Computational Fluid Dynamics Investigation of a Smelting Cyclone
Şerzat Safaltın, Sebahattin Gürmen
Department of Metallurgical and Materials Engineering, Istanbul Technical University, 34469 Maslak Istanbul, Turkey

Abstract
Cyclone type furnaces are used in smelting processes (e.g. Kivcet, Contop) as an alternative to vertical furnace components regarding to their lower volume and high rate of particle reduction mechanisms. Application of cyclone smelting furnaces are first introduced in 1950s with intention to react metal powders in high purity oxygen gas at elevated temperatures to reach good agreement by means of volume shrinking and rapid reactions. High temperature, thermal expansion and high gas injection velocities introduce turbulent flow fields which result as complexity in control of process. In this paper, finite volume method is used to numerically simulate the convective heat transfer in flow field of a high speed smelting cyclone plant.

1. Introduction
Autogenous or more commonly flash smelting process is based on concentrate smelting with initial ignition and continuous burning of reduction gas and pulverized raw materials into matte [1]. Commonly vertical shafts with burners are used for production of copper, zinc, lead and other relevant non-ferrous metals. Due to design considerations like volume, combustion rate and efficiency, cyclone chambers were suggested in mid 1950s. Smelting cyclones are recognized for their small volume, high gas velocities, reaction rates, thermal efficiencies and most importantly high smelting rates[1, 2]. Smelting cyclones consist of a large aspect ratio diameter-height chamber, equipped with one or more tangential tuyeres that act as an inlet of concentrate and reduction gas (high purity oxygen), and to openings on top and down of chamber. Top opening is used as an exhaust while bottom lets heavier products to enter matte region of furnace. Angled inclination of chamber walls enable combustion gases to form swirl while rapid expansion, high oxygen content and high temperature enable reduction reactions [1-3]. Due to geometry and high input velocities, swirl velocities reach upto 50 m.s⁻¹ during continuous reduction [1-2].

Process dependent approximations are considered with both analytical and practical studies in order to improve efficiencies while forming better understanding with related transport phenomenas. Especially, swirl mechanism is directly related to turbulent flow dynamics, but it is hard to visualize flow profiles, temperature regions and mass flow rates of input/output regions with practical observations. Modeling and simulation tools can be used to show related phenomenas with well known numerical methods based on finite approaches [4-6]. In this paper, flow fields, turbulent properties and thermal interactions of a smelting cyclone is studied with finite volume and steady-state approximations. The numerical method is based on the OpenFOAM (Open Field Operation and Manipulation) C++ library pack with state-of-art numerical schemes and solvers optimized for computational fluid dynamics calculations.

2. Experimental
In silico studies are divided into two parts. In the first part, model geometry definition, meshing, boundary conditions and parameters estimations are evaluated. Second part includes numerical simulation, parameter estimation and methodology.

2.1. Geometry, Meshing, Boundary Conditions and Material Properties
A 3-D geometry with approximately 4 m height, 2 m diameter size and rectangular shape intake is taken as a component of interest. Since the geometry has complex features like rectangular intake-cylinder cross section, exhaust region matches, angled conical surfaces etc., snappyHexMesh algorithm is used to obtain low skewness, high orthogonal homogeneous mesh structure with nearly 400000 cells, which is adequate for simulations.
2.2. Numerical Simulation
Thermophysical approach for pure mixture of a perfect gas which is sensible to internal energy is used as material model, coupled with Prandtl constant calculations into Reynolds averaged Navier-Stokes (RANS) formula. Primary constants are calculated given in Table 1. RANS is configured to $k-\omega$ SST (shear stress transport) turbulence model and computed with 2 separate solvers. Pressure is calculated with geometric algebraic multigrid solver with Gauss-Seidel smoother function, while velocity, turbulent kinetic energy, rate of dissipation near wall and rate of dissipation in large scale fluid is solved with smooth solver with symmetric Gauss-Seidel smoother function. Relaxation factors for velocity and turbulent constant are used for better error evaluation for iteration steps.

### Table 1. Primary constants for Reynolds Averaged Navier-Stokes and turbulence model

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Formula</th>
<th>Value[unit]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fully-Developed Velocity inlet</td>
<td>inlet value</td>
<td>10[ms⁻¹]</td>
</tr>
<tr>
<td>Hydraulic Diameter, $D_h$</td>
<td>$4\text{Area/Wetted Perimeter}$</td>
<td>0.685[m]</td>
</tr>
<tr>
<td>Turbulent Length Scale, $L$</td>
<td>$C_p0.75k1.5s$</td>
<td>0.04[m]</td>
</tr>
<tr>
<td>Reynolds Constant, $Re$</td>
<td>$UD_h/\nu$</td>
<td>$4.6\times10^5$</td>
</tr>
<tr>
<td>Turbulent Intensity, $I$</td>
<td>$0.16Re^{0.125}$</td>
<td>3.2%</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy in Freestream, $k$</td>
<td>$1.5(UL)^{0.5}$</td>
<td>0.15[m²s⁻³]</td>
</tr>
<tr>
<td>Specific Dissipation at wall, $w$</td>
<td>$C_p0.75kL$</td>
<td>18[s⁻¹]</td>
</tr>
</tbody>
</table>

3. Result and Discussion

3.1. Flow Fields
The plots in figure 2, 3 and 4 display the $x$, $y$ plane velocity fields and stream profile of overall cyclone smelter. Stream profile is integrated with Runge-Kutta integrator to volumetric cells in a virtual spherical region to initiate streamlines. Figure 2 and 3 shows high gas velocity near walls and cyclone flow formation in mid and outlet region of cyclone smelter. Figure 4 shows streamline profiles and velocities. Due to low aspect ration of outlet region, streamlines show angled incline at bottom.
3.2. Temperature Distribution

The plot in figure 5 shows the temperature profile after reaching thermally equilibria. It is clearly seen that due to very high velocities and turbulent energy dissipations, thermal characteristics are based on primarily convection rather than convection & radiation.

4. Conclusion

Swirls, turbulent fields and temperature profiles of a smelting cyclone modeled with finite volume method. steady state phenomena investigated with iterative solvers for RANS k-ω SST model. 3 dimensional 400000 cell mesh is structured with snappyHexMesh algorithm. Simulation results show that outlet geometry entities require optimization for better cyclone formation and thermal characteristics show that due to high velocities, heat transfer is based on primarily convective mechanisms. Model can further improved with kinematic particle cloud or discrete element methods and chemical reduction reactions.

References