

Numerical investigation of the influence of the nozzle bump on nozzle flow

Jinghan Ge ^{1, †}, Yuhe Mao ^{2, *, †}, Ruihan Wu ^{3, a, †}, Chongchu Zhang ^{4, b, †}

¹ Wenzhou High School International Division, Wenzhou, 325000, China

² School of Mechanical Engineering, University of Bristol, Bristol, BS8 1TH, UK

³ School of Chemical engineering, Dalian University of Technology, Dalian, 116000, China

⁴ School of Mechanical engineering, Hebei University of Engineering, Handan, 056000, China

* Corresponding Author Email: cd20143@bristol.ac.uk, wurhan@mail.dlut.edu.cn,
171849143@masu.edu.cn

[†]These authors contributed equally.

Abstract. Fluid Mechanics is the study of liquid or gas performance in motion or at rest. Many of the engineering disciplines apply fluid mechanics principles and concepts. With the development of technology, computer is involved in the field, which leads to computational fluid dynamics. In this paper, nozzles with various shapes of bump are used as the research object to investigate the influence of the bump on the nozzle flow via small disturbance approximation. Specifically, triangular, parabolic, irregular and combined shapes are utilized. Two different situations are also included in this paper, which are subsonic and supersonic case. The experimental results demonstrated that within reasonable parameters as the variable increases, the change becomes clearer. When area decreases in the same shape, the streamlines become fluctuant more extremely, especially the one near the boundary. The assumption that the fluid flows disorderly near the bottom is also verified by the simulation.

Keywords: CFD, python code, shape of nozzle, supersonic; subsonic.

1. Introduction

Research on fluid mechanics has a long history. For decades, one of the most popular ideas in the fluid mechanics is the idea that using computational structural dynamics (CFD) to solve problems. With the development of technology, computer has played an important role in CFD.

There are many successful cases about CFD. Blocken Bert mentioned that CFD has undergone a successful transition from an emerging field into an increasingly established field in urban physics research, practice, and design [1]. He also argues about the bright future of CFD in urban physics field. Except for aspect of urban physics, CFD also exists in food drying process. Malekjani, Narjes and Seid Mahdi Jafari declare that advanced computer modeling and simulation techniques can help in developing new dryers, modification of current systems, energy saving and process optimization [2]. For example, Hydrodynamics of fluid flow, heat and mass transfer during drying can be predicted using CFD [3]. As a result, CFD has a bright future in many fields, which is a very useful tool for dealing with many challenging issues.

Recently, Zhang et al. [4] used CFD to simulate the movement of ammonia and hydrogen sulfide pollutants in a public toilet under personalized ventilation, which gives people a lot of experience to fight against COVID-19. They used CFD to predict the air flow and measure the indoor air quality. Besides, with the recent COVID-19 outbreak, emergency department design must change in order to be adequately adapted to handle the epidemic. Furthermore, Alrebi et al. [5] examined the design of the emergency department at a university hospital using the CFD simulation based on the ANSYS CFX package. A coupled computational fluid dynamic–computational structural dynamics numerical simulation method is established based on an unstructured dynamic mesh and impact dynamics technology to solve the phenomena of ‘motion–collision–fragmentation–fragment motion’ for foreign objects such as fragile ice and hail entering the inlet [6]. In short, there are many fruitful results in

CFD field and most of them are practical to our daily life.

In this paper, the influence of the bump on the nozzle flow is studied via small disturbance approximation. 4 different boundaries, including triangular, parabolic, irregular, and combined shapes are utilized. For each shape, both supersonic and subsonic cases are investigated. Moreover, the influence of constant or variable inlet Mach number on the nozzle flow is studied.

The rest of this paper is organized as follows. The method is detailed with the implementation in Section 2. In Section 3, the results of the influence of the bump shape and inlet Mach number on the nozzle flow are presented. Finally, conclusions are drawn in Section 5.

2. Method

In this paper, the behaviors of fluid flow through a nozzle with different shapes of bump were compared. Specifically, bump shapes of triangular, irregular triangular, parabolic, and irregular shape combine sinusoidal with sine function are utilized for supersonic and subsonic cases. The functions $f(x)$ determining the triangular, irregular triangular, parabolic and combined shapes are expressed as Equation (1)-(4), respectively.

$$f(x) = \begin{cases} \beta x & (1.0 \leq x \leq 1.5) \\ -\beta x & (1.5 \leq x \leq 2.0) \end{cases} \quad (1)$$

$$f(x) = \begin{cases} \beta x & (1.0 \leq x \leq 1.5) \\ -2\beta x & (1.5 < x \leq 2.5) \end{cases} \quad (2)$$

$$f(x) = -\beta(x-1)(x-2) \quad (1.0 \leq x \leq 2.0) \quad (3)$$

$$f(x) = \begin{cases} \beta x & (1.0 \leq x \leq 1.5) \\ \beta \sin[\pi(x-1)] & (1.5 \leq x \leq 2.0) \end{cases} \quad (4)$$

The outlines of different shapes are demonstrated in Figure 1. The total length is 3 for all cases.

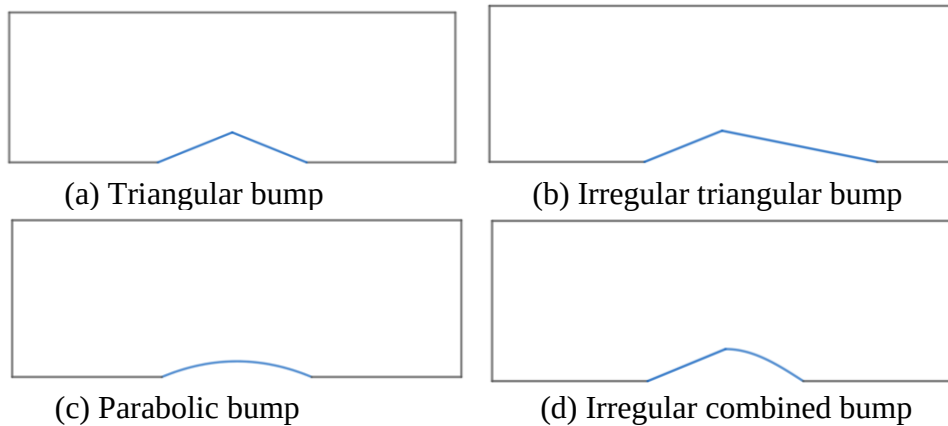


Figure 1. The outlines of different bump shapes

The small disturbance approximation is used. The vertex scheme was used to determine the value of potential function as shown in Figure 2.

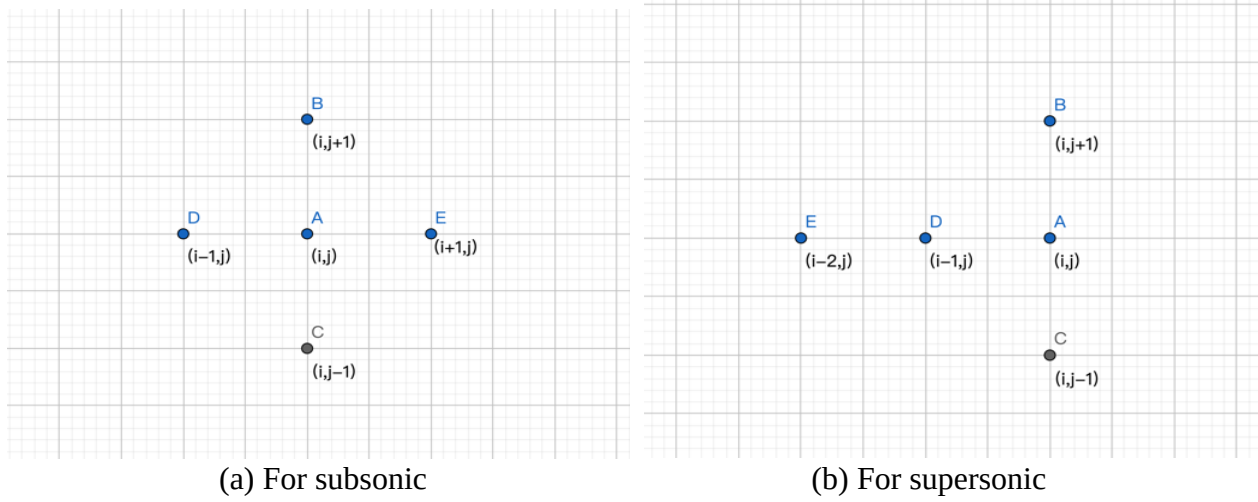


Figure 2. The diagram of the vertex schemes used for subsonic and supersonic case

For supersonic and subsonic cases with constant inlet Mach number, the expression of $\nabla\Phi$ is shown as:

$$\nabla\Phi = V_{\infty}x + \nabla\varphi \quad (5)$$

Where $\nabla\Phi$ is a divergence of vector potential, V_{∞} is perturbation velocity in the x-direction of uniform flow, x is the value in x-direction, $\nabla\varphi$ is a small perturbation velocity.

$$u = U_{\infty}(1 + \varphi_x) \quad (6)$$

$$v = U_{\infty}\varphi_y \quad (7)$$

Where u is the component of velocity in x-direction and v is the component in y-direction, φ_x and φ_y are the component of perturbation velocity in x-direction and y-direction, respectively. Equation (8) is the small disturbance equation:

$$(1 - M_{\infty}^2)\varphi_{xx} + \varphi_{yy} = 0 \quad (8)$$

Where $M_{\infty} = \frac{V_{\infty}}{a_{\infty}}$ the ratio of undisturbed onflow velocity V_{∞} and speed of sound a_{∞} is the Mach number of the incoming free stream.

For supersonic case, φ_{xx} and φ_{yy} are expressed as:

$$\varphi_{xx} = (\varphi_{i,j} - 2\varphi_{i-1,j} + \varphi_{i-2,j}) \quad (9)$$

$$\varphi_{yy} = (\varphi_{i,j+1} - 2\varphi_{i,j} + \varphi_{i,j-1}) \quad (10)$$

For subsonic case, φ_{xx} and φ_{yy} are illustrated as:

$$\varphi_{xx} = (\varphi_{i+1,j} - 2\varphi_{i,j} + \varphi_{i-1,j}) \quad (11)$$

$$\varphi_{yy} = (\varphi_{i,j+1} - 2\varphi_{i,j} + \varphi_{i,j-1}) \quad (12)$$

For supersonic and subsonic cases with Mach number is the function of φ , Equation (8) can be rewritten as:

$$[1 - M^2]\varphi_{xx} + \varphi_{yy} = 0 \quad (13)$$

With

$$k = 1 - M^2 = 1 - M_\infty^2 - (\gamma + 1)M_\infty^2 \varphi_x \quad (14)$$

$$\varphi_x = \frac{\varphi[i+1,j] - \varphi[i-1,j]}{2h} \quad (15)$$

Where M is the Mach number and M_∞ is the Mach number of the incoming free stream, k is a function about M with φ_x and the expression of φ_x is shown in Equation (14), γ is 1.4. In addition, the equations for φ_{xx} and φ_{yy} are same as Equation (9)-(12) represents above.

In this investigation, the number of constants used are as follows:

$$M_\infty = \begin{cases} 0.1 & \text{Subsonic} \\ 2.5 & \text{Supersonic} \end{cases}$$

To fulfill the simulation, an in-house python code is developed. At the beginning, we start with a nozzle shape function, which shows the shape of the boundary. This is the main part in this work. Then the error function will calculate the error of the true value and the estimated one for every 100 iterations, which can tell us whether the code is correct. If the values of error don't go down to zero, there may be bugs in the code. After that, it can output the velocity. Finally, plot functions are involved to draw figures like streamlines and the field of velocity.

3. Results and discussion

For all shapes of bump, Equation (13)-(15) for supersonic case were used to investigate the behaviors of fluids with various value of β . For all cases, maximum bump height used was consider as 0.1 metre. In addition, data of 0.05 mand 0.1 m were compared to investigate the effect of β variation to the behaviours of fluids for different bump shapes.

1) Triangular bump

In our research, we only focus on the small disturbance flow. As a result, the maxima of height should less than 10% of the total height. The total height is 1 m, so the maxim of β should be 0.2. Typical outlines of the simulation domain are depicted in Figure 3. The program will output the log(error)-iteration, the field of velocity, and streamlines.

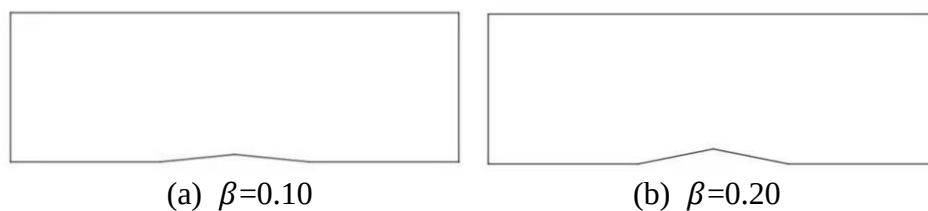


Figure 3. The outlines of triangular bump with different β

The streamline diagrams with changing in β are shown in Figure 4. And the figure of error is shown in Figure 5. It can be clearly seen in figure 4 that when β increases, the streamlines turn more curved. This shows that the area of the bump will have an important effect of the fluid. Due to the decrease of the area, the fluid will be disorder. Besides, it can be noted in Figure 5 that errors go down to zero at last. As iteration grows, the accuracy of the estimated value should increase, which is proved in Figure 5. This is also essential evidence to judge whether the code is correct.

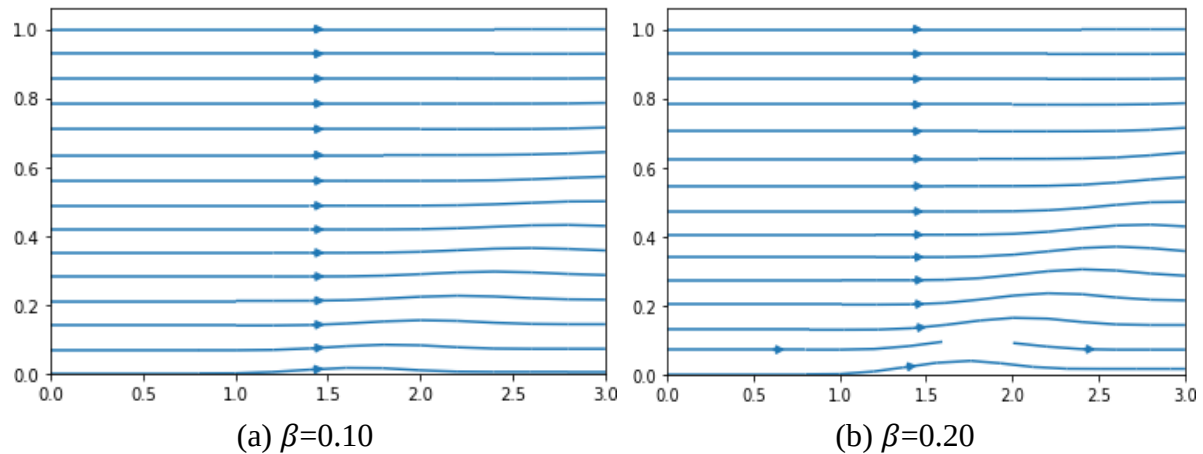


Figure 4. The diagrams of streamlines with changing in β

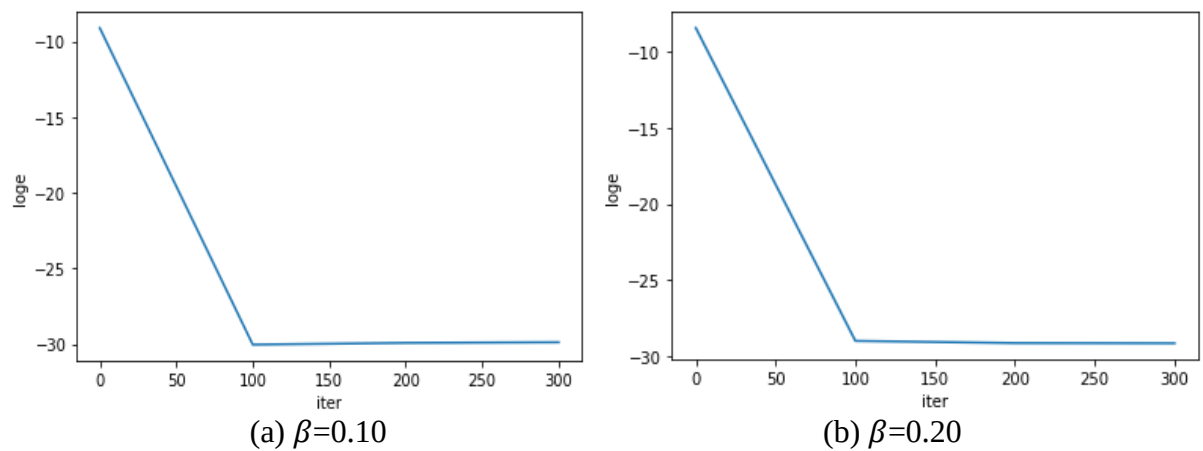


Figure 5. The figures of error against iteration with different β

2) Irregular triangular

About irregular shape, there have several variations as the change of β . In order to detect the change clearly so β changes very slight and the starting point is 0. Figure 6 show the outlines of the irregular triangular shape.



Figure 6. The outlines of irregular triangular bump with different β

Figure 7 show the streamline of this shape, with β equals to 0.10 and 0.20. The result illustrates that as β gets bigger, the streamline gets more curved.

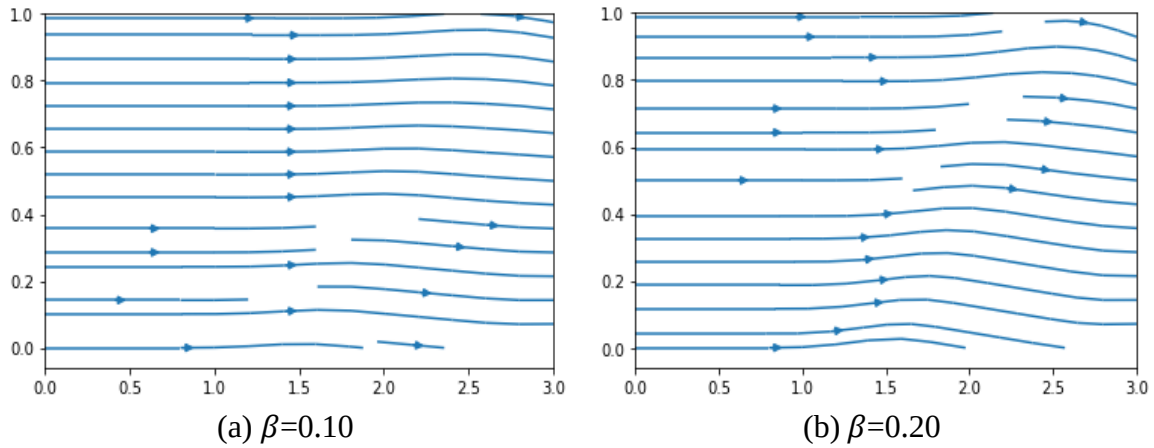


Figure 7. The diagrams of streamlines with changing in β

Figure 8 shows that the image of error with the different values of β . The trend of error is tended to 0, which means error could be ignored and the result is correct.

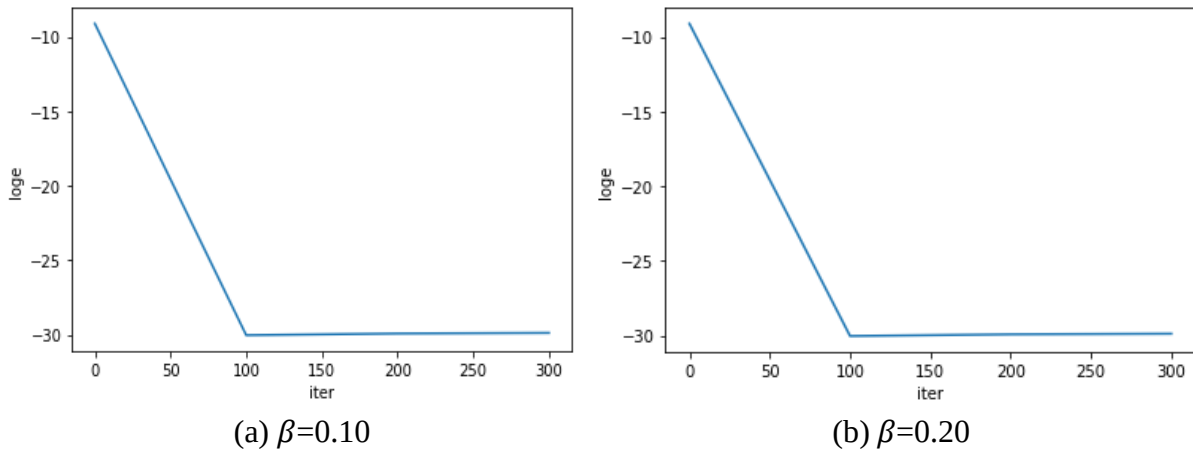


Figure 8. The figures of error against iteration with different β

3) Parabolic bump

In order to avoid the amplitude of bump higher than 10 percent of the total height of nozzle which should be 0.1m, the highest β was measured to be 0.4. In this part, all results were compared with values 0.2 and 0.4. To visualize the results with these two different β values, the figures of streamlines and error were compared. Also, the outlines for bump in nozzle with β values were shown to determine the differences of changing β value.

Figure 9 illustrates the outline of parabolic-shaped bump in the nozzle. It noted that the increase of beta value leads to the higher bump amplitude.

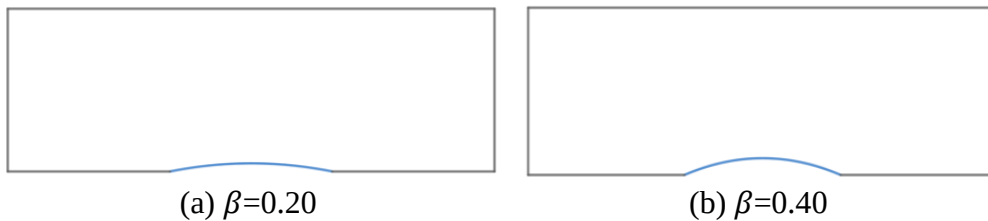


Figure 9. The outlines of parabolic bump with different β

The differences between streamlines diagrams are compared in Figure 10 when changing β value from 0.2 to 0.4. From Figure 10, it is clear to determine that due to the increase of β value with higher bump amplitude the streamlines of the fluids flowing would be more fluctuant.

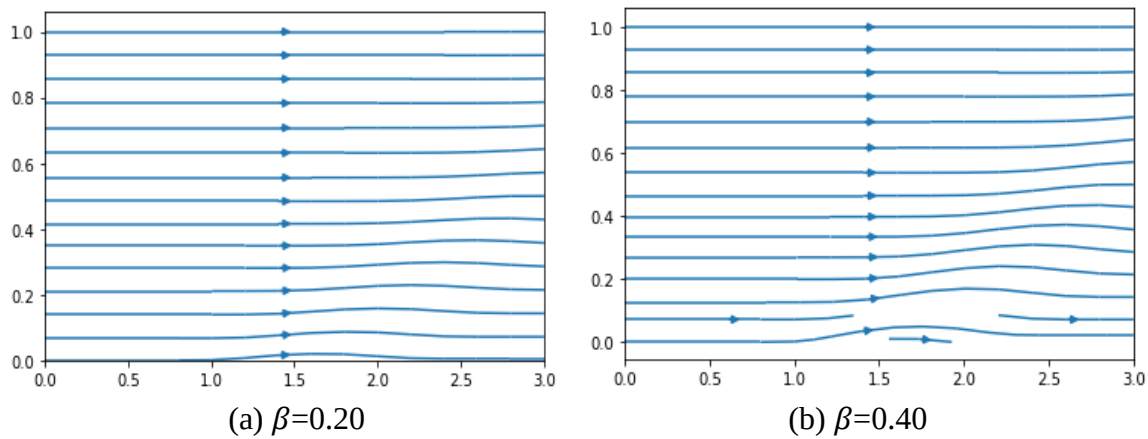


Figure 10. The diagrams of streamlines with changing in β

Figure 11 below presents the error varies when the increase of beta value occurs. By contrast, the differences between Figure 11(a) and (b) are negligibly small which demonstrates the error almost keep constant.

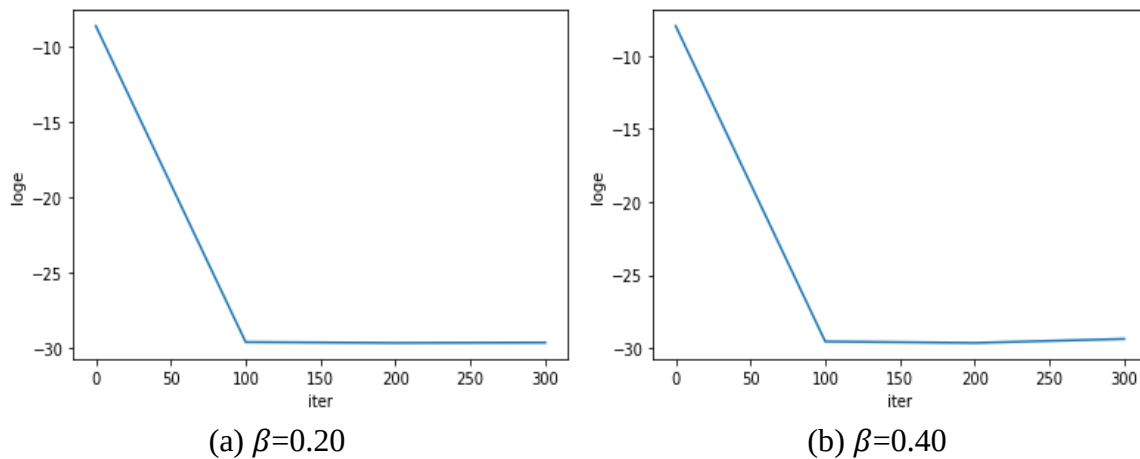


Figure 11. The figures of error against iteration with different β

4) Irregular shape combined triangular with sine function

About the combined shape, this is composed of a straight line and a sinusoidal line as shown in Figure 12.

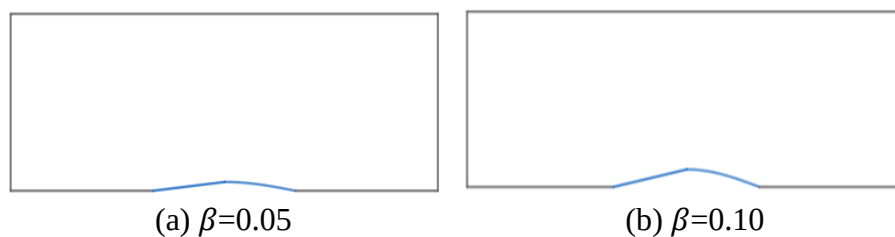


Figure 12. The outlines of parabolic bump with different β

Figure 13 below present the streamline diagrams of when beta is 0.05 and 0.1. From these two diagrams, it could be seen that as β rises, streamline becomes more fluctuant

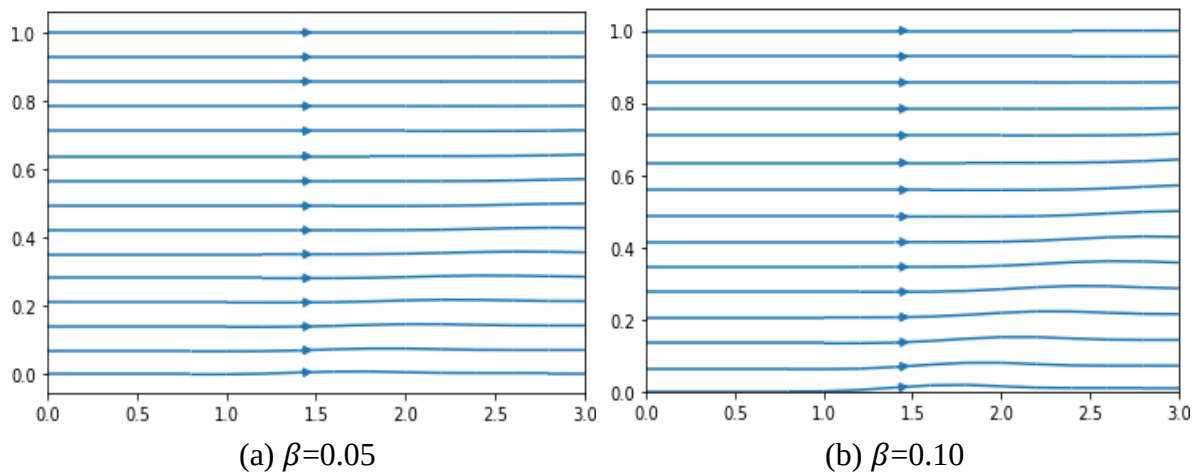


Figure 13. The diagrams of streamlines with changing in β

Figure 14 below present the error varies of different β . It shows that the value of errors is almost not change which can ignore about it.

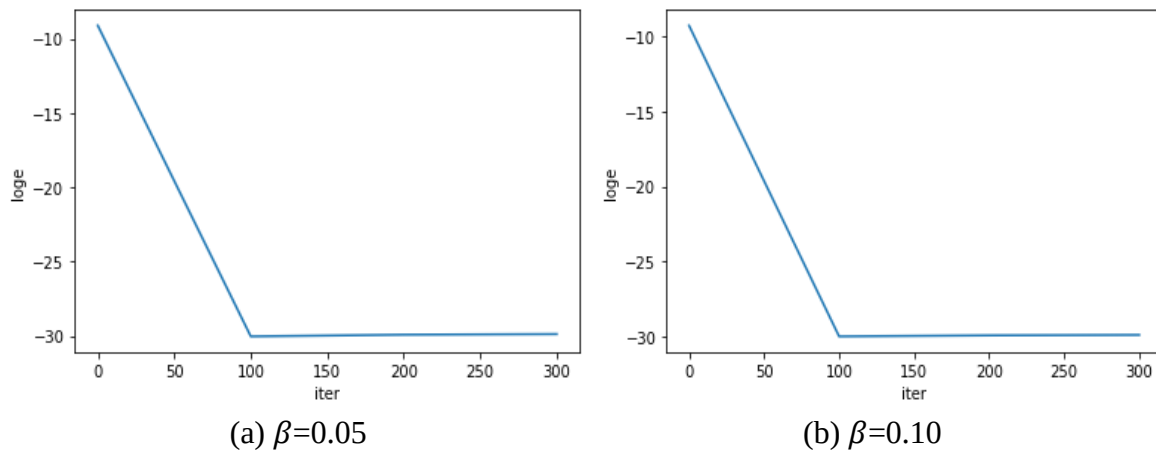


Figure 14. The figures of error against iteration with different β

In all, we will have these results:

Boundary are shown below (Figure 15):

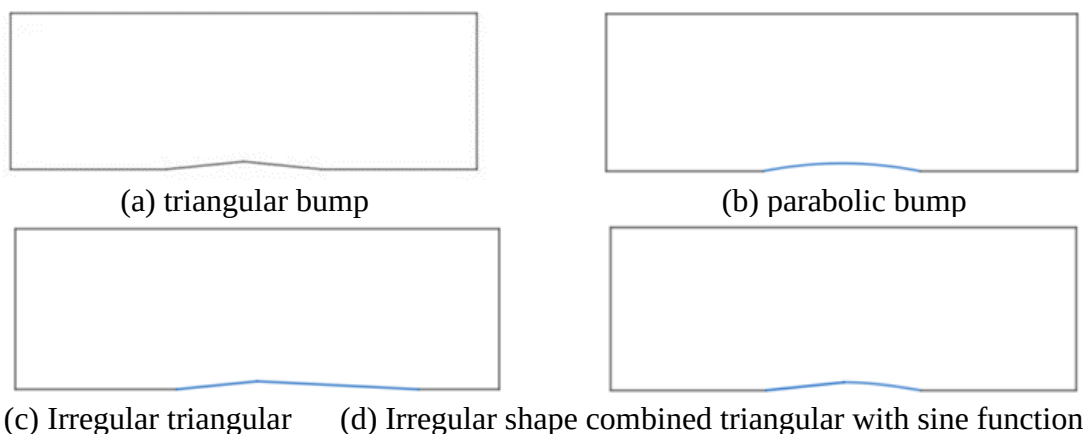


Figure 15. some shapes mentioned before

The log(error)-iteration figures are shown as Figure 16-19 below:

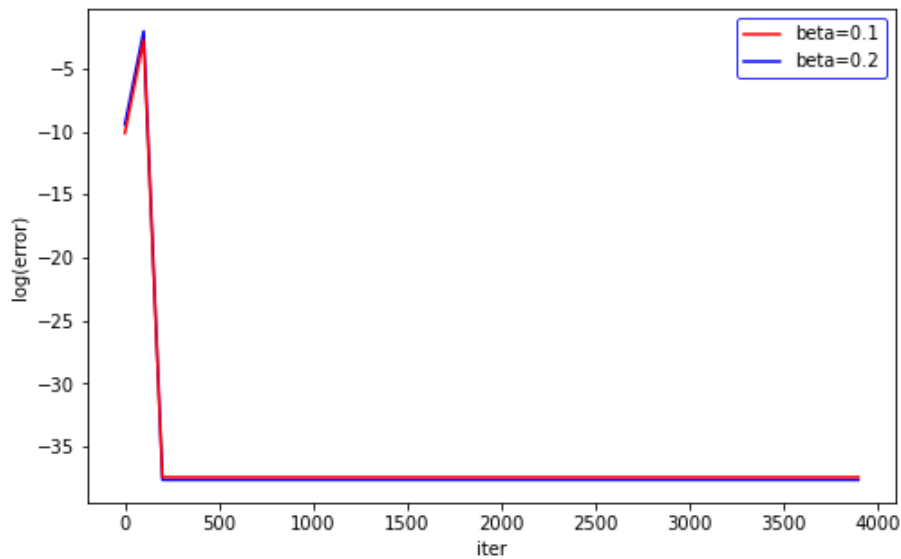


Figure 16. triangular shape

Red line shows conditions when $\beta = 0.1$ and blue line shows conditions when $\beta = 0.2$. They are very close.

Figure 17 shows the log(error)-iteration plot in parabolic shape.

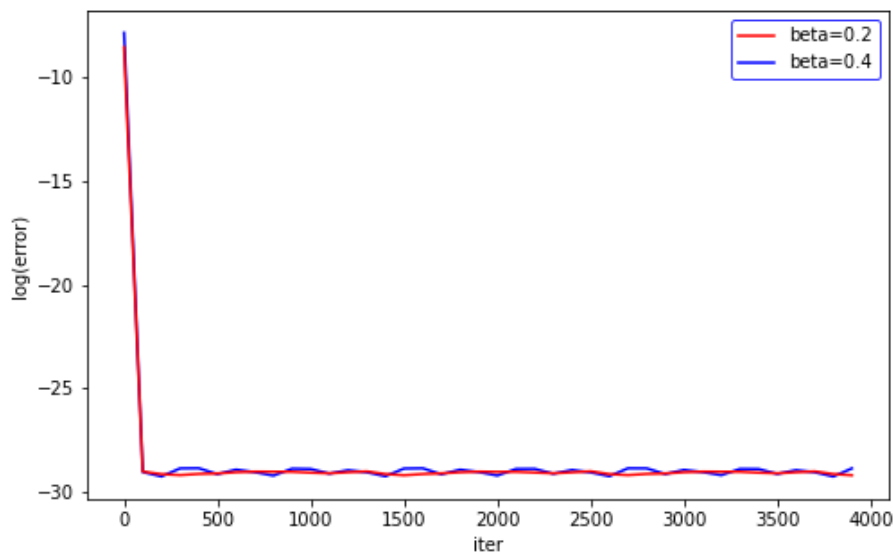


Figure 17. parabolic shape

In Figure 17, red line and blue line present the conditions when $\beta = 0.2$ and $\beta = 0.4$ respectively. Through comparing, it is clear to observe the differences between these two conditions are negligible.

Figure 18 below shows the log(error)-iteration plot in irregular triangular shape.

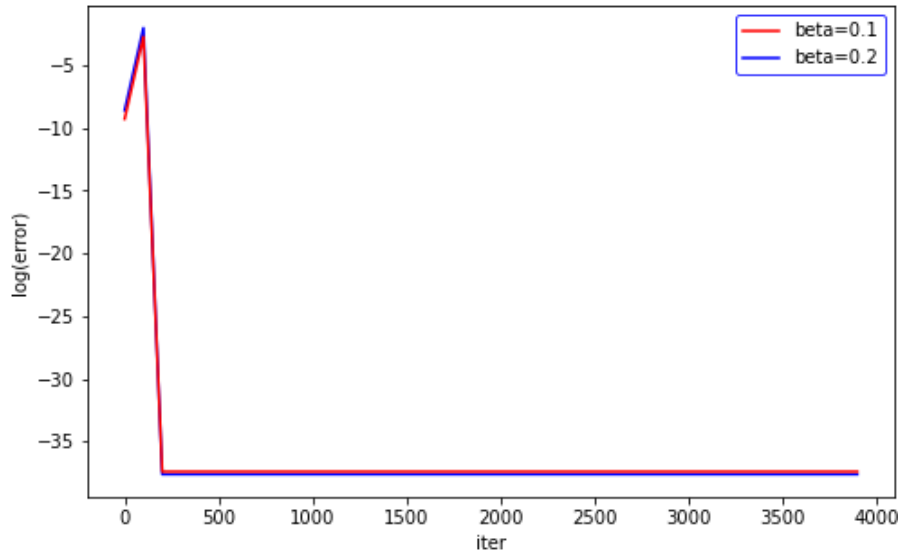


Figure 18. Irregular triangular shape

Red line shows the condition when $\beta = 0.1$ and blue line shows the condition when $\beta = 0.2$.

Figure 19 shows the log(error)-iteration plot in Irregular shape combined triangular with sine function.

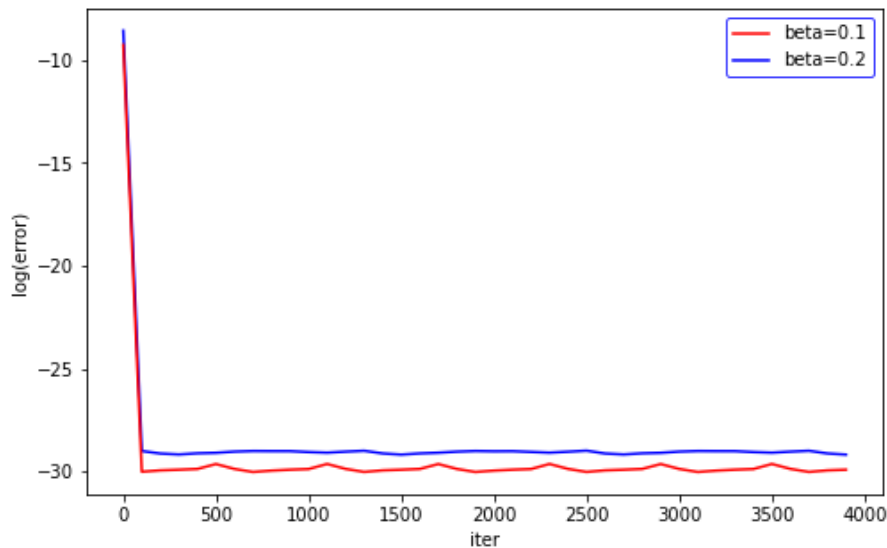


Figure 19. Irregular shape combined triangular with sine function

Red line illustrates the conditions when $\beta = 0.1$ and the blue one shows conditions when $\beta = 0.2$.

4. Conclusions and future work

In this paper, CFD is adopted to study whether the shape of the bump will influence the flow of the fluid. The main conclusions can be summarized as follows:

1) When increasing the value of β , the streamlines become more fluctuant. When the distance to the boundary is closer, this kind of effects becomes more extremely. This uncovers that the fluid near the boundary will change more obvious than other part.

2) For the same height (0.05\0.1) of four different shape bump, the streamlines of irregular shape have the most obvious change, while the combined shape gets the smallest change. The result shows the similar trend when the β equals to 0.1.

3) For every situation, the error all goes down to 0. Because increasing iteration means increase accuracy, it is not difficult to explain this phenomenon. This is also an important part to judge whether the code is correct.

In terms of future work, more shapes should be included into our research. Besides, more physic dynamics should be measured, such as pressure, velocity field. As a result, the python code should be added for these functions.

References

- [1] Blocken, Bert, "Computational Fluid Dynamics for Urban Physics: Importance, Scales, Possibilities, Limitations and Ten Tips and Tricks towards Accurate and Reliable Simulations.," *Building and Environment*, 91 (2015): 219 - 45. Web.
- [2] Malekjani, Narjes, and Seid Mahdi Jafari, "Simulation of Food Drying Processes by Computational Fluid Dynamics (CFD); Recent Advances and Approaches," *Trends in Food Science & Technology*, 78 (2018): 206 - 23. Web.
- [3] Malekjani, Narjes, and Seid Mahdi Jafari, "Simulation of Food Drying Processes by Computational Fluid Dynamics (CFD); Recent Advances and Approaches," *Trends in Food Science & Technology*, 78 (2018): 206 - 23. Web.
- [4] Zhonghua Zhang, Lingjie Zeng, Huixian Shi, Hua Liu, Wenjun Yin, Jun Gao, Lina Wang, Yalei Zhang, Xuefei Zhou, "CFD studies on the spread of ammonia and hydrogen sulfide pollutants in a public toilet under personalized ventilation," *Journal of Building Engineering*, Volume 46, 2022, 103728, ISSN 2352 - 7102.
- [5] Odi Fawwaz Alrebi, Bushra Obeidat, Ibrahim Atef Abdallah, Eman F. Darwish, Abdulkarem Amhamed, "Airflow dynamics in an emergency department: A CFD simulation study to analyse COVID-19 dispersion," *Alexandria Engineering Journal*, Volume 61, Issue 5, 2022, Pages 3435 - 3445, ISSN 1110 - 0168.
- [6] Baigang, Mi. "A CFD-CSD Coupling Method for Simulating the Dynamic Impact and Expulsion of Fragile Foreign Objects from the 'inlet-bypass Duct' Junction of a Turboprop Aircraft," *Engineering Applications of Computational Fluid Mechanics*, 16.1 (2022): 73 - 94. Web.