A MULTIGRID-SOLVER FOR THE COMPUTATION OF IN-CYLINDER TURBULENT FLOWS IN ENGINES

Barbara Ruttmann
Karl Solchenbach
Gesellschaft für Mathematik und Datenverarbeitung
Postfach 1240
D-5205 St. Augustin

SUMMARY

In the last 20 years in automobile engine design many interesting applications of computational fluid dynamics (CFD) techniques have arisen. One of them is the numerical simulation of in-cylinder flow and combustion processes in reciprocating engines. The mathematical model for the gas motion in the combustion chamber is the time-dependent, compressible Navier-Stokes equations combined with a standard turbulence model.

This paper presents the multigrid (MG)-program MGNS20 for the efficient numerical solution of the 2D-model without combustion. The time discretization is done implicitly. Therefore, the time-step size is not restricted by a stability condition. The non-linear elliptic system arising in each time-step is solved by a specially designed MG-method using relaxation techniques proposed by A. Brandt.

1. INTRODUCTION

The development of modern automobile engines must take into account the rapidly increasing energy-supply and ecological problems. Legislative constraints on pollutant emissions, the critical public opinion and considerations of economy by the customers require radical alternatives to existing designs. However, these requirements should not be met at the expense of poorer engine performance.

Engines are very sensitive to the fluid-dynamic processes within the cylinder. For instance, a large amount of turbulence is necessary to guarantee good fuel-air mixing and effective combu-
The flow behaviour depends strongly on the geometrical configuration of intake valve, spark plug, piston bowl etc. In order to improve the engine performance, many manufacturers try to optimize the design of the combustion chamber. Since building and testing of research engines is expensive and time-consuming, the role of numerical simulations as an aid to efficient engine design becomes more and more important.

The mathematical description of the physical processes within the cylinder must reflect the following features:
- complex geometric configuration with moving boundaries,
- real 3D instationary flows without symmetries,
- propagation of the flame front with steep spatial gradients of temperature and other quantities,
- dependence of the chemical reactions on the mixing and the turbulent components of the gas flow.

In order to reduce the complexity of the mathematical model, the combustion and chemical reactions are neglected in our application. The gas motion is described mathematically by the instationary, compressible, Reynolds-averaged Navier-Stokes equations combined with a second order turbulence model.

Early numerical methods for the solution of these equations are based on standard CFD techniques like the ICE-(Implicit-Continuous fluid-Eulerian-) method [10]. The ICE-technique treats the viscous terms explicity in time and therefore suffers from the well-known stability limitation of the time-step size which is very restrictive in case of flows with high turbulent viscosity. Computer programs which have been recently developed for 2D- and 3D-simulations apply a fully implicit time discretization without any stability restrictions. The large systems occurring are solved iteratively by ADI-methods [1] or the "Strongly Implicit Procedure" [9].

The aim of our work in this context is the development of an efficient algorithm using implicit time discretization in connection with a MG solution technique. MG-methods are known to yield the fastest algorithms for standard problems as Poisson-like equations [14], [17]. Also for special fluid dynamics problems (incompressible Navier-Stokes equations [13], compressible in-