Intake Manifold Optimization using CFD Analysis

Dr. H. Bensler, Volkswagen AG,
Powertrain Calculations Dept.
Wolfsburg, Germany

Abstract

A properly designed intake manifold is essential for the optimal functioning of an internal combustion engine. Traditional intake manifold optimization has been based on the direct testing of prototypes until satisfactory engine performance was obtained. This trial and error optimization method can be effective, but is expensive, time comsuming, and is not always successful. Moreover, this empirical method does not provide any of the vital information concerning the actual flow structure inside the intake manifold. Without this information, the design engineer can never really understand why a particular intake manifold performs correctly or not. The only feasible way to obtain this information within a reasonable amount of time and cost is to perform a 3-D computational fluid dynamics (CFD) analysis of the intake manifold. At VOLKSWAGEN AG (VW), this process has been greatly accelerated and the cost involved substantially minimized through the use of automatic mesh generation (AMG) for the creation of the 3-D CFD mesh. This paper will present a brief review of the functioning of an intake manifold along with a description of some basic design rules. Details concerning stationary and instationary 3D CFD analysis of intake manifolds based on AMG will be elaborated upon. Examples will be presented showing how the optimization of intake manifolds using CFD analysis based on AMG is giving VW engineers the competitive edge with regards intake manifold design providing answers to questions faster and cheaper than ever before.

1. Introduction

1.1 Intake manifold operation

An intake manifold (IM) is one of the primary components regarding the performance of an internal combustion engine. It consists typically of a plenum, a throttle body connected to the plenum, and depending on the number of cylinders individual pipes/runners which lead to the engine cylinders (Fig. 1). The task of an IM is to control how the air (and EGR) flow is distributed to the cylinders. The air flow to each cylinder is not necessarily identical even under steady engine operating conditions. This occurs due to the difference in the IM runner length and other geometric details along the air flow path to the cylinders. How the runners are connected to the plenum and the cylinder firing order also influence the air distribution of the IM. The air flow into each cylinder depends on the pressure in the plenum; moreover, pressure variations in the IM resulting from air flow momentum effects can substantially affect engine performance. [1]

During the operation of an engine, pressure waves propagate back and forth throughout the the IM to the individual cylinders. Pressure oscillations originating in one cylinder during the intake process produce pressure oscillations in other cylinders. The magnitude and position of the pressure waves is dependent of the runner length of the IM and the engine speed. These effects are relatively large and effect the change in the charge of the individual cylinders. As the engine speed increases, the amplitude of the pressure waves increases but the frequency decreases. These pressure waves affect positively or negatively the air distribution to the cylinders depending on their phase and amplitude. An uneven air distribution leads to non-uniform cylinder volumetric efficiency, overall power loss and increased fuel comsumption.

As shown by the study of Rozsas & Brandstetter [2], the volumetric efficiency for each of the individual cylinders of a 1.3 liter 4 cylinder gasoline engine was greatly influenced by the layout and volume of the IM. If one has a small plenum and an air supply from the side, the volumetric efficiency as a function of the engine speed varies greatly for each of the cylinders. By changing the air inlet to be symetrical, the volumetric efficiency for each cylinder becomes more favorable even with the same plenum volume. And by increasing the plenum volume 6 times, the symetrical inlet IM had almost equally distributed volumetric efficiencies for each cylinder at a given engine speed. This study thus showed that cylinders have equal influence on one another if the IM inlet is symetric, the IM pipes are of the same length, and if the runner orifices are the same distance from one another and from the throttle valve. This gives a uniform air distribution to the cylinders and the intake pulse / flow has the same intensity in all the cylinders.

1.2 Intake manifold design criteria

If one summarizes the above, one can derive certain IM design criteria which allow low air resistance and good air distribution to all of the cylinders:

1) A symetrically designed intake manifold allows uniform filling of the all the cylinders
2) Increasing the plenum volume helps to provide uniform cylinder filling
4) Cross-flows over the runner orifices in the IM plenum should be avoided
5) Unnecessary turbulence and eddies should be avoided in the inlet region of the IM

It should be noted that these ideal conditions cannot always be met due to the engine type and design and due to engine compartment space limitations.

1.3 Intake manifold testing

Traditionally, intake manifold design has been based on trial and error testing of prototypes. This empirical method can be effective, however, it is expensive, time consuming, and sometimes not always successful. Moreover, the method cannot provide any information regarding the flow structure inside the IM. Without this sort of information, the design engineer can never really understand why a particular IM works correctly or not.

Through the empirical method, the mass flow rate for each of the runners of the prototype IM is first optimized on a stationary-flow test rig. This ensures a uniform steady-state pressure drop over each of the IM runners, but it does not really indicate if the IM will perform well or not for an operating engine. The same principle applies to stationary intake port test rigs. These type of results can tell an engineer if the port has good steady-state filling capabilities or if it can provide a certain amount of steady-state swirl. But it doesn't really indicate what the in-cylinder flow structure will be in an operating engine. To make up for this lack of information, some sort of flow analysis should be performed.

2. Intake Manifold Optimization Process

2.1 Intake Manifold Design Process Chain

The process chain applicable to the design and optimization of an intake manifold can be seen in Fig. 2. As can be seen, the process starts with a one-dimensional (1-D) gas dynamics calculation of the entire gas track of the engine. One-dimensional gas dynamics codes are extremely useful in providing information with regards to the general gas flow and determining the gas track layout such as plenum volumes, runner lengths and diameters, valve diameters, etc... [3]. The advantage of a 1-D code is that many variations can be quickly calculated. Once a fine tuned model has been determined, the design engineer now has a geometric data base which provides the guidelines necessary for the construction of a CAD model. This CAD model should be created using a parametric modeller such that changes to the model can be quickly performed. It should be noted that 1-D codes cannot, however, provide detailed three-dimensional (3-D) information concerning what is occuring with the flow inside the IM. One can predict that there is a loss of power or torque at a given engine speed; but as to why this happens is sometimes out of the limits of the code due to its one-dimensional nature. This is where 3-D computational fluid dynamics (CFD) can provide useful information to answer these questions.

2.2 Stationary 3-D CFD Process Chain

The next step is to perform a 3-D CFD analysis of the flow in the intake manifold. There are two types of calculations to be performed:

1) stationary flow simulations where each runner of the IM is analyzed individually with a constant mass flow rate (one runner at a time)
2) instationary flow simulations at a specific engine speed where the entire IM is flowed through dynamically as is the case for an operating engine

The process chain for the stationary flow simulation can be seen in Fig. 3. To begin with, a completely closed 3-D surface model of the inside of the IM is obtained from the CAD model. Next one generates a numerical mesh "inside" this closed surface model. In the past, the generation of the numerical mesh using semi-automatic mesh generation (SAMG) tools was no trivial task. At VW the way this difficulty has been over come is through the use of a automatic mesh generation (AMG) tool which will be discussed in section 2.4. After the mesh has been generated, one performs the flow simulation ideally for each of the runners simultaniously on a multiprocessor workstation or supercomputer. The boundary conditions utilized for the inlet and outlets are either a constant inlet and outlet mass flow rate or a uniform pressure drop across each of the individual runners. After the flow simulations have been calculated a CFD postprocessor is utilized to visualize the flow structure in the IM and to determine the loss coefficients for the individual runners. One now has the results equivalent to what one would get from a steady-flow test rig, with the exception that one can now "see" what the flow structure is like inside the manifold. The advantage of a stationary simulation is that it is fast and provides the loss coefficient results that design engineers are accustomed to receive from experimental test rigs. Experience has shown however, that the steady-state flow structure inside an IM resembles that of an operating engine only in the inlet region of the plenum and perhaps for the runner closest to the inlet throttle body. Thus an instationary flow simulation is necessary to understand the flow structure inside the IM.

2.3 Instationary 3-D CFD Process Chain

In order to better understand an instationary IM flow simulation, its corresponding process chain can be seen in Fig. 4. To begin with, the same 3-D inner surface model of the IM is required. If one has already conducted a stationary simulation of the IM, the same numerical mesh can be used for the instationary analysis. The primary difference is that the boundary conditons are no longer constant but are time dependent. These boundary conditions are obtained from the 1-D gas dynamics analysis performed before the creation of the CAD model. An example of these time-dependent boundary conditions for a 4 cylinder gasoline engine are shown in Fig. 5. Using the 3-D CFD mesh and the instationary boundary conditions, one then uses a CFD solver to calculate the instationary flow. Since one begins the calculation with initial conditions for the pressure, temperature and turbulence in then IM based on assumptions, at least 2 complete 720 degree cycles need to be calculated to insure that all the IM runners are flowed through completely. Thereafter, videos of the flow strucutre inside the IM are created to give a "4-D" simulation of the flow. These videos help to visualize and better understand the instationary IM flow structure.

2.4 Automatic mesh generation of the 3-D CFD Mesh

Mesh generation is no longer an expensive and time consuming process. Due to the automatic mesh generation of the 3-D CFD mesh provided by the CFD code VECTIS [3], meshes of highly complex geometries are generated in a matter of hours on a workstation. The successful use of the automatic mesh generator depends on the existence of a completely closed and trimmed surface model of the IM. Given such a surface model, the usage of VECTIS is fast and simple using the following procedure:

1) read in the surface model
2) graphically define the inlets, outlets, and walls on the surface model
3) define a global mesh around surface model
4) run the automatic mesh generator
5) use the CFD solver to obtain the flow field results
6) visualize the results with the postprocessor

A more detailed description of how the VECTIS automatic mesh generator works can be found in [3].

To illustrate the advantage achieved through the use of automatic mesh generation, consider the example presented in Table 1. Here an estimate of the time and cost for a stationary IM calculation for one runner is compared for the case of semi-automatic mesh generation

(SAMG) and automatic mesh generation (AMG). Both cases start from the assumption that the CFD analyst has normal manual meshing skills and that a closed and trimmed surface of the IM exists. Depending on the complexity of the IM, SAMG should take from 3 to 10 days or more (for really complex IM geometries like those found in VW VR6 engine for example) to obtain a mesh; this requires from 24 to 80 man hours (Mhr) of tedious work (assuming an 8 hr work day). When considering AMG with VECTIS, there is only about one hour of engineer preparation time required for the mesh generator setup and about 5 hours workstation CPU time for the AMG. Both cases take 4 days CPU time to calculate on a single processor workstation and both require 1 day for the postprocessing. For the SAMG case, the total elapsed time is 8 to15 days; when using AMG, one is 1.5 to 3 times faster taking only about 5 days. In addition, the use of AMG is not only considerably faster and less expensive, but it makes the calculations engineer more efficient allowing more powertrain components to be analyzed at one time in parallel. Now the bottle necks are CPU availability and the time required for postprocessing in comparison to previous time investment required with SAMG.

3. 3-D CFD Analysis

3.1 Stationary Analysis

To illustrate the advantages of conducting a 3-D CFD analysis of an IM based on AMG, consider a recent analysis of an intake manifold for a 4 cylinder gasoline engine which was analyzed using the VECTIS automatic mesh generator. Upon receiving the closed and trimmed 3-D surface model of the IM, a 150,000 cell mesh was generated in 4.5 hours on a 150 MHz single processor Silicon Graphics workstation. In order to obtain quick results, a stationary flow simulation of runner 4 (nearest to the inlet) was calculated taking 4 days on a single processor workstation. A constant normalized mass flow rate of 0.80 (i.e. 80 % of the maximum instationary IM mass flow rate) was used as the inlet and runner 4 outlet boundary condition. Using the postprocessor, a cut was layed through the middle of the IM plenum showing the flow structure across the plenum and inlet (Fig. 6). The inlet is on a 90 degree angle to the plenum which results in a downward directed flow into the plenum with a magnitude of around 30 m/s (S1 in Fig. 6). A recirculation all along the wall under the inlet throttle body can be clearly seen (S2). This recirculation definitely causes a reduction of the effective cross-sectional area of the inlet thus leading to less filling potential for the cylinders. Observation of the flow in the plenum away from the inlet reveals three recirculation zones: 1) back left in the plenum (S3), 2) back middle of the plenum (S4), and 3) front right of the inlet flow (S5). One final recirculaton is found in the lower right corner of the plenum as well (S6); moreover, a large flow stream just above S6 is also present. Finally the last feature of the stationary flow is that a part of the inlet flow travels down the back of the plenum (S7).

3.2 Instationary Analysis

To see what the real flow structure in the IM was, an instationary 3-D CFD analysis was performed. The 1-D boundary conditions found in Fig. 5 at an engine speed of 3500 RPM were used for the simulation; moreover, the same mesh was used as was for the stationary calculation. As was previously stated, two 720 degree engine cycles are needed to perform an instationary simulation. A total of 8 days on a single processor R8000 Silicon Graphics workstation was required to calculate both cycles. Videos were then made of the flow for various section-cuts in the IM. In order to compare the stationary and instationary results, the boundary conditions should be as similar as possible. At 35 degrees crank angle after spark ignition (0 degrees), the normalized outlet mass flow rate of runner 4 also equalled 0.80. Fig. 7 thus shows the flow structure inside the plenum at the same section-cut as the stationary flow shown in Fig. 6. As can be seen, there are some similarities in the flow structures. As expected, a similar air velocity of 30 m/s flows through the inlet downward into the plenum (I1) (note that flow phenomenon I1 corresponds to S1 in section 3.1, I2 to S2, etc...). The recirculation around the wall under the throttle body is also present (I2). However, the flow in the plenum to the left of the inlet is considerably different. The recirculations I3 and I4 have basically disappeared and have moved way to left of the plenum, recirculation I5 has moved up and to the left with a much lesser swirl intensity and recirculation I6 and the strong flow stream above it has disappeared completely due to a large recirculation which now flows against the inlet flow. In addition, the flow down the back of the plenum I7 is also no longer present. Finally a new recirculation zone I8 has appeared just to the left of the inlet causing interference with the inlet flow.

From these calculations results, changes were incorporated into the 3-D model to improve the flow under the throttle body. These changes removed recirculations S2 and I2 in the inlet region of the plenum and increased the effective inlet cross-section allowing better filling of the cylinders to occur. Figures 6 and 7 reinforce the previous statement that stationary calulation results are only really applicable in the region of the inlet.

4. Conclusion

This paper has presented a brief review of how an intake manifold functions and some corresponding design criteria which can improve engine performance. Numerical results pertaining to stationary and instationary intake manifold flow simulations have been discussed showing their similarities and differences. The speed, effectiveness, and necessity of conducting 3-D CFD analysis of intake manifolds based on automatic mesh generation has been presented. Moreover, it has been shown that through the use of automatic mesh generation, the time and cost involved in performing such an analysis have been greatly reduced compared to semi-automatic meshing methods. In conclusion, the optimization of intake manifolds using 3-D CFD analysis has finally become fast and cost effective and is currently giving VW engineers the competitive edge with regards to intake manifold design and development.

Acknowledgements

The author would like to thank Dr. B. Beesten for providing the stationary flow results found in Fig. 6.

REFERENCES

[1] Heywood, J. B., Internal Combustion Engine Fundamentals, McGraw-Hill, New York, 1988.

[2] Rozsas, T. & Brandstetter, W., "Optimization of the charge process in modern motor vehicle spark ignition engines", SAE 885058.

[3] Bensler, H. P. & Oppermann, R., "CFD optimization of powertrain components", Proceedings of the IMechE Conference on Computers in Reciprocating engines and gas turbines, C499/057, London, England, Jan 9-10,1996.

	SAMG	AMG
Mesh generation time	3 to 10 days (24 to 80 man hrs)	0,20 days (WS CPU time) (1 man hrs)
Calculation time	4 days (WS CPU time)	4 days (WS CPU time)
Postprocessing time	1 day (8 man hrs)	1 day (8 man hrs)
Total:		
Total elapsed time	8 to 15 days	5,25 days
Total man hours	32 to 88 hrs	9 hrs

Table 1. Comparison of time and cost of a stationary intake manifold calculation for one runner using semi-automatic mesh generation (SAMG) and automatic mesh generation (AMG) (WS = workstation).

Intake Manifold Surface Model

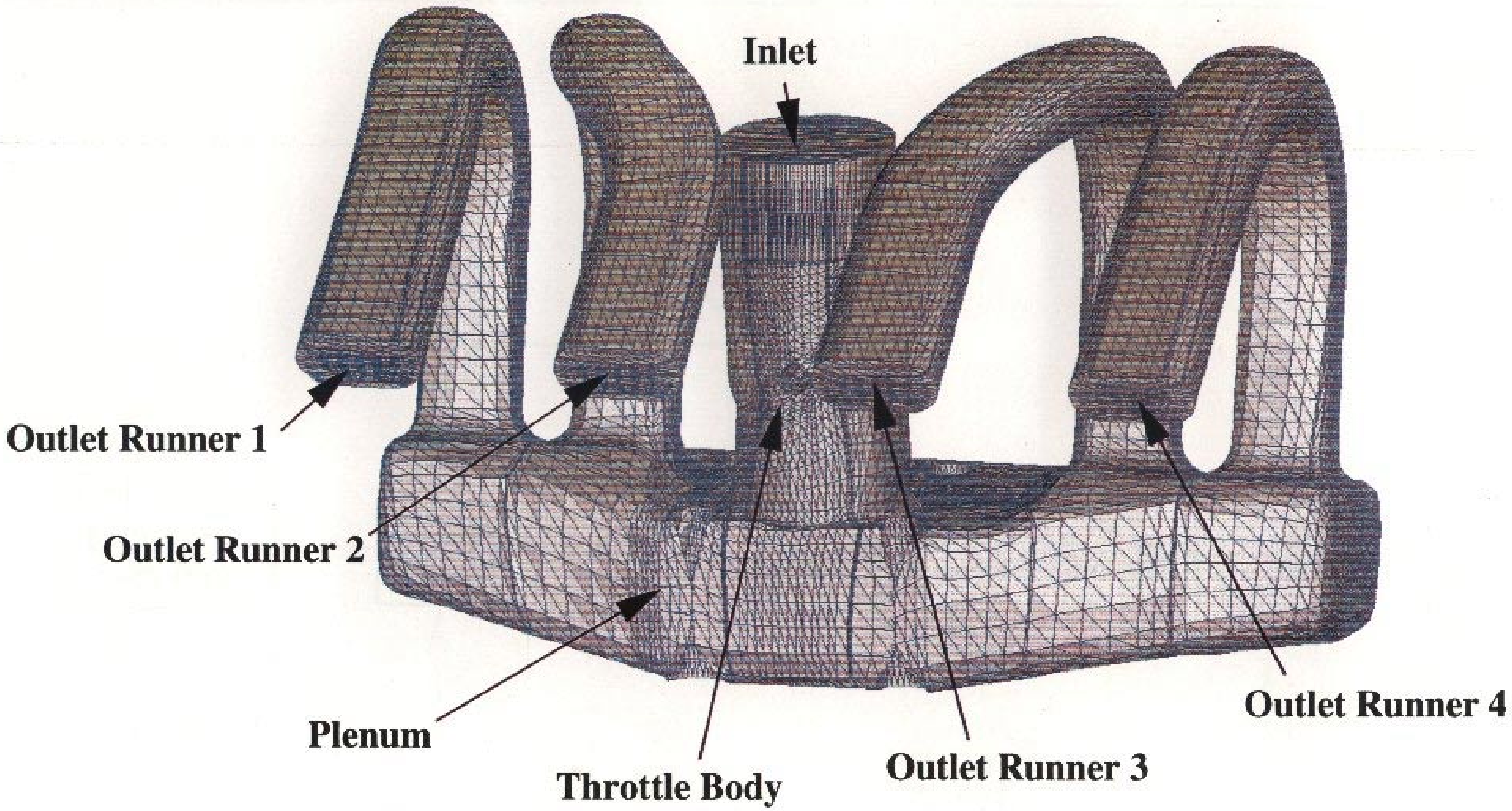


Fig. 1 Perspective view of an intake manifold surface model for a 4 cylinder gasoline engine.

VOLKSWAGEN AG
Research
and
Development

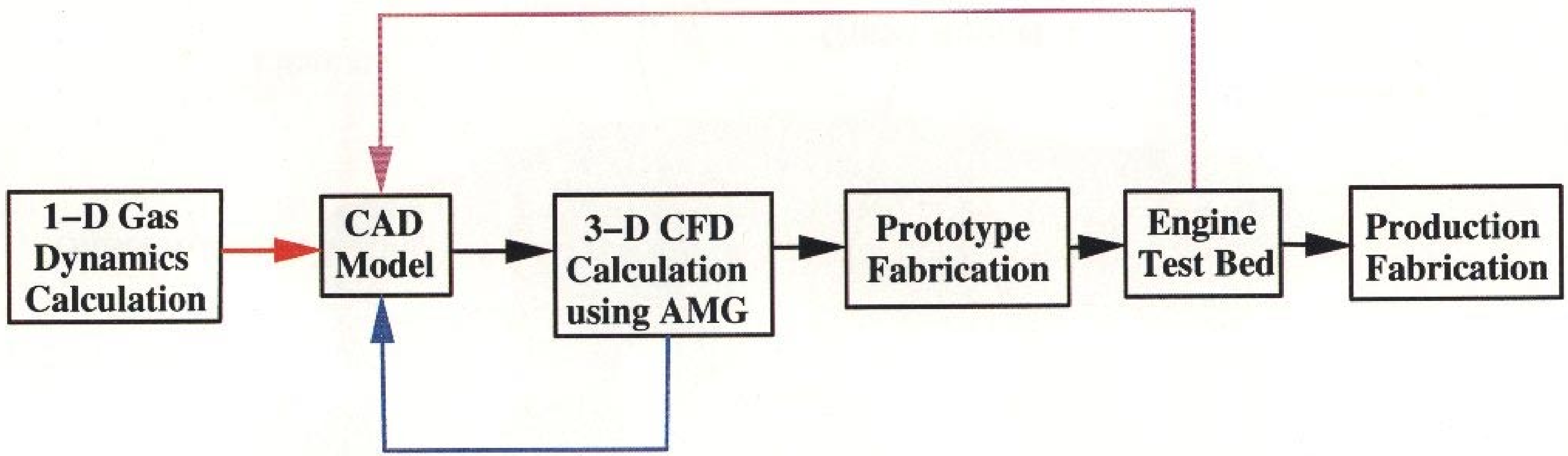


Fig. 2 Intake manifold design process chain
(AMG = automatic mesh generation).

VOLKSWAGEN AG
Research
and
Development

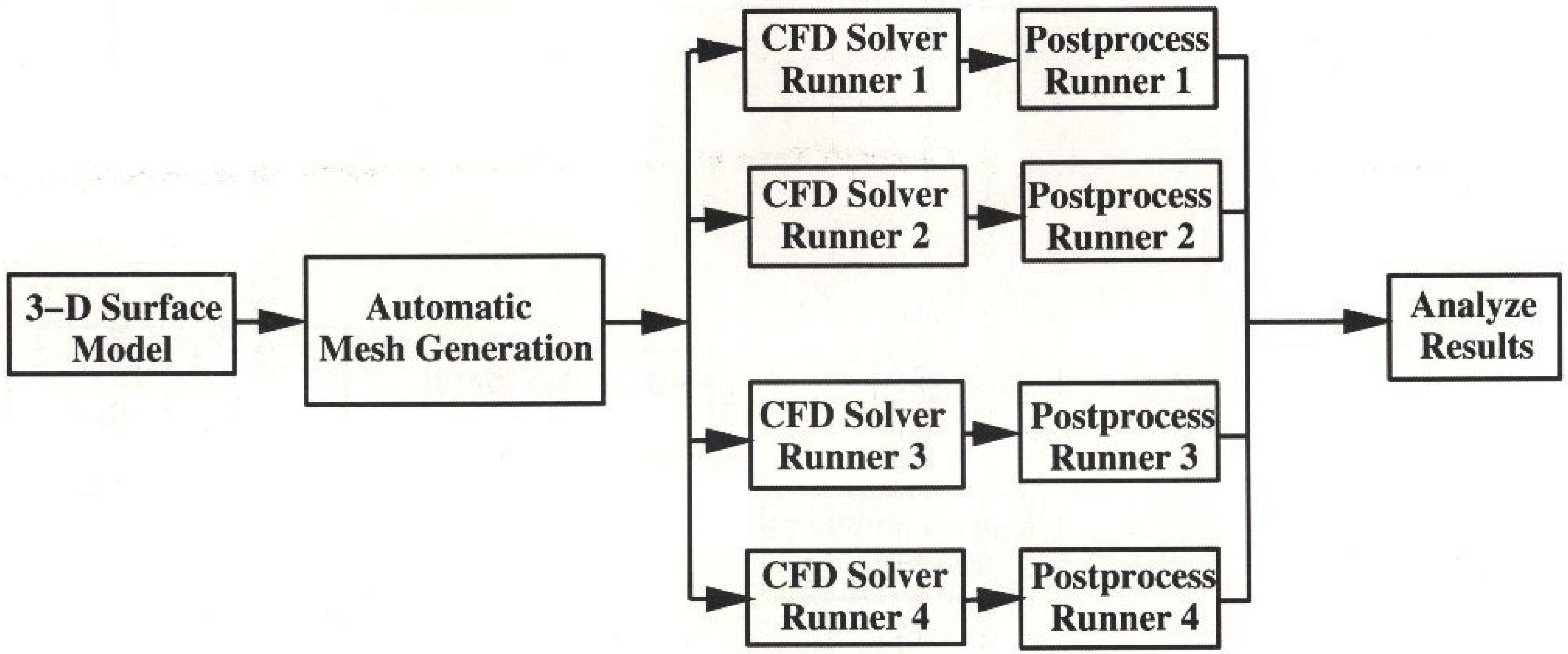


Fig. 3 3–D CFD optimization process chain: stationary flow.

VOLKSWAGEN AG
Research
and
Development

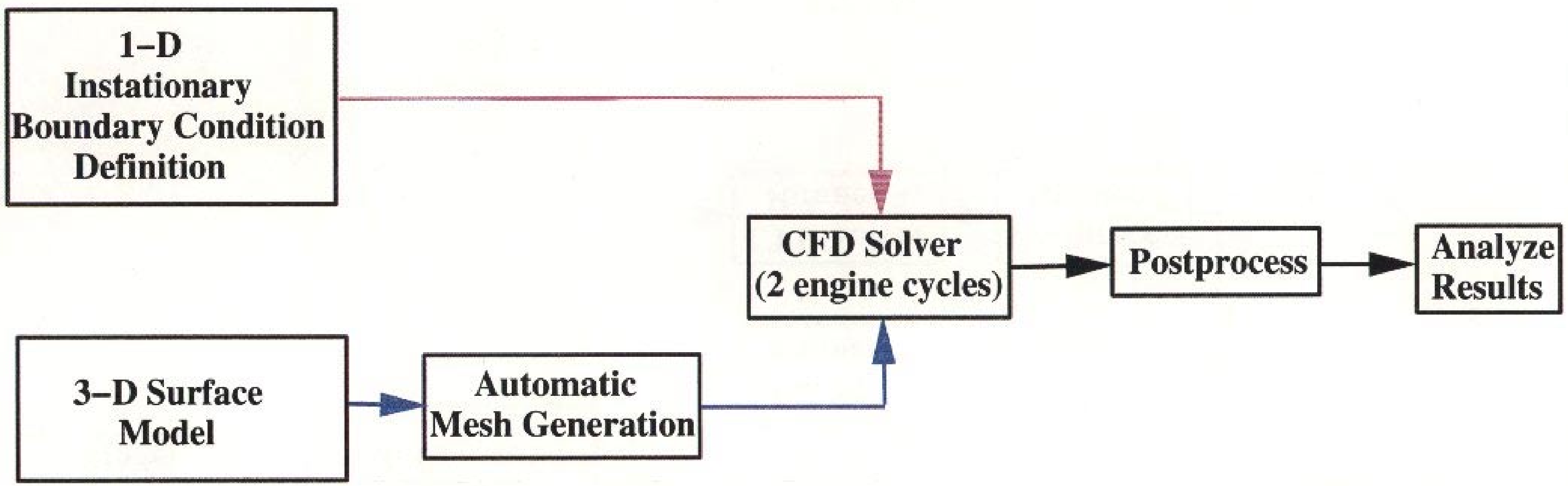


Fig. 4 3–D CFD optimization process chain:
instationary flow.

VOLKSWAGEN AG
Research
and
Development

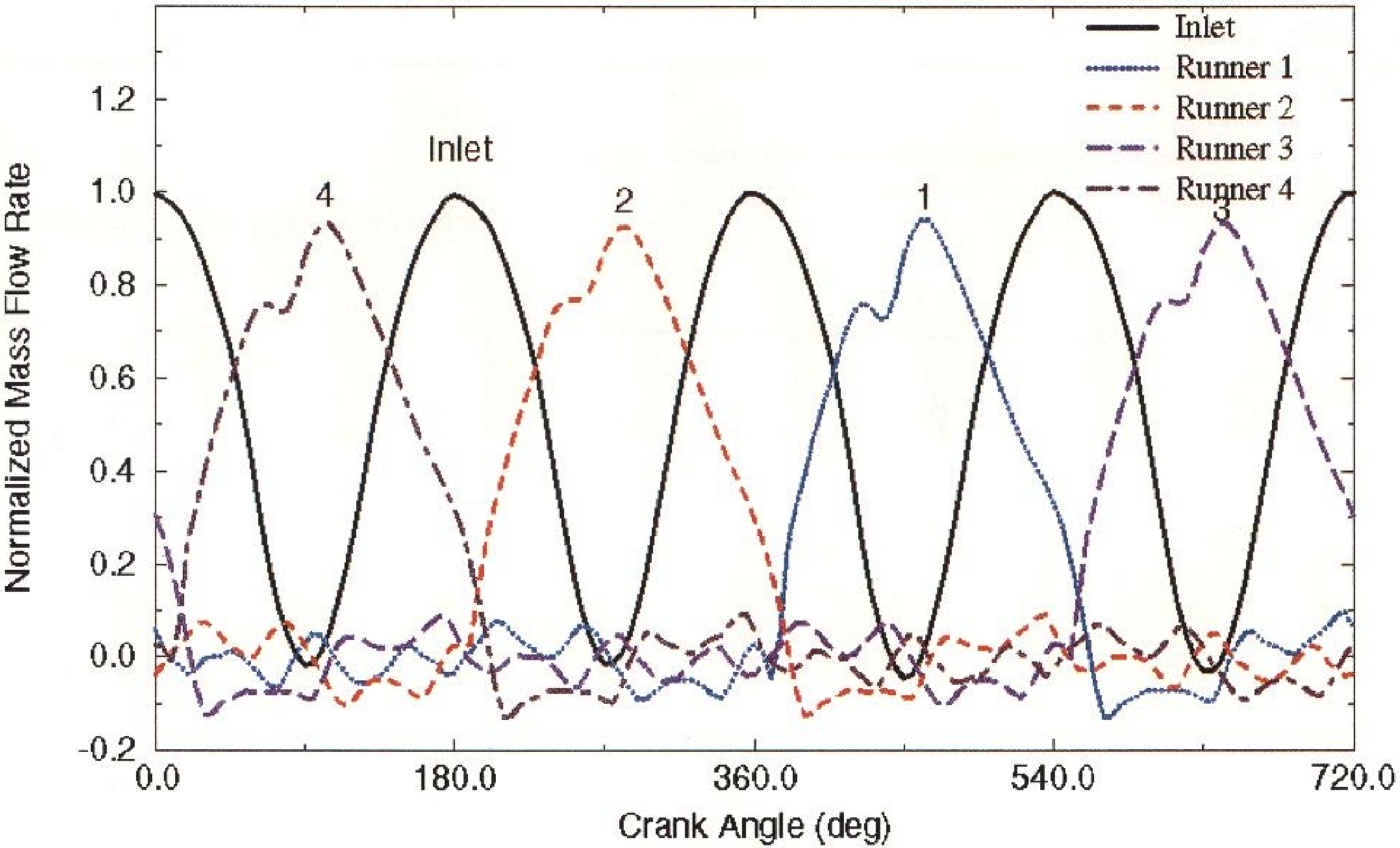
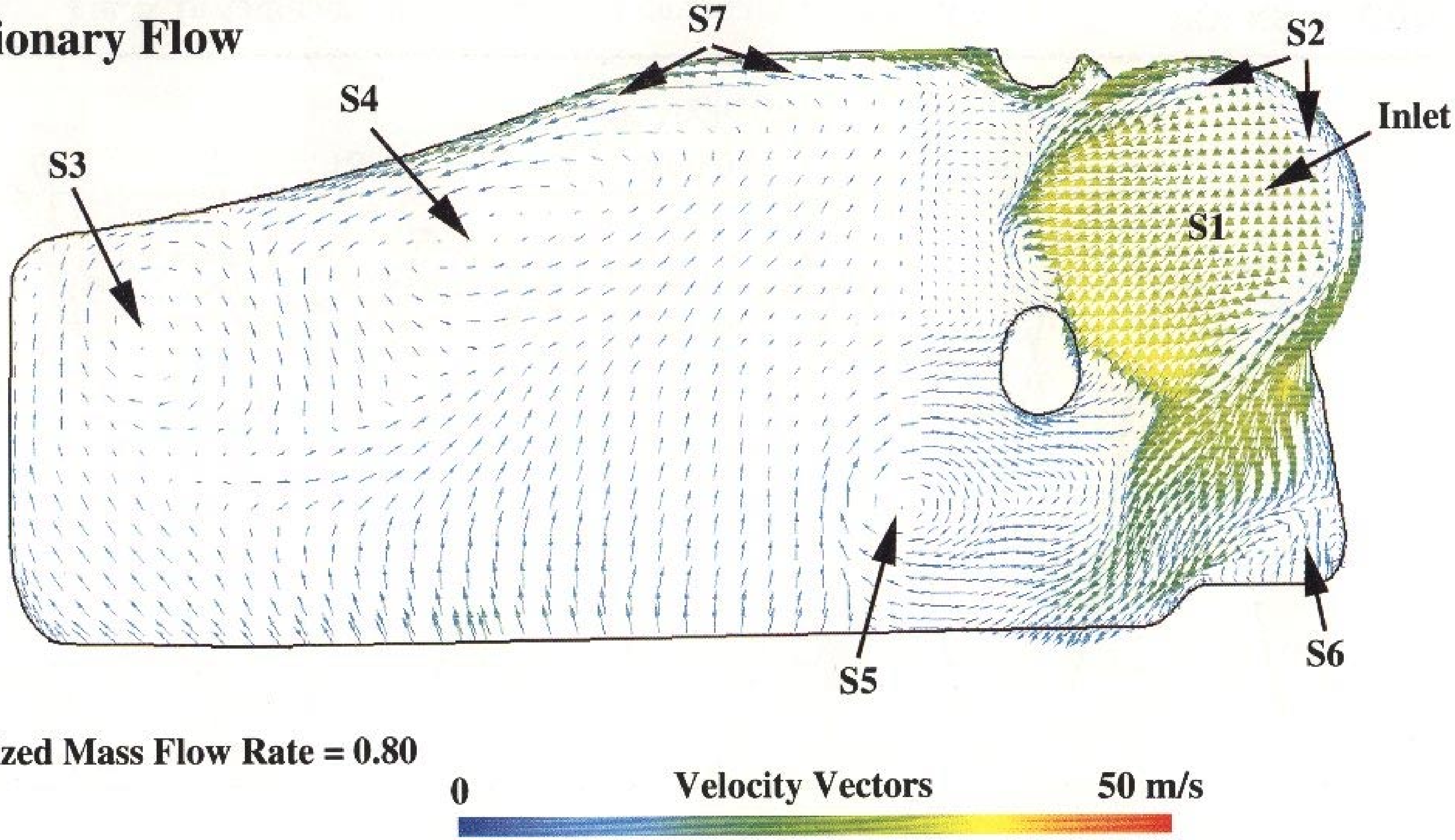


Fig. 5 Instationary intake manifold boundary conditions for a 4 cylinder gasoline engine resulting from a 1–D gas dynamics code, engine speed = 3500 RPM.

VOLKSWAGEN AG
Research and Development

Intake Manifold Plenum

Stationary Flow

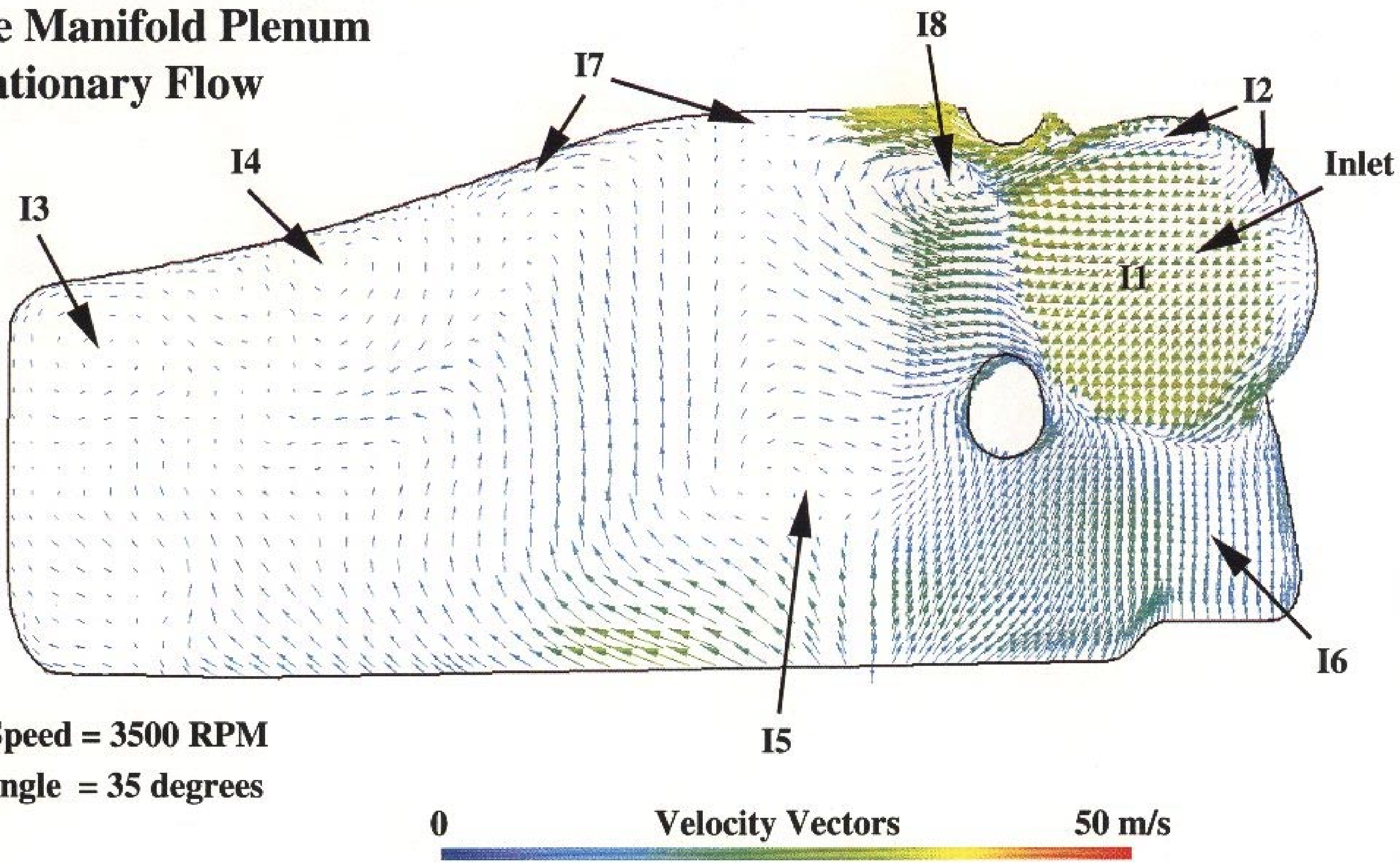


Normalized Mass Flow Rate = 0.80

Fig. 6 Stationary flow velocity vectors in the plenum of an intake manifold for a 4 cylinder gasoline engine.

VOLKSWAGEN AG
Research
and
Development

Intake Manifold Plenum
Instationary Flow



Engine Speed = 3500 RPM

Crank Angle = 35 degrees

Fig. 7 Instationary flow velocity vectors in the plenum of an intake manifold for a 4 cylinder gasoline engine, (engine speed = 3500 RPM).

VOLKSWAGEN AG
Research
and
Development