SIEMENS
Ingenuity for life

Automotive and transportation

Ford Otosan

Automotive manufacturer uses Simcenter STAR-CCM+ to rapidly produce better diesel engine designs

Product

Simcenter

Business challenges

Reduce the number of manufacturing prototypes early in the design phase

Produce accurate simulations early on and factor in heat transfer of multiple engine components

Use test and simulation to improve accuracy of temperature distribution

Keys to success

Use Simcenter STAR-CCM+ to improve simulation accuracy

Use co-simulation to couple multiphysics simulations with different time scales to achieve faster turnaround

Results

Used co-simulation capability in Simcenter STAR-CCM+ to rapidly produce better designs

Assessed the performance of engine subsystems and determined the loads and shape of the components and systems affected by complex multiphysics

Simulated multiple solid and fluid regions quickly and accurately

Siemens Digital Industries Software solution enables Ford Otosan to simulate multiple solid and fluid regions quickly and accurately

Using simulation early

Designing the next generation of highly efficient diesel engines requires rapid, robust and reliable performance predictions. This has led the powertrain computer-aided design (CAD)/computer-aided engineering (CAE) team at Ford Otomotiv Sanayi A.S. (Ford Otosan) to use Simcenter STAR-CCM+™ software to analyze the thermal exchange between hot engine gases and various critical solid engine components.

Using simulation-driven development and design has been crucial to the success of Ford Otosan's heavy duty engine, the Ecotorq 13 liter (L)/9L, with European Union (EU) 3, EU5 and EU6 variants. It is costly and time consuming to manufacture prototypes for testing before more mature designs are ready, so it is important to increase the use of simulation as early as possible in the design process.

Modern diesel engines are complex assemblies with tight packaging constraints, and due to higher temperatures as a result of turbocharging, they have largely nonuniform temperature distributions, which are challenging to predict. This need for increased simulation accuracy has led to the rise of multi-component, multiphysics computational fluid dynamics (CFD) simulation as a fundamental requirement for modern diesel engine design. To this end, Ford Otosan used co-simulation on two critical engine components, the engine exhaust manifold and the piston head, to see how it could help them achieve their design goals.

As Sinan Eroglu, the CFD supervisor of the powertrain CAD/CAE group, explains, "Using Simcenter STAR-CCM+ co-simulation enables coupling of multiphysics simulations with different time scales ranging from microseconds to thousands of seconds, providing faster and more accurate analyses and shorter turnover times for development and assessment of complex designs."

Engine exhaust manifold simulation
The function of the exhaust manifold is to bridge the gap between the engine structure and the exhaust after-treatment systems and allow the burnt in-cylinder gases to flow through it. Exhaust manifolds are designed and developed to provide smooth flow with low back pressure, while being able to withstand extreme temperatures.

"The aim is to come up with a durable manifold design, so accurate modeling of the warmup time is extremely important," says Eroglu. "It is a typical multiphysics problem in which there is a strong interaction between the fluid and solid domains, so the solvers for each of these have to be coupled. The main challenge arises from the nature of the pulsating flow behavior within the manifold, requiring a transient analysis to be conducted. The time step of this flow is on the order of 10 microseconds, and as the warmup period is 10 minutes, it is not feasible to run with this approach, so we sought alternative co-simulation methods."

The three approaches to the exhaust manifold shown in figure 1 were assessed.

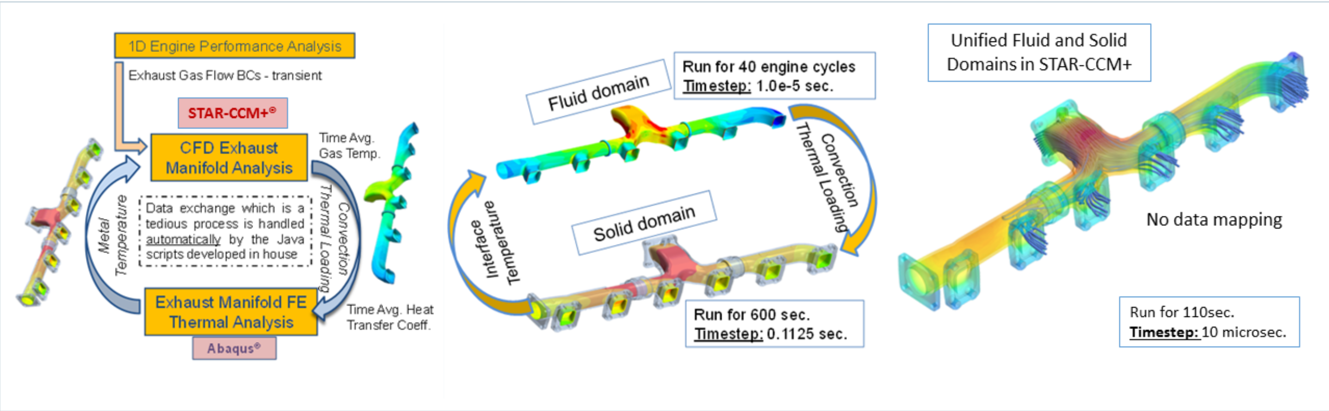


Figure 1: Coupling strategies: left – sequential coupling; middle – co-simulation; right – conjugate heat transfer.

Sequential coupling method

The first approach, which sequentially couples the finite volume solver in Simcenter STAR-CCM+ with the finite element solver in Abaqus, is driven by exchanging data between the two codes at the interface boundary of the fluid and solid domains. This allows a more complete structural and thermal stress analysis to be carried out on the manifold. The two models are pictured in figure 2. The data exchange is handled by internally developed Java scripts to conduct the process automatically. Each CFD run is conducted at the rated power condition for three engine cycles (2160° crank angle) with boundary conditions at the inlets and outlets provided by the one-dimensional engine simulation. At the end of the third CFD engine cycle, thermal load data (heat transfer coefficients and reference temperatures) are time averaged and mapped onto the finite element model (FEM), which is then run for 600 seconds (sec).

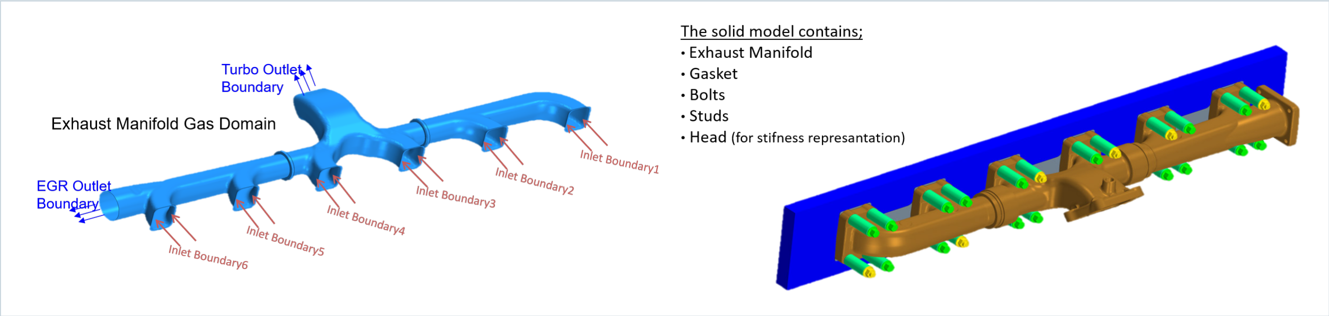


Figure 2: Fluid and solid models in Simcenter STAR-CCM+ and Abaqus.

This data is then fed back into the CFD model, updating the thermal distribution at the interface boundary, and the simulation is continued. Because of the data exchange between the fluid and solid models, the primary concern is temperature convergence, or at what point each separate fluid and solid model is up-to-date and energy is conserved throughout the whole system. This occurred after the third data exchange, as shown in figure 3.

"Using Simcenter STAR-CCM+ co-simulation enables coupling of multiphysics simulations with different time scales ranging from microseconds to thousands of seconds, providing faster and more accurate analyses and shorter turnover times for development and assessment of complex designs."

Sinan Eroglu
CFD Supervisor, Powertrain CAD/CAE Group
Ford Otosan

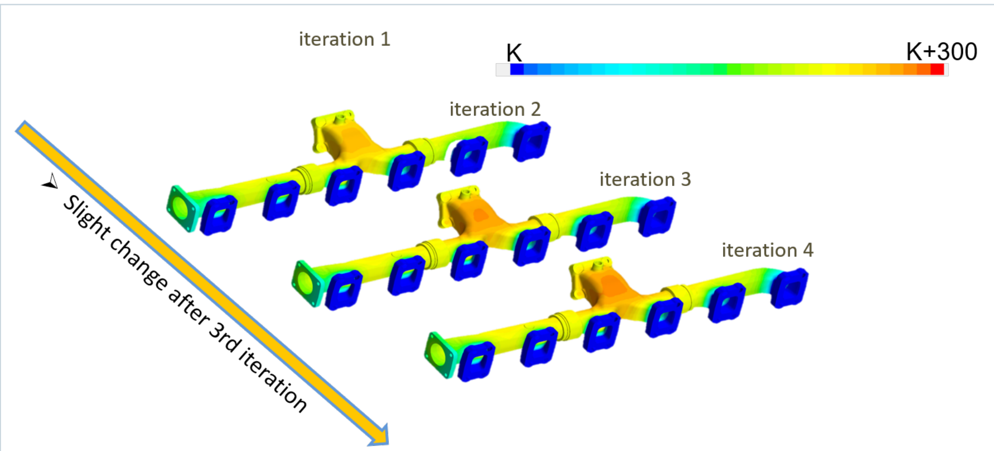


Figure 3: Simcenter STAR-CCM+ to Abaqus thermal data exchange convergence.

Co-simulation method

For cases in which a full thermal stress analysis is not required, a Simcenter STAR-CCM+ to Simcenter STAR-CCM+ co-simulation can be used: here, the finite volume method is used for both fluid and solid models to predict the thermal distribution. This allows the data transfer to be handled solely in Simcenter STAR-CCM+, permitting a more direct data transfer. Meanwhile, the simulation runs rather than transferring the averaged thermal load at the end of a complete cycle as is the case with the Simcenter STAR-CCM+ to Abaqus approach. In this instance, the data was transferred every five fluid time steps. Time steps of 10 microseconds (microsec) and 0.1125 seconds, respectively, were

"For the 10-minute exhaust manifold warmup case, the Simcenter STAR-CCM+ to Simcenter STAR-CCM+ co-simulation takes slightly longer to run than external coupling, but it gives more accurate results."

Sinan Eroglu
CFD Supervisor, Powertrain CAD/CAE Group
Ford Otosan

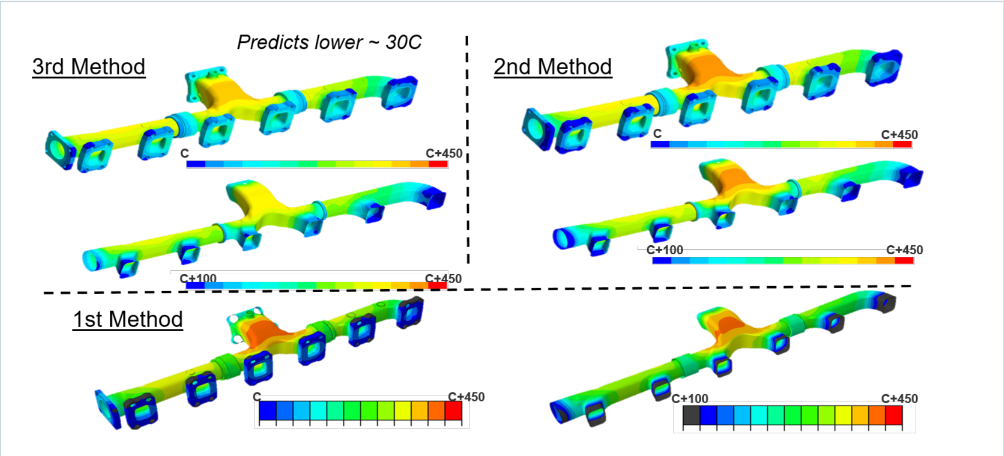


Figure 4: Temperature results at 110 seconds.

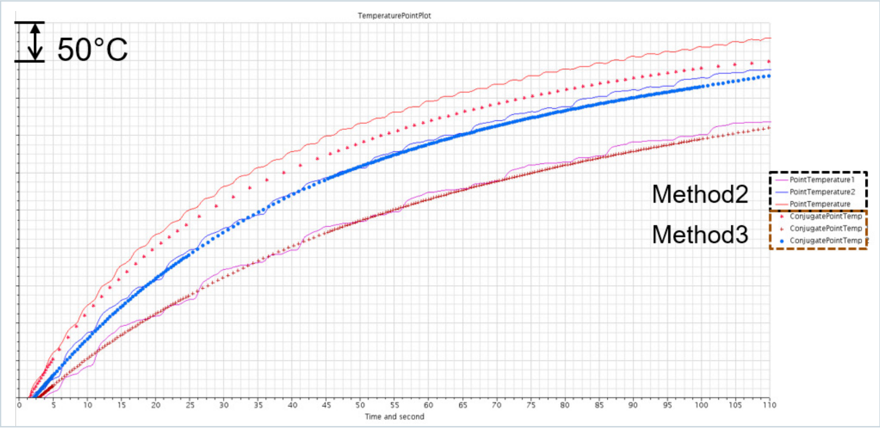


Figure 5: Warmup time for co-simulation versus the CHT method.

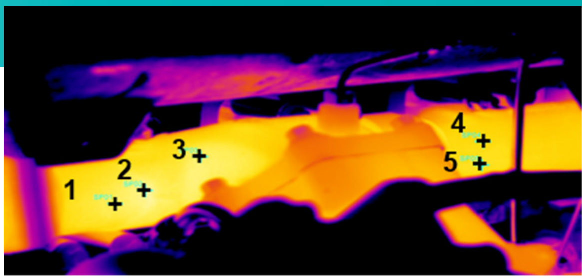
used in the fluid and solid models. There were 5,333 data exchanges between the fluid and solid models in order to ensure convergence.

The conjugate heat transfer method

Conjugate heat transfer (CHT) simulations are the most direct approach because the fluid and solid models are solved in a single Simcenter STAR-CCM+ simulation simultaneously, avoiding any data mapping. In these cases, the fluid domain time step takes precedence so that the flow structures in the manifold are resolved. The time step of 10 microseconds was applied to the entire model, which would have led to substantial run times for the solid model as it does not require such high temporal fidelity. Because of this, the CHT method had a 110-second time limit imposed on it.

Comparing methods

The three approaches are compared over the 110-second time limit in figure 4. In the neck region, the temperatures in the sequential (method 1) and co-simulation (method 2) approaches are over predicted compared to the CHT approach (method 3). Temperature time history plots at points randomly located on the manifold, shown in figure 5, demonstrate a 20° Celsius (C) to 25° C discrepancy between method 2 and 3 near the neck region, showing that method 2 has over predicted the temperature. However, the rate of warmup shows good agreement between the two.



Point	Method 2 CAE Test	%	Method 1 CAE Test	%
1	-8	-1.3	-59	-9.2
2	1	0.2	-50	-7.9
3	7	1.1	-57	-9.2
4	26	4.4	-30	-5
5	24	4	-33	-5.5
6	23	3.8	-32	-5.3
7	11	1.8	-39	-6.3
8	21	3.5	-32	-5.3

Figure 6: Thermal camera results at 600 seconds.

Comparing the physical test data from engine dynamometer testing in figure 6, thermal imaging results indicate there is, at worst, a 4 percent over prediction in temperature as a result of method 2 and an average over prediction of only 1.7 percent. However, comparing the method 2 thermal results to the physical data shows at worst a 9.2 percent under prediction and an average 7.3 percent under prediction. This shows the thermal analysis coupling approach conducted solely in Simcenter STAR-CCM+ is overall in close agreement with physical test data, although it is marginally conservative.

Piston cooling simulation

Given the close agreement of the co-simulation method in Simcenter STAR-CCM+ for fluid-to-solid thermal analysis, this methodology was carried out to analyze the effect of the oil jet aimed beneath the piston in order to cool it down. As shown in figure 7, the oil cooling jet is aimed beneath the piston head into the piston cooling gallery so that it is closer to the critical heat source of the piston crown where combustion thermal loads can act on it. The channel-like nature of the cooling gallery allows longer residence time of the oil, which aids heat transfer.

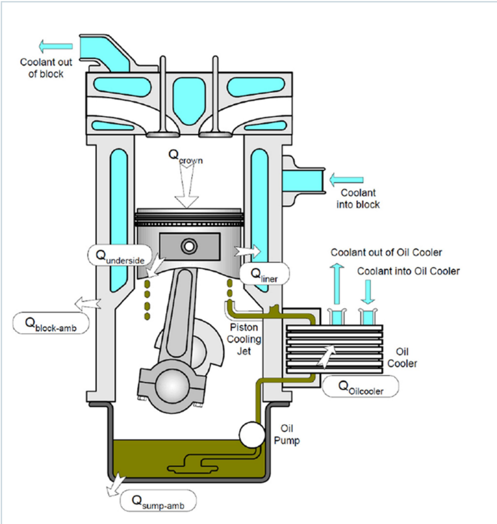


Figure 7: Schematic of piston cooling jet.

As technical specialist Serdar Güryuva points out, "As the specific power rating of diesel engines are increased by high pressure fuel injection and higher turbocharging pressures, thermal durability of the piston is increasingly important. Things such as lubricant quality deterioration – like coking, thermal cracking, carbon deposition, ring sticking and micro welding – are of concern. The lubricating oil serves as a secondary cooling medium for pistons to limit the temperature. It does so by the cocktail shaker effect inside the oil gallery, as the oil penetrates it and provides localized cooling to the piston head."

"The main challenge arises from the nature of the pulsating flow behavior within the manifold, requiring a transient analysis to be conducted. The time step of this flow is on the order of 10 microseconds, and as the warmup period is 10 minutes, it is not feasible to run with this approach, so we sought alternative co-simulation methods."

Sinan Eroglu
CFD Supervisor, Powertrain
CAD/CAE Group
Ford Otosan

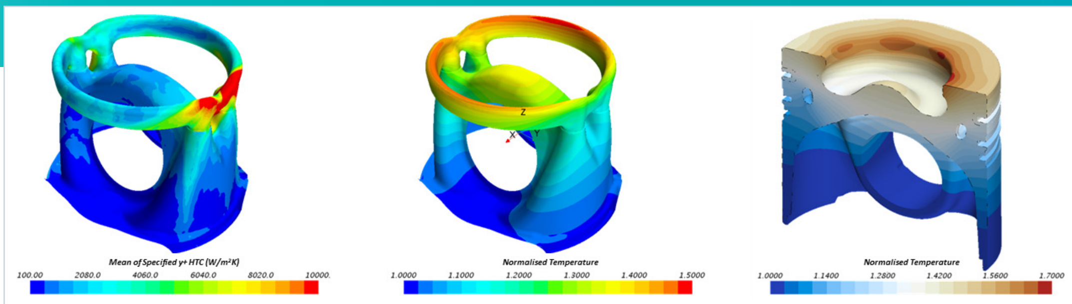


Figure 8: Heat transfer coefficient and normalized temperature on the fluid side (left and middle) and normalized temperature on the piston solid surface (right).

The Simcenter STAR-CCM+ co-simulation was carried out on a piston model that had the correct motion applied to it using piston velocity input data with constant temperature oil properties. An additional model was run with a static piston head and a relative motion applied to the inlet oil cooling volume so that no mesh motion was required. Only the inner liner, oil cooling jet nozzle and piston head were considered; the crankshaft and connecting rod were removed. Solid-to-fluid model data exchange occurred every two piston cycles, whereas in the last full cycle mean convective heat transfer was mapped from fluid to the solid. The thermal results from the Simcenter STAR-CCM+ co-simulation were compared against physical testing data at various sensor locations.

Results for the piston with the correct motion applied to it are pictured in figure 8, which shows the heat transfer coefficient and normalized temperature (as referenced with the crankcase temperature, T0=Temp/Tempcrank) for the fluid-to-solid interface, and the temperature on the outer piston head surface. This shows the high heat transfer in which the oil jet initially impinges on the oil gallery on the right hand side, and coincides with the coolest temperature within the oil gallery: T0=1.25.

As the oil moves around the gallery, the heat transfer is reduced and the temperature begins to rise again. For the complete solid piston model, the highest temperatures are inside the rim of the piston bowl, where the in-cylinder combustion thermal loads are acting. However, it is noticeable the white contour of T0=1.5 corresponds with the oil gallery. The correlation of the physical test results for both types of piston motion (full motion and static with oil jet relative velocity) are shown in figure 9. The results indicate good agreement between the physical test and the Simcenter STAR-CCM+ co-simulation.

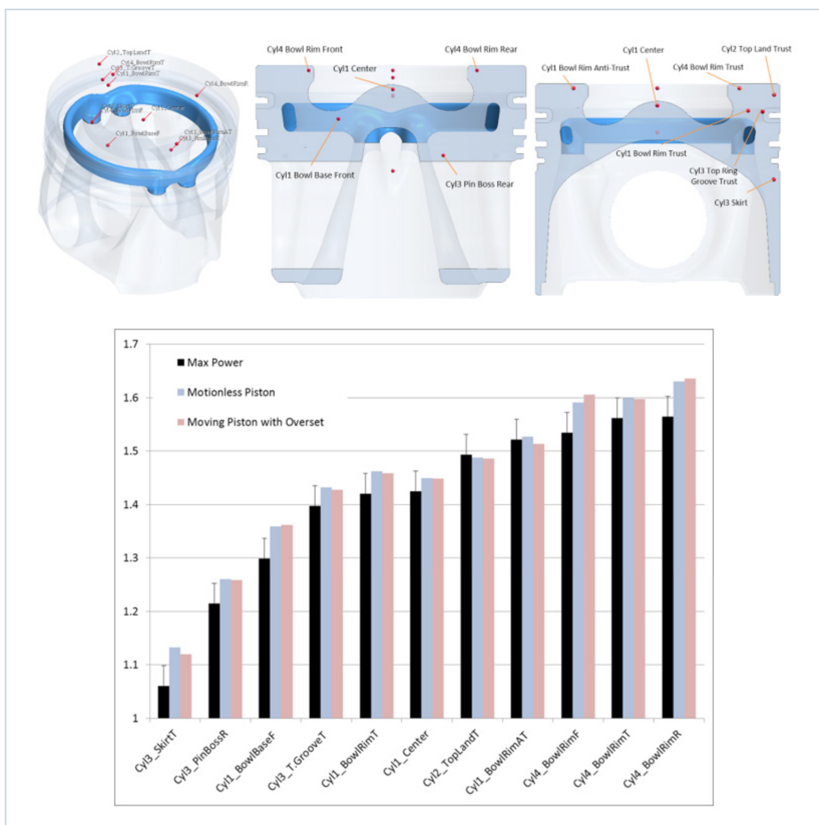


Figure 9: Temperature probe point comparison between CFD and physical test.

Solutions/Services

Simcenter STAR-CCM+
https://mdx.plm.automation.siemens.com/star-ccm-plus

Customer's primary business

Ford Otomotiv Sanayi A.S. (Ford Otosan) was founded in 1959 as a joint venture between the Ford Motor Company and Koc Holding. It is based in Turkey with five facilities employing 12,000 people producing the Ford Transit, the Transit/Tourneo Courier and Ford cargo trucks.
www.fordotosan.com.tr/en

Customer location

Gölcük
Turkey

Achieving better designs faster

With the higher thermal management demands being placed on engines, it is readily apparent that accurately simulating multiple solid and fluid regions can be handled quickly and accurately with Simcenter STAR-CCM+. This is especially true of problems in which highly nonuniform distribution of temperature and heat transfer coefficients exist, such as the exhaust manifold and piston head cooling simulations conducted by Ford Otosan.

Summarizing the results from these simulations, Eroglu says, "For the 10-minute exhaust manifold warmup case, the Simcenter STAR-CCM+ to Simcenter STAR-CCM+ co-simulation takes slightly longer to run than external coupling, but it gives more accurate results."

Due to the level of agreement in the oil jet cooling results, Güryuva is already aiming at a larger model: "As the piston wall temperatures affect the convective thermal loads, it is necessary to run a CHT analysis for the complete head and block system."

Sinan says, "The future of the diesel engine focuses on efficiency due to forthcoming CO2 regulations, resulting in the necessity to develop engines that withstand higher thermal and structural loads. Simcenter STAR-CCM+ provides the opportunity to accurately assess the performance of several engine subsystems and determine the loads and shape of the components and systems that are affected by complex multiphysics."

The bottom line is by using the co-simulation capability in Simcenter STAR-CCM+, Ford Otosan is able to achieve better designs faster.

"Simcenter STAR-CCM+ provides the opportunity to accurately assess the performance of several engine subsystems and determine the loads and shape of the components and systems that are affected by complex multiphysics."

Sinan Eroglu
CFD Supervisor, Powertrain CAD/CAE Group
Ford Otosan

Siemens Digital Industries Software

Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3333

66266-C4 1/18 A