Resistance loss analysis and structure optimization of conical diffuser in siphon rectifying device

J F Zhang, L H Xie and Xia Zhang
National Research Centre of Pumps, 301 Xuefu Road, Zhenjiang, Jiangsu, 212013, China
2211511041@stmail.ujs.edu.cn

Abstract. Siphon water conveyance is a commonly used way to transport water in high efficiency, and its core equipment is the submerged rectifying device at the entrance. Due to the complex structure inside the device, the local resistance loss is too high, which has a certain impact on the water delivery efficiency. By taking the single conical diffuser inside the siphon rectifying device as the research object, analyzing the mesh generation and the simulation strategy. Of which the target is reducing the resistance loss. Then the optimal plan is $y_0=10\mu m$, $N=20$, $R=1.3$, using $k\epsilon$ standard. And the analyzing of the head loss in each part of the conical diffuser is based on the simulation results. The obtained head loss of the diffuse part account for 40%-50% of the total loss. The parametric analysis of the conical diffuser structure is carried out for the diffuse part. Through the orthogonal design and the result analyzed, the optimal structural scheme is $L_1=36mm$, $L_2=74mm$, $D_1=17.5mm$, $D_2=24.6mm$, these optimal structure scheme were validated using numerical simulation results.

1. Introduction
With the rapid development of industry and agriculture, the scientific method of water diversion has been an important topic in the world. In different water transfer projects, there are some common applications: open channel diversion, tunnel diversion and pipeline diversion. The siphon water delivery system as a kind of commonly used water system pipeline water diversion, compared with other water supply system, has a lot of advantages, and it realizes the transportation and water resources safety high efficiency and energy saving. It realizes the scientific, safe efficient and energy-saving transportation of water.

The structure of the Siphon water delivery system diagram shown in figure 1

Siphon water delivery system consists of submerged rectifying device, vacuum suction device, water access pipe and other components. The working mode of siphon water delivery devices is, at the initial state, the siphon pipeline above the horizontal part of the tube are all air, then close the outlet pipe valve, open the pump interface valve top of the siphon, start the vacuum pump, and the air will be exhausted of the pipe, water from the siphon rectifying device will come into the tube, finally filled with the entire pipeline, so as to achieve the effect of continuous siphon water delivery.

The core of its hardware equipment is the submerged rectifying device, and its schematic diagram is shown in figure 2.

The installation of the rectifier at the inlet of the siphon device can effectively improve the inlet flow of the siphon device and eliminate the entrained-vortex and improve the performance of the
device. The structural optimization of the Siphon water delivery system is helpful to enhance its conveying capacity and improve its stability and safety.

Figure 1. The diagram of the siphon water delivery system

Figure 2. The diagram of the submerged rectifying device

In practical engineering applications, although it can improve the pipeline transport performance, because of its internal complex structure, there will be a certain local hydraulic losses, so that the overall pipeline transport performance have a certain impact. So there will be some effect to the transportation performance of the whole pipeline.

As the internal structure of the siphon rectifying device is a series of vertical conical diffuser, the structure is complicated. Therefore, the numerical simulation method is used to study the effects of the diameter of the inflow section and the outflow section, the length of the inflow section and diffuse part and working condition parameter to the resistance loss of single conical diffuser. Then the optimal parameters of the involute are obtained, which provide the basis for the optimization design of the internal structure of the siphon rectifying device.

2. Physical model and the calculation method

2.1. Physical model
In order to study the influence of single conical diffuser structure on the resistance characteristics of the siphon rectifying device, the involute structure can be divided into the inflow pipe section, the diffuse section and the outflow pipe section. parametric analyze the structure, the structural features
can be shown by the five structural parameters, figure 3 is the schematic diagram of the conical diffuser in simplified siphon rectifying device structure.

The parameters of the conical diffuser are shown in table 1.

| L1/mm | L2/mm | L3/mm | D1/mm | D2/mm |
|-------|-------|-------|-------|-------|
| 56    | 23.4  | 135   | 17.4  | 27.4  |

![Figure 3. The structure diagram of conical diffuser](image)

In the actual operating state, the conical diffuser is placed in a vertical state, the smaller section put downward, the water flow from the bottom into the inflow pipe section, outflow from the outflow pipe section.

2.2. Calculation of the head loss theoretical value

In order to provide the theoretical reference for numerical simulation, the theoretical head loss is calculated based on the empirical formula of the local loss coefficient of the conical diffuser and the calculation formula of the loss along the pipe.

The model can be divided into three sections according to the structure, the inflow pipe section, the diffuse section and the outflow pipe section. The head loss of the inflow section and the outflow section can be calculated by using the Darcy formula. The diffuse section can be calculated according to the local resistance coefficient of the conical diffuser, the formula 1 is the Darcy formula.

\[
h_f = 
\frac{\lambda l V^2}{d 2g}
\]  

(1)

2.2.1. The inflow pipe section and the outflow pipe section. The flow rate of the inlet of the inflow pipe section is \(v_{in}=1.2\text{m/s}\), the length is \(L_1=56\text{mm}\), the diameter is \(D_1=17.4\text{mm}\)

\[
Re = \frac{Vd}{\nu}
\]

(2)

According to the equation 2, in the inflow pipe section the \(Re=1.79\times10^4\), reference for the Moody diagram\(^{(1)}\), obtained that the local resistance coefficient of the inflow pipe section is about 0.025, then the head loss of it is about 0.003m.

In the same way, the flow rate of the inlet of the outflow pipe section is \(v_{out}=0.48\text{m/s}\), the length is \(L_3=135\text{mm}\), the diameter is \(D_2=27.4\text{mm}\), the Reymond number is \(1.14\times10^4\) according to the Moody diagram, obtained that the local resistance coefficient of the outflow pipe section is about 0.029, then the head loss of it is about 0.001m.

2.2.2. Diffuse section. The head loss of the diffuse section is calculated according to the local resistance coefficient of the conical diffuser\(^{(1)}\), the formula is

\[
h_f = \frac{\lambda l V^2}{d 2g}
\]  

(1)
The calculation of the local resistance coefficient referenced to this formula

\[
h_f = \frac{\xi}{2g} \frac{V^2}{2}
\]

The calculation of the local resistance coefficient referenced to this formula\(^{(1)}\)

\[
\xi = k \left( \frac{A_2}{A_1} - 1 \right)^2 + \frac{\lambda}{8 \tan \frac{\theta}{2}} \left[ \left( \frac{A_2}{A_1} \right)^2 - 1 \right]
\]

Where the coefficient \(k\) is define by the diffusion angle of the diffuse section \(\theta\), which is 25 here, obtained the \(k=0.62\), after the calculation the head loss of the diffuse section is about 0.03m.

2.3. The numerical simulation of the conical diffuser

Simulation for conical diffuser with smooth wall\(^{(2)}\), based on FLUENT, by using three-dimensional model, the hexahedral structured grid and different mesh generation strategy. Then the simulation results are compared with the theoretical calculation of the conical diffuser head loss. In this part we get the influence of different mesh generation strategies on the calculation results, then select the optimal mesh generation strategy which is suitable for the structure model. The grid generation of the entrance face is shown as follows.

![Figure 4. The grid generation of the entrance face](image)

The diameter of the inflow pipe section is \(D_1\), the outflow section is \(D_2\) the length of the inflow pipe is \(L_1\), the diffuse section is \(L_2\), the length of the outflow pipe section is \(L_3\), the distance from the bottom grid is \(y_w\), The number of the near wall grid nodes is \(N\). The O-block\(^{(3)}\) is used into the whole meshing domain, and the meshes in the boundary layer are local refinement. The method is to modify the block edge parameter at the near wall. The ratio of the vertical dimension of the first layer and the second layer is the grid gradient \(R\). Where the number of near-wall nodes is the number of nodes on the edge near the wall, the length of the selected edge in this case is 3.369 mm.

The medium in the calculation domain is water, of which the physical properties are shown in table 2.

The inlet boundary condition is velocity-in, the velocity is \(v_{in}=1.2\text{m/s}\), the outlet boundary condition is outflow, \(k-\varepsilon\) standard is selected for the turbulence model, the no slip shear condition is used on the wall boundary condition, the fluid pressure-velocity coupling is based on the SIMPLE algorithm, Using the second order upwind, the iterative convergence accuracy is \(1.0 \times 10^{-4}\).
Table 2. Physical properties of water

| Temperature | Density \( \rho \) kg/m\(^3\) | Heat capacity \( c_p \) \( \text{kJ/(kg} \cdot \text{K}) \) | Thermal conductivity \( \lambda \) \( \text{W/(m} \cdot \text{K}) \) | Kinematic viscosity \( \nu \) \( \text{m}^2/\text{s} \) | Dynamic viscosity \( \eta \) \( \text{Pa} \cdot \text{s} \) | Prandtl number \( \text{Pr} \) |
|-------------|-----------------|------------------|------------------|-----------------|-----------------|-----------------|
| 20          | 998.2           | 4.183            | 0.599            | 1.006 \times 10^{-6} | 1.004 \times 10^{-3} | 7.02            |

2.3.1. Influence of different near wall grid size on simulation results. The calculated grid is \( R = 1.3 \) and \( N = 20 \), and the near wall grid sizes \( y_w = 5\mu m, 10\mu m, 20\mu m, 50\mu m, 80\mu m \) and \( 100\mu m \) are used respectively. The results are shown in Fig5.

![Figure 5. Influence of different near wall grid on simulation results](image-url)

It can be seen from figure. 5 that the deviation of the calculated results increases with the increase of the mesh size in the near the wall. Therefore, under the condition of this model, the near wall grid size \( y_w = 10\mu m \) is suitable, the deviation is only 0.66%. And the above data can be used to obtain that the grid setting for the numerical simulation of the flow in the conical diffuser. The grid size \( y_w \) of the near wall has a great influence on the calculation results. If the unfavorable near wall grid size is chosen, the maximum error can reach 14%. And the influence of other parameters on the results is based on the selection of the near wall grid size \( y_w \), and the effect on the calculation results is limited. [4]-[7]

2.3.2. Influence of different number of the near wall grid nodes on simulation results. The setting of grid is \( y_w = 100\mu m, R = 1.3, N = 20 \), and the rest of the setting is same with above, respectively elect the number of different near-wall grid nodes as \( N = 15,20,25,30 \), and then change the near wall grid size to \( y_w = 10 \mu m \).

![Figure 6. influence of different near wall grid nodes number](image-url)
It can be seen from Fig. 6 that the number of near wall grids nodes with same near wall grid size will have a certain effect on the accuracy of the results \[8\], and select different numbers of near-wall grid nodes, the maximum error of each result and the minimum value is only 1% difference, it can be considered that the impact of the near wall grid nodes number on the accuracy of the results is very limited. After changing the near wall grid size, the smaller \(y_w\) can make the accuracy of the calculation result be improved effectively under the given model and boundary conditions. Although increasing the number of near-wall mesh nodes can slightly reduce the calculation result error, but when the near wall grid size \(y_w = 100\mu m\), due to the edge length limit, the number of nodes \(N\) cannot reach 30. if \(N\) has an influence on the computational time and computational space then it be given the reason since \(N\) influence is minimal. Based on the above analysis, we can select the number of near-wall grid nodes \(N = 20\) as the optimal option.

2.3.3. Influence of different grid gradual ratio on simulation results. The setting of grid is \(y_w = 100\mu m\), \(N = 20\), the remaining settings are the same as the previous part, respectively, the use of different grid gradual ratio \(R = 1.2, 1.3, 1.5, 1.8, 2\), and then change the near wall grid size to \(y_w = 10\mu m\), The results obtained as shown in figure 7.

![Figure 7. Influence of different grid gradual ratio](image)

It can be seen from Figure 7, in the case of the same near wall grid size, the influence of the grid gradual ratio on the calculation results are not large, the difference of the calculation results is less than 2%, and in the selection of the smaller near wall grid When the size is \(y_w = 10\mu m\), the accuracy of the calculation result is greatly improved. When compared with other options, \(R = 1.3\) is closer to the theoretical calculation, the error is only 0.4%. So the grid gradual ratio \(R = 1.3\) is selected as the optimal option.

2.3.4 Influence of different turbulence models on simulation results. The setting of grid is \(y_w = 10\mu m\), \(N = 20\), the remaining settings are the same as the previous part, the select of turbulence models is \(k-\varepsilon\) standard, \(k-\varepsilon\) RNG, \(k-\varepsilon\) realizable, \(k-\varepsilon\) low-re, \(k-\varepsilon\) RNG Enhanced, \(k-\varepsilon\) realizable Enhanced.

The simulation results are shown in the following figure:

It can be seen from Fig. 8 that the results of the standard \(k-\varepsilon\) model, the RNG \(k-\varepsilon\) model and the Realizable \(k-\varepsilon\) model, which are all high Reynolds number turbulence model, are similar in the same near wall treatment. But the \(k-\varepsilon\) low-Re model has a large error. And the combination of the Reynolds number in this study is a high value, so the choice of this turbulence model will have a greater error \[9\].

Comparing the different near wall treatments, the simulation results show that the standard wall functions are more accurate, and the error of using the enhanced wall functions is larger than 10%. Therefore, we use the standard \(k-\varepsilon\) model, and the standard wall functions is the optimal option.
3. Comparison of original plan and contract scheme

In this paper, the k-ε standard is chosen as the optimal turbulence model, the mesh generation strategy is $y_w=10\mu m$, $N=20$, $R=1.3$. In order to discuss the influence of conical diffuser resistances with different structural parameters, finish the numerical simulation of original plan with different structure at first. The structure parameters of the original plan are shown in Table 4 below:

| L₁/mm | L₂/mm | L₃/mm | D₁/mm | D₂/mm |
|-------|-------|-------|-------|-------|
| 36    | 79    | 135   | 17.5  | 22.7  |

Considering about the economic velocity in pipe and the different structural characteristics of the conical diffuser, choose 0.3 m/s, 0.9 m/s, 1.2 m/s, 1.5 m/s, 2m/s as the inlet velocities, comparing the difference between original plan and contract scheme in different inlet velocities.

From the figure, in different inlet velocities, the head loss increased gradually with the increase of the velocity, and in the same inlet velocity condition, the head loss of original plan is less than contract scheme, and with the increasing of inlet velocity, the difference of head loss is more, the head loss is about 20% at the maximum inlet velocity, thus the optimization of the conical diffuser structure parameters is beneficial to the decrease of the head loss.
Figure 10. Characteristic sections of the conical diffuser

For the further analysis of the various structural parameters on the effect of conical diffuser head losses, select the middle section of inlet section, diffuse section and outlet section, inlet and outlet as the five characteristic sections. Comparing the head loss between each characteristic section and the inlet.

Figure 11. Comparing the head loss between each characteristic section to the inlet

The diffuser percentage of each part of the head loss of total head loss, the figure shows that the main loss is produced in the divergent section, the losses accounted for the entire pipe loss of 40%-50%, therefore, the middle of gradually expanding the size of the optimization can effectively improve the whole hydraulic expanding the performance of the pipe\cite{10,11,12}.

4. Analysis of influence factors of a single conical diffuser

According to the structural parameters of conical diffuser, the five structure parameters were optimized. The length of outlet pipe for L3, can be regarded as the outlet extension part, when its length is five times larger than the diameter of the pipe, it can be regarded that the outlet flow pattern is stable, so in each scheme, take 135mm for L3. Then for the remaining four structural parameters, 4 factors and 3 levels orthogonal tables are established. The table of factor levels is shown in table 5:

| Table 4. Structural parameters of 4 factors and 3 orthogonal levels |
|---|---|---|---|---|
| L1/mm | L2/mm | D1/mm | D2/mm |
| 1 | 32 | 74 | 15 | 21 |
| 2 | 36 | 79 | 17.5 | 22.8 |
| 3 | 40 | 84 | 20 | 24.6 |

The four factors A, B, C and D were L1, L2, D1, D2, respectively, and each factor had three levels. The simulation test scheme, such as tables, consists of 9 groups.

| Table 5. Simulation test scheme of the structural parameters of 4 factors and 9 orthogonal levels |
|---|---|---|---|---|
| L1/mm | L2/mm | D1/mm | D2/mm |

8
Select $yw=10\mu m$, $N=20$, $R=1.3$ as mesh generation strategy, choose the turbulence model as $k$-epsilon standard, and use the same simulation strategy. The simulation results are as follows:

**Table 6.** Simulated head loss for the test scheme for the selected trial inlet velocities

|    | 0.3 m/s | 0.9 m/s | 1.2 m/s | 1.5 m/s | 2 m/s |
|----|---------|---------|---------|---------|-------|
| 1  | 0.00265 | 0.01755 | 0.02867 | 0.0418  | 0.06765 |
| 2  | 0.00255 | 0.01684 | 0.02745 | 0.0402  | 0.06541 |
| 3  | 0.00286 | 0.01898 | 0.03082 | 0.0448  | 0.07245 |
| 4  | 0.00378 | 0.02429 | 0.03929 | 0.0568  | 0.09102 |
| 5  | 0.00296 | 0.01908 | 0.03112 | 0.0453  | 0.07337 |
| 6  | 0.00265 | 0.01714 | 0.02806 | 0.0410  | 0.06643 |
| 7  | 0.00265 | 0.01714 | 0.02806 | 0.0410  | 0.06643 |
| 8  | 0.00316 | 0.02051 | 0.03327 | 0.0482  | 0.07153 |
| 9  | 0.00296 | 0.01898 | 0.03071 | 0.0445  | 0.07153 |

It is shown that, under the same inlet velocity, size relationship among the most simulation results of these schemes are alike, so we can select one certain simulated results under a certain inlet velocity as a representative, carry out orthogonal test analysis of each scheme.

Select the scheme in which the inlet velocity is 1.2 m/s, and K represents average value under corresponding results of different levels for every factor and rage $R$ is the difference between the maximum value and minimum value of same factor under different levels, for example in factors A, $R=k3-k2$. Every factor can be calculated according to the results of above calculation, calculation results are shown in the following table.

**Table 7.** Calculated average head loss to for selection of the optimal structure parameters

|    | A      | B      | C      | D      |
|----|--------|--------|--------|--------|
| k1 | 0.0306 | 0.0297 | 0.0299 | 0.0345 |
| k2 | 0.0289 | 0.032  | 0.0298 | 0.0302 |
| k3 | 0.0326 | 0.0306 | 0.0325 | 0.0276 |
| R  | 0.00364| 0.00235| 0.00269| 0.0068 |

From the table 8, according to the range of each factor , the biggest is factor A, then in successively is C, B, D. Therefore, the primary and secondary relationship for each factor is $A \rightarrow C \rightarrow B \rightarrow D$. We can consider that compare to other factors, the factor A has the greatest influence on the simulation results, so give priority to considering the value of A in the structural design.
When determining the level of each factor, we can compare value of k of each indicator, therefore the purpose of the optimization is to reduce the head loss, so it is ought to take the minimum value of each factor, it can be seen from the figure, the optimal structure parameters can be selected for A2, B1, C2, D3.

Table 8. Optimal structure parameters

| L₁/mm | L₂/mm | D₁/mm | D₂/mm |
|-------|-------|-------|-------|
| 36    | 74    | 17.5  | 24.6  |

The numerical simulation of the optimized structure is carried out by using the same simulation strategy as the previous simulation, and the simulation results are shown in the following table:

Table 9. Simulated head loss of the optimized structure

| Inlet velocity | 0.3m/s | 0.9m/s | 1.2 m/s | 1.5 m/s | 2 m/s |
|----------------|--------|--------|---------|---------|-------|
| Optimal scheme | 0.002  | 0.016  | 0.02643 | 0.038   | 0.062 |
| Scheme 6       | 0.002  | 0.016  | 0.02744 | 0.04    | 0.064 |
|                | 6      | 65     | 83      |         | 59    |

From the table, compare the simulation results of each designed structure scheme to the original plan, the head loss under different flow conditions all has been reduced. Making comparison to the best scheme among all structural schemes (scheme 6), There is still about five percents optimization at the maximum inlet velocity. In summary, it is considered that performance is optimized.

5. Conclusion

- For the numerical simulation of a single conical diffuser, grid generation strategy and the choice of turbulence model has great influence on the simulation results, among them, near wall grid size has great influence on the calculation results, when the influence of other factors is very limited. So in the simulation of single conical diffuser numerical, it is very important to select the appropriate the near wall grid size.
- The same type of turbulence models has limited impact on the simulation results, nevertheless due to the particularity of flow in conical diffuser, different near wall treatment methods has great
influence on the results of calculation, so it is also very important to select the appropriate near wall treatment of the turbulence model.

- By analyzing different grids generation and simulation strategies, the optimal scheme suitable for the study of this paper is selected, and the simulation of conical diffuser with multiple structures are carried out based on the optimal simulation scheme.
- The head loss in conical diffuser is mainly derived from the mid diffuse section, so the structural optimization of the diffuse section can greatly reduce the head loss of the whole conical diffuser.

References

[1] Mo Nairong. M 2000 Engineering Fluid Mechanics huazhong university of science & technology press

[2] Liu Ling, Lu Ping J 2013 Numerical Simulation on Resistance Loss and Structure Optimization of CFB-FGD Venturi Tube Environmental Science & Technology 36(9) 154-158

[3] He Yongseng, Jiang Guangbiao J 2006 Study on Diagnostic System for Numerical Prediction of Turbulent Flow in a Conical Diffuser: Effect of Model Functions and Grid and Reynolds Numbers Natural Science Journal of Xiantan University 28(2) 1-4

[4] Qin Wenjie, Hu Chenguang, Guo Liangping, et al J 2006 Effect of Near-Wall Grid Size on Turbulent Flow Solutions Transactions of Beijing Institute of Technology 26(5) 388-392

[5] Zhang Tao, Zhu Xiaojun, Peng Fei, et al. J 2013 Analysis of effect near-wall treatments on numerical computation of turbulent flow Journal of Naval University of Engineering (6)104-108

[6] Pan Weiguo, Nie Xuejun, Lei Junzhi, et al. J 2001 A numerical study on the structure and resistance characteristic of turbulent boundary layer Chinese Journal of Computational Mechanics 18(4) 393-396

[7] Li Xiaojun, Yuan Shouqi, Pan Zhongyong, et al, J 2012 Realization and application evaluation of near-wall mesh in centrifugal pumps Transactions of the Chinese Society of Agricultural Engineering 28(20) 67-72

[8] Li Qiang, Wei Jianqin, Xu Cangsu, J 2013 Influence of Grid Layer Number in Boundary Layer on Turbulence Kinetic Energy Distribution in Nozzle Vehicle Engine (3) 18-23

[9] Zhou Zhijun, Lin Zhen, Zhou Jun-hu, et al, J 2007 Application of different turbulent models in calculation of flow resistance in pipelines and comparison thereof Thermal Power Generation 36(1) 18-23

[10] Zhao Baofeng, J 1998 Study on local loss coefficient of diffuser Journal of Northeast Agricultural University (4) 367-370

[11] Li Qiong, Qi Er-rong, J 2007 Experiment Study on Flow Characteristic in Venturi Tube China Rural Water and Hydropower (11)65-67

[12] Zhao Baofeng, Jin Yingzi, Zhang Zhongxue, 1996 J Study on Loss Coefficient of Sudden Enlargement Increaser Journal of Heilongjiang Hydraulic Engineering College (4)31-33

[13] Gan G, Riffat S B. 2015 J Computational and experimental study of pressure losses in duct transitions International Journal of Energy Research 20(11):979-987.