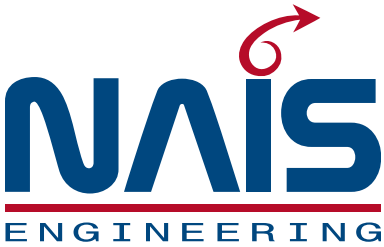

RESEARCH PROJECTS

For more information about our research projects please contact us at:
info@naisengineering.com

Or visit our web site at:
www.naisengineering.com

Setup of 1D Model for the Simulation of the Gear Pump Dynamic Behaviour

Abstract

External gear pumps are widely used in many different applications because of their relatively low costs and high performances, especially in terms of volumetric and mechanical efficiency. In the last years many efforts, in terms of mathematical models and experimental tests, were spent in order to limit energy losses and noise emissions. With the aim of deeply studying dynamic behaviour of external gear pumps and addressing their design, a 1D model was developed by means AMESim code. In the current modeling approach the following items were taken into account:

1. Recovery of the clearance between bearing block and casing;
2. The actual position of the gears computing the displacement of the bearing center;
3. The actual wear profile.

The model was then experimentally validated comparing simulated and experimental results for various delivering pressures, engine shaft speeds and gear teeth number (in particular pumps with ten and twelve teeth, characterized both by single and double contact were considered). Moreover all models developed were customized in order to simplify the computational environment for users.

Definition of a CFD Numerical Methodology for the Simulation of a Screw Compressor

Abstract

The goal of the present work was the study of the screw vacuum compressor. In order to achieve this goal, the best way was the numerical analisys because of the difficulties in performing experimental studies on the screw internal flow. A coupled 1D-3D simulation methodology was defined in order to characterize this component. Two machine types were considered:

- The first one was a *costant pitch* screw compressor;
- The second one was a *variable pitch* screw compressor.

Advanced Modelling of a New Diesel Fast Solenoid Injector and Comparison with Experiments

Abstract

Upcoming Euro 4 and Euro 5 emission standards are increasing efforts on injection system developments in order to improve mixture quality and combustion efficiency. The target features of advanced injection systems are related to their capability of operating multiple injection with a precise control of the amount of injected fuel, low cycle-by-cycle variability and life drift, within flexible strategies. In order to accomplish these tasks, injector performance must be optimised by acting on: optimisation of electronic, driving circuit, detailed investigation of different nozzle hole diameter configurations, assessment of the influence of manufacturing errors on hole diameter and inlet rounding on injector performance. An integrated 1D/3D methodology for the design of advanced new fast solenoid Common Rail (C.R.) injector for high speed diesel engines has been defined. The 1D model was used to address the injector design. Particular attention has been devoted in the simulation of the electromagnetic circuit. The functional relation between current and electromagnetic force at different air-gap values, as well as of fluid dynamics parameters was obtained by running an axial-symmetric model built into a finite-element code. Sub-models were introduced in order to account for the contribution of squish motion and actual fluid-dynamics force acting upward on needle surfaces. Multidimensional CFD simulations were extensively used in order to feed the model with the discharge coefficient of the orifices as a function of flow regimes and geometrical details.

Numerical Investigation of Critical Issues in Multiple-Injection Strategy Operated by a New C.R. Fast-Actuation Solenoid Injector

Abstract

The goal of this work was the investigation of the fuel mass variation injected with respect to nominal conditions in Common Rail injection systems for Diesel automotive applications. Two possible operating conditions have been considered: the consecutive injection of two injectors and the multiple shots of the same injector in the same engine cycle. An integrated experimental and numerical methodology has been used. Several experimental information were available in terms of instantaneous rail and pipe pressure and mass flow rate at different conditions. The 1D numerical model of the whole injection system was useful in addressing the questions remained unresolved in the post-experiments analysis. The experimental results show that injector performances are more related to pressure oscillations in injector connecting pipe rather than inside the common rail. A parametric analysis showed that the injected fuel mass variation with dwell time can be reduced by using shorter and larger injector connecting pipes.

Development of a Model for the Wall Film Formed by Impinging Spray Based ona Fully Explicit Integration Method

Abstract

A wall film model has been implemented in a customized version of KIVA-3 code. Under the hypothesis of thin laminar flow, the model simulates the dynamics of a liquid wall film generated by impinging sprays. Particular care has been taken in numerical implementation of the model. The major phenomena taken into account in the present model are: wall film formation by impinging spray; body forces, such as gravity or acceleration of the wall; shear stress at the interface with the gas and no slip condition on the wall; momentum contribution and dynamic pressure generated by the tangential and normal component of the impinging drops; film evaporation by heat exchange with wall and surrounding gas. The model doesn't consider the effect of the wavy film motion and suppose that all the impinging droplets adhere to the film. The governing equations have been integrated in space by using a finite volume approach with a first order upwind differencing scheme and they have been integrated in time with a fully explicit method. The model was validated using two different test cases reproducing PFI gasoline and DI Diesel engine wall film conditions.

CFD Analysis of Injection Timing Influence on Mixture Preparation in a PFI Motorcycle Engine

Abstract

The efficiency of engine operations, i.e. cold start, transient response and operating at idle, depends on the capability of the injection fuel system to promote a homogeneous mixture formation through an efficient interaction with engine fluid dynamics and geometry. The paper presents the development and the application of a methodology for running a CFD PFI engine simulation. A preliminary assessment of the wall-film and droplet-wall interaction sub models has been carried out in order to validate the methodology. Then a three-step numerical procedure has been adopted. The first two steps are aimed to properly initialize the secondary breakup model depending on the type of injector installed on board in order to achieve accurate predictions of spray characteristics. Then, in the third step, the CFD calculations have been used to investigate the mixture formation with the particular focus to achieve an homogeneous mixture formation without the contamination of the lubricant with fuel. A particular engine operating condition at low load is here discussed.

Assessment of a Numerical Methodology for LES Simulation of ICE Wall Bounded non-Reactive Flows

Abstract

The increasing of the overall engine efficiency is one of the most important target that the engine manufactures have to achieve during the next decade. In order to reach this goal is necessary to carefully understand the unsteady engine phenomena correlated with intake air flow and the air-fuel mixture process. The RANS methodology often doesn't allow a qualitative and quantitative accurate prediction of these phenomena, thus an alternative approach to the study of unsteady flow motions inside the engine has to be used. The aim of this paper is to present a complete overview of the influence of several parameters on the LES numerical simulation and show the potential and limits of LES technique in the simulation of actual IC engine flows. To define the numerical methodology, two test cases have been used. First a backward facing step case has been analysed to perform a preliminary parametric numerical test comparing two different LES models: Wall Adaptive Local Eddy-Viscosity (WALE) model and the one-equation Dynamic Model by Kim and Menon. The LES results are compared to experimental data in terms of mean axial and tangential velocities. Then, the best numerical settings, defined on the first case for the two LES models, have been applied to a more complex engine steady flow bench. As in the first case, numerical results obtained by LES simulations have been compared with available experimental LDA measurements of mean and rms fluctuations of both axial and tangential velocity components and with numerical predictions obtained by a RANS simulation of the same case. Using the LES approach, only a small part of the energy spectrum is modeled while the large scale motion is directly resolved.