space of the parent window. ANSYS FLUENT is able to read geometries generated in GAMBIT and model fluid flow within them. It can model various scenarios using computational fluid dynamics, including compressible and incompressible flow, multiphase flow, combustion, and heat transfer.

2.5 Mesh design

Grid generation is a key issue in flow simulation as it governs the stability and accuracy of the flow predictions. For the present case of flow through orifice, unstructured tetrahedral hybrid cells were used to discretize the entire flow domain. The size of the mesh was kept finer in the orifice region to capture the existence of high velocity and pressure gradients. The meshed geometry of the slotted orifice is shown in Fig. 2.1.

![Meshed slotted orifice geometry](image)

Fig.2.1 Meshed slotted orifice geometry (A) rectangular perforations with $l/w = 3.0$; (B) circular perforations.

3 MATERIALS AND METHODS

3.1 Overview

This paper describes the numerical studies to establish the effect of velocity and pressure distributions on the performance of the slotted orifice. Two sets of slotted orifices which were rectangular perforations and one with a circular perforation and a $\beta$ ratio of 0.40 were simulated in a 1.6 m horizontal pipe using the $k-\varepsilon$ turbulence model over a range of parameters, i.e. gas volume fraction (GVF) and gas mass flow rate. Besides, the pressure was also varied in the range of 240000 – 360000 Pa. The commercial CFD code, FLUENT 6.3 was used to model the wet gas flow.
3.2 Simulation Methodology

3.2.1 Geometry Details

In the present study, 5D upstream pipe length and 10D downstream pipe length were provided in the computational flow domain for specification of the boundary conditions. The reason for the longer downstream pipe length is to make sure that the entire flow profile is captured as it comes through the constriction. As the flow is considered to be fully developed, it does not matter if the upstream pipe is shorter than the downstream pipe. Generally, a pipe length of 100D–200D is needed for multiphase flow to fully develop. The one-dimensional (1-D) schematic layout of the computational flow domain is shown in Fig. 3.1. The geometries of the orifices were created in a GAMBIT 2.3.16 pre-processor.

![Fig. 3: Schematic layout of the flow domain used for the numerical study of orifices.](image)

Flow predictions were carried out to study the performance of different geometrical orifice flow meters consisting of a 5 mm plate with $\beta = 0.40$ in a Schedule 80 horizontal pipe (equivalent to an inner diameter of 105.74 mm). Figs. 1a and 1b show the general structure of the rectangular and circular orifices used in the flow investigation. The geometric details of the slots are summarized in Table 3.
Table 3
Dimensions of the orifices with $\beta = 0.40$.

<table>
<thead>
<tr>
<th>Case</th>
<th>Perforation</th>
<th>$l/w$</th>
<th>$D$ (mm)</th>
<th>$d/w$ (mm)</th>
<th>$l_1$ (mm)</th>
<th>$x_1$ (mm)</th>
<th>$x_2$ (mm)</th>
<th>No. of perforations</th>
</tr>
</thead>
<tbody>
<tr>
<td>1A</td>
<td>Rectangular</td>
<td>3.0</td>
<td>105.74</td>
<td>3.61</td>
<td>10.82</td>
<td>3.00</td>
<td>5.00</td>
<td>6 12 18</td>
</tr>
<tr>
<td>2A</td>
<td>Circular</td>
<td>-</td>
<td>105.74</td>
<td>7.05</td>
<td>-</td>
<td>3.00</td>
<td>5.00</td>
<td>6 12 18</td>
</tr>
<tr>
<td>3A</td>
<td>Standard orifice</td>
<td>-</td>
<td>105.74</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0 0 0</td>
</tr>
</tbody>
</table>

For the circular perforations, the geometry was first design on a graph paper to get the coordinates for each circle, and then the data was transferred into the GAMBIT 2.3.16 to draw the volume. In the Gambit, firstly the perforated orifice was drawn followed by the circles in the orifice. Then, pipes with the respective length 0.5 m for the upper stream and 1m for the lower stream were drawn and combine with the orifice. The data for the all drawing was tabulated as follows:

Orifices

Fig.3.1. Geometry of the perforated orifice.

Height: 0.005 mm
Radius: 0.050 mm