



SZÉCHENYI ISTVÁN UNIVERSITY
AUDI HUNGARIA FACULTY OF
VEHICLE ENGINEERING

TDK-THESIS

Machine learning-based approach in calculating pressure losses in Venturi meters

Levente Bodnár
Vehicle Engineering BSc

Supervisor:

Antal Bakonyi

Audi Hungaria Department of
Whole Vehicle Engineering

External supervisor:

Bence Péter Bocsi

AUDI Hungaria Zrt

Győr, 2024

Abstract

The importance of applied Fluid dynamics is a key factor to consider in the automotive, aviation or building service engineering industry. Computational Fluid Dynamics (CFD) is a simulation method which accurately predicts and calculates the fluid flow and its properties in a computer created environment. Furthermore, these simulations help engineers collect unmeasurable data. CFD is used for Research and Development (R&D) of new parts or constructions but also for predicting and preventing disasters.

On the other hand, as the R&D and technology advances, more and more complicated models are required to accurately simulate a scenario. These models require a huge amount of computational power and many hours of computational time. For this purpose engineers are working on alternative models that can simplify the complex geometries and reduce computational effort. Such models are surrogate-based models and data-driven models that accelerate the simulation process while maintaining small error in the results.

In the case of simulating closed pipe systems, CFD software have the capability of calculating the frictional losses in pipes. This phenomena occurs because of the friction between the fluid and the walls of pipes.

In my research I am investigating the losses in Venturi tubes with different geometries and with different mass flow rate of water in the system. For this purpose I use three different models: CFD, analytical and data-driven models. For closed pipe system simulation I choose Ansys Fluent software. These exported results serve as the learning database for the data-driven models in the Odyssee A-EYE software.

In the end, I compared the 3 methodology and found that the above mentioned neural network can be used as an alternate and time saving method for flow prediction in a Venturi tube.

Kivonat

Az alkalmazott áramlástan kulcsfontosságú tényező az autóiparban, a repülésben vagy akár az épületgépészetben. A numerikus folyadékdinamika (CFD) olyan szimulációs módszer, amely pontosan megjósolja és kiszámítja a folyadékáramlást és annak tulajdonságait egy számítógéppel létrehozott környezetben. Továbbá ezek a szimulációk segítenek a mérnököknek a mérhetetlen adatok gyűjtésében. A CFD-t új alkatrészek vagy konstrukciók kutatásához és fejlesztéséhez (K+F), de katasztrófák előrejelzésére és megelőzésére is használják.

A Kutatás és fejlesztés részleg előrehaladásával és a technológia fejlődésével egyre bonyolultabb modellekre van szükség egy jelenség pontos szimulálásához. Ezek a modellek hatalmas számítási teljesítményt és sok órányi számítási időt igényelnek. E célból a mérnökök olyan alternatív megoldáson dolgoznak, amelyek egyszerűsíthetik a bonyolult geometriákat és csökkenthetik a számítási igényeket. Ilyen modellek a helyettesítő modellek és az adatvezérelt modellek, amelyek felgyorsítják a szimulációs folyamatot, miközben az eredményekben csak kis hiba keletkezik.

Zárt csőrendszerek szimulációja esetén a CFD-k képesek a csövek súrlódási veszteségeinek kiszámítására. Ez a jelenség a folyadék és a csövek falai közötti illetve a folyadékrészecskék közötti súrlódás miatt következik be.

Kutatásomban a Venturi-csövek veszteségeit vizsgálom különböző geometriájú csövekben és a rendszerben átáramló különböző tömegáramú víz mellett. Ehhez három különböző modellt használok: CFD, analitikus és adatvezérelt modelleket. A zárt csőrendszer szimulációjához az Ansys Fluent szoftvert választom. Ezek az exportált eredmények szolgálnak tanulási adatbázisként az Odyssee A-EYE szoftverben lévő adatvezérelt modellekhez.

Végül összehasonlítottam a 3 módszertant, és megállapítottam, hogy a fent említett neurális hálózat alternatív és időtakarékos módszerként használható az áramlás előrejelzésére Venturi-csőben.

Contents

1. Introduction.....	5
2. Literature review	6
2.1. Computational simulations	6
2.2. Bernoulli's principle	7
2.3. Navier-Stokes equation.....	7
2.4. Comparison between confuser and diffuser	8
2.5. Venturi tube	9
3. Benchmark of the CFD software	10
3.1. openFoam.....	10
3.2. Ansys Fluent	11
3.3. Simcenter STAR-CCM+ CFD.....	11
3.4. Autodesk CFD 2024	11
3.5. SimFlow	12
3.6. Conclusion	12
4. Test study of the simulation	13
4.1. Geometry	13
4.2. CFD Simulation	14
4.3. Hexagon Odyssee	15
5. Results.....	16
6. Conclusion	25
References	26

1. Introduction

As technology advances more and more complex geometries surface and evolve in industry. Unfortunately there is not an accurate analytical model for every problem if they become too complex. Therefore some kind of numerical solution is needed. The most accurate way of calculating fluid flow around these difficult to numerically model geometries are Computational Fluid Dynamic simulations. The software uses some kind of Solver that approximates and leaves only the setup the boundary conditions for the user. These solvers can vary and use different formulas for the calculations. The computer does all the calculations. Unfortunately, the more complex the shape the harder the calculations are and the slower the solution is. These calculations require expensive equipment to run acceptably fast. For this purpose engineers are looking for methods that help reduce time and computational needs.

Such methods vary, for example some engineers from Volkswagen are currently working on an Artificial Intelligence accelerated CFD which speeds up the design and simulation process by modifying the geometry as the simulation runs. This ensures a parallel design and validation and speeds development processes up.[1]

Other methods use model reduction that replicate a similar result much faster. The drawback of these models is that they contain a small margin of error due to the reduction. The goal is to replicate the complex geometry with minimal error in the solution. The Reykjavik University together with the Iowa State University run a study on Surrogate-Based Airfoil Design with Space Mapping and Adjoint Sensitivity.[2]

Another method which uses Artificial Intelligence-aided models is a neural network based. These models use AI to predict and simplify the geometry. There are many neural networks and all of them are good for a different application. Sadik Yetkin et al. worked on a comparison article of the different neural networks on predicting aerodynamic coefficients.[3]

My topic is investigating the losses in Venturi tubes with different geometries and with different mass flow rate of water in the system. Using 3 calculation models: Analytical, Data-driven, and CFD model. Applying the CFD model as the base of the machine learning and as the most accurate one of the three, I compare the two other methods to find an accurate and fast solution and an alternative instead of the long CFD simulation.

2. Literature review

Venturi tubes are widely used for different purposes for example in a sandblasting gun or in cryogenic facilities as a measuring tool.[4] Design optimization of these shapes is a long process and requires a huge amount of computational time.

For overcoming the large computational time of optimal design or to shorten the situational efforts of CFD simulations there are several methods to substitute the process Jinrui Xiao Et al. from the School of Mechanical and Electrical Engineering, Guangzhou University used neural network for optimizing the design of a Venturi tube.[5] Sadik Yetkin*, Saleh Abuhanieh and Sahin Yigit from Turkey tested several AI. models for predicting aerodynamic coefficients. They found that regression trees; CatBoost, Bagging and XGB can reduce the number of CFD simulations needed for calculation.[3]

2.1. Computational simulations

Nowadays, computational simulations have a wide range of utilizations. Notably, in the field of mechanical engineering, these simulations play an important role in the exploration of the properties of elastic solids. They are crucial in vehicle design, structural stability analysis, and in ongoing efforts to enhance structural integrity. They are also important in chemical applications, numerical simulations serve as a valuable tool for optimizing reactions involving various substances, as seen in processes like combustion.[6]

With the above mentioned widespread on applications numerical solutions became very popular as an inexpensive tool of predicting scenarios and helping in the research and development processes as well as supervision of already existing designs. For example Computational Fluid Dynamic (CFD) simulation excels in its field. These simulations and calculations are based on principles and equations like the Navier-Stokes equation or Bernoulli's principle and accurately describe the fluid motions and interaction in real life simulations. Two dimensions and three dimensions is also useable and accurate.

In the world of motorsport like Formula 1 for example the amount of time spent on measurements is very limited and CFD simulations are the best tool to experiment and collect data.

2.2. Bernoulli's principle

Bernoulli's principle is a key phenomenon in fluid dynamics. This equation creates a relation between velocity of the fluid and the pressure of the fluid. This can be derived from Newton's second law of motion and the equation will look like as follows. [7]

$$\frac{v^2}{2} + gh + \frac{p}{\rho} = \text{constant}$$

This is the idea used for Venturi meters. This method is good in closed pipe systems where the fluid is incompressible the height difference of the fluid columns will give us the pressure difference in-between the two measurement points. Other applications of this theory can be for lift force calculations of airfoils or pitot tubes. Also carburetors use this phenomena to draw more air and fuel into the system.

2.3. Navier-Stokes equation

Navier-Stokes partial differential equation which describes the motions of viscous fluids. This is the most important equation for CFD simulations as it lets the computer understand fluid motions and interactions. By using this the computer is able to calculate for example velocity and other values and also predict and visualize flow patterns and turbulence.[7]

In my case the following Navier-Stokes equation is used for incompressible fluid with constant density and viscosity:

$$\frac{Du}{Dt} = \nu \nabla^2 u - \nabla \frac{p}{\rho} + \frac{1}{\rho} f$$

2.4. Comparison between confuser and diffuser

The two main parts of a Venturi tube are a confuser and a diffuser. These are conical shaped pipe parts with changing diameter sizes. A confuser is a constantly narrowing tube. This characteristic applies the Bernoulli principle. A diffuser is the opposite, it has a constantly widening radius and acts as the opposite way (**Fig.1**). As the diameter gets narrower the velocity of the fluid speeds up and the pressure drops. If we turn them and connect them with their smaller diameter side we achieve a Venturi tube.

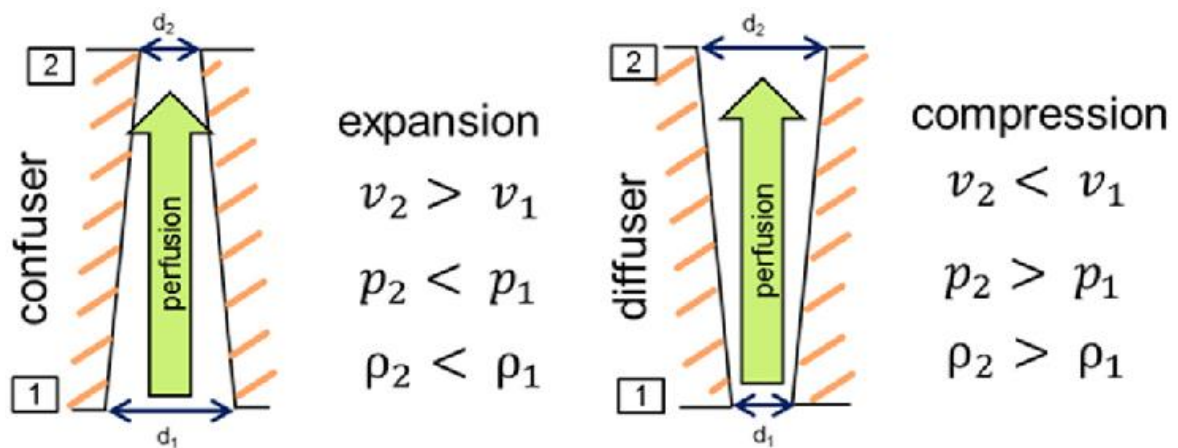


Fig.1: Comparison between confuser and diffuser[8]

2.5. Venturi tube

The simplest apparatus is a tubular setup called a Venturi tube or Venturi. The fluid flows through a length of pipe of changing diameter. To avoid undue aerodynamic drag, a Venturi tube typically has an entry cone of 20 degrees and an exit cone of 5 degrees. **(Fig.2)**

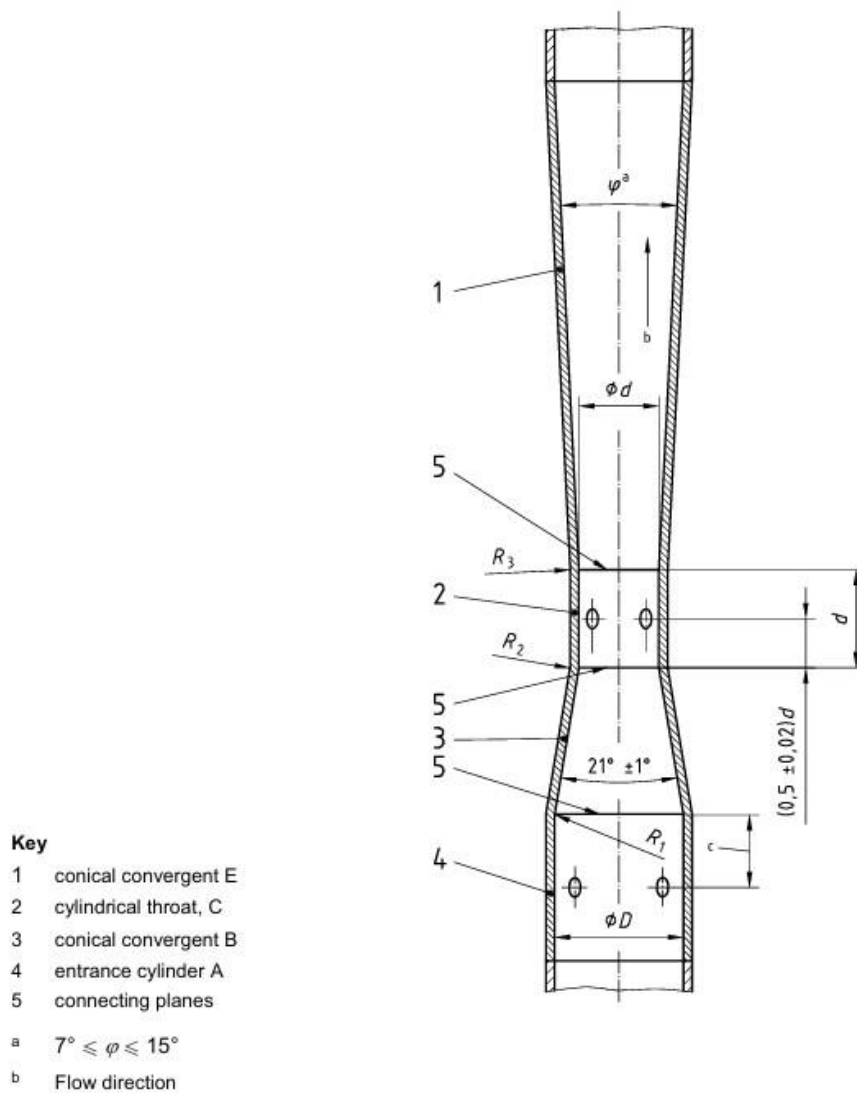


Fig.2: ISO_5167_4_2003 Venturi-tube standard[9]

3. Benchmark of the CFD software

To have accurate simulations and to understand if the results are real basic predetermined tutorial problems were run. These simulations had checked results and could be revised.

Since CFD software is a must in the industry there are many different versions and brands. To choose the best one for the research a benchmark problem was run in the following software and descriptions and documentations were read: openFoam, Ansys Fluent, Simcenter STAR-CCM+ CFD, Autodesk CFD 2024 and SimFlow. This benchmark was created with the sake of finding the best software specifically for this problem. The software had to meet the following criteria. The simulations had to be done in no more than an hour for the sake of reproducibility. Had to be available for educational use or had to be a freeware software. Other notes were easy to use interface and good helpline.

3.1. openFoam

This was the first software tried and this had the biggest impact on the learning curve. OpenFoam is a Linux based CFD software which mainly operated in the command prompt of the computer and requires a fair amount of programming knowledge.

The fact that it is Linux based means it is open source and free software. Also, it's a very well balanced program in terms of hardware requirements. Not too demanding and gets the job done.

On the other hand it is a Linux based program it is hard to install on a Windows PC. Also the user interface is almost not existing hard to set up and not a user-friendly program. It is not a good program to start learning to run CFD simulations.

3.2. Ansys Fluent

This is an industry tested and used software. It is made for Windows and is easy to operate.

This is a very complex software and very accurate while the interface and geometry integration is simple to use even for complex components. Ansys also has a good support and help line with frequently asked questions or we can ask our own. Besides all the pros, the hard to use interface difficult to start as a beginner and requires Ansys discovery for a simpler use. Also it is mainly designed for high budget companies and researchers and is demanding in terms of hardware.

3.3. Simcenter STAR-CCM+ CFD

This is an industry leading software used in the biggest companies and motorsport teams everywhere. The best program on the market very complex very good user interface and can handle any task.

However this is the most demanding in terms of hardware and also very expensive and hard to obtain software. Unfortunately no student nor free trial version available at the moment.

3.4. Autodesk CFD 2024

This is an American made software used mainly for university studies and maybe American companies. It has free student version without limitation. Easy to use and easy to learn interface and setup sequence. Good helpline with many tutorials for learning .Furthermore, Autodesk has a big community for help and sharing problems. It has integrated connection with Autodesk CAD (Computer Aid Design) software. On the contrary, because of the many graphical interface and visually appealing look it is a very computer demanding software.

3.5. SimFlow

This is an openFoam based CDF that has a friendly and easy to use interface and built in CAD design part for simple geometries. Although, openFoam is open-source this software only has a limited node number free version. It is easy to use and hardware friendly too.

Yet the 200000 mesh node limit in free version is not. Complex problems require a small inaccurate mesh or a symmetry to be calculated. Also not a lot of help or support for problem solving and on how to setup correctly a simulation.

3.6. Conclusion

The following Table represent the results of the benchmark and determines the best software for the research (**Table 1**). As the scores represent the best one of the five is the Ansys Fluent for this case of predicting pressure loss Venturi tube simulation.

Table 1: Benchmark points of different CFDs'

	OpenFOAM 	Ansys Fluent 	Simcenter StarCCM+ 	Autodesk CFD2024 	SimFlow 
Hardware requirement	Linux subsystem 4	Compatible with Win but demanding 6	Compatible with Win but demanding 6	Compatible with Win but demanding 6	Compatible with Win not demanding 10
License - Limitations	Open-source No limitations 10	Student License 512000 nodes 8	No free nor student version 1	Free student version no limitations 10	Student version limited to 200000 nodes 5.5
Compatibility	Exportable and importable data 9	Exportable and importable data 9	Exportable and importable data 9	Exportable and importable data 9	Exportable and importable data 9
Ergonomy	No user interface difficult to use 2	Difficult easy to use interface and design acceleration 10	Easy and friendly interface 8	Easy and friendly interface 10	Easy and friendly interface 10
Complexity	Great for simple simulations 5	Good for simple and complex sims 10	Can be used for almost any problem 10	Can be used for both 7	Can be used for both 8
Overall	6	8.6	6.8	8.4	8.5

4. Test study of the simulation

4.1. Geometry

The Geometry used for the simulations was created according to ISO 5167-4 standard of Venturi tubes. I choose 5 different inlet diameters (10mm, 20mm, 30mm, 40mm, and 50mm) and used the third of the inlet diameter as the choke diameter of the Venturi meters. (**Fig.3**) These Computer Aid Design models were then exported and using Ansys Discovery meshed and simulated.

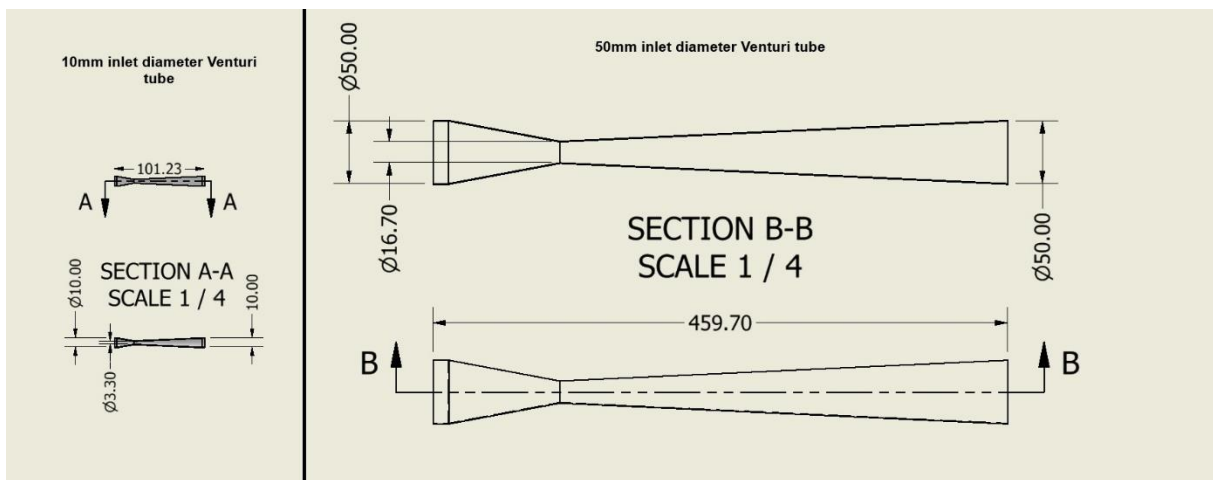


Fig.3: Smallest 10mm diameter Venturi tube (left) and biggest 50mm Venturi tube (right)

4.2. CFD Simulation

For the simulations Ansys Fluent and for the meshing and setup Ansys Discovery was used. **(Fig.4)** The following conditions were used. The geometry has an adaptive mesh and it is adjusting while the calculations are running for the most optimal grid and also for a short runtime. For the sake of simplicity and easy reproduction I opted for steady state simulations without temperature change and gravity. For the Venturi tubes non-slip walls were chosen. This means that a boundary layer will form and the pipe will act close to reality. **(Fig.5)** For further calculations the result values were exported.

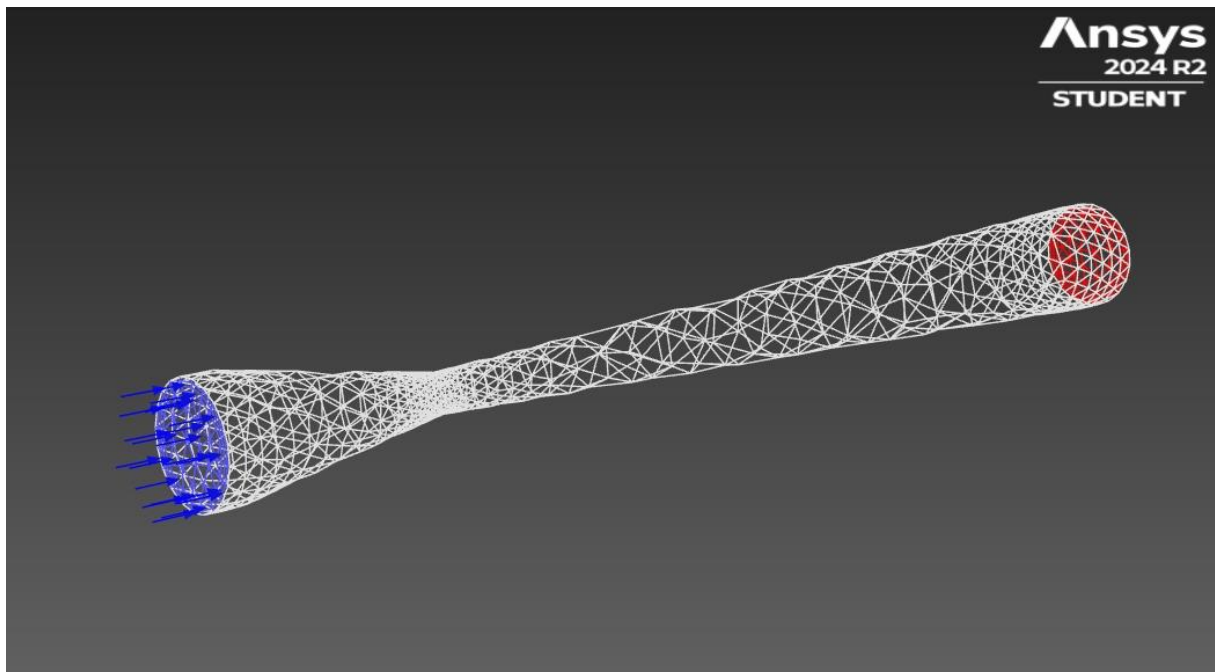


Fig.4: Visualization of the mesh of the 10mm inlet diameter Venturi tube

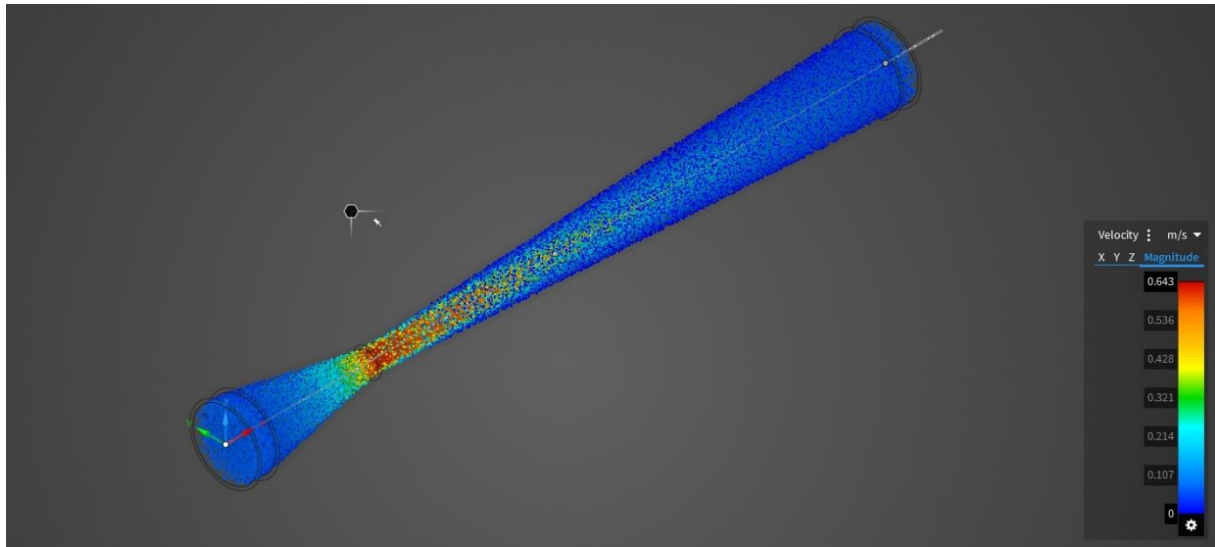


Fig.5: Velocity particle map of $1e-3$ flow rate in 10 mm inlet diameter Venturi tube

4.3. Hexagon Odyssee

For the data-driven model the built in “SIN-COS Function” mode of the software was used. This mode is a data-driven function fitting. (**Fig.6**) Although, this was a predetermined mode AEYE-manager module was needed to fine-tune it to fit my preferences. In the setting an extra input variable had to be added to make it two input function. Later for a more accurate result I started fine-tuning the calculations. INVD as well as RBF and pod-RBF calculating methods were tested.

Names (Optional)	Diameter	m_dot	Delta p
Load the names	Select values	Select values	
Test1	20	2	1362.57
Test2	20	4	5191.97
Test3	30	3	1446.46
Test4	40	2	-275.85
Test5	40	4	-84.71

Fig.6: Interface of the ODYSSEE A-EYE software

5. Results

In this section the results of the research will be presented. The results of the researched flows and diameters in the CFD software are predicted by a data-driven model in Hexagon ODYSSEE A-EYE. I created 25 CFD simulations to serve as the base point of the research and tried to replicate these results with the data-driven model as well as an analytical model. I used 20 resultant data as the learning database for the ODYSSEE model and the remaining 5 to create predictions.

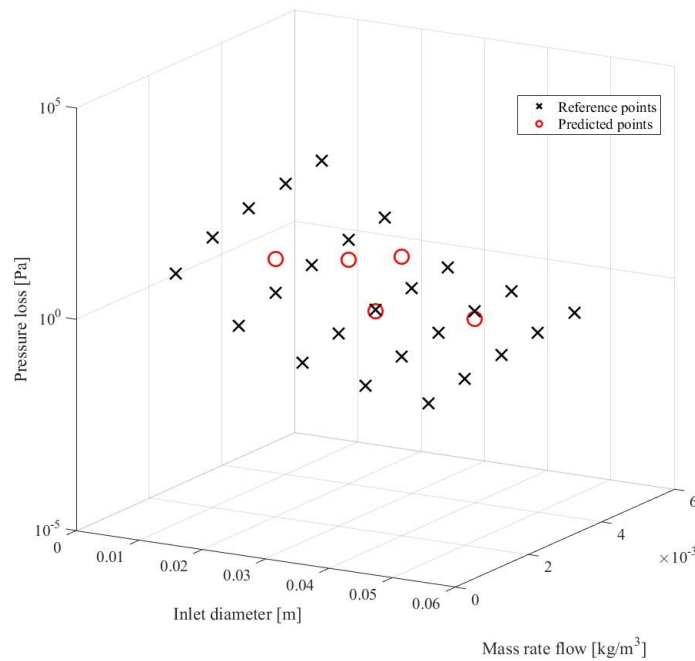


Fig.7: Plotting of the CFD and predicted data-driven values with INVD method and power parameter raised to 8

On **Fig.7** I am showing the INVD method with default settings. Here the power parameter is 2 and the distance parameter is also 2. In this case the mean square error value is 215,7171.

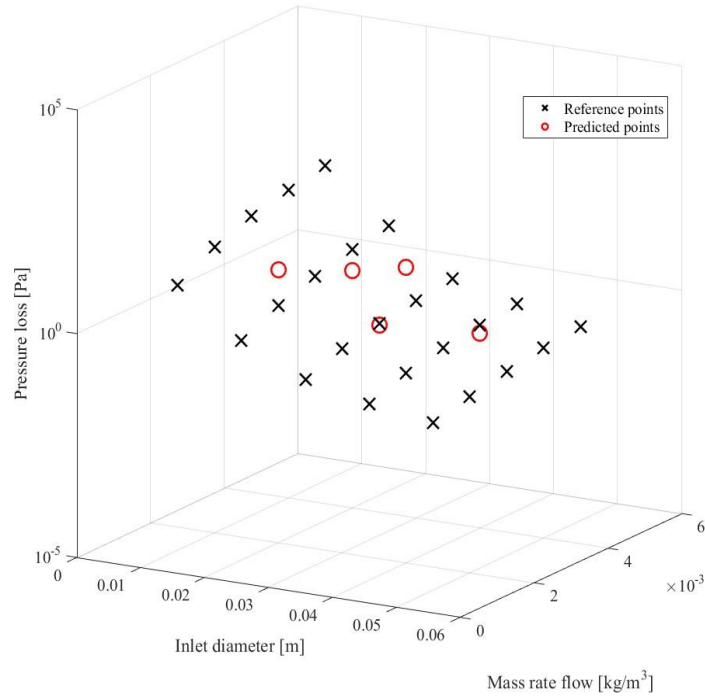


Fig.8: Plotting of the CFD and predicted data-driven values with default INVD method

In **Fig.8** the INVD method with power parameter changed to 8 is shown. Here the power parameter is 8 and the distance parameter is also 2. In this case the mean square error value is 215,717. As the results are the same it shows that in our case changing the power parameter does not affect the result.

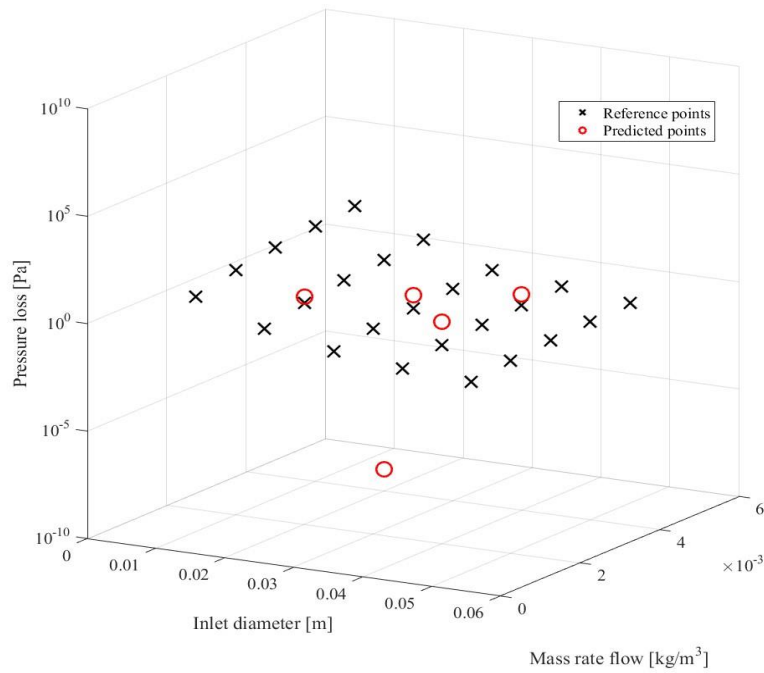


Fig.9: Plotting of the CFD and predicted data-driven values with default RBF method

In this case (**Fig.9**) I used the default RBF method to predict the calculated pressure loss. I used default RBF which uses Gaussian process and the sigma value is set to 0. In this case the error value is 11.758. This projects that the RBF method is better at predicting but can give negative values and those are not correct and real results. Unfortunately in the graph I could not handle negative due to the logarithmic approximation so the negative values are represented with a close to 0 number.

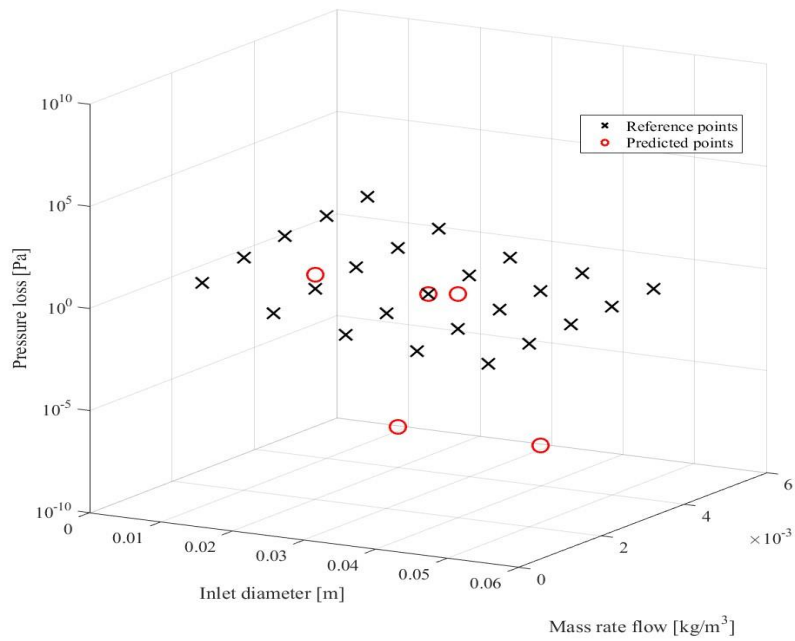


Fig.10: Plotting of the CFD and predicted data-driven values with RBF method where the sigma is -6

In the case of **Fig.10** I experimented with changing the sigma value of the default RBF method to -6. This gave me a worse result as it predicted more negative values than the default. The error value is 9.1236 which is smaller even with the two negative values.

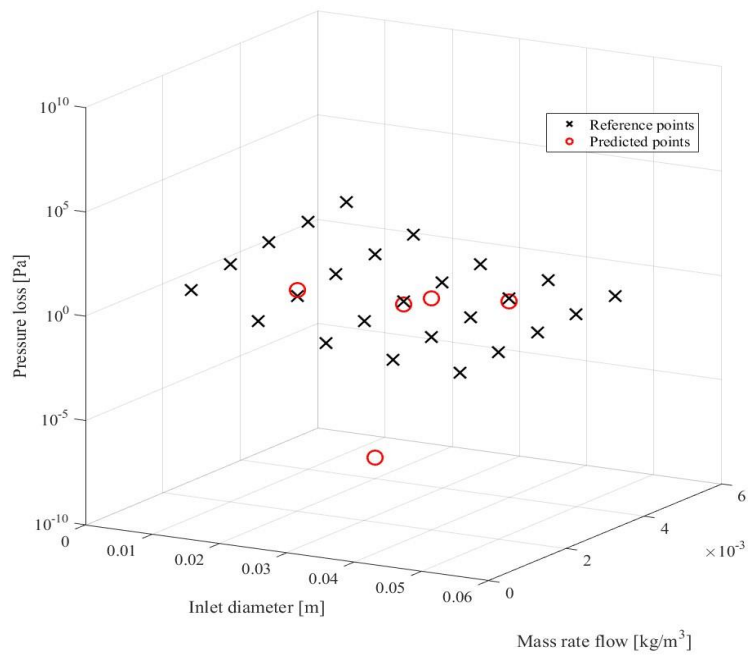


Fig.11: Plotting of the CFD and predicted data-driven values with RBF method where the sigma is 0.3

For (**Fig.11**) this case I changed the sigma parameter to 0.3 which gave an even better result as inly one negative value appeared and the error value of 4.1172.

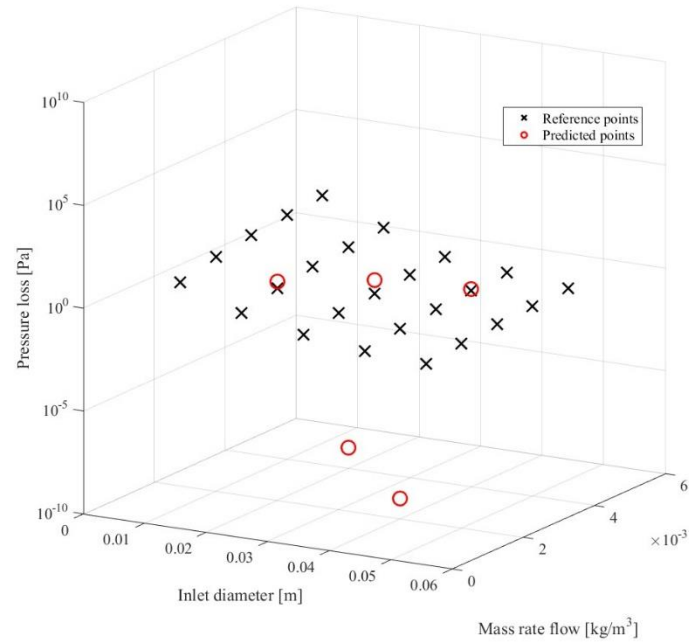


Fig.12: Plotting of the CFD and predicted data-driven values with RBF method where the sigma is 0.6

In the case when I raised the sigma value to 0.6 in the RBF method (**Fig.12**) I realized this is not the best route to go on in fine-tuning the model. The error value raised to 15.9694 and I got two negative values as the plot shows.

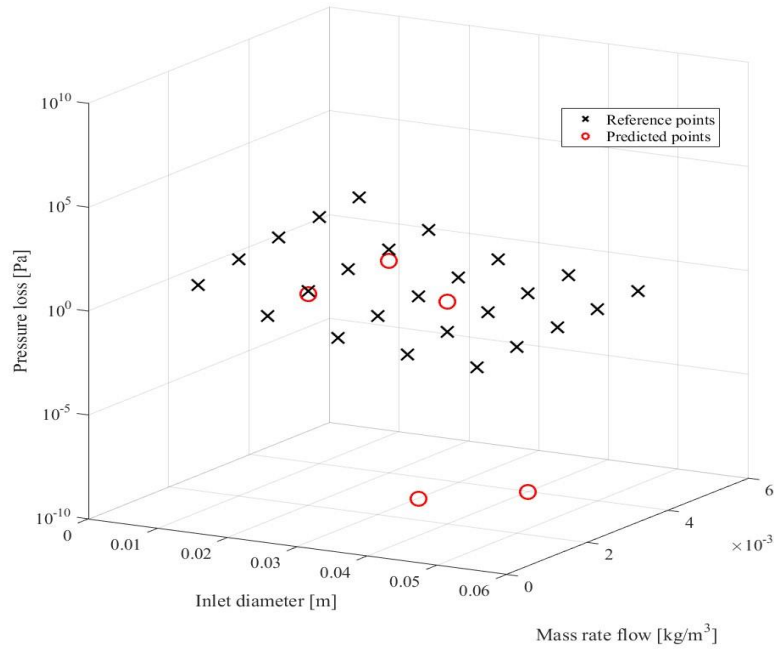


Fig.13: Plotting of the CFD and predicted data-driven values with RBF method where the sigma is 3

I also tried to set the sigma parameter to 3 but this even though gave the smallest error value 4.1402 it predicted two negative results which cannot occur in the real world. (**Fig.13**)

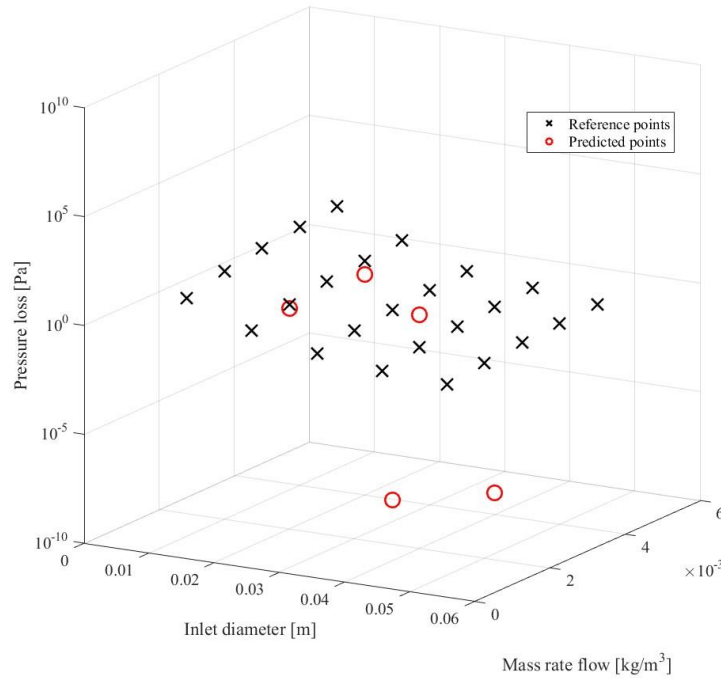


Fig.14: Plotting of the CFD and predicted data-driven values with POD_RBF method where the sigma is 6

I also tried a different RBF method called POD_RBF and changed the sigma value to 6 as well. (**Fig. 14**) This gave again two negative results but had a really accurate prediction in the other regions and resulted in an error value of 3.5398.

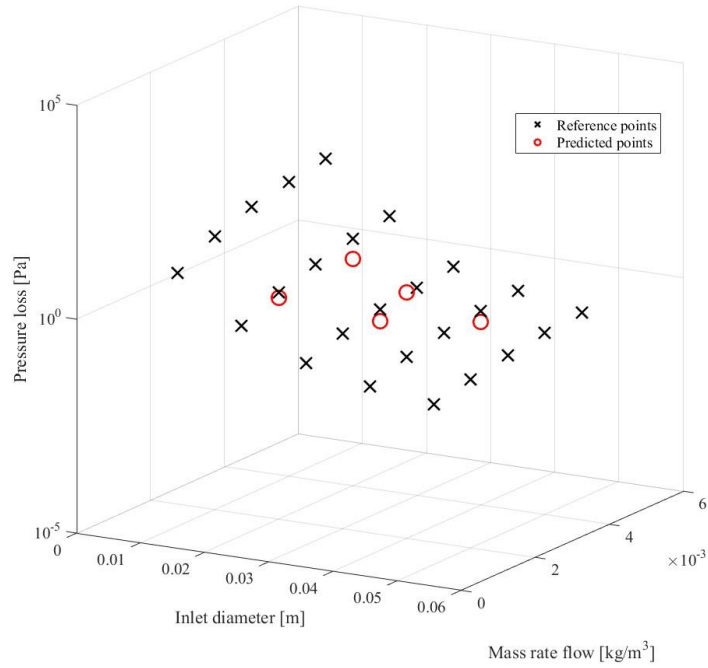


Fig.15: Plotting of the CFD and predicted data-driven values with POD_RBF method where the sigma is 15 and the mod parameter is 3

This final case (**Fig. 15**) seemed to be the best of all as it had no negative results and the error value is 3.0554. This method seems like the best approach at the moment although I think it still could be more accurate with a better tuning for this case of machine learning or with a more adjustable code.

6. Conclusion

In this thesis I investigated the losses in Venturi tubes with different geometries and with different mass flow rate of water in the system. For these CFD results I implemented a data-driven model to predict the results. After several different approaches I found that the POD_RBF method was the most accurate. However, the biggest drawback of RBF method is that it gives negative results which are completely invalid results as negative losses cannot occur. In the case of the drastically modified POD_RBF method the error value was 3.0554 which is the lowest of all. Also this case did not have negative resulting data too.

Unfortunately, Hexagon's Odyssee A-EYE does not offer too much customization in the mathematical code nor gives insight on how it works. For this problem in the future a handwritten code could work better. With that model I think a more accurate result can be achieved. Furthermore the data-driven and neural-network based model accuracy could be also tested and compared. Moreover this method can be compared with an empirical model like Hagen–Poiseuille equation based calculation. Also same test could be carried out for turbulent flow.

References

- [1] Nefi Alarcon, NVidia Developer, 2024.10.16. <<https://developer.nvidia.com/blog/volkswagen-accelerates-aerodynamics-concept-design-with-nvidia-v100-gpus-on-aws/>>
- [2] Yonatan A. Tesfahunegn, Slawomir Koziel, Leifur Leifsson and Adrian Bekasiewicz, (2015), Surrogate-Based Airfoil Design with Space Mapping and Adjoint Sensitivity, *Procedia Computer Science*, 51, 795-804
- [3] Sadik Yetkin, Saleh Abuhanieh, Sahin Yigit, (2024), Investigation on the abilities of different artificial intelligence methods to predict the aerodynamic coefficients, *Expert Systems With Applications*
- [4] Manfred Suesser, (2012), Performance of classical Venturi tubes for application in cryogenic facilities, *AIP Conference Proceedings*, Cryogenic Engineering Conference, Spokane, WA, USA, 2011.06.13-17., 1353–1362
- [5] Xiao, J.R., Liang, Z.W., Liu, X.C., Zhao, Z. and Xie, X.C., (2021), Design Optimization Analysis of Venturi Tube for Medium Conveying in Strengthen Grinding Process, *Engineering*, 13, 431-447
- [6] Santos Phd, Rômulo & Sales, Jorge, (2023), Mathematical analysis for simulation of incompressible fluid flow, *The Journal of Engineering and Exact Sciences*, 9. 12. 10.18540/jcecvl9iss12pp17392.
- [7] Frank M. White, *Fluid Mechanics*, 7th edition, McGraw-Hill, New York, NY, USA, 2011, 885
- [8] Zahedi, Ehsan & Woerz, Christoph & Reichardt, Gerd & Umlauf, Georg & Liewald, Mathias & Barz, Jakob & Weber, Rudolf & Förster, Daniel & Graf, Thomas. (2019). Lubricant-free deep drawing using CO₂ and N₂ as volatile media injected through laser-drilled microholes. *Manufacturing Review*. 6. 11. 10.1051/mfreview/2019011.
- [9] ISO, Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full — Part 4: Venturi tubes, ISO 5167-4, Geneva, Switzerland, 2003