A MATHEMATICAL MODEL FOR COMPRESSIBLE FLOWS OF IDEAL GASES THROUGH INFLOW CONTROL DEVICES USED IN ENHANCED OIL RECOVERY

Jean-Luc Olsen, Carlos F. Lange∗
Department of Mechanical Engineering, University of Alberta, Edmonton, Canada
∗carlos.lange@ualberta.ca

Abstract—Inflow control devices (ICDs) are used widely in Alberta to increase the oil yield and decrease both the economic and environmental costs of steam assisted gravity drainage (SAGD) based oil extraction. An ideal ICD is highly fluid selective: it creates pressure drop primarily through mechanisms that are sensitive to the fluid properties unique to undesired fluids, while creating minimal pressure drop through mechanisms strongly dependent on the fluid properties of desired fluids. Industry desires better criteria for comparing ICD designs that capture the ICD’s ability to be fluid selective towards allowing viscous bitumen while restricting water and gases, such as steam and butane (solvent), which tend to flow at higher velocities.

This paper proposes a mathematical model using two loss criteria to describe the performance curves of various simple ICDs operating in the compressible flow regime with ideal gases. The variability of these two loss criteria as functions of the reservoir pressure, molecular mass, and specific heat ratio of the ideal gas under the expected range of reservoir conditions is explored and compared to theoretical predictions and qualitative trends. The two criteria are linked to flow behavior in different parts of the ICDs, and the limits of their applicability are discussed. The ICD performance curves that the mathematical model is based on were obtained using a steady state approach in ANSYS CFX with total energy and SST turbulence models.

To show how the proposed compressible flow model could be used in industry, example comparisons between various simple ICDs are performed. In the first example comparison, ICDs are first sized to have the same flow resistance rating (FRR) of 0.8, then the flow rates of steam through each device are predicted using the proposed mathematical model. In a second example, following a similar method as an in-house code used by our industry partner, the required pressure drop to achieve a target mass flow rate through the best ICD design is calculated.

Keywords-component—compressible flow; SAGD; ideal gases; industry application

I. INTRODUCTION

Steam-assisted gravity drainage (SAGD) based oil production is a method used widely in Alberta to extract bitumen from deep oil reservoirs. SAGD uses two horizontal wells: an injection well to introduce steam into the reservoir, and a production well to drain heated bitumen from the reservoir. In SAGD oil production, inflow control devices (ICDs) can be used along the production well to increase the overall efficiency and lifespan of the well project. The primary mechanism for increasing well efficiency is controlling where oil is drained from the reservoir so to avoid steam from the injection well reaching the production well. A secondary effect that differs significantly between ICD designs is the fluid selectivity of the design. Fluid selectivity is how much an ICD design preferentially favours the flow of viscous fluids, like bitumen, over relatively thin fluids, such as water, steam, and butane (solvent). Both industry and the environment benefit from being able to identify highly fluid selective ICD designs and being able to predict their performance under the wide variety of fluid flow scenarios that can occur during oil extraction.

Initial work by Miersma et al. [1] has focused on the incompressible flow regime with single phase fluids. Miersma created criteria for comparing the fluid selectivity of ICDs by differentiating the effects of fluid viscosity and flow velocity on the overall pressure drop. These criteria, however, begin to lose accuracy when compressible effects become significant. Previous work by the present authors [3] expands upon Miersma’s work by including the compressible flow effects experienced by ideal gases flowing through ICDs up to choked conditions. The methodology used by the authors uses has been validated against experimental measurements of choked...
flows through supersonic ejectors [2]. Mesh independence of
the present method has been confirmed, and general trends
in flow rates and losses were observed in [3] and [4]. The
goal of the present work is to develop a usable functional
form for industry to quantitatively assess the performances
previously observed. The functional form should, ideally,
allow the user to identify fluid selective ICD designs, and
predict the flow rates of various gases through ICDs under
a dynamic range of conditions that would be exceedingly
difficult and expensive to individually compute numerically
or to measure experimentally.

II. CFD METHODOLOGY

All cases were run using the pressure-based ANSYS CFX
2020 R2 coupled solver. Previous work [2] found the $k-\omega$
with SST turbulence model to generally have the best balance
between accuracy and solving time, and for these reasons this
turbulence model is used. The simulations are steady-state, 3-
dimensional, single-phase, and use total energy and ideal gas
models for fluid properties. For advection schemes, a specified
blending factor of 0.5 is applied, which results in a half and
half blend of first order upwind differencing and second order
central differencing schemes to be used. This approach has
been validated [2] and mesh independence has been confirmed
[3].

A. Geometries

The overall geometry, shown in Fig. 1, represents a fraction
of the production well where there is one small pipe (tubing)
inside another, larger pipe (liner). The annular region between
the pipes is a high pressure zone at approximately reservoir
conditions. The inner pipe region is a low pressure zone from
where fluid is pumped to the surface. Connecting the two
regions are two ICDs that regulate the fluid flow rate.

For the current study, the four ICD designs shown in Fig. 2
are considered. Note that in Fig. 2, the top is the high pressure
annular region and the bottom is the low pressure inner pipe
region. All ICD designs have a minimal cross section area
equal to the area of a circle with a diameter of 3.8 mm.

B. Meshes

Meshes for these four geometries range from 12,895,243
elements to 20,810,480 elements. The meshes use structured
mesh in areas away from the ICDs, and tetrahedral mesh
around the ICDs to facilitate the meshing process of complex
ICD designs. Y+ values reach a local maximum of 16.6 in
the inner pipe region, and a local maximum 10.0 in the
ICDs themselves. Mean values within the ICDs, however, are
generally below 1, with the exception of a few cases where the
mean value goes up to 1.7. Y+ values upstream of the ICDs
are lower than the values inside the ICDs. The generally low
Y+ values mean that CFX’s automatic wall treatment method
for its $k-\omega$ with SST turbulence model mostly uses a low-
Reynolds model.

C. Well Conditions

According to Gates and Larter [5], the steam injection pres-
sures for the majority of SAGD operations range from around
1000-5000 kPa, with the average being around 2500 kPa. In-
jection temperatures considered by this source range from 180
to 250 °C. With consideration for the vaporization temperature
of steam and the fact that the likelihood of steam reaching
the production well increases when the pressure is low and
the temperature is high around the production well, boundary
conditions on the low end of the pressure range and the high
end of the temperature range are selected.

At the annulus inlet, pressure boundary conditions of 2100
and 1800 kPa are employed. At the inner pipe inlet and outlet,
pressure boundary conditions ranging from as high as 95% of
the annular pressure, down to 30% of the annular pressure
are employed. All inlets have fixed temperatures of 230 °C,
and use CFX’s default medium turbulence intensity of 5%. All
walls are adiabatic.

D. Fluid Properties

Inside oil wells, there can be emulsions of water, bitumen,
steam, and sometimes solvents such as butane. Phase change
can occur during the high pressure drops through ICDs, and
non-Newtonian behaviors may be observed in the bitumen.
At the current stage of the project, to simplify the analysis
of the basic compressible physics, an ideal gas assumption
is used. In total three fluids are considered. Fluid 1 is an approximation of steam at reservoir conditions, with the same molecular mass (18.015 kg/kmol) and a specific heat capacity ratio of around 1.3. The other two fluids are variations of fluid 1 that make them similar to butane. Fluid 2 has a lower specific heat capacity ratio of around 1.096, and the same molecular mass as steam. Fluid 3 has a higher molecular mass of 58.12 kg/kmol with an adjusted specific heat capacity to keep the same specific heat capacity ratio as fluid 1. All three fluids have a viscosity of 0.016 cP.

E. Mathematical Model

The proposed mathematical model to describe compressible flows of ideal gases through ICDs is shown in equations (1) to (3). $\Delta P$ is the pressure drop; $P_0$ is the total pressure upstream of the ICD, which is approximately equal to the upstream pressure since the upstream velocities for our geometry are always low, so the dynamic pressure component of the total pressure is always negligibly small; $P$ is the pressure downstream of the ICD, also called the back pressure; $P_R$ is the pressure ratio; $C_1$ and $C_2$ are the compressible coefficients being introduced to fit the curve to CFD results; $\dot{m}$ is the mass flow rate; $A$ is the cross sectional area of the narrowest portion of the ICD; $\rho_0$ is the total density upstream of the ICD, which for the same reasoning as the total pressure, is negligibly different from the upstream density; and $\gamma$ is the specific heat capacity ratio of the ideal gas.

$$\Delta P = P_0 - P$$ (1)

$$P_R = \frac{P_0 - C_1C_2\Delta P}{P_0}$$ (2)

$$\dot{m} = C_1A\sqrt{2\rho_0\gamma - 1\left(P_R\right)^\frac{2}{\gamma} - \left(P_R\right)^\frac{\gamma+1}{\gamma}}$$ (3)

Using the `curve_fit` function python from `scipy optimize`, which uses the Levenberg-Marquardt algorithm, a non-linear least squares fit of (3) to CFD data of mass flow versus pressure is created for each flow scenario. An example of the curves being fit to the data for each ICD is shown in Fig. 3. General trends in the relation between geometry and flow losses, and the relation between ideal gas properties and mass flow rate were explored previously [3] [4].

The coefficients found from curve fitting equation (3) to CFD data using fluid 1 and a reservoir pressure of 2100 kPa, as well as the $R^2$ of the fitted curve, can be found in Table I. The high $R^2$ values, all above 99.9%, indicate a very good fit between the proposed mathematical form and the numerical data.

From a mathematical perspective, $C_1$ does not need to appear in equation (2) for the Levenberg-Marquardt algorithm to find coefficients that fit the curve to the CFD data. However, to give the physical interpretation of $C_1$ as being losses before the choke point, including $C_1$ in (2) serves to take the decrease in the total pressure into account when considering the local conditions at the choke point. The higher the pressure drop, the greater the velocity, and the greater the losses in total pressure by the time the flow reaches the choke point of the ICD. The local total pressure and pressure at the choke point are believed to be what govern the flow rate through the ICD. Since local conditions inside the ICD are difficult to measure experimentally, while pressures in the reservoir and inside the production well are far easier to measure, the proposed mathematical model is written to use the easy to measure values and make use of the $C_1$ and $C_2$ coefficients to adjust them to the local conditions inside the ICD.

F. Pre-Choke Losses

The coefficient $C_1$ is being interpreted as representing losses in total pressure before the choke point, where a value of 1 represents 100% efficient flow, without losses, and a value of 0 represents a completely blocked passage. Looking at Table I, we can see that the 0° converging and the 0° converging-diverging ICD designs have $C_1$ coefficients just slightly below 1, which is to be expected since both designs have smooth, streamlined entrances. For the sharp-entranced 40° straight and 30° cone designs, the $C_1$ value is expectedly lower than the smoothed-entranced ICDs.

When changing either the reservoir pressure from 2100 kPa to 1800 kPa, or changing the fluid properties, the $C_1$ coefficient varies as shown in Table II. This table shows that under the range of pressures considered, the reservoir pressure has negligible impact on the $C_1$ coefficient. Fluid properties,
especially the specific heat capacity ratio, have a relatively larger impact on the calculated \( C_1 \) values, but the maximum change seen for the investigated ICD designs is on the order of magnitude of 2%. These variabilities are small enough to acceptably use the \( C_1 \) calculated from fluid 1 for other ideal gases within the considered range of variables.

### TABLE. II: Variability in \( C_1 \)

<table>
<thead>
<tr>
<th>Nozzle Design</th>
<th>Variable Changed</th>
<th>Reservoir Pressure</th>
<th>Molecular Mass</th>
<th>Specific Heat Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° Converging</td>
<td>-0.02%</td>
<td>0.06%</td>
<td>0.25%</td>
<td></td>
</tr>
<tr>
<td>0° Converging-Diverging</td>
<td>-0.07%</td>
<td>0.13%</td>
<td>0.53%</td>
<td></td>
</tr>
<tr>
<td>40° Straight</td>
<td>-0.04%</td>
<td>0.38%</td>
<td>2.03%</td>
<td></td>
</tr>
<tr>
<td>30° Cone</td>
<td>0.03%</td>
<td>-0.25%</td>
<td>0.80%</td>
<td></td>
</tr>
</tbody>
</table>

G. **Post-Choke Losses and Recovery**

The coefficient \( C_2 \) is being interpreted as the sum of losses and recovery in pressure after the choke point. A \( C_2 \) value below 1 represents net losses in pressure between the choke point and the inner pipe, while a value above 1 represents a net recovery. A \( C_2 \) value of 1 means that the pressure conditions at the choke point are the same as further downstream in the open area of the inner pipe. Looking at Table I, we can see that the ICD geometries whose narrowest cross-sections are adjacent to the inner pipe, the 0° converging and the 40° straight designs, have the \( C_2 \) values closest to 1. The 0° converging-diverging ICD, with its smooth expansion into the inner pipe, shows pressure recovery with its \( C_2 \) value of around 2. The 30° cone design is the only nozzle explored that shows pressure between its choke point and the inner pipe.

When changing either the reservoir pressure from 2100 kPa to 1800 kPa, or changing the fluid properties, the \( C_2 \) coefficient varies as shown in Table III. This table shows that under the range of pressures considered, the reservoir pressure has a small impact on the \( C_2 \) coefficient on the order of magnitude of less than 1%. Fluid properties again have a relatively larger impact on the calculated \( C_2 \) values, with the molecular mass changing the coefficient by up to 3.21% in the case of the 0° converging-diverging ICD, and the specific heat capacity ratio having the largest change of -5.6% for the 40° straight ICD. For the intended industrial application, these values are acceptable, and justify using one \( C_2 \) value to predict the flow rate of a variety of ideal gases.

### TABLE. III: Variability in \( C_2 \)

<table>
<thead>
<tr>
<th>Nozzle Design</th>
<th>Variable Changed</th>
<th>Reservoir Pressure</th>
<th>Molecular Mass</th>
<th>Specific Heat Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° Converging</td>
<td>-0.56%</td>
<td>1.44%</td>
<td>-0.98%</td>
<td></td>
</tr>
<tr>
<td>0° Converging-Diverging</td>
<td>0.15%</td>
<td>3.21%</td>
<td>-0.85%</td>
<td></td>
</tr>
<tr>
<td>40° Straight</td>
<td>0.85%</td>
<td>0.04%</td>
<td>-5.60%</td>
<td></td>
</tr>
<tr>
<td>30° Cone</td>
<td>-0.57%</td>
<td>1.07%</td>
<td>-1.65%</td>
<td></td>
</tr>
</tbody>
</table>

### III. EXAMPLE APPLICATIONS

#### A. Idealized Steam Flow Comparison

Following the method used by Miersma et al. [1], the nozzles are sized to all have the same flow resistance rating (FRR) of 0.8, i.e. to produce a specific flow rate of water at a given pressure drop. The specific FRR sized for uses a density of 998 kg/m³, a viscosity of 1 cP, a flow rate of 30 m³/d divided between 4 ICDs, and a pressure drop of 80 kPa. These conditions correspond to a water flow rate of 85.9 g/s per ICD. The diameters of the ICDs sized to this FRR can be found in Table IV.

With the ICDs sized to produce the same flow rate of water at a 80 kPa pressure drop, (3) can be used with the coefficients in Table I to calculate the flow rate of fluid 1, an ideal gas approximation of steam, through the ICDs. Assuming a reservoir pressure of 2 MPa, temperature of 230°C, and the same pressure drop of 80 kPa, the flow rates in the last column of Table IV are calculated. Between the four ICD designs, this calculation shows the 0° converging nozzle to be the best design under the given operating conditions because it has the lowest steam flow in Table IV. Having the lowest steam flow rate means being the most fluid-selective towards allowing water while preventing steam flow for these particular operating conditions. The 0° converging-diverging nozzle is the worst performing nozzle with approximately 8% more steam produced than the 0° converging nozzle.

### TABLE. IV: Idealized Steam Performance for Nozzles Normalized by Water Flow Rate

<table>
<thead>
<tr>
<th>Nozzle Design</th>
<th>Nozzle Diameter (mm)</th>
<th>Idealized Steam Flow Rate (g/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° Converging</td>
<td>2.97</td>
<td>7.98</td>
</tr>
<tr>
<td>0° Converging-Diverging</td>
<td>2.67</td>
<td>8.62</td>
</tr>
<tr>
<td>40° Straight</td>
<td>3.33</td>
<td>7.99</td>
</tr>
<tr>
<td>30° Cone</td>
<td>3.57</td>
<td>8.16</td>
</tr>
</tbody>
</table>

#### B. Pressure Drop Calculation

Our industry partner uses an in-house code for predicting well performance over time. The solver in this code finds the pressure drop required to produce a specific mass flow rate of fluid. To use (3) to calculate the pressure drop for a given mass flow rate requires an iterative method. First, for the known reservoir conditions and ICD losses, the maximum possible flow rate is calculated. For flow rates below the maximum, (3) is rearranged into the root finding problem shown in (4).

\[
\dot{m}_{err} = \dot{m}_s - \dot{m}_s - C_1 A \sqrt{2 \rho_0 \rho_0 \gamma \left( \frac{P_R}{\gamma - 1} - (P_R)^{\frac{\gamma}{\gamma - 1}} \right)} \quad (4)
\]

\( \dot{m}_s \) is the desired mass flow rate, and \( \dot{m}_{err} \) is the discrepancy between the desired mass flow rate and the mass flow rate predicted using a guess for the downstream pressure. Using the fsolve python function from scipy optimize, a root finding algorithm based on Powell’s method [6], with an initial guess of 90% of the reservoir total pressure, the solver quickly...
converges to the pressure required to produce the desired mass flow rate.

Taking the sized nozzles described in Table IV, the loss coefficients from Table I, and applying the above method to all possible mass flow rates for a reservoir at 2000 kPa and 230°C produces the pressure curves in Fig. 4.

All of the sized ICDs have similar performance at low flow rates, but differ significantly at higher flow rates where the pressure drop increases suddenly around a different flow rate for each nozzle. While the sized 0° converging nozzle produces the least amount of steam at a pressure drop of 80 kPa and the sized 0° converging-diverging produces the most steam at this pressure, the sized 0° converging-diverging nozzle has the lowest maximum flow rate. This highlights how fluid-selectivity is highly dependent on the operating conditions, so careful consideration of these conditions is required for selecting the ideal ICD design for a well project.

IV. CONCLUSIONS

A mathematical model to predict the flow rate of ideal gases through ICDs up to choked conditions has been proposed. The model uses two coefficients, $C_1$ and $C_2$, which describe the losses in total pressure before the choke point of the ICD, and the net losses and recovery in pressure after the choke point of the ICD respectively. The proposed coefficients show low variability with respect to reservoir pressures in the range of 1800 to 2100 kPa, and gas properties in the ranges explored, which correspond to approximations of steam and butane. Combined with the sizing method used by Miersma et al. [1] to size ICDs so that they produce the same amount of liquid flow under specific incompressible conditions, the proposed model can be used to predict how fluid-selective an ICD design is towards restricting an ideal gas while allowing the desired liquid under given operating conditions. Additionally, the proposed model can be solved numerically to find the pressure drop that produces a desired flow rate, so long as that flow rate is below the physical maximum.

V. FUTURE WORK

A more in depth analysis of the effect of Reynolds Number on the proposed loss coefficients is planned, along with an exploration of the fluid-selectivity of ICD designs over a range of operating conditions instead of only one. The curve fitting approach with consideration to how many simulations are needed and whether having more CFD data around the around choke point increases the accuracy of the loss coefficients could be explored. An optimization of the initial guess for the pressure when solving for the pressure required to produce a specific mass flow rate numerically could be performed. Additional physics, including real gas effects, phase change, and multi-phase flows are intended as parts of future projects.

ACKNOWLEDGMENT

The authors would like to thank NSERC and RGL Reservoir Management Inc. for providing funding for this research.

REFERENCES