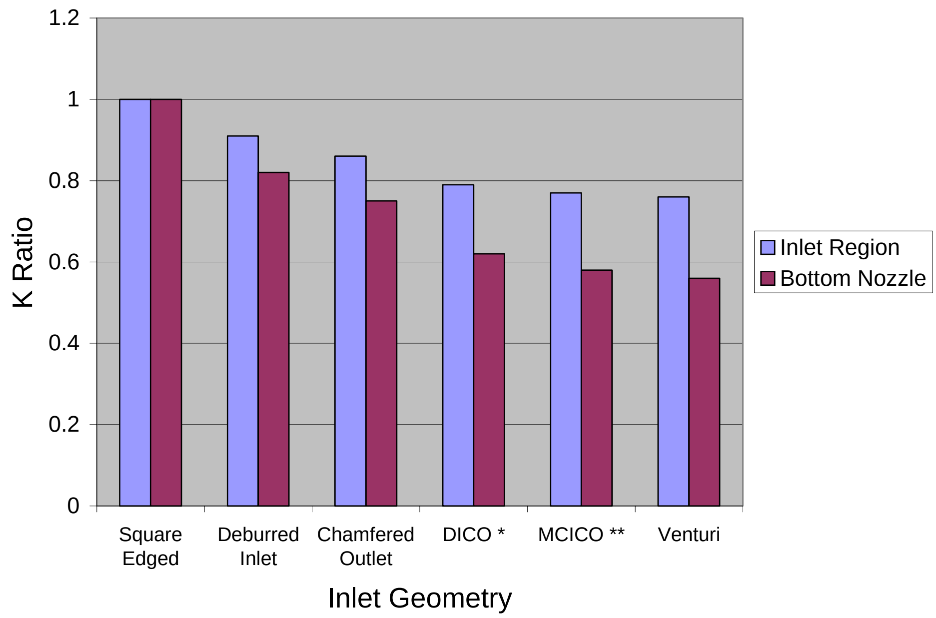

Figure 1 Loss Coefficient of Inlet Region with Various Flow Hole Shapes (w/o explicit scale)

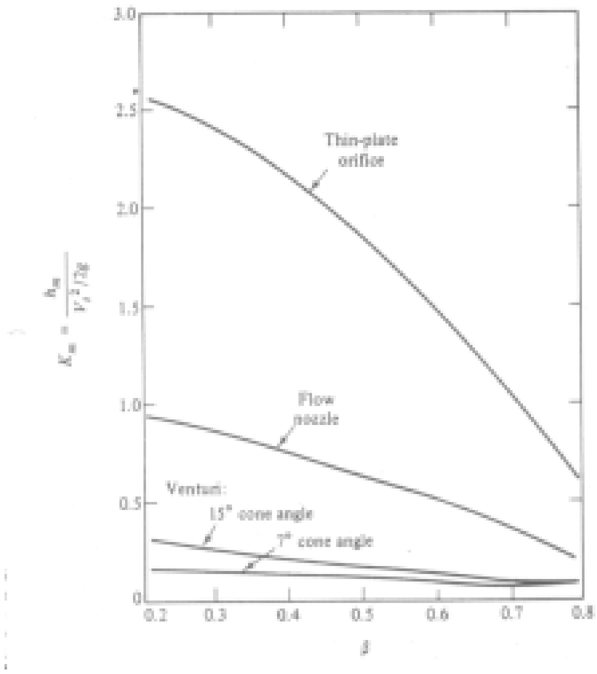

Figure 2 Nonrecoverable Head Loss in Bernoulli Obstruction Meter

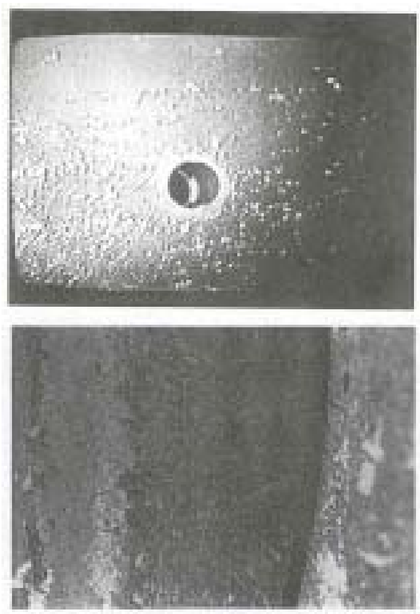

Figure 3 Example of Deposition on Flowing Surface

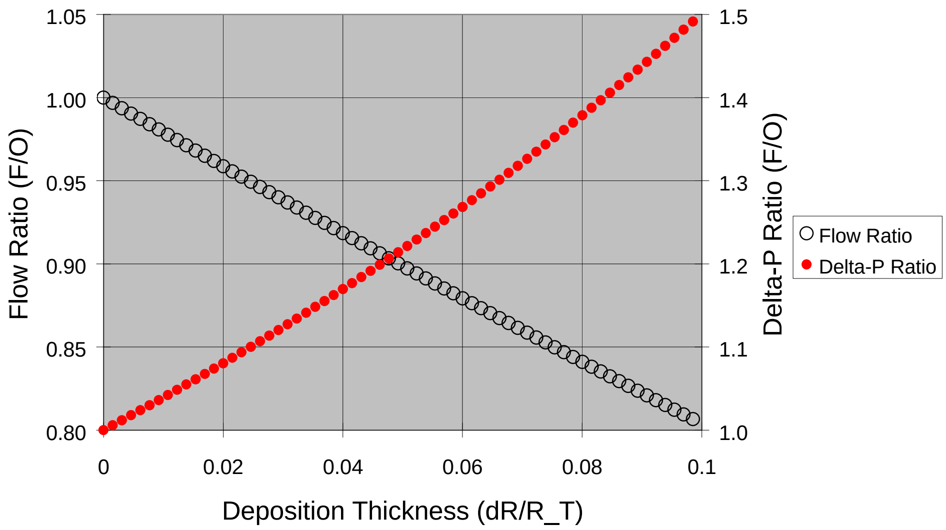

Figure 4 Result for Geometrical Distortion from Flow Relationship

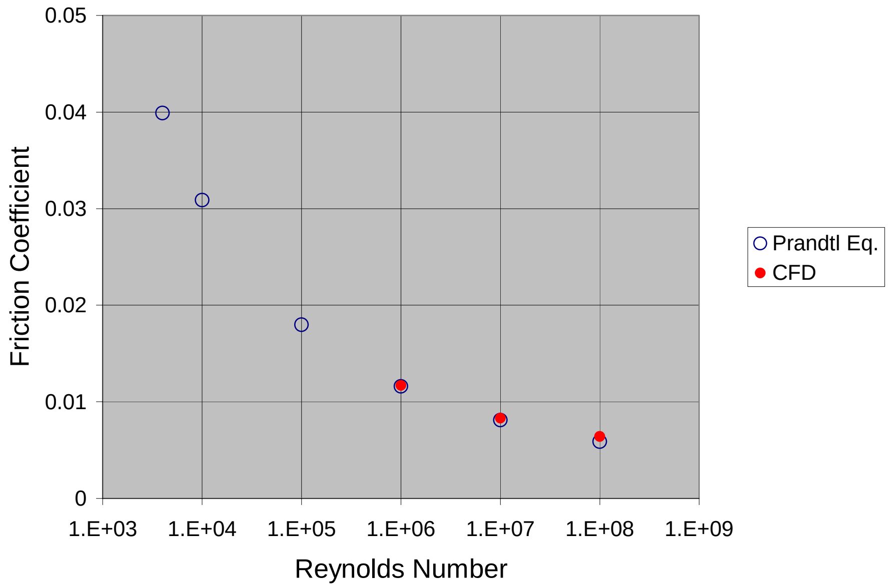

Figure 5 Benchmarking Results for Friction Coefficient vs. Reynolds Number

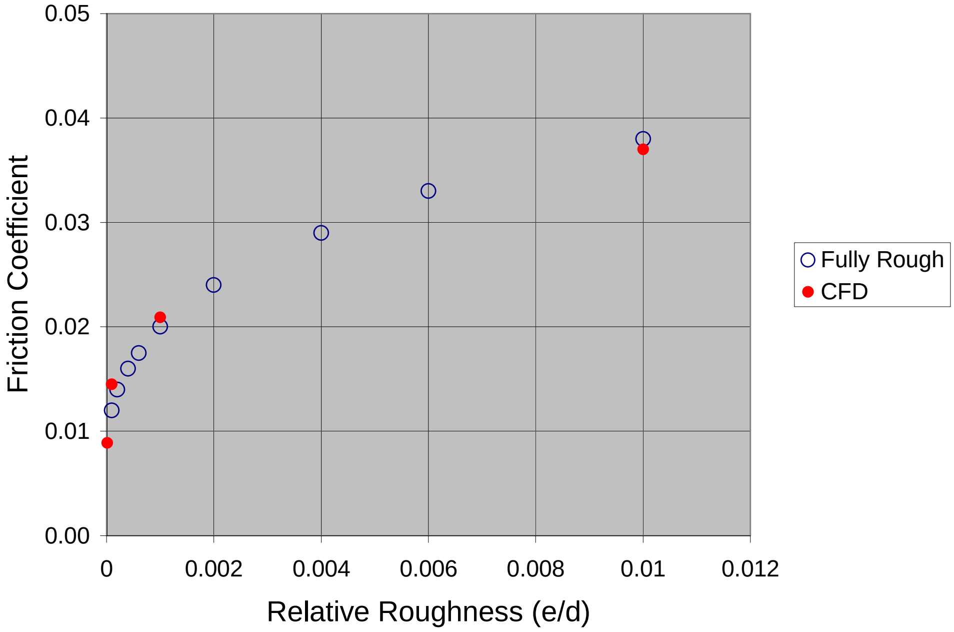

Figure 6 Benchmarking Results for Friction Coefficient vs. Relative Roughness

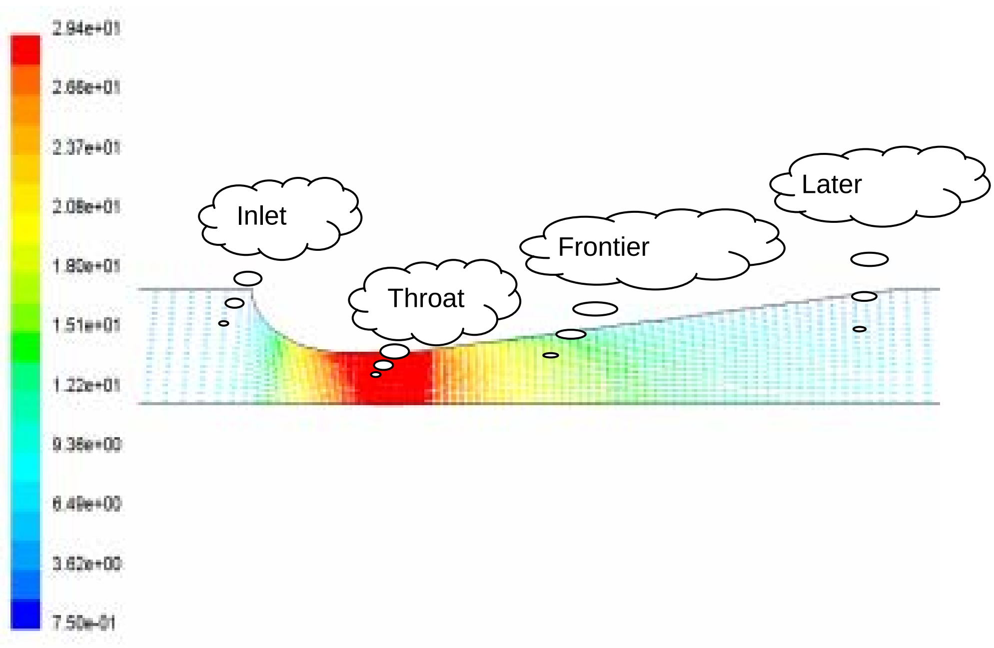

Figure 7   Flow Vector for Venturi with Smooth Surface

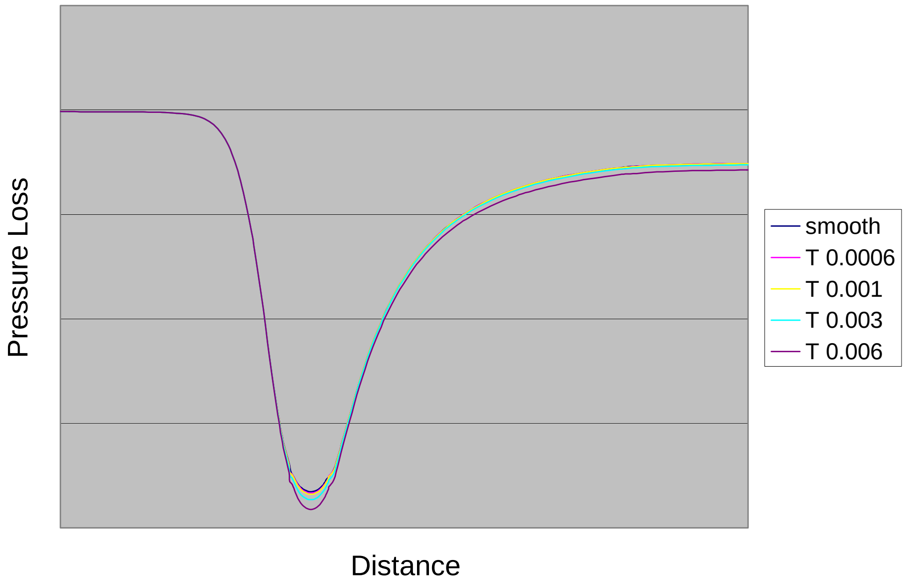

Figure 8　Pressure Change through Venturi (Thickness Model)

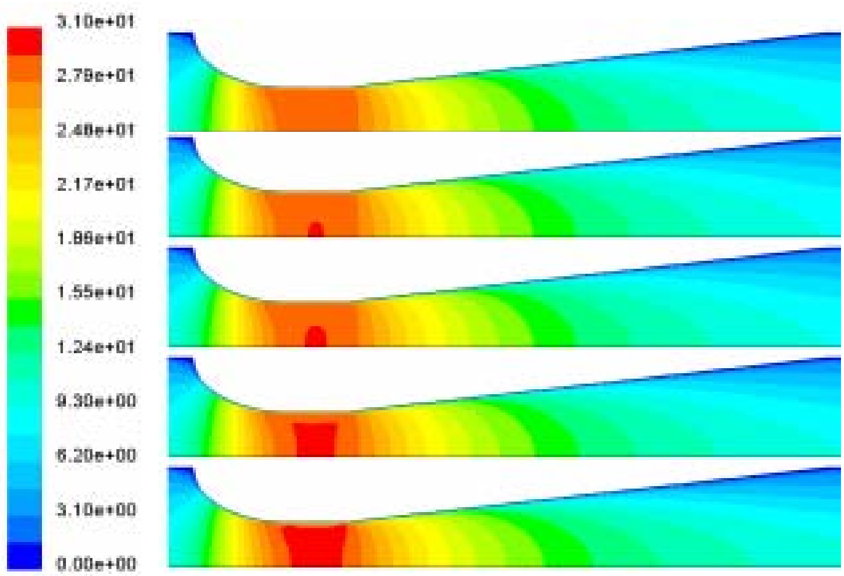

Figure 9　Velocity Distribution through Venturi

(Thickness Model, T = 0.0001, 0.0006, 0.001, 0.003, 0.006)

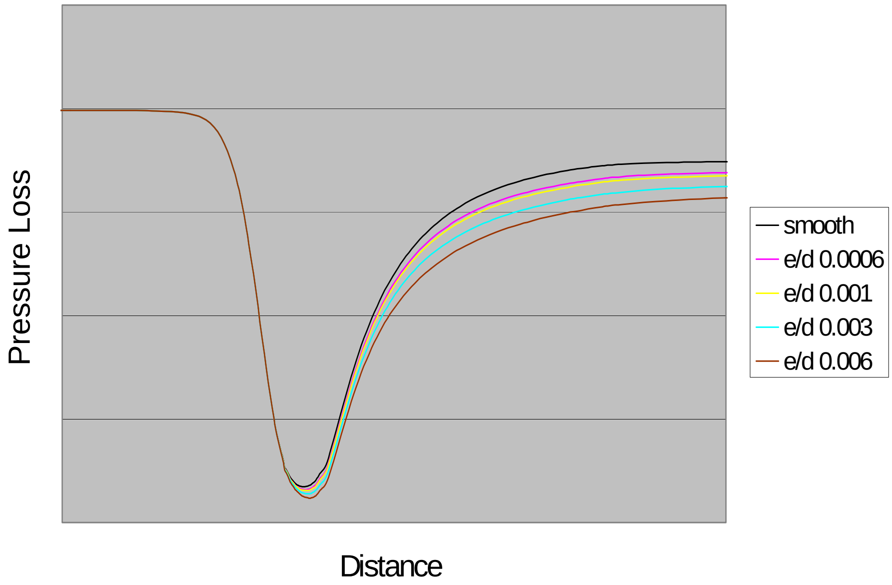

Figure 10 Pressure Change through Venturi (Roughness Model)

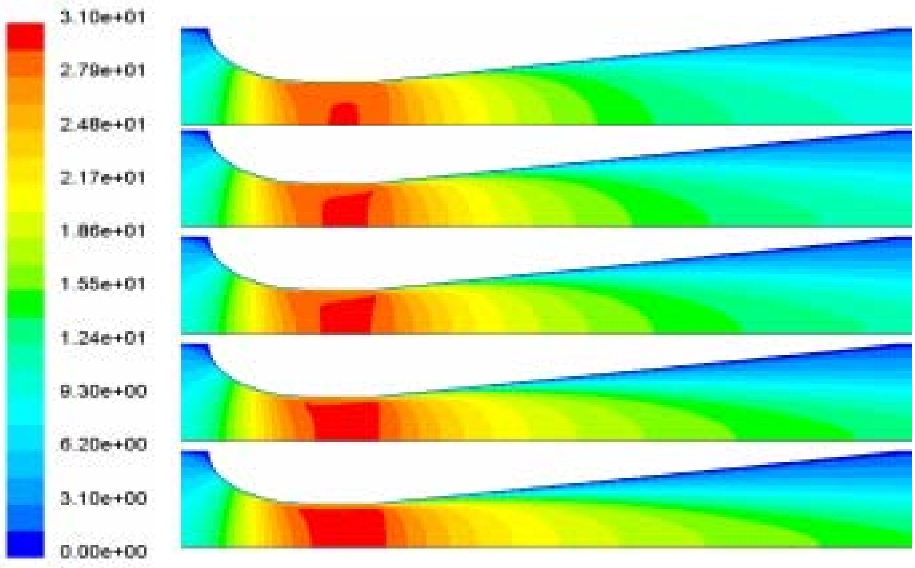

Figure 11 Velocity Distribution through Venturi

(Roughness Model, e/d = 0.0001, 0.0006, 0.001, 0.003, 0.006)

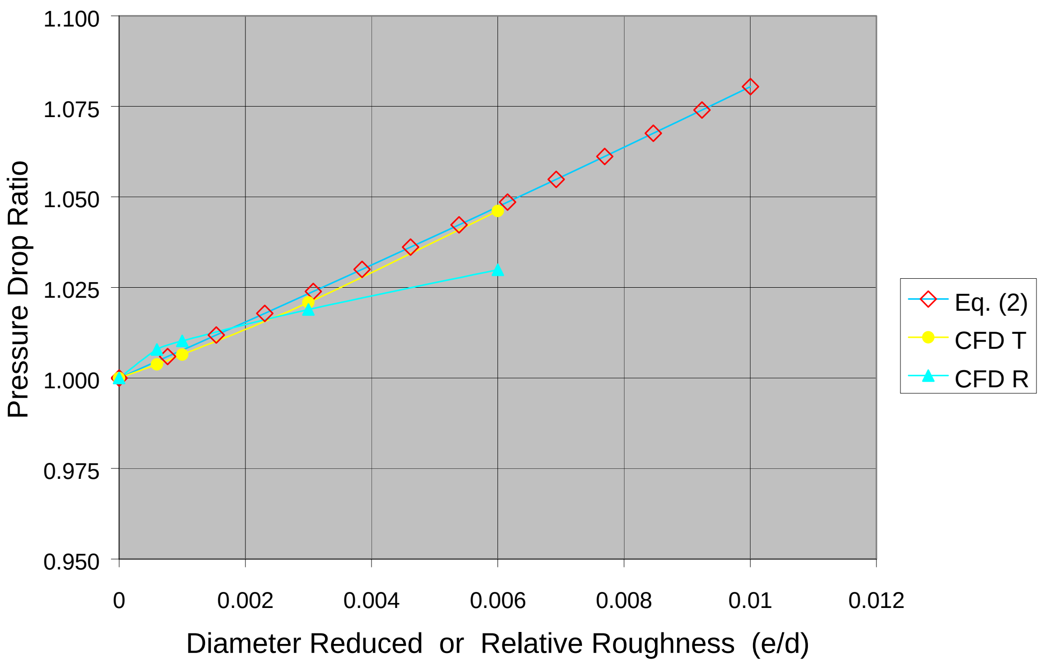

Figure 12 Comparison of Pressure Drop Ratio at Throat Region

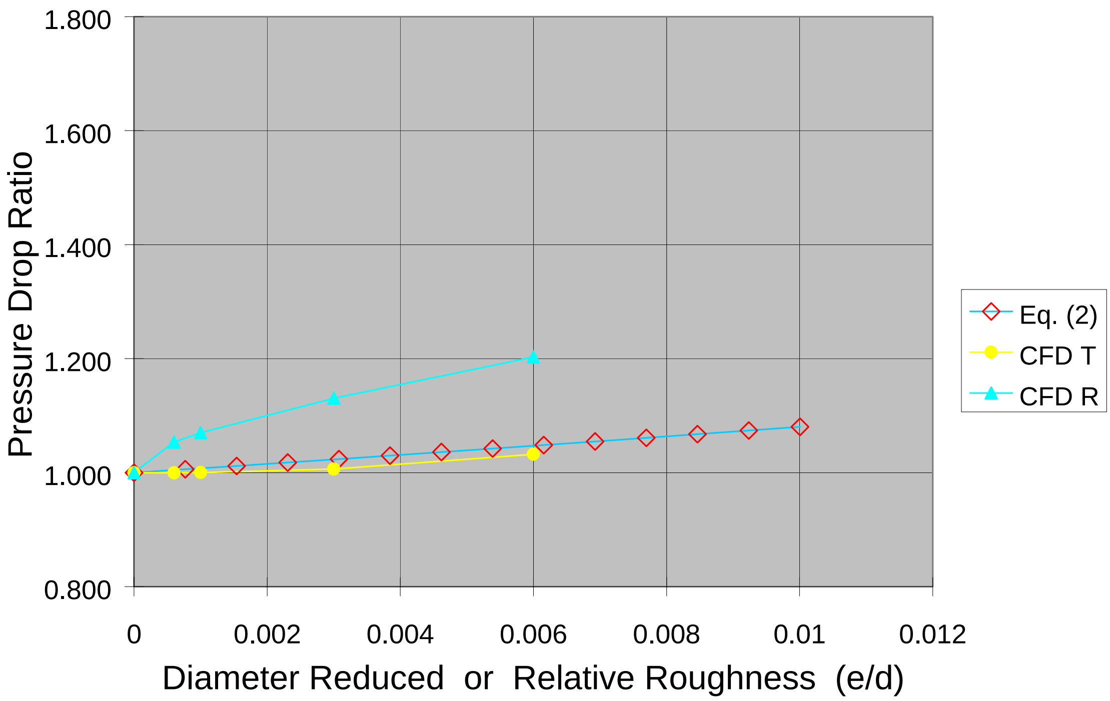

Figure 13 Comparison of Pressure Drop Ratio at Frontier Region of Diffuser

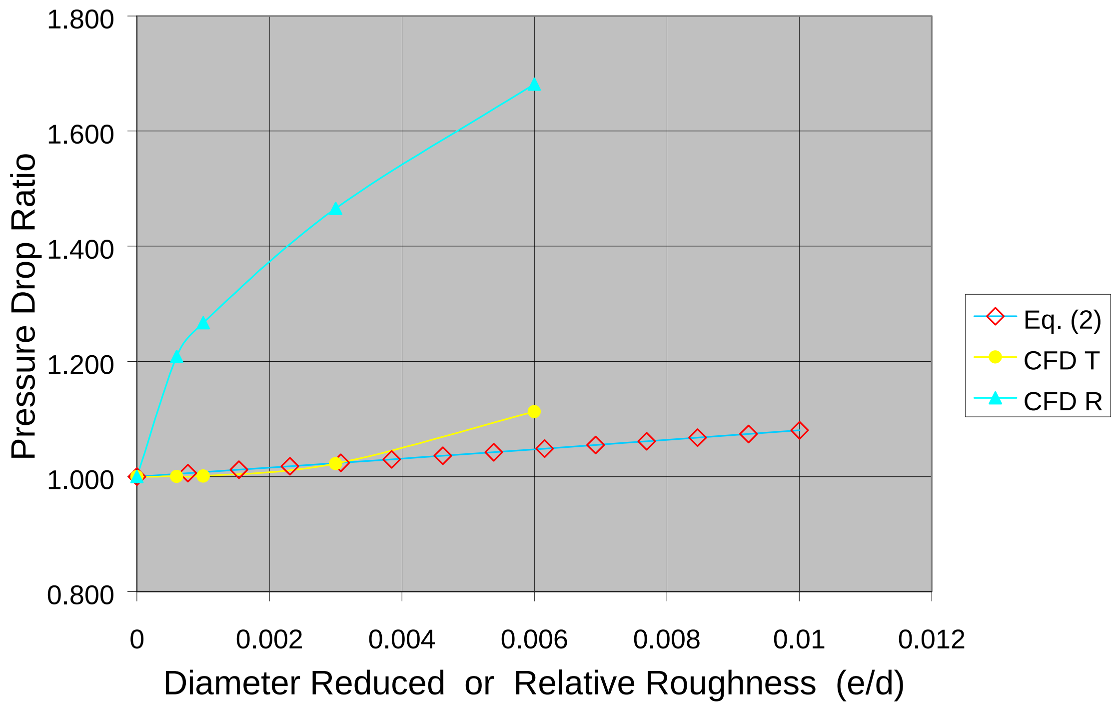

Figure 14 Comparison of Pressure Drop Ratio at Later Region of Diffuser