CHAPTER - 1
CHAPTER - 1

INTRODUCTION

Knowledge of the heat transfer from the working fluid to the combustion chamber walls is of major importance for engine development since it has a major influence on engine performance, efficiency, and emissions. Energy balances for automotive engines indicate that the percentage of the fuel heating value rejected to the coolant is of order 17% for full load and 30% for part load, respectively [48], which is comparable to the engine brake power. Furthermore, heat transfer affecting the average and peak gas temperatures, has a significant impact on the onset of knock and the emission formation processes in the combustion chamber as well as in the exhaust system (after burning of HC and CO).

Heat transfer rates, which can reach magnitudes of 8 to 10 MW/m² [48], and the heat fluxes to the coolant, are also responsible for engine block and head temperature distributions. In order to avoid metal temperatures and gradients that would cause fatigue cracking, bore distortion, high thermal stresses, and lubrication oil film deterioration at the cylinder liner, detailed knowledge of the locally resolved heat transfer coefficients is necessary.

Physical laws describing the heat transfer phenomena between the gas and surrounding wall in internal combustion engines, were the subject of many research activities in the last decade, but have not been resolved sufficiently to this day. The main reason for this lies basically in complex flow patterns in the engine's cylinder that have a three-dimensional, turbulent and instantaneous character, which makes it even more difficult to describe the heat transfer processes in a theoretical way.

The first scientific investigations were done by Nusselt, who divided heat transfer mechanisms in reciprocating engines into radiation, conduction and convection. Later, many equations were formulated to describe the time resolved and spatially averaged heat transfer coefficient. Based on the quasi-steady state assumption of convection mechanisms, empirical and semi-empirical equations describing the heat transfer coefficient are derived [7, 50, 86, 113] from the Nusselt,
Reynolds and Prandtl number relationship, which is found valid for several turbulent flows in simple geometries. Nevertheless, detailed spatially resolved heat transfer coefficients which are influenced by local flow and thermodynamic values in the fluid and the surface temperature of the wall can, obviously, not be obtained using these methods.

Accurate predictions of the time-dependent local heat transfer in the combustion chamber, therefore require the solution of the fully coupled, non-linear conservation equations of mass, momentum (Navier-stokes), and energy. Using appropriate numerical algorithms and today's computer resources, these equations can be solved efficiently, even for very complex problems. Since the accuracy of 3-D Computational Fluid Dynamics (CFD) codes is relatively high, their applications in research and industrial development increased significantly over the last decade.

Different models for turbulence effects, combustion, heat and mass transfer, etc. are necessary in order to enable simulations of real flow behaviors. The validity of such models has been proven for a variety of simple flow problems and is widely accepted in terms of feasibility and accuracy. It has been demonstrated, however, that in the case of complex flow structures, such as transient or highly turbulent behaviors as they occur in internal combustion engines, an appropriate employment and the general validity is strongly questionable or sometimes not existent at all [70].

It is the aim of this work to develop and assess the heat transfer model that predicts the spatially resolved, transient, turbulent and convective heat transfer from combustion products to the combustion chamber walls of reciprocating internal combustion engines.

First a literature survey is carried out to identify the heat transfer model with high potential to improve the analysis. Modifications were developed to the identified heat transfer models that take into account non-equilibrium effects of turbulent boundary layers as well as transient core pressure changes as they occur in the combustion chamber of reciprocating internal combustion engines.

To day the most common method to describe velocity and temperature distributions in turbulent
boundary layer flows is the logarithmic law of the wall. Assuming a certain velocity and thermal boundary layer profile, local flow values near the wall are determined using an integration technique. Within this thesis, the models validated are modified versions of the law of the wall method. Finally, heat flux and heat gradient from the working fluid to the combustion chamber walls is predicted in a four-valve spark ignition engine. The results are compared with the measured data available from literature in order to validate the model.