

Parametric optimization of an exhaust manifold using Isight, STAR-CCM+ and CATIA V5

Joel Davison

CD-adapco - London

Abstract

The work presented demonstrates how Isight may be used to drive STAR-CCM+ and CATIA V5 for parametric optimization. In this example, carried out for the CFD group of a major US automotive OEM, a DOE study was performed with the aim of minimizing the pressure drop along an exhaust manifold. Geometric modifications were carried out automatically in CATIA V5, and passed to STAR-CCM+ for flow analysis.

A key requirement from the OEM was to remove time-consuming manual iterations from the design process and this example case was setup to demonstrate the automatic optimization process using their existing design and analysis tools. Data is passed to and from CATIA using Design Tables linked with Excel, and STAR-CCM+ analysis was executed via the STAR-CCM+ Isight component managing all parameters, files and controlling the design exploration.

Keywords: CFD, optimization, automotive, CATIA, STAR-CCM+, Isight

1. Introduction

The current paper describes the modeling techniques required, and applied, to optimize an automotive internal combustion (IC) engine exhaust manifold design. A key requirement for success is the ability to easily and with minimal user interaction drive both a parametric design tool, in this case CATIA V5, and the engineering analysis tool STAR-CCM+. Consequently, Isight was deployed to encapsulate the optimized workflow. Such workflows using Isight have been demonstrated before at the SIMULIA Customer Conference by Otto & Jobusch from Rolls Royce (Otto, C, 2009).

The effectiveness and use of computational fluid dynamics (CFD) is strongly linked to the availability and cost of computer hardware. As this cost continues to be driven down, and accordingly the availability of high performance computers increases, so the amount of time the analysis tool spends solving numerical equations is reduced as well. A consequence of this reduced solve time is that the limit factor in the overall analysis is increasingly becoming dependent on the amount of time the engineer takes in interacting with the software itself.

To reduce the time an engineer takes interacting with the Computer Aided Engineering (CAE) tool the actions within this tool need to be automated. Both CATIA V5 and STAR-CCM+ have this ability in a number of different forms including the recording and playback of macros (small scripts containing application specific commands) and the capacity to use models as templates.

Leveraging these automation capabilities is relatively trivial in a single tool, however, the challenge is to effectively deploy disparate software tools in the automated workflow and easily allow them to communicate information with each other. To do this, a process automation tool is required. For this project Isight was chosen to drive the overall simulation workflow.

1.1 Background

The current case was produced in conjunction with SIMULIA to demonstrate the ability of Isight to integrate fluid flow analysis in STAR-CCM+ with parametric modeling using CATIA V5. The Computational Fluid Dynamics (CFD) group at a major automotive OEM was looking to expand their analysis capability to enable them to evaluate multiple designs in search of an optimized configuration more easily. A key goal of the CFD group was to significantly reduce the time taken to study the different design configurations by reducing manual interaction needed to setup and run the cases.

The case chosen to demonstrate this automated workflow is a four-port exhaust manifold built within CATIA V5. It is parameterized using four user-defined splines that describe the swept path of each runner through to the exit to the rest of the exhaust system. The splines themselves are defined by a Microsoft Excel spreadsheet with Cartesian coordinates specified for two points on each of the four splines.

The aim of the design of experiment (DOE) study is to find a design that minimizes the overall pressure drop of both the manifold as a whole but also additionally in each of the runner. To achieve this, the mid points of each of the splines are changed recursively to provide a modified sweep path for each of the runners.

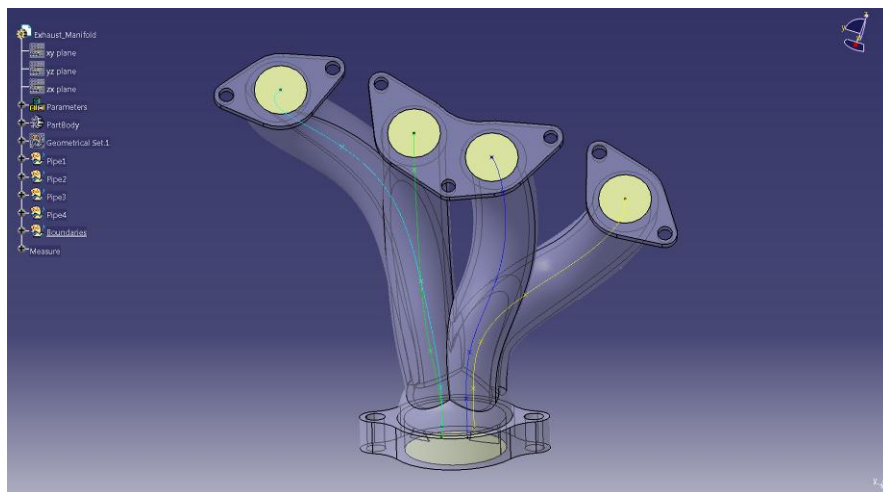


Figure 1. Starting manifold geometry showing the four splines used to define the runners

2. Model Setup

The optimization is run using a combination of three CAE tools: Isight 5.6 driving the workflow, CATIA V5 R20 controlling the geometric shape of the manifold and STAR-CCM+ v7.02 performing the flow simulation. Additionally Microsoft Excel spreadsheets are used to define the spline inputs and to read the runner lengths for post-processing.

2.1 Workflow

The workflow of each design point in the optimization study is as follows:

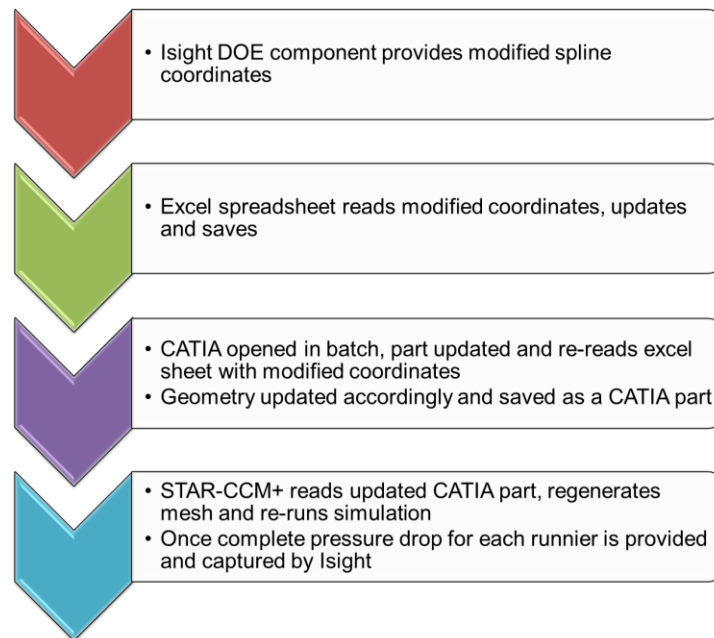


Figure 2. Overall analysis workflow

The above workflow is encapsulated in a DOE study using Isight. As well as the DOE component, specific Excel components are used to read/modify/save the input and output spreadsheets. A command line component is used to open CATIA in batch and to run the CATIA macro, and a STAR-CCM+ component is used for the flow analysis.

2.2 Isight Setup

The DOE component itself is setup using a L16 orthogonal array technique (as described by Ross, P, 1988), specifically chosen to maintain independence between the different factors, in this case the shape of the manifold runners. Four design points are specified for each of the factors as a percentage of the initial position of the spline mid-point. In this case values of -10%, -3%, +3%

and +10% of the starting value were used.. Four responses are prescribed for the study looking at the pressure drop down each of the runners.

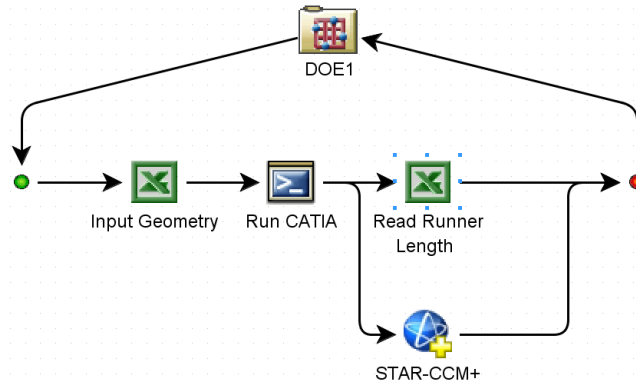


Figure 3. Isight design gateway setup

Input factors driven by the DOE component are each of the spline mid points. Responses defined as the pressure drop of each of the runners and the overall pressure drop of the entire system.

2.3 STAR-CCM+ Setup

STAR-CCM+ uses an unstructured finite volume solver to simulate flow, thermal and stress phenomena as described in the STAR-CCM+ user guide (CD-adapco). In this case only the flow element of the manifold is considered. The starting geometry passed from CATIA is shown below.

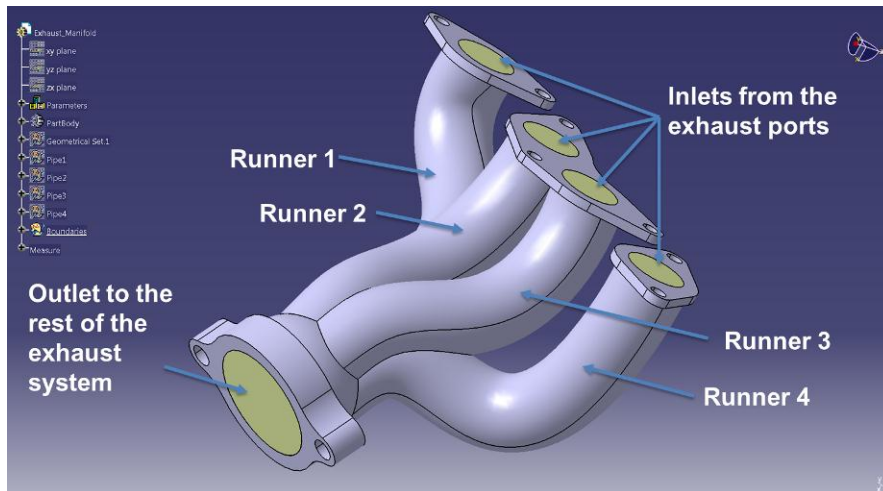


Figure 4. CATIA model setup read into STAR-CCM+

The geometry itself, in the form of a native CATIA V5 file (.catpart), is read into a pre-existing STAR-CCM+ template sim file (.sim is the native STAR-CCM+ format). The template sim file is in essence the same as any other simulation file as it has all the mesh, physics and post processing settings setup within it. The difference to a complete sim file lies in the absence of geometry.

The same blank sim file is used as a starting point for each of the design points in the study. Once the CAD has been read in to the template file, a Java macro is used to assign the relevant surfaces, identified by a patch number identifying each surface within the CAD model,) to the pre-defined boundaries.

The overall workflow within STAR-CCM+ is presented in Figure 5. The process is driven using the STAR-CCM+ Isight component which allows the user to choose which parameters to expose to Isight (as inputs and responses) and how to execute STAR-CCM+. In this case a Java macro is used to execute each of the steps seen in Figure 5.

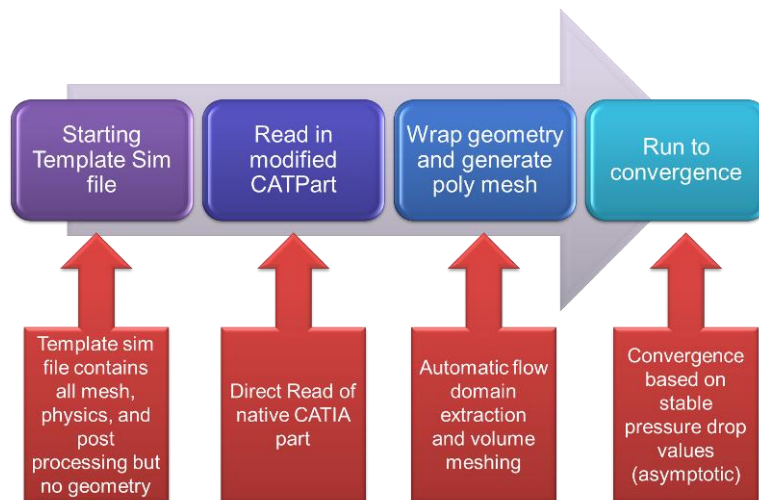


Figure 5. STAR-CCM+ workflow

2.4 Flow domain extraction

In Figure 4 it can be seen that, in common with most CAD geometries, the solid part of the manifold has been modeled. With this in mind it is necessary to not only read the geometry in to STAR-CCM+, but also to automatically extract the fluid domain. A number of different methods are available to create this flow domain, including automatic tools in STAR-CCM+ or manual creation by surface deletion and stitching in either CATIA or STAR-CCM+. The final selection was driven by the need to fully automate the process and negate possible sources of error in the setup process that may make the optimization study halt.

Considering this need for automation and robustness of the process STAR-CCM+'s surface wrapping facility was employed to extract the flow domain and hence to provide a surface for volume meshing. The surface wrapper itself is analogous to the inflation of a balloon within the geometry (or shrink wrapping for external flow cases) and reconstructs a clean, closed surface that is required to continue along the analysis pipeline.

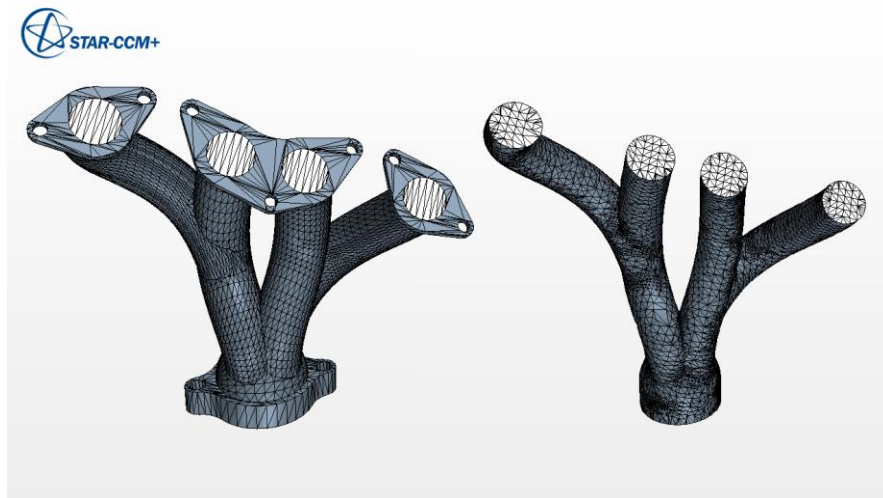


Figure 6. Imported (left) vs wrapped (right) geometry in STAR-CCM+

2.5 Meshing and Physics

With the flow domain complete, an unstructured polyhedral mesh is generated employing prismatic layers at the wall boundaries to accurately capture the boundary layer. The use of a polyhedral mesh is beneficial for flows such as those seen in this runner as it provides a suitable balance between accuracy and automation. The polyhedral structure reduces numerical dissipation ensuring greater accuracy than other meshes, for example tetrahedral, with an equivalent cell count (Benhamadouche, 2005).

The flow field is solved using STAR-CCM+'s segregated flow solver with the working fluid modeled as air obeying the ideal gas law. In reality the gas used will be a complex mix of different components representing the exhaust gases post combustion but in this case, and to aid convergence and run speed, compressible air is considered sufficient to deliver representative pressure drops.

The solution is solved steady state with the flow considered turbulent with a Reynolds number of around 10,000-15,000 and so the standard $k-\epsilon$ turbulence model is used. Convergence is judged based upon the pressure drop for the four runners varying less than one Pascal across 10 iterations. Boundary conditions and the polyhedral volume mesh are shown in Figure 7.

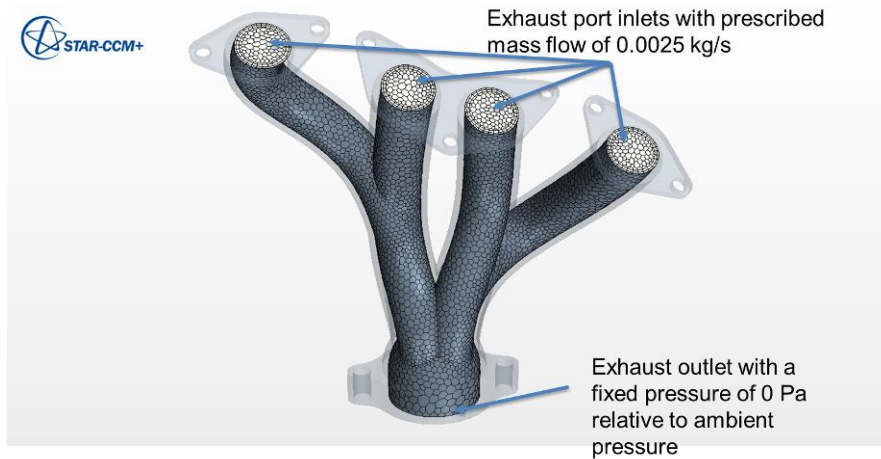


Figure 7. STAR-CCM+ volume mesh and boundary setup

3. Results

The DOE study was run with 16 design points. Out of these model points four failed due to the CATIA model not being able to regenerate. This regeneration failure was caused by the spline definition no longer generating a valid sweep path for one or more of the runners.

The initial design shows a total pressure drop across the system of 50.95 Pa. After 12 completed runs this is reduced to 47.97 Pa, a drop of 2.98 Pa or 5.85%. The most significant pressure reduction was seen down the third runner where the pressure drop was reduced from 11.75 Pa to 7.98 Pa, a reduction of 32.07%. The highest total pressure drop of all designs in the study was actually 92.93 Pa, 82.4.23 % higher than the initial design.

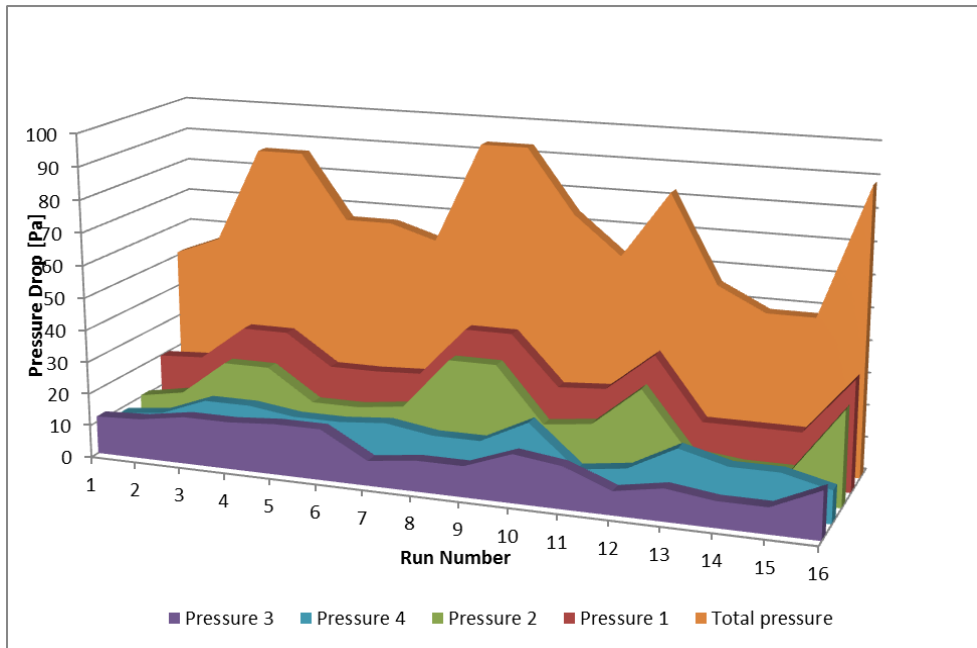


Figure 8. Pressure drop for each of the runners and total for the system across each design point

A total system pressure drop reduction of under 6% may seem relatively insignificant but a 32% reduction in a single runner is valuable given that in reality only one runner at a time is likely to be fully utilized. It is seen in Figure 8 that the first two runners generally have the greatest contribution to the overall system pressure drop. This also lends greater weight to the significance of a 32% drop in runner 3.

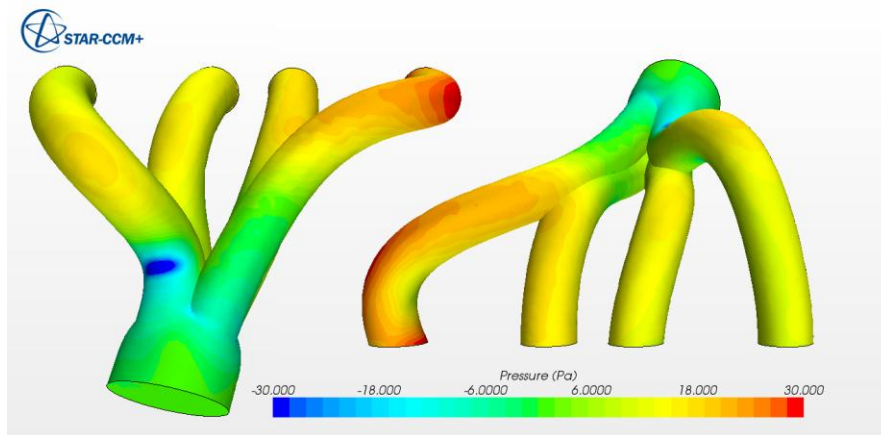


Figure 9. Pressure field in the initial design

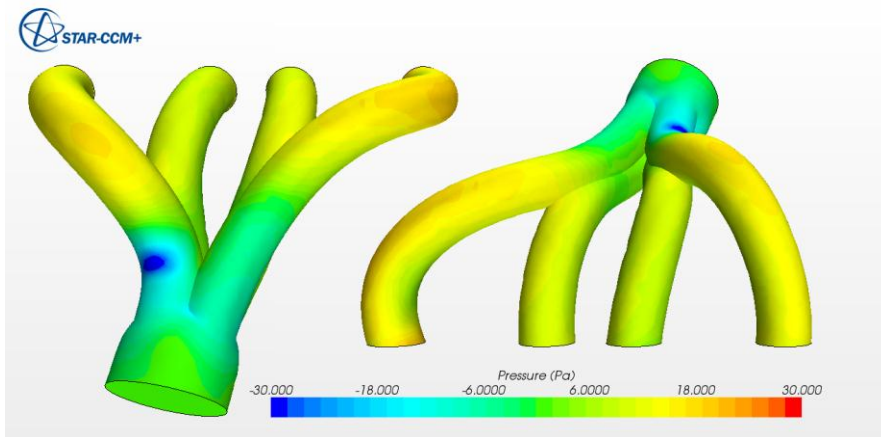


Figure 10. Pressure field in the optimized design

Analyzing the flow field of the initial design it can be seen that there is a significant pressure spike near to the inlet of runner one (see Figure 9). In the optimized design the magnitude is significantly lower, as evidenced by Figure 10.

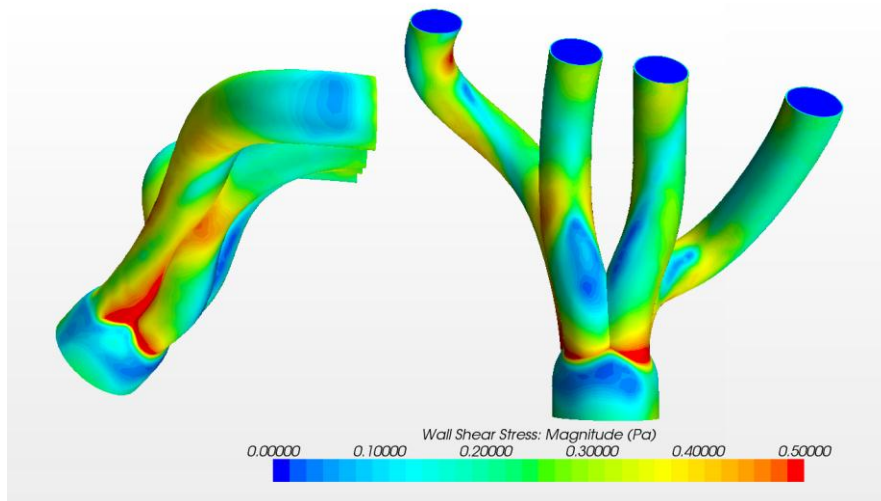


Figure 11. Wall shear stress magnitude in the initial design

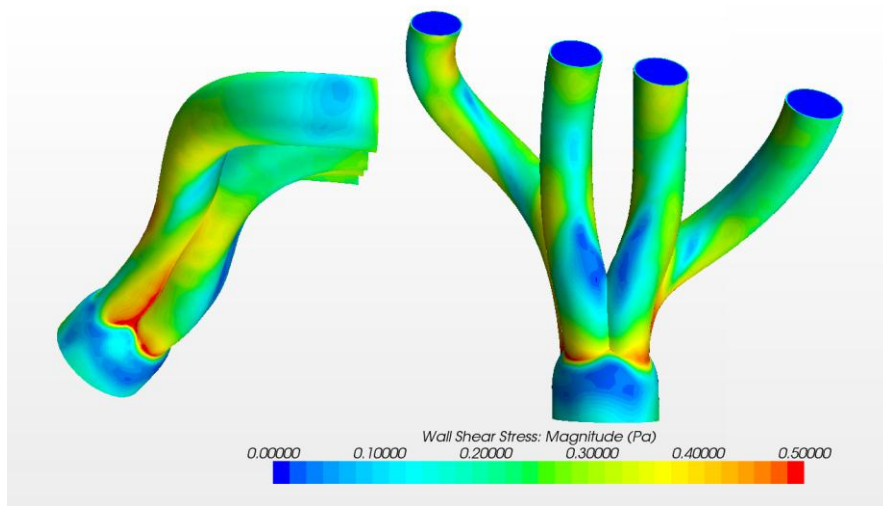


Figure 12. Wall shear stress in the optimized design

A possible source of this reduced pressure drop can be seen in Figure 11 and Figure 12 where wall shear stress is shown. In the improved design the overall distribution of the shear stress is lower and local areas of high stress are also reduced in both size and magnitude.

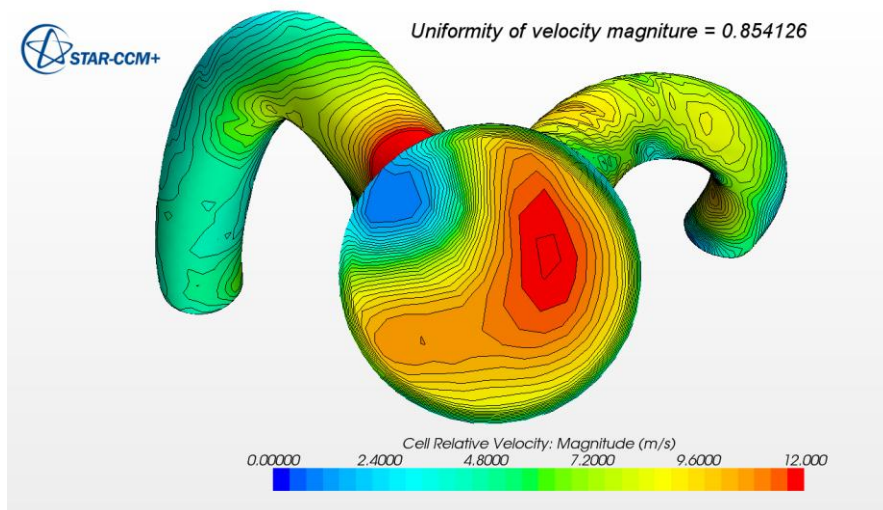


Figure 13. Velocity magnitude at the manifold exit in the initial design

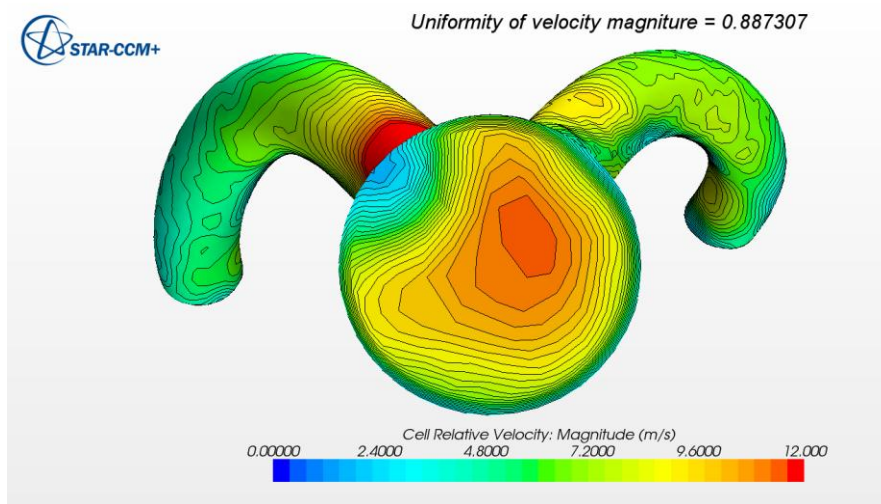


Figure 14. Velocity magnitude at the manifold exit in the improved design

Other effects of the improved design are an improved uniformity of the flow field at the exit as shown in Figure 13 and Figure 14. An enhancement of uniformity may have beneficial downstream effects improving the flow distribution as it enters an after treatment device or turbocharger.

3.1 Influence of mass flow rate on pressure drop

After the completion of the initial study an additional analysis was carried out to study the effects of varying mass flow on the system. The same parameters were used to control the shape of the manifold, with the mass flow rate studied at 10% less and 10% more than the baseline, 0.00225 kg/s & 0.00275 kg/s.

The results of the 0.00225 kg/s data show much the same behavior as the original test case with the same configuration leading to greatest reduction in pressure drop. In this case the magnitude of the reduction is significantly smaller though, 43.38 Pa to 42.48 Pa, a drop of only 2%. In this case the greatest improvement was seen in runner three with an improvement of over 30%, representing a reduction in pressure drop of 3.31 Pa (from 10.30 Pa to 6.99 Pa).

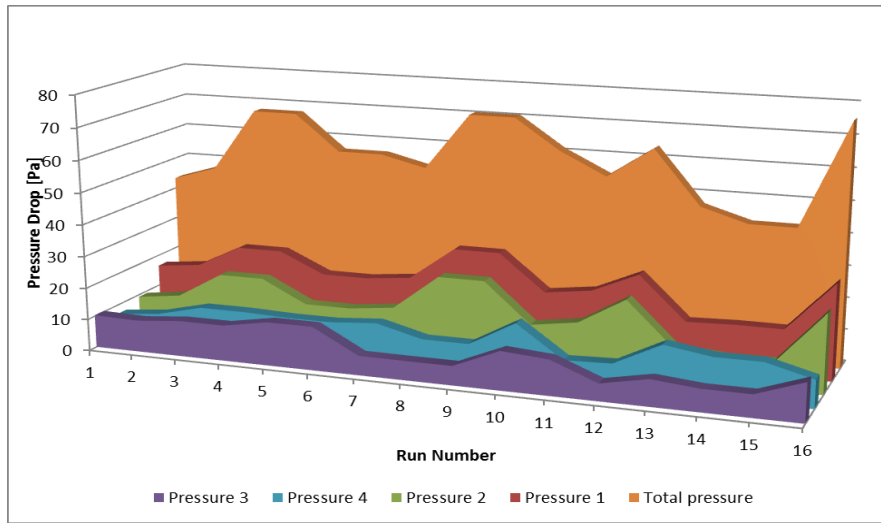


Figure 15. Pressure drop for each of the runners and total for the system for the 0.00225 kg/s study

The higher mass flow rate also exhibited similar behavior but again with a reduction in the relative differences between the initial and the optimized designs. This time the original design shows a pressure drop of 60.76 Pa and the optimized 60.17 Pa, a reduction of less than 1%. Runner three again showed the greatest improvement with a drop from 14.95 Pa to 17.25 Pa, a 33.16% reduction.

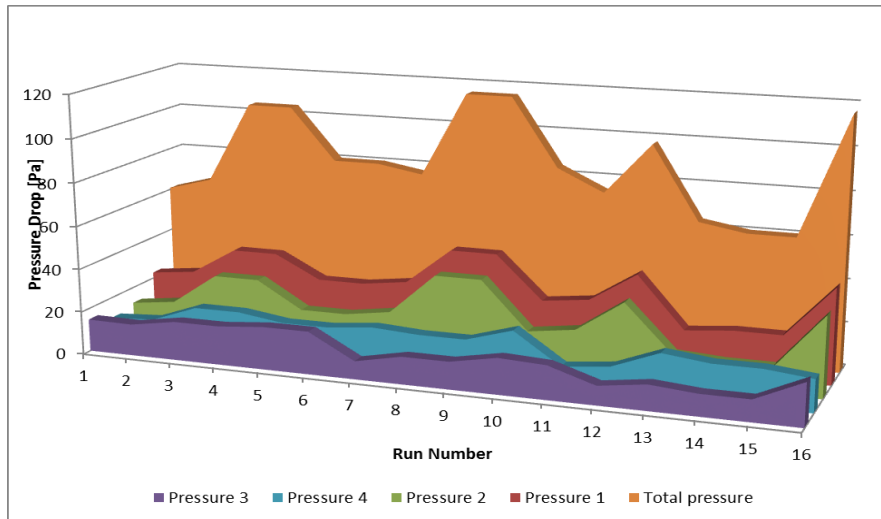


Figure 16. Pressure drop for each of the runners and total for the system for the 0.00275 kg/s study

4. Conclusions

This paper summarizes how Isight can be used to encapsulate multiple CAE tools to better explore the design space of an exhaust manifold. Both CATIA V5 and STAR-CCM+ have been integrated into the automatically run process demonstrating minimal user interaction and with it the benefits of automation.

Beyond the workflow benefits it also shows how the DOE study performed leads to an optimized design of an exhaust manifold with an overall reduction of pressure drop approaching 6% with a single runner reduction of over 30%. These improvements have come at relatively little cost to the end user beyond the time taken to setup the initial models and to run such a study manually would be prohibitively time consuming. Such a process gives the engineer confidence that the design they are producing for prototyping and testing is going to be closer to optimum. This in turn can result in cost savings throughout the engineering process through a reduced number of prototypes and a shorter testing phase and overall design cycle time.

5. References

- Benhamadouche, S., Laurence, D., Moulinec, C., & Peric, M. (2005). LES in a U-bend pipe meshed by polyhedral cells. *ERCOFTAC International Symposium on Engineering Turbulence Modelling and Measurements (ETMM6)*. Sardinia.
- CD-adapco. (n.d.). STAR-CCM+ User Guide.
- Otto, C., & Jobusch, C. (2009). Automated design processes with Isight and their. *Proceedings of the SIMULIA Customer Conference*. London.
- Ross, P, J. (1988). *Taguchi Techniques for Quality Engineering*. New York, NY: McGraw-Hill Publishing Company.

6. Acknowledgments

I would like to thank Malik Kayopv and Dwayne Dutton of SIMULIA who carried out the initial work on this project and provided valuable help to update and re-run the cases.

[Visit the Resource Center for more SIMULIA customer papers](#)