

Reed Valve CFD Simulation of a 2-Stroke Engine Using a 2D Model Including the Complete Engine Geometry

2010-32-0015

20109015

Published
09/28/2010

Dalibor Jajcevic, Raimund Almbauer and Stephan Schmidt
Graz University of Technology

Karl Glinsner and Matthias Fitl
BRP-Powertrain GmbH & Co KG

Copyright © 2010 SAE International and Copyright © 2010 SAE Japan

ABSTRACT

CFD has been widely used to predict the flow behavior inside 2-stroke engines over the past twenty years. Usually a mass flow profile or a simple 0D model is used for the inlet boundary condition, which replaces the complete intake geometry, such as reed valve, throttle, and air box geometries. For a CFD simulation which takes into account the exact reed valve geometry, a simulation of all above mentioned domains is required, as these domains are coupled together and thus interact. As the high speed of the engine affects the opening dynamic and closure of the reed valve, the transient data from the crank case volume and the section upstream the reed valve have an important influence on the reed petal dynamic and therewith on the sucked fresh air mass of the engine.

This paper covers a methodology for the transient CFD simulation of the reed petals of a 2-stroke engine by using a 2D model. It is a mathematical model, which describes the displacement and the dynamic behavior of a reed petal, applying a Fluid-Structure Interaction (FSI). This bending beam model of the petal is coupled with the commercial CFD Code Fluent via a user defined function. The used coupling and the mesh moving strategy is presented and discussed in detail. The 3D engine domain includes the complete engine geometry. The main focus lies on the requirements of a reed valve simulation for a 2-cylinder 2-stroke engine. Finally, test bench results of the fired engine were used for the validation of the gas dynamic behavior inside the engine.

INTRODUCTION

Due to a very simple working principle the reed valve is widely used for the control of fresh charge into the crankcase of a 2-stroke engine. Usually a variable mass flow profile or a simple 0D model is used for the inlet boundary condition of a CFD simulation of a 2-stroke engine. This boundary condition replaces the complete intake geometry, such as reed valve, throttle, and air box. For a CFD simulation with exact reed valve geometry also including the reed valve dynamic simulation, a simulation of all above mentioned domains is required, as these domains are coupled together and thus interact. This means that for the efficient simulation the transient data upstream the reed valve have an important influence on the reed petal dynamic and therewith on the engine performance. Figure 1 shows a typical configuration of the reed valve system with eight petals, investigated in this study.

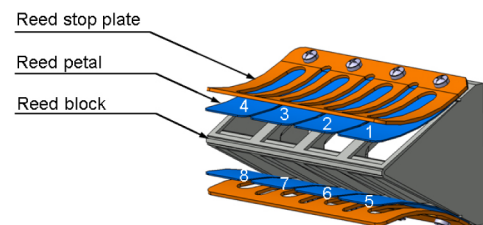


Figure 1. Reed valve system of a 2-stroke engine

Numerous investigations modeled the reed petals using different techniques in the last years. In some of them, a 1D approach based on the mass-spring system was used, other methods were based on the 2D approach using the beam theory, and in others the reed petals were modeled as a plate. Earlier studies proposed by Hinds and Blair [1] and Blair et al. [2] show a 2 dimensional model based on the beam theory. The motion of the reed petal is modeled as a cantilever beam with displacement at any point along the beam length equal to the superposition of the first three modes of free vibration. A testing procedure was developed for the validation of this approach permitting to record the petal lift displacements with a high speed camera. Presented results showed good correlation with measurements. Blair et al. [2] expanded the earlier study to provide further investigation of the used model taking into account different petal materials. Further, this model was improved by Fleck et al. [3] and [4], considering the variable petal width. Furthermore, the authors developed a technique to measure the reed valve lift on a fired engine. These measurements were obtained for a range of speeds and reed types and the results showed a good correlation with the calculated reed lift. Mitianiec and Bogusz [5] improved Blair's model and showed an application of this model with the reed petal as a plate. The authors were also interested in improving the load condition and therefore performed a 3D CFD simulation to determine the pressure profile on the petal surface. The steady improvement of the numerical methods together with the increase of computer power allows the simulation of increasingly complex technical problems and geometries. Therefore, some researchers applied 3D simulations in the last decade to investigate flow process through the reed valve where fluid and structural codes are coupled together. Cunningham et al. [6] and [7] used Blair's model for the reed petal movement. The employed 3D model consists of simplified reed valve geometry, the crankcase, and the intake ports. The rest of the engine geometry was replaced by a boundary condition stemming from a 1D simulation. Baudille and Biancolini [8] started the reed valve modeling using a finite element approach based on beam elements coupled with a 1D engine simulation. Thereafter, Baudille et al. [9] developed a 3D code based on the theory of bending beam taking into account the contact model. Therewith the influence of the stop plate was included in the simulation as well. Angeletti et al. [10] used Baudille's 3D model for the reed valve optimization with regard to the contact model - the used CFD model only includes the reed valve geometry and a small part of the intake ports. The rest of the engine geometry was replaced by boundary conditions.

In this paper Blair's model is used for the reed petal simulation, which is coupled with the CFD Code via a user defined function. In comparison to other models, the model does not need a large programming and computational effort, due to a small number of input- and exchange-data between the codes. The reed valve geometry was completely included

in the simulation without design simplifications; the engine geometry also included exhaust system, cylinder, crankcase, intake duct, throttle, and air-box geometries. Applying the above described models, the gas dynamic simulation inside the intake duct was carried out. The investigated engine is a 2-cylinder 2-stroke engine, which requires the simulation of the complete geometry including cylinders, exhaust system, and complete intake duct system. As the simulation of only one cylinder causes an incorrect gas dynamic behavior in the above mentioned domains and further leads to unrealistic results, the influence of both cylinders has to be taken into account. In order to reduce the calculation time the second cylinder was replaced by a new symmetry and time delayed boundary condition proposed by Jajcevic et al. [11]. This new boundary condition allows an efficient simulation of only one cylinder engine without data losses and simultaneously taking into account the effect of both cylinders. In order to validate the applied reed valve model and the new boundary condition, the simulations were conducted for two different engine speeds (7900 rpm wide open throttle - WOT and 6800 rpm low load - LL). Test bench results of the fired engine were used for the validation of the gas dynamic behavior simulation inside the engine.

2-STROKE ENGINE

The experimental engine is a 2-cylinder 2-stroke engine from the BRP-Powertrain Company, applying loop scavenging via five intake ports. The displacement of each cylinder is approximately 300 cm³. The engine can be also classified as a 2-stroke engine with reed valve controlled sucked air mass, crank case charge, Schnurle loop scavenging, variable exhaust port (RAVE - Rotax Adjustable Variable Exhaust), tuned exhaust pipe and high pressure gasoline direct injection (GDI). The intake ports are located symmetrically around the cylinder with the same opening time. The exhaust port consists of three ports, one main and two small lateral ports. In order to reduce the scavenging losses and to widen the speed range for effective exhaust tuning, the RAVE system controls exhaust port opening/closing time. Furthermore its use and position depends on the engine speed (low engine speed = RAVE system closed, high engine speed = RAVE system open). Due to a well-tuned exhaust system the reflected pressure wave reduces the scavenging losses. The engine has two separated intake ducts as well as two separate throttles. These intake ducts are further connected to an air-box volume (see [Figure 2](#)).

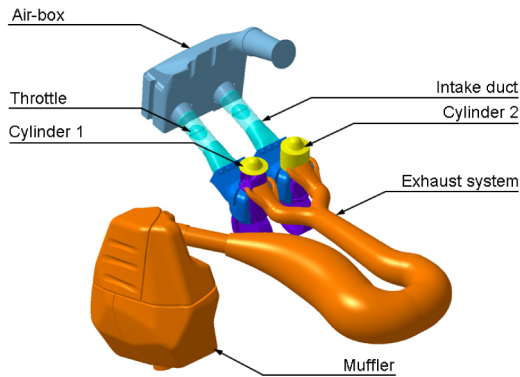


Figure 2. Flow geometry of the investigated engine

SIMULATION STRATEGY

As explained before, the investigation of a 2-cylinder 2-stroke engine requires the simulation of the complete geometry including cylinders, exhaust system, and the complete intake duct system. In this engine the cylinders are coupled together in a so-called “two-into-one” exhaust system. Furthermore, the intake and the exhaust ports are open simultaneously over a long period of the cycle and the gas dynamic behavior inside the exhaust port has a dominant influence on the scavenging process and on the mixture formation inside the cylinder. The simulation of only one cylinder causes an incorrect gas dynamic behavior inside the exhaust pipe and further leads to unrealistic results. For a simulation of the reed valve system the complete intake duct system simulation has to be considered, as the gas dynamic behavior inside the intake duct has an important influence on the reed valve behavior, so that for a correct simulation the transient behavior upstream the reed valve is needed.

However, in order to get reasonable results, the simulation of the complete and unaltered geometry is required. Another possibility is the replacement of the second cylinder by a corresponding boundary condition. In order to reduce the calculation time in this study, the second cylinder was replaced by the new symmetry and time delayed boundary condition proposed by Jajcevic et al. [11], taking into account the symmetry condition inside the engine.

This type of boundary condition can be used for time delayed as well as not time delayed technical problems. Figure 3 illustrates the application in a 2-cylinder 2-stroke engine simulation. In this case, cylinder 1 is completely calculated. For cylinder 2, the symmetric condition is used, so that it is replaced by the time delayed boundary condition. The boundary condition for the second cylinder is taken from the exhaust/intake cylinder 1 domain and for each cell respectively. This means, that for each cell in the exhaust/intake of cylinder 1 a symmetrically positioned cell in the domain of exhaust/intake cylinder 2 is found and marked. Simultaneously, all cell properties are read and saved in an

external output data file. Therewith, the correct boundary condition for each cell can be defined and the required flux can be calculated. For the flux calculation the time delayed data from the external data file are read for each cell respectively. In this work, the flux calculation occurs after the defined time delayed period. Explicitly, time delayed data are available and flux calculation is possible after this period. The advantage of this boundary condition is that the requested data are only provided by the simulation of cylinder 1. Neither a further variable profile, nor a 1D-model, nor measurement data for the replacement of the second cylinder are requested.

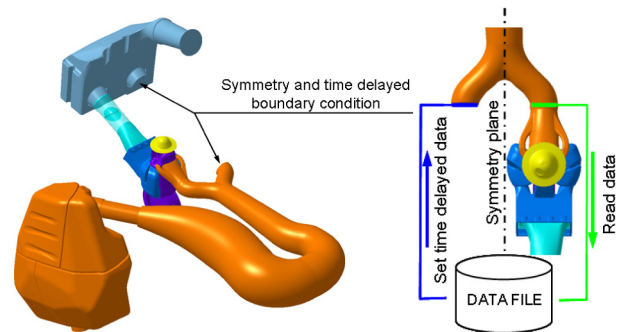


Figure 3. Application of the symmetry and time delayed boundary condition [11]

REED VALVE MODEL

The reed valve model code is written in the C programming language as a user defined function (UDF) and can dynamically be loaded with the Fluent solver. The source code containing the UDFs is directly compiled in Fluent and uses additional macros and functions that are supplied by Ansys [16]. Figure 4 presents the flowchart of the used model. The code consists of three main blocks - i.e. read data, reed valve model and transformation. The code is executed every time step by the Fluent solver. Using additional Fluent macros, the area averaged pressure on the upstream and downstream faces is read. After that the force is calculated from the pressure difference and the petal area. This force is further sent to the “Reed Valve Model” function. At the start of this function the new load condition is initialized and the beam lift position at time step $n+1$ can be calculated. The last step of the “Reed Valve Model” function is the calculation of the reed petal displacement along the reed length. The CFD code Ansys Fluent [12] uses a global Cartesian coordinate system for the flow calculation as well as a grid movement. In this work, a local coordinate system is used for the reed valve model (see Figure 5). Thus, an additional function for the coordinate transformation from local to the Cartesian coordinate system must be applied. Finally, the reed petal displacement can be submitted back to the Fluent solver and the node moving at time step $n+1$ can be evaluated.

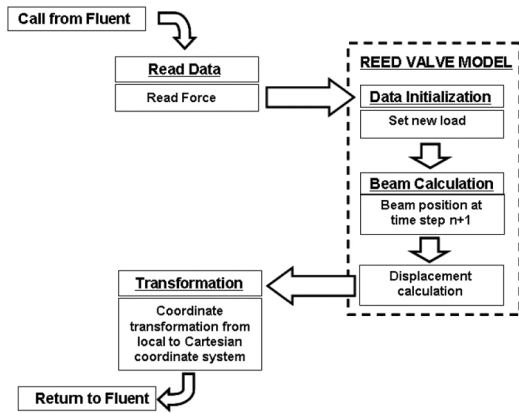


Figure 4. Flow chart of the reed valve model

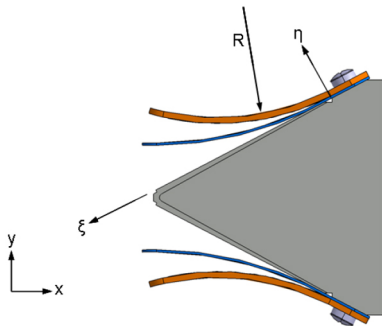


Figure 5. Local coordinate system for beam position calculation

In 1978 Hinds and Blair [1] published a 2D model based on the theory of transverse vibrations of a loaded beam. The authors deduced that the beam point motion can be described by following equation:

$$EI \frac{\partial^4 \eta}{\partial \xi^4} + \rho A \frac{\partial^2 \eta}{\partial t^2} = 0 \quad (1)$$

where, E is the young modulus, I the moment of inertia, ρ the density, A the cross-sectional area, η the lift and ξ the position of calculated beam point. A solution of [equation \(1\)](#) has the following form:

$$\eta = \phi(\eta) \cdot e^{i\omega t} \quad (2)$$

with

$$\phi(\eta) = A \cos \lambda \eta + B \sin \lambda \eta + C \cosh \lambda \eta + D \sinh \lambda \eta \quad (3)$$

and

$$\lambda = \sqrt[4]{\frac{\rho A \omega^2}{EI}} \quad (4)$$

Considering the boundary condition for a cantilever beam, e.g. start and end point condition, following equation can be obtained:

$$1 + \cos \lambda l \cosh \lambda l = 0 \quad (5)$$

[Equation \(5\)](#) is numerically solved and solutions of the first five roots of $\lambda_i l$ according to Hinds and Blair [1] are presented in [Table 1](#).

Table 1. Solution of $\lambda_i l$ according to [1]

$\lambda_1 l$	$\lambda_2 l$	$\lambda_3 l$	$\lambda_4 l$	$\lambda_5 l$
1.875	4.694	7.555	10.996	14.137

The vibration frequency of each moving point is given by

$$f_i = \frac{\omega_i}{2\pi} = \frac{\lambda_i l}{2\pi} \sqrt{\frac{EI}{\rho A l^4}} \quad (6)$$

Applying the same transformation, [equation \(3\)](#) can be written:

$$\begin{aligned} \phi_i(\eta) = & \cosh \lambda_i \eta - \cos \lambda_i \eta - \\ & - \frac{\sinh \lambda_i l - \sin \lambda_i l}{\cosh \lambda_i l + \cos \lambda_i l} (\sinh \lambda_i \eta - \sin \lambda_i \eta) \end{aligned} \quad (7)$$

The total displacement of each moving point is given by the following equations:

$$\eta(\xi, t) = \sum_{i=1}^n \phi_i(\eta) \cdot Z_i(t) \quad (8)$$

$Z_i(t)$ is obtained from:

$$\frac{d^2 Z_i}{dt^2} + 2\zeta_i \omega_i \frac{dZ_i}{dt} + \omega_i^2 Z_i = \frac{F_i(t)}{m_i} \quad (9)$$

where ζ_i is a damping factor, ω_i the vibration frequency, m_i the element mass, and F_i the force, which is dependant on the pressures upstream and downstream the reed petal and is set as a uniform forcing function. Fleck et al. [3] investigated various methods of calculating this forcing function and found that the best method was a simple uniform model, which considers that the pressure force is the same at any point on the reed petal. [Equation \(9\)](#) is solved by the four stage Runge-Kutta method according to Toro [14]. Zeng et al.

[15] used an alternative method, the first order accurate Euler-forward scheme, to solve this equation.

The movement of the reed petal is restricted - the petal maximum position is limited by the stop plate geometry and its minimum position by the reed block (see Figure 1). Therefore, the contact between the reed petal and the boundaries was taken into consideration. The main function of the stop plate is that it continuously reduces the free petal length during the opening phase. Therewith the elastic characteristic is automatically changed depending on the contact point position. E.g. Hinds and Blair [1] changed the effective free length by a contact between the reed petal and the stop plate. The new length is calculated from the distance between the contact point and the free reed end point. Fleck et al. [3] models this contact in a simple way by using the product of a coefficient of restitution and the time derivative of the principle coordinate in order to describe the rebound velocity of the reed from the reed stop. For the maximum petal position and a contact between the reed petal and the reed stop plate, it is assumed in this study that the reed lift at any point cannot be larger than the stop plate height. The reed stop plate geometry can be described with a circle function given by:

$$\eta_{\max}(\xi) = \left| \sqrt{R^2 - (\xi - \xi_C)^2} - \eta_C \right| \quad (10)$$

where R is the radius of the stop plate curve and ξ_C and η_C the circle center position in the local coordinate system. With this function the stop plate is completely described along its length. If the reed lift is larger than the value of $\eta_{\max}(\xi)$, the moving of all points in this region is stopped. In a next step, the max contact point position is found. For all beam points with a position smaller than the max contact point position the time derivative component of the principle coordinate is set to zero and the reed position is set to the position of the stop plate. For the beam points with a position larger than the max contact point position - the free length of the reed petal - the velocity component is multiplied by a coefficient c given by the following linear equation:

$$c = \frac{c_{\max}}{N} (i - i_{\max}), \quad i \geq i_{\max} \quad (11)$$

where $c_{\max}$ is the max damping coefficient ($c_{\max} = 0.7$ is used in this study), N the total number of the beam points, i the actual beam point and $i_{\max}$ the position of the max contact point.

The experimental verification of the simulation model is a necessary step to validate its correctness and is carried out with a Laser Doppler Vibrometer (further LDV). Nagy et al. [17] uses LDV for the valve lift measurement in hermetic piston compressors. An LDV instrument measures the

velocity laser beam direction component of a moving reflective surface (see Figure 6). In the case of a moving surface - due to the Doppler Effect - the frequency of the reflected light compared to the source light is shifted. This frequency shift is evaluated by an interferometer and converted to a voltage signal. This signal is a function of the velocity of the moving surface. Finally, the reed valve lift can be obtained from the velocity function.

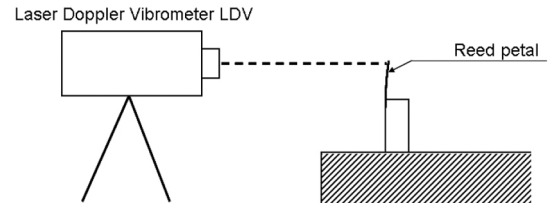


Figure 6. Laser Doppler Vibrometer - LDV

The start displacement of the reed petal at position $\xi = L$ is set to approximately 6.2 mm. The reed petal is disengaged and the free vibration of the cantilever beam can be measured.

Figure 7 shows the comparison between the measurement and simulation results in the case of a free cantilever beam vibration. A good prediction of the simulation model can be seen, which further confirms the accurateness of the reed valve model.

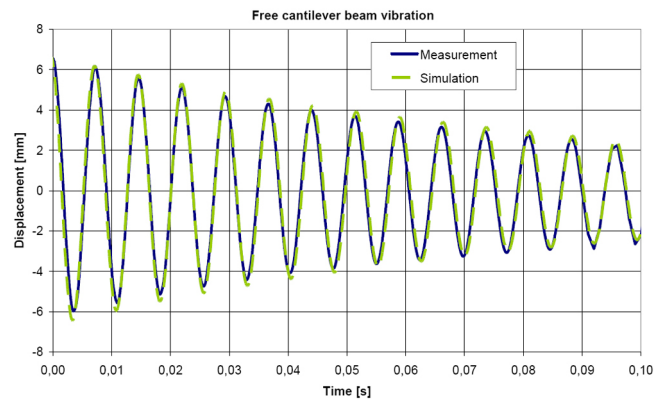


Figure 7. Free cantilever beam vibration

MOVING MESH STRATEGY

For the reed valve simulation a moving mesh is required and for this purpose the Fluent re-meshing method is used (see Ansys [12]). Fluent marks cells that violate the set "skewness" or size criteria and locally re-meshes the marked cells or faces. As soon as the new created cells or faces satisfy the skewness criterion, the mesh is locally updated with the new cells. For the efficient application of this meshing strategy the reed valve system is divided into two domains (see Figure 8). These two domains are completely