**WDWCWWCA Method for Designing and Studying Engine Intake System Based on CAD/CAE/CFD Integration 2011**

The similarity principle and pressure wave theory, the structure parameters of target engine intake system were defined referencing the similar engine. Then, the intake system parameters were input into GT-power, and the intake system performance was simulated and optimized. Next, the 3-D structure model was designed based on CAD software and optimized values. After that, the key parts of intake system were analyzed and optimized by using CAE and CFD. As a result, a new intake system has been designed. The whole processes of designing and studying intake system included parameters define, performance optimization, structure design, flow calculation, noise and vibration analysis, were coupled together. And the feasibility and effectiveness of the idea was validated through practical example. The study results have provided guidance for designing intake system of engine.

**Numerical study of fluid flow and effect of catalytic converter volume in optimisation of diesel oxidation catalyst in a CI engine using CFD 2013**

Catalytic converter has become a necessity to achieve low emissions in all the C I engine driven vehicles. The design of catalytic converter has become critical which requires a thorough understanding of fluid flow inside the catalytic converter. In the present work, an attempt has been made to design and optimise the Close Coupled Catalytic converter using commercial Design software (IDEAS) and CFD tool (CFX). Catalytic converter has been designed for 1.4 L, 0.8L and 1L to achieve BS IV Norms. CFD study has been conducted for all three designs. The substrate region was modelled as a porous medium. The governing equations namely conservation of mass, momentum were solved for analysis. The predicted numerical results were validated with those available in literature. The analysis involved determining back pressure across the converter system for a given mass flow rate. The numerical results of Pressure drop, Space velocity, Velocity profile and Uniformity index were studied to optimise the volume of Catalytic converter.

**Transient CFD simulation of gasoline intake characteristics 2011**

It is clear that flow characteristics are crucial for reflecting intake performance of gasoline engine. Based on AVL Fire, 3-D and transient flow in the port and cylinder has been simulated by using CFD code for a 476Q gasoline engine. Boundary conditions, the variable pressures, were employed by 1-D simulation with using Boost software. The effects of flow coefficient and turbulence kinetic energy on intake performance were investigated. A correlation between intake loss coefficient and flow coefficient was presented by the principles of fluid mechanics. Furthermore, the flow coefficient was discussed at various valve lifts. The result shows that the transient CFD simulation can truly reflect the intake characteristics. The intake loss coefficient and flow coefficient are related with geometric parameters of intake port. In addition, high turbulence kinetic energy is associated with the rotational pattern of flow velocity vector.

# Computational Fluid Dynamics Technology Applied in Flow Analysis in Diesel Engine's Cooling Water Jacket 2010

# Using computational fluid dynamics software, the fluid flow in cooling water jacket of HPD diesel engine is studied. Results show that all parts in A Row cylinders cooling water jacket have a better cooling effect, such as in cylinder block and in fire deck. And cooling effect for each cylinder is uniform. Total pressure loss through cooling water jacket is 32.6kPa. The size of water up-holes is better for cooling cylinder head, which can also provide boundary conditions for further calculating temperature field of cylinder head.

# CFD Technology Used to Optimize Fuel Injector Design of Railway Diesel Engine 2010

In order to improve the performance of 16 V280 diesel engine, the three-dimensional numerical models of the flow field in the nozzle which belongs to the injection system was built, and different detailed flow structures was captured under different pressure conditions. Through the analysis of the flow field, the unsuitable position of the nozzle was located, and the methods which combine CAD / CAM / CFD were used to optimize the structure of the nozzle. The analysis results indicate that with the increase of the injection pressure; the flow turbulence intensity increase; the more energy get lost, which caused by turbulence; the coefficient of flow decrease. After the spray nozzle structure was changed into sphere pin valve, the current capacity under the same level of injection pressure was improved greatly.

# CFD analysis of fluid flow and heat transfer of an automotive radiator with nano fluid 2013

Internal Combustion engines in the automobile applications are becoming highly power-packed with increasing power to weight and/or volume ratio. Further, the space available under the bonnet is also decreasing due to the ever increasing demand of small cars by the customers. Nano fluids are a new class of heat transfer fluids engineered by dispersing nanometre - size solid particles in traditional heat transfer fluids. In the present study a Nano fluid is used as a coolant in a radiator model and is analysed for evaluating the fluid flow and heat transfer characteristics. A radiator model is modelled in CATIA modelling software and is meshed using a pre-processing software GAMBIT. Three volumes are created i.e., coolant, radiator and air and overall pressure, temperature and velocity distribution of coolant and air are analysed and presented by using a Computational Fluid Dynamics (CFD) environment software FLUENT. Results have shown that the rate of heat transfer is better when Nano fluid (Al2O3 + Water) is used as coolant, than the conventional coolant. Velocity distribution graphs shown that the radiator design have to be optimized to eliminate water stagnation.

# Heat exchanger heat transfer coefficient and CFD modeling 2019

Heat exchanger performance assessment with a high precision level is one of the key success factors for OTEC. Evaluating heat transfer coefficient with high precision in OTEC working conditions - specifically low heat flux, low mass flux, and large dimensions - is not possible by applying existing methods, models, and heat transfer laws. Initial analysis shows that heat transfer is driven by thermal and hydraulic parameters, which are linked, and therefore require 2D or 3D numerical models to be accurately modelled.Naval Energies and Naval Group have therefore defined and applied a heat transfer qualification method to correlate a 3D CFD thermo-hydraulical model based on ANSYS Fluent code with reduced scale testing on a water/ammonia flooded shell & tube evaporator. The results meet the high expectations as an uncertainty of less than 10% was achieved on the shell side heat transfer coefficient of the flooded shell & tube evaporator in 2-phase flow conditions, lowering the uncertainty of Rankine Cycle electrical production to 2%.

# Measurements and CFD Modeling of Temperatures in the Engine Compartment of a Hybrid Electric Vehicle 2017

In this article the temperature distribution within the engine compartment of a hybrid electric vehicle is experimentally and numerically investigated. The aim of this study is to develop a simulation model that captures the thermal behaviour of the electrical components for different driving conditions. For the experimental part, temperature sensors are placed at various locations inside cooling hoses as well as on the hoses and on various components. Using the commercial computational fluid dynamics (CFD) software, a complete vehicle simulation is set up for the same model. A comparison between the measurements and the numerical results shows good results. The increase in cooling media temperature when passing through the CIDD (Combined Inverter and DC/DC converter) is determined with a 10% deviation, also the CIDD surface temperatures are well predicted. For the Electric Rear Axle Drive (ERAD) the surface temperatures lie within the requested interval for the majority of measurement points, especially on the exterior of the cooling channel around the electric machine.

**Study on combustion and NOx emission characteristics of a DI diesel engine operating on pistache seed biodiesel/diesel and methanol/diesel blends 2011**

The experiments of the emissions of diesel engine fueled with the pistache seed biodiesel diesel fuel and methanol - diesel fuel were performed on YTR3105 diesel engine. The NOx emission and combustion characteristics of the engine fueled with the two kinds of blend fuels were investigated, the adiabatic flame temperature was calculated. The effects of burning delay, burning duration and air-fuel ratio on NOx emission were discussed. The results show that the NOx emission of the two kinds of fuels increases with the increase of engine load. With the increase of the blending ratio, the NOx emission of engine fueled with pistache seed biodiesel and the pure diesel engine are equal to or slightly increase. The NOx emission of methanol - diesel fuel decreases with the increase of the blending ratio. The adiabatic flame temperature of Methanol - diesel fuel is far lower than the pistache seed biodiesel - diesel.

**Numerical simulation on effects of nozzle hole cone angle on combustion and emissions in a diesel engine 2011**

The combustion process and emission characteristics in diesel engine mainly depend on the mixture formation in cylinder. Spray characteristic is an important factor which affects the mixture formation and combustion process. Further, nozzle hole cone angle of the fuel injector directly influence the spray characteristic. In this paper, flow evolution process, spray distribution, fuel/air ratio equivalence ratio distribution, temperature distribution and emissions formation are analyzed. The results show that: mixture formation quality can be improved if selecting the nozzle hole cone angle appropriately, and therefore combustion process will be improved and emissions will be reduced.

**Effect of Exhaust Gas Recirculation on Homogeneous Charge Compression Ignition combustion model in Diesel Engine 2016**

In this study, the modeling of the effect of exhaust gas recirculation on Homogenous Charge Compression Ignition (HCCI) in Diesel Engine has been investigated. The efficiency and emission reduction of the engine were an important point of view from study charge compression ignition. Furthermore, the greater adoption of the diesel engine has been inhibited by the increasing public awareness on pollution as well as the rising cost associated with the relatively high number of particulate matter (PM) and oxides of nitrogen (NOx) of its emission. The increase of compression-ignition (CI) engine fuel efficiency accompanied by lower emissions has encouraged intensive studies on the improvements of the diesel engine. In this paper, we describe the recent progress achieved in the research of advanced combustion and fuels to improve the fuel efficiency of diesel or CI engines. An Exhaust Gas Recirculation (EGR) system was arranged to be fitted in the engine. The study on the prospect of EGR application in lowering the engine-out NOx emissions in a heavy-duty diesel engine has been conducted by employing a refined 1D fluid-dynamic engine model generated in Boost Software Tools and the control of engine cycle parameters. The conclusion show that the engine with an opening valve of exhaust gas recirculation 0.3 and a compression ratio of 20 generated the highest volumetric efficiency.

**Research on fuel injection strategy of start process for the electronic diesel engine 2014**

The start process of multi-cylinder diesel engine has been experimental investigated in this paper. With a fixed injection duration, the start experiment is performed with different injection timing. Then, with a fixed injection timing, the experiment of the starting moment is performed with different injection duration. Based on the above experiment data and the calculation method, the matching strategy for the transmission system is put forward as well as corresponding injection strategy.

**Modal analysis of the cylinder block of 4D25G diesel engine 2011**

The 3-D models of the cylinder block of 4D25G Diesel Engine, were modeled with Pro/Engineer and reasonably simplified. Elements were applied to mesh 3-D geometric models, and then the FEM model was constructed. Using the model, a modal analysis was done to get the basic vibration mode and the nature frequency of the cylinder block. Through analysis for results, we can learn dynamic characteristic of the cylinder block. It can provide the direction in which the vibration and noise could be further reduced through the structural improvement.

**Technologies of emissions control for NOx and PM on diesel engine 2011**

With the increasingly serious problems of environmental pollution, the technologies of diesel engine are confronted with great challenges on reducing emissions to meet stringent emission legislations. As the major pollutants emitted by diesel engines, it becomes a very important issue to reduce the amount of NOx and PM. Based on pre-treatment and post-treatment principles, the paper mainly introduces several technologies to reduce the emissions of NOx and PM. These technologies include CO 2 in-cylinder injection, using oxygenated fuels, NIS and MK et al.

**Investigation of the effects of different diesel fuel cetane numbers on exhaust emissions in a single cylinder direct injection diesel engine 2017**

There are limited studies to reveal the effect of local diesel fuel types especially in Kurdistan region of conducted to evaluate the effect of different types of diesel fuel having different cetane number, which are from Kirkuk, Dura, Basra refineries and with ethanol blended diesel fuel 8%. The other samples of diesel fuel are from crude oil in TaqTaq oil field in Koya, refined in both Kar and Bazyan refineries. For this purpose the fuel with different value of cetane number were tested in a direct injection diesel engine at variable speed and load. The study showed that the concentration emission of co, NOx and HC are reduced by 6%, 7% and 16%, respectively, while CO2 are increased by 3% for fuel cetane number 50. The reason of decreasing emissions are due to complete combustion of these fuel. Result also show that Koya diesel fuel gives the worst harmful emissions of CO. The result shows that the exhaust emission of co is in general decreased with increasing cetane number, and its value is increased at maximum engine torque for all engine speeds. Ethanol blended fuel with cetane number 54 gives minimum emission of CO at engine speed n= 2200 rpm. NOx emissions of the higher cetane number fuel are decreased by 20% and co of this fuel is reduced to 58%. In general the ethanol blended fuel gives slightly lower HC and CO.

**Automation of diesel engine test procedure 2016**

Leading engine manufacturers carry on investigations and R&D work to improve reliability and durability of internal combustion engines (ICE), particularly, diesel engines. Diesel engine examination and testing are the main methods for verifying manufacturing quality of parts and assembly components, units and engine in whole, accuracy of assembling, correspondence of main diesel engine characteristics to the requirements of technical specifications. The types of diesel engine test procedures are regulated by the state standards (GOST) and international standards (ISO), which define the procedures for engine commissioning and requirements to engine performance standards. Manufacturers continue to improve the construction of engines and performance indicators even after their commissioning and installation. A current diesel engine test procedure is a complex and time-consuming process that can be compared with experimental studies. For this reason, automation systems for engine testing (AST) are created. The need for constant improvement of performance standards of diesel engines raise the costs with respect to test procedures in the course of development of new engine prototypes. In particular, high costs are associated with a mismatch between a level of automation of manufacturing and R&D works. Therefore, automation of test procedures is one of the main goals to be achieved in order to improve the level of technology at production and quality of manufactured diesel engines.

**Effect of altitude height on combustion noise of DI-diesel engines 2011**

Taking the YN4100QB diesel engines as the research object, acoustic power of the radiation noise from the surface of the engine has been tested and calculated, which utilizes the measuring method of superficial acoustic pressure levels based on the simulation system of atmospheric pressure. The engine's combustion noise, mechanical noise and inlet noise have also been successfully separated with the experiment of the noise source identification of diesel engines. All the measures are taken to probe into the influencing mechanism of atmospheric pressure on combustion noise of DI-diesel engines. Comparisons are made about the difference of combustion noise in diesel engines so as to obtain its variation regulation with different altitudes.

**Experimental Study on Plateau Matching Performance of Turbocharger and Vehicle Diesel Engine 2010**

In order to improve the plateau adaptability of CY4102BZLQ-A2A turbocharged diesel engine, the matching performances of the diesel engine with TB28 and HP60 turbochargers at different altitudes (0 m, 2,000 m, 3,000 m, 3,500 m, 4,000 m) were investigated on the engine test bed for simulating high altitude (1ow atmosphere pressure). Results show that HP 60 turbocharger is more suitable to match the engine in terms of engine performance, intake air characteristics, turbocharger rotational speed etc. TB28 turbocharger is over speed when the engine runs at 3,500 m altitude and 2,600 r/min rotational speed, while HP60 turbocharger is over speed when the engine runs at 4,000 m altitude and 2,800 r/min rotational speed. No matter which turbocharger is selected to match the engine, the performance of the engine will deteriorate at different extents. The intake air flow mass and the air-fuel ratio decrease by 7.1-10.5% and 7.2-10.3% respectively with altitude increasing every 1,000 m. By contrast with 0 m altitude, engine torque and power at 4,000 m altitude decrease within 1% and the specific fuel consumption increases by 3.2-3.5% at the maximum torque speed, while the engine torque and power decrease by 4.6-5.3% and 4.5-5.2% respectively and the specific fuel consumption increases by 5.3-11.5% at higher and lower engine rotational speed.

**Simulation Research on Two Stage Injection of a Heavy Duty Diesel Engine 2010**

The16V280ZJ diesel engine is the power unit of Chinese main railway diesel locomotive. This article has established this diesel engine electron injection system's complete machine model using the GT software. Then a series of combustion cases of heavy duty D.I. diesel engine are calculated based on typical split injection modes. Simulation parameters are modified by comparison with given experimental results. Calculation results effectively demonstrated the effects on heat release rate by different injection velocity, fuel distribution and pulse interval. Therefore, injection optimization under multi-injection mode is summarized.

**Diesel engine dynamics modeling based on heat release rate identification 2011**

Heat release rate is one of the important factors affecting diesel engine characteristics. This paper presents a diesel engine dynamics model based on identification of heat release rate. Instantaneous engine speed and cylinder pressure can be obtained by numerical simulation of air intake and exhaust properties, in-cylinder combustion process and crank rotational dynamics by using the identification results. Simulation results of the engine speed and cylinder pressure well coincide with experimental ones for different external loads and nominal engine speed.