

CFD Topology and Shape Optimization of Ford Applications using Tosca Fluid

Dr. Anselm Hopf

Ford Motor Company, Aachen, Germany

Abstract: *Modern engine architectures need compact designs driven by vehicle package constraints. Furthermore, an engine should fit into several vehicle applications and parts have to be modified to match all package configurations. The optimization of the individual parts and the system by Computational Fluid Dynamics (CFD) may help to minimize or even delete these drawbacks. Meanwhile, there are several powerful optimization methods for three-dimensional flows on the market. While comparing their strengths and weaknesses, efficiency and easy handling are also important for engineers in research and product development. In this paper, a combined optimization strategy using CFD topology optimization followed by a shape optimization is presented using the software tools Tosca Fluid and STAR-CCM+. At the end, an initial design of the flow-optimized parts has been generated (bionic design). So, the designer starts with an optimized design solution. At Ford, several applications have been optimized with Tosca Fluid and some of them are presented here in detail.*

Keywords: *CFD, Topology Optimization, Tosca Fluid, Numerical Simulation, Flow Uniformity, Pressure Drop Reduction, Flow Balancing, Shape Optimization, STAR-CCM+, Adjoint Solver, Water-Pump Inlet, Ports, Integrated Exhaust Manifold, Bionic Port Design, and Flow Splitter.*

1. Introduction – Optimization of 3D Flows (CFD)

In the last few years, several flow problems have been optimized at Ford using modern optimization techniques for three-dimensional flows. Before presenting some of them, an overview of the most used CFD optimization methods is given.

Meanwhile, there are several powerful optimization tools for 3D flows on the market. The strengths and weaknesses of each tool have been compared; however, efficiency and easy handling are also important for engineers in research and product development.

Figure 1 gives a rough overview of optimization methods and tools for 3D flow phenomena.

As first approach, it is a common practice to optimize flows by hand, based on engineering knowledge and experience. Due to the empirical approach, this method may be called a 0-dimensional optimization problem.

A more systematic approach is the parameter optimization, where the engineer has to define parameters. These parameters are often sizing parameters like diameters, radii, cone angles, etc. The parameters are changed during the optimization. Many CFD runs are calculated and are often solved using Design of Experiments (DoE) "(Siebertz, 2017)." This approach is very powerful

regarding e.g. multi-objective runs (Pareto front) and transient CFD analyses. A drawback is the need to define parameters. The optimized solution will only lie within the defined parameter space. So, the method could not deliver an extended optimized or new design, e.g. if the engineer missed a parameter or the defined parameters could not model a certain or unknown shape.

Optimization of 3D flows (rough classification, incomplete):

0) Optimization by hand and engineering knowledge

1) Parameter optimization

- Sizing, parameters to be defined: e.g. diameter, radii, cone angle, location of gasket holes
- Change parameters → 1D optimization problem
- Numerous runs often solved with DoE
- e.g. modeFRONTIER, HyperStudy, Optimate+, HEEDS, Isight

2) Surface/Shape optimization

- Surfaces have to be identified which should be optimized
- Move nodes/vertices → 2D optimization problem
- Either morphing of flow domain surface (again 1D parameter level) solved with DoE or
- Adjoint/gradient solvers need high numerical accuracy
- e.g. CFD-code specific morpher, ANSA, Sculptor, *Adjoint*

3) Topology optimization

- Design space has to be defined
- Switch on/off finite volumes → 3D optimization problem
- Result is a stepped brick model which will be smoothed
- e.g. *TOSCA Fluid*

Combination

Figure 1. Overview / classification of 3D flow optimization.

Another optimization class is the surface or shape optimization. Here, surfaces have to be identified, which should be optimized. The optimizer has to move nodes or vertices. Therefore, it is a 2-dimensional optimization problem. Often morphing techniques are used to modify and move the local surface. The morpher may also stretch the attached volume mesh. These problems are also solved by many CFD runs and DoE. In the last few years, adjoint solvers have been implemented into CFD codes, which may calculate the sensitivity of the flow depending on so-called cost functions like pressure drop "(STAR-CCM+, 2010)." The CFD mesh is morphed in a further step based on this sensitivity result. Due to calculation of gradients, adjoint solvers need high numerical accuracy and good convergence.

A further optimization class is the topology optimization. A so-called design space has to be defined and will be meshed. During the optimization run, the finite volumes will be switched on or off. The result is a stepped brick model, which will be smoothed afterwards. Working on topology means a 3-dimensional optimization problem. This method does not need any parameterization of the geometry. One of the first commercial software systems for CFD topology optimization is Tosca Fluid "(Tosca Fluid, 2012)."

2. Used CFD Optimization Strategy

Often a combination of the optimization methods described above is very successful. In this paper, the combination of CFD topology optimization with a following shape optimization using an

adjoint solver has been applied. Finally, an optimized 3D design of the flow region has been determined as input for a first CAD design. Figure 2 illustrates the workflow of the combined CFD topology and shape optimization.

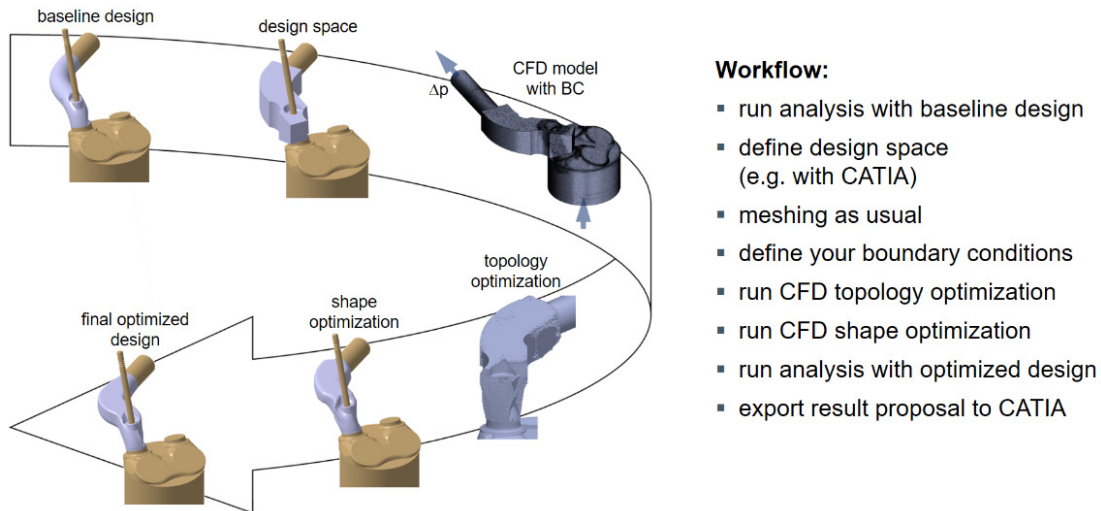


Figure 2. Workflow of combined CFD topology and shape optimization.

First of all, a possible flow region, the so-called design space, has to be defined, which is colored in grey. The design space may be created within a CAD system like CATIA or by extrusion of a surface mesh including modifications by hand. After standard meshing the model (here with STAR-CCM+), the boundary conditions have to be assigned. Now, the model is ready for the topology optimization with Tosca Fluid. The result is a stepped brick model of the optimized design, which has to be smoothed. The smoothed surface is directly used to setup a shape optimization with the adjoint solver. Finally, the baseline and the optimized design will be analyzed and compared. At the end, the smoothed surface is exported as starting geometry for the CAD designer.

2.1 CFD Topology Optimization with Tosca Fluid

Topology optimization is well established and integrated in Ford product development for structural analysis and design. In the past, no commercial topology optimization tool for fluid flows was available. Tosca Fluid may optimize fluid flows by CFD topology optimization in an early design stage and will deliver a first design proposal.

The roots of Tosca Fluid have been developed at Daimler research in the early 2000s under the name "AutoDuct" "(Klimetzek, 2006), (Moos, 2004)." One of its inventors, Dr. Klimetzek, explains the methodology of CFD topology optimization by looking to the nature of a river delta. Zones of backflow and recirculation have been eliminated by accumulation of sand, called sedimentation. Tosca Fluid is using the same bionic idea and thus called bionic optimization method. CFD topology optimization has been also implemented using mathematical adjoint methods, but this topic is out of scope of this paper, see e.g. "(Othmer, 2006)." Up to now, Tosca Fluid has been implemented for STAR-CCM+ and ANSYS Fluent.

Following applications may be solved with Tosca Fluid so far:

- Pressure loss reduction (e.g. air ducts, water jacket, oil paths)
- Flow efficiency by uniform velocity distribution (e.g. pump-inlets, elbow pipes)
- Flow balancing tasks (e.g. manifolds, flow splitter)

But how does CFD topology optimization work? The procedure will be described by means of an exhaust port example. A so-called design space has to be defined, which is colored grey, shown in Figure 3. It is a possible flow region, which connects the given inlet and outlet cores. The design space is usually limited by clearance planes to other parts like e.g. the cut-outs of head bolt, spark plug, and valve guide. The design space will be assembled to so-called frozen parts like inlet and outlet cores colored in gold, which should not be modified by the optimizer. After standard meshing with STAR-CCM+, pressure drop boundary conditions like used in the baseline model have been assigned.

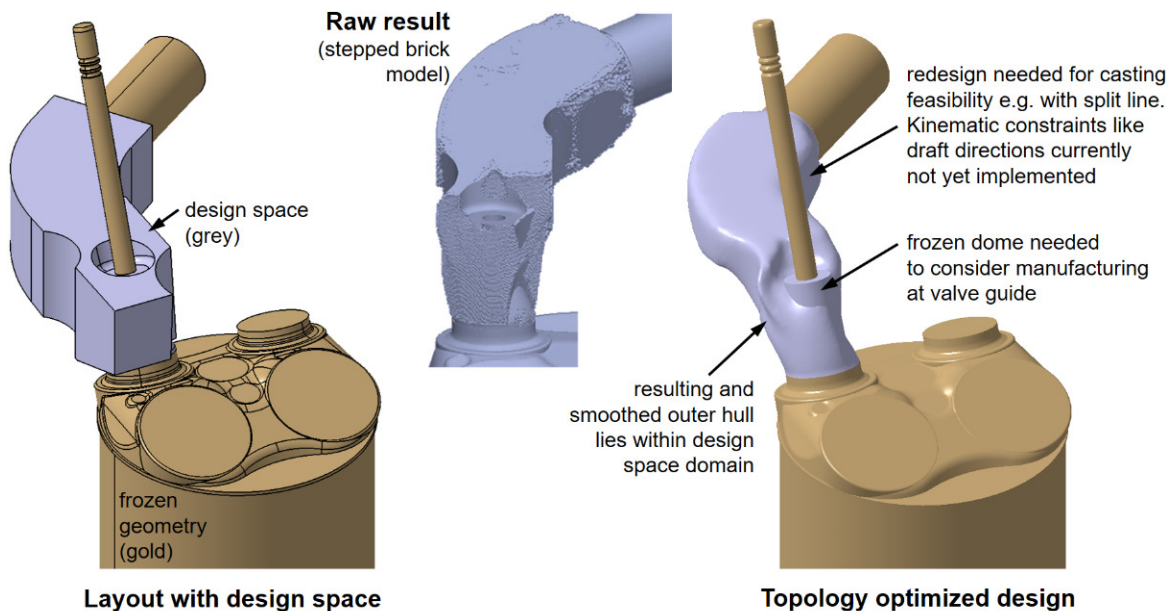


Figure 3. CFD topology optimization – Design space and optimized design.

During the optimization run, the finite volumes will be switched on or off. The result is a stepped brick model of the optimized design, which has to be smoothed (see Figure 3). The resulting surface could be smoothed with the Tosca software in a post-processing step as well. The smoothed surface is shown on the right-hand side of Figure 3. By definition, the resulting outer hull of the flow lies within the design space domain and implicitly fulfils the kinematic constraint of border violation. Conveniently, the smoothed surface is directly used for a CFD validation run to see the optimized flow. Nevertheless, a redesign of the exhaust port may be necessary for casting feasibility, for example to add a split line. Currently, kinematic constraints like draft directions are not yet implemented in Tosca Fluid.

CFD topology optimization with Tosca Fluid has shown its strengths in several Ford CFD applications. Pressure drops have been reduced and flow uniformity and mass flow rates have been substantially increased. Nevertheless, the optimization can be further improved using a shape optimization for fine-tuning afterwards.

2.2 CFD Shape Optimization

For the shape optimization, the Adjoint Solver of STAR-CCM+ has been used, which is available for the user since STAR-CCM+ version 9 "(STAR-CCM+, 2010)." In this example, the pressure drop has been defined as objective function or as a so-called adjoint cost function. The sensitivities will be calculated at control points, which are colored grey, shown in Figure 4.

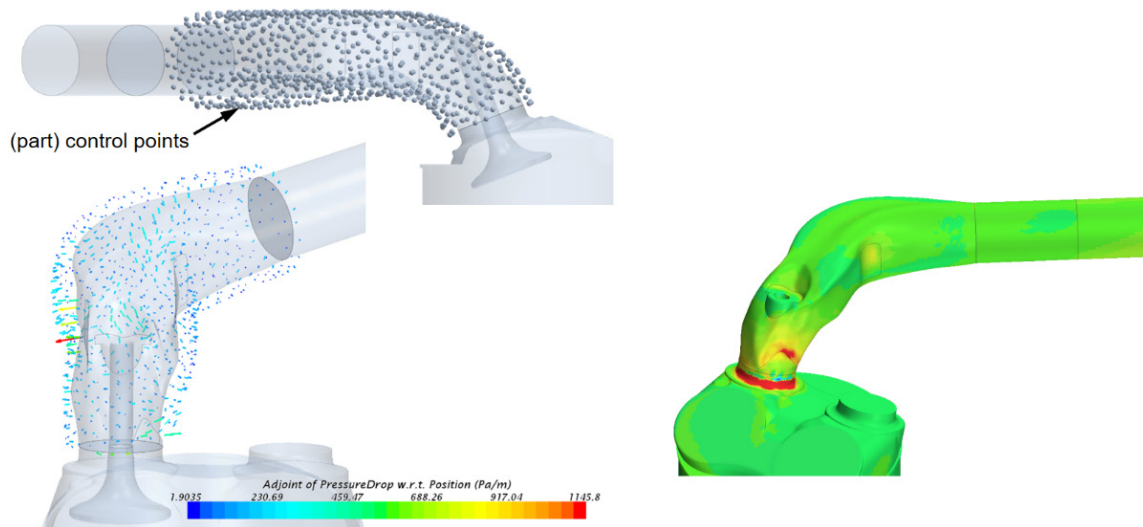


Figure 4. CFD shape optimization – Control points and adjoints / gradients.

Finally, the mesh is deformed and morphed in dependency of the local control point and based on its sensitivity multiplied with a morphing factor. The local mesh deformation is done on the surface and on the flow volume as well. Only small deformations should be used to avoid bad morphed cells, which will abort the CFD job. After a certain grade of deformation, the CFD model needs to be remeshed to guarantee best cell quality for further optimization runs. Furthermore, adjoint solvers need high numerical accuracy and good convergence due to calculation of gradients. The usage of Tosca Fluid and the Adjoint Solver of STAR-CCM+ is currently restricted to steady-state and single-phase flow physics. But this is often sufficient to have a first layout and impression of the new hull of the flow region. After the shape optimization run with the adjoint solver, the CFD engineer will receive a highly smoothed surface for an improved flow. In some Ford applications, the additional improvements by the shape optimization lie in the magnitude order of those of the topology optimization.

The optimization procedure presented above has been applied to several applications at Ford in the field of powertrain and vehicle aerodynamics. In the following, some of these Ford applications are described in more detail.

3. Optimization of a Water-Pump Inlet

One of the first Ford applications to test Tosca Fluid was the development of the water-pump inlet for the Ford 2.0L EcoBlue diesel engine with four cylinders "(Hopf, 2014)."

Often an engine should fit into several vehicle applications. There are different car types with different sizes, a longitudinal North-South installation or transversal East-West installations. At the end, some parts have to be modified to match all these package configurations. A side view on a cylinder block with the water-pump and the turbo system is shown in Figure 5. Driven by such package constraints, the shown coolant pipe to the water-pump inlet had to be redesigned resulting in a 180 degree coolant direction change (U-turn) in front of the water-pump inlet. The U-flow shaped design will generate an inhomogeneous and non-uniform inlet flow at the water-pump entry at higher mass flow rates. This will reduce water-pump efficiency and might cause cavitation at the impeller.

For this case, the first-in-industry software tool for fluid flow topology optimization called Tosca Fluid has been used, coupled with the CFD code STAR-CCM+.

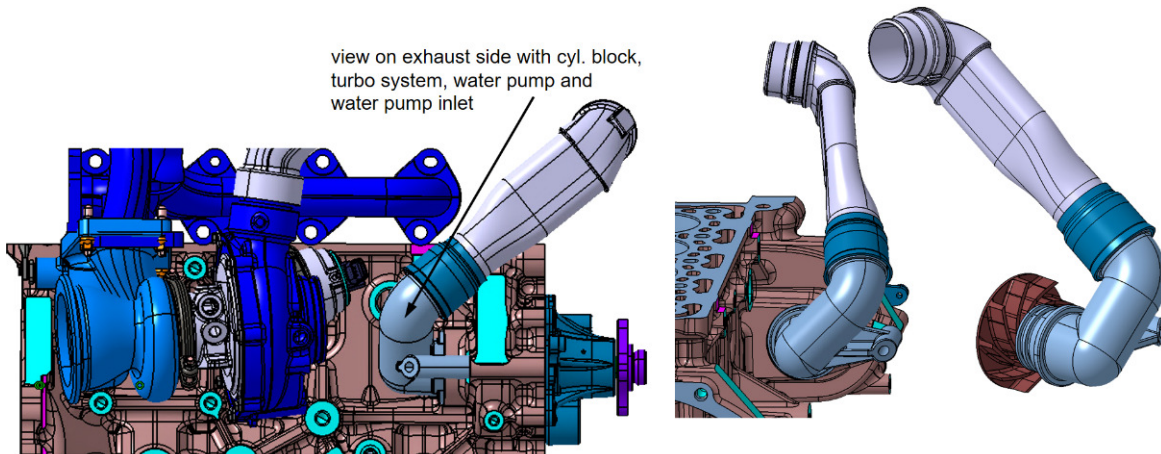


Figure 5. Water-pump inlet of an engine.

To analyze the current situation, the flow region has been extracted out of the pipe. After import of the geometry, the model has been meshed with polyhedron cells and two prism layers (see Figure 6). At the outlet, the CFD mesh has been extruded to reduce the flow turbulence for stable flow conditions at the outlet boundary. Water/glycol has been used as coolant material properties. Boundary conditions at the inlet are mass flow inlet with maximum flow and pressure outlet condition at the outlet. The CFD simulation has been setup with following conditions: Incompressible, steady-state, realizable $k-\epsilon$ turbulence model with two-layer, all y^+ wall treatment, and 2nd order segregated flow scheme.

Figure 6 shows the streamlines colored with the magnitude of the flow velocity. As expected, the flow will be pushed to the outer walls at high mass flow rates. After the corner in front of the outlet, a flow separation with a large recirculation zone can be observed.

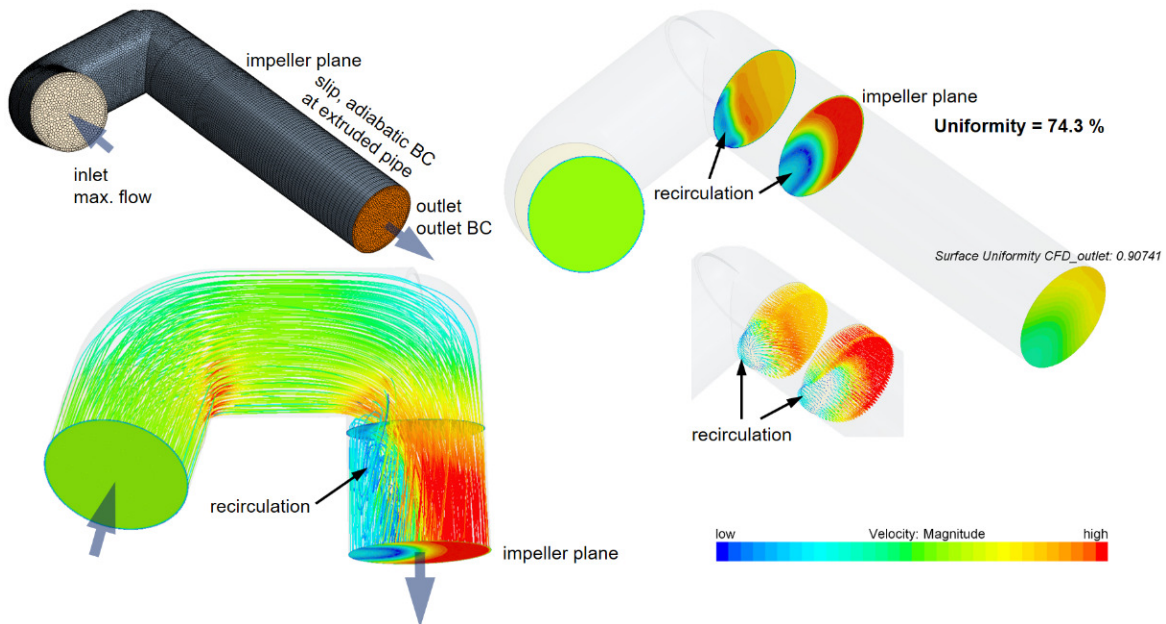


Figure 6. CFD mesh, boundary conditions, and CFD results of baseline model.

Within the impeller plane, there is a very high flow gradient, which will reduce water-pump efficiency and might cause cavitation at the impeller. The uniformity index at the impeller plane is 74 %, which should be increased. The main task for this problem is to minimize or to avoid the recirculation to increase the flow uniformity.

Using topology optimization, the next step is the definition of the design space shown in Figure 7.

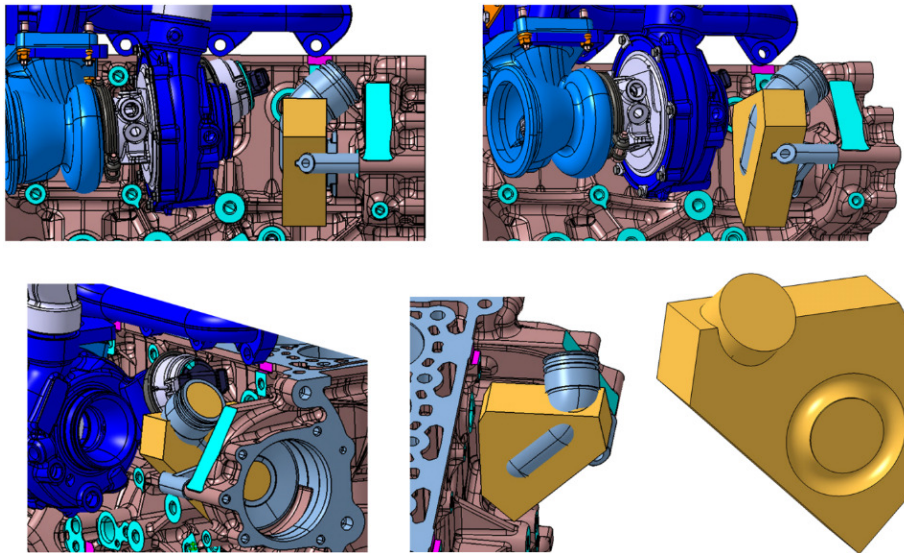


Figure 7. Design space for topology optimization.

After standard meshing with STAR-CCM+, the boundary conditions have to be assigned. In order to force the optimizer to generate a uniform velocity distribution within the impeller plane, a constant velocity field is set at this plane via a mass-flow inlet boundary condition. In front of the outlet, a trumpet-shaped design has been installed for a smooth outlet flow (shown at the right bottom of Figure 7). This trumpet shape is a result and a lesson learnt from former topology optimization runs to keep the flow attached.

Figure 8 illustrates the CFD results (streamlines and velocities) of the baseline design versus the topology optimized design. After the optimization, the critical recirculation zone and other backflow zones have been eliminated. The topology optimized design shows a much more homogenous and uniform velocity distribution at the impeller plane. One reason is the larger inner outlet radius. Another reason is the reduced main-jet velocity by a larger cross-section in front of the outlet radius. The flow may better follow the radius at lower local flow speeds. The overall maximum velocity has been reduced about 15 % compared to the original design. Looking on the velocity distribution of the impeller plane section-cut shows much improvement. The flow is more homogenous and uniform. The uniformity index rises about 20 % to 95 % at the impeller plane. A further result is the pressure drop, which has been reduced about 75 %.

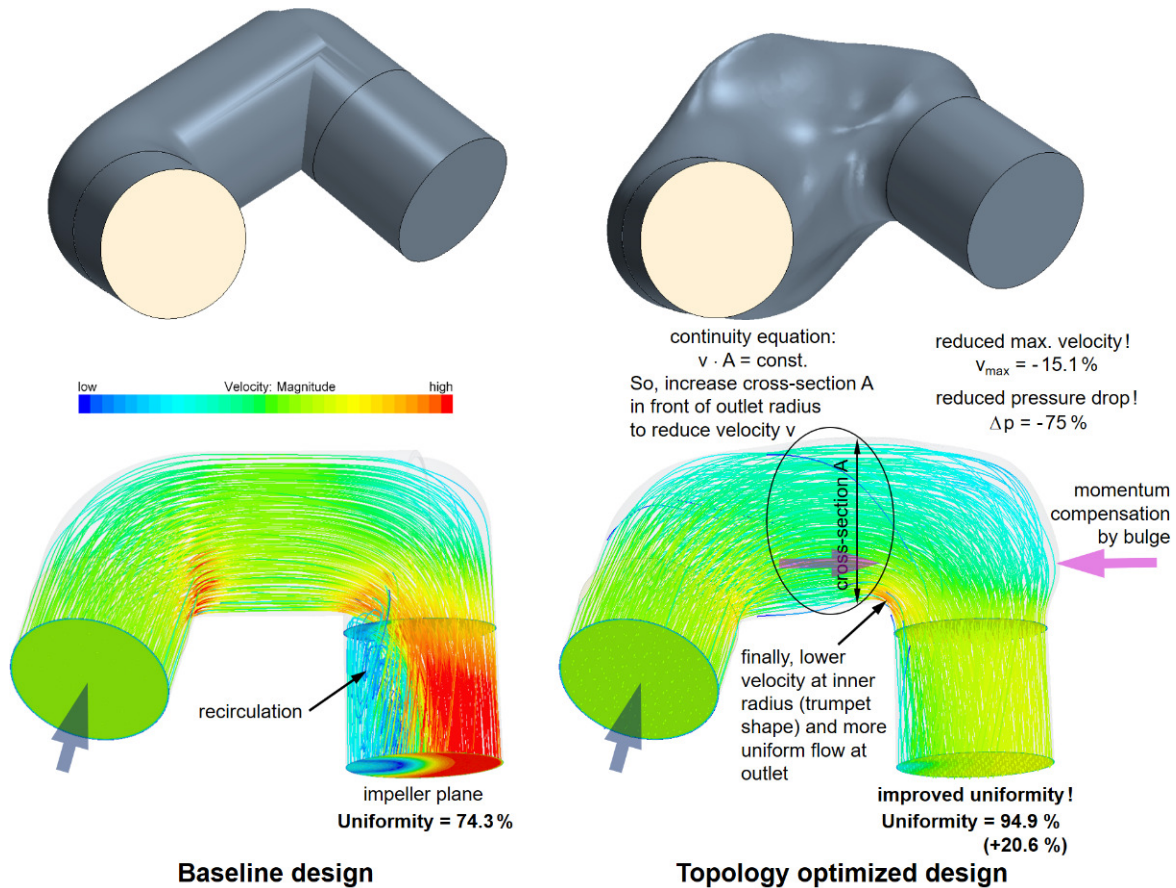


Figure 8. CFD results of baseline vs. topology optimized design.

These results of the topology optimization are promising. Meanwhile, the package situation and engine configuration has been changed. The optimization has been run with the updated configuration to support the design to a feasible solution. The flow domain of a feasible late CAD design is shown in Figure 9 including a picture of the final Ford series design.

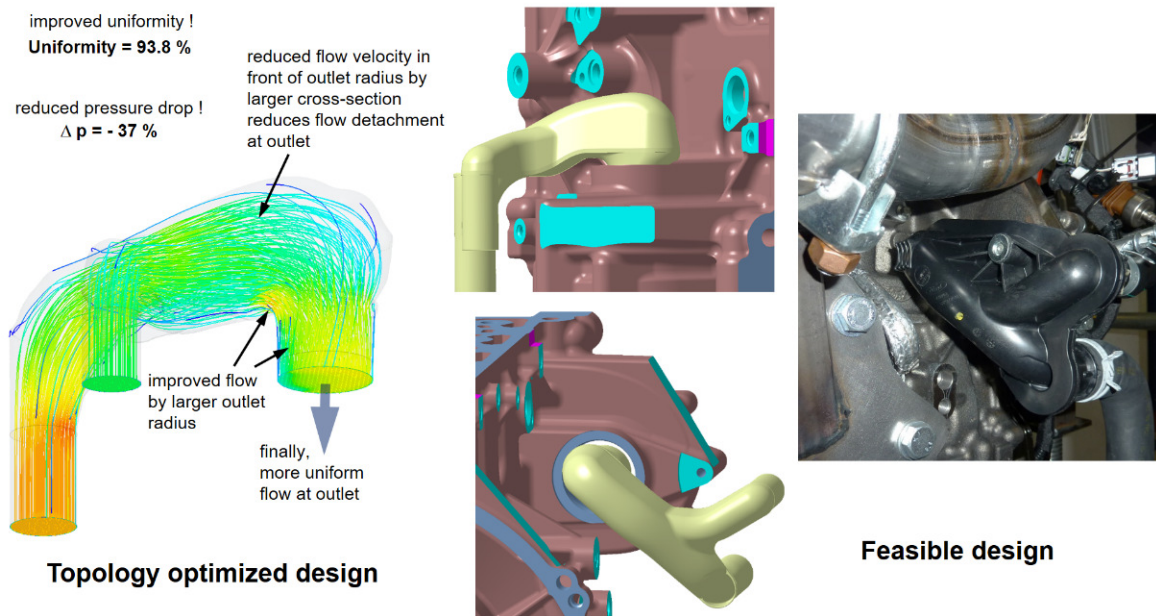


Figure 9. CFD results of topology optimized design and feasible design.

4. Optimization of Ports in Integrated Exhaust Manifolds

Another Ford application to test Tosca Fluid and the adjoint solver under air/gas conditions was the optimization of exhaust ports to increase the mass flow rates at a given pressure drop.

Modern engine architectures do require gas flow handling on small space. Driven by such package constraints, new cylinder-head designs are guiding the exhaust gas to the turbocharger system via an integrated exhaust manifold (IEM). Key benefits of the IEM are improved weight, cost reduction, fuel-efficient stoichiometric operation of gasoline engines ($\lambda = 1$), and the possibility of running exhaust gas temperatures up to 1050°C "(Borrmann, 2008), (Ernst, 2011)." Meanwhile, sophisticated boosting systems with two turbo chargers give an improved maximum low-end torque at very low engine speeds and a faster transient response. At Ford, an advanced boosting concept has been investigated, where two turbo chargers are served via two separate integrated exhaust manifolds "(Kramer, 2014), (Weber, 2018)."

Figure 10 shows a view inside the cylinder head with two IEMs colored in red. One IEM connects all left-sided exhaust valves and the other IEM connects all right-sided exhaust valves, each of them ending separately at an exhaust flange of the cylinder head. To cool the hot exhaust ports, a water jacket is needed. It is shown in Figure 10 colored in transparent grey and is made out of several cores, which are also connected via vertical drillings.

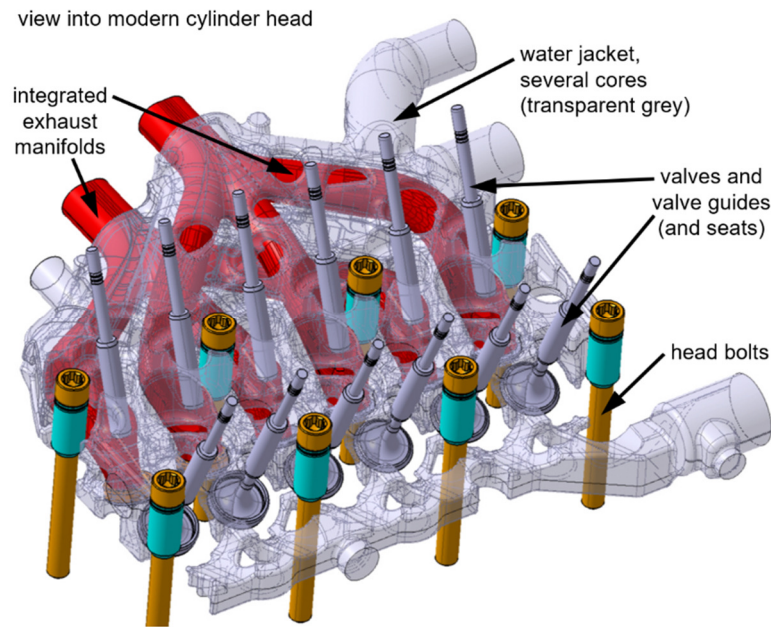


Figure 10. Integrated exhaust manifolds within an engine cylinder head.

Furthermore, the valves, valve guides, and valve seats are shown colored in grey including the head bolts in orange, which connect the cylinder head and block. Especially, the outer exhaust ports of the IEMs are running through a package labyrinth and are heavily bent within smallest space. Furthermore, the port cross-sections are reduced by valve stem, spark plug clearance, valve guide, and head bolt cut-outs. These issues lead to higher pressure drops causing a reduction of mass flow rates and flow efficiency. At the end, higher emissions, increased fuel consumption, and gasoline engine failures by higher knock sensitivity could be the consequences.

To improve this situation, a combined optimization strategy has been used based on CFD topology optimization and a following shape optimization.

To analyze the current situation, the flow through each exhaust port has been calculated. A classical design for exhaust ports is a tube with a constant circular cross-section including some cut-outs around spark plug, valve guide, and head bolt. This design has been taken as baseline model for further comparison. Port 1, which is the very left port in Figure 10, has several strong bends along its flow path and is taken as an example to demonstrate the optimization procedure presented in this paper. The flow region has been extracted out of the cylinder head and block. The block bore, the combustion chamber for four valves, and one exhaust port are shown in Figure 11. After import of the CATIA geometry into the CFD code STAR CCM+, the model is meshed with polyhedron cells and two prism layers. Mesh refinements are made around the opened exhaust valve (see Figure 11). At the outlet, the CFD mesh has been extruded to reduce the flow turbulence for stable flow conditions at the outlet boundary. Here, slip and adiabatic boundary conditions have been used at the wall.

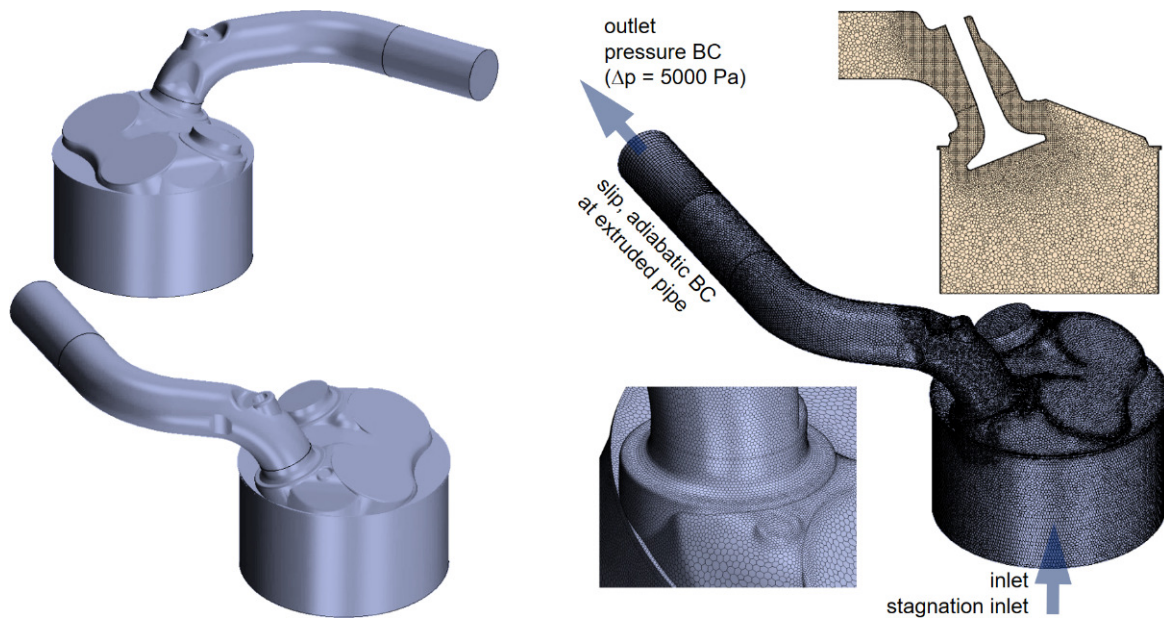


Figure 11. CATIA design, CFD mesh, and boundary conditions of baseline design.

During engine operation, the hot exhaust gas is pushed out of the cylinders in intervals. This is a highly transient event. Furthermore, the exhaust gas can exceed 1000°C at full load operation. These boundary conditions cannot be realized on flow bench test-rigs. Therefore, exhaust ports are measured with the same procedure as used for intake ports. A pressure drop, for example 5000 Pa, is set and the resulting mass flow rate will be determined under steady-state conditions at room temperature. The boundary conditions in the CFD model are stagnation inlet at the inlet and a pressure outlet condition at the outlet (see Figure 11). The chemical material properties of air have been set as ideal gas at 25°C according to the flow bench conditions.

The flow results of the CFD analysis for the baseline design are shown on the left-hand side in Figure 13. There are higher flow velocities at the sharp bends and in regions with smaller cross-sections, e.g. around the exhaust valve. Typically, these higher local flow velocities indicate a higher pressure drop and thus a lower mass flow rate. Reasons for this are flow separations after the bends and therefore a reduced effective cross-section for the flow and a reduced port flow efficiency by a deteriorated flow.

Now, the main task was the optimization of the port shape to maximize the mass flow rate at a given pressure drop. The combined optimization strategy using CFD topology and shape optimization has been applied to this example called bionic port design "(Hopf, 2016), (Hopf, 2017)." Different result stages during the optimization of exhaust port 1 are shown in Figure 12.

Figure 13 shows the CFD results in terms of streamlines and flow velocities as a comparison between the baseline design and the optimized designs. The topology optimized design shows reduced flow velocities especially around the bends. The main reason is the reduced main-jet velocity by larger cross-sections in front of the corners. The flow may better follow a radius at lower local flow speeds.

The CFD results of the (topology and) shape optimized design are shown on the right-hand side in Figure 13. This design shows a smoothed outer surface with further reduced flow velocities especially around the bends. Again, main reason is the reduced main-jet velocity by larger cross-sections in front of the corners. Furthermore, the cross-sections around the exhaust valve have been enlarged by the shape optimizer. Finally, the mass flow rate of the exhaust gas of the first port has been increased by more than 8 % compared to the baseline design.

According to the simulation steps described in the last few chapters, the new optimized surfaces of all exhaust ports have been determined. Figure 14 shows the fully optimized geometry versus the baseline design. Additionally, the streamlines and flow velocities of ports 1 and 6 are shown in this figure.

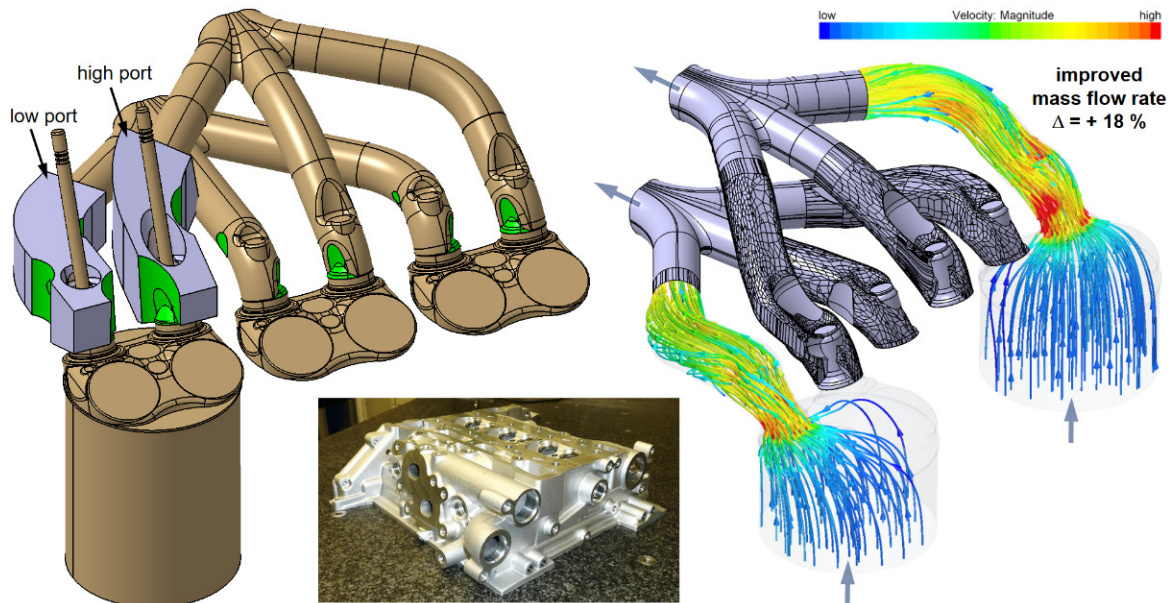


Figure 14. Definition of design spaces for all exhaust ports and CFD results of topology and shape optimized design.

Looking on all exhaust ports, the mass flow rate or flow coefficient could be improved up to 18 % using the proposed optimization strategy. Meanwhile, flow bench measurements with the built cylinder head have verified the numerical results.

At the end, the mass flow rates through all exhaust ports have been substantially improved using the combined strategy of CFD topology and shape optimization (bionic port design). Finally, the smoothed surfaces of all ports have been exported as starting geometry for the CAD designer.

5. Optimization of a Cast Flow Splitter

In the next example, a cast flow splitter within an oil circuit of a Ford engine has to be optimized, which is shown in Figure 15. An incoming flow is split by a ratio of 45/55 to two drilled outlet pipes, which are connected via a cast passage.

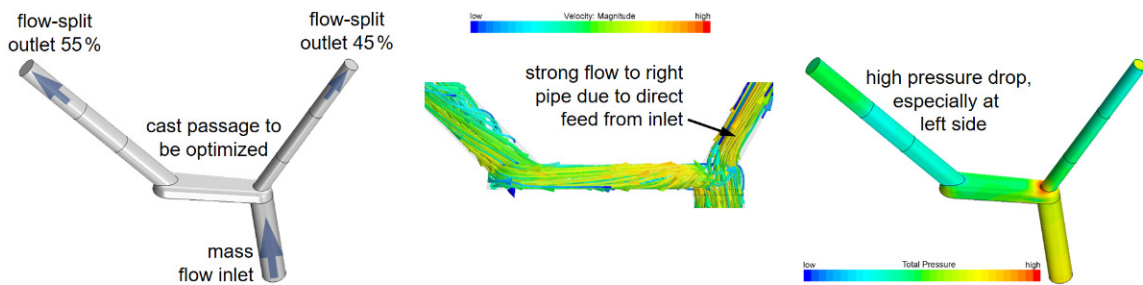


Figure 15. Flow splitter with CFD results of baseline design.

For the flow, full mass flow rate at the inlet, the flow split at the outlets, and material properties of oil 5W20 at 140°C have been used. The CFD analysis of the baseline design shows a high pressure drop at the left outlet indicating a high drag through the cast passage (see Figure 15).

The neighboring parts do not allow much room to define the design space, which is shown in Figure 16 colored in magenta. On the right-hand side of this figure, the raw result as well as the smoothed surface of the topology optimized design are illustrated.

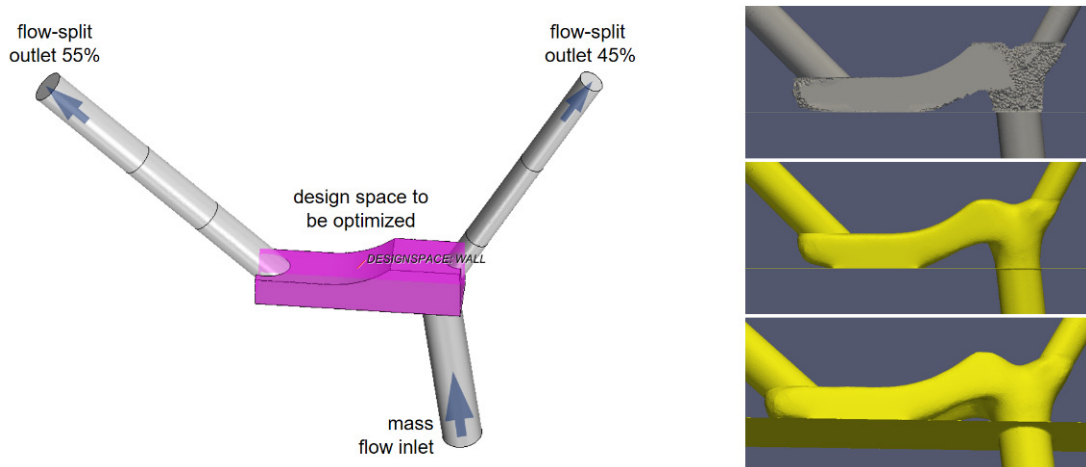


Figure 16. Design space and Tosca Fluid results.

A rerun of the smoothed topology optimized design give the CFD results shown in Figure 17.

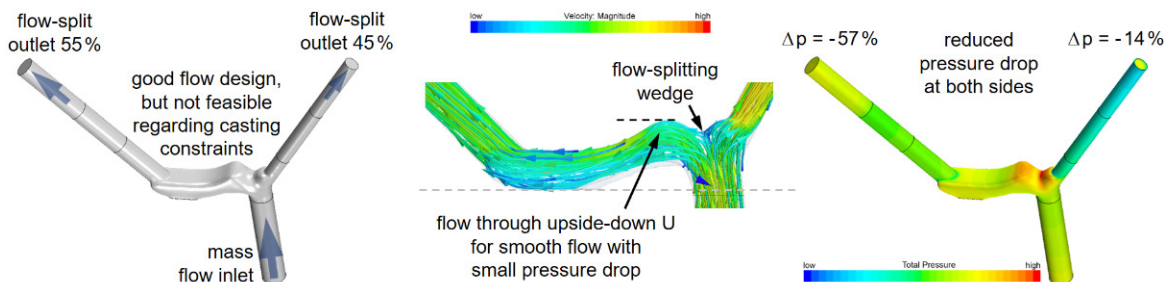


Figure 17. CFD results of topology optimized design.

The incoming flow is split at a wedge over the inlet. Especially, the flow to the left side is running with lower drag through the curved passage. The pressure drops to both outlets have been reduced. The topology optimized design has a good flow behavior but is not feasible design. The "hill" at the bottom side cannot be cast. Here, casting constraints like the definition of casting angle and draft direction are missing. Therefore, these manufacturing conditions have been added by hand shown in Figure 18. The feasible (cast) design shows a slightly reduced flow quality compared to the topology optimized design. Main reason is one recirculation zone positioned at the "hill" described above.

At the end, the flow of the feasible design could be improved with Tosca Fluid and the pressure drop at the left outlet has been substantially reduced compared to the baseline design.

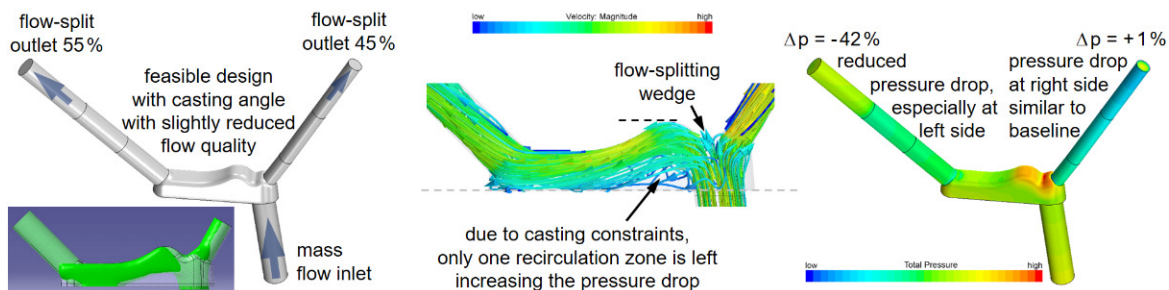


Figure 18. CFD results of feasible (cast) design.

6. Summary / Conclusions

In this paper, several Ford applications have been presented showing the optimization of three-dimensional flow problems of engine parts.

The proposed solution is a combined optimization strategy using CFD topology optimization with Tosca Fluid and a following shape optimization with the Adjoint Solver (bionic design). It is easy to setup and post-process the optimization runs. The CFD engineer may generate an initial design of a new CFD optimized part and give it to the designer. So, the designer may start with an optimized design solution. The new surfaces of the parts may also serve as an idea generator of new flow shapes, which may help to extend the parameter set for parametric optimizations. In the age of additive manufacturing, a 3D print of the parts is quickly possible.

For Ford, this new optimization workflow is highly efficient, reduces development time, improves result quality, and may reduce the number of expensive and time-consuming test-rig measurements.

7. References

1. Borrmann, D., Kuhlbach, K., Friedfeldt, R., Mehring, J. et al., "Cylinder Head Integrated Exhaust Manifold IEM Applied for Gasoline Downsizing Concepts," presented at 17th Aachen Colloquium, pp. 1001-1017, Aachen, Germany, 2008

2. Ernst, R., Friedfeldt, R., Lamb, S., Lloyd-Thomas, D. et al., "The New 3 Cylinder 1.0L Gasoline Direct Injection Turbo Engine from Ford," presented at 20th Aachen Colloquium, pp. 53-72, Aachen, Germany, 2011
3. Hopf, A., Hitchings, A., and Routledge, L., "CFD Topological Optimization of a Car Water-Pump Inlet Using TOSCA Fluid and STAR-CCM+," presented at STAR Global Conference, Vienna, Austria, 2014
4. Hopf, A., "Finding the Perfect Flow - Port Development Using CFD Topology Optimization and Adjoint Solver," presented at STAR Global Conference, Prague, Czech Republic, 2016
5. Hopf, A., Bartsch, G., Krämer, F., and Weber, C., "CFD Topology and Shape Optimization for Port Development of Integrated Exhaust Manifolds," SAE Technical Paper 2017-01-1339, doi:10.4271/2017-01-1339, 2017
6. Klimetzek, F.R., Paterson, J., and Moos, O., "AutoDuct: Topology Optimization for Fluid Flow," presented at 1. Konferenz für Angewandte Optimierung in der Virtuellen Produktentwicklung, FE-Design, Karlsruhe, Germany, 2006
7. Kramer, U., Klein, R., Hofmann, C., Stoffels, H. et al., "Extreme Downsizing of CNG Engines – Opportunities and Challenges," presented at 1st International Conference – Advanced Fuels for Sustainable Mobility, Haus der Technik, Nuerburgring, Germany, 2014
8. Moos, O., Klimetzek, F., and Rossmann, R., "Bionic Optimization of Air-Guiding Systems," SAE Technical Paper 2004-01-1377, doi:10.4271/2004-01-1377, 2004
9. Othmer, C., de Villiers, E., and Weller, H.G., "Implementation of a continuous adjoint for topology optimization of ducted flows," AIAA-2007-3947, 2006
10. Siebertz, K., van Bebber, D., and Hochkirchen, T., "Statistische Versuchsplanung: Design of Experiments (DoE)," VDI-Buch, 2017
11. STAR-CCM+ User Manual (Version 7.04.006), CD adapco Group / Siemens, 2010
12. Tosca Fluid User Manual (Version 2.3), FE-Design / Dassault Systèmes Simulia Corp., 2012
13. Weber, C., Kramer, U., Friedfeldt, R., Ruhland, H., and Krämer, F., "Development of a New Combustion Engine Dedicated to Methane Operation," 39th International Vienna Motor Symposium, 2018

8. Acknowledgments

Some results of this work have been performed as part of the research project "GasOn" assigned by the European Commission within the Horizon 2020 framework under grant agreement No. 652816 and have been conducted at Ford-Werke GmbH in Cologne and Aachen (Germany).

Some results of this work have been performed as part of the research project "THOMSON" assigned by the European Commission within the Horizon 2020 framework under grant agreement No. 724037 and have been conducted at Ford-Werke GmbH in Cologne and Aachen (Germany).