3D numerical study of a feed jet in a rotating flow-field

D J Jiang and S Zeng
Department of Engineering Physics, Tsinghua University, Beijing 100084, China
E-mail: jiangdj@tsinghua.edu.cn

Abstract. A contribution of a feed drive in the total counter-current flow in a gas centrifuge for isotope separation is an important problem in optimization of its separation performance. A 3D model is used to simulate flow structures of a feed jet in a rotating flow-field. By using a CFD code, the details of a feed jet are obtained under the axial feed jet boundary condition. It is demonstrated that because of the vacuum regime in the region near the axis of rotation, the results of numerical simulation of a CFD code bring errors. The 3D DSMC simulation is a feasible method to overcome this problem in the future.

1. Introduction
A gas centrifuge (GC) is a rapidly rotating hollow cylinder, in which an isotope mixture is separated by the centrifugal force in a radial direction. Further separation in a GC is carried out in the axial direction by means of the axial counter-current flow. Usually, it excites in a working chamber of a GC to get higher separation efficiency by three possible ways: the thermal drive by a temperature gradient on a sidewall of a centrifuge machine, the mechanical drive takes place by the mechanical braking of a rotating flow by a stationary scoop, and the feed drive induces by retarding influence of the ingoing flow to the GC [1].

A process gas is introduced into the spinning rotor from a stationary central post and removes from stationary pipes called scoops located at either end of the rotor. Note that originally the supersonic feed jet flow locates in the rarefied flow region near the axis of rotation (a vacuum core). Fu et. al [2] applied the eigenvector method to study the radial feed jet. The conclusion was that the feed flow speed becomes the subsonic one on the boundary between a vacuum core and a viscous gas. Roblin et al. [3] used the direct simulation by the Monte-Carlo method (DSMC) to simulate the flow field of the radial feed jet, and the feed conditions on the viscous boundary were modified according to the results of the simulation.

As it turned out, DSMC is a feasible method to make a numerical simulation of the rarefied flow in the vacuum core, but very time consuming one. Based on our previous work of the 3D radial feed jet in a GC [4], in this paper the 3D flow structure of the axial feed jet in the rapidly rotating flow is investigated by the CFD method.

2. Calculation procedure
The control 3D Navier-Stokes equations of the feed jet flow are as follows,

\[
\frac{\partial U}{\partial t} + \frac{\partial (F - F_r)}{\partial x} + \frac{\partial (G - G_r)}{\partial y} + \frac{\partial (H - H_r)}{\partial z} = 0
\]
where the footnote “ν” means a viscous term,

\[
F_{\tau} = \begin{bmatrix}
0 \\
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz} \\
k \frac{\partial T}{\partial x} + u \tau_{xx} + v \tau_{xy} + w \tau_{xz}
\end{bmatrix},
\]

\[
G_{\tau} = \begin{bmatrix}
0 \\
\tau_{yy} \\
\tau_{yx} \\
\tau_{yz} \\
k \frac{\partial T}{\partial y} + u \tau_{yy} + v \tau_{yx} + w \tau_{yz}
\end{bmatrix},
\]

\[
H_{\tau} = \begin{bmatrix}
0 \\
\tau_{zz} \\
\tau_{zx} \\
\tau_{zy} \\
k \frac{\partial T}{\partial z} + u \tau_{zz} + v \tau_{zx} + w \tau_{zy}
\end{bmatrix},
\]

Here \(u, v, w\) are the Cartesian velocity vector components, \(\rho\) is the density of a process gas, \(p\) is the pressure of the process gas, and \(e\) is the total energy per unit volume.

The Finite Volume Method (FVM) was adopted to discretize the control equations and the 2-order implicit upwind scheme was used to capture the shock waves. The realizable turbulence model was applied for simulation the turbulence caused by a supersonic feed jet.

3. Analysis model and calculation conditions

The analysis model and coordinate system are shown in figure 1. The process gas flows into a GC is passing through the feed hole in a stationary stand. In order to evaluate the interaction between a feed flow and a rotating gas, a rotating rotor wall is in considered in the 3D approach. Only the flow region passing through the feed hole in a stationary stand. In order to evaluate the interaction between a feed pass and a rotating gas, a rotating rotor wall is in considered in the 3D approach. Only the flow region below the feed inlet boundary plane is calculated to induce the computation workload. Also the flow in a waste chamber is fully not considered.

Using the Iguassu model centrifuge [5], the calculation domain is the central area near a feed hole. All other parameters used in calculation are listed in table 1.

The hexahedral mesh system is generated in the simulation procedure. The typical grids are demonstrated in figure 2. We refined the grid near the rotor and the feed inlets to employ the parameters variation of this area. 2,016,000 hexahedral cells were used in this simulation.

| Parameter             | Value          |
|-----------------------|----------------|
| Radius of rotor \(r_a\) | 6 cm           |
| Height of rotor \(8r_a\) |               |
| Angular velocity \(\omega\) | 10\(^4\) rad/s |
| Peripheral speed       | 600 m/s        |
| Wall pressure          | 13 300 Pa       |
| Average temperature \(T_0\) | 300 K        |

Table 1: Parameters of the Iguassu Centrifuge
Figure 1. Analysis model for the feed inlet, stationary feed pipe, rotating rotor wall, and end plate in absence of a withdraw chamber.

Figure 2. Generated meshes in an angular section

Figure 3. Meshes in an axial section

The boundary condition settings used in calculation are shown in Table 2.

| Boundary          | Boundary Type | Conditions setting               |
|-------------------|---------------|----------------------------------|
| Feed inlet        | Pressure inlet| Pressure, velocity, temperature  |
| Up boundary       | Rotating wall | Rotating speed; temperature; No-slip |
| Inner Boundary    | Station wall  | temperature; No-slip             |
| Rotor wall        | Rotating wall | Rotating speed; temperature; No-slip |
| Outlet Boundary   | $T_0 = 300 \text{ K}$ | Pressure, temperature           |

The initial pressure was given according to the rotating rigid body assumption as following:


\[ p = p_0 \exp \left( \frac{M \Omega^2 r^2}{RT_0} \left( \frac{r^2}{r_0^2} - 1 \right) \right). \]

The initial temperature is fixed as the uniform distribution of 300 K. Also, the velocity in the radial and axial directions is set as zero. The tangential velocity is established according to the formula below

\[ v_\theta = \omega r. \]

4. Results analysis and discussion

Expand waves were captured in simulation, which can be seen from the flow distribution in the section x=0 in Figure 4. The following phenomena can be observed:

1. In axial direction, the expansion waves appear near the feed inlet, but disappear rapidly.
2. The gas flowing into the gas centrifuge affects with the rotating gas, so the tangential velocity decreases near the feed jet flow.
3. The stream lines composited by the radial and axial velocity note that the feed gas flows into the centre region when it enters into gas centrifuge.

![Figure 4](image_url)

(a) Mach number (b) Tangential velocity (c) Pressure

Figure 4. The parameters of the flow distribution in the x=0 section.

The obvious deformation of the feed jet in the tangential direction is clearly visible. Bethink that only the axial feed jet (without the radial and tangential velocities) was simulated in this research. The affected range in axial range by feed jet is only about 1 cm, which is smaller than expected. It means that there is no much effect on the counter-current by feed jet.
Two phenomena need to be discussed:

1. The affected flow region is quite small, especially in axial direction and the edge of the explosion wave is clear. In fact, the influenced area should be larger due to the low pressure in centre region.

2. The gas flows into the centre area as soon as it enters the rotating flow field. The reason is the loss of the axial momentum of the gas and the radial gradient of the pressure. It means that the influence of the feed jet on the flow field in the gas centrifuge is limited.

At present few study papers on 3D axial feed jet in rotating rarefied flow were published, so it is necessary to perform simulation using other method in order to verify the simulation results. A feasible way is using 3D DSMC method to do this, which needs higher calculation ability.

5. Conclusions
The 3D numerical simulation was carried out to study the feed jet flow in the gas centrifuge for isotope separation. Using CFD technique, the flow structure of the feed jet was obtained. Simulation result notes that the influence of feed jet is limited, and the feed gas flows into centre region immediately when it entered GC. The expand waves decrease in axial direction; the affected range is only about 1 cm.

Because of the lack of the similar numerical results and experimental data of the axial feed jet, more study work should be done using other methods, such as 3D DSMC.

Acknowledgement
This research was carried out in the framework of the project No 1151101155 supported by NSFC.

References
[1] Doneddu F, Roblin F and Wood H G 2000 Sep. Sci. Technol. 35 1207-21.
[2] Fu R et al 1995 Tsinghua Sci. Technol. 1 61-5.
[3] Roblin P et al 2000 22nd International Symposium on Rarefied Gas Dynamics (Sydney Australia).
[4] Jiang D J, Zeng S 2008 Int. Workshop on Separation Phenomena in Liquids and Gases (Brazil).
[5] Kai T et al 2000 J. Nuc. Sci. Technol 37 153-65.