Proceedings of the Korean Nuclear Society Autumn Meeting
Yongpyong, Korea October 2002

An Analytical Evaluation for the Pressure Drop Characteristics of Bottom Nozzle Flow Holes

S.G. Yang, H.J. Kim, H.T. Lim, E.J. Park, K.L. Jeon
KEPCO Nuclear Fuel Co.

Abstract

An analytical evaluation for the bottom nozzle flow holes was performed to find a best design concept in terms of pressure drop. For this analysis, computational fluid dynamics (CFD), FLUENT 5.5, code was selected as an analytical evaluation tool. The applicability of CFD code was verified by benchmarking study with Vibration Investigation of Small-scale Test Assemblies (VISTA) test data in several flow conditions and typical flow hole shape. From this verification, the analytical data were benchmarked roughly within 17% to the VISTA test data. And, overall trend under various flow conditions looked very similar between both cases. Based on the evaluated results using CFD code, it is concluded that the deburring and multiple chamfer hole features at leading edge are the excellent design concept to decrease pressure drop across bottom nozzle plate. The deburring and multiple chamfer hole features at leading edge on the bottom nozzle plate have 12% and 17% pressure drop benefit against a single chamfer hole feature on the bottom nozzle plate, respectively. These design features are meaningful and applicable as a low pressure drop design concept of bottom nozzle for Pressurized Water Reactor (PWR) fuel assembly.

1. Background

Generally, a pressure drop of fuel assembly loaded in Pressurized Water Reactor (PWR) core is the one of critical design factors. In new fuel assembly developing project, a basic requirement of newly developing fuel assembly design is improving thermal/hydraulic performance of fuel assembly by adding new design features. These design features have high flow resistance characteristics, e.g. a high flow blocked mixing vane and intermediate flow mixer (IFM) grid. The high resistance fuel assembly affects to Reactor Coolant System (RCS) flow rate, hold-down spring force, and mixed core departure from nucleate boiling (DNB) performance, etc.. Therefore, the one of major design concepts of fuel assembly components is focusing on minimizing pressure drop.

Many activities have been done to search the best design concepts for bottom nozzle in a pressure drop point of view because bottom nozzle is a high resistance component. The bottom nozzle with large radius flow holes, slot type flow holes, chamfered flow holes, or Jedinstvo type flow holes have been already introduced in PWR fuel assembly. Even though these design features have a lot of pressure drop benefits, the development of bottom nozzle having additional pressure drop benefit is still needed to minimize pressure drop impact to the relative design area. Based on this necessity, the hydraulic test and analytical evaluation should be performed to select the best design concept among several bottom nozzle candidates. For the analytical evaluation, computational fluid dynamics (CFD) code was selected as a best way. The applicability and efficiency of CFD code is already well known in several commercial areas. As a specific tool, FLUENT 5.5[1] , developed by Fluent Inc., is applied to evaluate for pressure drop of bottom nozzle plate.

A purpose of this study is to find the best design feature of bottom nozzle flow hole using an analytical tool. For this analysis, the applicability of FLUENT 5.5 code was verified by benchmarking study with Vibration Investigation of Small-scale Test Assemblies (VISTA) test data in several flow conditions and typical flow hole shape. And then, the several kinds of hole were selected as a candidate and FLUENT 5.5 code was used to evaluate pressure drop of bottom nozzle plate as well.

2. Analytical Tool

2.1 Flow Hole Shapes of Bottom Nozzle Plate

The geometry of bottom nozzle plate including VISTA test housing shows in Figure 1. The bottom nozzle plate is 2.95 inches square and 0.560 inch thick. The dimension of flow housing is 3 inches square. Figure 2 presents a typical flow hole shape. The shape of bottom nozzle hole includes the 15° inlet chamfer, 10° outlet chamfer as well as 0.1925 inch radius features. For the deburring and multiple chamfer study, the geometry of flow hole is given in Figures 3 and 4, respectively.

2.2 Numerical Model

For the CFD analysis, the flow is assumed 3-dimensional, turbulent and incompressible single-phase water. The control volume is a 1/8-th bottom nozzle plate because the ratio of flow area is exactly the same as the full bottom nozzle plate. As a pre-processor, GAMBIT 1.3.0 [2] has been used to make 1/8-th bottom nozzle plate geometry and to create mesh of control volume. After meshing process, the quality of mesh was checked and the boundary zone was defined. For the numerical analysis, the mesh file was generated by GAMBIT option. Based on mesh file created by GAMBIT, FLUENT 5.5 code was run to get the pressure drop data across the bottom nozzle plate. The typical numerical models used in FLUENT 5.5 code are described in Table 1. These parameters were changed a little bit depending on the objective of study, e.g. benchmarking, deburring, multiple chamfer.

2.3 Nodal Resolution Study

The objective of nodal study is to check the sensitivity of solution in terms of control volume, mesh size, and convergence criteria. From the results of nodal study, the optimized model was set up for further study.

The first sensitivity parameter is inlet length. Based on the 1/8-th bottom nozzle plate with inlet and outlet chamfered holes, the inlet length below bottom nozzle plate was changed from 43.627" to 1.5". 43.627" inlet length is exactly same position of VISTA flow straightener. 2.625" inlet length is the position of VISTA P1 pressure transducer. However, outlet length was fixed as a 7" because the outlet has to have a proper length due to jetting flow of downstream. As an inlet velocity boundary condition, the uniform velocity distribution was used. The pressure outlet boundary condition was set as well. After FLUENT 5.5 code run with various inlet lengths, the result given in Figure 5 shows that pressure drop difference is similar for all cases, and the maximum difference is roughly 1.9%. Furthermore, it was realized that the pressure drop difference with 43.627" and 2.625" inlet length is very close to each other in case of using uniform velocity boundary condition. Also, the result of relative comparison between 8.69" (the position of VISTA P2 pressure transducer) and 7" outlet length shows that both cases are closely matched together.

The mesh size study for 1/8-th plate model having 2.625" inlet and 8.69" outlet length was done using FLUENT 5.5 code. Mesh size was changed from 0.02" to 0.01" by 0.002" step. Due to the mesh size decreasing, the total number of mesh giving on control volume was dramatically increased from 210,000 to 820,000. Figure 6 shows that the smaller mesh size gives low pressure drop difference. The maximum pressure drop difference is roughly 5.0%. Generally, a small size mesh is more accurate than a large one.

For the convergence criteria study, the same 1/8-th plate model with fine mesh size (0.01") was selected. The typical factors for convergence criteria are listed up in Table 1. There is no convergence criterion for energy equation because current bottom nozzle plate model doesn't have any heat source across the control volume. The variation of pressure drop difference was checked at 0.001, 0.0001, and 0.0004. Figure 7 shows that a maximum pressure drop difference is only 0.4%. It means that pressure drop difference within this model is not much dependent to the convergence criteria above 0.001.

Based on the three kinds of nodal study results, 1/8-th plate model with fine mesh size (0.01") and 0.0001 convergence criteria was chosen for further study.

3. Evaluation and Results

3.1 VISTA Benchmarking Study

VISTA test with various flow holes of bottom nozzle plate was completed. VISTA test data for benchmarking study were come from VISTA testing. A typical shape of flow hole for benchmarking study is given in Figure 2. This shape of flow hole is matching to the VISTA test plate number 8 described in Reference 2. From the nodal study results, 1/8-th plate model with fine mesh size (0.01") and 0.0001 convergence criteria was used for this study.

The first shot for the benchmarking study was starting using 1st order upwind scheme[3] . The results show that pressure drop difference is roughly 31% in high flow cases and 8% in the low flow cases, respectively. The next step for benchmarking study is 2nd order upwind scheme[3] option to get more accurate results than 1st order upwind scheme option. At the same flow rate, FLUENT 5.5 code was run using 2nd order upwind scheme. Since all runs were completed, only 3%~17% difference was shown in all flow ranges. It means that 2nd order upwind scheme has better accuracy than 1st order upwind in case of comparing to the VISTA test data. The detail results are given in Figure 8. The current model using 2nd order upwind scheme provides good validation of VISTA testing data to estimate the pressure drop difference of bottom nozzle plate. Figures 9 and 10 are the typical velocity contour about the chamfered hole and whole control volume, respectively.

3.2 Deburring Study

The detail dimensions for deburring at inlet and outlet chamfer location are described in Figure 3. The bottom nozzle plate model for this study is exactly the same as the VISTA benchmarking model. Four positions (A, B, C, and D) across the flow hole were deburred.

FLUENT code was run with same boundary condition. The results showed that deburring at inlet region is dominant compared to the deburring at outlet region in terms of pressure drop. Furthermore, deburring at the leading edge of inlet chamfer is most effective to get more pressure drop benefit. The combination effect having two and four deburring is good matching to the sum of each deburring effect. Based on the Table 2, the maximum pressure drop benefit is roughly 12% and 16% on the inlet deburring and combination deburring, respectively.

3.3 Multiple Chamfer Study

Figure 4 shows the detail dimensions for multiple chamfer at inlet location. The bottom nozzle plate model for this study is exactly same to the VISTA benchmarking model. The dimension of multi-chamfer was set up by the function of X, Y and depth.

FLUENT 5.5 code was run with same boundary condition. The results showed that X=0.008 in., Y=0.020 in., and depth=0.070 in. multi-chamfer is most effective a pressure drop point of view. Since this geometry may give break through problem between holes, X dimension of second chamfer was reduced to the 0.004 inches. Based on this geometry, X=0.004 in., Y=0.020 in., and depth=0.0555 in. multi-chamfer is best concept to get 17% pressure drop benefit described in Table 3.

4. Conclusion

Based on the evaluated results using CFD code, it is concluded that the deburring and multiple chamfer hole features at leading edge are the excellent design concept to decrease pressure drop across bottom nozzle plate. The deburring and multiple chamfer hole features at leading edge on the bottom nozzle plate have 12% and 17% pressure drop benefit against a single chamfer hole feature on the bottom nozzle plate, respectively. These design features are meaningful and applicable as a low pressure drop design concept of bottom nozzle for Pressurized Water Reactor (PWR) fuel assembly

5. References

[1] FLUENT User' s Manual, Fluent Inc., 2001
[2] GAMBIT User' s Manual, Fluent Inc., 2001
[3] S.V. Patankar, "Numerical Heat Transfer and Fluid Flow", University of Minnesota, 1980

감사의 글

본 연구는 과학기술부 원자력연구개발 중장기 계획사업인 웨스팅하우스형 개량핵연료 개발과제의 지원으로 수행되었으며 이에 감사드립니다.

Table 1 : Typical Numerical Model for FLUENT 5.5 Code

<table>
<tr><th>Model</th><th colspan="2">Description</th></tr>
<tr><td rowspan="2">Physical models</td><td>Solver</td><td>Solver = segregated, Formulation = implicit,
Space = 3-D, Time = Steady-state,
Velocity Formulation = absolute</td></tr>
<tr><td>Viscous model</td><td>Standard k-ε model
Standard wall function</td></tr>
<tr><td rowspan="3">Boundary condition</td><td>Inlet</td><td>Velocity-inlet
Velocity maginutude = 4.563 m/s
Turbulence intensity = 4%
Hydraulic diameter = 0.1511 in.</td></tr>
<tr><td>Outlet</td><td>Pressure-outlet
Gauge pressure = 0 psi
Turbulence intensity = 4%
Hydraulic diameter = 0.1511 in.</td></tr>
<tr><td>Wall</td><td>No heat generation
Wall roughness height = 0.0 in.
Wall roughness constant = 0.5</td></tr>
<tr><td rowspan="2">Solution Control</td><td>Under-relaxation factor</td><td>Pressure = 0.3, Momentum = 0.7,
Turbulence Kinetic Energy = 0.8,
Turbulence Dissipation Rate = 0.8,
Viscosity = 1, Density = 1, Body Force = 1</td></tr>
<tr><td>Discretization</td><td>Pressure = Standard,
Pressure-Velocity Coupling = SIMPLE
Momentum , Turbulence Kinetic Energy,
Turbulence Dissipation Rate = First Order Upwind
& Second Order Upwind</td></tr>
<tr><td>Convergence control</td><td>Residual (convergence criteria)</td><td>Continuity = 0.0001
X, Y, Z - Velocity = 0.0001
K, ε = 0.0001</td></tr>
</table>

Table 2 : Deburring Study Result

Deburring Position	Pressure Drop Difference – CFD (2nd Order)		
	DP (psi)	Difference (%)	Normalized Difference w/ VISTA Results (4.75/5.94)
No(baseline)	5.295	0.0	0.0
A	4.585	12.0	10.7
B	5.176	2.0	1.8
C	5.206	1.5	1.3
D	5.253	0.7	0.6
A + B	4.523	13.0	11.7
A + B + C + D	4.357	15.8	14.2

Table 3 : Multi-chamfer Study Result

Inlet Chamfer Type	X (in.)	Y (in.)	Depth (in.)	DP (psi)	Diff.(%)
Narrow single chamfer	0.000	0.007	0.070	5.295	0.0
Deburring	0.009	0.0087	0.070	4.585	13.4
Two chamfer	0.008	0.005	0.070	4.585	13.4
Two chamfer	0.008	0.010	0.070	4.498	15.1
Two chamfer	0.008	0.020	0.070	4.404	16.8
Two chamfer	0.008	0.030	0.070	4.433	16.3
Two chamfer	0.008	0.040	0.070	4.483	15.3
Two chamfer	0.008	0.050	0.070	4.518	14.7
Two chamfer	0.008	0.060	0.070	4.538	14.3
Wide single chamfer	0.008	0.070	0.070	4.606	13.0
Two chamfer	0.004	0.020	0.070	4.695	11.3
Two chamfer	0.002	0.020	0.070	4.939	6.7
Two chamfer	0.004	0.020	0.0555	4.580	13.5
Two chamfer	0.004	0.020	0.0760	4.592	13.3
Triple chamfer (at L. edge)	0.004	0.020	0.0555	4.651	12.2

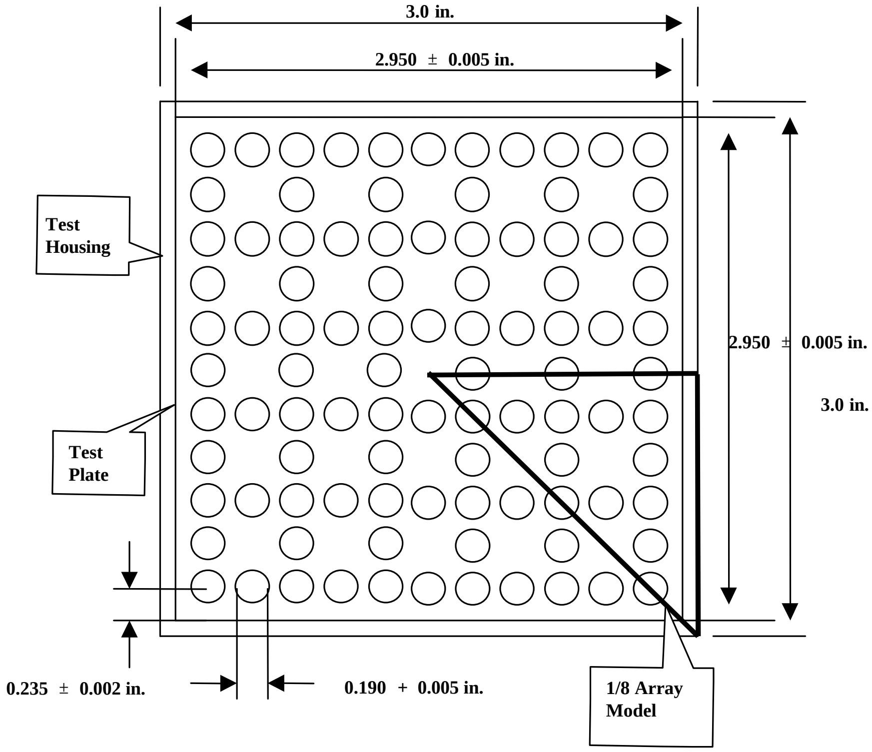

Figure 1 : Bottom Nozzle Plate Geometry

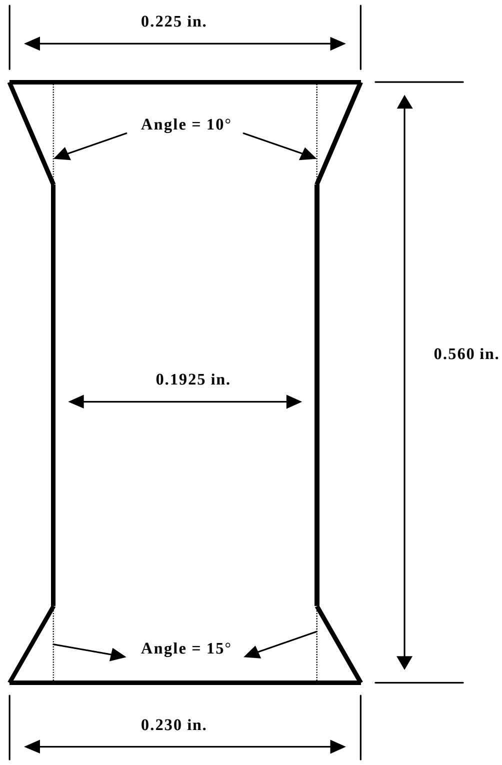

Figure 2 : Bottom Nozzle Plate Flow Hole Geometry

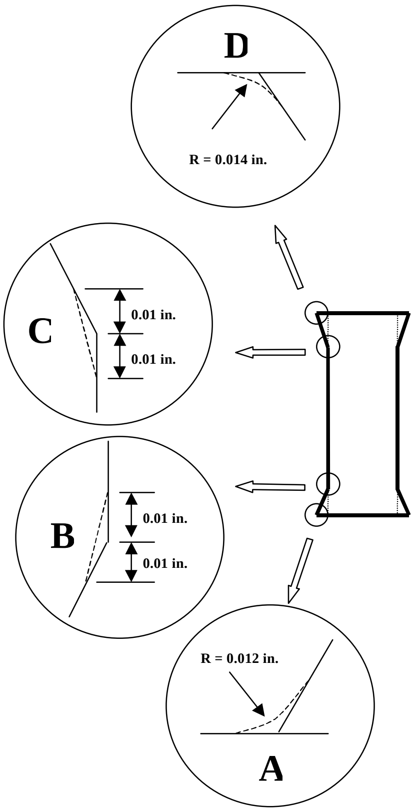

Figure 3 : Flow Hole Geometry for Deburring Study

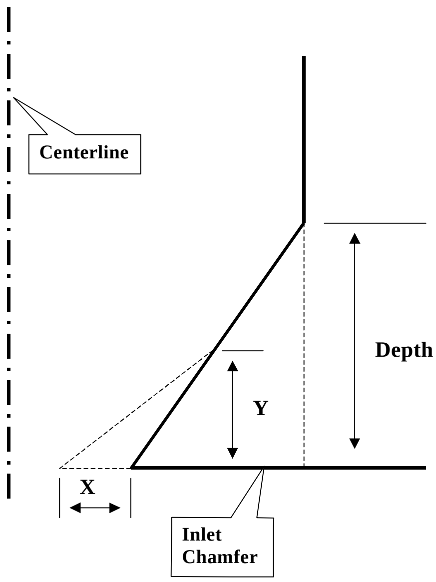

Figure 4 : Flow Hole Geometry for Multi-Chmafer Study

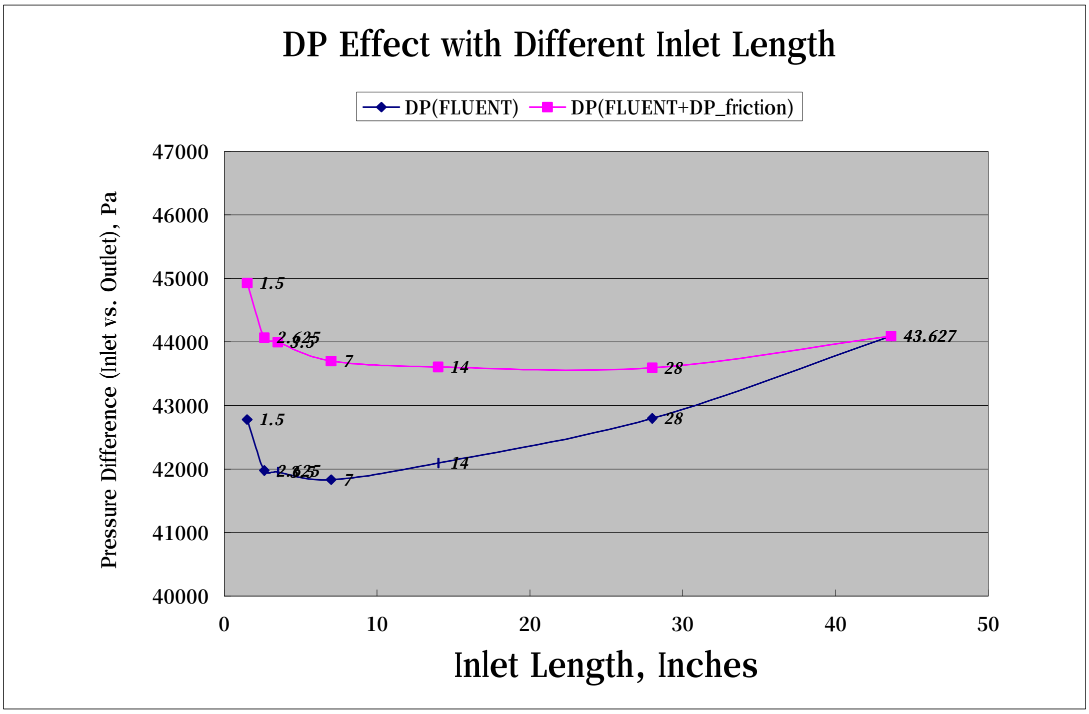

Figure 5 : Pressure Drop Difference with Various Inlet Length

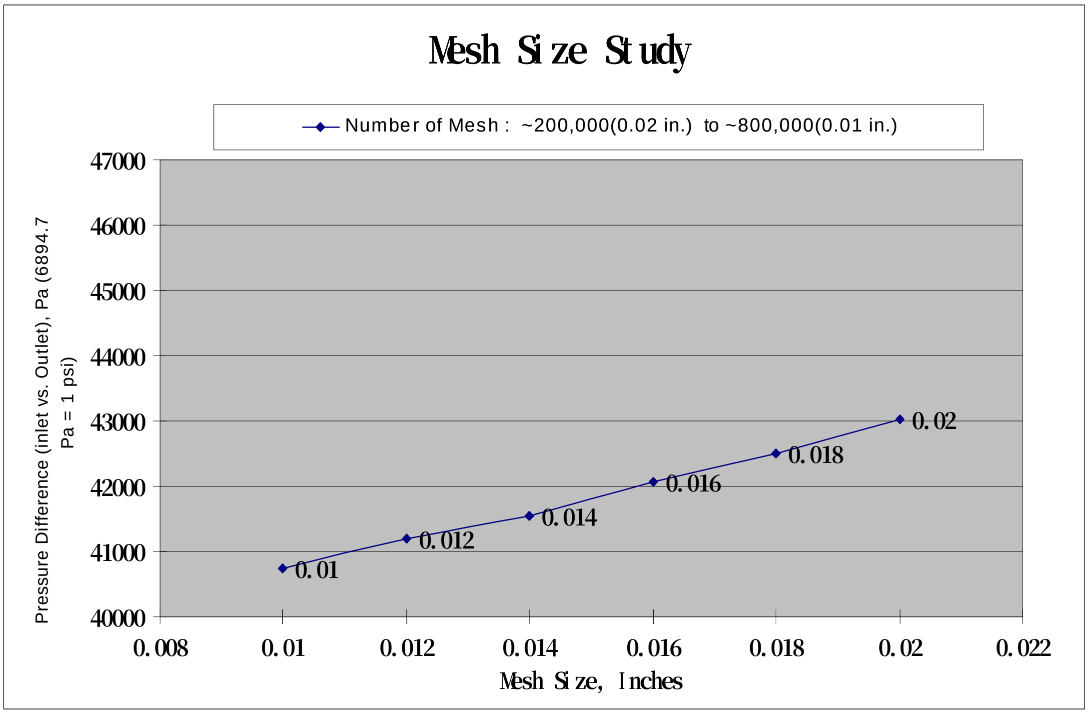

Figure 6 : Pressure Drop Difference with Various Mesh Size

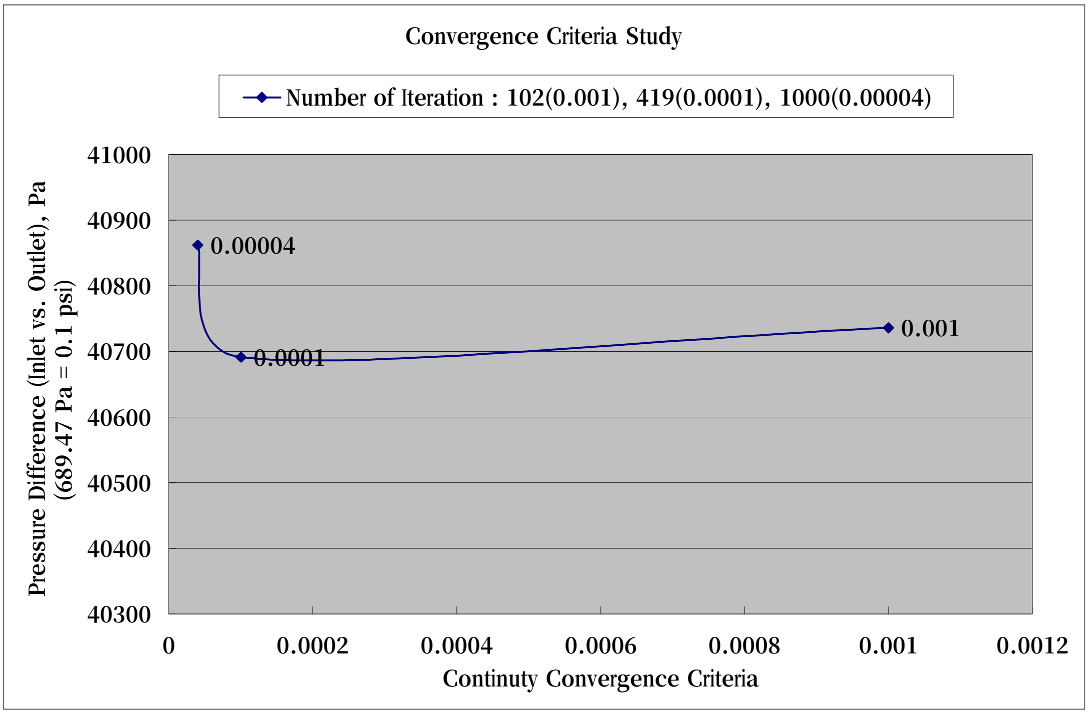

Figure 7 : Pressure Drop Difference with Various Convergence Criteria

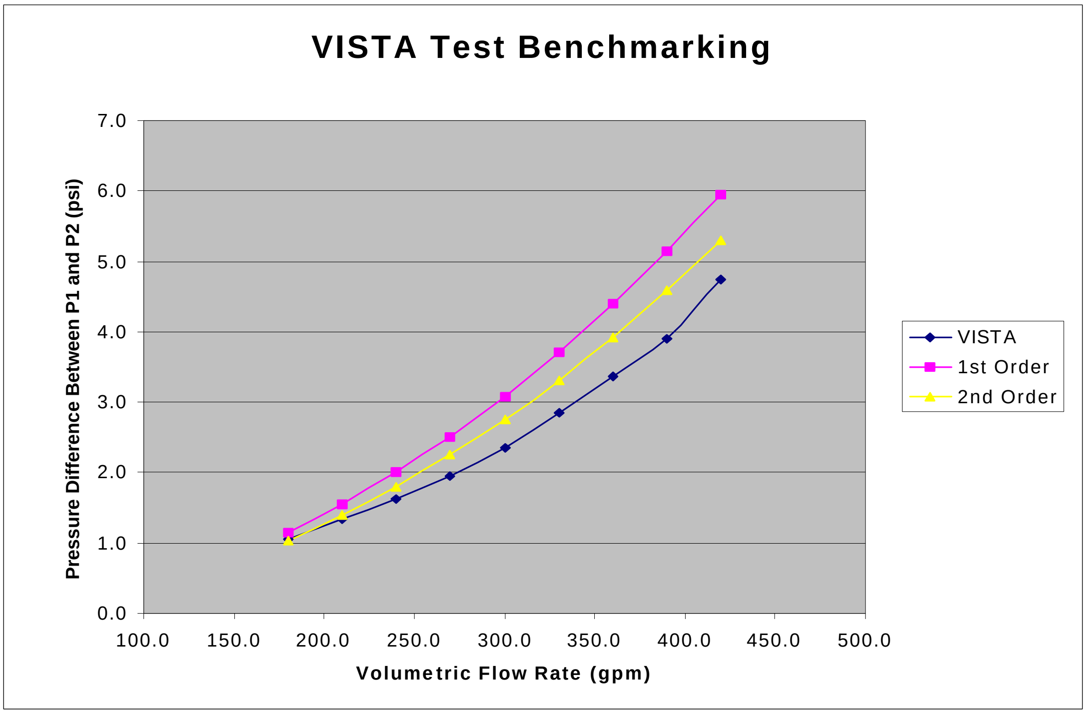

Figure 8 : Pressure Drop Difference Based on the VISTA Benchmarking Study

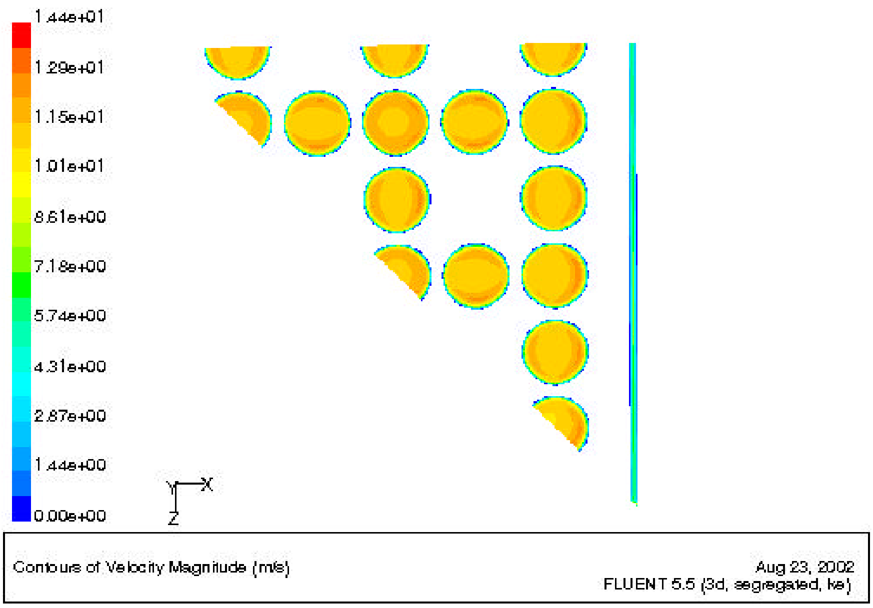

Figure 9 : Typical Velocity Contour at the Chamfered Hole Region

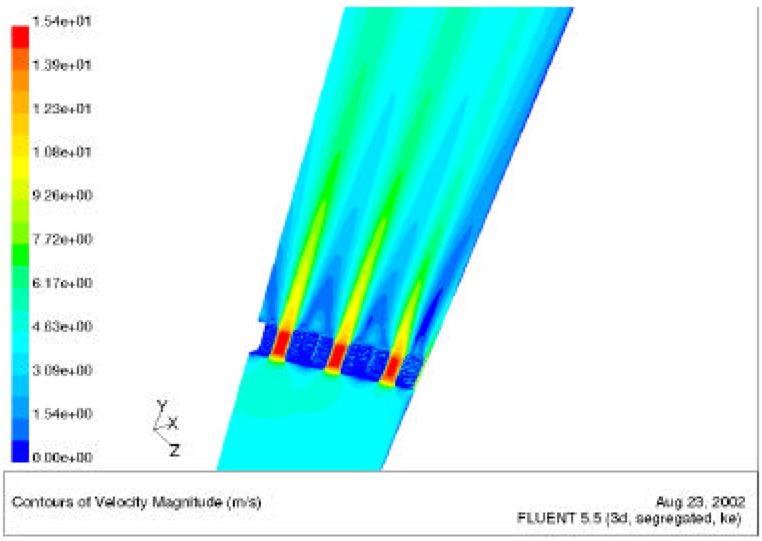

Figure 10 : Typical Velocity Contour at the Whole Control Volume