FLEXe

Future Energy System

RESEARCH REPORT
NO D4.2-12
HELSINKI 2016

Jukka Kiijärvi
University of Vaasa

Seppo Niemi
University of Vaasa

Zero Wall Shear Model

CLIC

Solution Architect for Global
Bioeconomy & Cleantech Opportunities

CLIC INNOVATION OY
ETELÄRANTA 10
P.O. BOX 10
FI-00131 HELSINKI,
FINLAND
CLICINNOVATION.FI

ISBN 978-952-7205-03-7

CLICK Innovation
Research report no D4.2-12

Jukka Kiijärvi
University of Vaasa

Seppo Niemi
University of Vaasa

Zero Wall Shear Model

Name of the report: Zero Wall Shear Model

Key words: diesel, fuel injection, nozzle, cavitating flow, outlet flow area

Abstract

The aim of this report is to study computationally the one-dimensional turbulent cavitating flow in the nozzle of a diesel injector. The cavitating flow in the nozzle is often computed with the slug flow model. In this model the outlet flow area is assumed the same as the geometric cross sectional area of the nozzle. The zero wall shear model is the other model used to compute the flow in the nozzle. The equations of this model are derived in detail. Two Scilab scripts were written for the computation. As a result, the slug flow model gives a clearly too low flow velocity and a too large flow area at the outlet. In cavitating turbulent flow the zero wall shear model is recommended for computing the flow velocity and area at the outlet of the nozzle.

Helsinki, October 26, 2016.

Contents

Symbols

$\vec{A}$	area vector
A	area
A_c	minimum flow area
A_g	geometric cross sectional area
A_2	flow area in section 2
C_c	coefficient of contraction
C_d	coefficient of discharge
C_{dt}	coefficient of discharge in turbulent non cavitating flow
CS	control surface
CV	control volume
F_{B_x}	body force in x-direction
F_{S_x}	surface force in x-direction
F_x	net force in x-direction
g	gravitational acceleration
K	Nurick's cavitation number
K_{cr}	critical Nurick's cavitation number
$\dot{m}$	mass flow rate
$\dot{m}_{\text{th}}$	theoretical mass flow rate
p	pressure
p_c	pressure at the minimum flow area
p_v	vapor pressure
p_1	pressure at section 1
p_2	pressure at section 2
u	x-component of flow velocity
u_c	x-component of flow velocity at the minimum flow area
u_g	according to geometric flow area calculated x-component of flow velocity at the outlet of nozzle
u_{th}	theoretical flow velocity in x-direction
u_2	x-component of flow velocity in section 2
V	velocity
$\vec{V}$	velocity vector
V_c	flow velocity at the minimum flow area
$\rlap{-}V$	volume
$\dot{\rlap{-}V}$	volume flow rate
z	height
ρ	density

1 Introduction

The emissions and efficiency of an internal combustion engine depend on the combustion in the cylinder. In a diesel engine combustion is controlled by means of fuel injection. During a short time the fuel is injected through small nozzles using high pressure and velocity into the combustion chamber.

The fuel flow in the nozzle affects the spray and diffusion of fuel and air in the cylinder. High pressure and velocity are needed to achieve good atomization, adequate penetration of the spray and good mixing of fuel and air.

Because a great pressure difference exists over the injector nozzle, the flow cavitates in the nozzle almost all the time during the injection. Cavitation is a very complicated phenomenon. In a simple cavitation model of an injector nozzle it is assumed that vapor pressure exists at the smallest cross sectional flow area.

Combustion is often simulated with a computational fluid dynamics code. In that case the flow velocity from the outlet of the injector nozzle must be known as a function of time. To compute the flow velocity, the flow area at the outlet of the nozzle must be known, too.

The aim of this study was to solve both the flow velocity and area at the outlet of a diesel nozzle. A one-dimensional model was used. The flow was assumed to cavitate in the nozzle.

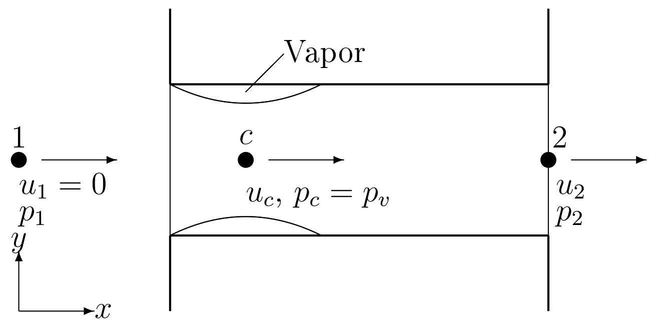

Figure 1. Schematic of the flow in Nuricks's model.

2 Models

In deriving the Nurick's and the zero wall shear model the Fox-McDonald-Pritchard [Fox10] solution methodology was used. The methodology was modified a little.

2.1 Nurick's model

Nurick's model is needed in deriving zero wall shear model. Nurick [Nur76] has derived an equation for the coefficient of discharge in the diesel injector nozzle. In this equation the discharge coefficient depends on the coefficient of contraction and Nurick's cavitation number. The coefficient of contraction is calculated by dividing the minimum flow area by the geometric cross sectional area. Nurick's cavitation number is a function of the pressure before and after the nozzle and the vapor pressure of the fuel.

2.1.1 Schematic

In Figure 1 the schematic of the flow in the Nurick's model is given.

2.1.2 Governing equations

The Bernoulli equation is

$$p + \frac{1}{2}\rho V^2 + \rho g z = \text{constant}, \tag{1}$$

where

p	pressure
ρ	density
V	flow velocity
g	gravitational acceleration

z height.

The mass flow rate is calculated with the expression

$$\dot{m} = \rho \dot{V} = \rho V A, \tag{2}$$

in which $\dot{m}$ means mass flow rate, $\dot{V}$ volume flow rate and A cross sectional area.

2.1.3 Assumptions

In deriving Nurick's model the following assumptions were made:

1. The flow is turbulent.
2. The flow cavitates in the nozzle.
3. No losses exist.
4. Liquid is incompressible.
5. The flow is one-dimensional.
6. Steady state equations can be used.
7. When one looks from the inlet or outlet of the nozzle, vapor accumulates ring-shaped on the wall of the nozzle.
8. At the smallest flow area (the section c in Figure 1) the pressure equals to the vapor pressure.

At full load, the injection lasts in a vehicle diesel engine 1–2 ms and in a medium speed diesel engine approximately 5 ms . The injection pressure can vary 150 MPa or more during the injection period. So it is obvious that the flow in the nozzle is time dependent. The nozzle is very short compared to the injection pipes and the injection time is short, too. Thus, it can be assumed that the properties of the liquid in the nozzle at a certain moment don't change. Schmitt [Sct66] has shown by experiments and calculation that in diesel nozzles the liquid flow can be assumed quasi-static. So the steady state equations are allowed to use in liquid nozzle flow (Assumption 6).

2.1.4 Solution

The liquid flows from a container through the nozzle into the cylinder. The section 1 is chosen so that there the flow velocity $u_1 \approx 0$ (Figure 1, p. 7). The pressure p_1 exists in the section 1.

Because the streamlines must be continuous, the streamlines at the inlet are unable to turn abruptly to the direction of the nozzle walls. For this reason the flow diverges from the walls and the flow area decreases in the nozzle near the inlet. The smallest flow area A_c locates in the the section c. This section is also called vena contracta. In the section c, the pressure p_c is the lowest and the velocity u_c the highest. When the pressure p_c decreases enough, cavitation begins in the vena contracta. Now, in the section c, the vapor pressure exists (Assumption 8).

The ratio of the smallest flow area A_c to the geometric cross sectional area A_g is defined as the coefficient of contraction C_c or

$$C_c = \frac{A_c}{A_g} \tag{3}$$

From the previous expression and Equation (2) the mass flow rate becomes

$$\dot{m} = \rho A_c u_c = \rho C_c A_g u_c, \tag{4}$$

in which ρ means the density of the liquid. When the flow velocity in the section c is solved from the previous equation, it comes

$$u_c = \frac{\dot{m}}{\rho C_c A_g} \tag{5}$$

Next the Bernoulli equation (1) is used to the sections 1 and c. The section 1 was chosen so that the flow velocity there is $u_1 = 0$. The pressure difference between the sections 1 and c caused by the height difference is insignificant. In this way, from the Bernoulli equation and mass flow rate an expression is derived for p_1 or

$$p_1 = p_c + \frac{1}{2}\rho u_c^2 = p_c + \frac{1}{2}\rho \left(\frac{\dot{m}}{\rho C_c A_g}\right)^2 \tag{6}$$

The mass flow rate is solved from the last equation and it is remembered that $p_c = p_v$ (Assumption 8), then it becomes

$$\dot{m} = C_c A_g \sqrt{2\rho(p_1 - p_v)} \tag{7}$$

When the flow has no losses, the Bernoulli equation is between the sections 1 and 2

$$p_1 = p_2 + \frac{1}{2}\rho u_{\mathrm{th}}^2 \tag{8}$$

where u_{th} means the theoretical flow velocity. This velocity is calculated using the geometric cross sectional area.

Now the theoretical flow velocity is solved from the previous equation

$$u_{\mathrm{th}} = \sqrt{\frac{2}{\rho}(p_1 - p_2)} \tag{9}$$

The coefficient of discharge C_d is defined to be the ratio of the real and theoretical mass flow rates

$$C_d = \frac{\dot{m}}{\dot{m}_{\mathrm{th}}} = \frac{A_g C_c \sqrt{2\rho(p_1 - p_v)}}{\rho A_g u_{\mathrm{th}}} = \frac{C_c\sqrt{2\rho(p_1 - p_v)}}{\rho u_{\mathrm{th}}}, \tag{10}$$

where $\dot{m}_{\mathrm{th}}$ means theoretical mass flow rate.

By setting Expression (9) of the theoretical flow velocity into Equation (10) the coefficient of discharge becomes

$$C_d = \frac{C_c\sqrt{2\rho(p_1 - p_v)}}{\rho\sqrt{(2/\rho)(p_1 - p_2)}} = C_c\sqrt{\frac{p_1 - p_v}{p_1 - p_2}} \tag{11}$$

In Nurick's model the expression under the square root is called cavitation number K i.e.

$$K = \frac{p_1 - p_v}{p_1 - p_2} \tag{12}$$

Because the cavitation number has many definitions, K is called here the Nurick's cavitation number.

Finally the coefficient of discharge can be given as

$$C_d = C_c\sqrt{K} \tag{13}$$

2.2 Slug flow model

At the nozzle outlet the velocity in cavitating flow is often computed as a mean velocity. This is calculated using mass flow rate, density and geometric cross sectional area of the nozzle. It is thought that the flow touches the walls of the nozzle rather soon after the cavitation region (Figure 2). The flow has developed uniform at the outlet. There are large gas bubbles, which move with the flowing liquid. This kind of flow is called slug flow. The slug flow model may describe non-cavitating turbulent and slightly cavitating flow in the nozzle rather well [Sch97].

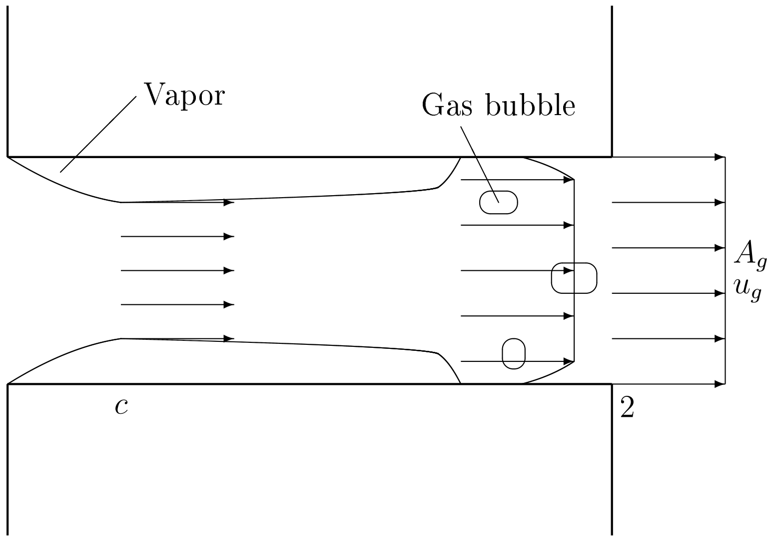

Figure 2. Slug flow model.

2.3 Zero wall shear model

According to Schmidt [Sch97] the flow in the nozzle cavitates strongly almost all the time during the injection (Figure 3). For this reason the vapor region on the wall extends nearly to the whole length of the nozzle. When the cavitation region ends, the flow velocity near the wall remains very low. Therefore, it can be assumed that after the cavitation region there is only insignificant shear force that affects the main flow. Since the main flow is assumed frictionless, the velocity profile is uniform.

Schmidt [Sch97] has derived equations for the flow velocity and flow area at the outlet of the nozzle, when the flow is cavitating in the nozzle. Equations of the outlet flow velocity and area are derived using the conservation of mass and the momentum of a control volume chosen in the nozzle. The following derivation is based on the Schmidt's dissertation [Sch97].

2.3.1 Schematic

Figure 4 shows the schematic of the zero wall shear model.

2.3.2 Governing equations

The momentum equation for the control volume in the x-direction is [Fox10]

$$F_x = F_{S_x} + F_{B_x} = \frac{\partial}{\partial t} \int_{\mathrm{CV}} u\rho \, d\forall + \int_{\mathrm{CS}} u\rho \vec{V} \cdot d\vec{A}, \tag{14}$$

where

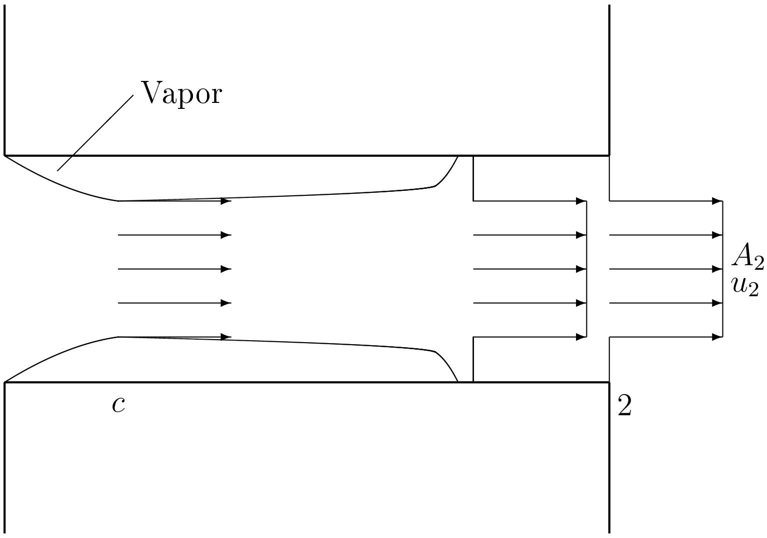

Figure 3. Zero wall shear model flow in the nozzle.

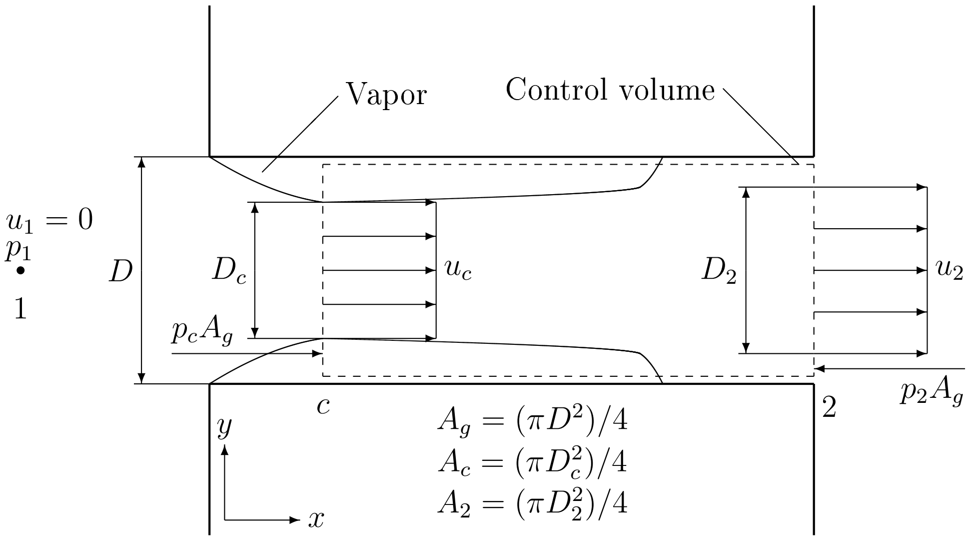

Figure 4. Schematic of zero wall shear model.

F_x	net force in x-direction
F_{S_x}	surface force in x-direction
F_{B_x}	body force in x-direction
t	time
CV	control volume
u	x-component of velocity
ρ	density
$\forall$	volume
CS	control surface
$\vec{V}$	velocity vector
$\vec{A}$	area vector.

When the flow is uniform at each inlet and outlet, the last term of the previous equation can be written in sum form

$$F_x = F_{S_x} + F_{B_x} = \frac{\partial}{\partial t} \int_{\mathrm{CV}} u\rho \, d\forall + \sum_{\mathrm{CS}} u\rho \vec{V} \cdot \vec{A} \tag{15}$$

The equation for conservation of mass is [Fox10]

$$\frac{\partial}{\partial t} \int_{\mathrm{CV}} \rho \, d\forall + \int_{\mathrm{CS}} \rho \vec{V} \cdot d\vec{A} = 0 \tag{16}$$

2.3.3 Assumptions

In the zero wall shear model it is assumed that

1. Steady state equations can be used.
2. Flow cavitates much in the nozzle.
3. Vapor builds up seen from the axle of the nozzle ring-shaped on the wall of the nozzle.
4. There is no shear force from the wall to the main flow. Thus the flow is frictionless.
5. Velocity profile at vena contracta and at the outlet of the nozzle is uniform due to frictionless flow.
6. Flow is one-dimensional.
7. The liquid is incompressible.

8. At vena contracta in the section c (Figure 4, p. 12), no momentum transfers through the vapor to the control volume. This assumption follows from the fact that the density of vapor is small compared to the density of the liquid.

9. The control volume is stationary.

10. Pressure at vena contracta in the section c is equal to the vapor pressure.

2.3.4 Solution

The Reynolds transport theorem was used in deriving the zero wall shear model equations. More information of this theorem is given in the reference [Fox10, pp. 92–160] and in many other fluid mechanics books.

The effective flow velocity at the outlet of the nozzle approximates the flow velocity from the nozzle. The effective flow velocity is calculated using the effective flow area. The equations of the effective flow velocity and area were solved from the momentum equation and conservation of mass.

Momentum equation: At first, the momentum equation for the chosen control volume was created. On the left surface of the control volume (Figure 4, p. 12) acts the pressure force $p_c A_g$ and on the right pressure force $p_2 A_g$. p_c means the pressure in the section c and p_2 pressure in the section 2. A_g stands for the geometric cross sectional area of the nozzle. According to Assumption 10, the pressure in the section c is equal to the vapor pressure p_v. So the net surface force acting on the control volume is

$$F_{S_x} = p_c A_g - p_2 A_g = A_g(p_c - p_2) = A_g(p_v - p_2) \tag{17}$$

No body forces acting on the control volume in x-direction exist. Thus the body force in x-direction is

$$F_{B_x} = 0 \tag{18}$$

Because the flow was assumed quasi static (Assumption 1)

$$\frac{\partial}{\partial t} \int_{\mathrm{CV}} u\rho \, d\forall = 0 \tag{19}$$

At the vena contratcta in the section c, the velocity and area vectors show to opposite directions. So, in this section, the scalar product of velocity and area is negative. Here it is allowed to use the sum form in the last term of Equation (14). On the right surface of the control volume, the velocity and area vectors have the same direction. Their scalar product is therefore positive. In this way it comes

$$\int_{\mathrm{CS}} u\rho\vec{V}\cdot d\vec{A} = -\rho u_c^2 A_c + \int_{A_2} u\rho\vec{V}\cdot d\vec{A} \tag{20}$$

By substituting the terms (17), (18), (19) and (20) into Equation (14), it is received

$$A_g(p_v - p_2) = -\rho u_c^2 A_c + \int_{A_2} u\rho\vec{V}\cdot d\vec{A} \tag{21}$$

Applying the Bernoulli equation (1) to the sections 1 and c

$$p_1 = p_v + \frac{1}{2}\rho u_c^2, \tag{22}$$

and solving the square of the flow velocity at the smallest cross sectional flow area, it comes

$$u_c^2 = \frac{2}{\rho}(p_1 - p_v) \tag{23}$$

In addition, it is known (Expression (3)) that the smallest flow area is

$$A_c = C_c A_g, \tag{24}$$

in which C_c is the coefficient of contraction.

By solving the momentum in the section 2 from Equation (21) and substituting the expressions of the velocity (23) and area (24), we receive

$$\begin{aligned}\int_{A_2} u\rho\vec{V}\cdot d\vec{A} &= \rho u_c^2 A_c + A_g(p_v - p_2)\\ &= \rho\frac{2}{\rho}(p_1 - p_v)C_c A_g + A_g(p_v - p_2)\\ &= 2C_c A_g(p_1 - p_v) + A_g(p_v - p_2)\end{aligned} \tag{25}$$

The velocity out of the nozzle and the outlet flow area are approximated with an effective flow velocity u_2 and an effective flow area A_2. The effective velocity and area must fulfill both the momentum and conservation of mass equations. When the effective velocity and area are substituted into the momentum Equation (25), it comes

$$\rho u_2^2 A_2 = 2C_c(p_1 - p_v)A_g + (p_v - p_2)A_g \tag{26}$$

Conservation of mass: Next, the conservation of mass equation was formulated using the governing equation (16). The flow was assumed quasi static (Assumption 1), so

$$\frac{\partial}{\partial t}\int_{\mathrm{CV}} \rho \, d\forall = 0 \tag{27}$$

In the section c, the velocity and area vectors show in opposite directions. Thus the scalar product is negative. In the section c, the following mass flow rate $\dot{m}$ goes through the left control surface:

$$\int_{\mathrm{CS}_c} \rho \vec{V} \cdot d\vec{A} = -\rho u_c A_c = -\dot{m} \tag{28}$$

At the outlet of the nozzle, the velocity and area vectors show to the same direction. Hence, here the scalar product is positive. The mass flow rate through the outlet of the nozzle is

$$\int_{\mathrm{CS}_2} \rho \vec{V} \cdot d\vec{A} = \rho u_2 A_2 \tag{29}$$

The conservation of mass equation of the control volume becomes

$$-\dot{m} + \rho u_2 A_2 = 0 \tag{30}$$

Effective flow velocity: When the mass flow rate $\dot{m}$ Equation (7) is substituted to previous Formula (30), it is received

$$-C_c A_g \sqrt{2\rho(p_1 - p_v)} + \rho u_2 A_2 = 0 \tag{31}$$

Solving the effective flow area from this equation, it comes

$$A_2 = \frac{C_c A_g \sqrt{2\rho(p_1 - p_v)}}{\rho u_2} \tag{32}$$

By substituting the expression of the effective flow area into Equation (26), we get

$$\rho u_2^2 \frac{C_c A_g \sqrt{2\rho(p_1 - p_v)}}{\rho u_2} = 2C_c A_g (p_1 - p_v) + A_g (p_v - p_g) \tag{33}$$

Solving the effective flow velocity from this, the formula for u_2 is received

$$\begin{aligned} u_2 &= \frac{2C_c(p_1 - p_v) + (p_v - p_2)}{C_c\sqrt{2\rho(p_1 - p_v)}} \\ &= \frac{2C_c p_1 - p_2 + (1 - 2C_c)p_v}{C_c\sqrt{2\rho(p_1 - p_v)}} \end{aligned} \tag{34}$$

Effective flow area: Next, the effective flow area was determined. When the effective flow velocity Equation (34) is substituted into Equation (32), for the effective flow area it comes

$$\begin{aligned} A_2 &= \frac{\left[C_c A_g \sqrt{2\rho(p_1 - p_v)}\right]\left[C_c \sqrt{2\rho(p_1 - p_v)}\right]}{\rho[2C_p p_1 - p_2 + (1 - 2C_c)p_v]} \\ &= \frac{2\rho C_c^2 (p_1 - p_v)}{\rho[2C_p p_1 - p_2 + (1 - 2C_c)p_v]} A_g \\ &= \frac{2C_c^2 (p_1 - p_v)}{2C_c p_1 - p_2 + (1 - 2C_c)p_v} A_g \end{aligned} \tag{35}$$

Ratios: Now an equation was derived for the ratio of the effective flow area and the geometric cross sectional area. The vapor pressure p_v is much lower than the pressure before and after the nozzle. That is why the vapor pressure is assumed $p_v \approx 0$. The dependence between the pressures p_1 and p_2 is needed. When the vapor pressure is approximated zero, the Nurick's cavitation number is solved from Expression (12)

$$K = \frac{p_1}{p_1 - p_2} \tag{36}$$

Solving the pressure p_2 after the nozzle from this equation, it comes

$$p_2 = \frac{K - 1}{K} p_1 \tag{37}$$

The ratio of the cross sectional areas is solved from Equation (35)

$$\frac{A_2}{A_g} = \frac{2C_c^2 (p_1 - p_v)}{[2C_c p_1 - p_2 + (1 - 2C_c)p_v]} \tag{38}$$

Because $p_v = 0$, the area ratio is

$$\frac{A_2}{A_g} = \frac{2C_c^2 p_1}{2C_c p_1 - p_2} \tag{39}$$

The solution of the pressure p_2 (37) is substituted into Equation (39)

$$\begin{aligned} \frac{A_2}{A_g} &= \frac{2C_c^2 p_1}{2C_c p_1 - [(K - 1)/K] p_1} \\ &= \frac{2C_c^2}{2C_c - (K - 1)/K} \end{aligned} \tag{40}$$

Finally, the ratio of the average velocity u_g and the effective flow velocity u_2 was determined. In both cases, the same mass flow rate flows through the nozzle

$$\dot{m} = \rho A_g u_g = \rho A_2 u_2 \tag{41}$$

From this it follows that

$$\frac{u_g}{u_2} = \frac{A_2}{A_g} \tag{42}$$

Thus the ratio of the average velocity and the effective flow velocity is equal to the ratio of the effective flow area and the geometric cross sectional area of the nozzle.

3 Computed results

The coefficients of discharge were calculated using Equations (12) and (13). The vapor pressure was assumed to be zero. The area and velocity ratio were solved with Equations (40) and (42). For computing the results two Scilab scripts were written.

In calculations two nozzles were used. The first nozzle has slightly rounded inlet edge and the second one rounded inlet edge. In non-cavitating turbulent flow the discharge coefficient is constant. The values of contraction and discharge coefficient for non-cavitating turbulent flow are rough approximations. The coefficient of contraction depends strongly on the geometry of the nozzle [Sch97, p. 54].

For a slightly rounded inlet edge nozzle the coefficient of contraction C_c is 0.666. The coefficient of discharge C_{dt} in non-cavitating turbulent flow is assumed to be $C_{dt} = 1.16C_c = (1.16)(0.666) \approx 0.773$ [Kii15]. The critical Nurick's cavitation number K_{cr}, when the flow begins or ends to cavitate, is solved from Equation 13

$$K_{\mathrm{cr}} = \left(\frac{C_{dt}}{C_c}\right)^2 = \left(\frac{0.773}{0.666}\right)^2 = 1.35$$

Schmidt [Sch97] has used this nozzle to verify the zero wall shear model. The values of the slightly rounded inlet edge nozzle are typical to an old diesel nozzle.

The coefficient of contraction C_c of the rounded inlet edge nozzle is approximated to be 0.796. The coefficient of discharge C_{dt} in non-cavitating turbulent flow is $C_{dt} = 1.12C_c = (1.12)(0.796) \approx 0.890$ [Kii15]. The critical cavitation number K_{cr} of this nozzle is 1.25. The coefficients of the rounded inlet edge nozzle are representative of a modern diesel nozzle.

3.1 Coefficient of discharge

The coefficient of discharge of the two nozzles as a function of the Nurick's cavitation number is given in Figure 5. The short horizontal lines at the ends of curves show the constant coefficient of discharge in turbulent non-cavitating flow.

The rounded inlet edge nozzle begins to cavitate at a smaller Nurick's cavitation number than the slightly rounded edge nozzle.

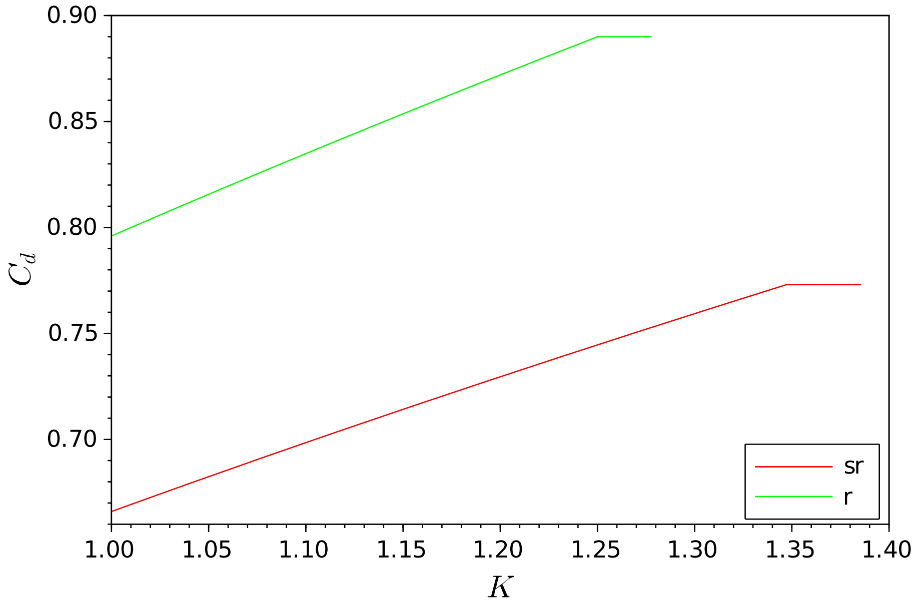

Figure 5. Coefficient of discharge C_d as a function of the Nurick's cavitation number K. sr slightly rounded inlet edge nozzle, r rounded inlet edge nozzle.

3.2 Area and velocity ratio

The ratio of the effective flow area and geometric flow area is shown in Figure 6 as a function of the Nurick's cavitation number. The ratio of the average velocity and the effective velocity is equal to the area ratio as given in Equation (42). When the coefficient of discharge reaches the value of the turbulent non-cavitating flow, the curves end.

The slightly rounded inlet edge nozzle has a smaller area ratio A_2/A_g than the rounded inlet edge nozzle. If the effective flow area is assumed accurate, then the error of the less rounded edge nozzle is greater compared to the more rounded edge nozzle.

In the real injection process it is estimated that the injection pressure p_1 is 150 MPa and the pressure p_2 in the combustion chamber 15 MPa. Then the Nurick's cavitation number K is 1.11. In this case for the slightly rounded inlet edge nozzle $A_2/A_g = 0.721$ and for the rounded inlet edge nozzle $A_2/A_g = 0.849$. When the geometric cross-sectional area is used instead of the effective area, the error in the slightly rounded inlet edge nozzle is 39 % and in the rounded inlet edge nozzle 18 %.

The slightly rounded inlet edge nozzle has cavitation in larger Nurick's

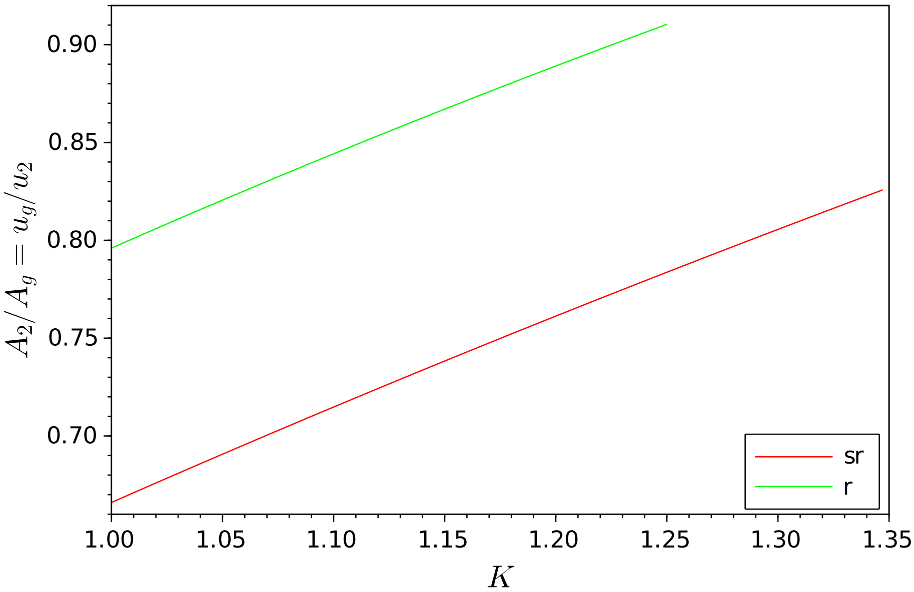

Figure 6. The ratio of effective flow area A_2 and geometric area A_g as a function of the Nurick's cavitation number K. u_2 is effective flow velocity and u_g velocity computed using the geometric area. sr slightly rounded inlet edge nozzle, r rounded inlet edge nozzle.

cavitation number region than the rounded inlet edge nozzle.

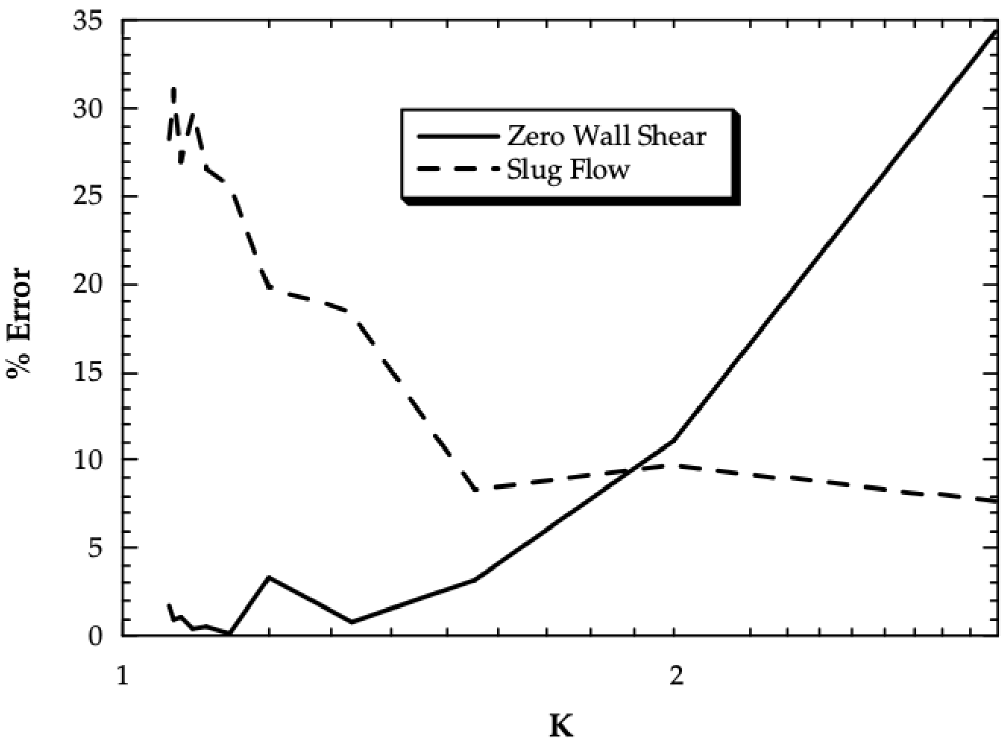

Figure 7. The error of the exit momentum of the zero wall shear and slug flow model compared to the results of a two-dimensional model. The error is shown against the Nurick's cavitation number K. [Sch97, p. 58].

4 Discussion

Schmidt [Sch97, pp. 57–58] has compared the exit momentum of the zero wall shear model, the slug flow model and a two-dimensional model. The two-dimensional model was assumed the most accurate. The exit momentums of the zero wall shear model and the slug flow model were verified to the exit momentum of the two-dimensional model. Schmidt's results are shown in Figure 7.

Schmidt used in the verification slightly rounded inlet edge nozzle. Its coefficient of contraction $C_c = 0.666$, discharge coefficient of turbulent non-cavitating flow $C_{dt} = 0.773$ and the critical Nurick's cavitation number $K_{\mathrm{cr}} = 1.35$ are given in Section 3.

In Figure 7 at the Nurick's cavitation number $K = 1.34$, the error of the zero wall shear model begins to increase and the error of the slug flow model begins to decrease. This may indicate that $K = 1.34$ is the real Nurick's critical cavitation number of the slightly rounded edge nozzle. The estimated Nurick's critical cavitation number 1.35 is almost the same.

The error of momentum of the zero wall shear model varies from 0 to 3.4 % in the cavitation region. In cavitating flow the error of the slug flow model is 17.8–31.0 %. Schmidt [Sch97, pp. 55–59] has also validated the zero wall shear model using the measured results from different references. It can be concluded that in the cavitation region the zero wall shear model is much more accurate than the slug flow model.

In the modern injection systems, the injection pressure can be very high. It is possible that the flow between the needle and seat cavitates. The effect of this cavitation on the flow in the nozzle was not considered in this study.

During cavitation the effective outlet flow area of the slightly rounded and the rounded edge nozzle is smaller than the geometric cross sectional area of the nozzle (Figure 6). According to this, the geometric cross sectional area can't be used as the outlet flow area in cavitating flow.

If the flow is non-cavitating turbulent, it is unknown, how to compute the effective flow area and velocity at the outlet of the nozzle. The zero wall shear model may be valid for a certain time in this kind of flow or ends, when the effective flow area becomes equal to the geometric cross sectional area. One other possibility is that the flow area at the outlet changes abruptly from the zero wall shear model value to the geometric cross sectional area between cavitating and non-cavitating turbulent flow.

The zero wall shear model can also be applied to the cavitating liquid flow in other diesel like nozzles.

5 Conclusions

1. The equations of the Nurick's and zero wall shear model were derived in a detailed way.

2. During the cavitating flow in the nozzle the zero wall shear model was more accurate than the slug flow model. When cavitating flow in the nozzle is computed one-dimensionally, the zero wall shear model is recommended.

3. The slightly rounded inlet edge nozzle cavitates within larger range of the Nurick's cavitation number than the rounded inlet edge nozzle.

4. The compatibility between the zero wall shear model and non-cavitating turbulent flow is unknown.

5. Experiments are needed to validate the zero wall shear model and search the compatibility with non-cavitating turbulent flow.

6. The zero wall shear model can be utilized when improving combustion process in a diesel engine.

7. Basically turbulence and cavitation are three-dimensional. Therefore, three dimensional computational fluid dynamics may give more knowledge of the flow in the nozzle than one-dimensional model.

Acknowledgements

This work was carried out in the research program Flexible Energy Systems (FLEX^{e}) and supported by Tekes – the Finnish Funding Agency for Innovation. The aim of FLEX^{e} is to create novel technological and business concepts enhancing the radical transition from the current energy systems towards sustainable systems. FLEX^{e} consortium consists of 17 industrial partners and 10 research organisations. The programme is coordinated by CLIC Innovation Ltd. `www.clicinnovation.fi`.

References

[Fox10] Fox, R., Pritchard, P., McDonald, A., *Introduction to Fluid Mechanics.* Seventh Edition, Asia, John Wiley & Sons, 2010, 754 pp.

[Kii15] Kiijärvi, J., *Diesel Fuel Injection System.* Material of the course ENER2030 Fluid Mechanics in University of Vaasa. 2015.

[Nur76] Nurick, W., *Orifice Cavitation and Its Effects on Spray Mixing.* Journal of Fluids Engineering, vol. 98, 1976, pp. 681–687.

[Sch97] Schmidt, D., *Cavitation in Diesel Fuel Injector Nozzles.* Dissertation, University of Wisconsin-Madison, 1997, 182 pp.

[Sct66] Schmitt, T., *Untersuchungen zur stationären und instationären Strömung durch Drosselquerschnitte in Kraftstoffeinspritzsystemen von Dieselmotoren.* Dissertation, Fakultät für Maschinenwesen und Elektrotechnik der Technishen Hochschule München, 1966, 124 pp.