REPUBLIC OF TURKEY
YILDIZ TECHNICAL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

EXHAUST MANIFOLD THERMAL AND FLOW ANALYSIS

İPEK DUMAN

MSc. THESIS
DEPARTMENT OF MECHANICAL ENGINEERING
PROGRAM OF HEAT AND PROCESSING

ADVISER
ASSOC. PROF. ALP TEKİN ERGENÇ

İSTANBUL, 2016

REPUBLIC OF TURKEY

YILDIZ TECHNICAL UNIVERSITY

GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

EXHAUST MANIFOLD THERMAL AND FLOW ANALYSIS

A thesis submitted by İpek DUMAN in partial fulfillment of the requirements for the degree of **MASTER OF SCIENCE** is approved by the committee on 14.10.2016 in Department of Mechanical Engineering, Heat and Processing Program.

Thesis Adviser

Assoc. Prof. Alp Tekin ERGENÇ

Yıldız Technical University

Co- Adviser

Dr. Sinan EROĞLU

Ford Otosan

Approved By the Examining Committee

Assoc. Prof. Alptekin ERGENÇ

Yıldız Technical University ________________________

Prof. Cem SORUŞBAY

Istanbul Technical University ________________________

Dr. Sinan EROĞLU

Ford Otosan ________________________

Assist. Prof. Orkun ÖZENER

Yıldız Technical University ________________________

ACKNOWLEDGEMENTS

Firstly, I would like to express my sincere gratitude to my advisor Assoc. Prof. Alp Tekin ERGENÇ and my co-adviser Dr. Sinan EROĞLU for the continuous support of my master study and related research, for their patience, motivation, and immense knowledge. Their guidance helped me in all the time of research and writing of this thesis.

My sincere thanks also goes to Rıfat Kohen YANAROCAK, who provided me an opportunity to carry out thermal investigation of the Ecotorq 12.7 L engine exhaust manifold at Gölcük engine dynamometer laboratory, and who gave access to the research facilities.

Last but not the least, I would like to thank my family; my mother, Nilay DUMAN, and my father, Güngör DUMAN, for supporting me spiritually throughout writing this thesis and my life in general.

This master thesis includes the heavy duty diesel engine exhaust manifold thermal distribution analyses and verification with test results.

It is the time you have wasted for your rose that makes your rose so important.

October, 2016

İpek DUMAN

TABLE OF CONTENTS

Page

LIST OF SYMBOLS

T	Temperature
$\bar{T}$	Averaged temperature
$\overline{HTC}$	Averaged heat transfer coefficient
h_f	Fluid heat transfer coefficient
K_s	Solid thermal conductivity
q	Heat flux
y+	Y plus (Dimensionless wall distance)
ε	Emissivity

LIST OF ABBREVIATIONS

BC	Boundary Condition
CA	Crank Angle
CAE	Computer Aided Engineering
CFD	Computational Fluid Dynamic
CHT	Conjugate Heat Transfer
CPU	Central Processing Unit
EGR	Exhaust Gas Recirculation
FE	Finite Element
FEA	Finite Element Analysis
FSI	Fluid Structure Interaction
HD	Heavy Duty
HP	Horse Power
HTC	Heat Transfer Coefficient
RANS	Raynolds Averaged Navier Stokes
RPM	Revolutions per Minute
YTU	Yıldız Technical University

LIST OF FIGURES

LIST OF TABLES

ABSTRACT

EXHAUST MANIFOLD THERMAL AND FLOW ANALYSIS

İpek DUMAN

Department of Mechanical Engineering

MSc. Thesis

Adviser: Assoc. Prof. Alp Tekin ERGENÇ

Co-adviser: Dr. Sinan EROĞLU

Nowadays, reducing the fuel consumption, increasing the performance and efficiency of engine are the significant customer demands for the automotive industry. To meet the customer expectations and overcome this engineering challenge, new generation diesel engines are developed by using direct injection technology (common rail), turbocharger application considering the downsizing trend. Due to the downsizing, turbocharging, injection technology development and obligation of compliying with the emission regulations, engine out exhaust gas temperature reaches over 800 °C for diesel engine.

Through the engine operating condition, exhaust manifold is exposed high thermal stresses due to the elevated exhaust gas temperature. Exhaust manifold metal surface temperature increase very quickly and extreme transient temperature gradient can occur. Thermomechanical fatigue cracks are observed as a result of the cyclic manner thermal loading.

To come up with a robust and durable engine exhaust manifold design, it is required well developed CAE methodology to predict accurate transient temperature distribution on exhaust manifold. Then the predicted metal temperature is used to carry out the thermo-structure durability analysis evaluating the design of exhaust manifold.

In this study, investigated the feasibility of three different CAE approaches called conjugate heat transfer analysis, sequential coupling and co-simulation to assess the temperature field of heavy duty diesel engine exhaust manifold.

Apart from the numerical calculation, performed thermal camera measurement called thermal survey on the Ecotorq 12.7 L heavy duty diesel engine at Gölcük dynamometer laboratory to validate the CAE results.

Key words: Heavy duty diesel engine, exhaust manifold, temperature prediction, thermal assessment, computational fluid dynamics (CFD), conjugate heat transfer, thermal camera

YILDIZ TECHNICAL UNIVERSITY

GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

ÖZET

EGZOZ MANİFOLDUNDA TERMAL VE AKIŞ ANALİZİ

İpek DUMAN

Makine Mühendisliği Anabilim Dalı
Yüksek Lisans Tezi

Tez Danışmanı: Doç. Dr. Alp Tekin ERGENÇ
Eş Danışman: Dr. Sinan EROĞLU

Otomotiv endüstrisinde son zamanlarda, yüksek performans'a sahip, daha az yakıt tüketimi olan verimli motorların üretilmesi en büyük müşteri talepleri arasında yer almaktadır. Bütün bu müşteri beklentilerini karşılamak ve zorlu mühendislik probleminin üstesinden gelebilmek amacıyla yeni nesil dizel motorlarda common rail yakıt enjeksiyon ve turboşarj sistemleri kullanılmaya başlanmıştır, motorlarda hacim küçültme yoluna gidilmiştir. Gelişen yeni nesil motorlarda yukarıda bahsedilen teknolojilerin kullanılmaya başlanmasıyla birlikte ister istemez yanma sonucu açığa çıkan egzoz gaz sıcaklıkları da dizel motorlarda 800 °C'ye kadar ulaşmıştır. Egzoz emisyon değerlerinin (regülasyonlar) sağlanması da gaz sıcaklıklarının artmasında ayrıca bir rol oynamaktadır.

Egzoz manifoldu, motor çalışma koşullarınd yükselen egzoz gaz sıcaklıklarından dolayı şiddetli termal gerilmelere maruz kalmaktadır. Manifold metal yüzeyinde gerçekleşen ani sıcaklık artışı ile birlikte büyük sıcaklık gradyanları oluşmaktadır. Tekrarlı ısıl yüklemelerin gerçekleşmesi sonucunda ise termomekanik yorulma kaynaklı çatlaklar gözlemlenmektedir. Dayanımı yüksek ve gürbüz bir manifold tasarımı geliştirilmesi sürecinde yüzeydeki sıcaklık dağılımını doğru tahmin edecek analiz yöntemleri önem kazanmaktadır. Egzoz manifoldu üzerinden elde edilen sıcaklık dağılımları daha sonrasında ısıl dayanım analizlerinde kullanılmaktadır.

Bu çalışma kapsamında, dizel ağır ticari araç motorunun egzoz manifoldu üzerindeki sıcaklık dağılımının incelenmesi amacıyla 3 farklı nümerik analiz metodu, bileşik ısı transfer analizi, sıralı-bağlı analizler ve eş simulasyon, incelenmiştir.

Ayrıca Gölcük dinamometre laboratuvarlarında, Ecotorq 12.7 litre ağır ticari araç dizel motoru üzerinde termal kamera ile sıcaklık ölçümleri gerçekleştirilmiştir ve nümerik analizler ile validasyon sağlanmıştır.

Anahtar Kelimeler: Ağır ticari araç motoru, egzoz manifoldu, sıcaklık dağılımı, hesaplamalı akışkanlar dinamiği, bileşik ısı transferi, termal kamera

CHAPTER 1

INTRODUCTION

1.1 Literature Review

In the automotive engineering, exhaust manifold which is mounted on cylinder head of an engine, collects the exhaust gas from the cylinders and dispose it to the exhaust after treatment system. In the Figure 1.1, is shown the schematic view of exhaust manifold disassembled from engine head [1].

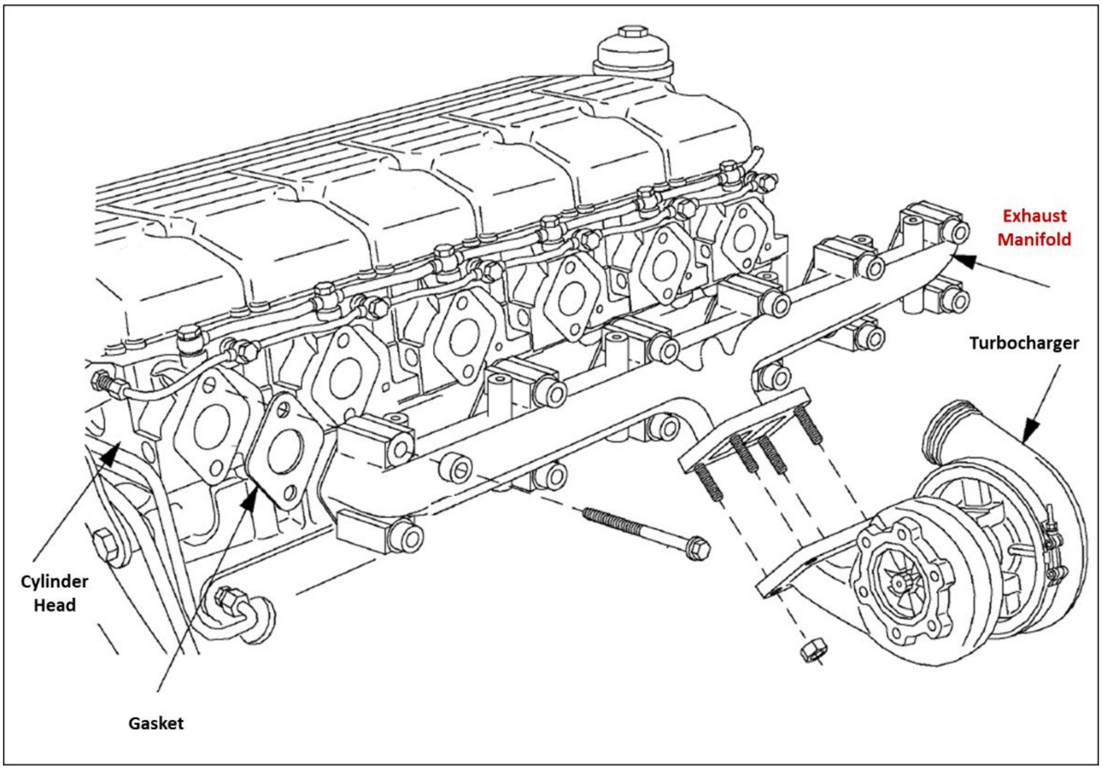


Figure 1.1 Engine Exhaust Manifold Schematic View [1]

The exhaust manifold plays significant role in engine performance. Especially, exhaust emission system efficiency and fuel consumption depend on manifold design [2].

Nowadays, increase of engine performance and overall system efficiency is the most struggling question seen in the automotive industry. In addition to this, safety, fuel economy and price are the demands from the customer. To satisfy all these market expectations regarding to the emission and performance of engine, the temperature of the exhaust gas disposed from the engine becomes too high [3]. During the engine operation, exhaust manifold is subjected to severe thermal cycling under harsh working conditions, resulting from high temperature exhaust gases, which also show transient behavior. So the automotive exhaust manifolds are designed and developed to supply minimum backpressure and turbulence and must be able to withstand extreme heating under high temperature and cooling down under low temperature.

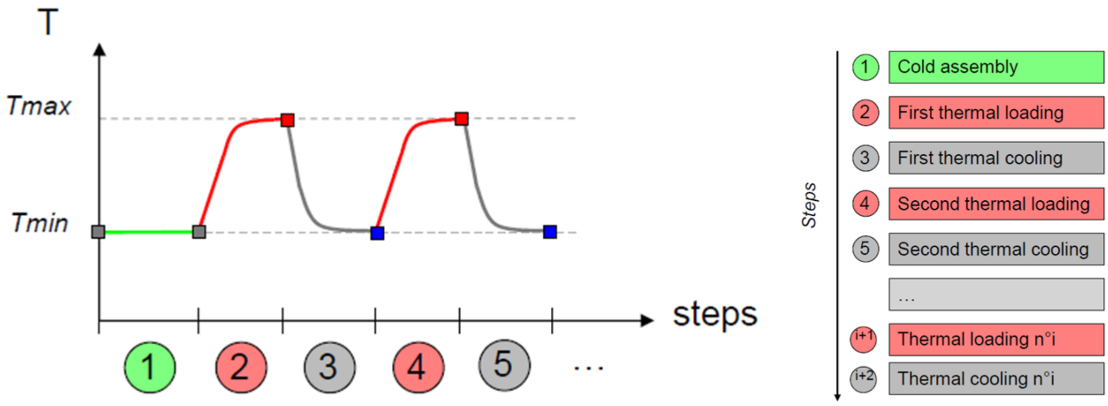


Figure 1.2 Engine Cyclic Thermal Loading [6]

The exhaust manifold metal temperature can rapidly increase from ambient temperature up to about 900 °C in a short time. Since the manifold is only cooled through the conduction with the cylinder head via thermal contact and natural convection with its surroundings, and radiation, it produces large transient thermal gradients [4]. The larger temperature variation causes larger thermal stress on the structure. Exhaust manifold cracks from Thermomechanical Fatigue (TMF) are often seen on highly loaded engines. Due to cyclic thermal loads, thermo-mechanical fatigue cracks on exhaust manifold and gasket sealing failure are often observed during engine durability tests, such as thermal shock test [5].

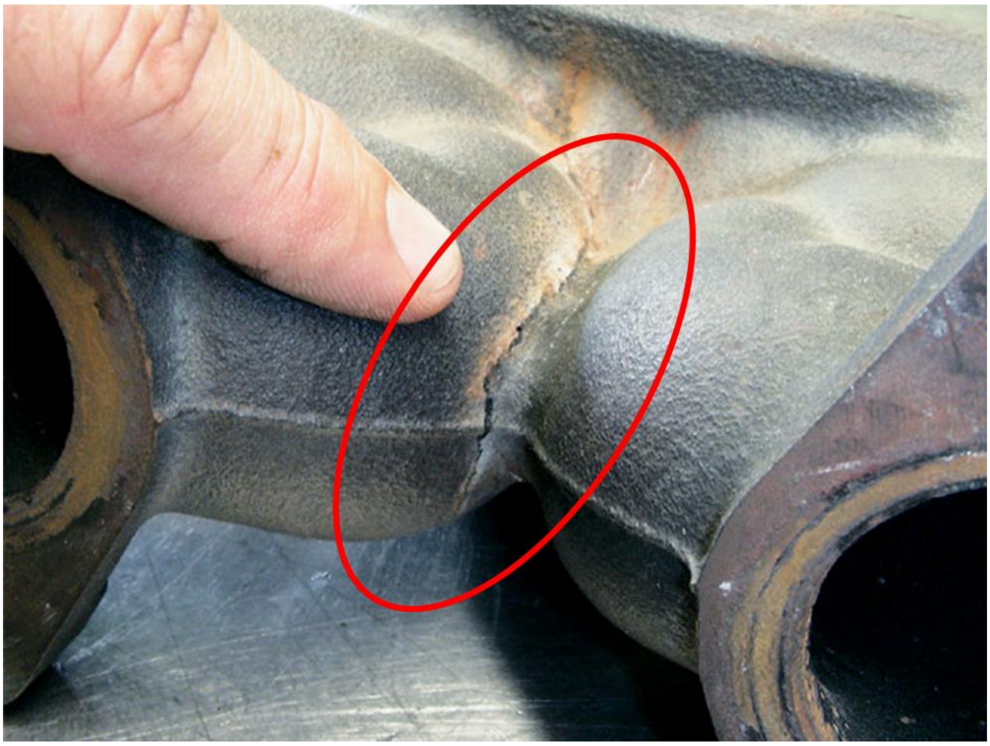


Figure 1.3 Thermal Crack on Exhaust Manifold

There are three different failure mechanisms of thermo-mechanical fatigue in cast iron exhaust manifold related with the high temperature levels. First one is the creep which is the flow of materials at high temperatures. Fatigue is crack growth and propagation due to repeated loading. Oxidation is a change in the chemical composition of the material due to environmental factors.The oxidized material is more prone to crack generation. [6]

Therefore, to come up with a robust and durable design is a challenging task. In early stages of the design process, advanced and robust CAE methods are required for the durable design of exhaust manifolds.

The use of advanced simulation technologies enables the design and analyses engineers to identify critical locations in an early phase of development and to meet measures in order to remove local structural weaknesses [7]. This minimizes the need for expensive hardware testing, thus reducing the overall product development cycle time and cost.

As a scope of this thesis, performed literature survey to find out related studies providing boundary conditions (temperature field) for thermal fatigue analysis.

Zhi-En Liu and Xue-Ni Li (2014), carried out transient fluid-solid thermal coupling simulation by using the serial coupling method. Bi-directional fluid-solid coupling simulation between CFD software Star CCM+ and FEA software Abaqus is achieved and transient temperature and thermal stress distribution are obtained. Also compared the thermal stress difference between the transient and steady state calculation. Results show that the difference of the temperature distribution is large and this effect the position of maximum stress point on manifold. This study demonstrated that transient analysis method is more effective to predict thermal fatique life of exhaust manifold. [8]

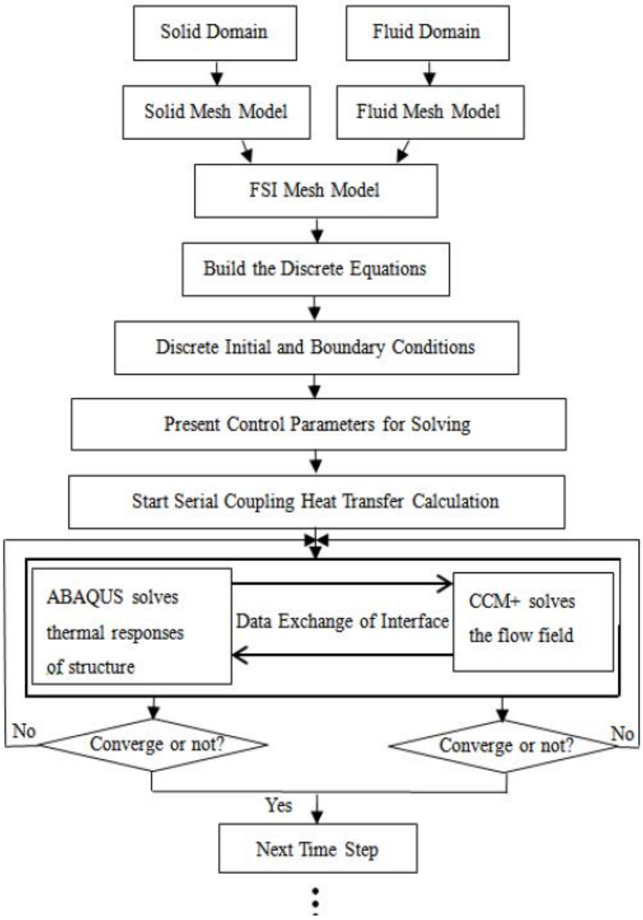


Figure 1.4 Serial Coupling of Star CCM+ and Abaqus Flowchart [8]

Mamiya, Masuda and Noda (2002) developed CAE approach involving computation fluid dynamic (CFD) and finite element (FE) to simulate the temperature distribution in the early stage of design and estimate the thermal fatigue life of engine exhaust manifold. Considered the internal and external flow field to simulate the temperature distribution under real operating conditions. In house cfd code was used for the external air-flow simulation to predict heat transfer coefficient between air and manifold wall at various vehicle speed. Performed unsteady gas flow calculation using Star CD tool to predict the heat transfer coefficient between exhaust gas and manifold wall under various operating conditions. The predicted heat transfer coefficients were used in thermal FE analysis to predict temperature field. The boundary conditions of the transient gas dynamic analysis was derived by the engine cycle simulation on GT Power as inlet velocity and outlet pressure. [9]

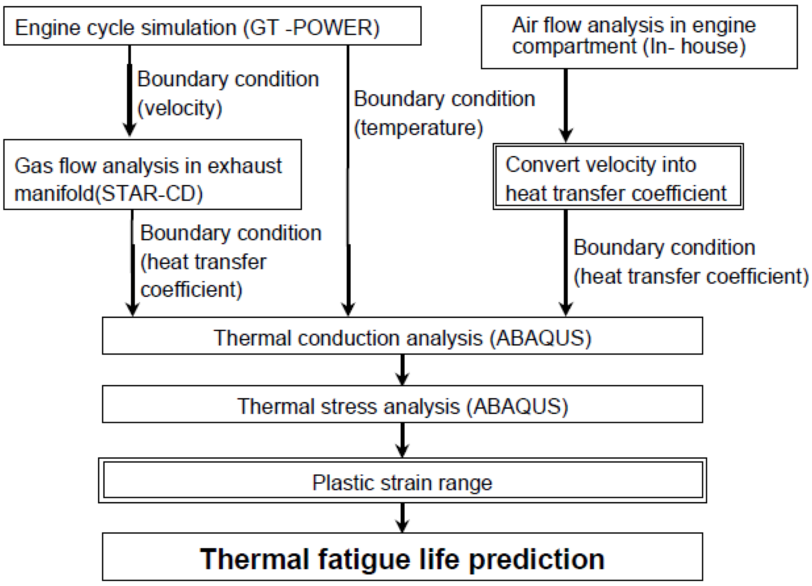


Figure 1.5 The Procedure of Thermal Analysis [9]

Fan, Kuba, Nakanishi (2004), investigated three different approaches to couple a stress analysis system and a FVM CFD system to analyze a FEM thermal stress field. In the first method, the grid and temperature distribution of CFD were used directly in thermal stress analysis taking the whole advantage of mesh compatibility. In this approach hybrid mesh is limited to use. Flow and temperature field is solved using CFD software and resulting film coefficient between the solid-fluid and ambient temperature was interpolated to the corresponding mesh of solid surfaces generated for thermal stress analysis regarding to the second method. The calculations were divided into the 3 stages, thermal flow analysis for fluid side, thermal analysis in solid part for temperature field and stress analysis in solid part. With this method, fluid and solid domain calculations are performed separately. In the last method, both flow and temperature field was solved using CFD software and predicted metal temperature field is interpolated to the nodes of finite element based model for thermal structure analysis. According to this study, third approach (volume mapping) is seen the easiest and most accurate method compared with other two methods. [10]

1.2 Objective of the Thesis

Temperature distribution of the manifold is the most significant boundary condition to drive the thermomechanical fatigue analysis which predict the life of the component. Therefore, to perform the thermal stress analysis accurately, well predicted thermal distribution should be provided. The main objective of this thesis is development of robust CAE approach to obtain the temperature distribution on solid structure of exhaust manifold called thermal analysis. Three different CAE approaches are investigated within the scope of the thermal analysis. Apart from the numerical calculations, performed experimental study with thermal camera to validate the results.

1.3 Hypothesis

Accurate temperature distribution of exhaust manifold plays an important role at thermomechanical fatigue life prediction. Within this thesis study, explained the three different CAE methodology called sequential coupling, co-simulation and conjugate heat transfer analysis respectively. Considering the product development process, conjugate heat transfer analysis come up robust temperature prediction method, on the other hand it requires more computational run time. Similar level of accuracy could be achieved at Star CCM+ Co-Simulation analysis with reduced computational run time.

CHAPTER 2

THERMAL CAE MODELLING

The ability to accurately predict metal temperature of exhaust manifold is very important for robust design of the exhaust manifold. In view of this, the first step in the manifold development is the accurate metal temperature prediction of the manifold. The main challenge arises due to the multi-physics nature of the problem where there is a strong interaction between the fluid and solid domains. The solution requires the coupling of fluid and solid domains. The presence of pulsating flow behavior within the exhaust manifold caused by the sequential firing of each cylinder makes the CAE solution more difficult and requires the analyses to be run as transient.

Conjugate heat transfer simulations are commonly used method to model multiphysics problem as exhaust manifold. In the conjugate heat transfer approach, solid and fluid domains are modelled and solved in a single instance CFD analysis. As it is known, using conjugate method for modelling the multi-physics problem like exhaust manifold is the most accurate approach. In the conjugate approach, both solid and fluid domains are modelled together in CFD environment and solved together. Conjugate approach solves the temperature field of the structure simultaneously with the fluid flow. However, the pulsating flow behavior within the manifold makes the user apply very small time steps for the flow solver to converge, between 1.0e-5 second and 1.0e-4 second. With this time step, modelling the entire heat-up period of manifold which is 600 second using conjugate approach is not feasible in terms of computational run time, which takes over 5 months to run with 48 CPUs. So alternative methods are investigated to obtain the temperature distribution of the exhaust manifold. In the following sections, these thermal modelling approaches will be explained in detail.

2.1 Method 1- Sequential Coupling Approach

This method is the most widely approach used in the industry called sequential coupling method (decoupled) which requires the coupling of CFD and FE solver. In the sequential coupling approach, the thermal analysis of the manifold is performed by employing an iterative process between CFD and FE analysis. It requires a coupling between these two solvers in fluid and solid domains are calculated separately. Also there should be a proper data exchange process on fluid-solid interface between CFD and FE models through this method.

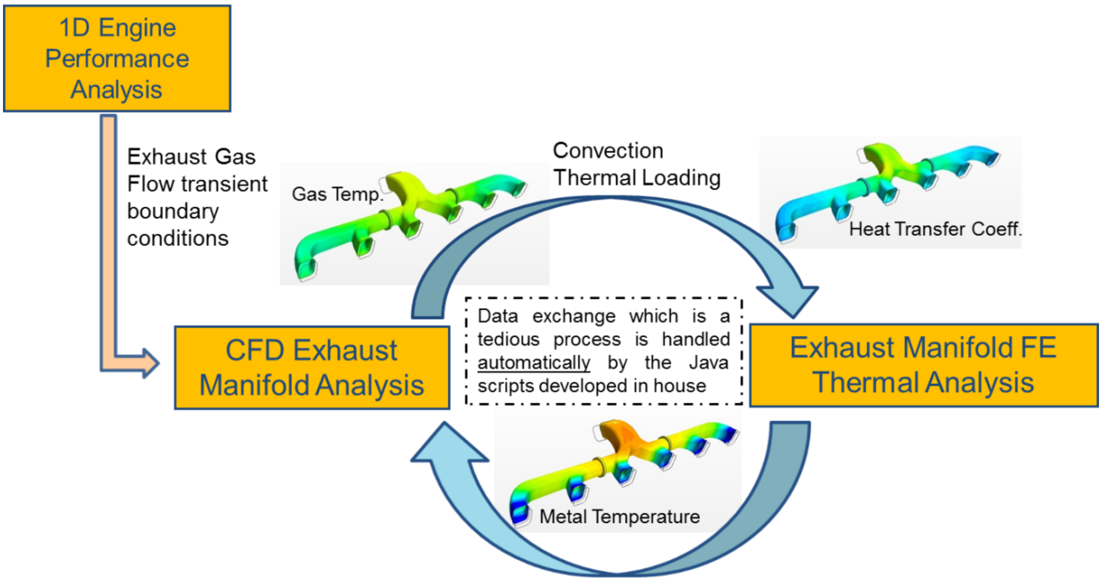


Figure 2.1 Sequential Coupling Approach Schematic Diagram

The aim of the 3D CFD simulation of the exhaust manifold is to examine the flow behavior and provide the time averaged heat transfer coefficient (HTC) and associated film temperature on the fluid solid interface boundary to the FE thermal analysis. Due to the presence of pulsating flow behavior within the exhaust manifold during the engine operation, the CFD analyses are performed as transient to come up with more realistic solutions. The iterative loop starts with uniform wall temperature assumption on the fluid domain side and CFD analysis is run for 3 engine cycles and then time averaged results which are heat transfer coefficient and fluid temperature in the last cycle are mapped onto fluid interface of solid mesh. To automatize the data exchange, which is a tedious task, process between FE and CFD solvers, in house developed Java scripts are used. After running the solid thermal analysis in FE solver, the temperature distribution outputs obtained on the fluid interface of the structure is mapped onto the wall of fluid domain (CFD model) and CFD analysis is rerun. After each going back and forth between CFD and FE analyses, the total heat transfer rate on the fluid interface calculated from each analyses are monitored and this iterative approach continues until heat transfer rate on the interface convergence is achieved.

2.1.1 CFD Modelling

- Since the exhaust gas flow characteristic is heavily varying with time inside the manifold through the engine operating cycle, carried out the CFD analysis which in transient mode.
- In the CFD domain, as the boundary conditions; at the runner inlets, mass flow inlet boundary condition type and at the outlet to the turbo and EGR (exhaust gas recirculation) boundaries pressure boundary condition type were defined as shown in Figure 2.2.

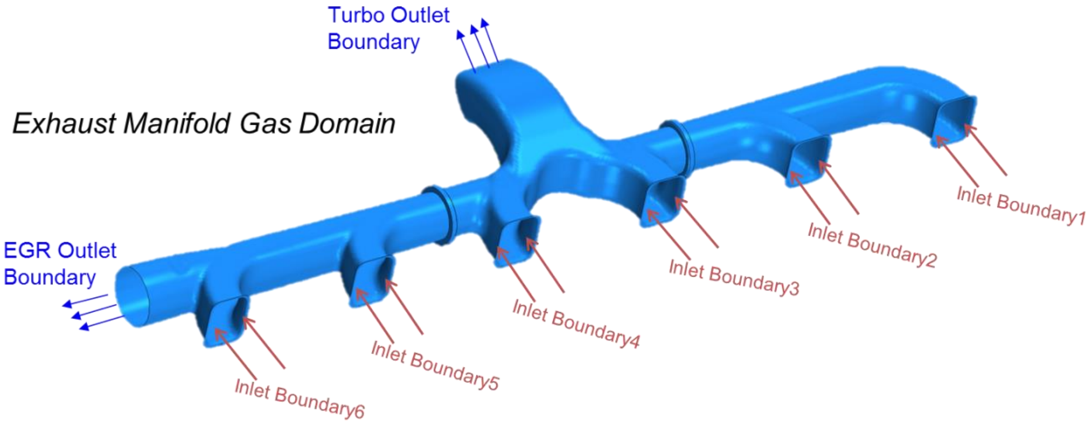


Figure 2.2 Exhaust Manifold CFD Domain

- Assumed the gas flow as compressible taking account of significant changes in the density.
- Selected the k-epsilon turbulence model which RANS based turbulence model with high y+ wall treatment. High y+ wall treatment assumes that the near-wall cell lies within the logarithmic region of the boundary layer.
- CFD model used in the calculation consists of ~290.000 trimmer mesh. The resulting mesh is composed predominantly of hexahedral cells with trimmed cells next to the surface. Produced flow aligned grids in the exhaust manifold gas domain with using trim mesher.

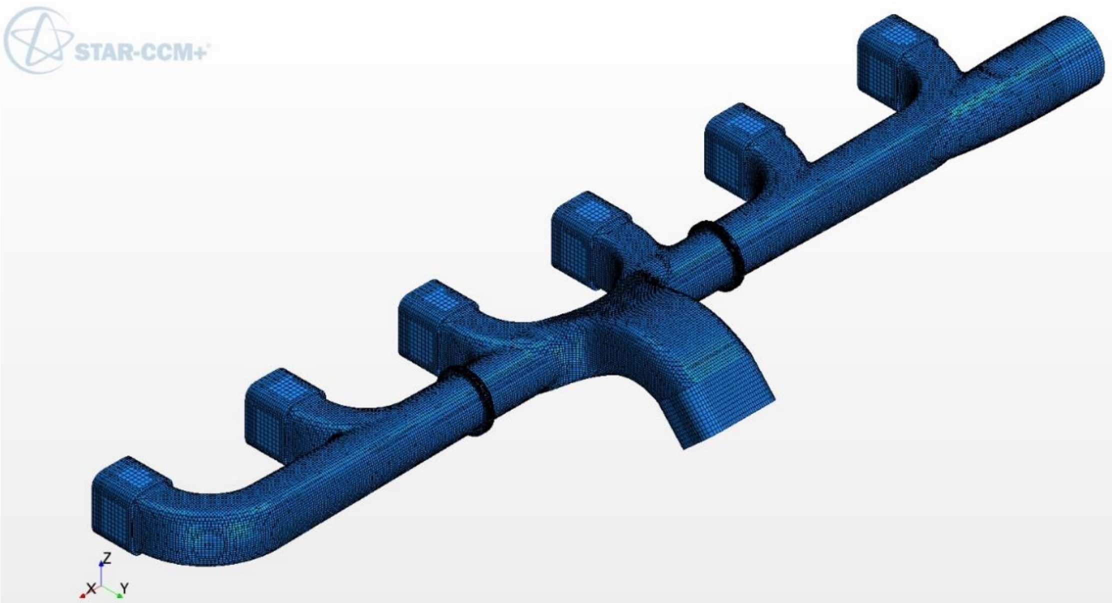


Figure 2.3 Fluid CFD Domain Trimmer Mesh

The boundary conditions required for 3D CFD analysis of exhaust manifold is obtained from the 1D CFD engine performance analyses over complete engine cycle as a function of time. The time dependent boundary conditions are provided from the GT Power software via help of the engine performance team at Ford Otosan.

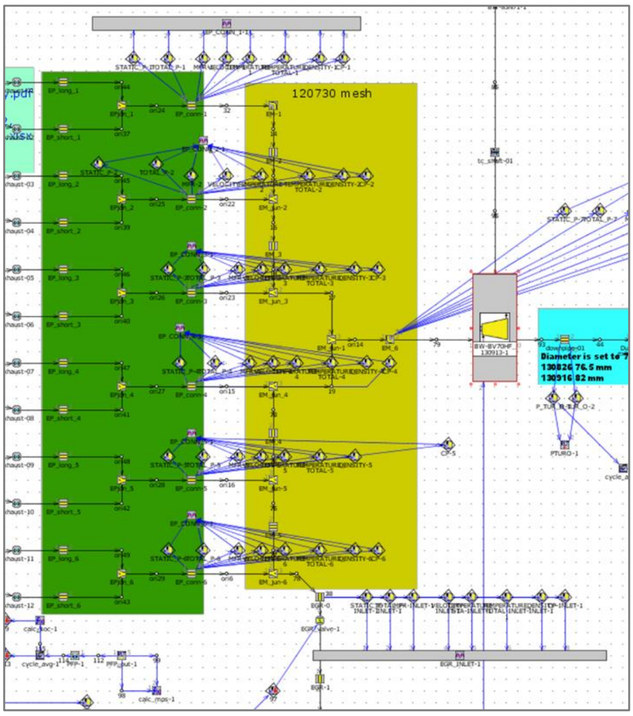


Figure 2.4 Engine Performance Model at GT Power

The crank angle resolved boundary conditions including exhaust gas mass flow rate, temperature and pressure at each runner and outlet under full load condition shown as the graphs below.

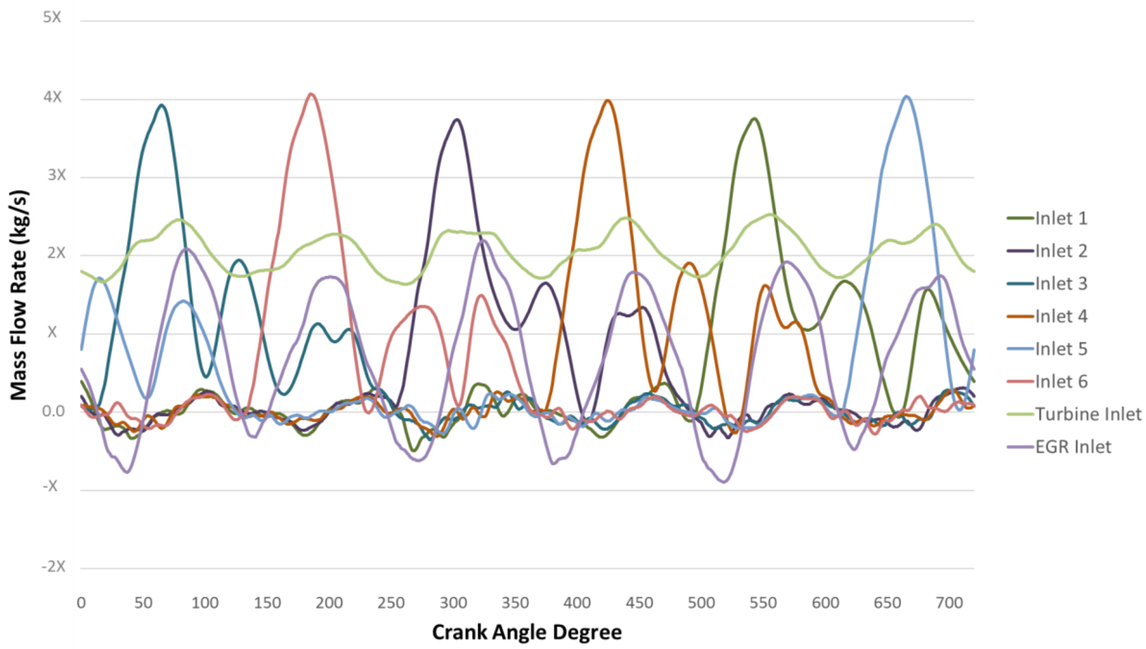


Figure 2.5 Mass Flow Rate BC obtained from 1D CFD Analysis

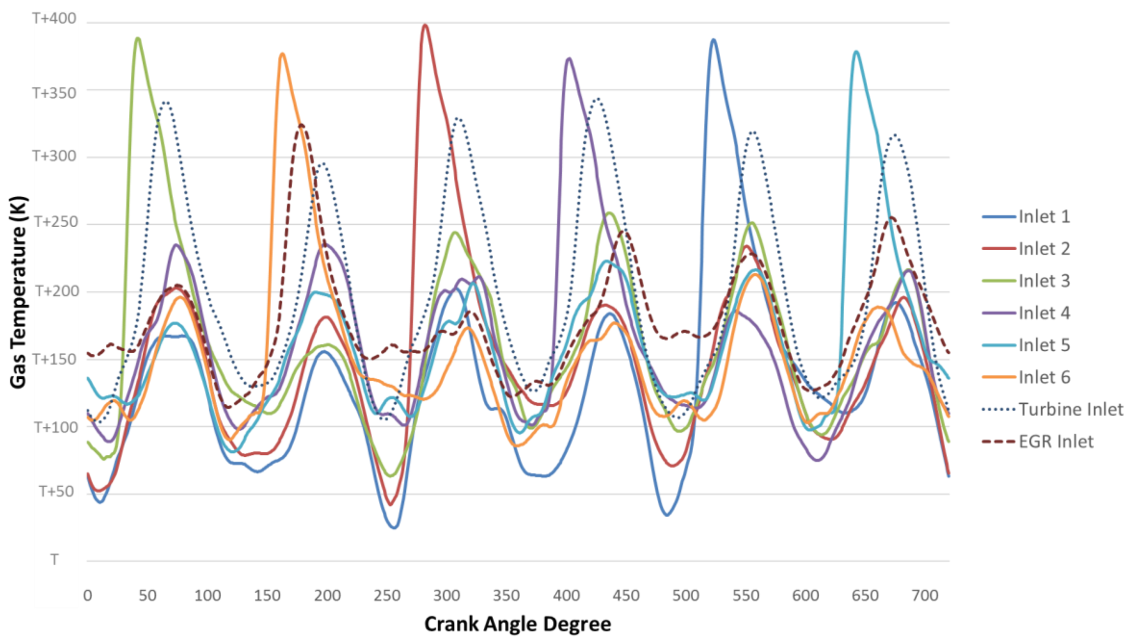


Figure 2.6 Exhaust Gas Temperature BC obtained from 1D CFD Analysis

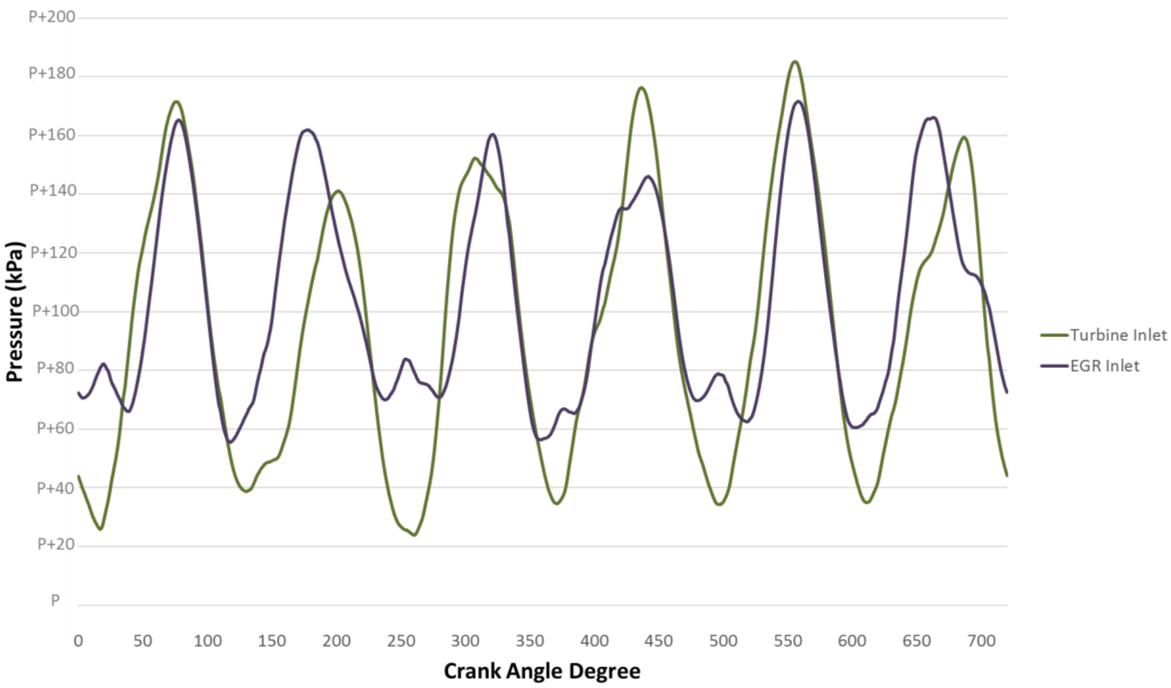


Figure 2.7 Pressure BC obtained from 1D CFD Analysis

2.1.2 Solid Structure Modelling

On the finite element model, thermal model of solid structure consists of the exhaust manifold, gasket, bolts, studs and representative head which is modelled on the Abaqus for the heat transfer calculation.

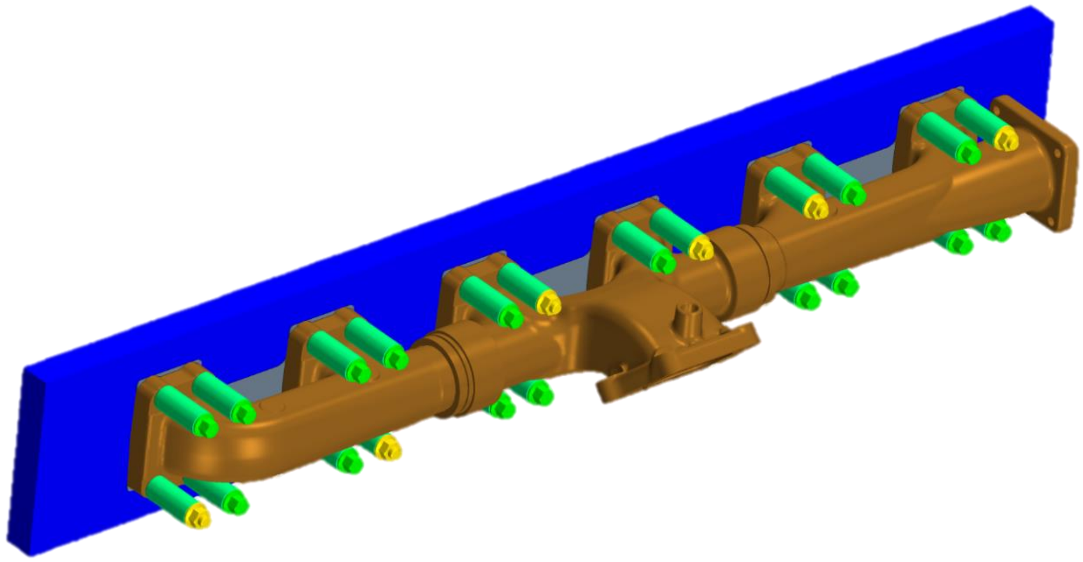


Figure 2.8 Solid Structure Model

Abaqus has solved this heat transfer problem involving conduction, forced convection, and radiation in transient mode.

2.1.3 Data Mapping and Time Averaging

Initially the transient CFD analysis is performed by assuming uniform wall temperature or uniform convection loading on the wall boundaries of the fluid exhaust gas domain. Then time averaged, non-uniform surface heat transfer coefficient (HTC) and fluid temperature distribution obtained from 3D CFD analysis are mapped onto the fluid interface of the exhaust manifold structure using the macros developed in Otosan which is explained in the following paragraphs. After running the FE thermal analysis, the temperature distribution results obtained on the fluid interface of the structure is mapped back onto the wall of fluid domain (CFD model) and CFD analysis is rerun. This process is repeated until the heat transfer at the interface converges.

In house developed java scripts carry out the following below;

- Time averaging of convection thermal loading,
- Mapping the related data from one to another,
- Generating the contour plots necessary for the post-processing without any need of user interference.

The key point of coupling for thermal analysis is how to use the flow field data obtained from CFD analyses stage as input data for FE thermal analysis. The CFD model is run for 3 engine cycles (2160 Crank Angle) each engine cycle represents 2 crank revolutions, 720 CA. Only the last engine cycle data is time averaged and provided as an input data of FE analysis. The reason why time averaged HTC distribution is mapped onto FE thermal model is that the instantaneous results lead to misleading and inaccurate solid temperature prediction. As seen from the Figure 2.5, inlet mass flow rate distribution in one engine cycle, cylinder 5 which the last fired cylinder effects on instantaneous HTC distribution.

On the Figure 2.9, the instantenous and time averaged htc distribution is shown to reveal the difference.

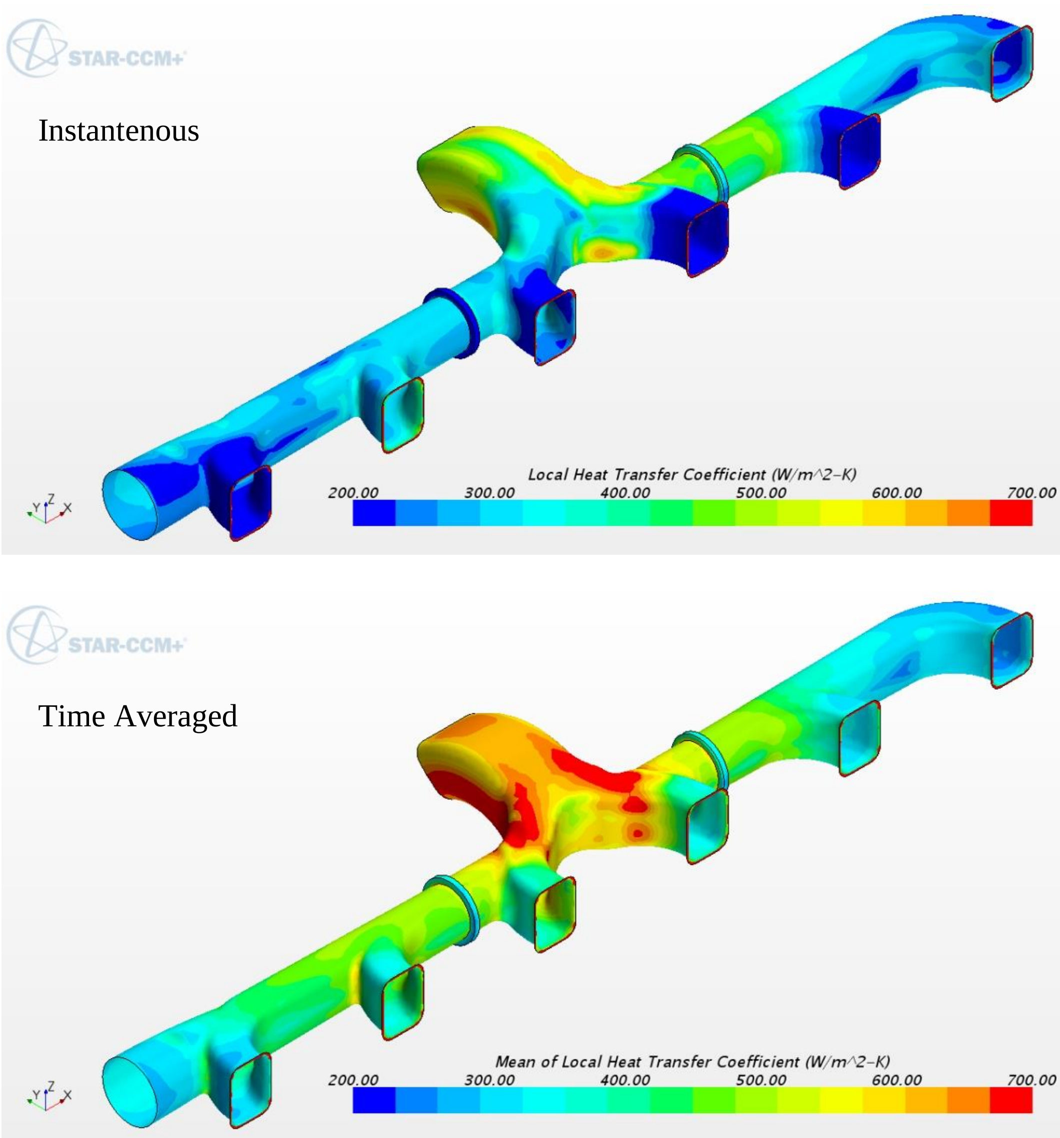


Figure 2.9 Instantaneous and Time Averaged HTC Distribution at the End of 3rd Cycle

Standard averaging method is used over 720˚CA for Heat Transfer Coefficient (HTC) distribution necessary for FE thermal analysis.

$$\overline{HTC_i} = \frac{\sum_{i=1}^{720} \overline{HTC_i}}{720} \tag{2.1}$$

For the gas side reference temperature distribution which will be provided to FE thermal analysis; weighted averaging method is used. The effect of using standard averaging versus weighted averaging for the gas side HTC reference temperature on the interface convection thermal loading was assessed and shown leading to significant differences in the results.

$$\bar{T}_i = \frac{\sum_{i=1}^{720} T_i * HTC_i}{720 * \overline{HTC_i}} \tag{2.2}$$

2.1.4 Results of Sequential Coupling Method

In this study, 5 iterations between CFD and FE solvers were carried out which took almost 2 days with 16 CPUs for CFD solver and 32 CPUs for FE solver. The change of time averaged local HTC and weighted averaged reference gas temperature distribution with iterations between the CFD and FE solver on figures below.

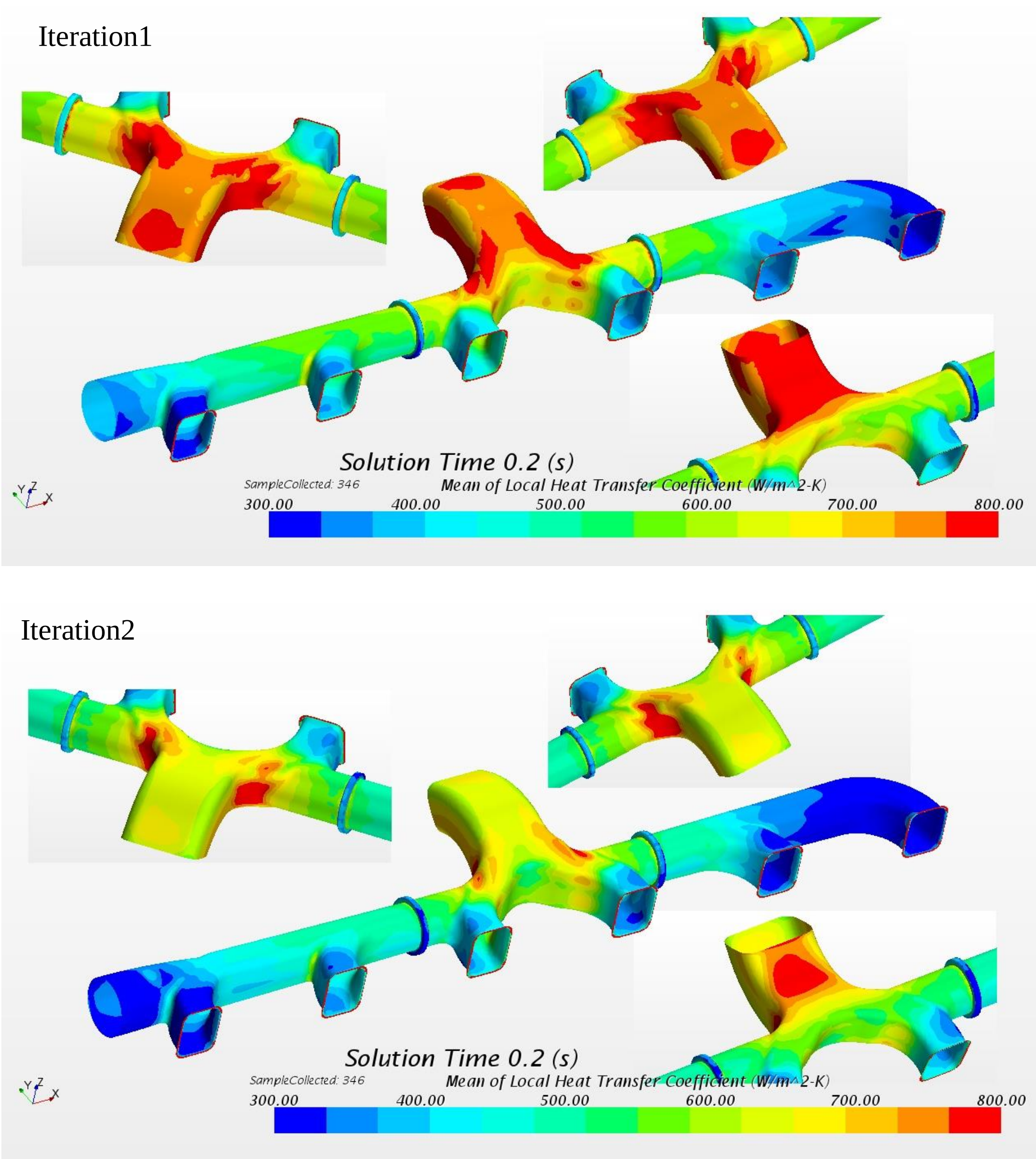
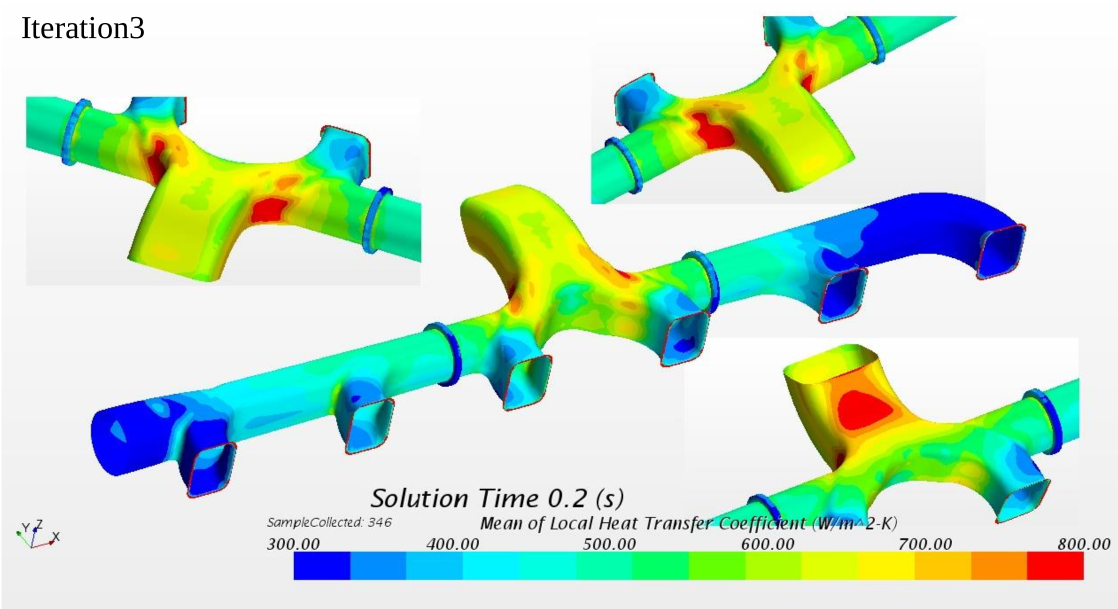


Figure 2.10 Local HTC Distribution between CFD and FE Solver



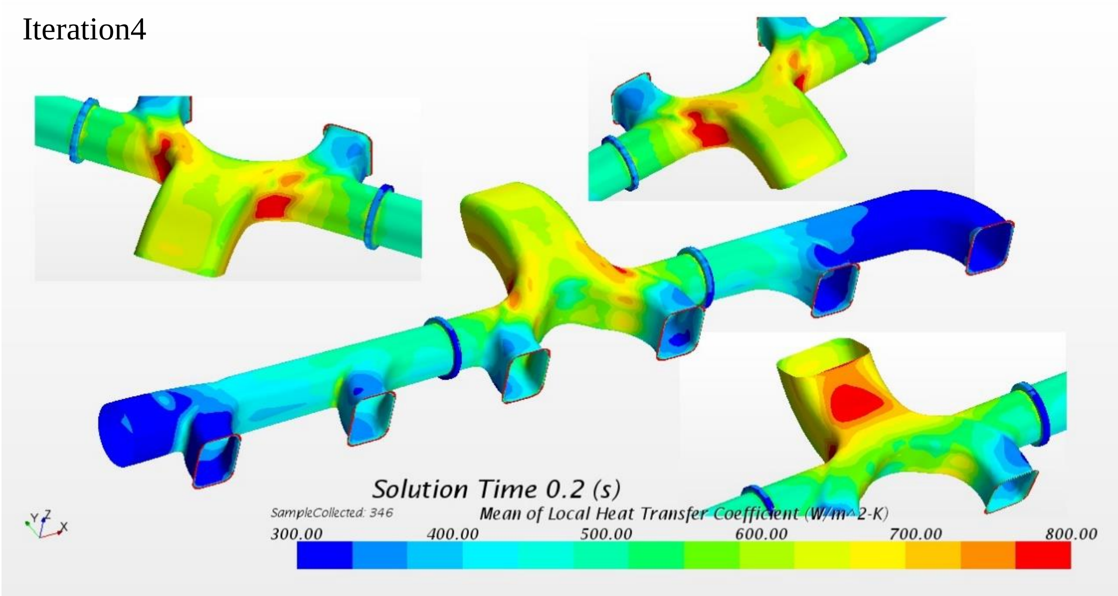


Figure 2.10 (Cont’d)

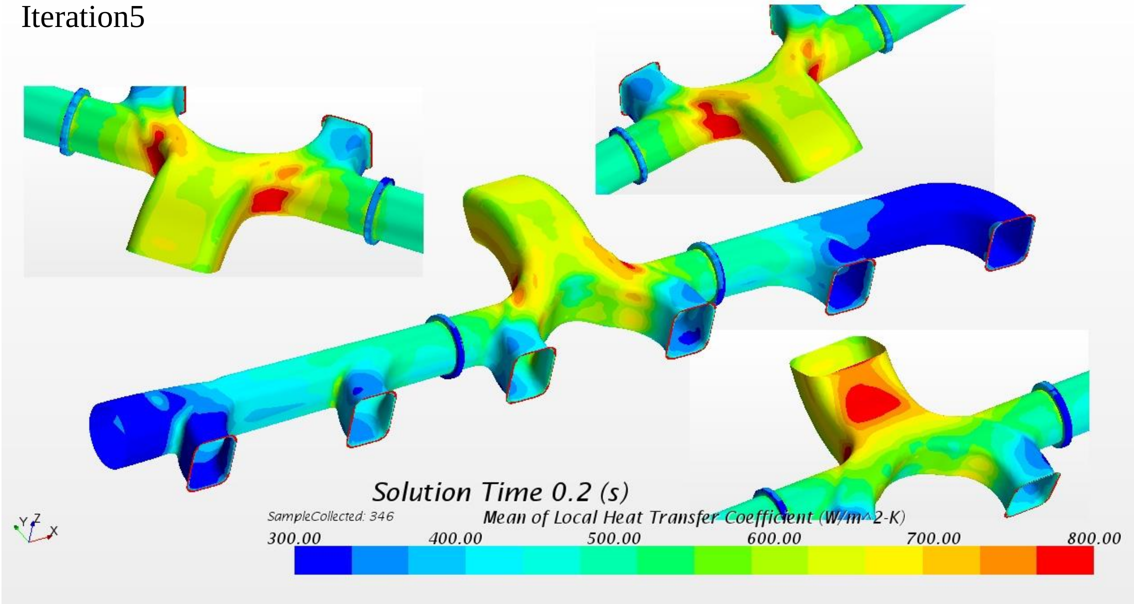


Figure 2.10 (Cont'd)

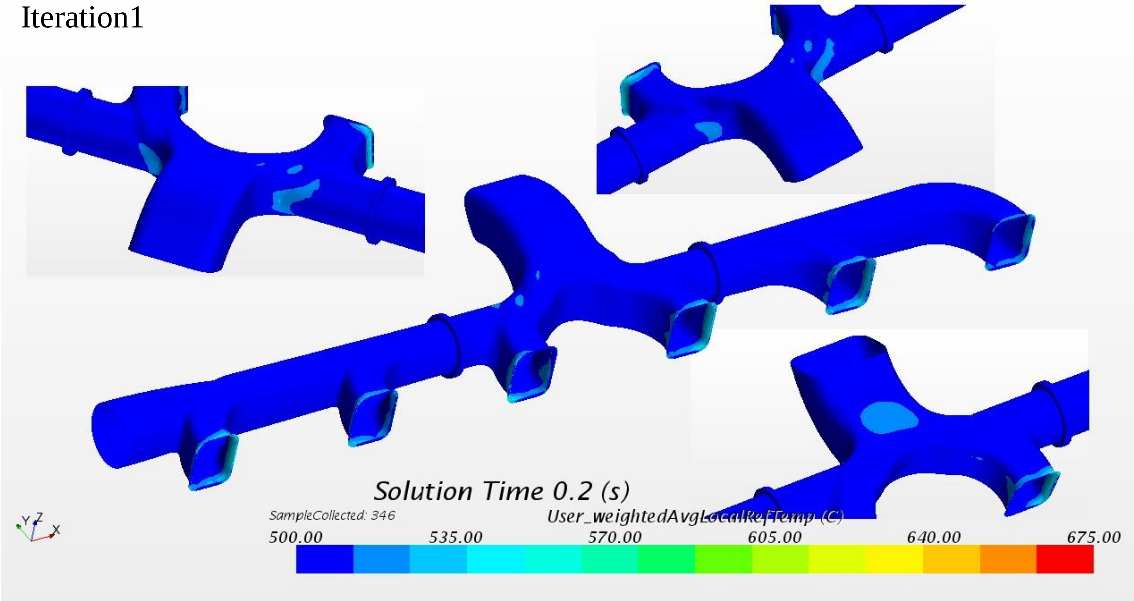
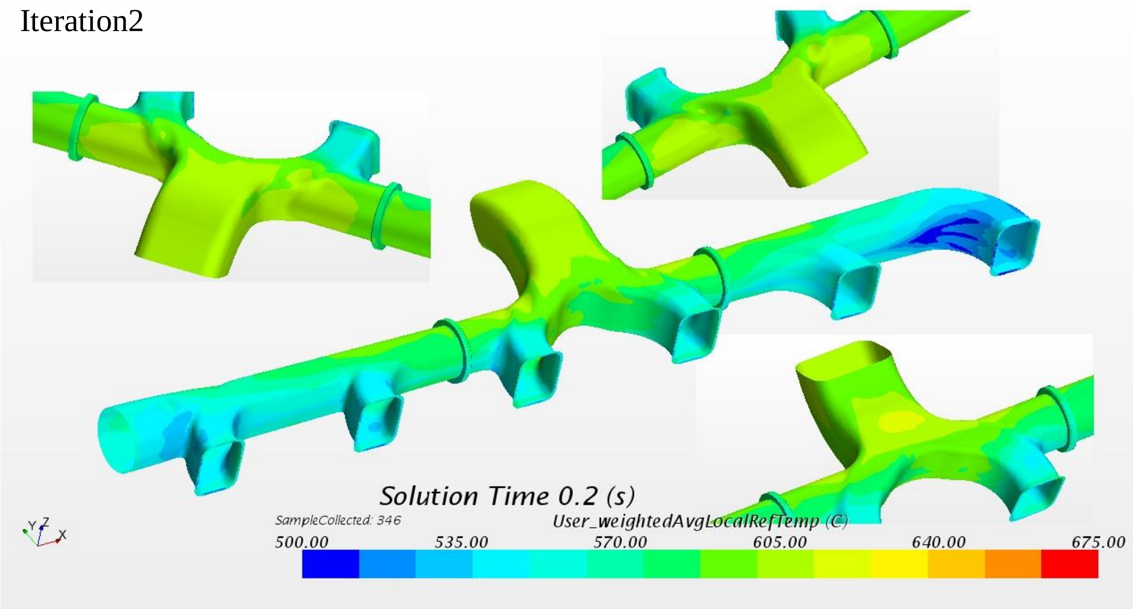


Figure 2.11 Weighted Averaged Reference Temperature Distribution



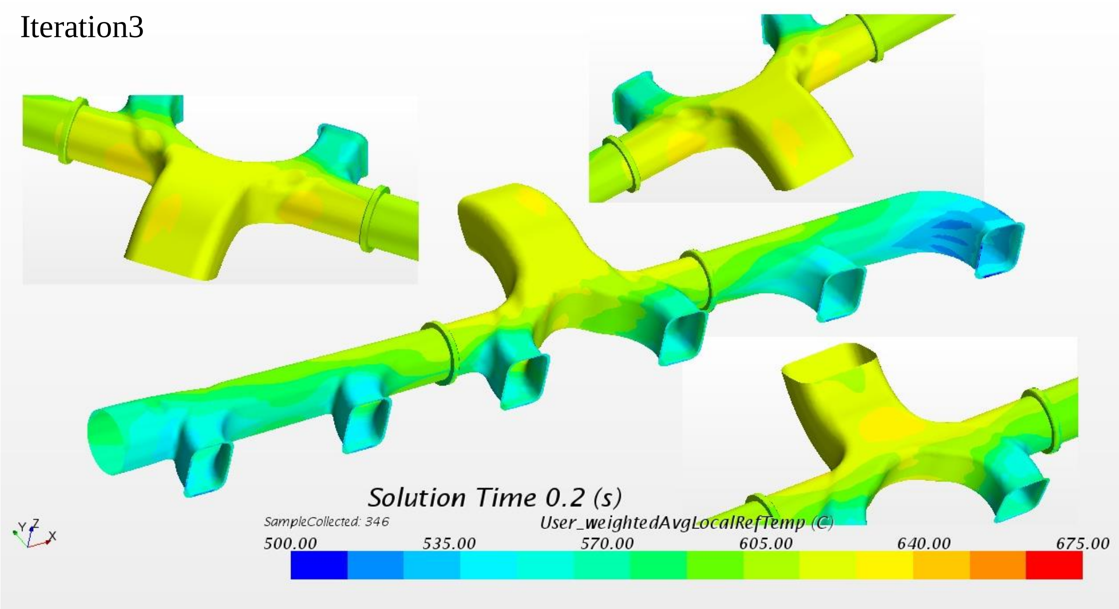
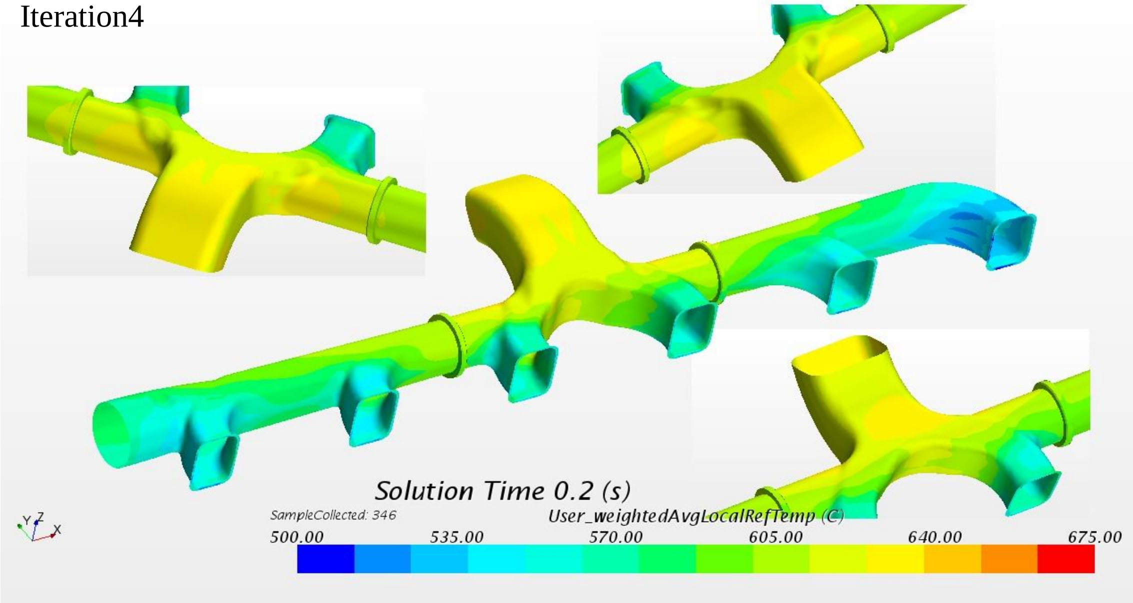


Figure 2.11 (Cont'd)



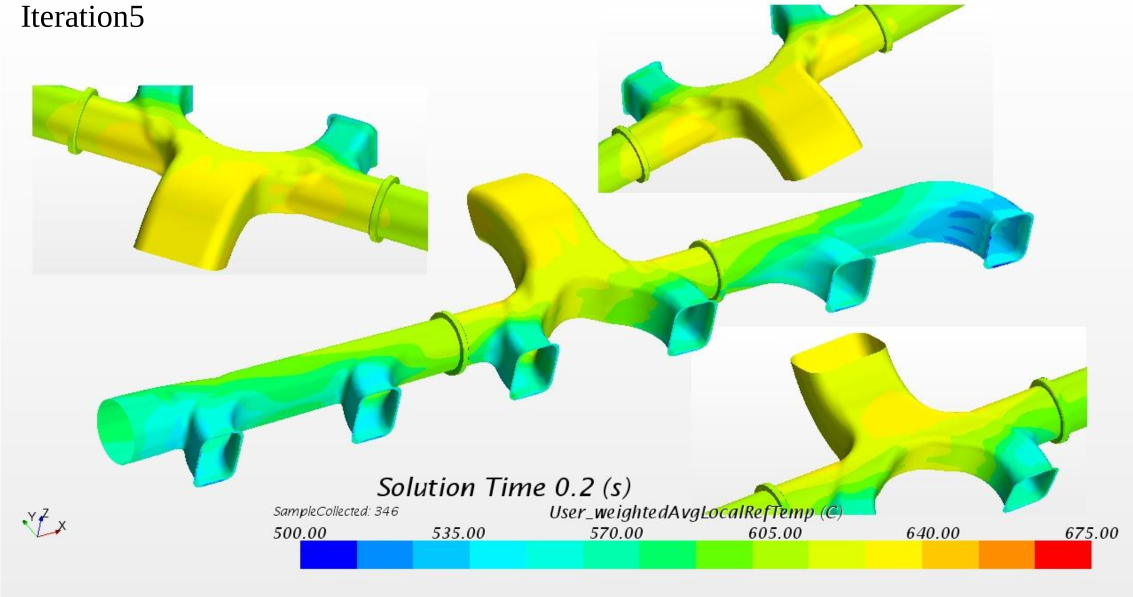


Figure 2.11 (Cont'd)

As it is seen from the Figure 2.10 and Figure 2.11, the HTC distribution changes slightly after the second iteration, however HTC fluid reference temperature changes with iterations since they are heavily affected by the wall temperature distribution. The mean of local reference gas temperature is the temperature of the cell (grid) next to the wall boundary. Heat transfer rate and surface averaged temperature on interface monitored at each iteration to check the convergence are plotted on the graph.

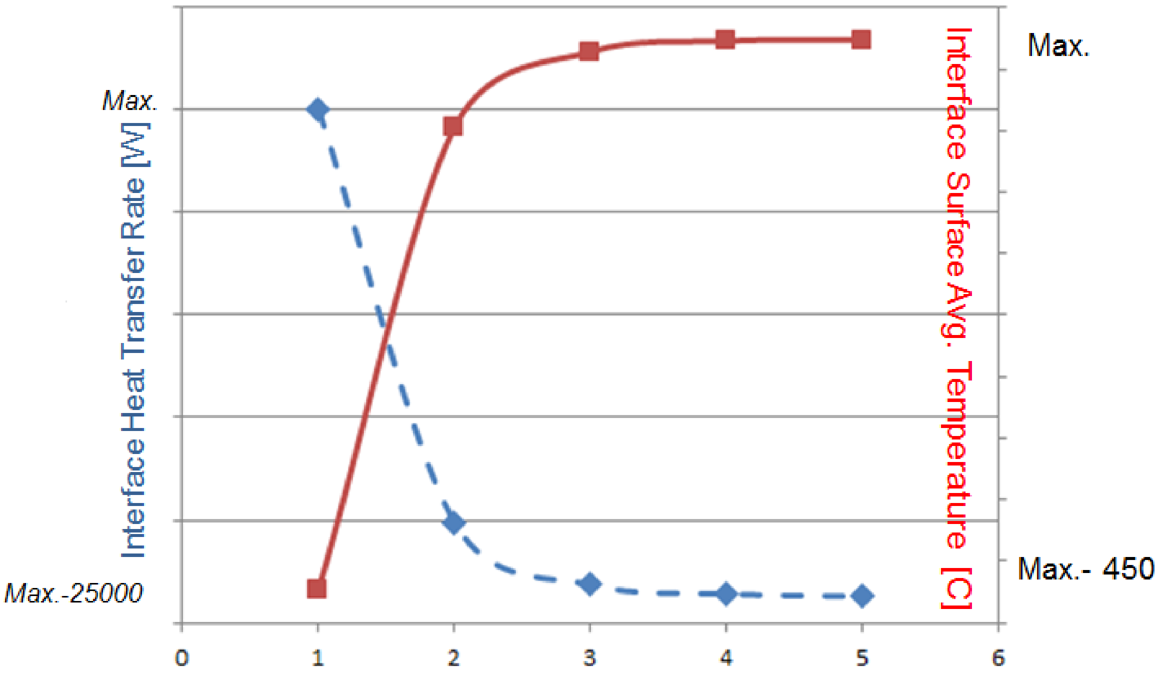


Figure 2.12 Heat Transfer Rate & Surface Averaged Temperature Change with Iteration

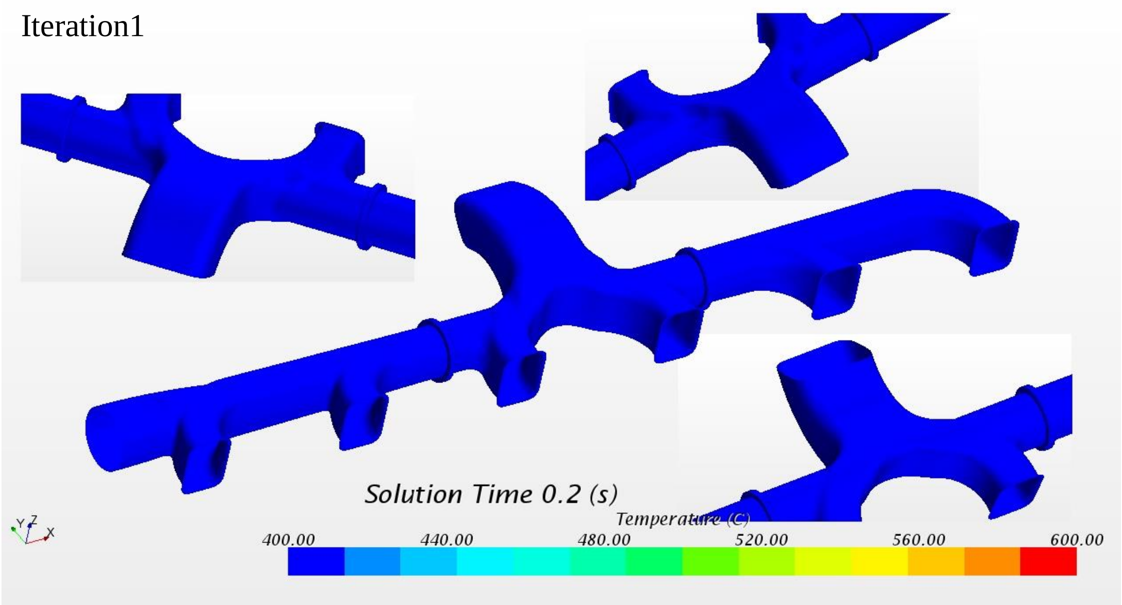
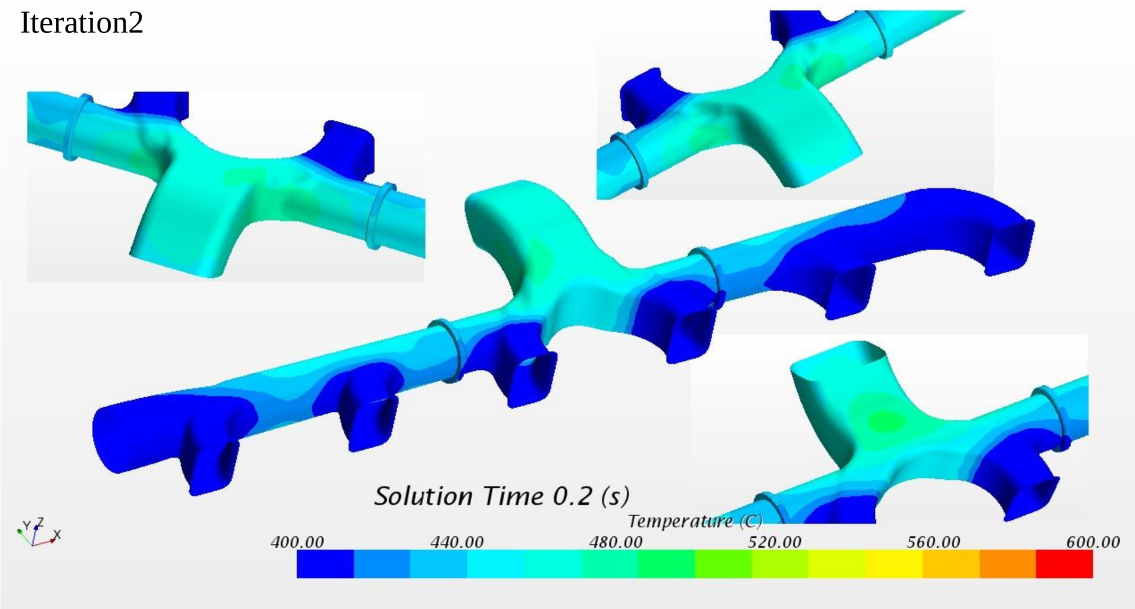


Figure 2.13 Temperature Distribution on Interface at each Iteration



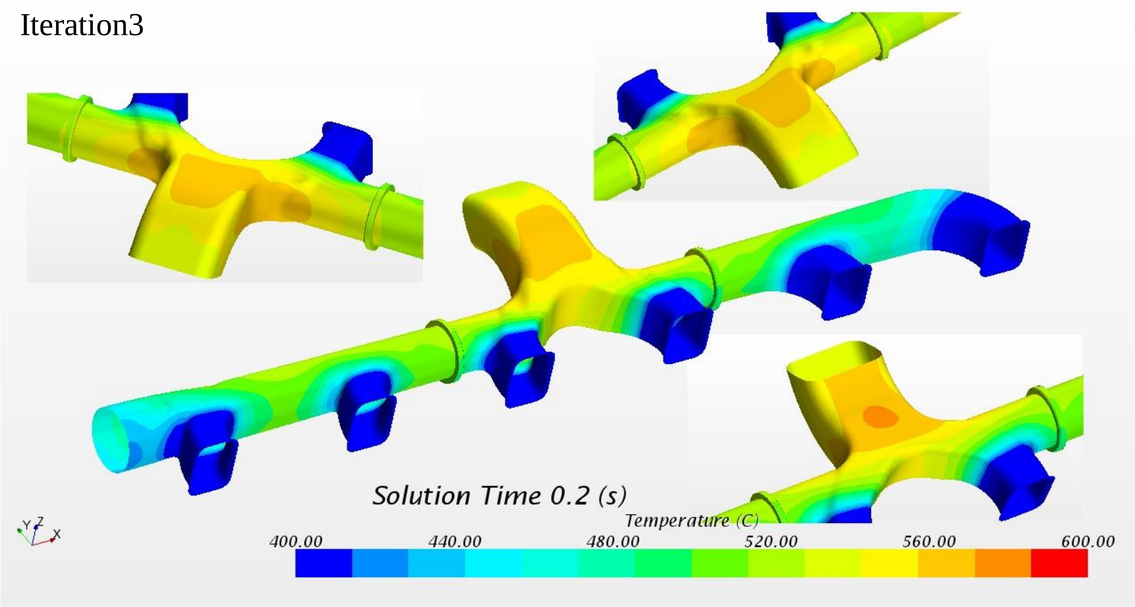
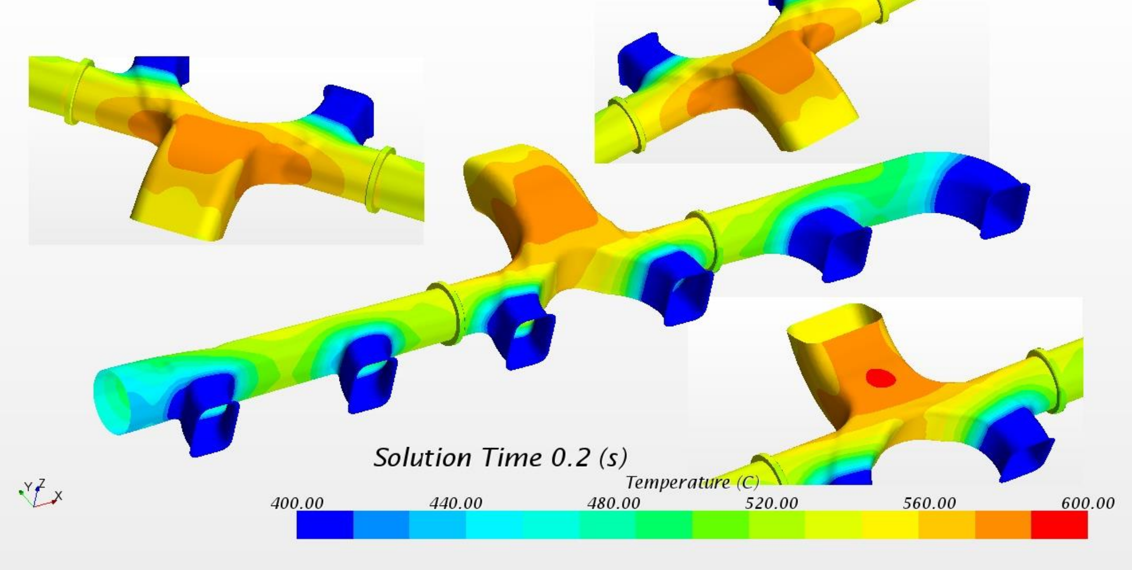


Figure 2.13 (Cont'd)

Iteration4



Iteration5

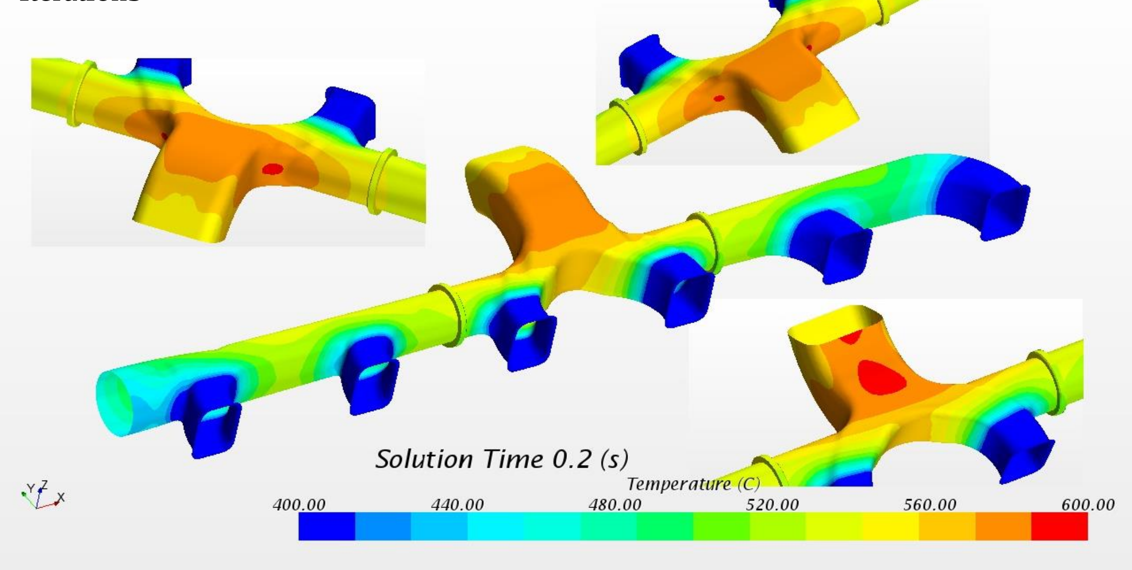


Figure 2.13 (Cont'd)

At peak power condition, the interior wall of the exhaust manifold is exposed to exhaust gas with temperature up to 750 °C seen on the Figure2.6.

Temperature results show that at least 3 iterations (going back & forth between CFD Solver & FE Solver) need to be done to achieve convergence in terms of temperature distribution.

Figure 2.14 shows the temperature distribution of the manifold structure obtained from thermal analysis at the end of the warm up period. The EGR exit from one side of manifold leads to slightly unsymmetrical temperature distribution between the two banks of exhaust manifold. Highest temperature levels are observed around the turbo entrance area (neck region) which is the most thermally critical region.

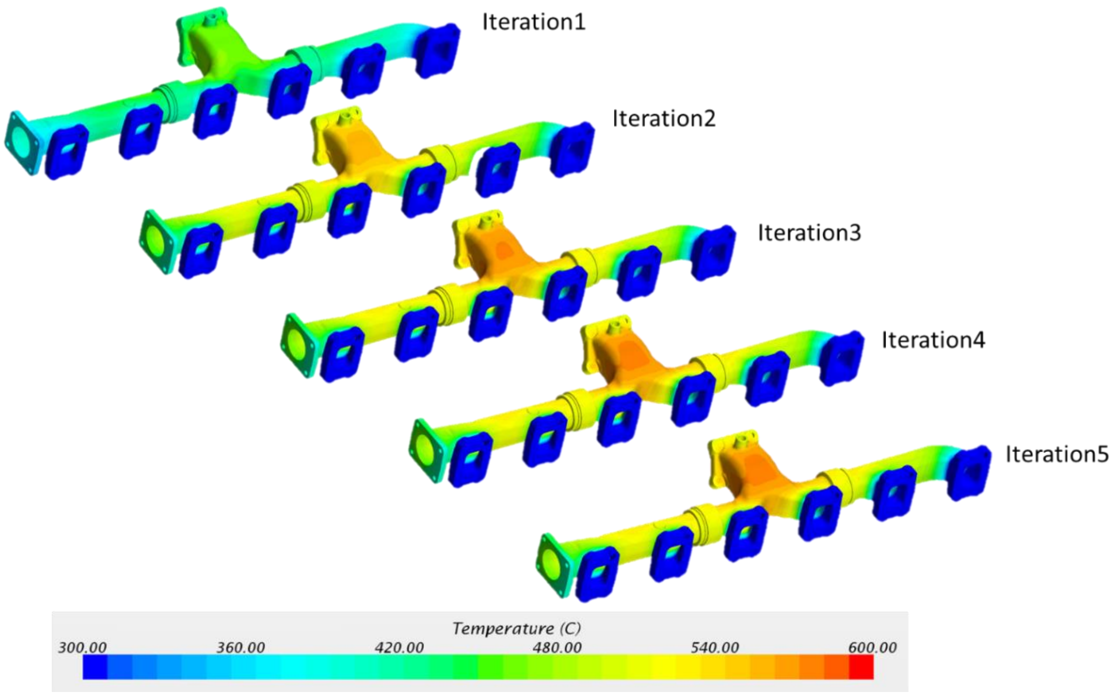


Figure 2.14 Temperature Distribution Outer Surface of Exhaust Manifold @600 Second

2.2 Method 2- STAR-CCM+ to STAR-CCM+ Co-Simulation

Conjugate heat transfer simulations are commonly used to model both solid and fluid domains in a single instance of simulation file. In these simulations, it is not possible to account for the different time scales present in the solid and fluid domains. Pushed out burned gas passes through the exhaust manifold immediately, but manifold itself takes minutes to reach its maximum temperature because the thermal inertia of the solid is high compared to the fluid typically. For scenarios where the time scales of the thermal conduction in the solid are much longer than the time scales in the fluid, STAR-CCM+ to STAR-CCM+ co-simulation provides a particular advantage. The analyses with widely varying time scales, STAR-CCM+ to STAR-CCM+ co-simulation allow the user to treat the fluid and solid domains separately. The solid and fluid simulations must exchange data at a frequency that captures the slow change in the solid temperature. [11]

A solid thermal simulation generally requires less iterations per time step to reach convergence criteria than is required by the fluid simulation. By allocating a separate simulation for the solid thermal analysis, the computational time (CPU time) needed to solve the overall solid-fluid analysis can be considerably smaller when the co-simulation method is used. Also similar level of accuracy is achieved when compared the properly set up co-simulation and the fully-implicit conjugate simulation. Lastly, Co-Simulation model allows you to couple two running simulations in an automated manner. The Solution fields are passed and mapped between the two simulations at pre-defined intervals with no user intervention. [12]

In the exhaust manifold, the hot exhaust gases are being discharged along the different runners independently depending on which cylinder is firing. In this instance, it is necessary to run a Co-simulation between two transient simulations, continuously updating the temperature and heat transfer coefficients as a result of the continuously changing flow field. To predict the heat up of an exhaust manifold is an important simulation capability. The problem is complicated, given the vastly different time-scales that are involved in the fluid physics and the warm up of the solid physics.

In the result section, it will be given the comparison of CHT and Co-Simulation approaches. In the current section, it will be described how the exhaust manifold warm up process is modelled using Star-CCM+ Co-simulation technique.

In this approach, two transient simulations are built; one CCM simulation containing the fluid domain and one CCM simulation containing the solid domain. Another flexibility of this method, generated volume mesh is not needed to be conformal so different mesh densities and mesh types could be used for each simulation. Therefore, Used same mesh structure (trimmer cell) as used in the method1 (sequential coupling) for exhaust gas region. On the other hand, similar mesh type (polyhedral cell) was selected for solid manifold region as used in the method 3 (conjuagte heat transfer analysis) through the calculation. Specified the fluid domain as leading simulation and solid domain as lagging simulation. Thermal field data is exchanged across the fluid and solid interface. In this technique, time intervals of data exchange on interface are determined by user. Each simulation waits for another one to reach the defined exchange step. It works without the requirement to manually create data mappers in each simulation or to create any Java macros that automate the exchange between two instances of STAR-CCM+.

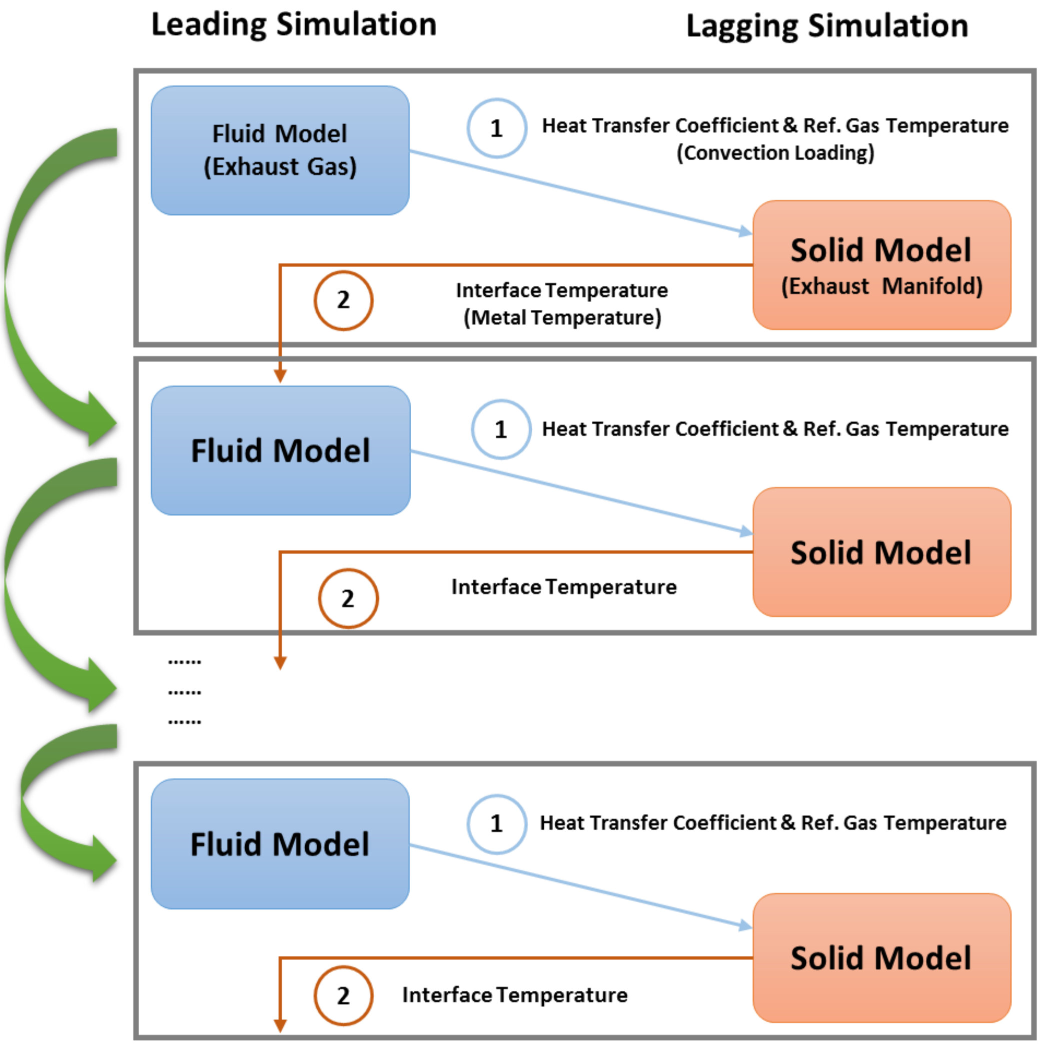


Figure 2.15 Star-CCM+ to Star-CCM+ Co-Simulation Approach Schematic Diagram

The diagram as shown above explains the two-way STAR-CCM+ to STAR-CCM+ co-simulation process. After the leading simulation runs for a specified number of iterations, a mapping operation occurs and then the convection loads (heat transfer coefficient and reference gas temperature) on the interface is transferred to the waiting lagging simulation. The lagging simulation uses the data as boundary condition and runs. Once the lagging run is completed, the temperature on the interface is fed back to leading simulation. This process continues to run and send data until their individual stopping criteria are satisfied.

2.2.1 Modelling Details

- Both solid and fluid analyses are performed in transient mode on high performance computer.

- Thermal load datas are exchanged between fluid and solid somains at the interface; set up the data exchange frequency 1time step for solid domain and 5 time steps for fluid domain.
- Fluid domain run time is determined as 40 engine cycle corresponds to 2,667 seconds with 5,00e-05 sec. as time step.
- Modelling the exhaust heat up process of exhaust manifold takes 600 second with 0.05625 second time scale.
- Defined five random points in the solid manifold domain to monitor the temperature behavior and check the convergence through the heat transfer calculation apart from residuals.

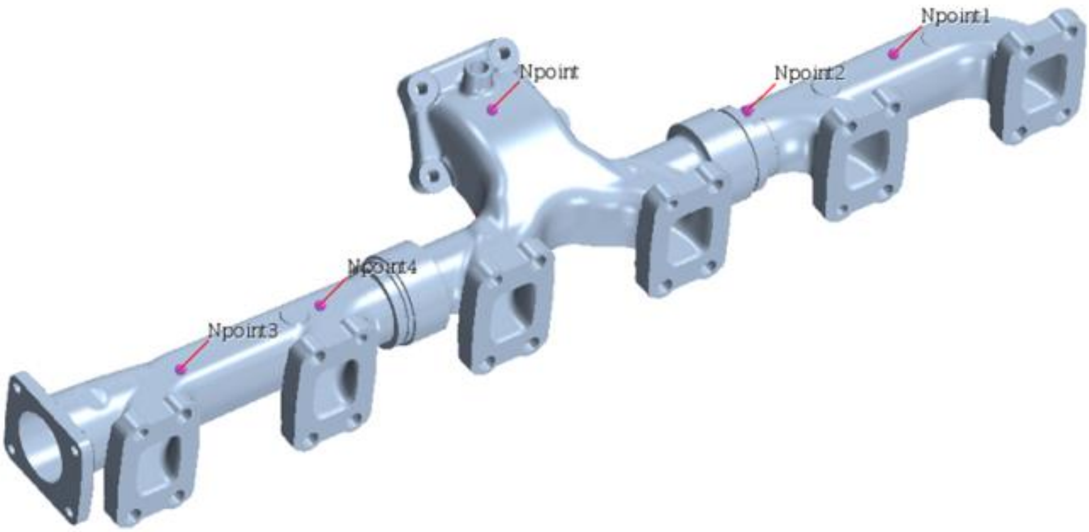


Figure 2.16 Monitored Points on Exhaust Manifold to Check Convergence

2.2.2 Sensitivity Study 1

Firstly, sensitivity study was performed for the parameters such as exchange frequency, time step, number of inner iteration and mesh type for the fluid-solid domains to obtain the optimum settings for Co-Simulation methodology. The details of this sensitivity study can be seen on the table 2.1. The change of these parameters, which come up with computational run time cost were shown to have negligible affect (~ 0.5%-2%) on the results, so the base case results are used for the rest of study.

Table 2.1 Co-Simulation Sensitivity Study 1 Parameters

		Time Step	**Run Time**	**Total Time Step**	**Exchange Time**	**Total Exchange**	**Inner Iteration**	**CPU Time**
		Second	Second	#	Second	#	#	Hours
BC	*Fluid*	0,00005	0,066667	1333	0,00025	267	15	X
	Solid	0,45	600	1333	2,25	267	8	
SC1	*Fluid*	0,00005	0,066667	1333	0,00005	1333	15	~2X
	Solid	0,45	600	1333	0,45	1333	8	
SC2	*Fluid*	0,00005	0,066667	1333	0,00005	1333	15	~2X
	Solid	0,09	600	1333	0,45	1333	8	
SC3	*Fluid*	0,00005	0,066667	1333	0,00025	267	15	~1.5X
	Solid	0,45	600	1333	2,25	267	50	
SC4	*Fluid*	*0,00005*	*0,066667*	*1333*	*0,00025*	*267*	*15*	*~1.2X*
	Solid	*0,45*	*600*	*1333*	*2,25*	*267*	*8*	

- BC: Base Case, fluid domain with trim cells
- SC1: Exchange frequency increased (set up the exchange frequency as 1 time step for each solid and fluid simulation)
- SC2: Solid domain time step decreased from 0,45 second to 0,09 second, exchange frequency increased
- SC3: Solid domain inner iteration increased from 15 to 50
- SC4: similar settings with base case but changed fluid domain volume mesh with polyhedral cells.

The effect of the increasing data exchange frequency with 5 time steps on results could be considered as negligible. Presented the interface temperature difference, local reference temperature difference and local heat transfer coefficient difference between the base case and sensitivity case on the figure 2.17.

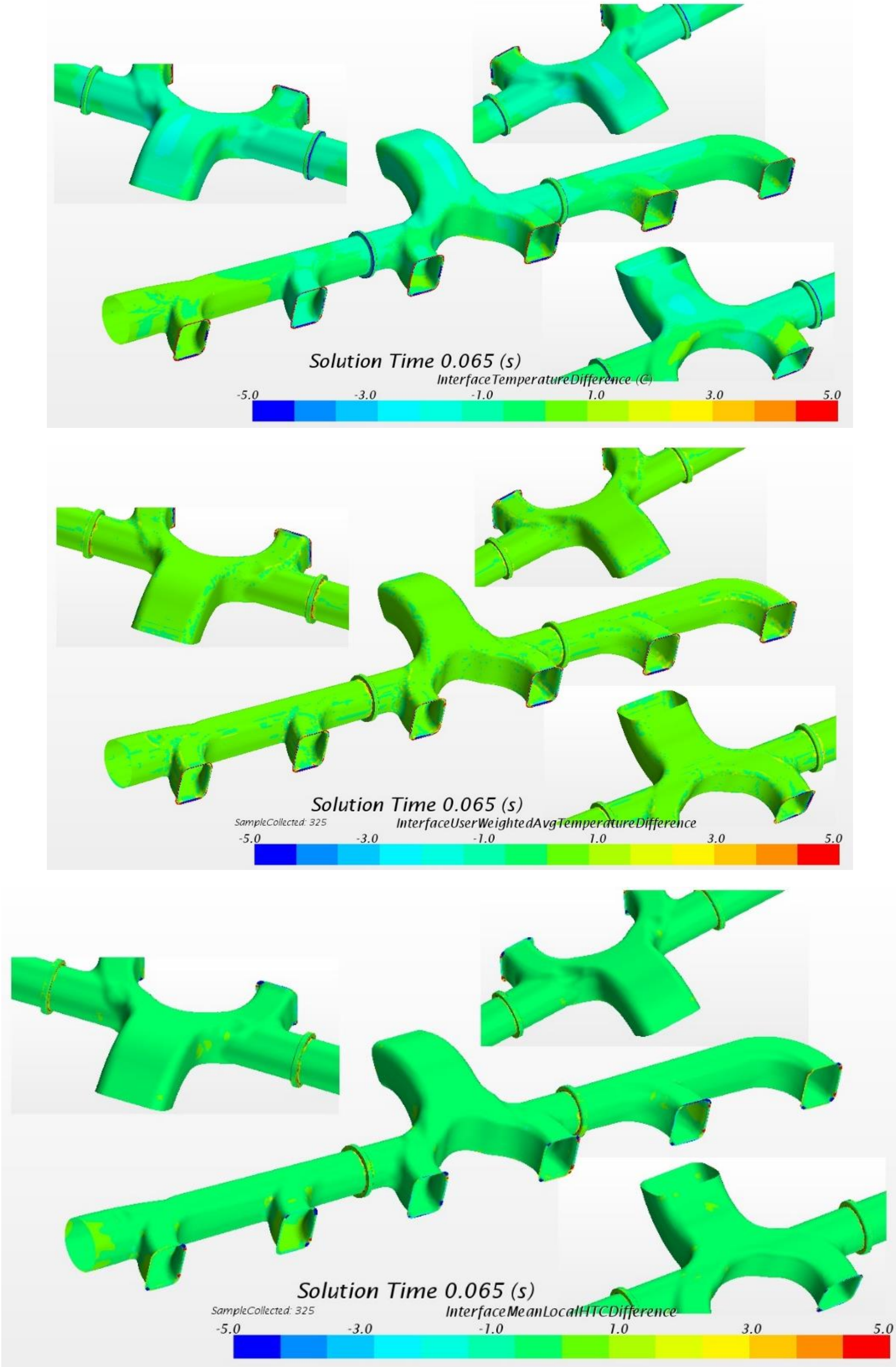


Figure 2.17 Difference Between Base Case (BC) and Sensitivity Case1 (SC1)

2.2.3 Sensitivity Study 2

The key point of the Co-Simulation method is; how many engine cycle of fluid domain has to be coupled with the warm up period of solid domain. Modelling the whole warm up period of exhaust manifold requires ~9000 engine cycle (consider that 1 engine cycle is 0,06667 second at 1800 rpm working condition and heat up period is 600 second) which means it is not feasible for Co-Simulation run in terms of computational time. Therefore, sensitivity study was carried out for the coupling of solid domain heat up period with the 1 engine cycle, 20 engine cycle and 40 engine cycle fluid domain simulation which come up computational run time cost. Computational run time of each cases are displayed on the table below.

Table 2.2 Co-Simulation Sensitivity Study 2 Details

Case Number	Number of Engine Cycle for Fluid Simulation	Computational Run Time
Case1	1 Engine Cycle (0,0667 second)	0.6 days
Case 2	20 Engine Cycle (1,333 second)	3 days
Case 3	40 Engine Cycle (2,667 second)	6 days

Monitored the temperature value of random points during the exhaust manifold solid side analysis is shown on the figures below. On the other hand, observed the maximum and volume averaged temperature of structure domain through the warm up period.

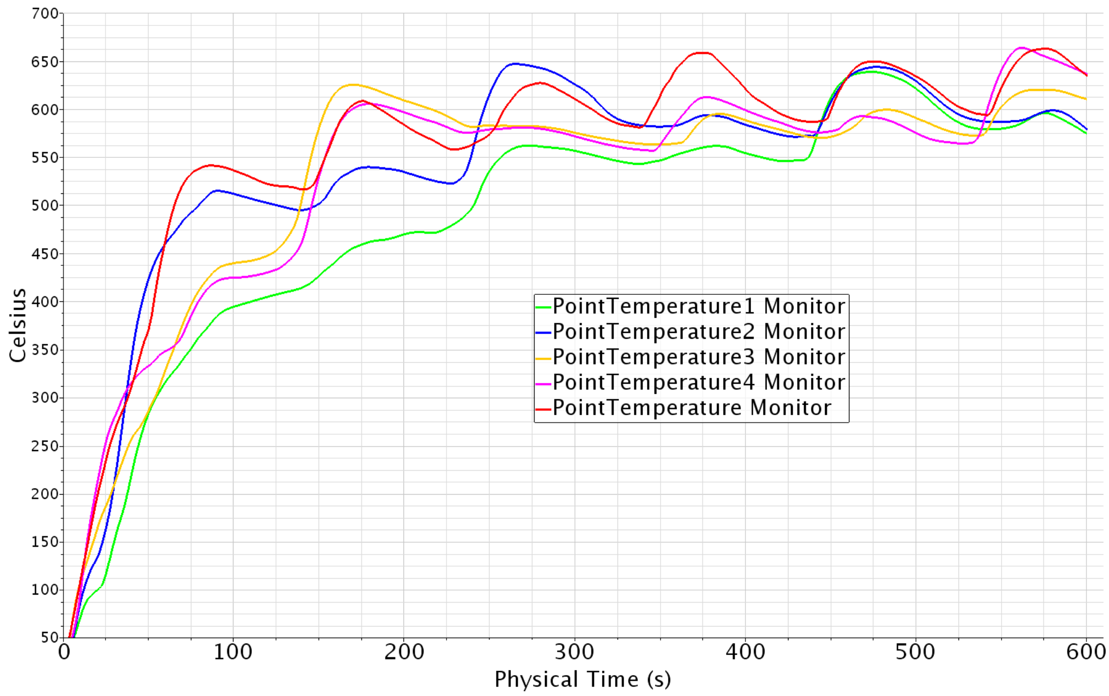


Figure 2.18 Temperature Behavior of Points at Case 1

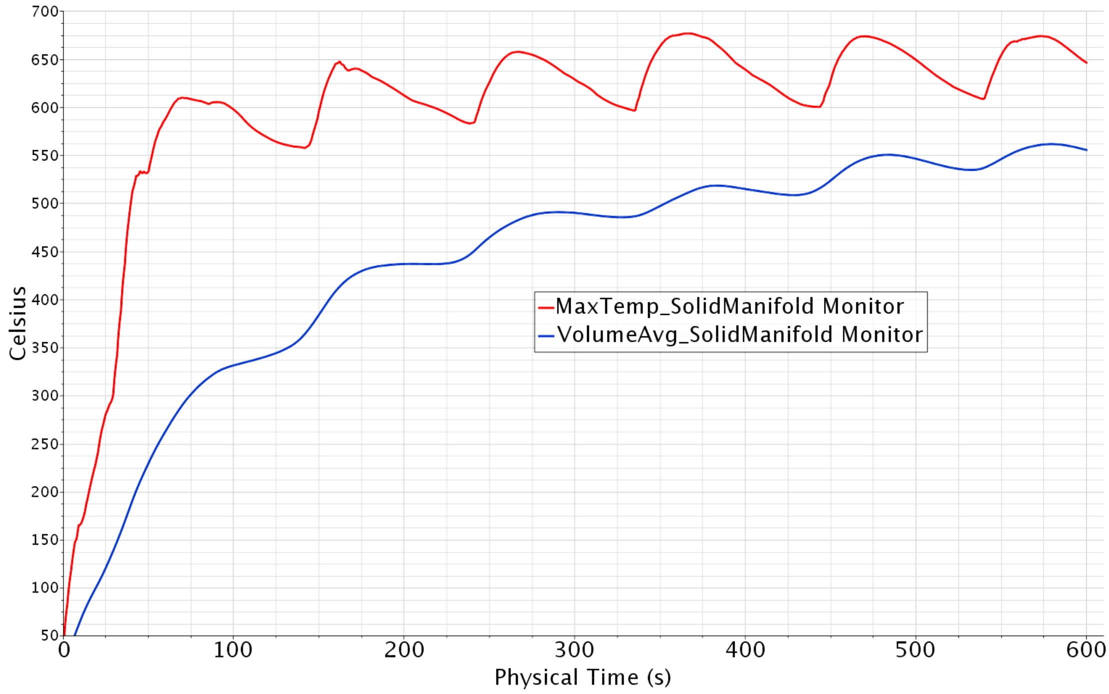


Figure 2.19 Maximum and Volume Averaged Temperature of Manifold at Case 1

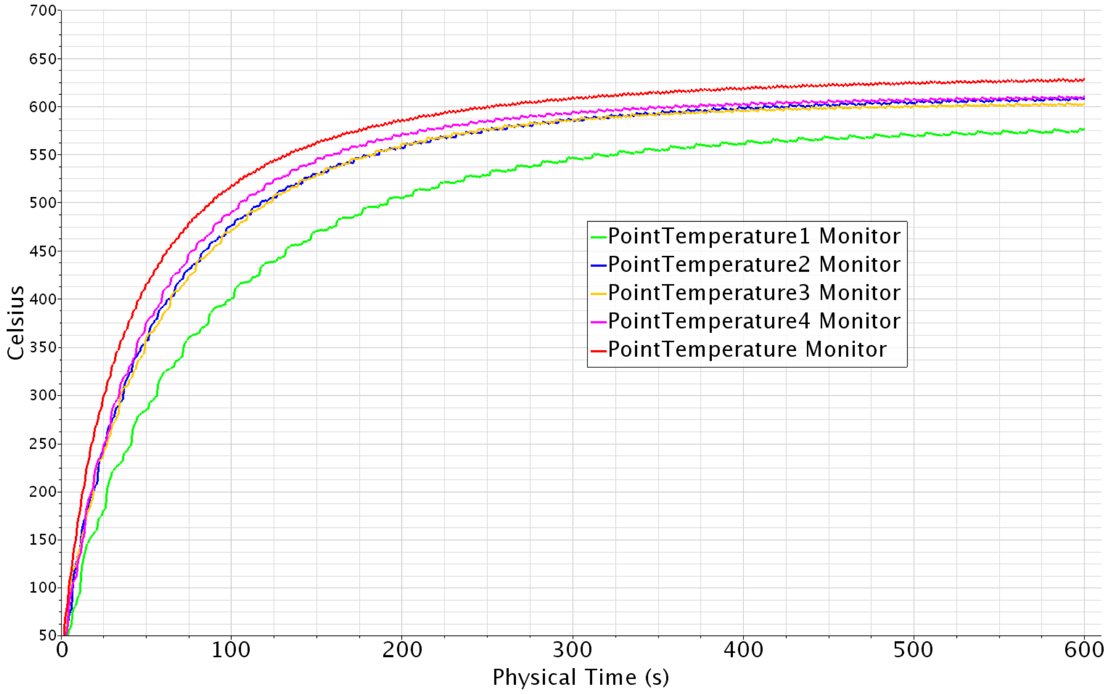


Figure 2.20 Temperature Behavior of Points at Case 3

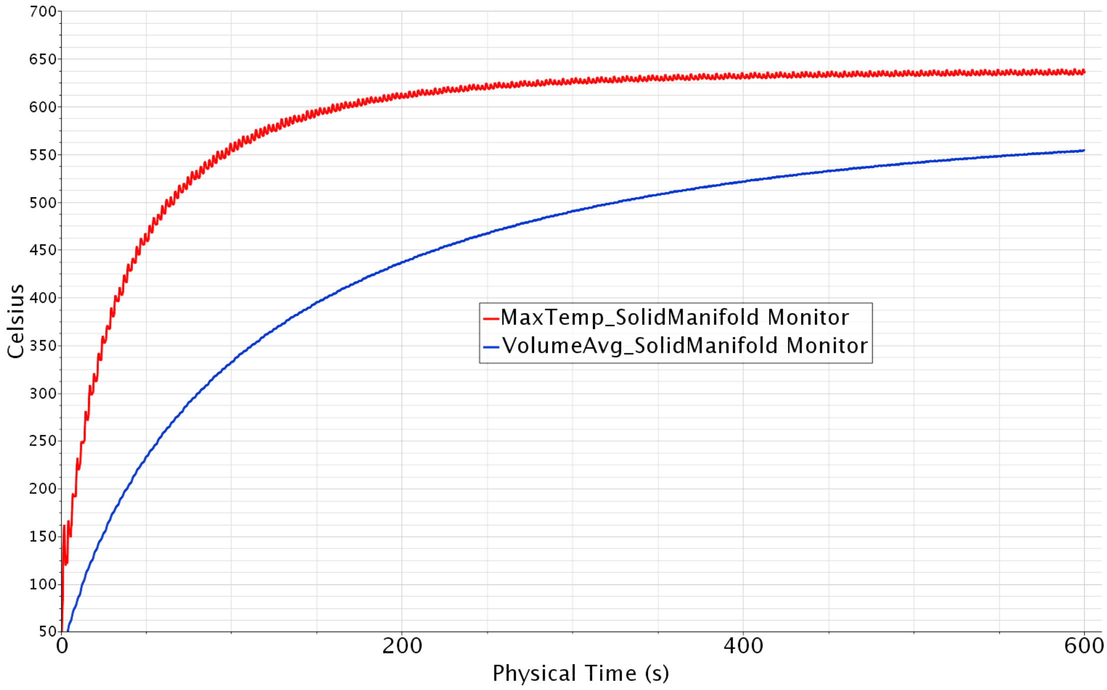


Figure 2.21 Maximum and Volume Averaged Temperature of Manifold at Case 3

The temperature of every point within the solid domain is expected to increase until the steady state condition is reached. But as seen from the results, in case1, at some time intervals, temperature decrease which is an odd behavior, is observed. This duration corresponds to the turbo outlet mass flow decrement period. This behavior does not represent the reality. But, when the case settings are considered, this makes sense, because the duration where the turbo outlet mass flow decreases is about 0.009sec., corresponding to the solid domain run time period of 35-40sec., which leads to reduction in temperature of solid due to conduction through the solid. To get rid of that behavior, in the 2nd case, the run time for fluid domain increased to 20 engine cycles which comes with a computational run time cost, and an improvement is observed but still there is room for further improvement. In the 3rd case, the run time for fluid domain increased to 40 engine cycles, the odd behavior almost diminished. The warm up behavior of the 3rd case seems reasonable as seen from the figure 2.20 and figure 2.21. So, the higher the number of engine cycle for fluid domain run coupled with the solid domain, the more realistic case is the third case settings achieved good compromise between the run time cost and accuracy level.

2.2.4 Results of Co-Simulation Method

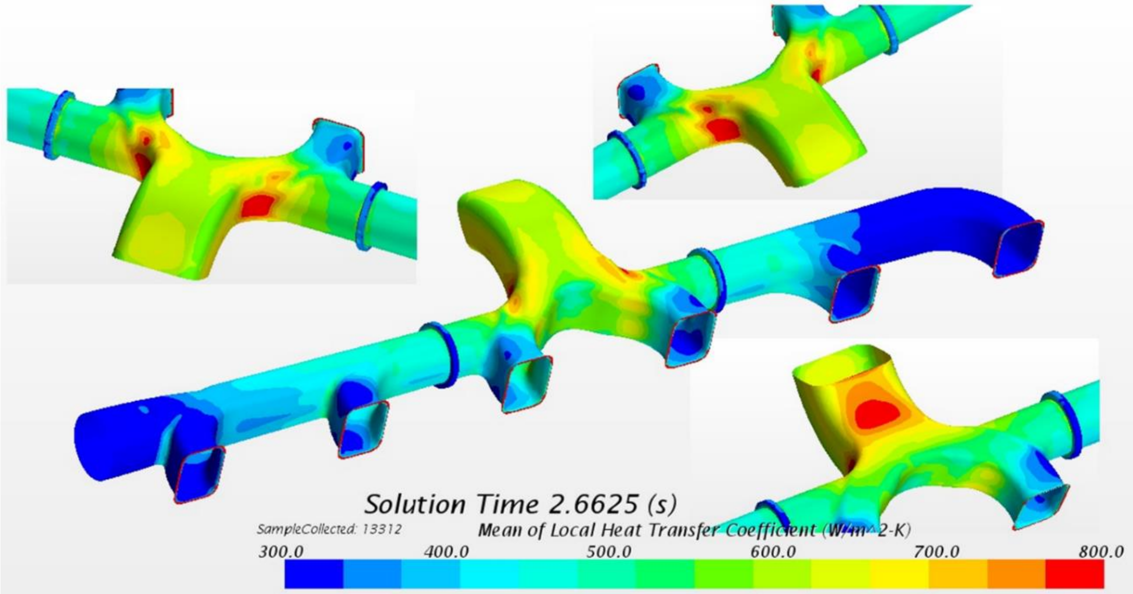


Figure 2.22 HTC Distribution on Interface at 40th Engine Cycle

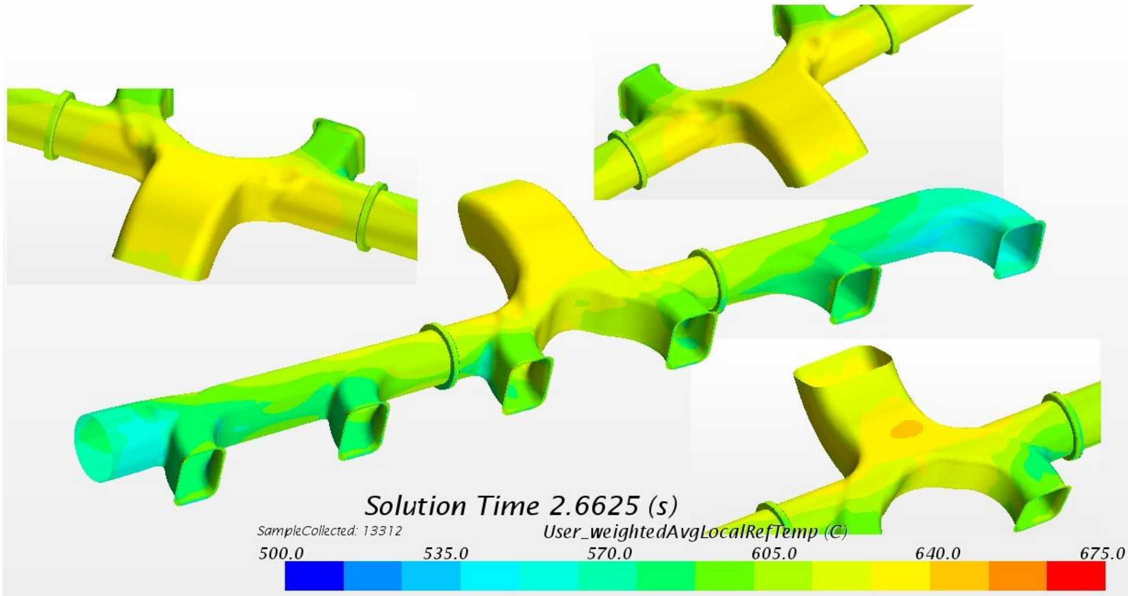


Figure 2.23 Reference Temperature Distribution on Interface at 40th Engine Cycle

2.3 Method3- Conjugate Heat Transfer (CHT) Approach

Conjugate heat transfer refers to the ability to compute conduction of heat through solids domain, coupled with convective heat transfer in a fluid domain. [13]

This approach involves simultaneous heat transfer in both a solid and a fluid which means single STAR-CCM+ simulation can obtain solutions in both the solid and the fluid domain. Fluid and solid equations are implicitly coupled and are solved simultaneously. The surface conditions (temperature and energy) are performed at the fluid-solid interface stated below;

$$T_{solid|surface} = T_{fluid|surface} \tag{2.3}$$

$$q_{solid|surface} = q_{fluid|surface} \tag{2.4}$$

$$-K\frac{\partial T_{solid}}{\partial n}|_{surface} = h_f(T_{surface} - T_{ref}) \tag{2.5}$$

With the purpose of comparison during the methodology development, built up conjugate heat transfer model just for one case and run for 110 second with a gradually increasing time step starting from 1.0e-5 second up to 1.0e-4 sec. which used 32 CPUs. The mesh model of fluid domain contains 350k polyhedral cells. On the other hand, solid domain which consists of manifold, bolts, gasket, studs and simplified engine head just for stiffness purposed for the subsequent durability assessment involves 380k polyhedral cells. Volume mesh of domains can be seen from the Figure 2.24 and Figure 2.25 respectively. All interfaces are fully conformal in the model. Conformal interfaces guarantee a 100% face match between the two regions/boundaries participating at an interface. This conformal match will guarantee that the physics models assigned for each region communicate at each and every cell face within this interface.

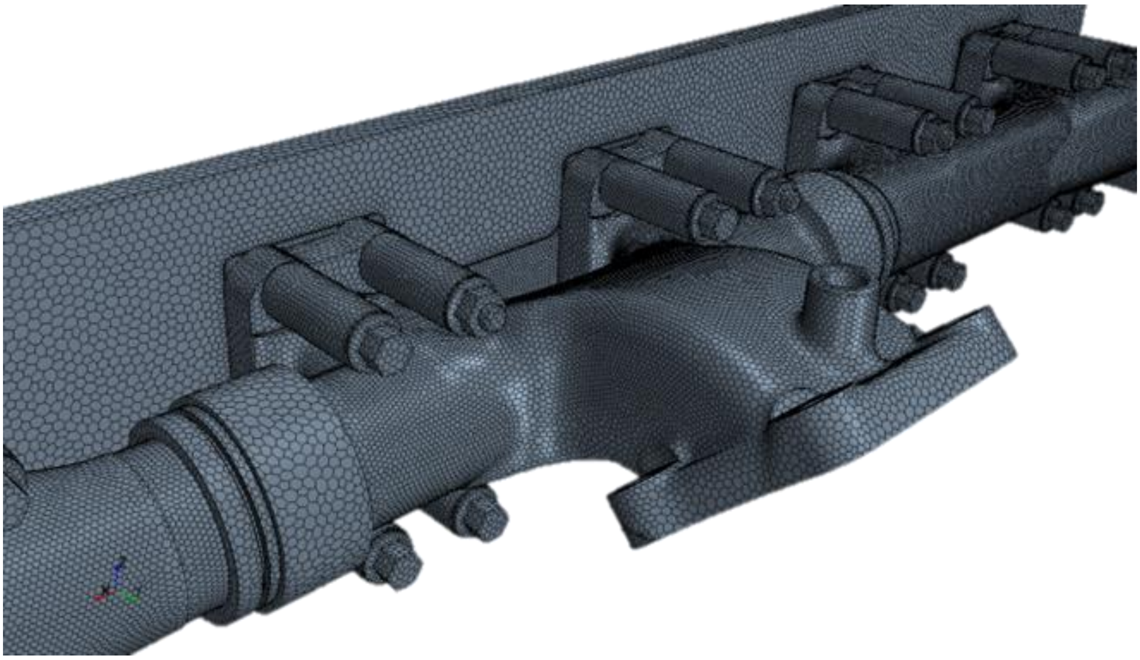


Figure 2.24 Volume Mesh of Solid Domain at CHT Simulation

Figure 2.25 Volume Mesh of Fluid Domain at CHT Simulation

CHT eliminates the need to define thermal boundary conditions on structure-fluid interface such as iterative data exchange process between CFD & FE is avoided. However, it is not feasible to run the case for a time period due to very long run times.

The case is run only for one-sixth of total time period with 48 CPUs which takes almost a month.

2.3.1 Results of Conjugate Heat Transfer Method

Figure 2.26 and figure 2.27 show the temperature results on the interface and outer surface of exhaust manifold respectively at 110 second.

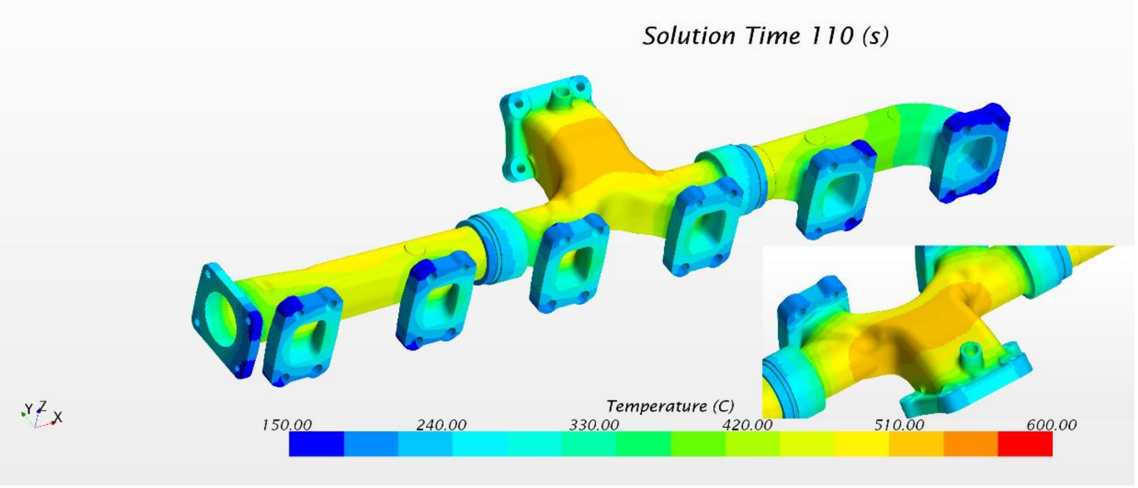


Figure 2.26 Manifold Outer Surface Temperature @110 Second

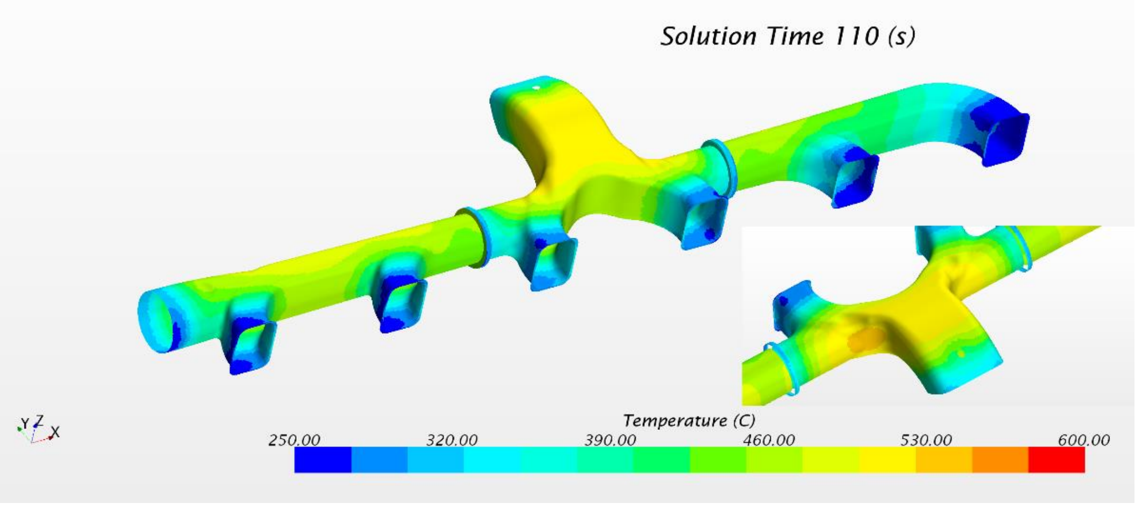


Figure 2.27 Interface Temperature Results of Conjugate Approach @110 Second

2.4 Comparison of Thermal Modelling Approaches

The temperature predictions (both fluid and solid domain) of three methods at 110. second are compared in the first two figures below respectively. Scalar contours is based on the same color scale.

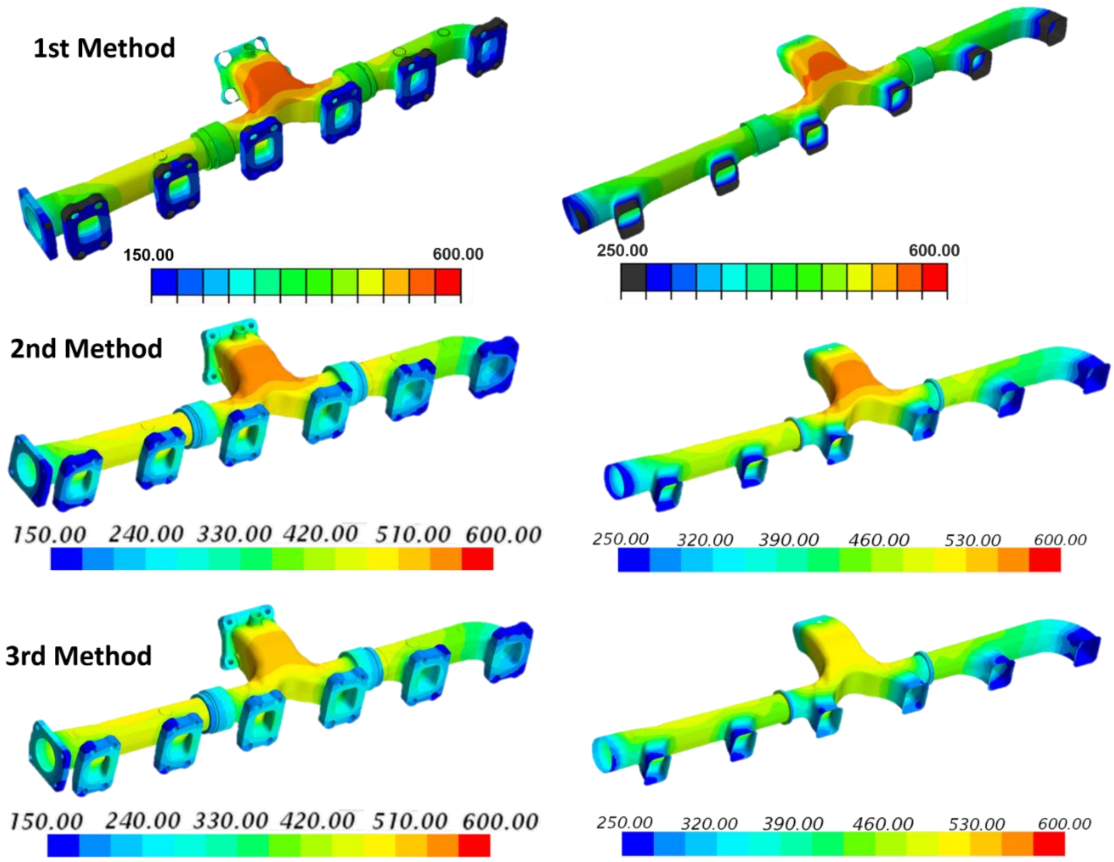


Figure 2.28 Temperature Distribution Comparison of Method 1, Method2 and Method3 @110 Second

As seen from the temperature distribution contour plot, conjugate heat transfer approach (3^{rd} Method) come out to predict ~30 °C lower temperature than the other two methods.

Moreover, compared the warm up behavior of Method 2 and Method 3 for 110 second by using the points within the manifold structure, maximum and volume averaged temperature change of solid with time and displayed on the following plots respectively. Method 2 predicts higher temperature up to 25~30 °C compared with the Method3.

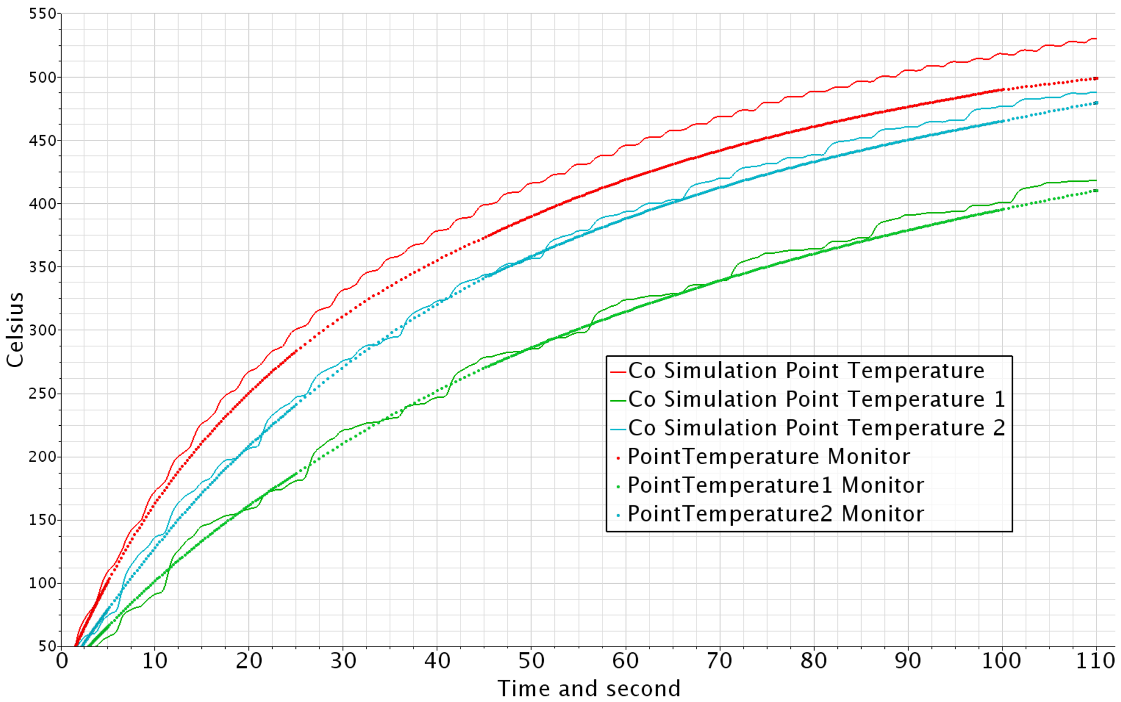


Figure 2.29 Comparison of Co-Simulation and Conjugate Method (The temperature of points within the exhauts manifold)

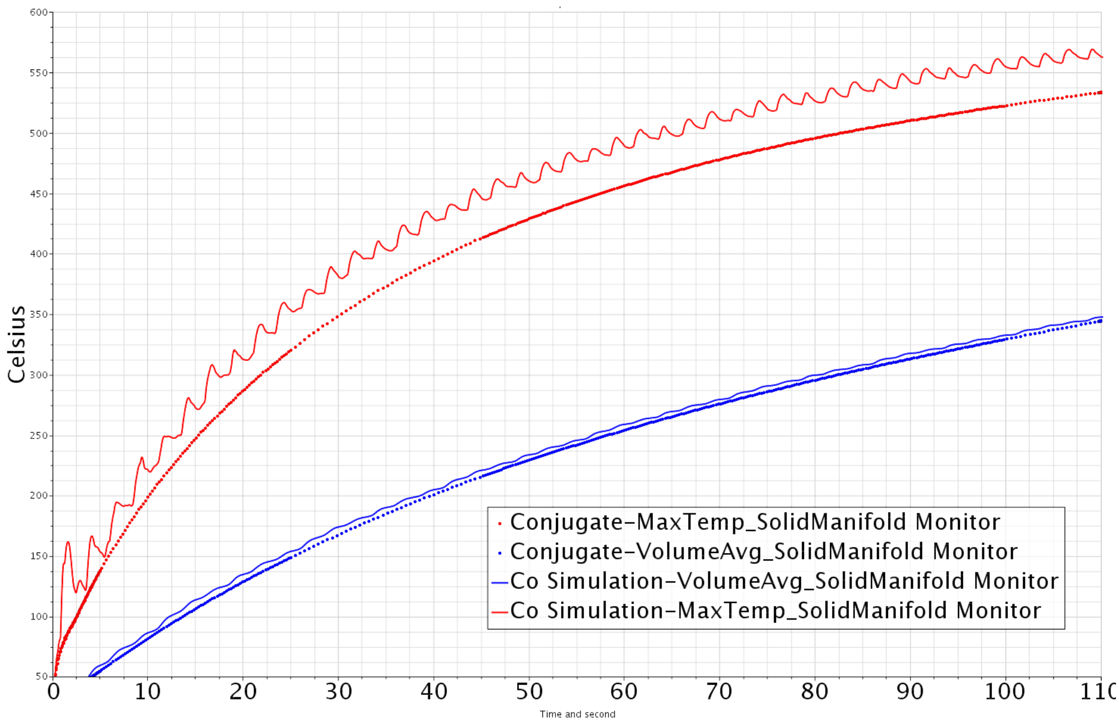


Figure 2.30 Comparison of Co-Simulation and Conjugate Method (Maximum and Volume Averaged Temperature of Manifold Structure)

On the other hand, temperature prediction of Method1 and Method 2 are compared at the end of complete warm up period (at 600 second). It is shown from the Figure 2.31, Method 1 tends to underpredict temperatures by about 60-70 °C. The temperature results of monitored points are presented on the Table 2.3.

Table 2.3 Temperature Differences on Monitored Points (Method1 and Method2)

Method / **Point Number**	**Temperature Difference (°C)**
Point1	55.40
Point2	70.50
Point3	66.76
Point4	64.60
Point5	65.70

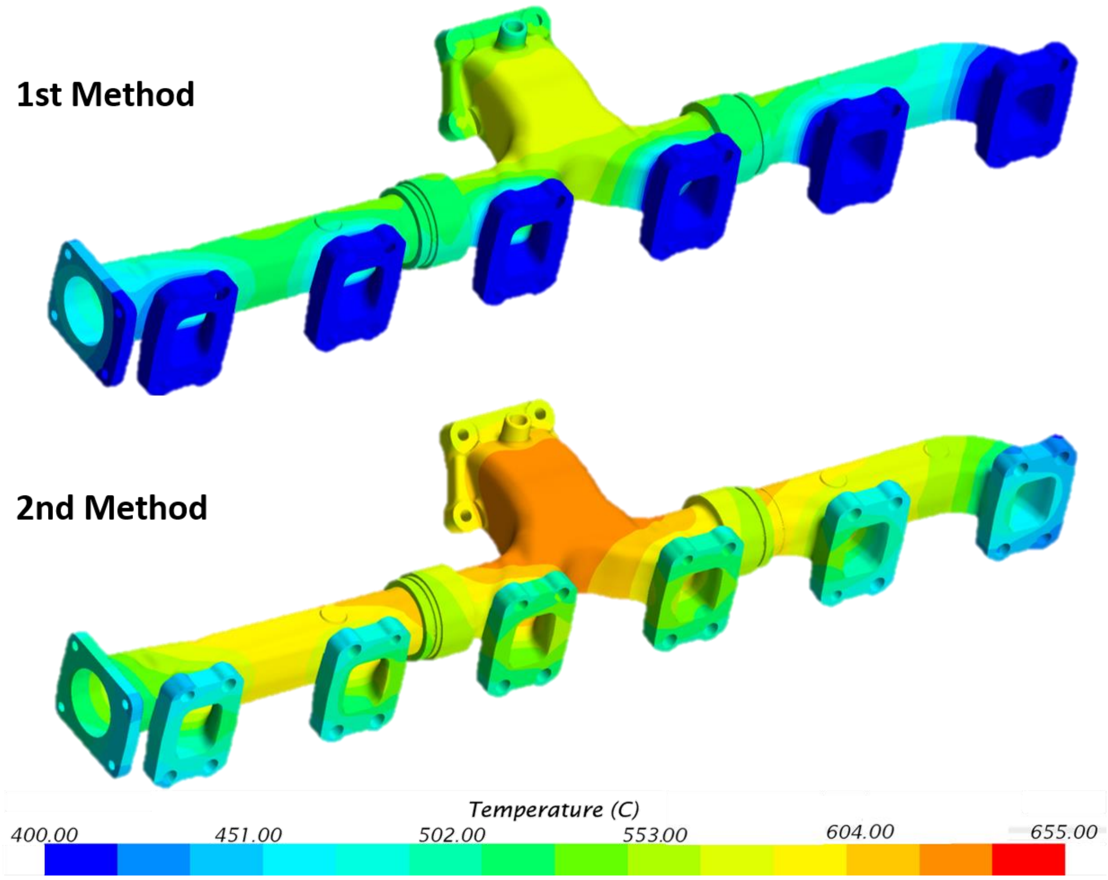


Figure 2.31 Exhaust Manifold Skin Temperature from Method 1 and Method 2 @600 sec.

CHAPTER 3

THERMAL SURVEY

To validate the CAE results, performed an experimental study at Ford Otosan Gölcük dynamometer laboratory. Monitored the whole warm up period of exhaust manifold with infrared camera called thermal survey investigation. Thermal camera measurement taken with am EMC cycle comprised of rated power (1800 rpm at full load) and high idle condition (2150 rpm at no load) from exhaust manifold of a 13L engine using 480 PS (HP) calibration from two different camera angles.

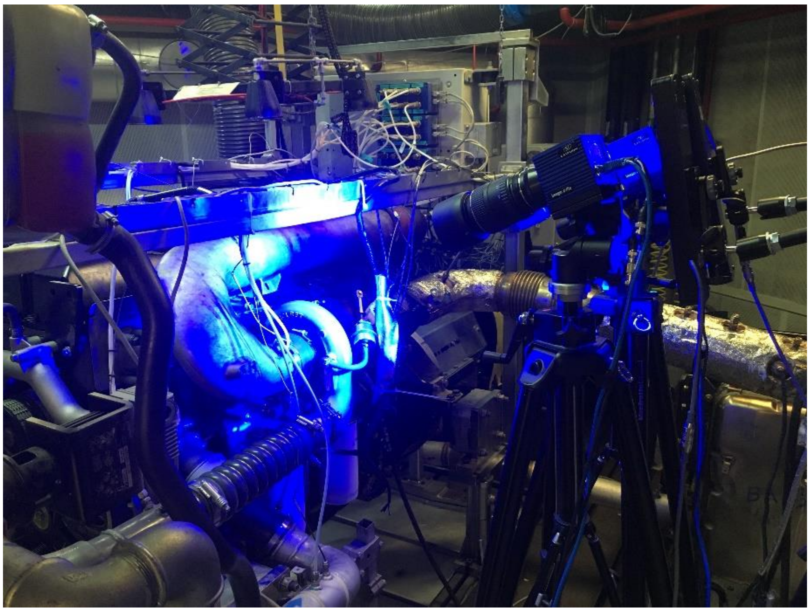


Figure 3.1 Thermal Survey Investigation at Gölcük Dynamometer Test Center

3.1 Thermal Camera

A thermal imaging camera is a reliable non-contact instrument which is able to scan and visualize the temperature distribution of entire surfaces of machinery and electrical equipment quickly and accurately. Our eyes are detectors that are designed to detect electromagnetic radiation in the visible light spectrum. All other forms of electromagnetic radiation, such as infrared, are invisible to the human eye.

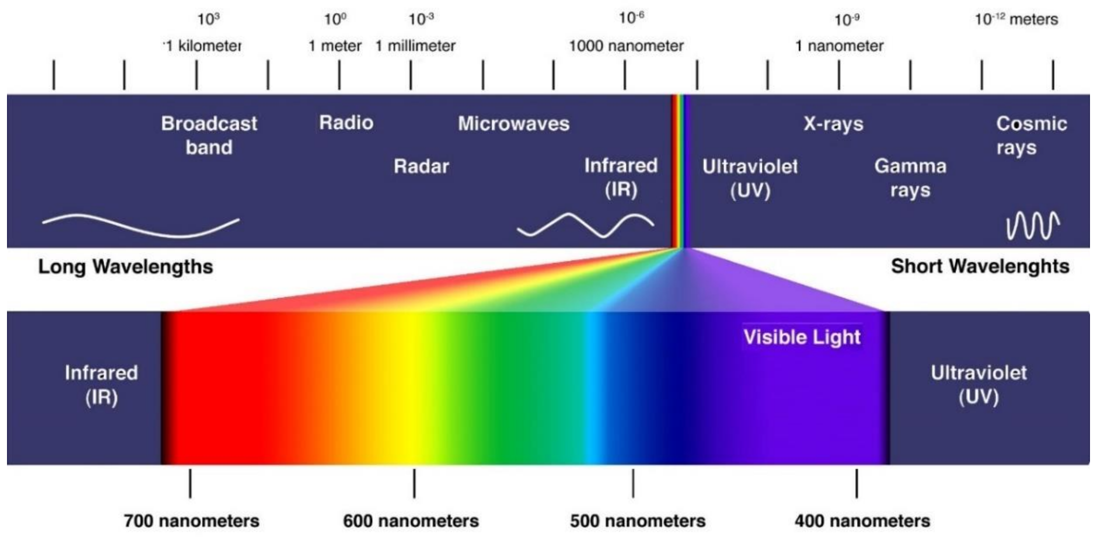


Figure 3.2 Electromagnetic Spectrum

Infrared radiation lies between the visible and microwave portions of the electormagnetic spectrum. The primary source of infrared radiation is heat or thermal radiation. Any object that has a temperature above absolute zero (-273.15 °C or 0 Kelvin) emits radiation in the infrared region. The warmer the object, the more infrared radiation it emits. A thermal imaging camera records the intensity of radiation in the infrared part of the electromagnetic spectrum and transforms it to a visible image.

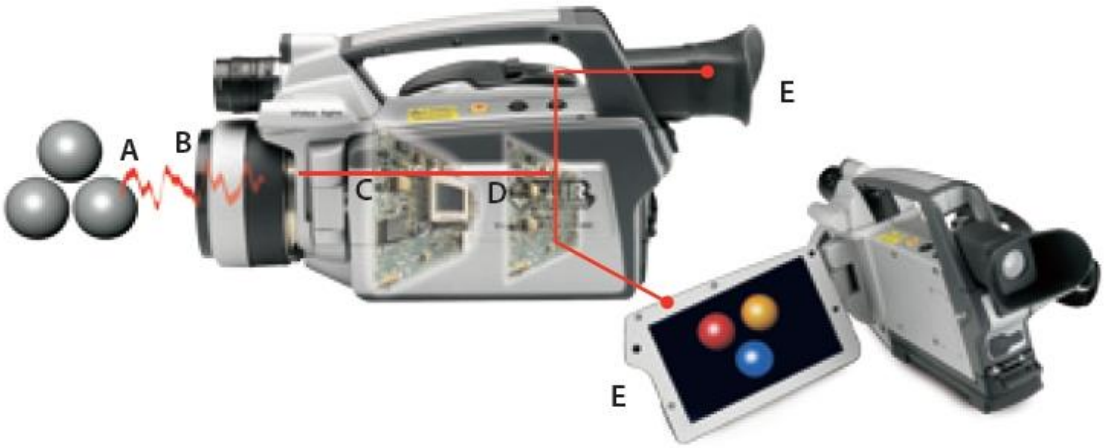


Figure 3.3 Thermal Imaging Camera Working Principle

Infrared energy (A) coming from an object is focused by the optics (B) onto an infrared detector (C). The detector sends the information to sensor electronics (D) for image processing. The electronics translate the data coming from the detector into an image (E) that can be viewed in the viewfinder or on a standard video monitor or LCD screen. Infrared thermography is the art of transforming an infrared image into a radiometric one, which allows temperature values to be read from the image. So every pixel in the radiometric image is in fact a temperature measurement. In order to do this, complex algorithms are incorporated into the thermal imaging camera. [14]

The main aim of this study is to assess and measure exhaust manifold surface temperature on maximum heat rejection conditions. The engine run at 1800 rpm 2000 Nm (rated power) conditions for 10 minutes until surface temperatures are stabilized. Taken measurement with FLIR thermal camera seen on the figure 3.4 from exhaust side of the engine.

Figure 3.4 FLIR Thermal Camera used Experimental Study

3.1.1 Thermal Camera Validation

Instrumented skin thermocouple onto the middle part of exhaust manifold, high temperature distribution region, to validate thermal camera result. (see Figure 3.5)

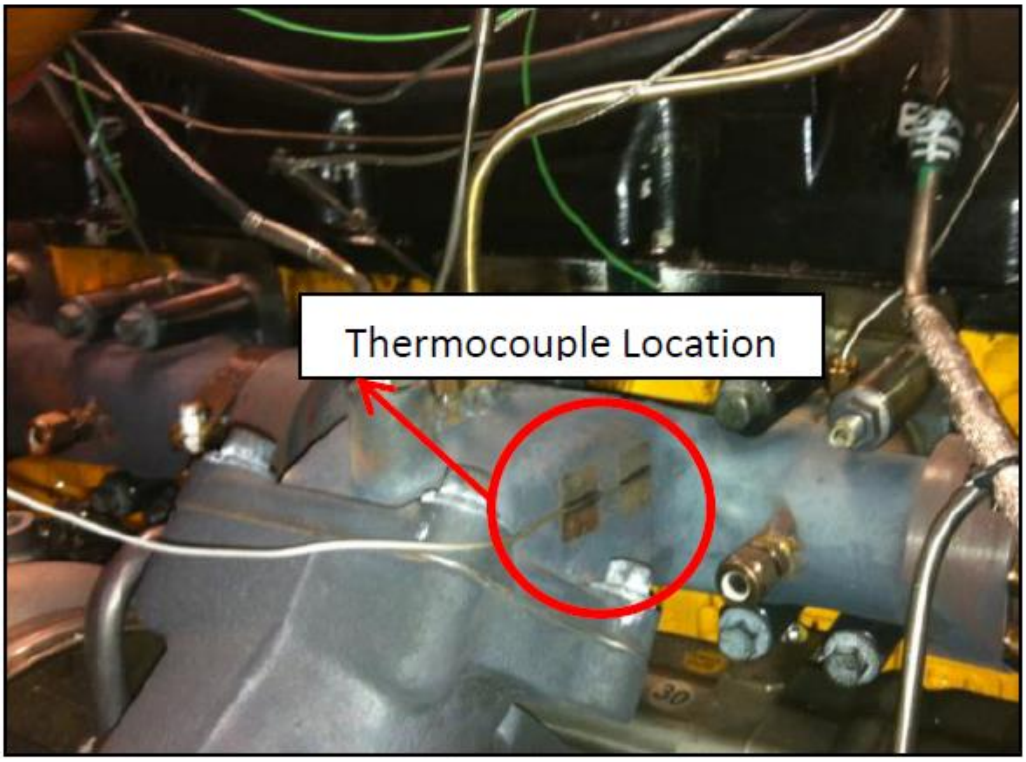


Figure 3.5 Skin Thermocouple Instrumentation on Manifold

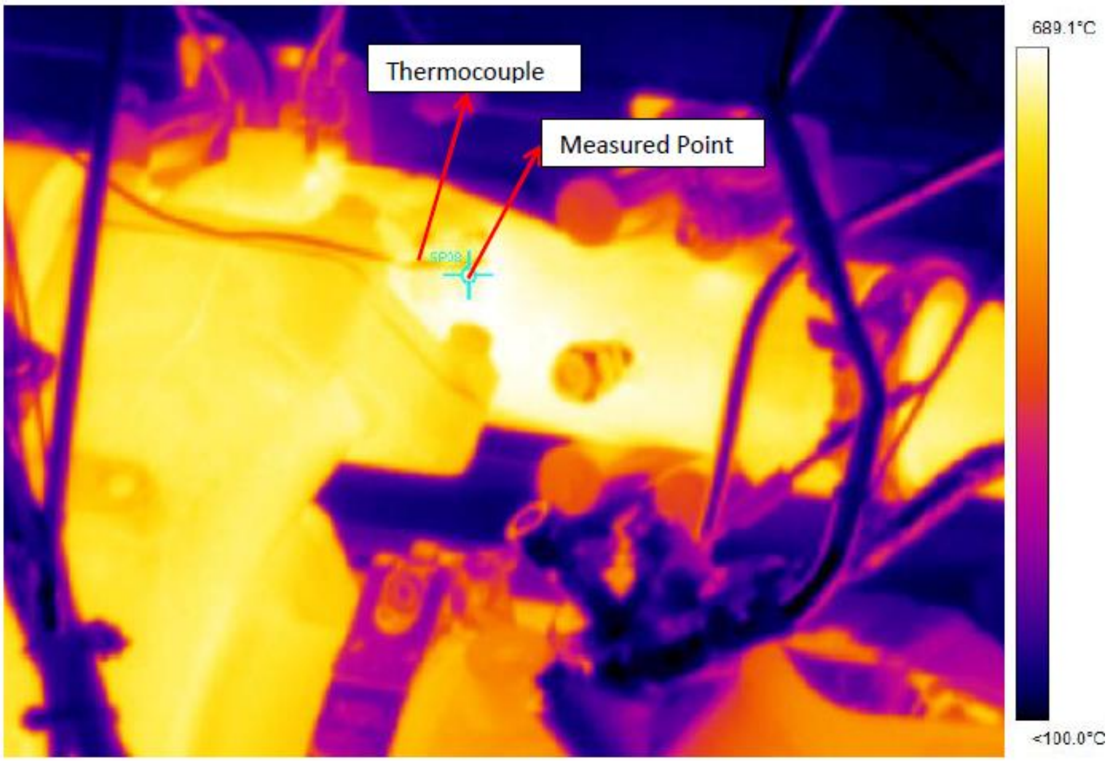


Figure 3.6 Thermal Camera Results for Thermal Camera Validation

To read the correct temperature values, one important thing needs to be taken into account, and that is a factor known as emissivity. Emissivity (ε) is the efficiency with which object emits infrared radiation. During the thermal survey investigation, emissivity value of the thermal imaging camera has predefined as 0,92 which has also tuned before.

Table 3.1 Thermal Camera Validation Results

Point	**Temperature (°C)**
Skin Thermocouple	T
Measurement Point	T + 5

3.2 Thermal Camera Results

ThermaCAM Researcher Professional 2.10 is being used to post-process the measurements.

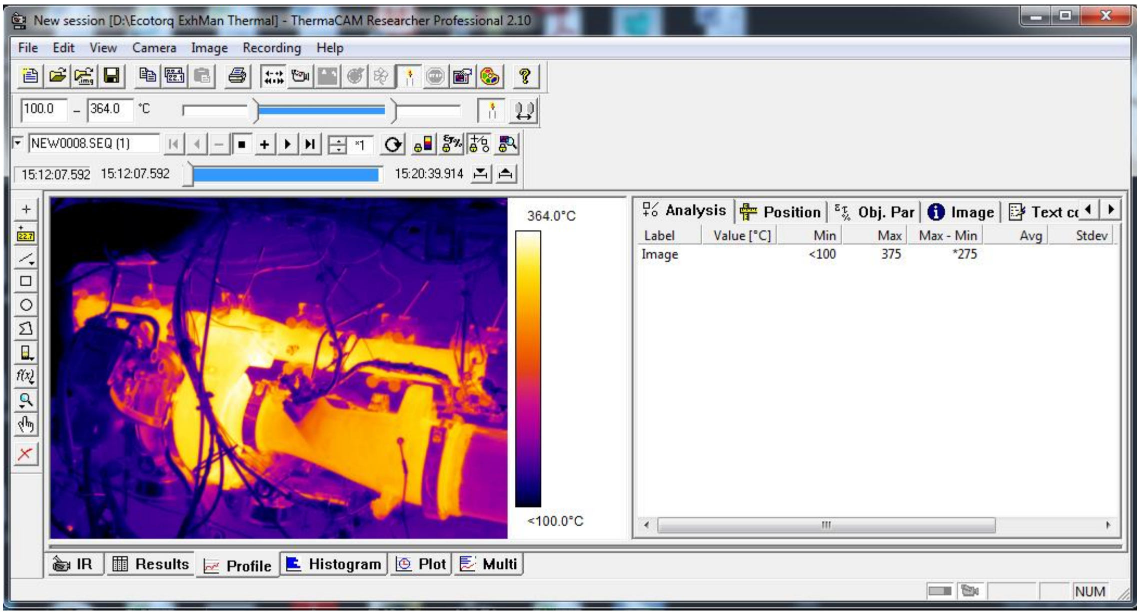


Figure 3.7 ThermaCAM Researcher Professional User Interface

In the following contour plots represent the metal temperature outer skin of the exhaust manifold at the end of warm up period (at 600 second). Apart from the thermal camera post-procesing results, predicted metal temperature at Method1 and Method2 are shown for the comparison purpose.

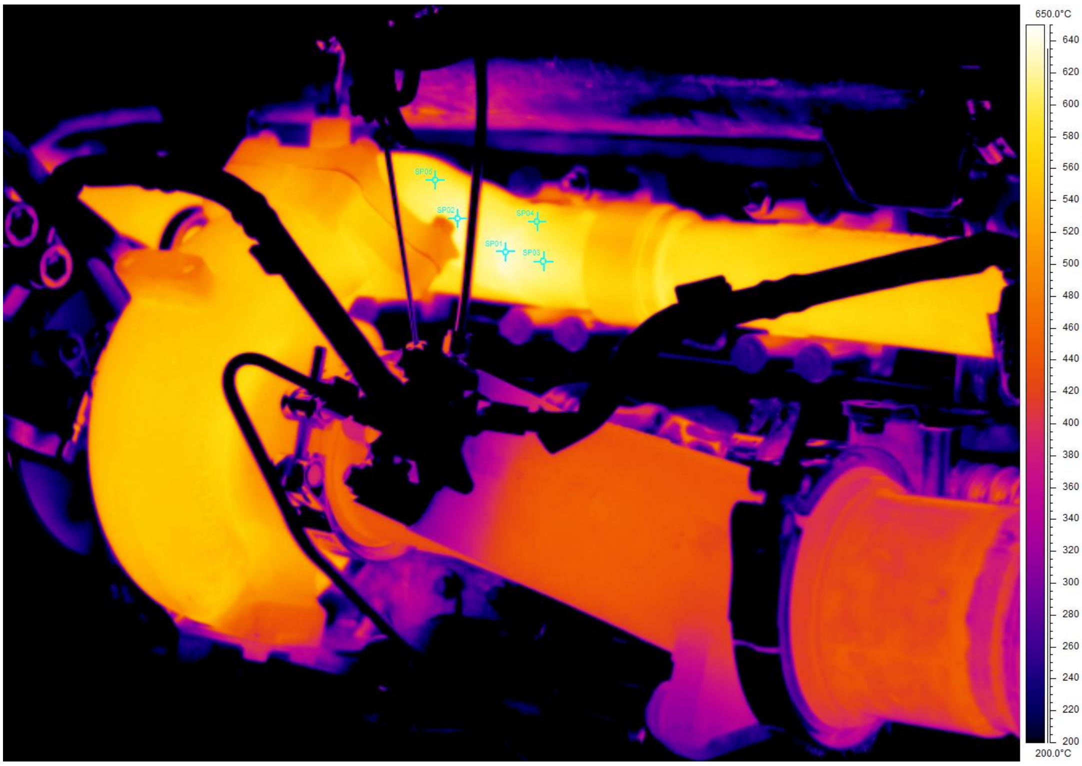


Figure 3.8 Thermal Camera Results at the end of Warm Up Period (View 1)

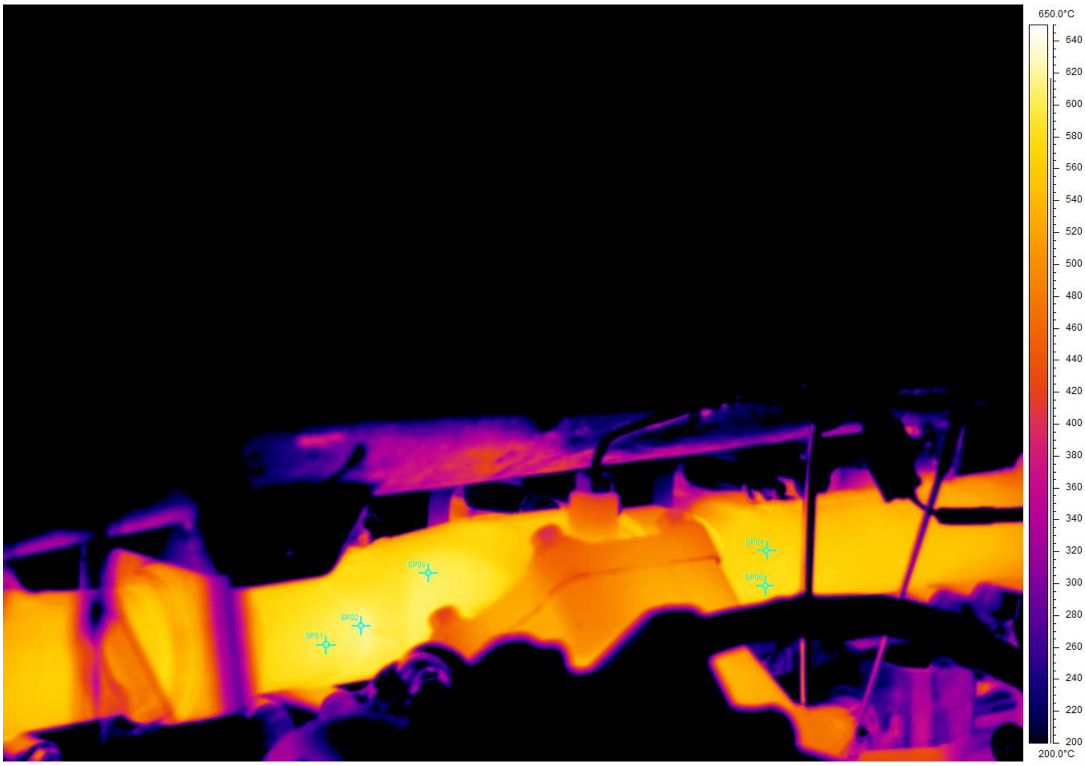


Figure 3.9 Thermal Camera Results at the end of Warm Up Period (View 2)

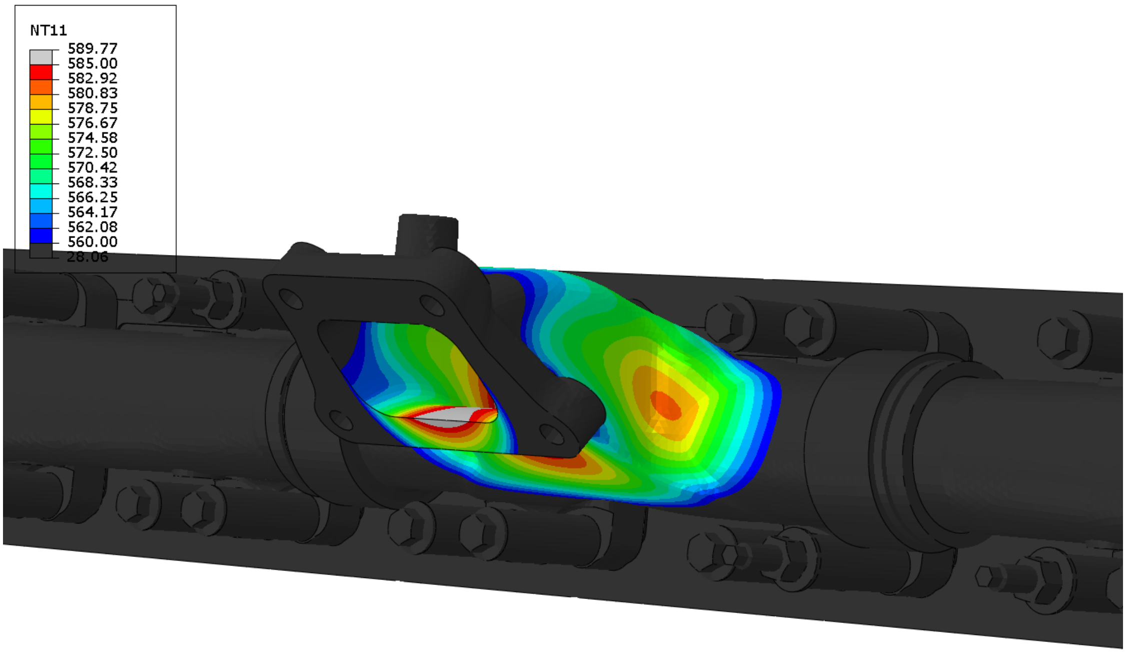


Figure 3.10 Metal Temperature Results comes from Sequential Coupling (View1)

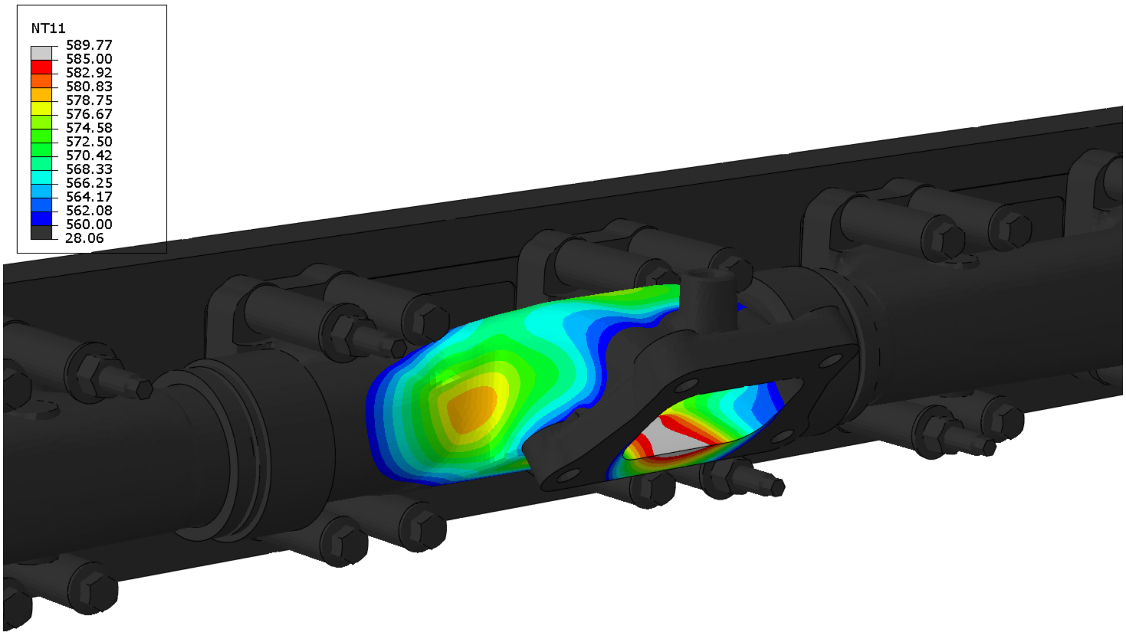


Figure 3.11 Metal Temperature Results comes from Sequential Coupling (View2)

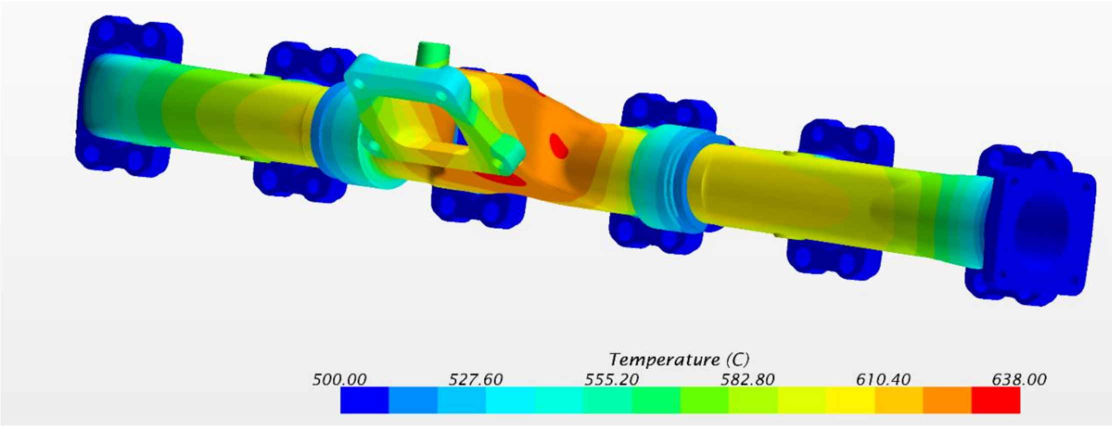


Figure 3.12 Metal Temperature Results comes from Co-Simulation (View1)

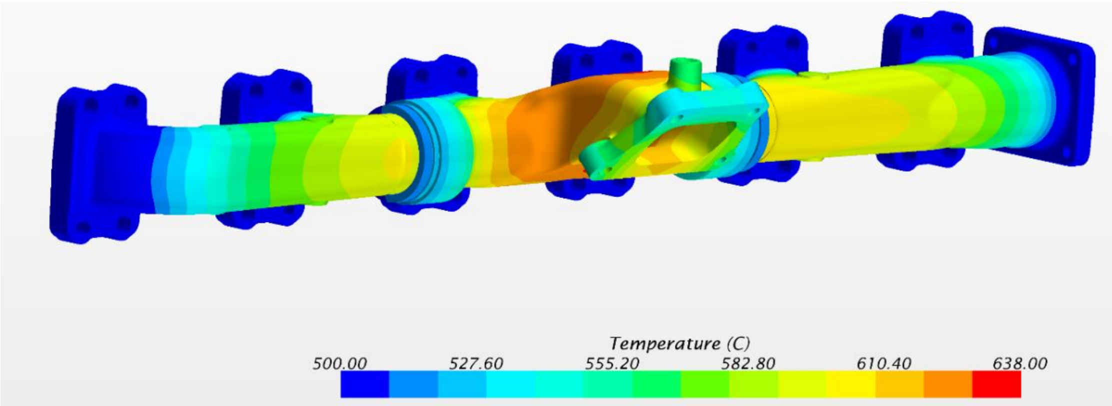


Figure 3.13 Metal Temperature Results comes from Co-Simulation (View2)

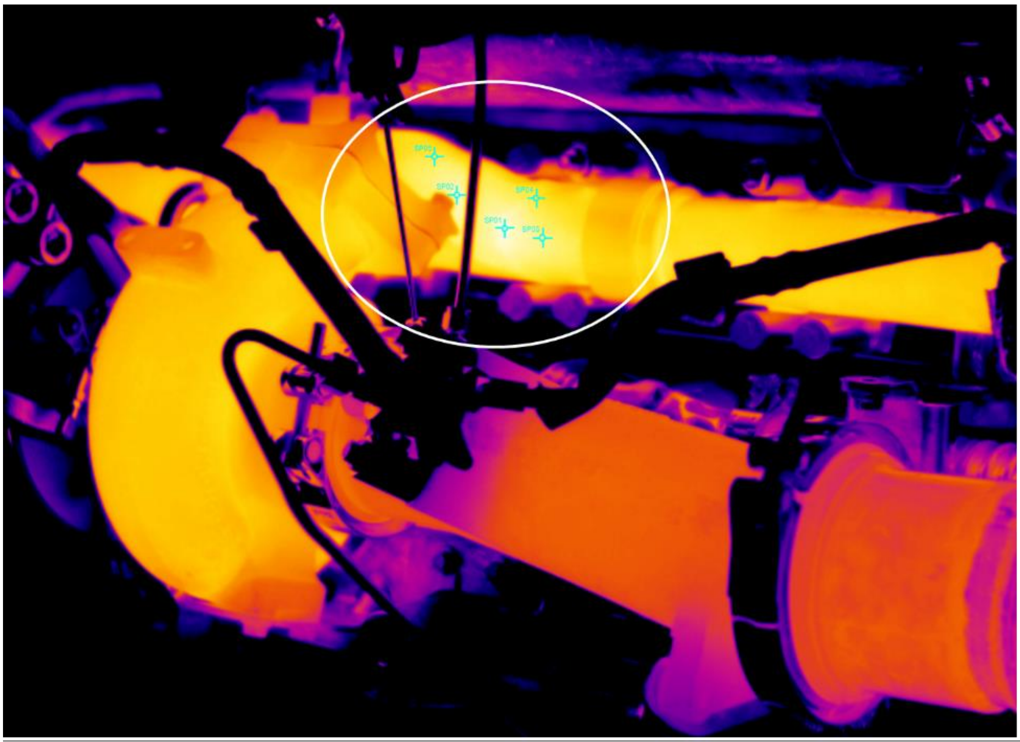


Figure 3.14 Monitored Temperature Points at Thermal Survey Post-Processing

To observe the heat up process of exhaust manifold with thermal camera, carried out post-processing graphically by selecting points onto the exhaust manifold skin and monitored the temperature value.

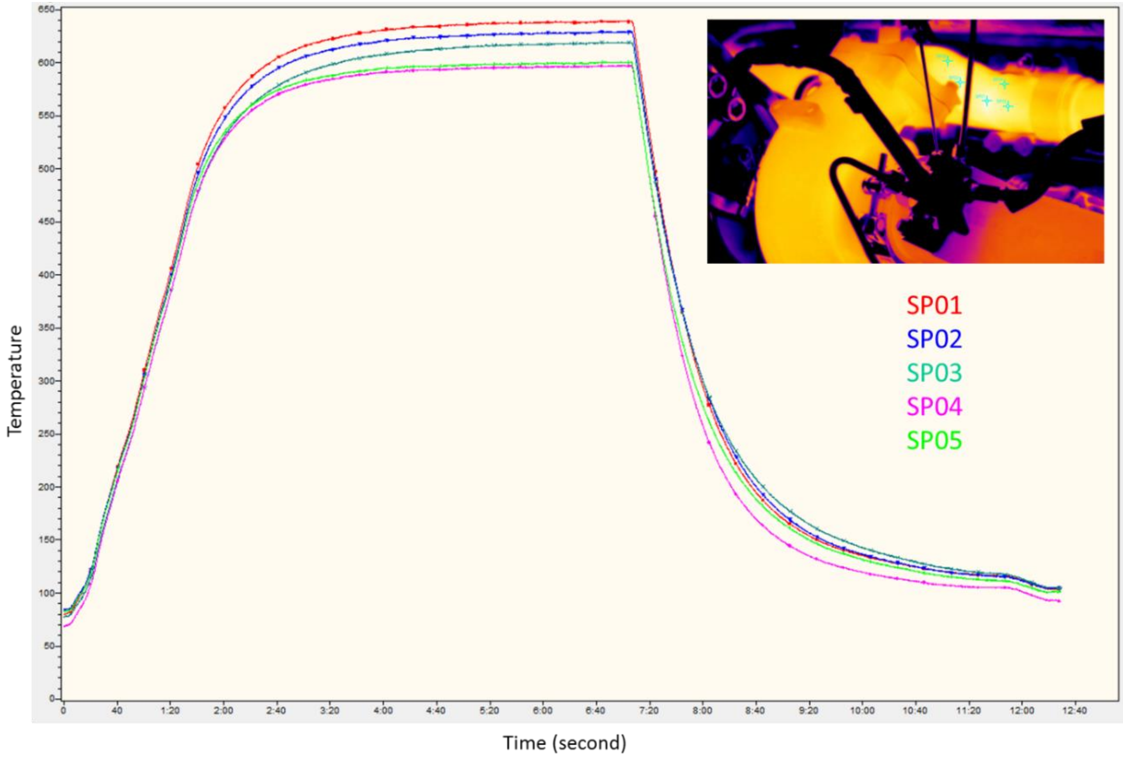


Figure 3.15 Heat Up Behaviour of Measurement Points from Thermal Camera

3.3 Comparison the Thermal Camera and CAE Results

It has been determined 8 points which are critical in terms of temperature to compare the test and CAE results (Sequential Coupling and Co-Simulation method). Listed the temperature variance (delta temperature and percentage) with the test data and CAE results of highlighted points on the table 3.2 below. As seen from the results, sequential coupling method (Method 1) underpredicts up to the %10 in terms of temperature compared with measured data. Co-simulation method overpredicts the tempeature values up to %4 which is close to the experimental result. [15]

Table 3.2 Temperature Deviation between Thermal Camera and CAE Results at Highlighted Points

Temperature Difference	**Method 1**		**Method 2**	
	CAE-Test		**CAE-Test**	
Point Number	**Celcius**	**%**	**Celcius**	**%**
Point 1	-59	**-9,2**	-8	-1,3
Point 2	-50	-7,9	1	0,2
Point 3	-57	**-9,2**	7	1,1
Point 4	-30	-5,0	26	**4,4**
Point 5	-33	-5,5	24	4,0
Point 6	-32	-5,3	23	3,8
Point 7	-39	-6,3	11	1,8
Point 8	-32	-5,3	21	3,5

CHAPTER 4

RESULTS AND DISCUSSION

In this thesis, the theoretical study aims to investigate the feasibility of three different CAE approaches used for the simulation of exhaust manifold fluid flow and the accompanying thermal distribution. To predict the heat up of an exhaust manifold is an significant analysis capability. The problem is challenging due to the pulsating fluid flow inside the engine exhaust manifold which is required to perform each analysis in transient mode. Also computational time should be taken into account while carrying out the analyses. It lasts approximately 5 months to model the entire thermal heat up period of exhaust manifold with transient conjugate heat transfer method, so investigated another alternative methods without compromising on accuracy. It is also realized that different time-scales that are involved in the fluid physics and the heat up of the solid physics.

To validate the outcome CAE solutions, performed an experimental study to measure the manifold skin temperature with thermal imaging camera. The predicted metal temperature is then used to carry out the thermo-structure durability analyses to evaluate the design of HD diesel engine exhaust manifold which is not out of the scope of this study.

The main findings of this thermal assessment study can be summarized as below;

- Star CCM+ gives an opportunity to treat the fluid and solid domains separately by running the analysis with different time scales. One of the most important key point of the Co-simulation approach is to decide the how many engine cycle should be coupled for fluid domain. It has been determined 40 engine cycle as analysis time for fluid simulation at the end of sensitivity study.

- Star-CCM+ Co-Simulation method results which is up to ~4% higher when compared with thermal camera results means good agreement with experimental output.

- Calculation time of Co-Simulation method, applied 40 engine cycles, is about 1/35 of the CHT method. Co-simulation run time lasts 5 days with 24 CPUs for fluid side, 24 CPUs for solid side. In terms of computational run time, Co-Simulation method becomes to appear more reasonable than conjugate approach.
- Compared the computation run time of method1, sequential coupling method run time is approximately 1/75 of conjugate heat transfer analyis.
- Temperature predictions obtained from method1 (Sequential Coupling) are ~9%-10% lower than thermal survey results.
- Time period; temperature results of second method are higher than the first method (up to 50-60 °C), however first method predicts faster warm up period.
- One-sixth of time period, temperature results of third method (conjugate heat transfer approach) is about 25-30 °C lower than compared with other two CAE approaches.
- Star-CCM+ Co-Simulation method stands out with run time saving and obtaining more reasonable metal temperature distribution compared with experiment output.
- Represented the comparison between three three different CAE approach while predicting metal temperature distribution of heavy duty diesel engine exhaust manifold in terms of run time, pre-processing effort and accuracy on table below.

Table 4.1 CAE Thermal Modelling Approaches Comparison Table

Criteria	Sequential Coupling	Star CCM+ Co-Simulation	Conjugate Heat Transfer
Run Time	+	++	+++
Pre-Processing	++	++	+
Accuracy	+	++	+++

REFERENCES

[1] Caterpillar, Exhaust Systems Application and Installation Guide, http://stor-erik.com/wp-content/uploads/2013/03/Exhaust-Systems-LEBW4970-05.pdf, 10 March 2015.

[2] Zhang, X., Luo, Y. and Wang, J. (2011). "Coupled Thermo-Fluid-Solid Analysis of Engine Exhaust Manifold Considering Welding Residual Stresses", Transactions of JWRI, Special Issue on WSE2011.

[3] Smith, T. J., H. J. Maier, H. Sehitoglu, E. Fleury, and Allison, J. A. (1999). "Modeling High Temperature Stress Strain Behavior of Cast Aluminum Alloys", Metallurgical and Materials Transactions, 30A, 133-146.

[4] Hazime, R. M., Dropps, S. H. and Anderson, D. H., Ali M. Y., (2003). "Transient Non-Linear FEA and TMF Life Estimates of Cast Exhaust Manifolds", SAE Technical Paper 2003-01-0918.

[5] Chen M., Wang Y., Wu W., Xin J., (2014). "Design of the Exhaust Manifold of a Turbo Charged Gasoline Engine Based on a Transient Thermal Mechanical Analysis Approach", SAE Technical Paper, 2014-01-2882.

[6] Gocmez, T., (2014). "Exhaust Manifold Training: Design, Calculation and Testing for Ford Otosan", Training Notes 2014-02-20.

[7] Gocmez, T., Deuster, U., (2009). "An Integral Engineering Solution for Design of Exhaust Manifolds", SAE Technical Paper 2009-01-1229.

[8] Liu, Z., Li, X., Hou, X., Wang, C., Wu Yan, F., (2013). "Numerical Simulation for Exhaust Manifold Based on the Serial Coupling of STAR-CCM+ and ABAQUS", Research Journal of Applied Sciences, Engineering and Technology 6 (20): 3903-3909.

[9] Mamiya, N., Masuda, T. and Noda, Y., (2002). "Thermal Fatigue Life of Exhaust Manifolds Predicted by Simulation", SAE Technical Paper 2002-01-0854.

[10] Fan, Q., Kuba, M., Nakanishi, C. , (2004). "Coupled Analysis of Thermal Flow and Thermal Stress of an Engine Exhaust Manifold", SAE Technical Paper 2004-01-1345.

[11] Star-CCM+ Version 10.02 User Manual.

[12] Brown, A. , https://steve.cd-adapco.com/articles/en_US/Video/STAR-CCM-to-STAR-CCM-Co-Simulation-where-both-Fluid-and-Solid-simulations-are-transient-in-nature, 20 August 2016.

[13] Bakker, A., Applied Computational Fluid Dynamics Lecture 13: Heat Transfer, http://www.bakker.org/dartmouth06/engs150/13-heat.pdf, 08 July 2016.

[14] FLIR, Thermal Imaging Guidebook for Building and Renewable Energy Applications,http://www.flirmedia.com/MMC/THG/Brochures/T820325/T820325_EN.pdf, 15 July 2016.

[15] Eroglu, S., Duman, I., Ergenc, A.T., Yanarocak, R.,(2016). “Thermal Analysis of Heavy Duty Engine Exhaust Manifold Using CFD”, SAE Technical Paper 2016-01-0648.

CURRICULUM VITAE

PERSONAL INFORMATION

Name Surname	: İpek DUMAN
Date of birth and place	: 15/04/1989-Bursa
Foreign Languages	: English
E-mail	: ipekduman89@gmail.com iduman1@ford.com

EDUCATION

Degree	Department	University	Date of Graduation
Undergraduate	Mechanical Engineering	Yıldız Technical University	2012
High School	Math &Science	Nişantaşı Nuri Akın Lisesi	2007

WORK EXPERIENCE

Year	Corporation/Institute	Enrollment
2012- …	Ford Otosan Engineering Center	Product Development Engineer
2011-2012	Arçelik Research&Development Center	Part-time Project Student

PUBLISHMENTS

Papers

[1] Eroglu, S., Duman, I., Ergenc, A.T., Yanarocak, R.,(2016). “Thermal Analysis of Heavy Duty Engine Exhaust Manifold Using CFD”, SAE Technical Paper 2016-01-0648, 2016.