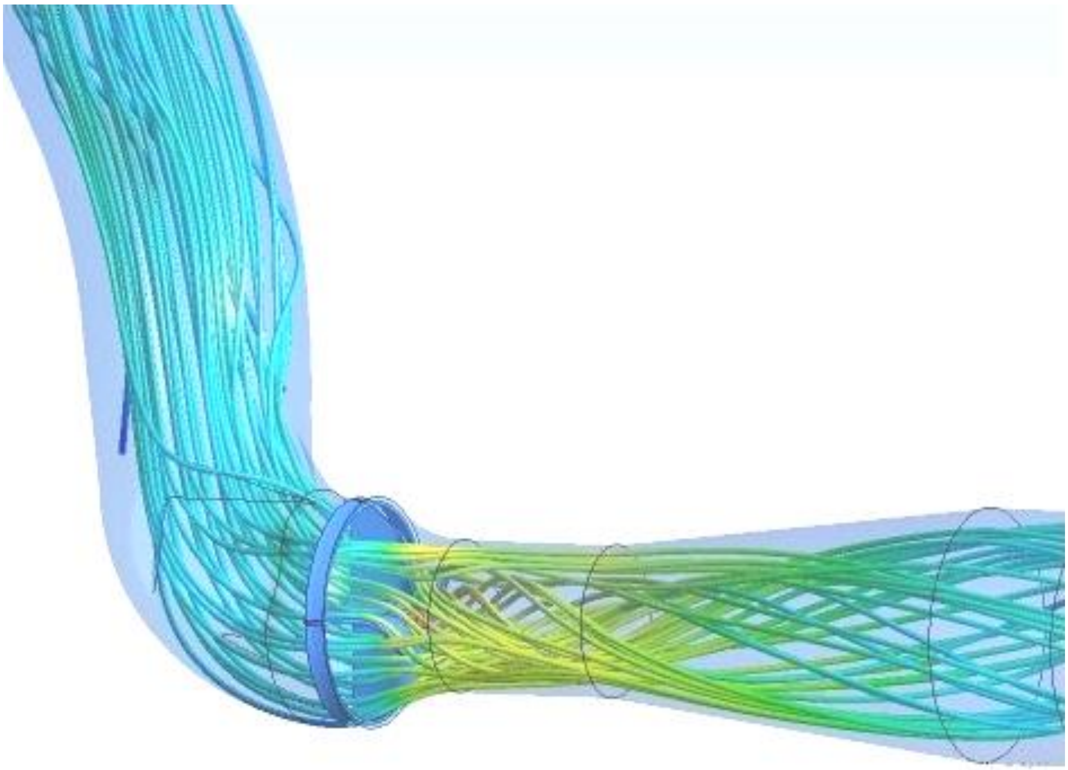
CHALMERS



Swirling Airflow Influence on Turbocharger Compressor Performance

Master's Thesis in Automotive Engineering

JUSTINAS PECIURA

Department of Applied Mechanics
Division of Combustion
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2014

Master's Thesis 2014:80

MASTER’S THESIS 2014:80

Swirling Airflow Influence on Turbocharger Compressor Performance

Master’s Thesis in the Automotive Engineering

JUSTINAS PECIURA

Department of Applied Mechanics
Division of Combustion
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2014

Swirling Airflow Influence on Turbocharger Compressor Performance
Master's Thesis in the Automotive Engineering
JUSTINAS PECIURA

Master's Thesis 2014:80
ISSN 1652-8557
Department of Applied Mechanics
Division of Combustion
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Cover:
Computational Fluid Dynamics simulation of swirl generating device.

Chalmers Reproservice
Göteborg, Sweden 2014

Swirling Airflow Influence on Turbocharger Compressor Performance
Master's Thesis in the Automotive Engineering
JUSTINAS PECIURA
Department of Applied Mechanics
Division of Combustion
Chalmers University of Technology

Abstract

Master Thesis work was done at Volvo Cars, Powertrain Installation department, Air Induction System group. Scope of the thesis was to investigate clean side duct and optimize it for compressor performance. After literature review work was narrowed down and focused on swirling airflow and its influence on turbocharger compressor performance from efficiency and NVH point of view.

Throughout thesis work CFD simulations were used to design swirl generators and investigate swirl behaviour in the ducts. Then, flow bench experimental tests were done to validate CFD simulations and define best settings for swirl modelling. Finally, experimental tests were done in turbocharger test rig to investigate swirl influence on compressor performance.

Thesis outcome was a set of guidelines for Air Induction System group, that covers swirl creation methods and swirl effect on compressor performance. Results showed that swirling motion can move compressor map improving surge or choke limit. In addition, small amounts of positive swirl can also improve compressor efficiency up to 0.8%.

DET ÄR JU BARA LUFT, HUR SVÅRT KAN DET VA?

Acknowledgements

Foremost I would like to express my gratitude to Henrik Nyberg and Niclas Bratt from Air Induction System, that gave me this opportunity to do Master Thesis work at Volvo Cars.

I would like to thank my supervisors Henrik Nyberg from Volvo and Sven B Andersson from Chalmers, also CAE Engineers Johan Brunberg, Oskar Bergman and all Air Induction System group for all the advices, guidance and support that was given to me throughout my work.

Also I would like to thank Ragnar Burenius from Volvo and Per Ericsson from SAAB/NEVS for guidance and all the experimental turbocharger tests that were done for my thesis work.

Justinas Peciura, Göteborg 04/7/14

Contents

1

Introduction

1.1 Background

Environmental requirements for internal combustion engines are reducing their size , safety requirements - reducing the size of the engine compartment, as a result modern - day engine has become highly limited in packaging space and size. At the same time fuel consumption and high power requirements from the customers are increasing. Common solution to overcome these challenges is to use air charging. Usually, engines with high power requirements can have more than one air charging device, such as turbocharger or supercharger. As a result air intake for internal combustion engines is becoming a complex system that includes number of highly complicated shape ducts.

For the same reasons turbochargers are also decreased in size. Reduction of turbocharger size narrows their range which is highly limited by surge and choke working conditions. Modern engine restrictions demand both a proper choice of turbocharger and Air Induction System, as they can significantly influence turbocharger compressor performance.

1.2 Scope

Scope of thesis was to investigate Air Induction System clean side duct. A duct which connects air filter box and turbocharger inlet and find out what are important air flow parameters for turbocharger compressor performance and how they are influencing it.

Literature review showed that investigating all flow parameters will not be possible to be done during the period of master thesis, therefore thesis work was narrowed down only to swirling airflow investigation.

Outcome of thesis work was a set of guidelines for Air Induction System group for beneficial application of swirling flow and initial ideas for swirl generation.

2

Literature review

LITERATURE REVIEW SUMMARIZES information found in research, articles and patents that has been done to investigate possible air intake system improvements and proposes promising improvement paths for further investigation.

2.1 Intake influence

2.1.1 Performance

A choice of turbocharger for the engine starts with matching their performance maps. Common issue also mentioned by [1, 2, 3] and [4] is that turbocharger compressor performance maps are experimentally measured using straight line intake with minimum pressure losses and high flow uniformity, which in real conditions are not common. Air inlet system can highly influence turbo compressor map and has to be taken into account when choosing right compressor. [1] experimentally tested turbocharger performance for compressor:

- without air inlet
- with clean side ducts
- full air inlet

Results showed that only clean side duct can reduce efficiency of turbo compressor performance by 3%.

Influence of clean side duct can be seen in the figure 2.1, air inlet pressure drop increases velocity of the flow, which pushes the compressor map to the left and reduces high mass flow performance. [5] has also confirmed that bigger air filter sizes can influence compressor performance. In his case bigger air filter (9 liters) was more beneficial then smaller (5 liters).

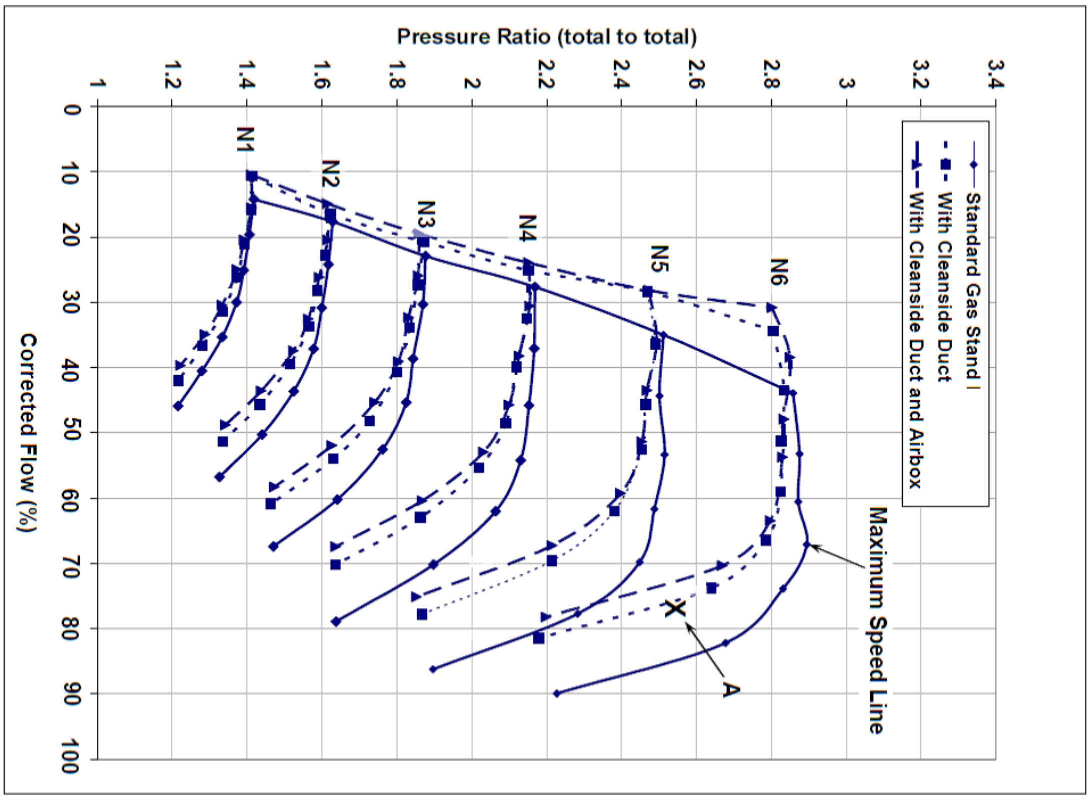


Figure 2.1: Intake system influence on turbocharger compressor performance map done by Ford

2.1.2 NVH

Turbocharger introduction into modern engines also added another noise source in the vehicle. Common turbocharger noise found in modern engine is so called whosh noise. Whoosh noise is a broadband flow-type noise typically evident under full load acceleration on turbocharged engine. Whoosh noise excitation is due to the generation of turbulence within the turbocharger compressor and commonly found when compressor is working in a region close to surge.

According to [6] whoosh noise is not simply encountered where compressor operation line is closest to surge line, therefore it is not useful to predict the level of whoosh noise by comparing these lines. It is important to take into consideration a marginal surge region, so called soft surge, as it is enough to encounter whoosh noise when operating line is close to that region.

It is logical that moving operating line further from marginal surge region will reduce whoosh noise and thereby improve NVH of the engine. [6] investigated surge noise reduction by reducing excitation at source. To do this several compressor variants were

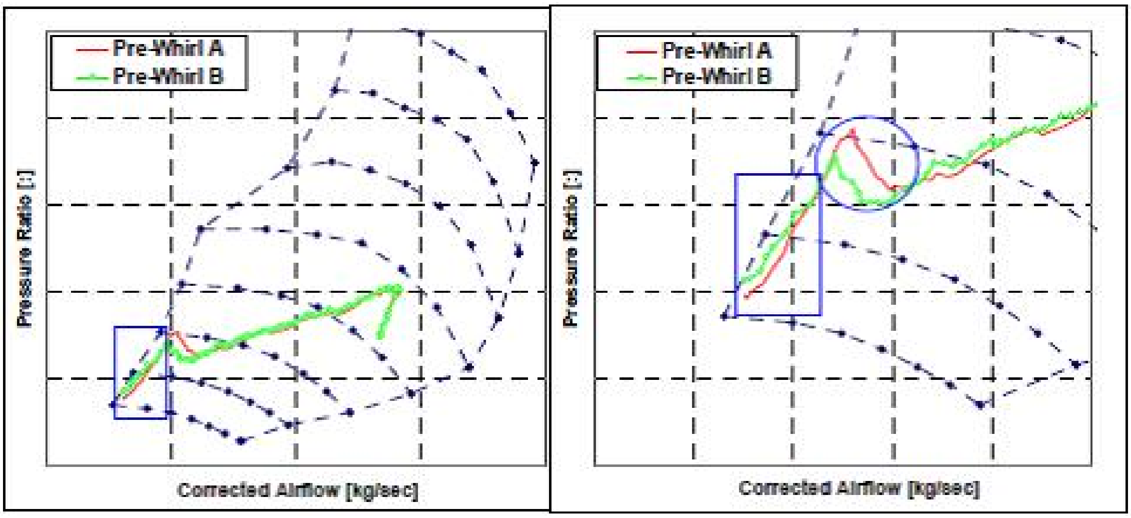


Figure 2.2: Compressor operational lines with different inlet swirl

investigated with the potential to give similar engine performance while allowing full load operation to occur further from the marginal surge region. Concluding that in better cases reduction of 3dB(A) in 4-mic average SPL at full load were achieved.

Research done by [7, Ford] investigated whoosh noise improvement possibilities on a turbocharged V6 engine. Among other methods, [7] was altering air flow to the compressor with an introduction of pre-whirl. Two prototype pre-whirl devices were used to evaluate their effectiveness to reduce the whoosh noise. Prototype A had positive and prototype B negative pre-whirl. Negative pre-whirl is preferable, because it reduces flow separation at the compressor blade, thereby improving surge margin.

Results shown that prototype B showed noise improvement only in the range of 2000–2500 rpm. However at engine speeds lower than 2000 rpm there is only a minor improvement on whoosh noise. In addition prototype B introduced stronger pressure pulsation and temperature rise at the inlet.

Operational lines shown in figure 2.2 with different swirl show that prototype B moved operational line closer to surge line at speed below 2000 rpm, while in the region 2000–2500 rpm it moved away from the surge line and improved the whoosh noise.

It was concluded that changes occurred because of pressure changes at both compressor inlet and outlet. It can be seen that prototype B has higher pressure drop at compressor inlet and that is what gives a change in surge margin.

To conclude it can be seen that control of air flow for compressor inlet can improve not only performance of turbocharger, but can also have significant influence on its NVH.

2.2 Flow parameters

Turbocharger compressor performance parameters (stage efficiency, mass flow range, pressure ratio capability) are influenced by the compressor inlet flow. Volvo clean side duct design guidelines described important flow parameters, that should be taken into account while designing air intake system:

- Flow capacity
- Pressure drop
- Pressure distribution
- Velocity distribution and magnitude
- Pre-whirl
- Flow eccentricity
- Vortex index

However, understanding how each of these parameters influence compressor performance and separating their influences is not possible. Therefore in most of the cases these parameters are categorized into three main attributes that define air inlet flow:

- Flow uniformity (pressure distribution, velocity distribution and magnitude, flow eccentricity)
- Pressure drop (flow capacity, pressure loss)
- Flow rotation (pre-whirl, vortex index)

Correct control of these parameters in the inlet will supply turbocharger with the flow that can significantly alter turbocharger performance for specific conditions. However, as [2] concluded, there will always be a tradeoff between different flow parameters - increased velocity will increase pressure drop, also induced flow rotation will usually increase pressure drop in the main flow - and it is up to the designer to consider what is required for his specific system.

2.2.1 Uniformity, pressure drop, velocity

Main three parameters used to evaluate air intake system at Volvo are uniformity, pressure drop and velocity. Most extensive work on these parameters was done by [2] were eleven different 90° turbocharger inlet elbow configurations were investigated. [2] measured pressure loss coefficient and flow uniformity index for velocity distribution. Out of these eleven configurations four were chosen for comparison:

- worst case
- lowest pressure drop
- moderate pressure drop and high uniformity index

- highest uniformity index

Four concepts were experimentally tried in a turbocharger rig to define how different air inlet performance parameters influence turbocharger compressor performance.

It was concluded that high uniformity index improves turbocharger performance by increasing pressure ratio especially in the low mass flow region and it does not have any negative effects. High velocity will improve surge line, but increase pressure drop and will reduce overall pressure created by the turbocharger, in addition it moves choke line to the left, which is disadvantage, and therefore there is always a trade-off between high pressure loss and low discharge coefficient (high velocity).

Effect on different inlet conditions on the compressor map can be seen in the 2.3. To conclude, it can be said that the higher the uniformity of turbocharger inlet the more beneficial it is. It is also seen that even though pressure drop increases velocity which is beneficial for surge, it is also disadvantageous for choke and it is more beneficial to keep pressure drop as low as possible.

2.3 Improvements

Engine packaging constraints increased number of bends, conical shapes and non – circular shapes in clean side ducts and made them less optimal for the air flow. To evaluate changes in the compressor map it became crucial to understand how different shapes affect air flow parameters mentioned in [2] to be able to design and optimize inlet system.

2.3.1 Conical surfaces

[5] did a research on different inlet configurations. Part of the investigation was to find out how tapered or non-tapered turbocharger inlet cones affect their performance. It was found out that generally tapered inlet cones are increasing pressure ratio at all mass flows and also moves compressor map to the left by increasing flow velocity.

However, it is also known that steep conical shapes can create separation and vortices, which creates large pressure losses in the inlet. Different sources recommend different maximum angles for conical shapes variating from 5° to 15°, however no data about Reynolds number influence in those tests was found. [8] investigated flow separation in conical shapes with 5° and 10° and 30° angles. In addition they focused on different flow Reynolds numbers and their influence on separation. It was concluded that separation is highly influenced by Reynolds number. Higher Reynolds number flow can be attached to higher conical shape angle. For Reynolds number of 2000 separation occurs already at 5° angle, while with higher Reynolds numbers separation at 5° does not occur. This shows that not only the geometry can influence flow parameters, but also flow itself is important.

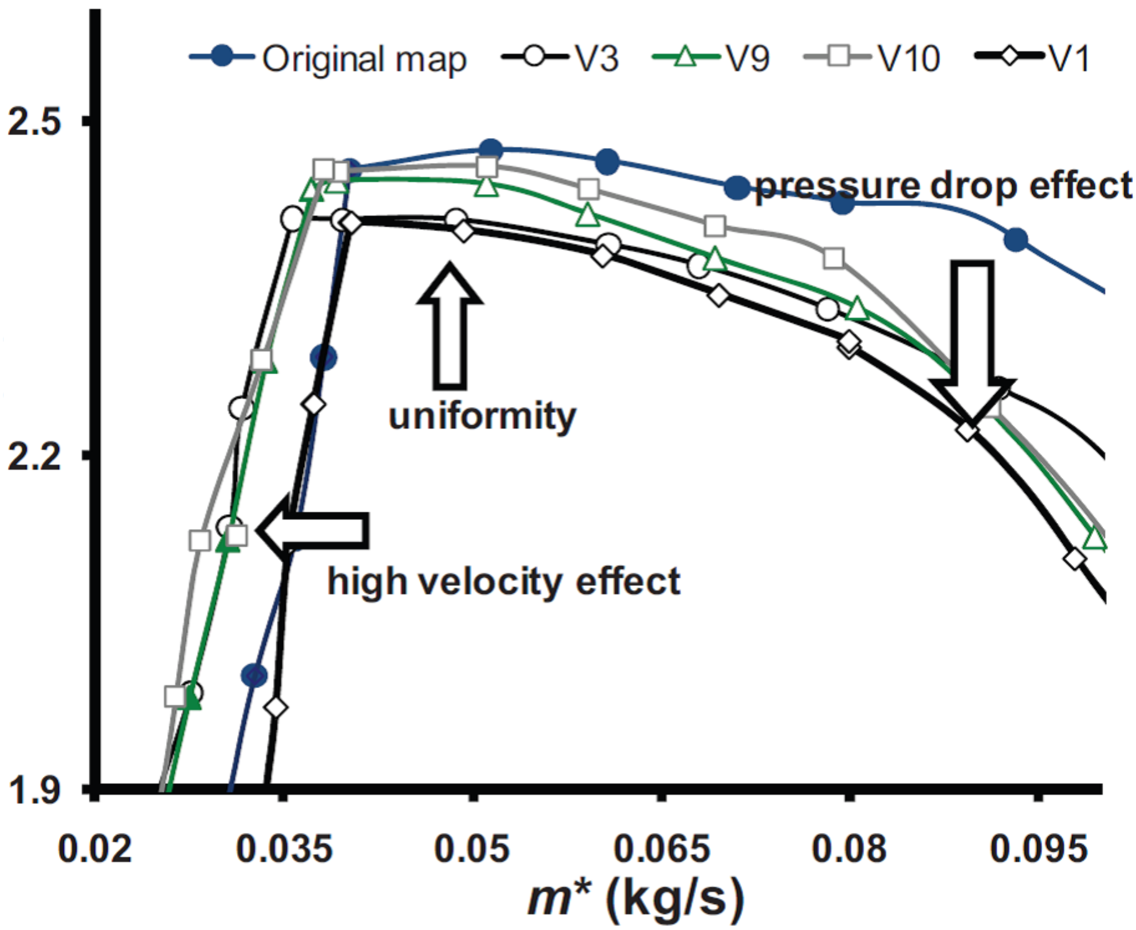


Figure 2.3: Flow uniformity, pressure drop and flow velocity influence on compressor map.

2.3.2 Non - circular shapes

Ducts inside passenger vehicle ventilation system usually have unusual shaping because of strict packaging. Same problems are becoming evident in intake system of the car. Research conducted by [9] used CFD simulations to investigate pressure drop for eight most common non-circular shapes with different bend angles.

From Figure 2.6 it can be seen that Rectangle B gives the best results, while Rectangle A gives the worst even though it is the same shape, just turned by 90° . This can be explained that high pressure drop losses usually occur near the inner wall. However, it was also seen that the curvature of outer wall and slope of top and bottom plays significant role in secondary flows and hence flow separation and pressure loss. Also as expected pressure loss is increased with increased bend angle, but it also highly depends on the shape of the duct.

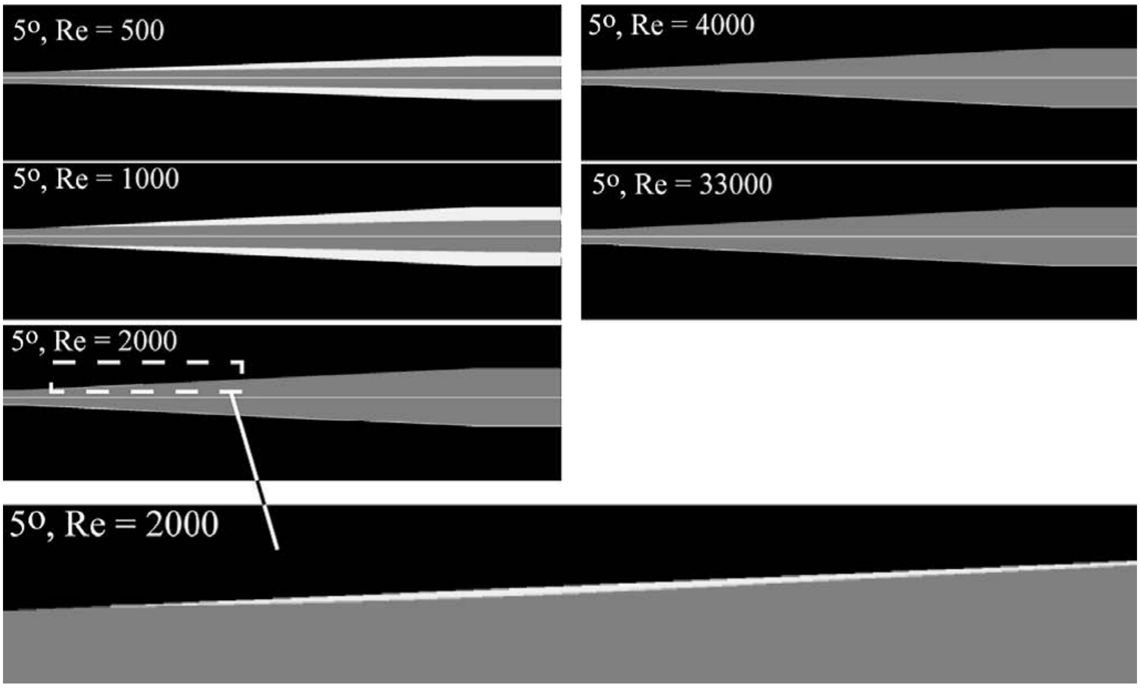
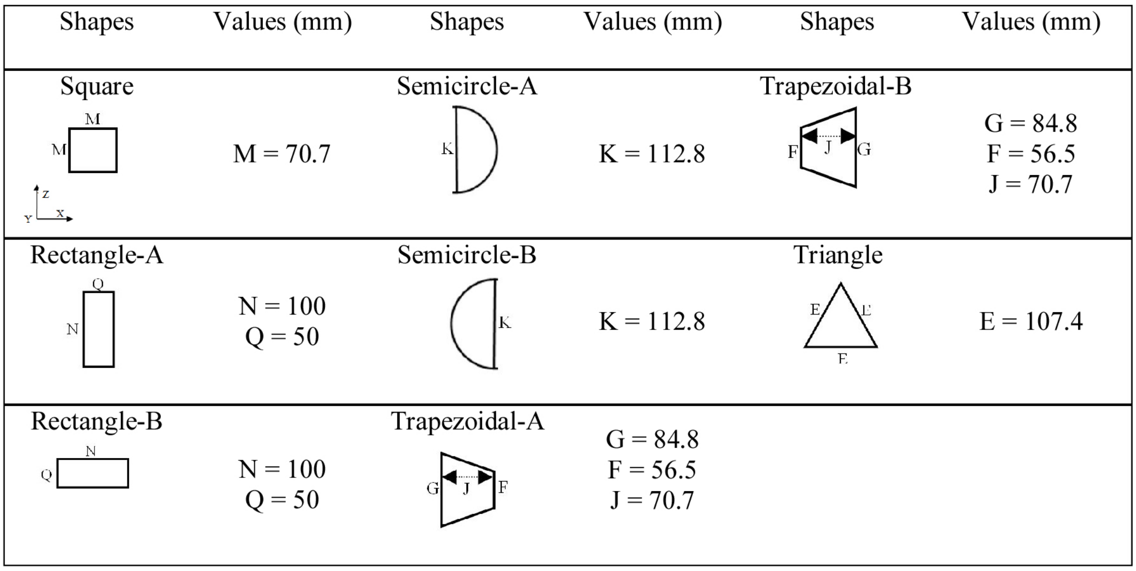


Figure 2.4: Reynolds number influence on flow separation in conical expansion.



Shapes	Values (mm)	Shapes	Values (mm)	Shapes	Values (mm)
Square	M = 70.7	Semicircle-A	K = 112.8	Trapezoidal-B	G = 84.8 F = 56.5 J = 70.7
Rectangle-A	N = 100 Q = 50	Semicircle-B	K = 112.8	Triangle	E = 107.4
Rectangle-B	N = 100 Q = 50	Trapezoidal-A	G = 84.8 F = 56.5 J = 70.7		

Figure 2.5: Shapes tested in [9] research

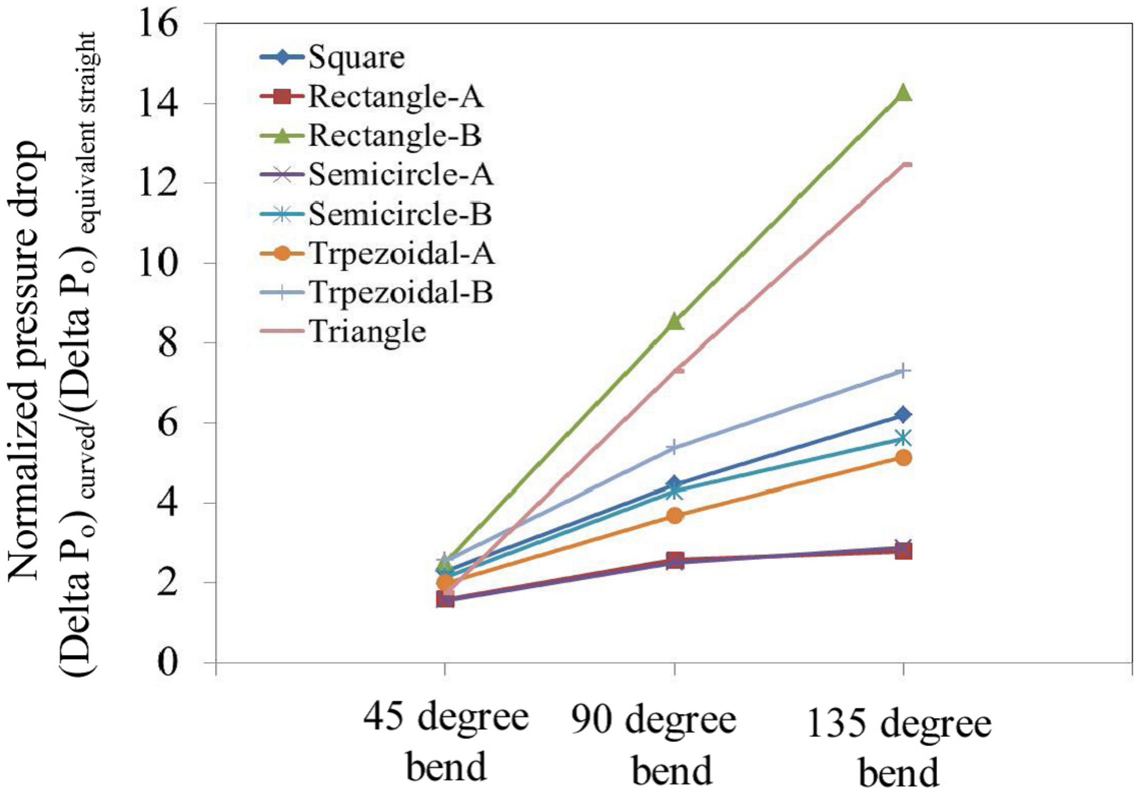


Figure 2.6: Results of different shapes tested in [9] research

2.3.3 90° bend

A common trend in turbocharged engine is to have a 90° bend just before inlet of the compressor. Because of limitations in space and tight curves this bend usually causes highest losses in the intake system.

Two different design proposals were investigated to improve air flow in a 90° bend. First solution made by [10] investigated curve flow physics. As outer curve has higher velocity than inner it also has lower pressure because of continuity. Therefore this pressure difference between outer and inner surface creates separation and secondary flow. It is beneficial to move inner and outer curves closer to reduce the pressure difference made by the bend; then negative effects of bend will be reduced. Solution with vane inside the middle of the duct was also investigated. It will separate flow and consequently divides pressure difference into two parts.

Another duct patented by [11] uses similar techniques as [10] ducts but the shape is more adapted for turbocharger compressor. Additional improvement was to create flow attachment for the inner surface by making an opposite bend just before the turn. By expanding duct sides it is also narrowing a passage between outer and inner curves, while keeping the same airflow area.

[3] investigated three different pipes before turbocharger compressor, one straight

for comparison and two 90° bend inlet configurations. It was concluded that distortion caused by bend-pipe inlet decreases the choke limit to lower mass flow rate. It was also found out that bend pipe with longer axial distance created larger distortion on the impeller inlet than shorter pipe and as a consequence it reduces turbocharger compressor performance more significantly. Distortions made by inlet bend pipe are harmful for compressor blades, especially with increased mass flow.

2.3.4 S - shape

Another common ducting shape - S – shape - could be also found in Volvo systems, therefore investigation of research was done on S shape internal flow.

A research was done on the exhaust system S shapes in DP filter by [12]. It was concluded that for S – shapes it is convenient to use unipolar sigmoid function as it gives better results compared to sharp 90° angles. [12] also investigated swirling flow effect in the DPF and it was concluded that it reduces pressure drop by 10% and improves flow uniformity by 30% with a swirl between 10 m/s and 20 m/s.

A study done by [13] tried swirling flow for different S – shaped ducts and measured their pressure losses. It was concluded that overall static pressure recovery with swirl increases by 40%, moreover flow uniformity is also improved for a swirling intake flow.

2.3.5 Pre - whirl guide - vanes

[1], [6], [14], [15], [16], [17], [18], [19], [20], [21] and [22] were investigating inlet pre-whirl influence on turbocharger compressor performance. Even though it is not yet widely used in industry, only BMW, Toyota and Ford did some work on this, but can be seen as a promising concept that can extends turbocharger compressor performance.

Good work for introduction with the topic was done by [1]. Initially they have investigated different air inlet system features and their influence on turbocharger compressor performance. It was found out that inlet configuration can have high influence on compressor stage efficiency, flow range and pressure ratio capability. A part of the research was to investigate swirl influence in the intake system, therefore a flow modifier was included to create a positive swirl. Overall results showed that air inlet system has high influence on turbocharger performance and that inclusion of flow modification devices can improve compressor performance noticeably.

Investigation by [14] was done with different static pre-rotation devices. Initially five different devices were tested with different number of vanes and different intake shapes. It was concluded that all of the inlet guide vanes reduced turbo - lag significantly, by increasing rotational speed of the turbocharger. Their effect was most significant on part loads, where 40% improvement in turbo acceleration time was seen.

An interesting development was done by [15]. Variable inlet guide vanes and casing treatment synergy effect was investigated and concluded that casing treatment and VIGV gives better results when they are combined than separately and overall surge flow rate was reduced by 59%.

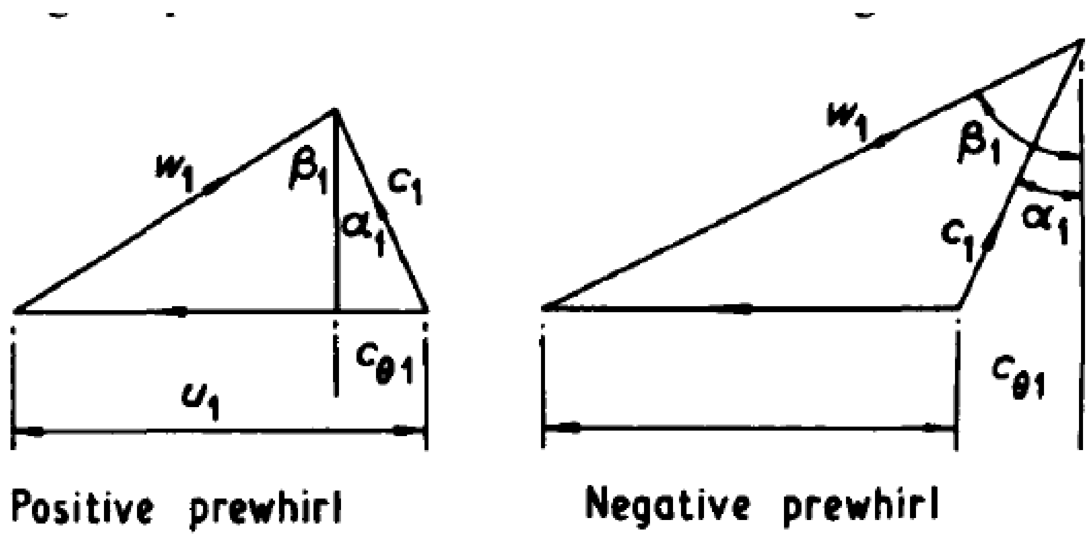


Figure 2.7: Positive and negative swirl velocity triangles

Extensive research on different angles of inlet guide vanes was done by [16]. It was concluded that stable operation can be extended to low flow rates by introducing positive pre-whirl angle. It was also mentioned that curved pre-whirl device blades have increased pre-whirl effect with smaller attack angles. However these curved blades were not able to be set straight and create a zero pre-whirl which limits their operation for high mass flow rates were pre-whirl is disadvantage, because of pressure drop.

Development of a commercial prototype for industry was done by [17]. Same results were found again. Positive pre – whirl will extend compressor operating range into lower mass flow rates and the higher the attack angle, the more advantage it gets. In addition, negative pre-whirl angle was investigated were it was found out that it can reduce turbocharger speed at high engine speeds and extend choke range, however at the same time it reduces compressor efficiency. Finally, by trying different concepts with more vanes and longer device it was concluded that increasing angle of rotation and shortening device is more efficiently than increasing the number of vanes.

Research done by [18] confirmed same results for a surge line improvement. In addition, they looked more extensively in a local effect of the pre-whirl by investigating flow after different distances from pre – whirl device. These results showed some non – symmetric flow at the pre – whirl vane angle of $> 60°$. Consequently, poor velocity distribution downstream may have had compressor efficiency drop.

Research done by [19] confirmed previous works and most widely and clearly showed how different pre – whirl angles influence compressor map 2.8. [19] changed angle from $0°$ to $70°$ to see effect on surge performance at low mass flow range and turbocharger compressor efficiency. It has also included $10°$ degree negative pre – whirl which showed improvement in choke line and moved top efficiency points to higher mass flow. Finally, it was concluded that marginal (up to $20°$ positive pre – whirl) angles increases compressor efficiency by around 2%.

Different research on the pre – whirl influence on a turbocharger compressor was done by [20]. Turbocharger inlet was investigated as annular diffuser which is affected by pre – whirl. Different angles of pre – whirl and different AR (area ratio) were tried. It was concluded that as the flow proceeds downstream, the longitudinal velocity decreases irrespective if flow is swirling or not as well the effect of swirl gradually decays as the flow proceeds downstream. The main cause influencing velocity profile is boundary conditions. Maximum velocity in annular diffuser is closer to the hub for non – swirling flow and is pushed closer to the casing in swirling flow. Swirl also pushes stall from the casing wall closer to the wall. Finally, pressure recovery is faster with introduction of swirl.

It can be seen that swirl can significantly influence turbocharger compressor performance, therefore an investigation was done to evaluate known ways for swirl creation. Number of patents were done for conventional IGV devices like [23] and [24].

Another conventional, but more advanced concept was also done by [24] which used inlet guides vanes with a hub-less solution, which is reduced pressure drop along the pre – whirl device.

Quite similar to guide vanes pre – whirl device, but working in a different way was SGD (swirl – generator device) made by [21]. The difference of SGD device compare to other IGV that it relies on radial inlet and centripetal vanes and this strongly influences conditions at the compressor inlet. Main difference lies on a special flow distribution, with low velocities at the center and high velocities at the periphery of the compressor inducer.

Different pre – whirl device was done by [22], were pre – whirl was induced by pressurized air. The main advantage of this method is no obstacles in a flow path and less pressure drop. As well it can have better control control possibilities as it will not have moving mechanical parts. Aerodynamically induced pre - whirl holes used 70° angle with diameter of 0.003 m and airflow of 0.00471–0.0994 and improved surge margin by 84% at low speeds and 23% at higher speeds.

Finally, patent by [11] introduced a way when one turbocharger in a serially coupled turbochargers is creating a pre – whirl rotation for a second one and removes any need of inlet guide vanes. It could also be a common solution in twin turbocharging systems, which are becoming more popular in modern engines.

2.3.6 Optimization methods and solvers

Initially research use CFD analysis as initial step and experimental testing afterwards. [25] confirmed CFD compatibility for a pressure drop optimization and accuracy by comparing CFD results with experimental tests.

More advanced methods used different computerized optimization tools. [26] used Hyper Study Tools for Clean Side Duct optimization and proved it to be a useful tool. Also interesting approach was done by [4] were they investigated a Bionic approach instead of numerical. The main idea of bionic approach is to use physical information of the flow field to optimize the duct path by letting flow “finding” its way.

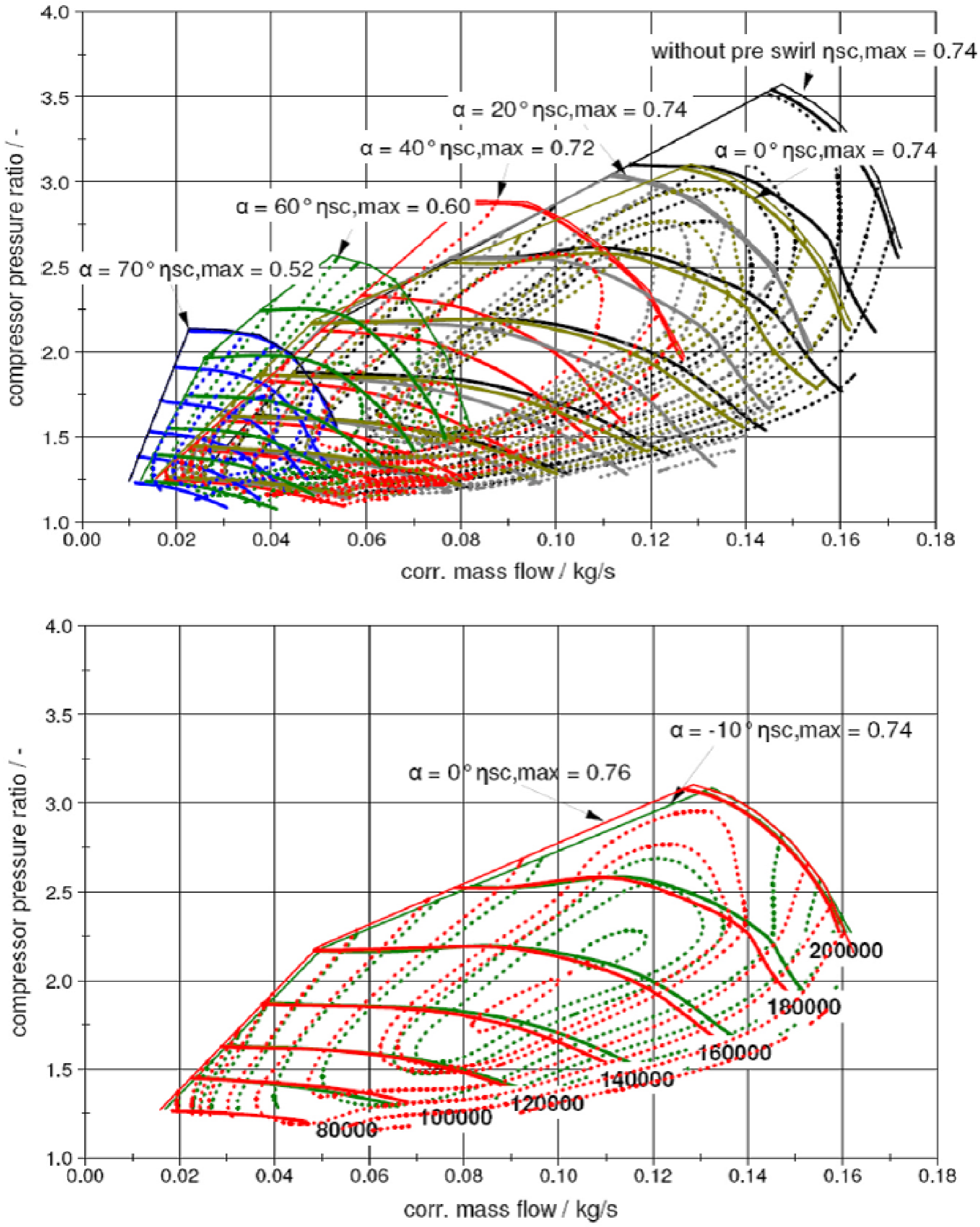


Figure 2.8: Swirl influence on the compressor map.

2.4 Conclusions

Literature review showed many possible ways to improve air intake system performance for compressor. Three paths were suggested for Air Induction System that could be further investigated.

- Swirl investigation
- 90° bend improvement
- Investigation of different duct shapes

It was decided to focus thesis work on swirl investigation as it was least known and quite important parameter that could be influencing air inlet system performance, but at the moment was not evaluated at Air Induction System at all.

3

Theory

THESIS PROJECT IS FOCUSED on the turbocharger compressor inlet airflow, therefore theory part will cover an introduction to turbocharging in the modern internal combustion engines, background on centrifugal compressors, swirling flow theory together with pre - whirl and Computational Fluid Dynamics for turbulence modelling.

3.1 Turbocharging

To overcome environmental, safety and power requirements a common solution in the modern engines is air charging. Air charging can be done by supercharger, turbocharger or other different ways.

The most common way that uses part of wasted energy is turbocharging. Turbocharger has a turbine and compressor part connected to each other by shaft. Turbine part is spin by exhaust gasses from the engine and it spins compressor part, which compresses the air coming into the engine.

As more air can be pushed into the cylinder, it means that more fuel can be injected and more power generated in the engine. For every engine, power is limited by amount of air coming into the cylinders, therefore turbocharging extends these limits.

As it was mentioned - turbocharger is using part of exhaust gasses, which otherwise would be wasted, therefore it increases overall engine efficiency and as a result reduces fuel consumption and harmful exhaust gas emissions.

3.2 Compressor

A centrifugal compressor consists of a rotating impeller followed by a diffuser. Fluid is drawn in through the inlet casing into the eye of the impeller. The function of the impeller is to increase the energy level of the fluid by whirling it outward, thereby

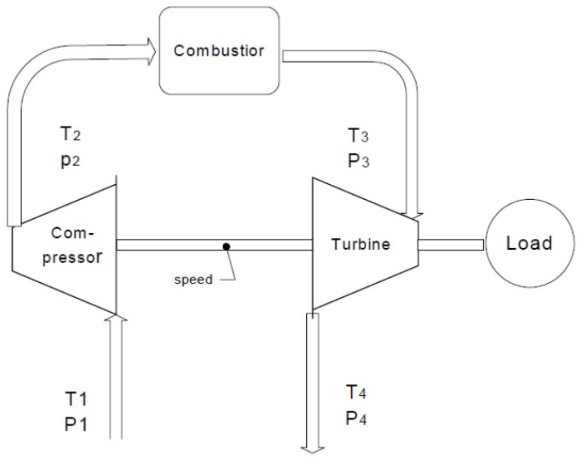


Figure 3.1: Simple turbocharger systematic view.

increasing the angular momentum of the fluid. Both the static pressure and the velocity are increased within the impeller. The purpose of the diffuser is to convert the kinetic energy of the fluid leaving the impeller into pressure energy. Outside of the diffuser is a scroll or volute whose function is to collect the flow from the diffuser and deliver it to the outlet pipe.

The hub is the curved surface of revolution of the impeller a–b; the shroud is the curved surface c–d forming the outer boundary to the flow of fluid. At entry of the impeller, the relative flow has a velocity w_1 at angle b_1 to the axis of rotation. This relative flow is turned into the axial direction by the inducer section or rotating guide vanes as they are sometimes called. The inducer starts at the eye and usually finishes in the region where the flow is beginning to turn into the radial direction.

3.3 Pre - whirl

Air flow coming into the compressor can be rotated in order to influence compressor performance. So called positive pre-whirl is when flow is rotated into the same direction as compressor wheel rotation, and negative pre-whirl is opposite.

At low engine speeds, the compressor impeller speed is too high in relation to the air flow velocity, so the incidence angle of the blade tip is away from the design point. Adding pre-whirl in the rotational direction of the impeller (positive pre-whirl) can help to improve the incidence angle and the inlet Mach number is reduced. Since the flow already has a rotational component before entering the compressor wheel, the relative

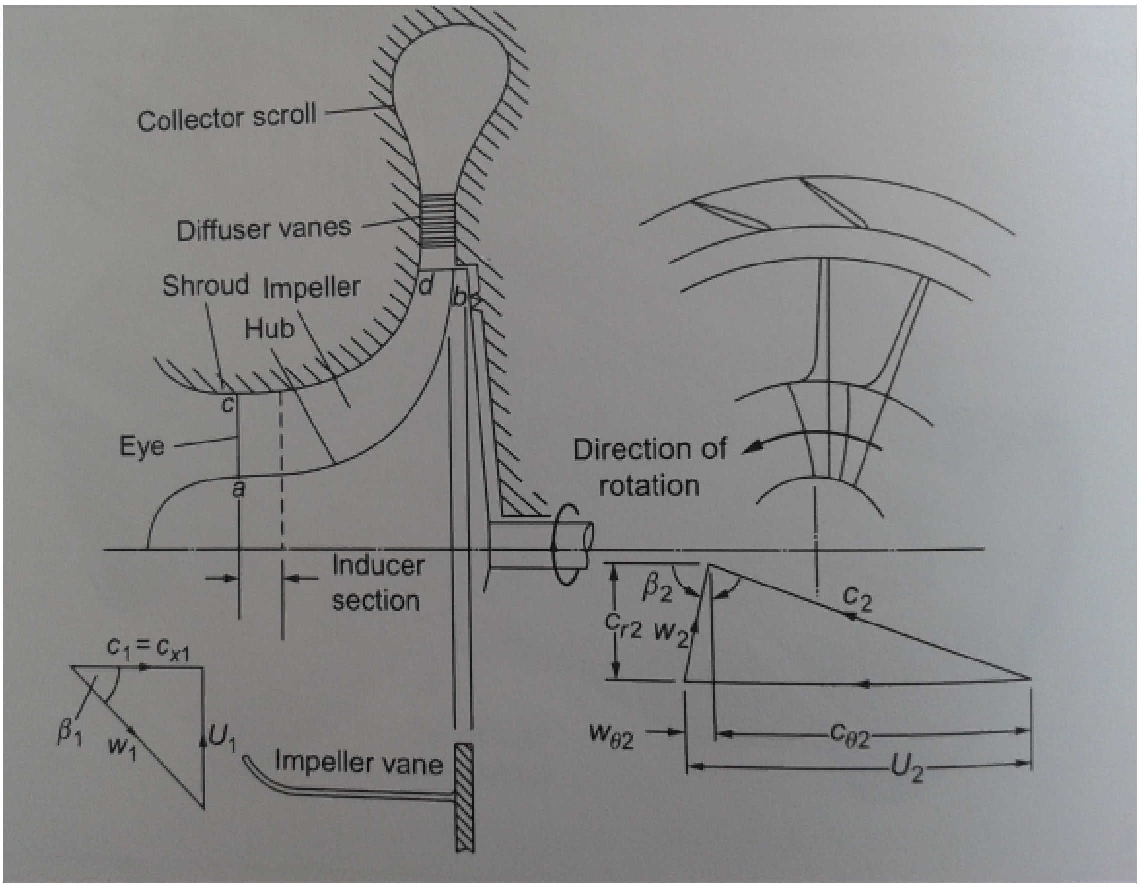


Figure 3.2: Compressor velocity triangles.

velocity, critical mainly at the wheel tip, is reduced. Inlet Mach numbers can be in the range of 1.5. This speed is associated with energy losses, reducing the surge margin. The lower the inlet Mach number is, the smaller the energy loss can be. If the incidence angle is optimised and therefore the energy loss resulting from flow separation can be avoided again the surge margin is increased. Hence the compressor wheel is able to speed up. The flow already includes a whirl component, so the work done by the impeller on the flow can be reduced, which leaves more energy to speed up the impeller wheel. This can help to improve transient response and increases the pressure ratio for a given amount of turbine energy.

At very high engine speeds and mass flow rates the incidence angle is not optimum, because the compressor wheel is spinning too slowly compared with the inlet flow speed. Negative pre-whirl (i.e. against the direction of the impeller) is then used to improve the relative flow angle towards the impeller again. At very high turbo speeds the flow losses in the diffuser section increase too. A second source of losses is the relative Mach number at the compressor impeller tip for high mass flow rates which increases when applying pre-whirl. Also mechanical losses increase with turbo speed. All these combined increase the losses to a level where the choke limit of the compressor is reached.

3.4 Swirl ratio

It is important to have a good analytical value to determine the magnitude of flow rotation. Research read in literature review commonly uses vane angle to measure swirl, however vane angle is not a good analytical value because of two reasons:

- Vane angle does not show exact angle of flow rotation, different vanes can create different amounts of swirl
- Naturally, every intake system will create some amount of swirl and it is not possible to evaluate how much of swirl system has by using vane angle measurement

Therefore, instead of vane angle it was chosen to use swirl ratio.

$$SR = \frac{Axialmomentumofaxialflow}{Tangentialmomentumofaxialflow} \tag{3.1}$$

CFD software Fluent was not able to give it as post processed value, therefore some calculations needed to be made. Firstly, tangetial velocity had to be calculated from the given data of coordinates and velocity vectors:

$$v_r = \frac{v \times (z - z_0) - w \times (y - y_0)}{\sqrt{(z - z_0)^2 + (y - y_0)^2}} \tag{3.2}$$

Swirl ratio was calculated for chosen plane by integrating axial momentum of axial flow and tangential momentum of tangential flow over the surface area. Denominator is multiplied by area radius of the swirl rotation in order to make results dimensionless and to be possible to compare it in different cases.

$$\frac{\oint \rho \times \sqrt{(y - y_0)^2 + (z - z_0)^2} \times \sqrt{u^2} \times v_r}{R \times \oint \rho \times u^2} \tag{3.3}$$

3.5 CFD

3.5.1 Turbulence models

Turbulence model is the construction and use of a model to predict the effects of turbulence. Averaging is often used to simplify the solution of the governing equations of turbulence, but models are needed to represent scales of the flow that are not resolved.

Realizable k-ε model

Realizable k-ε model is a recent development and differs from the standard k-ε model in two important ways:

- The realizable k-ε model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, ε has been derived from an exact equation for the transport of the mean - square vorticity fluctuation.

RNG k-ε model

RNG model was developed using Re-Normalisation Group (RNG) methods by [Yakhot] to renormalise the Navier - Strokes equations, to account for the effect of smaller scales of motion. In the standard k-ε model the eddy viscosity is determined from a single turbulence length scale, so the calculated turbulent diffusion is that which occurs only at the specified scale, whereas in reality all scales of motion will contribute to the turbulent diffusion. The RNG approach, which is a mathematical technique that can be used to derive a turbulence model similar to the k-ε results in a modified form of the ε equation which attempts to account for the different scales of motion through changes to the production term.

k-ω SST model

The SST k-ω turbulence model is a two-equation eddy-viscosity model which has become very popular. The shear stress transport (SST) formulation combines the best of two worlds. The use of a k-ω formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer, hence the SST k-ω model can be used as a Low-Re turbulence model without any extra damping functions. The SST formulation also switches to a k-ε behaviour in the free-stream and thereby avoids common k-ω problem that the model is too sensitive to the inlet free-stream turbulence properties. Authors who use the SST k-ω model often merit it for its good behaviour in adverse pressure gradients and separating flow. The SST k-ω model does produce a bit too large turbulence levels in regions with large normal strain, like stagnation regions and regions with strong acceleration. This tendency is much less pronounced than with a normal k-ε model though.

Reynolds Stress Model

The Reynolds Stress Models (RSM), also known as the Reynolds Stress Transport models, are higher level, elaborate turbulence models. The method of closure employed is usually called a Second Order Closure. This modelling approach originates from the work by Launder. In RSM, the eddy viscosity approach has been discarded and the Reynolds stresses are directly computed. The exact Reynolds stress transport equation accounts for the directional effects of the Reynolds stress fields.

Enhanced wall treatment

Enhanced wall treatment is a near-wall modelling methods that combines a two-layer model with enhanced wall functions. If the near-wall mesh is fine enough to be able to resolve laminar sublayer, then the enhanced wall treatment will be identical to the traditional two-layer zonal model. However, the restriction that the near-wall mesh must be sufficiently fine everywhere might impose too large computational requirement. Ideally, then, one would like to have a near-wall formulation that can be used with coarse meshes (usually referred to as wall-function meshes) as well as fine meshes (low-Reynolds-number meshes). In addition, excessive error should not be incurred for intermediate meshes that are too fine for the near-wall cell centroid to lie in the fully turbulent region, but also too coarse to properly resolve the sublayer. To achieve the goal of having a near-wall modelling approach that will possess the accuracy of the standard two-layer approach for fine near-wall meshes and that, at the same time, will not significantly reduce accuracy for wall-function meshes, FLUENT can combine the two-layer model with enhanced wall functions.

4

Method

THIS PART OF THE THESIS WORK describes continuity of the thesis project, includes the reasoning behind the decisions that were made and in some sections results that greatly influenced next steps of the work could be mentioned, however main part of the results are in the Results section.

4.1 Introduction

Literature and theory review gave a base to define thesis aim. It was decided to investigate swirling air flow as it was least known phenomena in AIS and usually not considered during design process. However literature review already showed that if correctly used it can greatly influence turbocharger compressor performance parameters.

4.2 Swirl ratio

Literature review showed that high amounts of positive swirl are disadvantageous as they greatly reduce turbocharger efficiency, in addition even small negative swirl are also giving negative effects. Taking into account information found in the literature, that swirl of greater than $+40°$ of vane angle is negative and that swirl up to $+20°$ can give positive efficiency effects, it was decided to investigate swirl range between +40 and 0. In addition range was extended into the negative swirl of $-10°$ to evaluate its efficiency drop effect.

It was concluded that swirling flow used in this range can improve compressor efficiency, contrary swirling flow outside this range could significantly reduce efficiency of turbocharger.

The range $+40°$ to $-10°$ taken from literature however couldn't be used as a guideline for a number of reasons:

- Vane angle does not directly show the air flow angle. Different vane shapes can create flow rotation that can have different swirl magnitude.
- Bends and complex shapes themselves can create rotation of the flow. This rotation will also influence compressor performance and it is not clear what vane angle it represents.

Instead of vane angle it was decided to use swirl ratio. As it was not clear what is the exact relation between vane angle and swirl ratio, first thing was to find a relationship between them.

4.3 Swirl vanes

Swirl generator looking similar as in [19] was designed with vane angle range from -10 to +40. This range was simulated and swirl ratio values were found for each vane angle. A range of investigation was changed from vane angles to swirl ratio. Range of -10° to +40° was covering swirl ratio range of -0.06 to +0.18. It was expected to get best efficiency at approximately 20° of vane angle, which represented 0.09 swirl ratio. Evaluating possible error in vane shape and CFD simulations it was decided that range of -0.06 to 0.15 will be sufficient for further investigation.

A design of initial vanes was improved because of two reasons. Firstly, initial vanes had straight shapes that were causing a lot of pressure drop. To separate swirl ratio and pressure drop influence on the compressor map, pressure drop had to be reduced. Secondly, it was important to realize how much swirl different vanes with the same angle are creating and confirm idea that two vanes with the same angle cannot be compared.

4.4 Benchmarking

To finalize swirl ratio investigation range it was also important to get an idea what swirl ratio is common in design process. Benchmark was done on these Volvo intake systems.

- System 1
- System 4
- System 5

System 1 was chosen, because all the test were planed to be done with LP turbocharger from this system. System 4 was found interesting as it's shape was naturally creating a lot of swirl. System 5 was in interest as it gets highly swirling airflow from supercharger, in addition this system had two solutions - with and without air flow straightener.

Furthermore a BMW intake system with OEM swirl generating device was ordered. To test what influence this device has on the turbocharger, System 1 was redesigned to fit BMW swirl device and both of these systems were tested in the turbo rig.

System 2 was also tested using CFD tools, however because of innovative flexible design it was time consuming to evaluate how exactly vanes are bending and how much swirl they are creating at different air flows. Therefore only one case with maximum angle and 720 kg/h mass flow was tried.

Pressure drop of System 2 was also measured in the flow rig, to determine what effect it would have on the intake system pressure drop compared to System 1.

4.5 Flow bench testing

To validate simulation results and the amount of swirl that swirl generators are giving experimental tests were done in flow rig.

Firstly, vanes with improved design were manufactured for testing using rapid prototyping. Then, two tests were conducted - pressure drop and swirl ratio.

4.5.1 Post processing

Swirl ratio tests were done using swirl measurement equipment for internal combustion engines. An output measured by this equipment was momentum created by swirling airflow created on the spinning honeycomb. These results had to be post processed to convert it to swirl ratio.

Mass flow was converted to axial momentum rearranging conservation of linear momentum equation.

$$\sum F = \sum \frac{d_{mi}}{d_t} \times V_i = \dot{m} \times V_i = \dot{m} \times \frac{\dot{m}}{\rho \times A} = \frac{\dot{m}^2}{\rho \times A} \tag{4.1}$$

$$axial\, momentum = \frac{\dot{m}^2}{\rho \times \pi \times R^2} \tag{4.2}$$

Tangential momentum was calculated from measured data. While calibrating it was found out that measurement scale is showing 100, when there is a 10 gram weight on the lever. As lever length was 10.45 cm it was calculated that

$$T = m \times g \times r = 0.01 \times 9.81 \times 0.1045 = 0.01025145\, N \cdot m = 10.25145\, N \cdot mm \tag{4.3}$$

$$tangential\, momentum = \frac{SW \times 1.025145}{10000} \tag{4.4}$$

From this it can be known that measured data should be multiplied by 1.025145 and divided by 10000 to get exact momentum created by swirling airflow.

Combining these to equations final equation for post - processing was derived:

$$SR = \frac{\frac{SW \times 1.025145}{10000}}{\frac{R \times \dot{m}^2}{\rho \times \pi \times R^2}} = \frac{0.0001025145 \times \rho \times \pi \times R \times SW}{\dot{m}^2} \tag{4.5}$$

4.6 Simulations

Results from CFD simulation didn't match well with experimental results, therefore an improvement of CFD calculations was needed. Four different turbulence models were tried for swirl modelling: k-ε realizable, k-ε RNG, k-ω SST and Reynolds Stress Model.

4.6.1 Meshing

Initial mesh at AIS uses two wall layers with settings found in Table 4.1, which in various simulations gave Y+ values between 10 and 50. As results from CFD needed to be improved, different wall functions were tried. For enhanced wall function a requirement of Y+ values is to be close to 1. Calculation of first layer size for these Y+ value of 1 was done and found out that aspect ratio should be 0.04. Simulations were run using new mesh settings specified in table 4.1. However, as it was difficult to converge solution with such fine mesh aspect ratio of first layer was changed to 0.08.

4.6.2 Turbulence models

Basic CFD settings at Air Induction System uses Realizable k-ε turbulence model with standard wall function. This is a good model for laminar flow and shows a good match with experimental tests for pressure drop results.

As swirling flow is more complicated and causes more turbulent flow this model seemed not sufficient, this was also confirmed later in experimental results. Therefore it was decided to try different turbulence models. Fluent manual suggested to use k-ε RNG and Reynolds Stress Model as a good models for swirl so these two were included. In addition k-ω SST model was tried as number of articles suggested it as a good model for channel flows and adverse pressure gradients.

- k-ε Realizable
- k-ε RNG
- k-ω SST
- Reynolds Stress Model

While comparing initial results from these four models with experimental results it was found out that swirl ratio results differs by 15%-20%, which was considered to big.

First improvement was to try non equilibrium wall function with Reynolds Stress Model as it showed good swirl ratio results. It has improved solution and difference was reduced to 11%. Same wall function was tried with k-ε RNG model, but result difference was still 19%.

As it looked like different wall functions are not influencing solution enough it was decided to improve mesh and use more than basic two layers. After some runs aspect ratio of mesh was set to 0.08 and in total 13 layers were added to the walls. This improvement showed considerably better results for k-ε RNG model and swirl ratio difference was reduced to only 6,38%.

Finally it was decided to try Enhanced Wall Function for RSM and k-ε Realizable models with improved mesh. Results were also considerably better as RSM model showed only 8,25% difference and k-ε only 5,15%. It was also important to check the pressure drop match with experimental results. k-ε realizable model showed best match with experimental results by having only 0,32% difference.

Table 4.1: Layers for k-e realizable with enhanced wall function

Aspect ratio	0.04	0.2
Growth factor	1.2	1.2
No. of layers	8	2
Additional outer layers	5	0
Last aspect ratio	0.4	0

Initial simulation was run with baseline Fluent settings found in table 4.2. However solution with enhanced wall function was diverging, therefore Courant number and relaxation factors were changed to values shown in table 4.2.

Table 4.2: Courant number and relaxation factors for k-e realizable with enhanced wall function

Flow Courant Number	50	200
Momentum Explicit Relaxation Factor	0.6	0.75
Pressure Explicit Relaxation Factor	0.5	0.75
Density Under-Relaxation Factor	0.8	1
Turbulent Viscosity Under-Relaxation Factor	0.8	1
Energy Under-Relaxation Factor	0.8	1

4.6.3 CFD with turbocharger inlet

Simulations that has been done to test CFD settings and create connection between vane angle and swirl ratio were using straight inlet and straight outlet as can be seen in Figure 4.1. The reason for this was that flow bench for swirl was able to run only with a duct connection of 76 mm diameter.

However tests that were planned to be done in turbocharger rig will have different inlet, that can be seen in Figure 4.2. The main difference between these inlets are

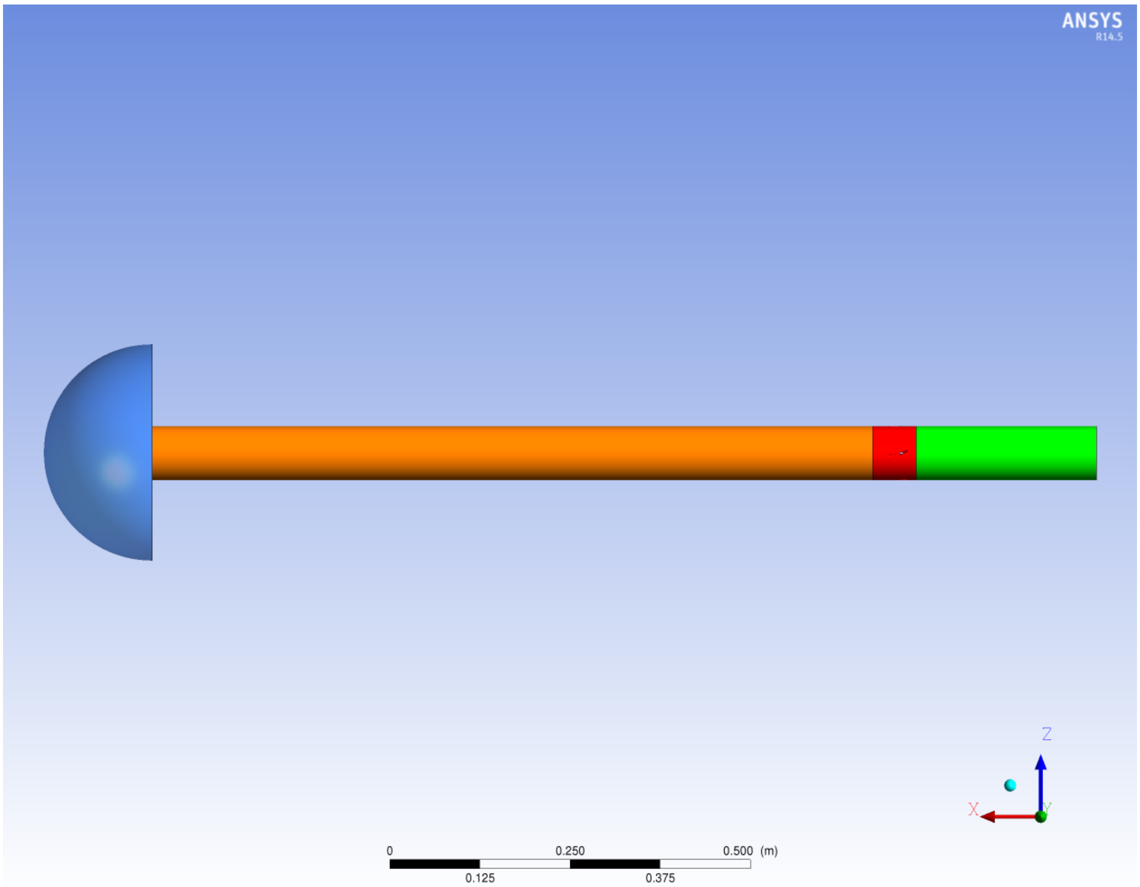


Figure 4.1: Test setup for flow bench

conical connection to the compressor inlet. To finalise swirl ratio that will be created in turbocharger test rig CFD simulations with the same swirl generators and updated and validated turbulence model were done.

Results showed that swirl devices in the turbocharger test rig will create 40% less swirl because of conical inlet. Swirl generators were covering the range of swirl between -0.06 and 0.11 of swirl ratio. As it was expected that optimum swirl for efficiency should be around 0.09 of swirl, it was decided that this range is a little bit to narrow and because of small errors top efficiency point can be out of the tested range. Therefore addition swirl device with 45° angle was manufactured, which expanded covered swirl range to 0.13. This was decided to be sufficient range and everything was prepared for experimental testing.

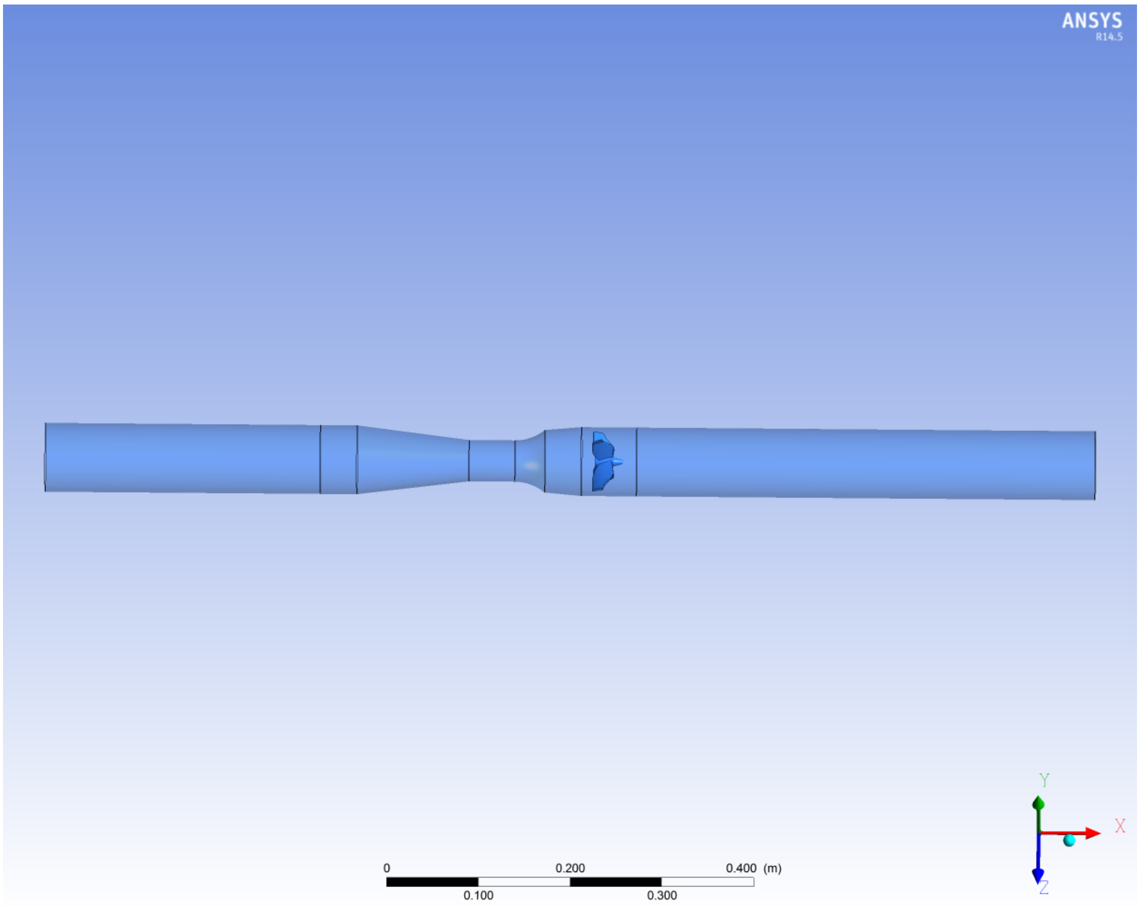


Figure 4.2: CFD simulation setup for testing in turbcharger bench

4.7 Turbocharger Test Rig

As swirl generator CFD model and swirl ratio was confirmed by flow bench tests the last step was to define swirl ratio influence on turbocharger compressor performance. To conduct experimental tests master thesis project was continued in AVL Turbocharger Test Rig at SAAB/NEVS.

Test cell setup can be seen in the Figure 4.3. Measurements on the compressor side were taken in 3 places: before swirl device, after exhaust T junction and just before compressor wheel. First two were used to measure most important parameters: pressure, temperature, mass flow and calculate compressor performance parameters. Measurement point before compressor wheel was checking the temperature and was used to determine when compressor reaches surge region. In addition the control of compressor speed was done by measuring parameters in exhaust side and also compressor speed sensor was included.

Compressor speed was chosen to cover the range from 40 000 rpm to 160 000 rpm.

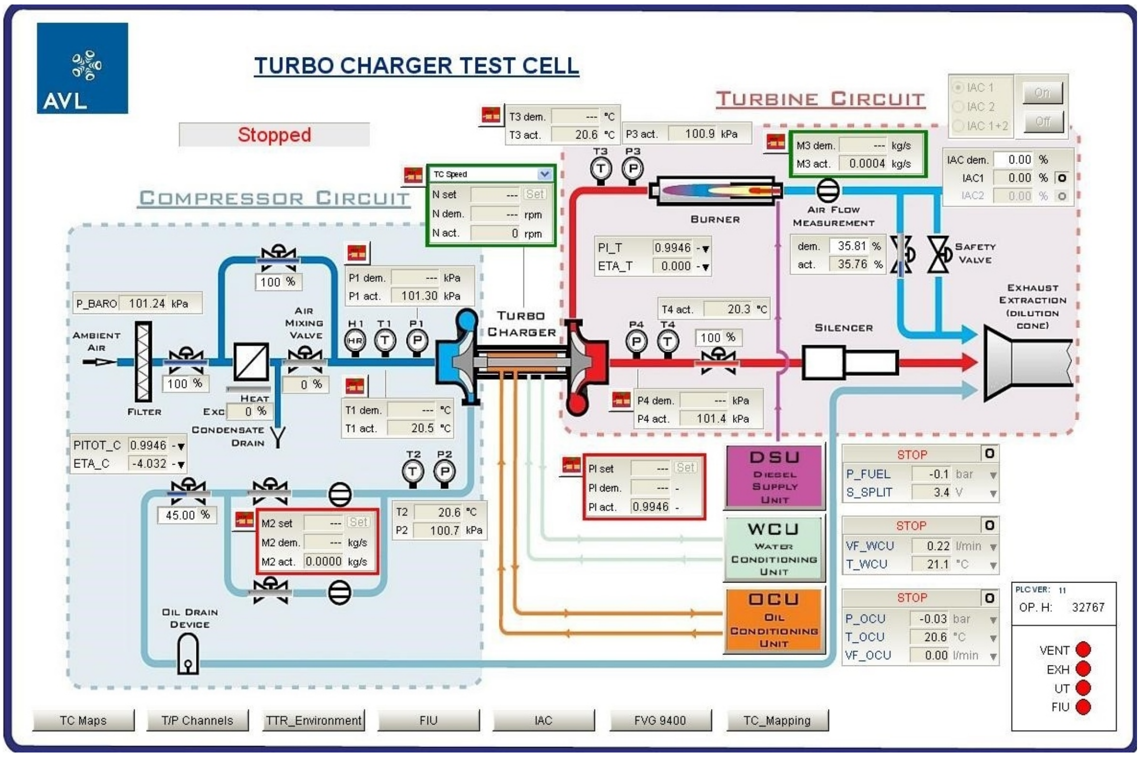


Figure 4.3: Schematic of test cell

Lower line was limited by testing possibilities, as it is hard to control and measure results lower than speed values of 40 000 rpm. Top limit was set by looking at compressor operating area. Full limit operating line was reaching 150 000 rpm range, therefore going above 160 000 rpm was not necessary.

First tests were done with baseline and #35 swirl generators in order to determine if results are as expected and experimental tests cover needed range for the results. It was expected that #35 will have swirl magnitude that is to high and efficiency improvement will be lower than baseline. However as efficiency improvement seemed to be at its peek it was decided to increase range a little bit and add swirl device #45. As #45 showed sign of efficiency decrease compared with #35 it was concluded that boundary value for positive pre - whirl is reached.

Further it was necessary to do the same test with the negative pre - whirl. Swirl #P8 was tested and showed significant reduction in efficiency. As results matched expectations, no further increase in positive swirl was needed.

Boundary condition for both limits were set and next step was to find maximum efficiency area. Swirl devices of #12, #20, #0 were tested to cover the range from -0.06 to 0.13 of swirl ratio.

Swirl device of #0 was an additional baseline. As map with no swirl also was not creating any pressure drop, therefore results could have been influenced by this difference.

Even though #0 created a small amount of swirl because of curved vanes, but it was creating similar pressure drop as other devices and was a good map to compare to.

5

Results

RESULTS OBTAINED DURING THESIS WORK can be summarized into 3 categories - Computational Fluid Dynamics, Flow Bench testing and Turbocharger test rig results. Results are organized in the order as they were obtained throughout master thesis project and trying to keep same flow as it is in Method section.

5.1 CFD Simulations

5.1.1 Benchmarking

Table 5.1 summarizes results from benchmarked systems. Three systems were benchmarked using CFD simulations. As results were obtained from simulations with base AIS CFD settings they could have up to 20% error, however it is enough to show approximate swirl ratio values, trends and get a basic idea that swirl is present in intake systems and values are big enough to make influence on compressor performance.

5.1.2 Swirl ratio for flow bench

Table 5.2 summarizes swirl generator simulations for vane angle range from 0° to 40°. Simulations for these vane angles were done in the same setup as will be tested in flow bench to be able to compare CFD results with experimental tests. Results showed that these devices are covering range from 0 to 0.18 of swirl ratio.

Same simulations were done with improved vane design. Pressure drop results in 5.2 showed that for maximum case of 40° pressure drop was reduced from 450 Pa to 100 Pa. As this was really small pressure drop it was assumed that improved design is sufficient and pressure drop effect on compressor map will be insignificant. Swirl ratio results in Figure 5.1 and Table 5.3 confirmed that different swirl device can create totally different magnitude of swirl. At vane angle of 10° swirl ratio difference is 57% and at 40° 19%.

Table 5.1: Summary of benchmarked system swirl results

System	Swirl ratio
System 1	0.0266
System 2	0.1762
System 4	0.0759
System 5 (14w07)	0.0448
System 5 (14w11)	-0.0830
System 5 (14w19), with flow straightener	0.1007
System 5 (14w19), without flow straightener	-0.0815
System 6	-0.04
System 7	0.024

Table 5.2: Swirl ratio for vane angles from 0 to 40 degrees

		Vane angle								
		0	5	10	15	20	25	30	35	40
Distance from vanes, mm	0,01	0,0102	0,0363	0,0646	0,0864	0,1059	0,1137	0,1371	0,1559	0,1666
	0,05	0,0039	0,0350	0,0627	0,0850	0,1046	0,1167	0,1452	0,1740	0,1981
	0,1	0,0032	0,0341	0,0600	0,0800	0,0970	0,1080	0,1350	0,1639	0,1874
	0,2	0,0023	0,0328	0,0561	0,0728	0,0901	0,1051	0,1326	0,1618	0,1861
	0,3	0,0017	0,0320	0,0544	0,0696	0,0884	0,1041	0,1313	0,1602	0,1840
	0,4	0,0013	0,0314	0,0539	0,0686	0,0874	0,1028	0,1293	0,1572	0,1799
	0,5	0,0010	0,0308	0,0536	0,0677	0,0861	0,1009	0,1267	0,1535	0,1754

Table 5.3: Swirl ratio comparison for different design

		Vane angle		
		35	35B3	35B4
Distance, mm	0,01	0,1559	0,1945	0,2204
	0,05	0,1740	0,1927	0,2189
	0,1	0,1639	0,1910	0,2173
	0,2	0,1618	0,1881	0,2144
	0,3	0,1602	0,1854	0,2117
	0,4	0,1572	0,1825	0,2075
	0,5	0,1535	0,1784	0,2023

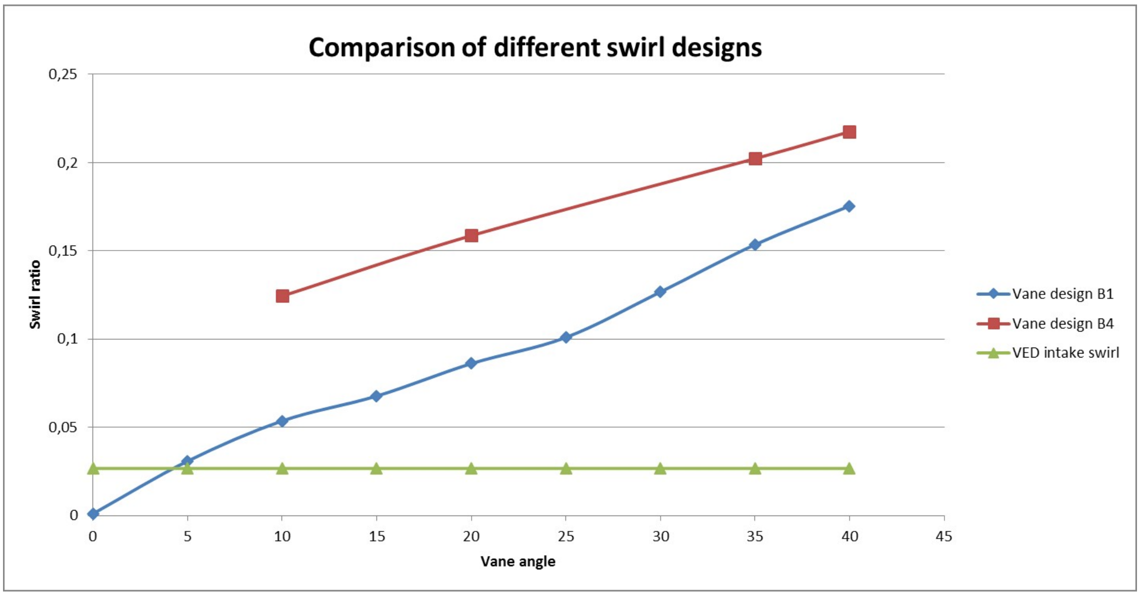


Figure 5.1: Comparison of swirl ratio created by initial and final vane design. System 1 swirl ratio is included as baseline for reference.

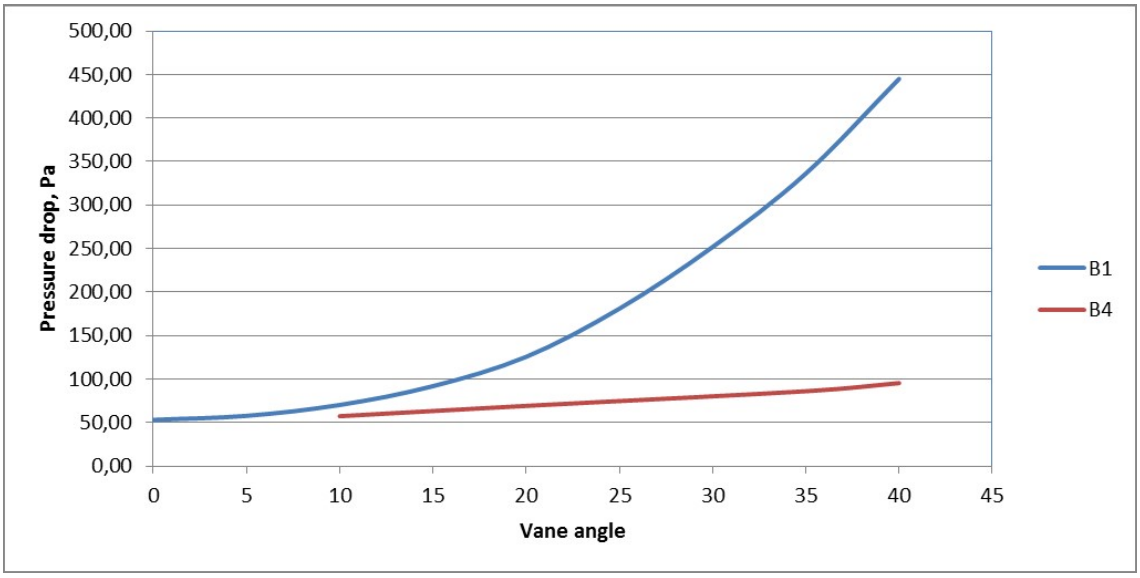


Figure 5.2: Pressure drop of B1 and B4 swirl generator

5.1.3 Swirl ratio for turbocharger test rig

Swirl device #20 was simulated with setup that can be seen in the figure 4.2. After the bench conical inlet, same as in the compressor, is attached and then conical expansion is attached. Swirl change while flowing through this outlet can bee seen in figure 5.3. Results show that reduction of diameter from 76 mm to 40 mm gives a swirl reduction of 40%. This had to be taken into account for testing in turbocharger rig. If swirl will be reduced to much it will not give required effect on the compressor performance.

Therefore swirl generators with vane angles from 0 to 45 were simulated in the setup that will be used in turbocharger test rig. Main difference between flow bench and turbocharger test rig was outlet. Flow bench had straight outlet and turbocharger bench had a conical compressor inlet. As it can be seen from Figure 5.4 swirl devices will cover swirl ratio range from -0.06 to 0.13.

5.1.4 Swirl ratio in 90° bend

Results of swirl behaviour in 90° bend can be seen in Figure 5.5.

For marginal swirl ratios (up to 0.2) pressure drop in the bend is not increasing compared to no swirl condition, also swirl decay is so small that can be neglected. When swirl ratio is more than 0.2 pressure drop starts to increase significantly and for swirl ratio of 0.3 is already two times bigger. Swirl decay is also increasing with higher ratios.

Investigation for swirl generator positioning was also done for 90° bend. Generator positioning can be seen in Figure 5.6. Results for swirl creation in 90° bend can be seen in Figure 5.7 and Table 5.4. Pressure drop is not affected for cases 10, 70 and 90. However there is a trade off between swirl ratio in the outlet and uniformity. Case

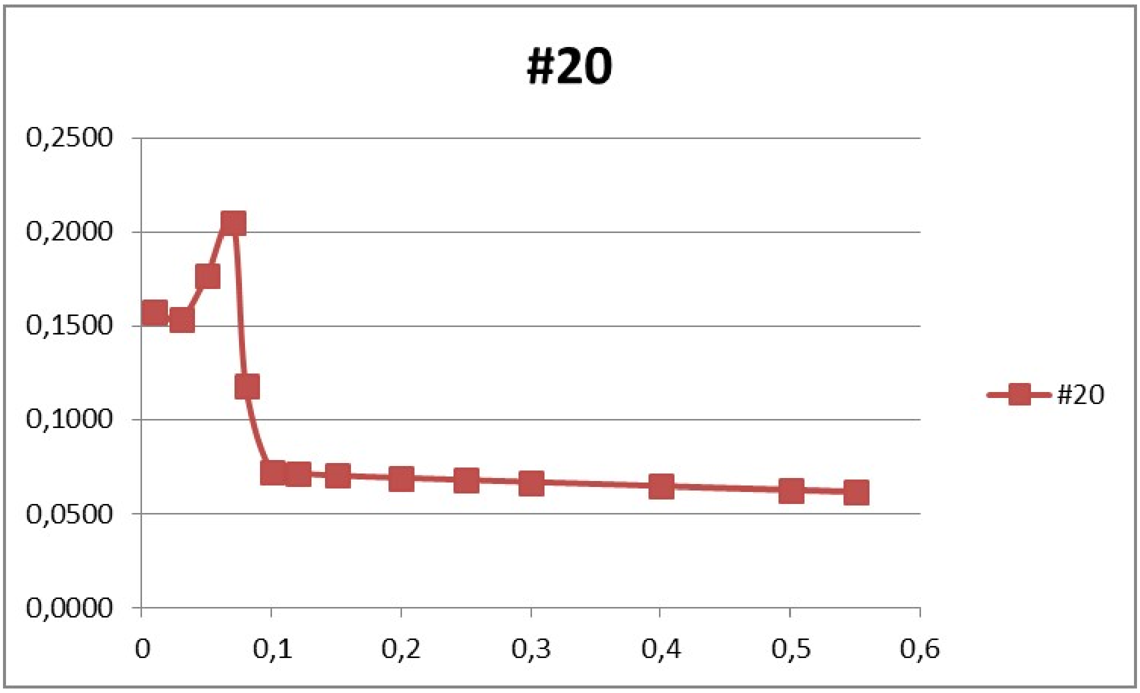


Figure 5.3: Swirl ratio change in the turbocharger rig setup.

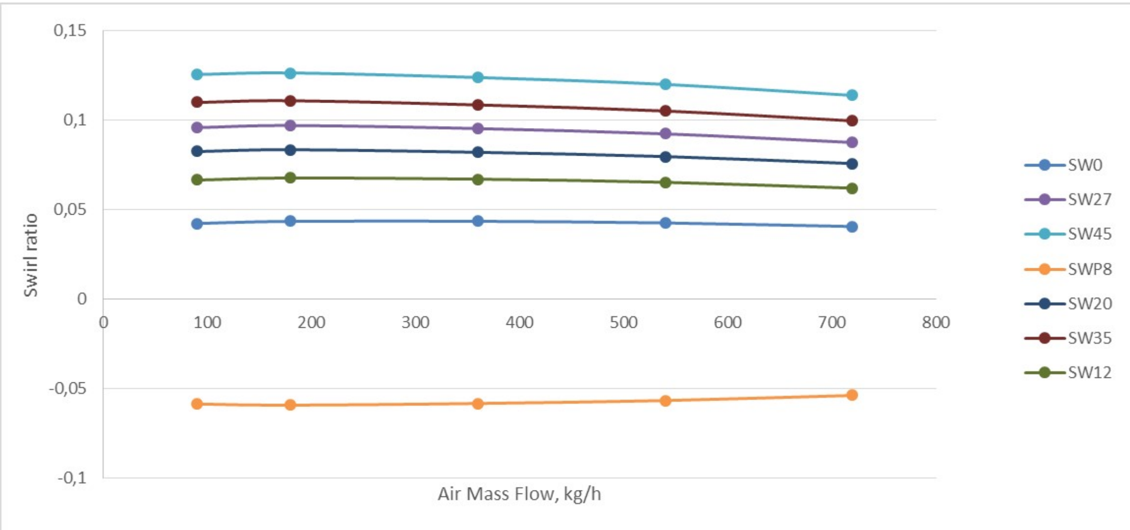


Figure 5.4: Swirl created in turbocharger test rig.

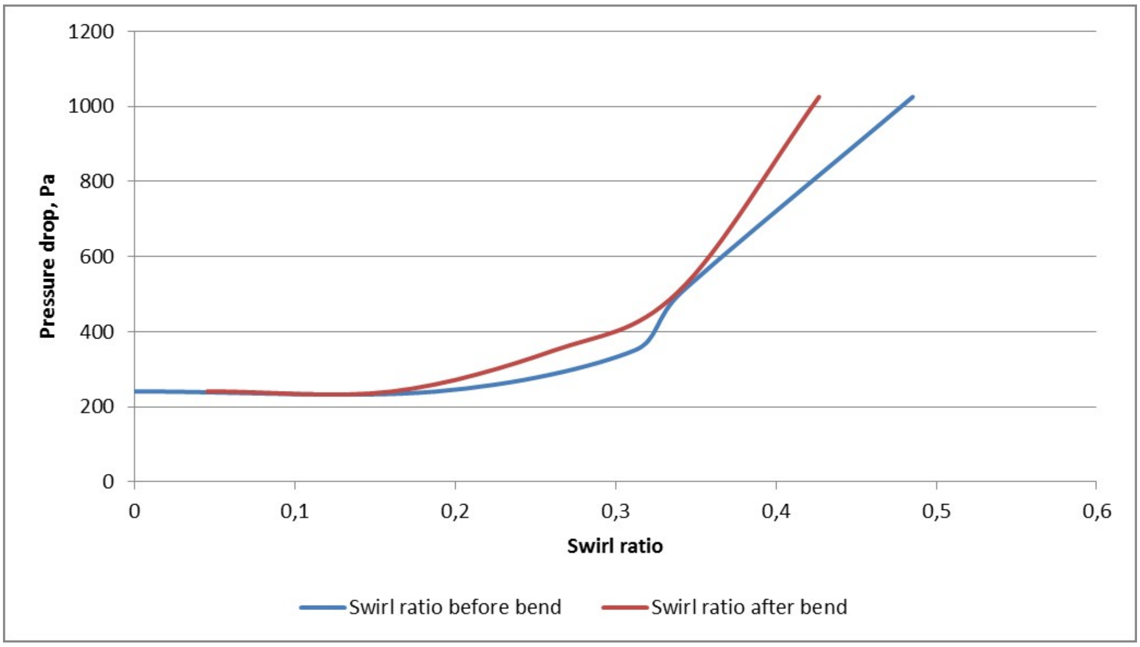


Figure 5.5: Swirl and pressure drop results in 90° bend.

Table 5.4: Pressure drop and uniformity results for swirl generators in 90° bend

Swirl generator position	pressure drop	flow uniformity
10	472.40	0.9262
50	537.65	—
70	472.26	0.9594
90	471.42	0.9605
90s	337.76	0.9610
0s	373.86	0.9514

90 gives least swirl but best uniformity and case 10 is opposite. Case 50 gives highest pressure drop because of it's position, which gives a lot of separation. Cases 90s and 0s have straight vanes instead of angled, therefore can be compared only to each other. It can be seen that it is beneficial to have flow straightener before the bend instead of having it after as both pressure drop and flow uniformity results are better.

5.1.5 Initial CFD simulations

Results from CFD simulations with different turbulence models and change in mesh settings can be seen in the Figure 5.8. It can be seen that swirl ratio modelled by

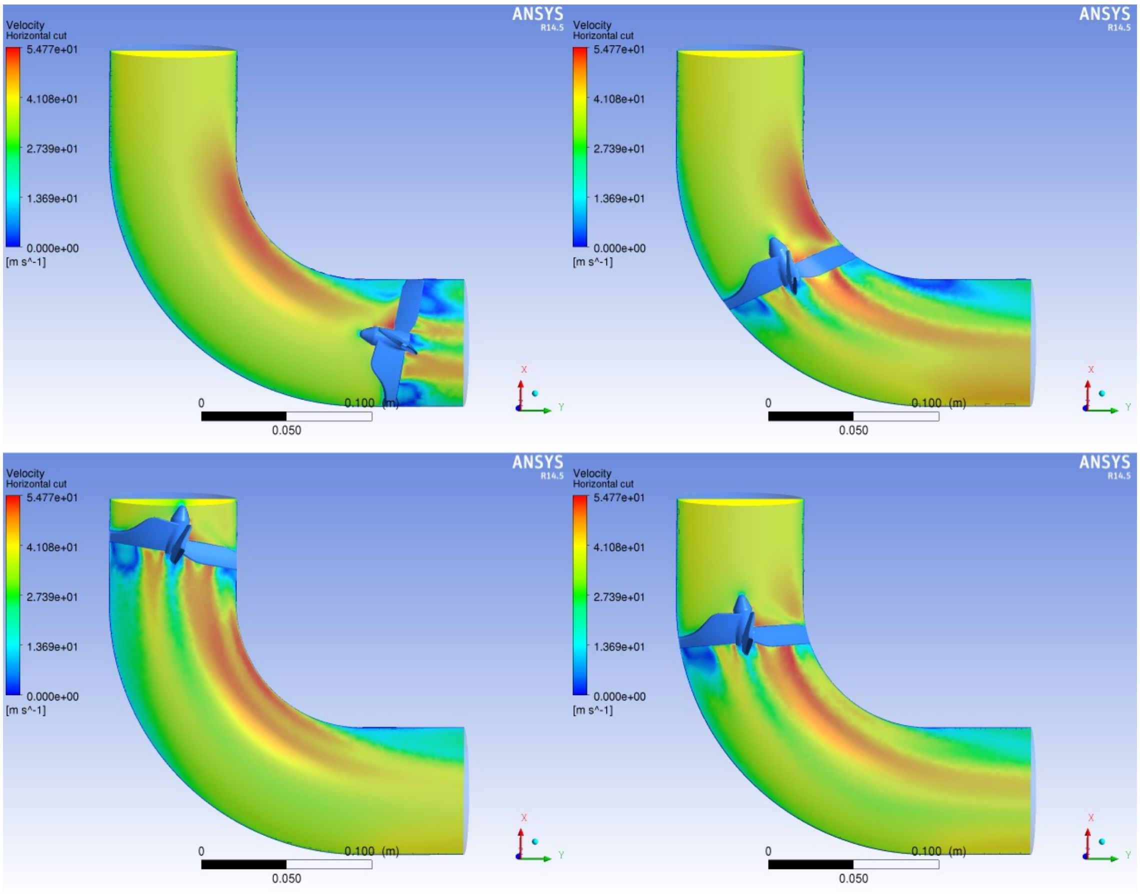


Figure 5.6: Swirl generator positioning in 90° bend.

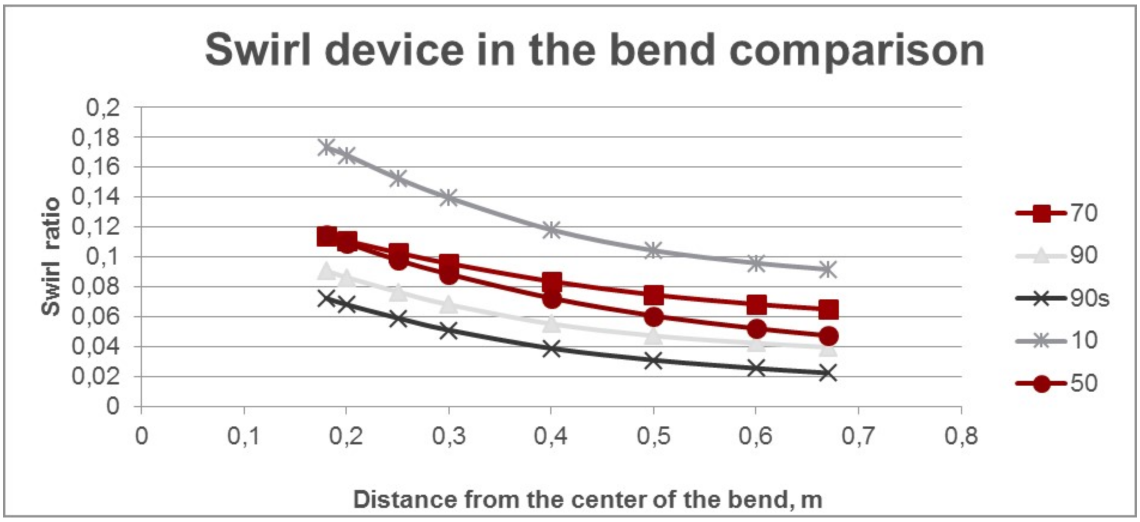


Figure 5.7: Swirl generator results in 90° bend.

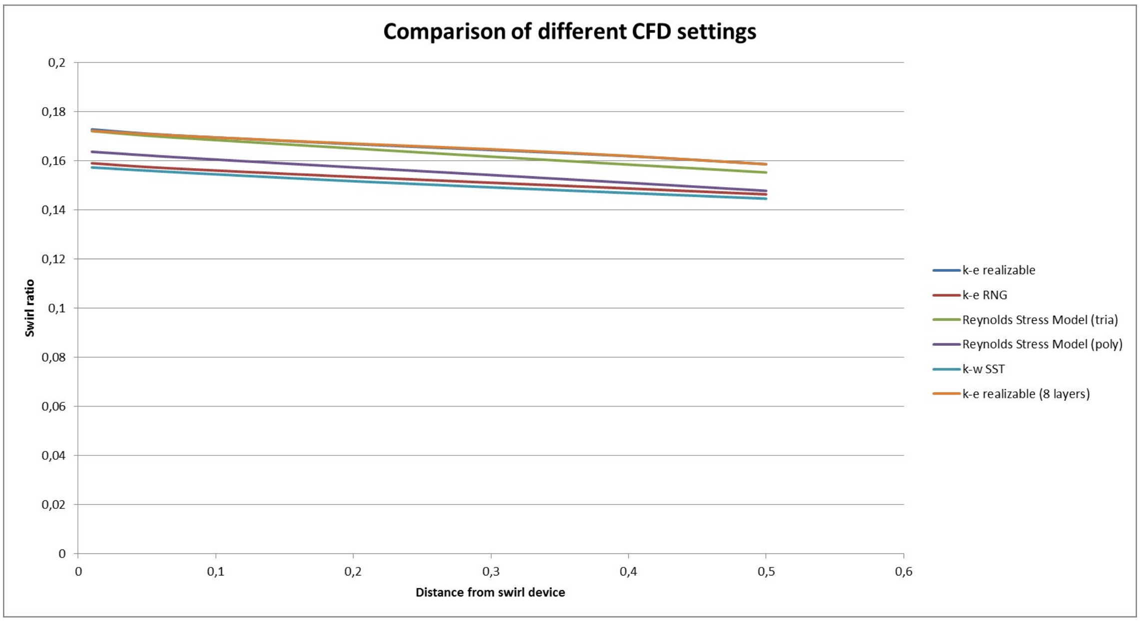


Figure 5.8: Comparison of swirl results with different turbulence models and mesh settings.

different turbulence models can have difference up to 10%. It was also found out that difference in pressure drop between Reynolds Stress model and two equation models was up to 15%.

5.2 Flow bench

Two tests were done in flow bench - swirl ratio and pressure drop. Initial swirl results are summarized in Figure 5.9.

Figure 5.9 shows measured magnitude of swirl. These results were corrected by subtracting swirl that is created naturally in the flow bench (Utan swirl) from the results with swirl generators. Corrected results can be seen in the Figure 5.10.

Relationship between swirl ratio and vane angle in this experimental setup can be seen in Figure 5.11. As these models were made from rapid prototyping they had quite poor tolerances, therefore results have a bit wavy line, however it could be assumed that relationship is linear.

Range covered by swirl devices is from 0.1 to 0.24 for swirl ratio. The reason why it is higher than range which is a target for thesis (0 to 0.15) is that they have different setups. These devices will be placed in turbocharger test bench setup where they have conical inlet and reduce their swirl by 40%, that will change the range of 0.1 - 0.24 to approximately 0.06 - 0.144. However simulations with exact results are described later in this chapter.

It was noticed that with increase of mass flow swirl ratio is also increasing. Influence of mass flow on swirl ratio can be seen in Figure 5.12.

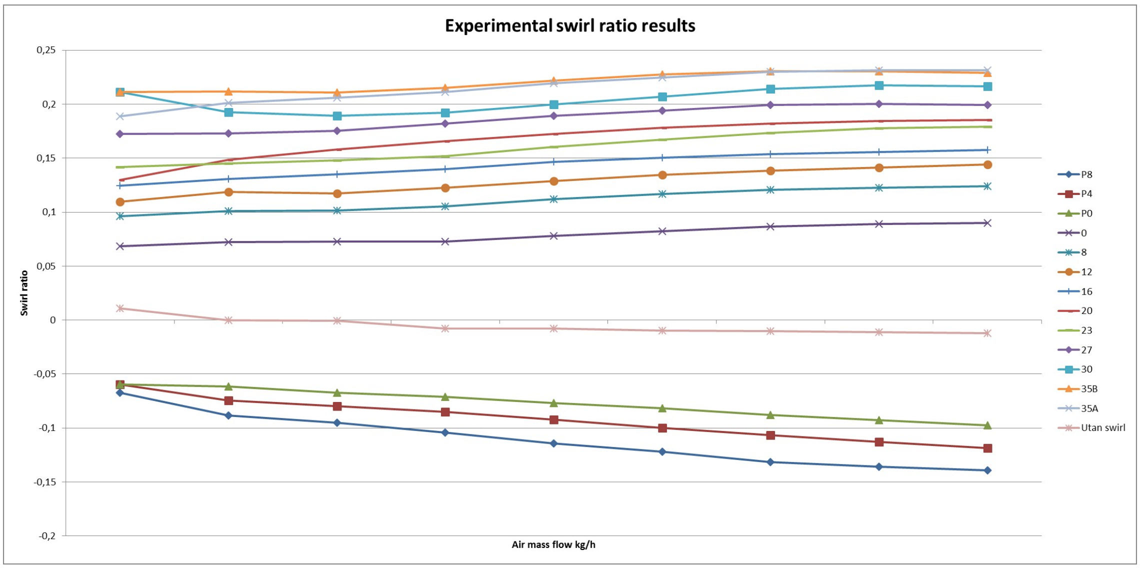


Figure 5.9: Initial results of swirl in the flow bench after post processing.

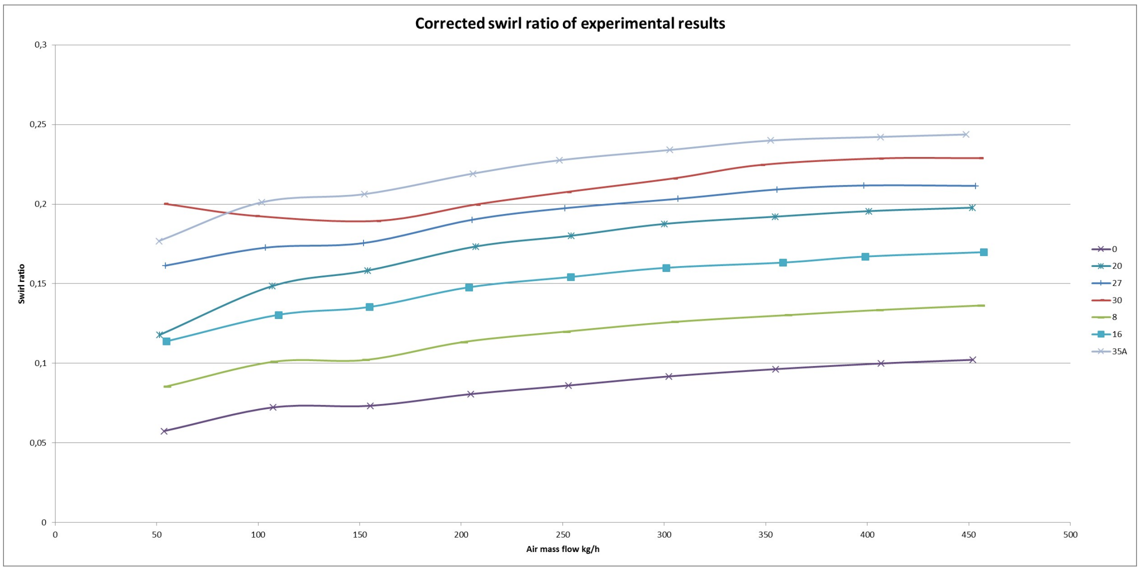


Figure 5.10: Corrected results of swirl in the flow bench.

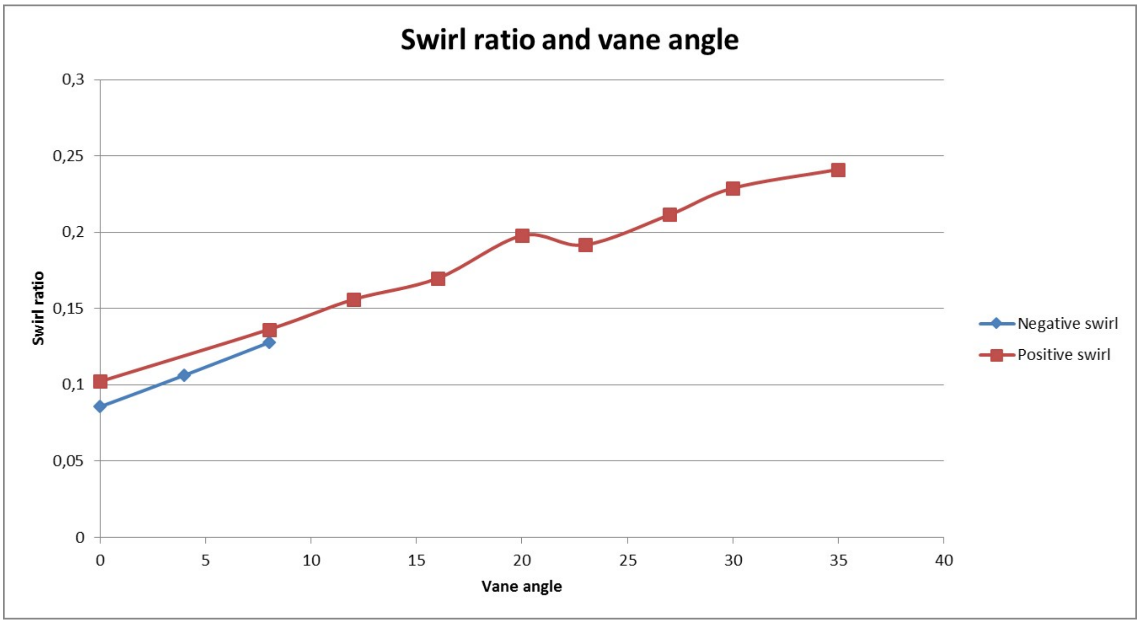


Figure 5.11: Corrected results of swirl in the flow bench.

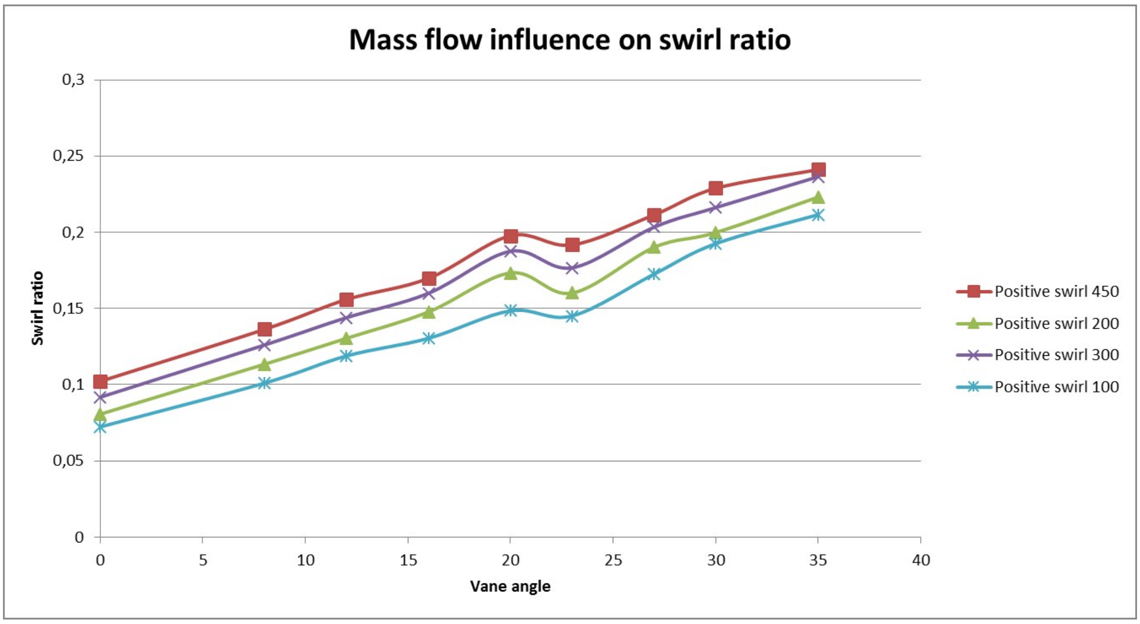


Figure 5.12: Mass Flow influence on swirl ratio.

Table 5.5: Comparison of CFD and experimental results

CFD settings	SR	Difference	Δ p	Difference
k-ε RNG	0,1909	21,66%	53,85	5,7%
k-ε Realizable	0,2054	15,72%	52,70	6,18%
Reynolds Stress Model	0,2021	17,08%	46,34	17,5%
k-ω SST	0,1931	20,76%	53,20	5,28%
Non equilibrium wall function				
k-ε RNG v3	0,1877	22,98%	53,85	4,13%
k-ε RNG v4	0,1969	19,21%	—	—
Reynolds Stress Model v2	0,2161	11,35%	49,53	11,81%
Reynolds Stress Model v3	0,2159	11,42%	49,54	11,81%
13 layers, aspect ratio 0.08				
Reynolds Stress Model v5	0,1796	26,3%	63,63	-13,29%
k-ω SST v2	0,1851	24,51%	53,20	5,28%
k-ε RNG v5	0,2282	6,38%	57,62	-2,58%
Enhanced wall treatment				
Reynolds Stress Model	0,2236	8,25%	46,79	16,71%
k-ε Realizable (ewf)	0,2311	5,15%	55,99	0,32%
Experimental	0,2437	—	56,17	—

5.2.1 CFD and Flow rig comparison

Results in Table 5.5 show the comparison of CFD results with different settings and experimental values. Results are shown for mass flow value of 450 kg/h as it was maximum value reachable by experimental equipment.

Initial results showed almost 20% deviation from experimental results. From four initial models Reynolds Stress Model and k-ε showed best results with difference from 15% - 17%.

All models had basic settings used at AIS (Table 4.1). As initial settings showed poor results a non-equilibrium wall function was tried and showed some improvement. Reynolds Stress Model results were 11% off and it was closest to experimental. However it seemed that for other models change of wall function didn't have significant influence.

To improve sensitivity to different wall function it seemed that best way is to refine the mesh. In addition requirement for enhanced wall function was to have Y+ values close to 1 and basic settings with two layers seemed more suitable for laminar flow then complex swirling flow.

Table 5.6: Maximum and average efficiency of initial runs

Swirl generator	Maximum efficiency	Average efficiency
Baseline	0.7579	0.6326
#35	0.7604	0.6387
#45	0.7575	0.6450
#20	0.7589	0.6451
#P8	0.7454	0.6250
System 1	0.7140	0.6044
System 2	0.7465	0.6051
Baseline #2	0.7300	0.6039
#P0	0.7273	0.5706
#0	0.7318	0.5936

Finally enhanced wall function with refined mesh was tried for RSM and k-ε realizable models. k-ε showed best match so far as it was only 5% off from experimental results.

5.3 Turbocharger gas stand

Initially comparison was done with average and maximum efficiencies seen Table 5.6. These results gave nice insight into efficiency change trend. However more in-depth analysis of the results was also conducted using Volvo turbocharger gas stand post-processing code were full compressor maps were plotted.

5.3.1 Full intake system tests

Two full intake systems were test in turbocharger test rig because of two reasons. Firstly, System 1 was measured for a comparison with compressor map that has no intake system, that shows a difference that can be made by intake system. Secondly, System 2 was measured. Flexible vane swirl generator changes its vane angle with increased air mass flow and should adjust swirl to changing conditions. Figure 5.13 compares maps with and without swirl device. Two effects can be noticed; surge line has moved to the left and efficiency of compressor map increase by 0.5%.

5.3.2 Surge line

Results in the Figure 5.14 shows three compressor maps plotted on top of each other. Only speed lines, surge line and choke line, which is equal to 50% efficiency is shown.

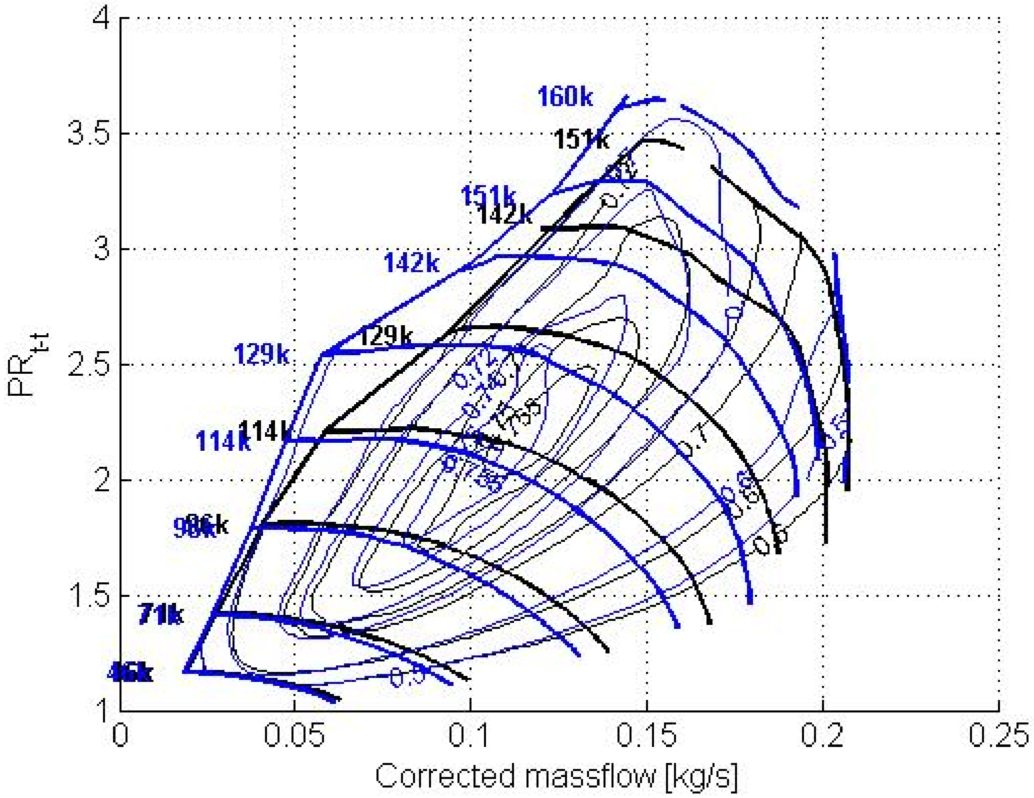


Figure 5.13: Comparison of System 1 and System 2

It can be seen that positive swirl (green map) shifts compressor map to the left. While negative swirl moves map to the right. Positive swirl had a magnitude of 0.13 and negative - -0.06.

Surge lines seen in the results are not parallel and in some areas closer to each other. This is because it is hard to define surge conditions and for different cases these conditions could differ, which affects exact surge line points. It is more accurate to look at speed lines, which are quite stable and shows how compressor map is shifted and confirms that positive swirl shifts map to the left and negative to the right.

5.3.3 Efficiency

Main focus in thesis work was on compressor efficiency change. It was expected to reach best efficiency at around 20° of vane angle, which is approximately 0.1 of swirl ratio.

Results in Figure 5.15 shows efficiency points for compressor full load curve with different maps. As it was expected, positive swirl is improving efficiency and negative is reducing. Best efficiency points in this case were reached at 0.11 of swirl ratio. Further increase in swirl ratio to 0.125 had a small drop in efficiency, therefore it was assumed

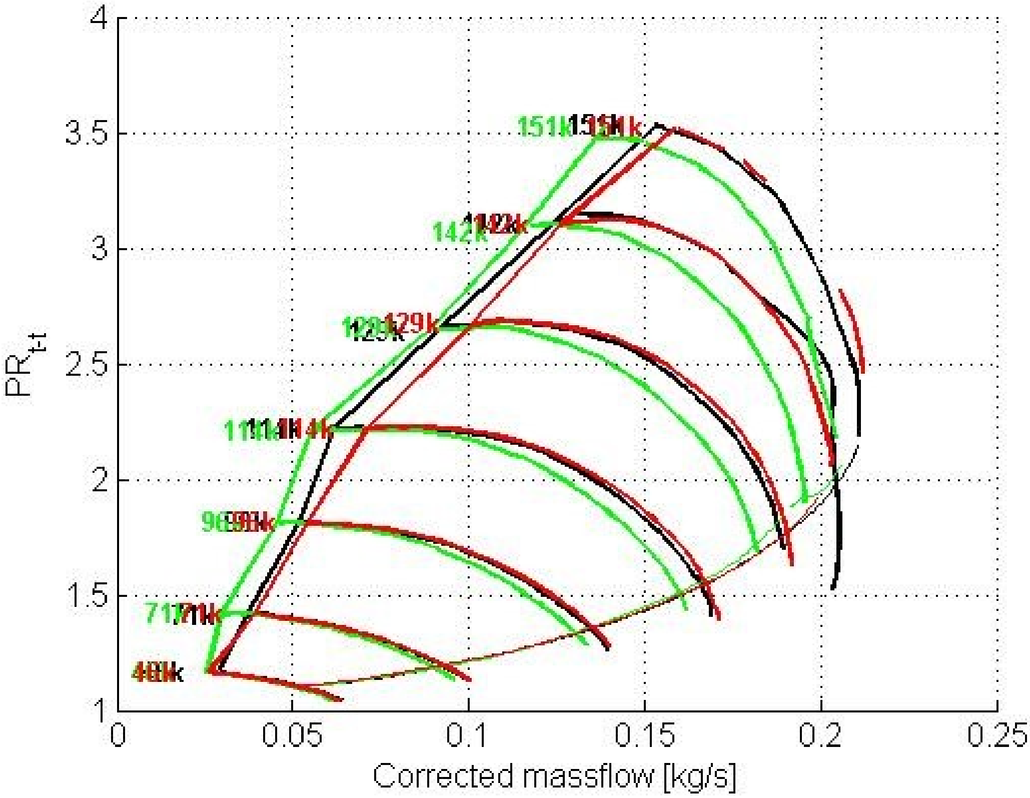


Figure 5.14: Surge line comparison

that 0.11 of swirl ratio is at the best efficiency point.

-8 and 35

Results in Figure 5.16 shows the efficiency difference between two compressor maps. One with swirl ratio of 0.11 - best efficiency map and another one with swirl ratio of - -0.06, which has worst efficiency. Black line is a full load operating curve. By comparing these two maps it can be seen that while compressor is operating in full load conditions it works in 1.5% - 2.5% better efficiency range.

Results in Figure 5.17 shows compressor maps of -0.06 and 0.11 plotted on top of each other. It can be seen that map with positive swirl of 0.11 not only increases maximum efficiency, but also expands efficiency "islands" in all directions. Looking at efficiency line of 0.7 it can be seen that there is a trend for efficiency to shift towards lower mass flow and lower pressure ratio area.

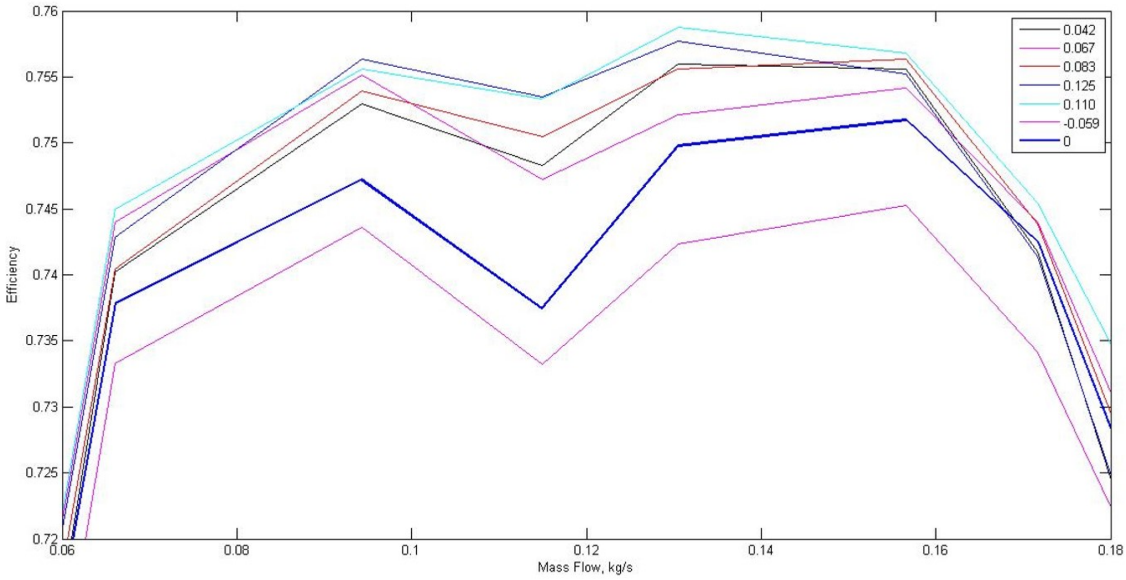


Figure 5.15: Efficiency comparison for full load curves

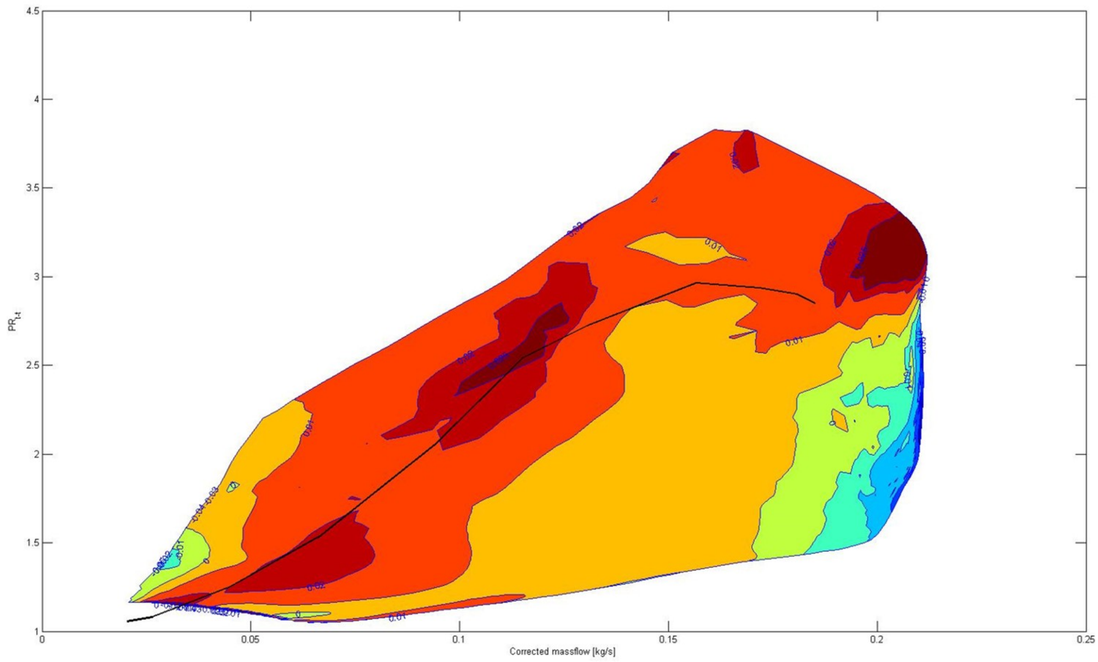


Figure 5.16: Efficiency comparison for 0.11 and -0.06 swirl ratio maps

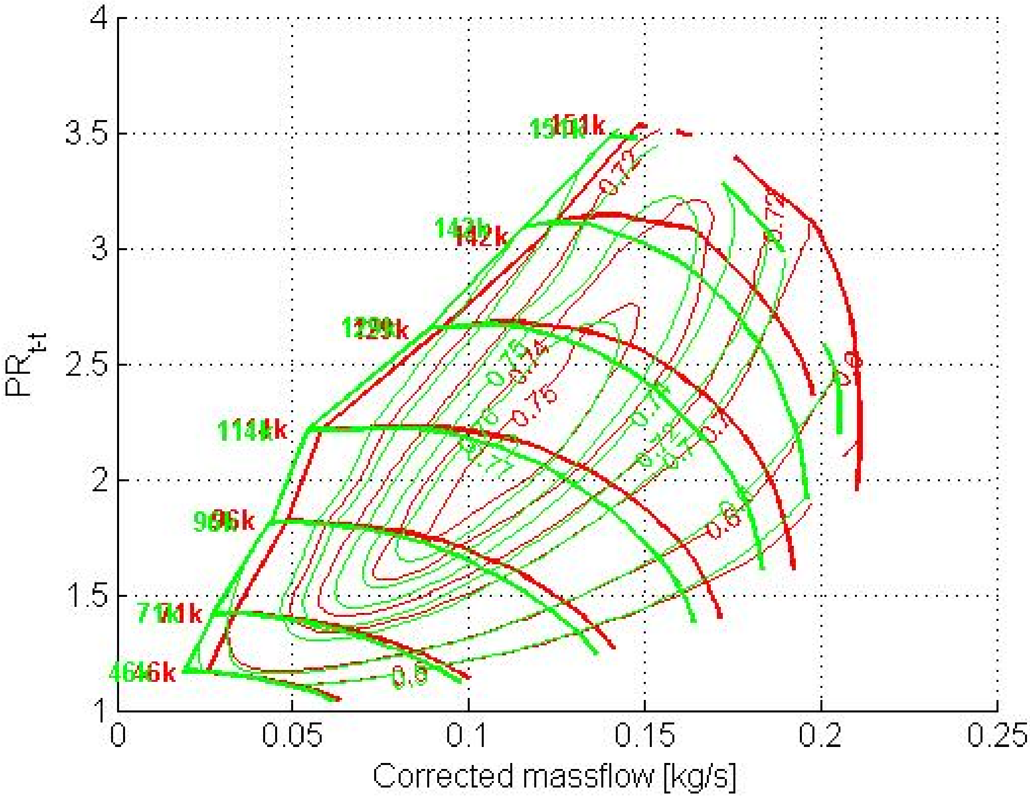


Figure 5.17: Efficiency comparison for 0.11 swirl and -0.06 swirl ratio maps

0 and 35

Figure 5.18 shows comparison of compressor maps with no swirl and 0.11 swirl. Again efficiency improvement can be clearly seen and the trend for efficiency "islands" to move towards lower mass flow and lower pressure ratio area is evident.

0 and P8

Figure 5.19 shows comparison of compressor maps with no swirl and -0.06 swirl ratio. Two trends can be seen - efficiency is lower for negative swirl, efficiency "islands" are smaller. It can be strange that no expected efficiency "island" movement to the right is evident, however it could have been overcome by efficiency reduction and be the reason why it is not seen.

Map movement

Figure 5.20 shows compressor surge, choke (0.5 efficiency), 0.76 efficiency and speed lines for all tested positive swirls. A few trends can be seen from this comparison, firstly,

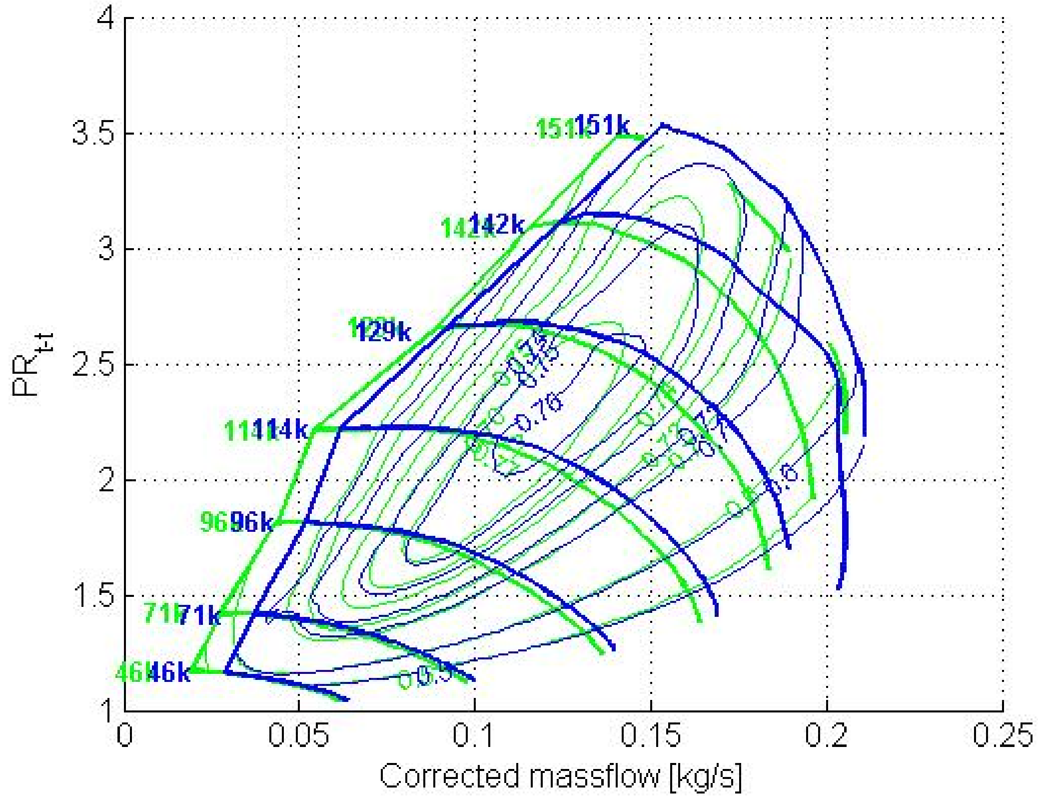


Figure 5.18: Efficiency comparison for 0.11 swirl and no swirl ratio maps

speed lines clearly show that with increasing amount of swirl map is shifted towards the left side, secondly increasing amount of swirl is expanding the 0.76 efficiency islands. An islands with 0.082 (#20) of swirl ratio seems to be the biggest and shifted upwards, however it doesn't follow the overall trend, so it could be assumed that there was some errors in measurement. From other three maps it can be seen that red islands with swirl ratio of 0.11 is biggest and confirms the initial results that ratio of 0.11 gives the maximum efficiency.

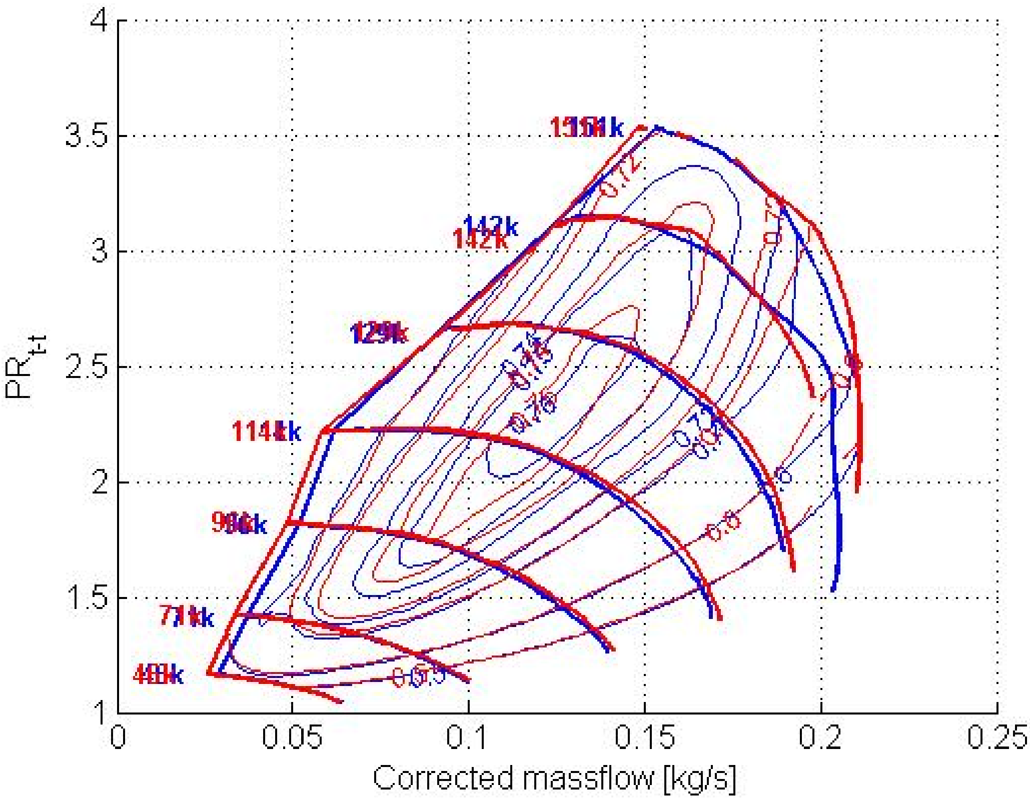


Figure 5.19: Efficiency comparison for -0.06 and no swirl ratio maps

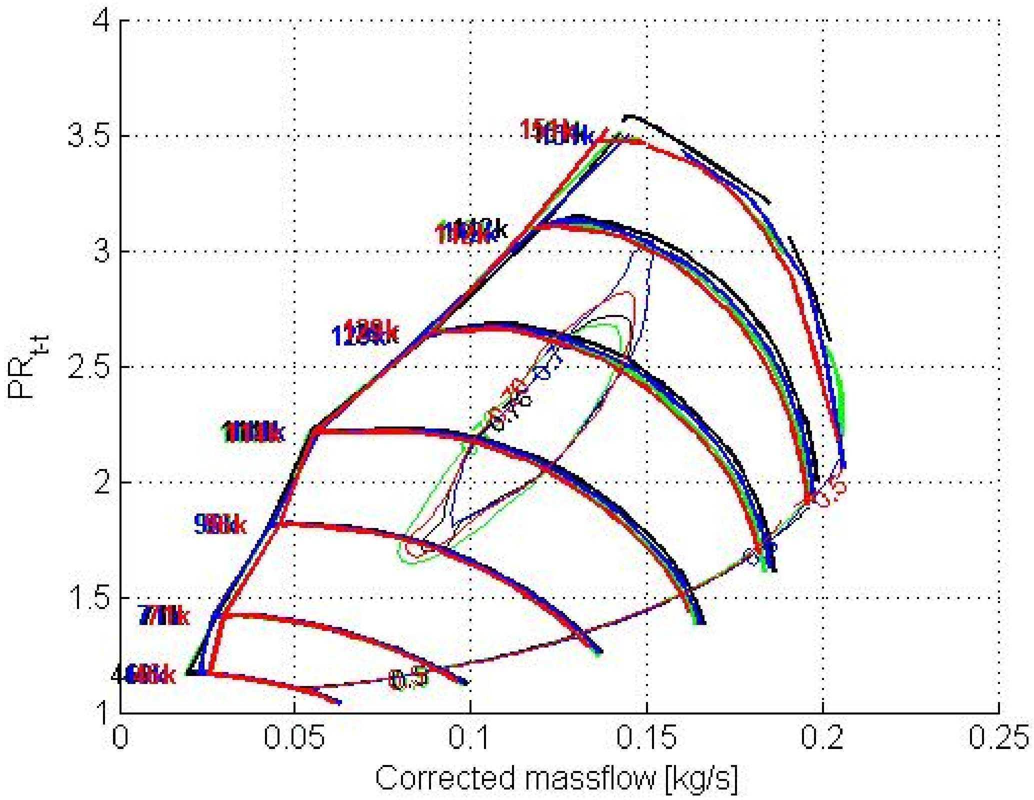


Figure 5.20: Comparison of maps with increasing amount of swirl

6

Discussion and conclusions

Master thesis results showed that small amounts of swirl could give positive effects on compressor efficiency. Even small negative swirl of -0.06 of swirl ratio is giving negative effects. Marginal amounts of positive swirl can improve efficiency by up to 0.8%. For given compressor it was found out that best efficiency improvement is gained at swirl ratio of 0.11, which matches results of other conducted experiments where efficiency improvement was evident at around 0.1 of swirl ratio.

In addition it was confirmed that positive swirl moves compressor map to the left side and improves surge line and negative swirl moves compressor map to the right side and could improve choke line.

Finally, to understand how important this topic is for AIS a number of air intake systems were benchmarked to find out how much swirl is usual in common air intake system.

It was seen that benchmarked intake systems can have from -0.0830 to 0.1007 of swirl ratio and affect compressor efficiency by up to 2%, therefore it is important to evaluate swirl of the system during design process.

Another important change of this thesis compared to other investigations was swirl measurement method. Most of the articles used vane angles to measure its influence on turbocharger compressor. However vane angle was not a good analytical value that could be used to compare different experiments, therefore swirl ratio was used in this report. In addition swirl behaviour in the ducts was investigated to see how swirl is decaying in different shapes found in air intake system.

Finally, a number of swirl creation ideas were suggested for the design engineers, to have an idea which of them are better. It was found out that small devices with vanes have a big advantage in simplicity of design, manufacturing and packaging even though they are creating a little bit higher pressure drop.

7

Recommendations

THIS CHAPTER SUGGESTS FUTURE WORK that can be done in area covered by master thesis project. Four areas of investigation are proposed that potentially can improve or extend project and give promising results.

- Engine 1D simulations simulations with new compressor maps
- Low Pressure Exhaust Gas Recirculation system for swirl creation
- NVH testing
- Turbocharger response time test

7.1 Engine 1D simulations

To find out compressor efficiency improvement influence on engine performance, 1D engine gas exchange process simulations should be done.

Two simulations were done during this master thesis project. However, the only change that has been done was compressor map change and no optimization or calibration for new compressor maps has been done. Simulations fully adapted to new compressor maps would show more accurate results of engine performance change.

7.2 Low Pressure EGR

Air injection idea mentioned in Appendix C could also be used as a part of Low Pressure Exhaust Gas Recirculation system. This idea will not only reduce exhaust emissions but could also improve compressor working efficiency.

7.3 NVH testing

NVH section in Literature Review chapter analysed how swirling airflow in the intake can affect whoosh noise caused by turbocharger. Some testing was planed to be done in master thesis presentation, unfortunately it was not possible to do them because of tight NVH testing schedule. Two adjustments were designed for System 1 and System 3 intake system. System 1 was redesigned to fit BMW swirl device and System 3 has a designed swirl device that fits system and creates 0.1 of swirl ratio. As this ratio is also giving improvement in compressor efficiency, it would be also interesting to see how much improvement it could give in noise caused by compressor.

7.4 Turbocharger response time

Number of articles mentioned that improvement in turbocharger response time can be seen with swirling flow in the inlet. [14] mentioned that at full load conditions turbocharger response time can be reduce by up to 24%, and part load conditions even by 40%. It would be interesting to determine if swirling flows that increase compressor efficiency also reduce compressor response time.

Bibliography

[1] G. C. T. Morris, The effect of air inlet system features on automotive turbocharger compressor performance, Tech. rep., Ford Motor Company Limited (2010).

[2] J. R. Serrano, X. Margot, A. Tiseira, L. M. Garcia-Cuevras, Optimization of the inlet air line of an automotive turbocharger, International Journal of Engine Research 14 (2013) 92–104.

[3] D. Li, C. Yang, M. Zhou, Z. Zhu, H. Wang, Numerical and experimental research on different inlet configurations of high speed centrifugal compressor, Science China Technological Sciences 55 (2012) 174–181.

[4] O. Moos, F. R. Klimetzek, R. Rossmann, Bionic optimization of air-guiding systems, Polymer 2013 (2004) 08–15.

[5] J. Galindo, F. Arnau, A. Tiseira, R. Lang, H. L. PhD, T. G. MD, Measurement and modeling of compressor surge on engine test bench for different intake line configurations (04 2011).
URL `http://dx.doi.org/10.4271/2011-01-0370`

[6] D. Evans, A. Ward, Minimising turbocharger whoosh noise for diesel powertrains (05 2005).
URL `http://dx.doi.org/10.4271/2005-01-2485`

[7] C. Teng, S. Homco, Investigation of compressor whoosh noise in automotive turbochargers (05 2009).
URL `http://dx.doi.org/10.4271/2009-01-2053`

[8] E. Sparrow, J. Abraham, W. Minkwycz, Flow seperation in a diverging conical duct: effect of reynolds number and divergence angle, International Journal of Heat and Mass Transfer 52 (2009) 3079–3083.

[9] A. K. M, K. Kanniah, Flow and pressure drop characteristics for various non-circular curved ducts used in automobile hvac system with different bend angles (04 2013).
URL `http://dx.doi.org/10.4271/2013-01-1645`

[10] D. Guentert, D. Todd, W. Simmons, Flow tests of an naca-designed supercharger inletelbow and the effects of varous components on the flow characteristics at the elbow outlet, Tech. rep., National Advisory Committee for Aeronautic (1946).

[11] P. Anschel, S. Roby, Duct for changing direction of flow, particularly for turbocharger compressor inlet, wO Patent App. PCT/US2008/079,978 (Apr. 23 2009). URL `http://www.google.com/patents/WO2009052170A2?cl=en`

[12] L. Ma, M. Paraschivoiu, J. Yao, L. Blackman, Improving flow uniformity in a diesel particulate filter system (05 2001).
URL `http://dx.doi.org/10.4271/2001-01-1944`

[13] Ramazan, Cfd simulation of swirling effect in a s - shaped diffusing duct by swirl angle 20, International Journal of Engineering Research and Applications 3 (2013) 51–59.

[14] M. Toussaint, P. Podevin, Guide-vanes upstream the impeller of centrifugal compressor, 5th European Conference on Turbomachinery Fluid Dynamics And Thermodynamics.

[15] H. Uchida, A. Kashimoto, Y. Iwakiri, Development of wide flow range compressor with variable inlet guide vane, R&D Review of Toyota CRDL 41 (2006) 9–14.

[16] F. Wallace, A. Whitfield, R. Atkey, Experimental and theoretical performance of a radial flow turbocharger compressor with inlet prewhirl, Proceeding of the Institution of Mechanical Engineers 189 (1975) 177–189.

[17] N. Fraser, T. Fleischer, J. Thornton, J. Rueckauf, Development of a fully variable compressor map enhancer for automotive application (04 2007).
URL `http://dx.doi.org/10.4271/2007-01-1558`

[18] H. Mohtar, P. Chesse, A. Yammine, J. F. Hetet, Variable inlet guide vanes in a turbocharger centrifugal compressor: Local and global study (04 2008).
URL `http://dx.doi.org/10.4271/2008-01-0301`

[19] S.-J. Schnorpfeil, S. Pischinger, P. Adomeit, S. Bowyer, Potentials of variable compressor pre swirl devices in consideration of different sealing concepts (04 2013).
URL `http://dx.doi.org/10.4271/2013-01-0934`

[20] B. B. Arora, B. D. Pathak, Effect of swirl on the intake of turbocharger (2008).

[21] J. Galindo, J. Serrano, X. Margot, A. Tiseira, N. Schorn, H. Kindl, Potential of flow pre-whirl at the compressor inlet of automotive engine turbochargers to enlarge surge margin and overcome packaging limitations, International Journal of Heat and Fluid Flow 28 (2007) 374–387.

[22] N. Kyrtatos, N. Watson, Application of aerodynamically induces prewhirl to a small turbocharger compressor, Journal for Engineering for Power 102 (1980) 943–950.

[23] E. Benisek, Pre-whirl turbo charger apparatus, uS Patent 3,922,108 (Nov. 25 1975).
URL `https://www.google.com/patents/US3922108`

[24] E. Hermann, R. Stec, D. White, J. Tomlin, W. Ronzi, R. Khami, R. Bonhard, B. Boddakayala, Inlet swirl control for turbochargers, uS Patent 8,177,498 (May 15 2012).
URL `https://www.google.com/patents/US8177498`

[25] R. Devi, P. Saxena, B. Walter, B. Record, V. Rajendran, Pressure reduction in intake system of a turbocharged-inter cooled di diesel engine using cfd methodology (06 2004).
URL `http://dx.doi.org/10.4271/2004-01-1874`

[26] W. S. A. Leong, A new approach to optimizing the clean side air duct using cfd techniques (2007).

[27] N. Zloch, C. Roemuss, Axial torque governor for a turbo-supercharger for internal combustion engines, uS Patent 4,780,055 (Oct. 25 1988).
URL `http://www.google.com/patents/US4780055`

[28] R. Cheng, For burning fuel efficiently and with minimal emission of pollutants, uS Patent 5,735,681 (Apr. 7 1998).
URL `http://www.google.com/patents/US5735681`

A

Swirl generation

RESULTS OF SWIRL INFLUENCE INVESTIGATION showed that it has noticeable influence on the compressor performance and needs to be controlled at the compressor inlet. To give some swirl generation ideas for the designer a number of swirl devices were designed and simulated to investigate which methods are more advantageous. Some of ideas are taken from the literature, others are original.

A.1 Initial shape ideas

A number of different patents are registered for swirl generation. The most popular and probably easiest way to create a swirl is guide vanes. Guide vanes is also the most popular choice used in the research as they are easy to design, manufacture and package. Also it gives a possibility to design a guide vanes that can be controlled by changing their angle and as a result changing generated swirl.

Main investigation in this thesis work was also done using swirl generator with four guide vanes. Initially a design with straight vanes was designed to repeat conditions in the experiment done by [19]. Later straight vanes were improved to reduce pressure drop and increase generated swirl as can be seen in Figure A.1.

Swirl generated by improved vanes was increased by 19% – 57% and changed range of swirl from 0.0536 – 0.1754 (straight vanes) to 0.1244 – 0.2175 (curved vanes) and can be seen in the Figure 5.1.

At the same time curved vanes improved air flow going around them by reducing separation and amount of caused turbulence (FigureA.2). Air flow improvement reduced pressure drop (Figure 5.2) by 18% for vane angle of 10° and 78% for vane angle of 40° .

A different type of guide vanes were inspired by Low - Swirl Burner patented by [28]. Even though it was designed to stabilize flame, it looked like a good solution for a turbocharger inlet. The advantage of this device for compressor inlet was that it's center part was not creating any swirl. For a compressor it seemed like a good solution

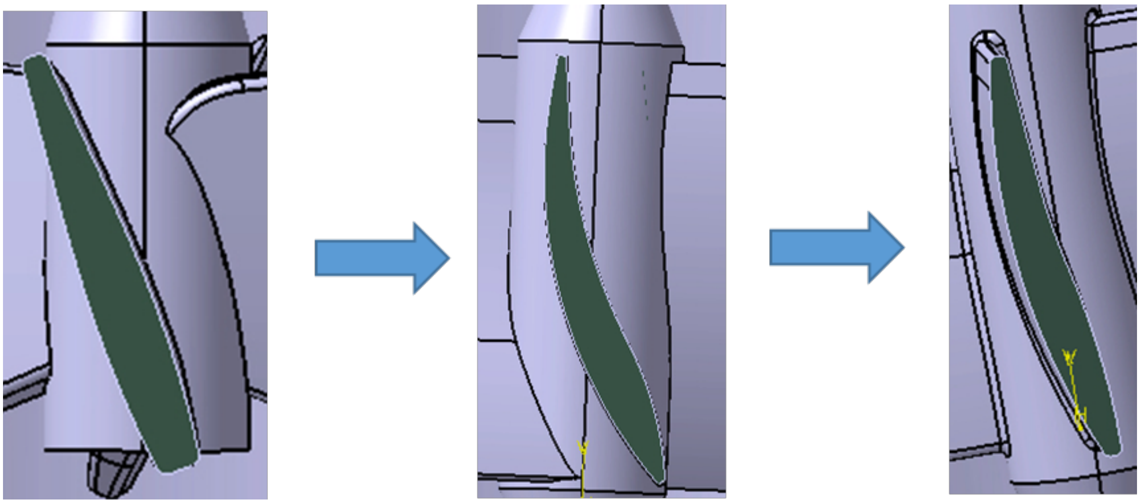


Figure A.1: Improvement of swirl generator used in experimental setup.

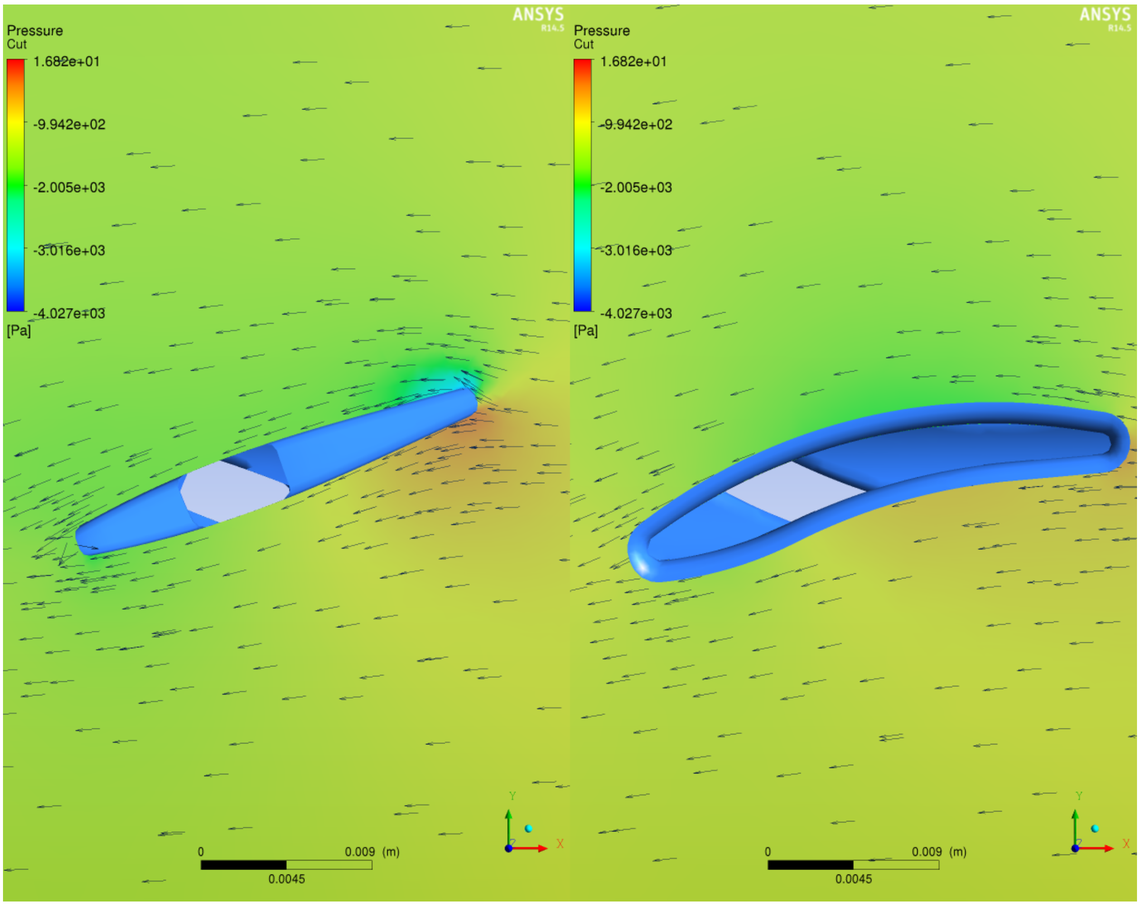


Figure A.2: Airflow comparison for straight and curved vanes.

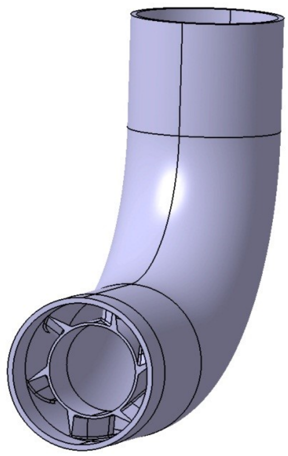


Figure A.3: Low - Swirl Burner swirl generator in the 90° bend.

as compressor wheel center part is blocked by a shaft and not producing any work. Therefore it was assumed that no swirling airflow would be needed for this part and Low - Swirl Burner would give needed effect with lower pressure drop as swirl is generated only on the outside of swirl generator.

A model for this device can be seen in Figure A.3. CFD results for this swirl generator showed that is created 507 Pa of pressure drop. 90° bend itself is creating 243 Pa of preesure drop, therefore it could be assumed that this swirl generator is itself is creating only around 264 Pa of pressure drop while creating 0.266 of swirl ratio.

Even though swirl devices with guide vanes are creating sufficient swirl they are still creating enough pressure drop to reduce the improvement on the turbocharger that can be achieved by swirling airflow.

A way to remove the need of guide vanes could be by separating part of the air flow and inducing swirl with it. A channel that take part of the flow and rotates around main flow was designed and can be seen in Figure A.4.

CFD simulation results showed that swirl generated by this system is 0.049 and is not enough to make any significant influence on compressor performance. The main reason for this was that flow was not separating into the rotating channel.

Looking more in-depth into CFD results in Figure A.6 left side it can be seen that velocity in the channel is much smaller than in the main flow. As this rotating channel is longer than straight section it is giving higher pressure drop (Figure A.7) left side. Higher pressure in the channel is the main reason why flow is not going into it and is not creating the swirl afterwards.

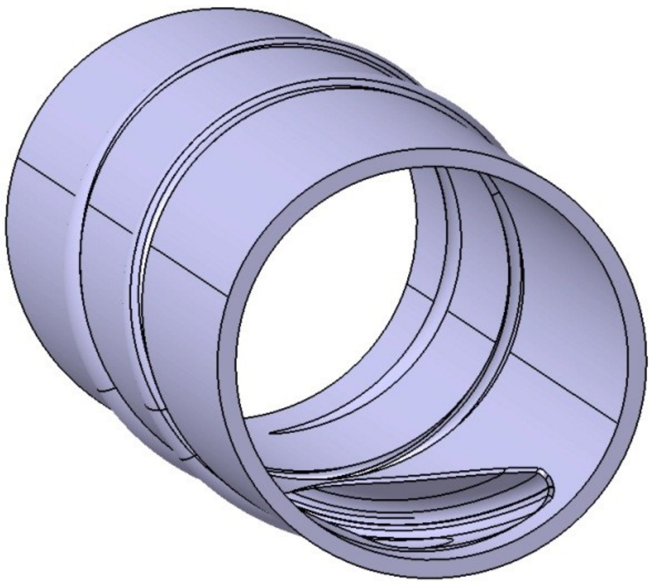


Figure A.4: Swirl generator with flow separation.

To force flow into the channel and see what affect it gives, a swirl generator with the same idea but closed channel was designed and can be seen in Figure A.5. Results showed that closed channel giving worse results. Pressure drop in this channel has increased even more as it can seen in the Figure A.7 right side, therefore flow is reduced even more and swirl created by this solution was only 0.008.

A possible way to reduce pressure drop created by guided vanes was to increase the distance where swirl is created. In the guided vanes swirl is usually created in a really short distance, therefore flow has to be turned quite fast and that creates high pressure losses.

A solution was a spiral shaped duct in a straight long distance seen in Figure A.8. Results showed that swirl created by this ducts was 0.1939 with a pressure drop of 357 Pa.

Another solution can be seen in Figure A.9. Channel was transformed into 4 tubes that are rotating around center axis. This solution gave a swirl ratio of 0.104 with a pressure drop of 275 Pa.

As most popular shape in the intake system is 90° bend it was decided to investigate some solutions were swirl is created with a help of this bend. Two ideas were tried that can be seen in A.10. Solution on the left has used the same spiral duct adapter to 90° bend and solution on the right has used two guide vanes. Results showed that spiral duct in the 90° bend created a swirl of 0.299 with a pressure drop of 604 Pa. As it was mentioned, naturally 90° bend creates a pressure drop of 243 Pa, therefore it can be assumed that pressure drop of swirl generation was 361 Pa. Results seemed much better than a solution were spiral duct was used in the straight line, with almost the

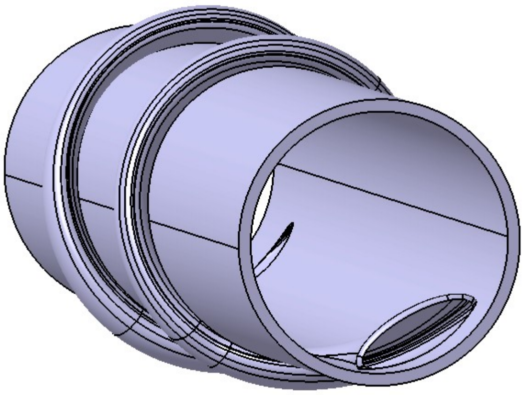


Figure A.5: Swirl generator with improved flow separation.

same pressure drop it gave 54% more swirl. Solution on the right created a swirl ratio of 0.22, however these vanes were causing a lot of separation and a total pressure drop in the duct was 1172 Pa which was much more than in any solution before. Subtracting pressure drop created by the bend it can be assumed that this pressure drop was 929 Pa, which is still much higher than other solutions.

A solution used by [21] tried guide vane design. However instead of implementing vanes axially to the flow they did it in 90° bend and turned the flow at the same time as creating swirl. This device was called SGD (Swirl Generation Device) instead of IGV (Inlet Guide Vanes) and worked on a bit different princple.

A similar device was designed and can be seen in Figure A.11. One significant advantage of this device was that it also uses 90° bend to generate swirl and reduce the amount of space needed by the bend. Results showed that swirl created by this device was 0.448 and pressure drop of 2241 Pa. As this was also using 90° bend it can be assumed that device itself is creating 1998 Pa of pressure drop.

Table A.1 summarizes results from different swirl devices. Pressure drop/Swirl ratio shoes how much pressure is needed to create a swirl ratio of 0.1 and is good approximation of how good is swirl generator. Results showed that guide vanes give the best results for swirl creation in addition they are easier to design, manufacture and package as other solutions need more space. A good solution is also spiral duct in the 90° bend as it gives pressure drop close to the guide vanes. However, solution like this is hardly manufacturable and it is complicated to adjust amount of created swirl.

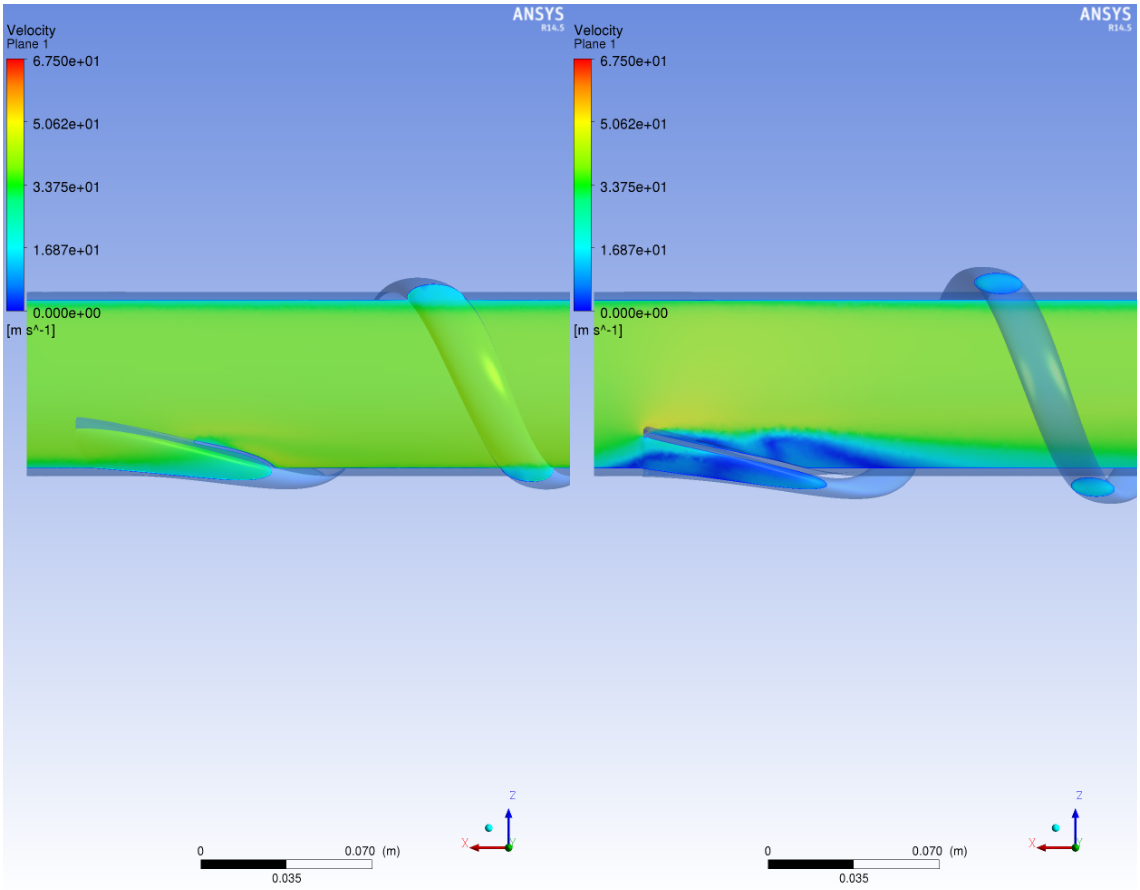


Figure A.6: CFD velocity results of swirl generator with flow separation.

Table A.1: Comparison of swirl generation devices

Swirl Device	Swirl ratio	Pressure drop, Pa	Pressure drop/Swirl ratio
B4	0.2175	95.42	43.8
SD12	0.2664	264	99.0
SD1	0.0495	99.35	200.7
SD2	0.0083	107.94	1300.4
SD4	0.1939	357.77	184.5
SD14	0.1043	275.26	263.9
SD6	0.2992	361	120.6
SD7	0.2286	929	406.3
SD11	0.4483	1998	445.6

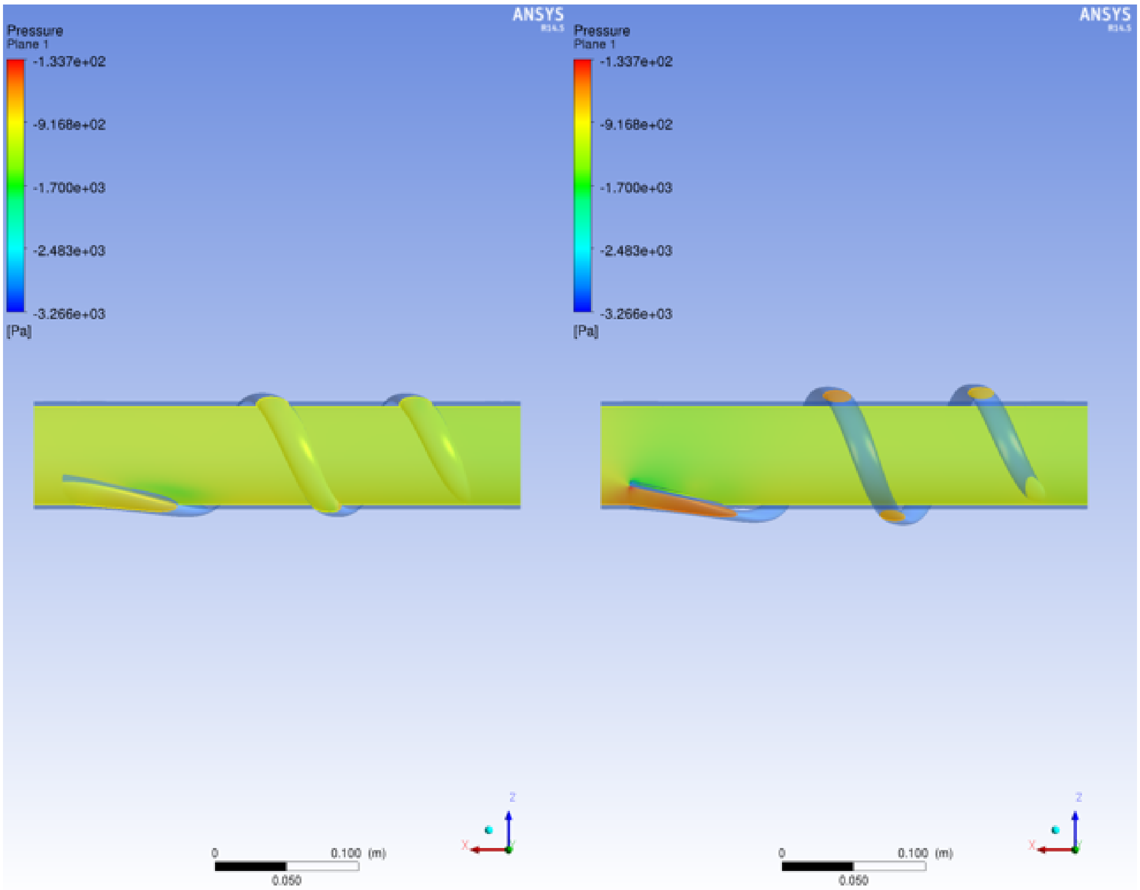


Figure A.7: CFD pressure results of swirl generator with flow separation.

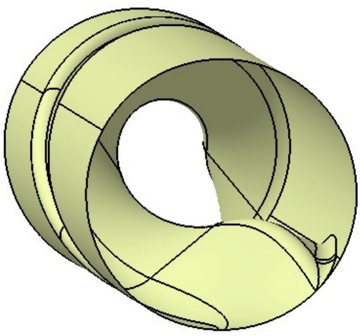


Figure A.8: Spiral shaped duct for swirl generation.

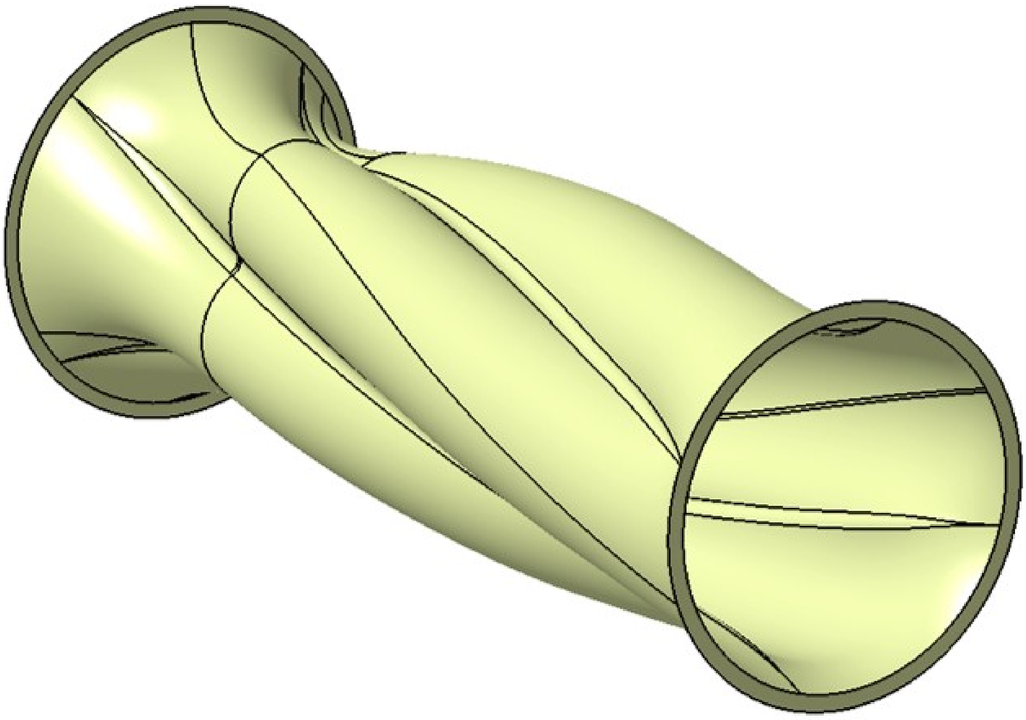


Figure A.9: Tube shaped duct for swirl generation.

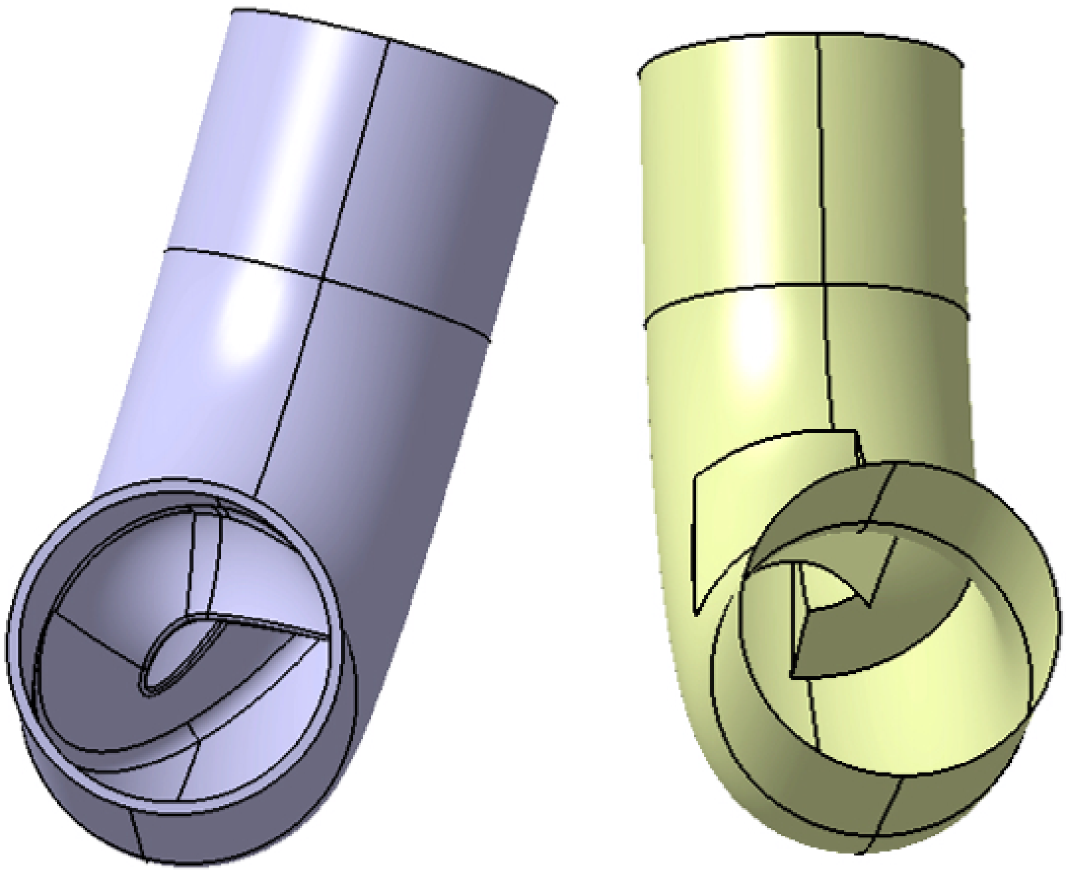


Figure A.10: Spiral shaped duct and guide vanes for swirl generation in the 90° bend.

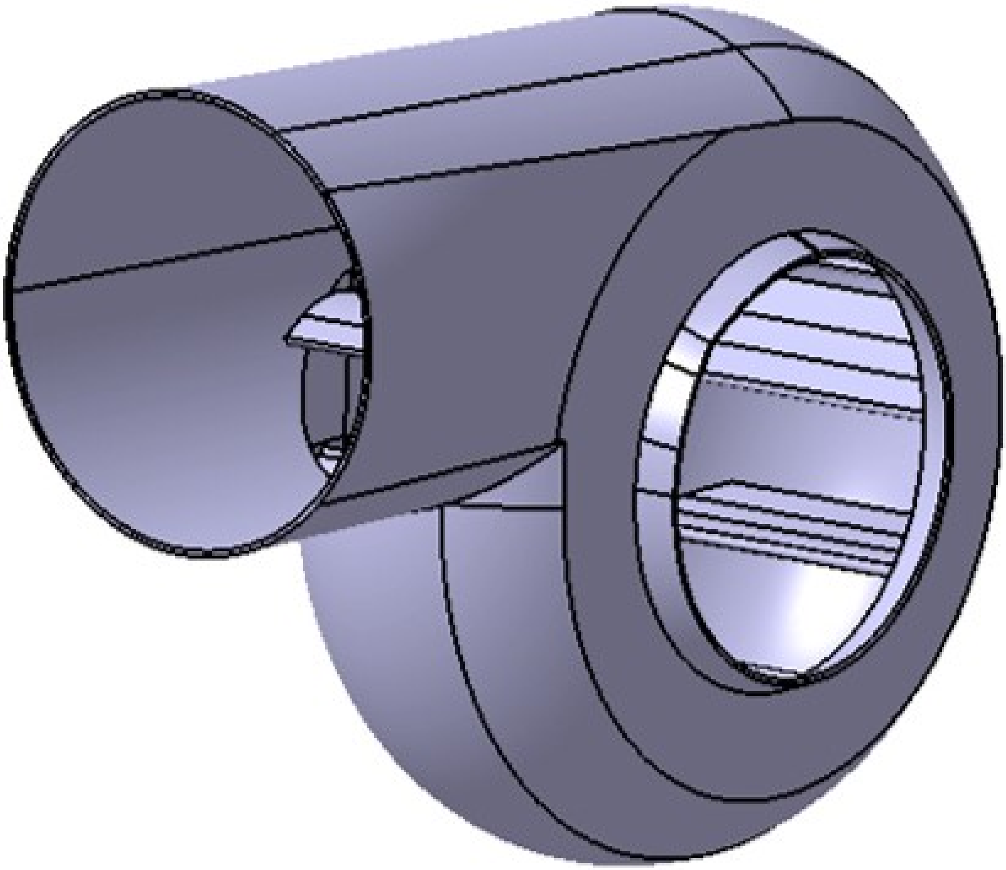


Figure A.11: Swirl Generation Device (SGD).

B

Air injection

Research done by [22] investigated air induced swirl possibility. This method would have one big advantage - no pressure drop. However this kind of swirl generation system is much more complicated and needs a source of high pressure air. An idea would be to use exhaust gasses as a part of Low Pressure Exhaust Gas Recirculation system and inject these gasses into the mainstream of air inlet. This would give double effect - reduce harmful exhaust emissions and increase compressor efficiency, moreover as compressor map is shifted to the left this area could be used for low mass flow when compressors are usually working close to surge conditions.

B.1 Initial tests

Initially air injection was designed and simulated together with other swirl creation solutions and was designed in 90° bend (Figure X)to test its possibility to create swirl. To have a reasonable injection pressure it was decided to use exhaust gasses exiting the cylinder to use in swirl creation. Chosen pressure was 1.7 bar and temperature 600° C. System created a swirl of 0.44 swirl ratio.

A second simulation was for an idling condition of the engine. During thse conditions compressor low RPM range is affected by surge conditions, which can be improved with the swirl. Air injection pressure this time was 1.14 bar and air mass flow in the intake was reduced from 720 kg/h to 150 kg/h. Results showed significant increase in swirl (up to 1.23). So it seemed that injection pressure is to big and had to be reduced. Assuming that exhaust gas flow will be throttled air injection pressure was reduced to 3760 Pa. Swirl ratio was reduced to 0.62. During initial tests it was clear that this system was different aspects like:

- No pressure drop
- Creates high amounts of swirl

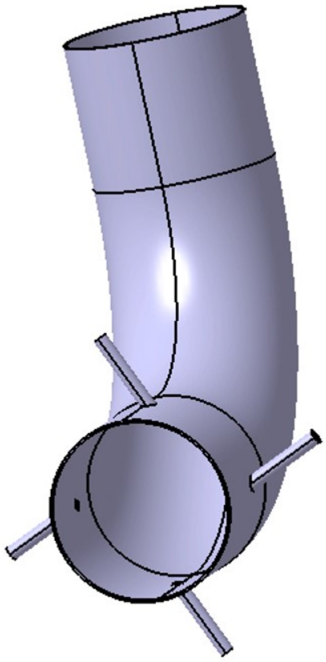


Figure B.1: Air injection for swirl creation.

- Can be easily controlled

and needs to be investigated differently.

B.2 Injection parameters

Initial investigation of air injection showed quite good potential of this system and possibility to use it together with exhaust gas recirculation systems. It was also clear that this system would need a lot of development from exhaust, intake systems, controls and also overall engine optimization and calibration for the system work. This was out of the scope and hardly possible to finish within thesis period, therefore it was decided to focus investigation only on intake side and evaluate injection parameter influence on swirl creation. Initial system used in the previous CFD simulations was used as a baseline which parameters can be found in Table B.1 and four parameters were changed:

- number of injectors (3, 4, 5, 6)
- injector radius (4, 8, 12, 16)
- injection angle (15, 30, 45, 60)
- injection pressure (470, 940, 1880, 2632, 3760, 7520)

Table B.1: Initial air injector parameters

Number of injectors	4
Injector radius	4
Injection angle	30
Injection pressure	1880

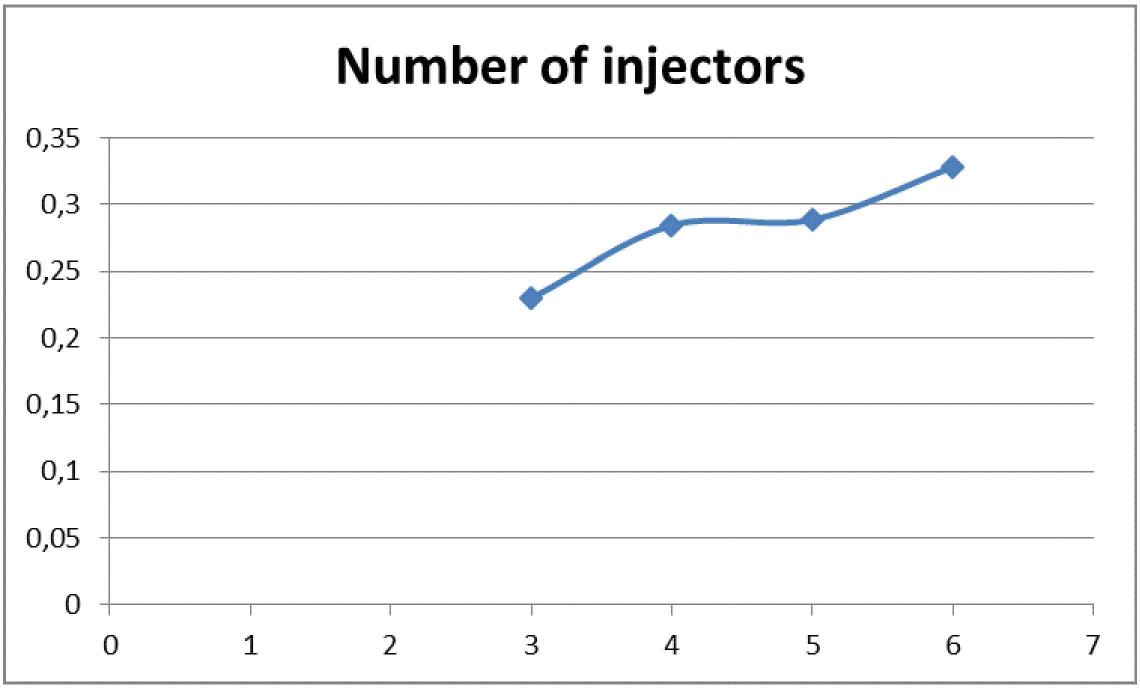


Figure B.2: Injector number influence on swirl.

Figure B.2 shows that increasing number of injectors is increasing the amount of created swirl. Figure B.3 shows how swirl is affected by the angle of injector and for this particular case angle of 45° gives best swirl ratio. Finally B.4 shows that with increase of pressure amount of created swirl is also increased. Different injector radius was tried for swirl induction. However increased injector radius with the same pressure was giving to much mass flow and that caused back flow in the intake system and significant reduction of mass flow going into compressor, therefore results could not be finalized.

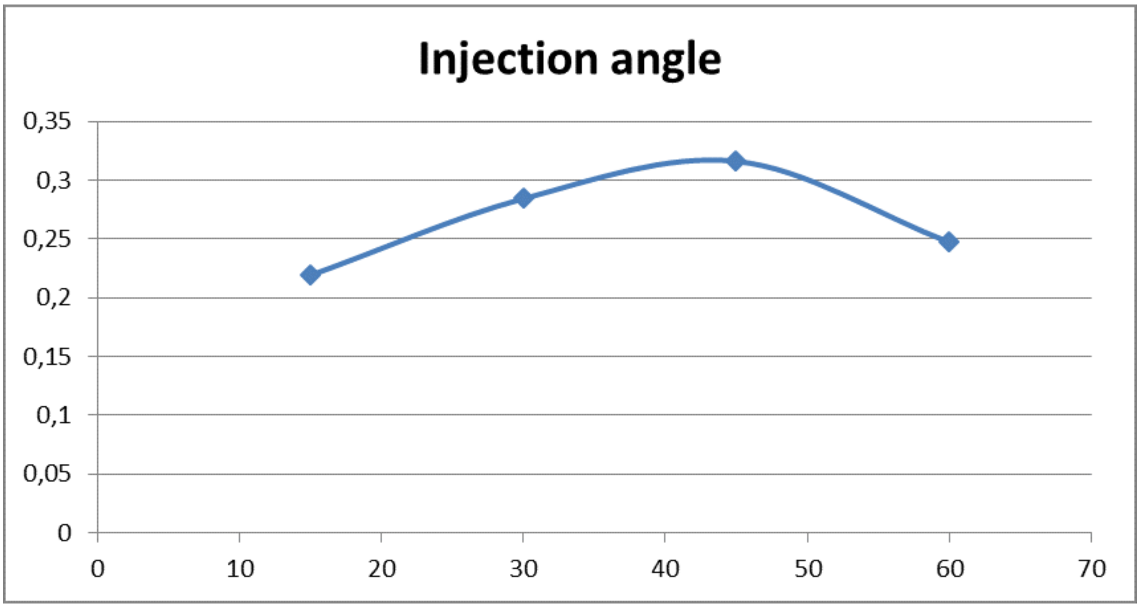


Figure B.3: Injector angle influence on swirl.

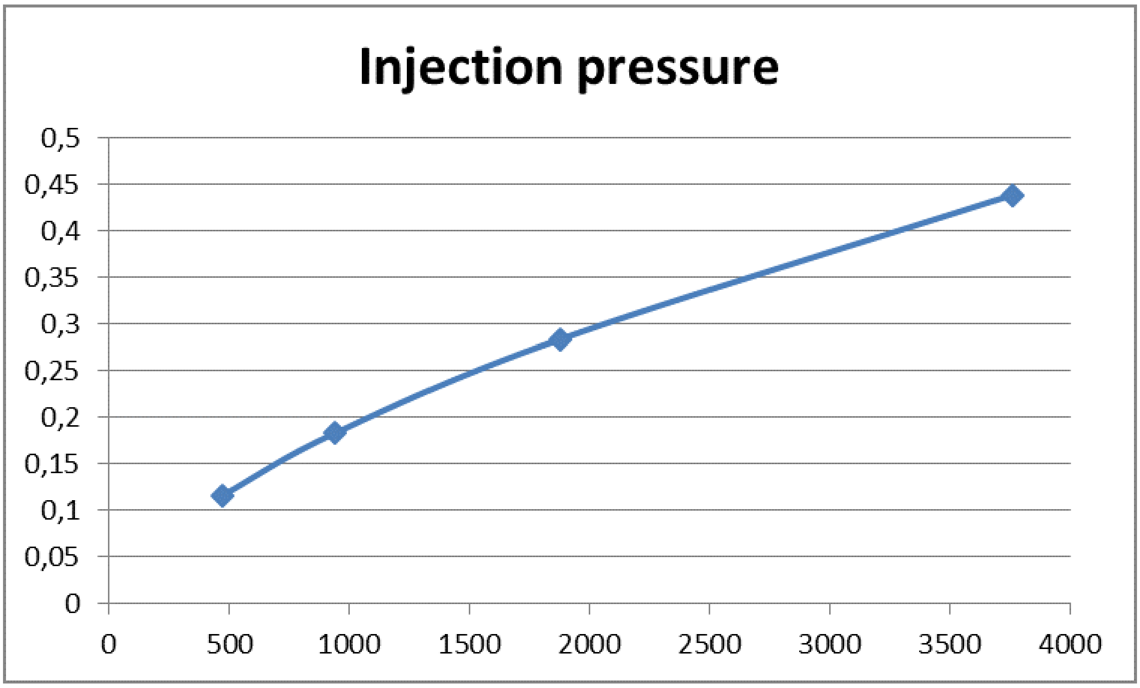


Figure B.4: Injector pressure influence on swirl.

C

NACA bend

DURING LITERATURE REVIEW some research was found on 90° bend improvements. 90° bend is one of the most common bends in the air intake systems that causes most of the pressure drop and usually designer spends a lot of time while trying to optimize it. Therefore a small investigation on possible solutions was also done in this master thesis project. [10] optimized 90° bend for NACA aplication which later was also used in automotive applications, [11] registered a patent on improved 90° bend which reduces separation on the inner radius and total pressure drop in the bend. Finally [21] also mentioned the need to improve the flow through 90° bend and designed a different swirl device which uses 90° bend for swirl creation. [10] and [11] had a lot of similarities between their designs. They concluded that separation in the 90° bend is caused by pressure difference between outer and inner radius. To reduce this pressure difference a distance between outer and inner radius should be reduced. At the same time they tried to keep the same flow area of the ducts. A results was that perfectly round ducts was squeezed to oval form. An example of this solution can be seen in Figure C.1. To investigate the amount of shape change needed for pressure drop improvement a range of similar ducts were designed. Initial radius of the ducts was 37.5 mm and flow area of this duct was kept for all the solutions. A range from 33 mm to 70 mm was simulated to see what pressure drop it gives. Results can be seen in the Figures X and Y. Figure X shows how separation is reduced with changed shape. From figure Y can be seen that if duct radius is changed from 33 mm to 45 mm a pressure drop reduction is around 20 Pa, which in this case is almost 10%. Further change of the ducts starts to increase pressure drop and even go beyond. This happends because changed ducts have bigger surface area and that increases friction losses. From this investigation it can be concluded that it is most efficient to ovalize duct by changing its initial radius with a coefficient of 1.2.

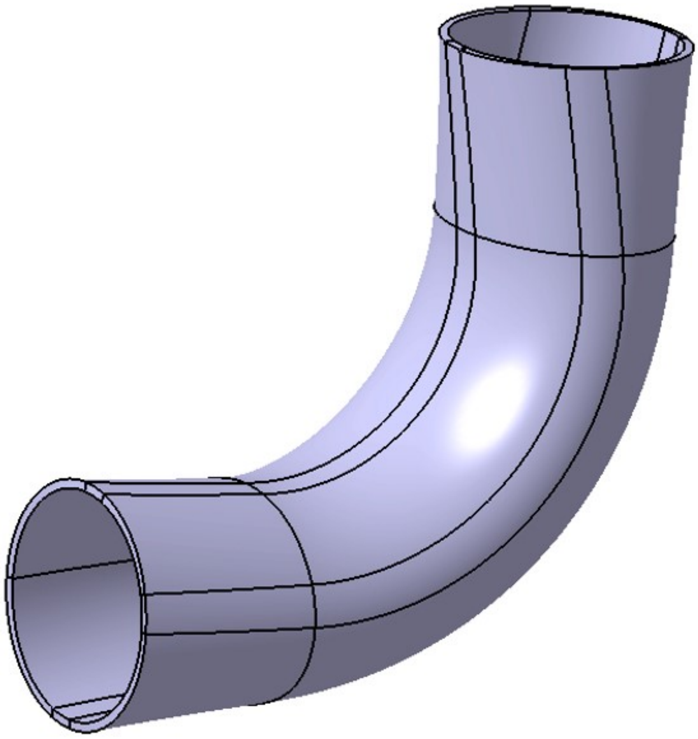


Figure C.1: NACA type 90° bend.

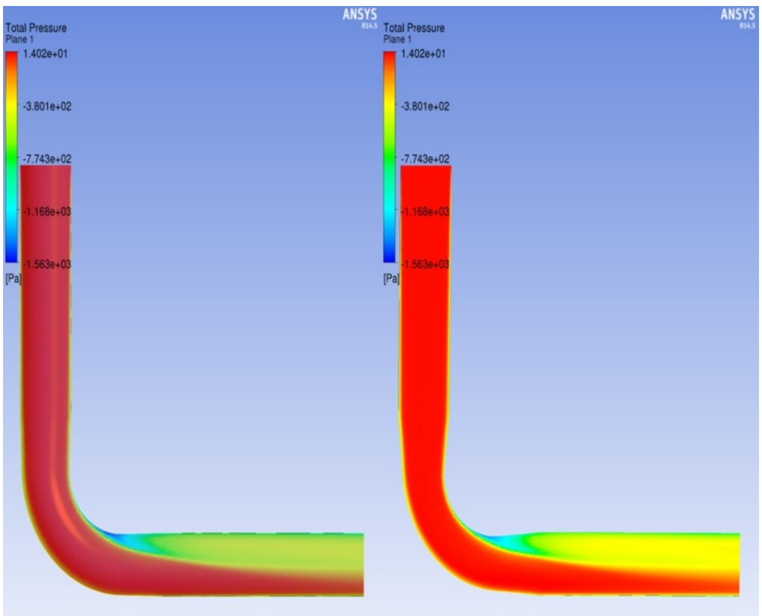


Figure C.2: Flow separation of different NACA type 90° bends.

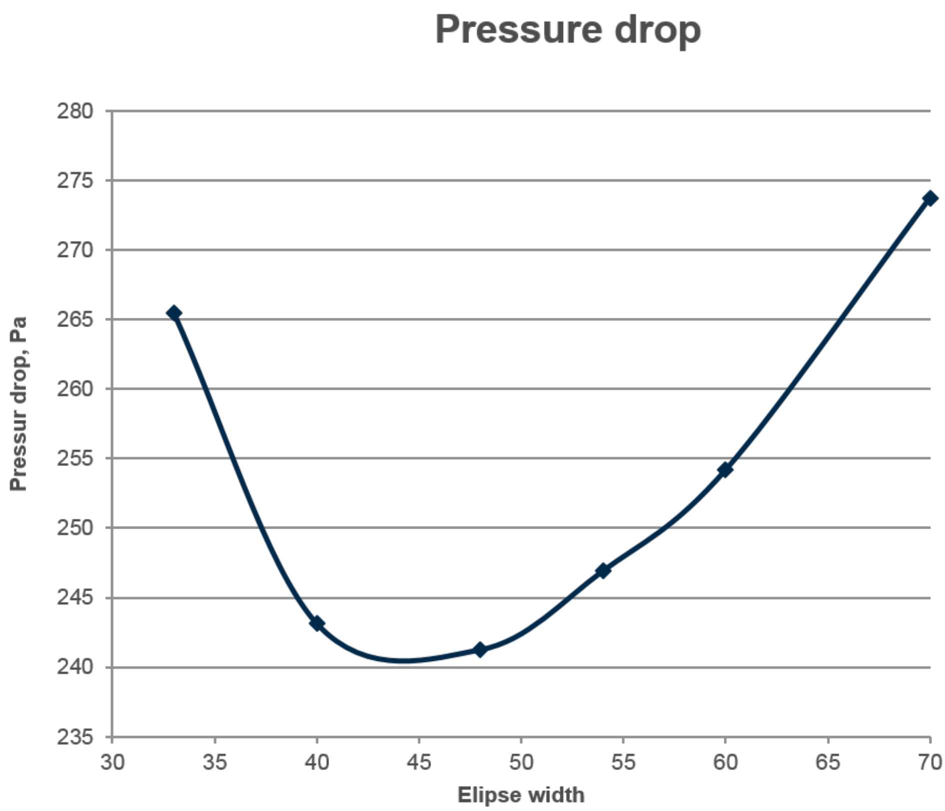


Figure C.3: Pressure drop of different NACA type 90° bends.