CFD-Based Optimization of Heat and Mass Transfer Characteristics in a Co-Current Spray Dryer for Silica Slurry Drying

Bhaskar H B[1], Harish T N[2], Praveenkumar M[3]
[1] *Department of Mechanical Engineering, Sri Siddhartha Institute of Technology, Tumakuru, Karnataka, India*
[2] *Department of Mechanical Engineering, Sri Siddhartha Institute of Technology, Tumakuru, Karnataka, India*
[3] *Department of Mechanical Engineering, GM Institute of Technology, Davanagere, Karnataka, India*

***Abstract*—Spray drying is one of the most efficient industrial processes for converting liquid feed into dry powder and is extensively employed in the food, pharmaceutical, chemical, and ceramic industries. The drying performance of a spray dryer is governed by complex interactions among airflow, heat transfer, turbulence, droplet evaporation, and particle transport, making experimental investigation both time-consuming and expensive. Computational Fluid Dynamics (CFD) provides an effective alternative for analysing these coupled transport phenomena and optimizing dryer performance.**

This paper presents a numerical investigation of a pilot-scale co-current spray dryer used for silica slurry drying. A three-dimensional model of the spray dryer was developed and simulated using ANSYS Fluent. The airflow inside the chamber was modelled using the standard *k–ε* turbulence model, while silica slurry droplets were tracked using the Euler–Lagrange Discrete Phase Model (DPM). The study evaluates velocity distribution, pressure variation, temperature field, turbulent kinetic energy, particle trajectories, droplet evaporation, and heat transfer characteristics within the drying chamber. Numerical predictions are compared with representative experimental data to assess the reliability of the CFD model.

The simulation results show that the hot air undergoes rapid cooling immediately after contacting the atomized slurry because of intense evaporation, followed by a gradual temperature reduction towards the outlet. The predicted velocity field demonstrates stable co-current flow with limited recirculation, while the particle trajectory analysis confirms effective dispersion of droplets within the drying chamber. The heat transfer coefficient and evaporation behaviour indicate efficient moisture removal under the selected operating conditions. Validation results demonstrate good agreement between CFD predictions and experimental observations, supporting the applicability of CFD as a reliable tool for analysing and optimizing industrial spray drying processes.

***Index Terms*—Computational Fluid Dynamics, Spray Dryer, Silica Slurry, Heat Transfer, Mass Transfer, ANSYS Fluent, Discrete Phase Model, Particle Evaporation.**

I. INTRODUCTION

Spray drying is a well-established thermal drying process widely employed for converting liquid feed materials into dry powder products in a single continuous operation. Owing to its high drying rate, short residence time, and ability to produce powders with controlled particle size and moisture content, spray drying has become an essential process in the food, pharmaceutical, chemical, ceramic, and advanced materials industries. The process involves atomizing a liquid feed into fine droplets and exposing them to a stream of hot air, resulting in rapid evaporation of moisture and the formation of dry particles. Its capability to handle solutions, suspensions, emulsions, and slurries makes spray drying one of the most versatile industrial drying technologies. These characteristics are described in the introductory chapter of your project report.

Among various industrial applications, drying of silica slurry is particularly important because the final powder quality directly influences the performance of ceramic products, catalysts, fillers, and specialty materials. The drying performance depends on several interacting phenomena, including airflow distribution, turbulence, heat transfer, mass transfer, droplet evaporation, particle transport, and residence time inside the drying chamber. Any imbalance in these parameters may result in incomplete drying, excessive wall deposition, poor powder recovery, non-uniform particle size distribution, and increased energy consumption. Your report identifies these interacting transport phenomena as the key engineering challenge for silica slurry spray drying.

Traditionally, optimization of spray dryers has relied on experimental investigations. Although experimental studies provide valuable information, direct measurement of velocity, temperature, pressure, turbulence, and droplet behaviour inside the drying chamber is difficult because of the high operating temperatures and limited accessibility. Furthermore, conducting repeated experiments under different operating conditions increases both development time and cost. Consequently, numerical simulation has become an effective approach for analysing internal dryer behaviour and supporting process optimization.

Computational Fluid Dynamics (CFD) enables simultaneous prediction of airflow characteristics, pressure distribution, temperature field, turbulence, particle trajectories, and droplet evaporation within a spray dryer. By solving the governing equations of fluid flow, heat transfer, and particle transport, CFD provides detailed information that is difficult to obtain experimentally. The combination of the Reynolds-Averaged Navier–Stokes (RANS) equations, the standard **k–ε** turbulence model, and the Euler–Lagrange Discrete Phase Model (DPM) provides a practical framework for simulating industrial spray drying processes. Your methodology chapter adopts this numerical approach for the present investigation.

The present study focuses on the CFD analysis of a pilot-scale **co-current spray dryer** used for drying silica slurry. A three-dimensional model of the dryer was developed using Siemens NX and analysed using ANSYS Fluent. The investigation examines velocity distribution, pressure variation, temperature profiles, turbulence characteristics, particle trajectories, droplet evaporation, and heat transfer inside the drying chamber. Numerical predictions are compared with representative experimental observations to evaluate the reliability of the developed CFD model. This scope is consistent with the methodology and results documented in your report.

The findings of this study demonstrate how CFD can be used to understand the complex transport phenomena occurring during spray drying and provide engineering guidance for improving dryer performance, enhancing drying efficiency, minimizing wall deposition, and reducing the need for extensive experimental trials. The methodology presented in this work also offers a useful framework for future CFD-based design and optimization of industrial spray drying systems.

Novel Contributions of the Present Work

The major contributions of this study are:

- Development of a three-dimensional CFD model of a pilot-scale co-current spray dryer for silica slurry drying.
- Numerical prediction of airflow, pressure, temperature, turbulence, and particle transport using ANSYS Fluent.
- Investigation of droplet evaporation and heat transfer characteristics through the Euler–Lagrange Discrete Phase Model.
- Validation of CFD predictions using representative experimental results.
- Identification of operating conditions that improve drying efficiency and reduce wall deposition.
- Demonstration of CFD as an effective engineering tool for spray dryer design optimization.

II. LITERATURE REVIEW

Spray drying has been extensively investigated over the last two decades because of its importance in producing high-quality powders with controlled particle size, moisture content, and flow characteristics. Recent studies have increasingly adopted Computational Fluid Dynamics (CFD) to understand the complex interaction between airflow, heat transfer, droplet evaporation, turbulence, and particle transport inside spray dryers. The literature reviewed in your thesis shows that CFD has become an effective tool for optimizing spray dryer design and operating conditions while reducing dependence on expensive experimental investigations.

Several researchers have developed CFD models to predict airflow behaviour and temperature distribution inside spray dryers. These studies demonstrated that the internal flow pattern significantly influences particle residence time, evaporation rate, and drying efficiency. The use of Reynolds-Averaged Navier–Stokes (RANS) equations together with the standard **k–ε** turbulence model has been widely accepted for predicting turbulent airflow in industrial spray dryers because it provides a reasonable balance between computational cost and prediction accuracy.

Other researchers have incorporated the Euler–Lagrange Discrete Phase Model (DPM) to simulate droplet motion, particle trajectories, and moisture evaporation. Their investigations showed that droplet size distribution, atomizer characteristics, inlet air temperature, and airflow velocity have a considerable influence on drying behaviour and powder quality. CFD simulations also revealed that improper airflow distribution may create recirculation zones, increase wall deposition, and reduce drying efficiency.

Recent investigations have focused on improving heat and mass transfer inside spray dryers through optimization of operating parameters such as inlet air temperature, feed flow rate, atomizer design, chamber geometry, and slurry properties. Numerical studies reported that higher inlet temperatures generally enhance moisture removal; however, excessive temperatures may increase wall deposition or affect product quality. Similarly, slurry viscosity has been identified as an important parameter influencing atomization quality, droplet breakup, and evaporation rate.

Although considerable research has been carried out on spray drying of food products, pharmaceutical formulations, and dairy materials, comparatively fewer studies have investigated **silica slurry drying** in a **pilot-scale co-current spray dryer** using a comprehensive CFD model. Many published investigations concentrate only on airflow analysis or single-droplet evaporation, while relatively few simultaneously analyse velocity distribution, pressure variation, temperature field, turbulence characteristics, particle trajectories, droplet evaporation, and heat transfer under realistic operating conditions. Your report highlights this gap and motivates the development of a more comprehensive CFD model.

In the present work, a three-dimensional CFD model of a pilot-scale co-current spray dryer is developed using Siemens NX and analysed using ANSYS Fluent. The standard **k–ε** turbulence model is employed for the continuous air phase, while silica slurry droplets are modelled using the Euler–Lagrange Discrete Phase Model. The investigation integrates airflow prediction, heat transfer analysis, particle tracking, droplet evaporation, and validation against representative experimental observations within a single numerical framework. This integrated approach extends the scope of previous studies and provides practical guidance for improving industrial spray dryer performance.

Research Gap

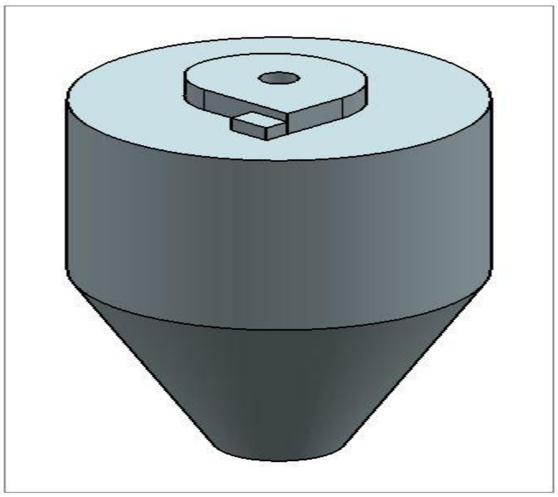
The review of the available literature indicates that:

- Most published studies focus on food, dairy, or pharmaceutical spray drying rather than silica slurry applications.
- Many investigations analyse only one aspect of the drying process, such as airflow, droplet evaporation, or heat transfer, instead of combining all major transport phenomena.
- Limited research is available on pilot-scale co-current spray dryers operating under conditions representative of silica slurry drying.
- Experimental investigations alone cannot provide detailed information on the internal velocity, temperature, pressure, turbulence, and particle transport fields.
- Therefore, there is a need for a comprehensive CFD model capable of simultaneously predicting fluid flow, heat transfer, droplet evaporation, and particle trajectories to support process optimization and improve drying efficiency.

III. METHODOLOGY

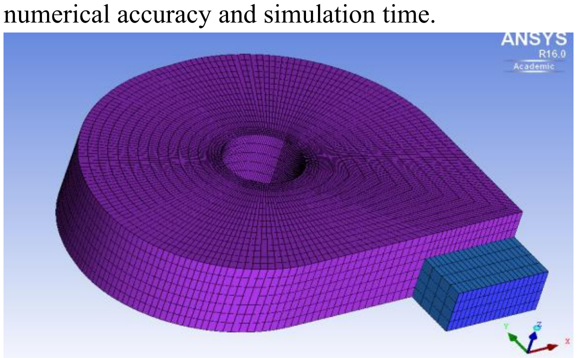
A. Geometry Development

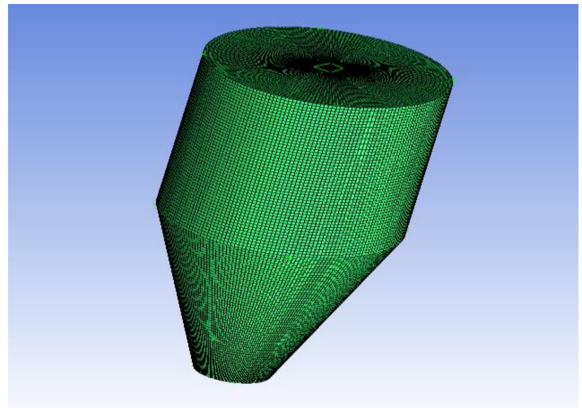
A three-dimensional model of the pilot-scale co-current spray dryer was developed using Siemens NX based on the actual dimensions of the experimental setup described in the project report. The computational model includes the drying chamber, hot air inlet, atomizer, feed inlet, powder outlet, and exhaust outlet. The geometry was designed to reproduce the physical configuration of the spray dryer while maintaining sufficient geometric accuracy for CFD analysis. After completion of the CAD model, the geometry was imported into ANSYS Fluent 16.0 for preprocessing and numerical simulation. This modelling approach follows the geometry development described in the thesis.



B. Computational Mesh Generation

The computational domain was discretized using an unstructured mesh composed of tetrahedral elements with local refinement near the atomizer, air inlet, and chamber wall regions. Mesh refinement in these areas improves prediction of high velocity gradients, droplet dispersion, and evaporation behaviour. A mesh independence assessment was performed to ensure that further refinement produced only negligible changes in the predicted flow variables while keeping the computational cost reasonable. The final mesh provided an appropriate balance between numerical accuracy and simulation time.





C. Governing Equations

The airflow inside the spray dryer was assumed to be three-dimensional, turbulent, and incompressible. The numerical model solves the governing equations for conservation of mass, momentum, and energy.

Continuity Equation

$$\frac{\partial \rho}{dt} + div.(\rho u) = 0 \ldots\ldots$$

where

- ρ = density (kg/m³)
- **V** = velocity vector (m/s)

Momentum Equation

$$\rho\frac{Du}{Dt}, \rho\frac{Dv}{Dt}, \rho\frac{Dw}{Dt} \ldots\ldots$$

where:

- P = static pressure
- μ = dynamic viscosity
- F = body force

Energy Equation

$$\rho\left[\frac{\partial h}{\partial t} + div.(hV)\right] = -\frac{Dp}{Dt} + div(k\ grad\ T) + \Phi \ldots\ldots$$

where:

- T = temperature
- Cp = specific heat
- k = thermal conductivity
- Sh = heat source term

These equations govern the transport of mass, momentum, and thermal energy throughout the spray drying chamber and form the basis of the CFD simulation described in the report.

D. Turbulence and Particle Models

Turbulent airflow was simulated using the standard k–ε turbulence model**,** which provides reliable prediction of recirculation zones and overall flow behaviour in industrial spray dryers. The transport equations for turbulent kinetic energy (k) and turbulence dissipation rate (ε) were solved together with the momentum equations.

Silica slurry droplets were modelled using the Euler–Lagrange Discrete Phase Model (DPM). In this approach, the continuous phase (hot air) is solved in an Eulerian framework, while individual droplets are tracked in a Lagrangian reference frame. Heat and mass transfer between the droplets and the surrounding air were included to simulate moisture evaporation during the drying process. This modelling strategy is consistent with the methodology adopted in your thesis.

E. Boundary Conditions

The numerical simulation was carried out under representative operating conditions corresponding to the pilot-scale spray dryer.

The principal boundary conditions applied were:

Parameter	Value
Drying air inlet temperature	473 K
Air inlet velocity	As specified in the experimental setup
Feed material	Silica slurry
Turbulence model	Standard k–ε
Particle model	Euler–Lagrange DPM
Solver	Pressure-based, steady-state
Pressure outlet	Atmospheric pressure

The hot drying air entered through the top inlet, while silica slurry droplets were injected through the atomizer in the same direction, establishing a **co-current** flow configuration. Moisture evaporation, particle transport, and heat transfer were computed simultaneously during the solution process.

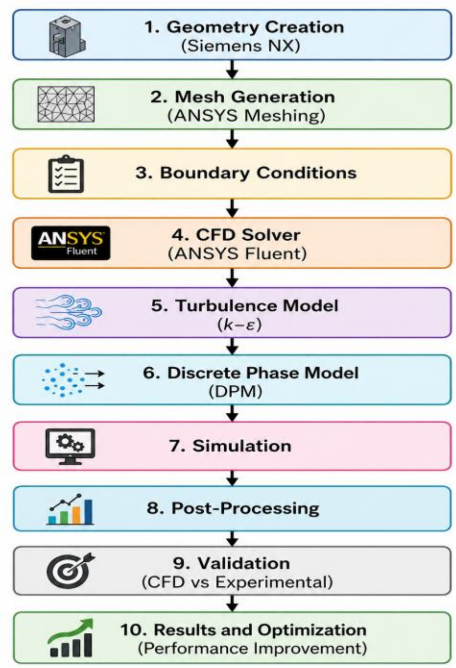
F. Numerical Solution Procedure

The governing equations were solved using the pressure-based solver available in ANSYS Fluent 16.0**.** Pressure–velocity coupling was achieved using the SIMPLE algorithm. Second-order discretization schemes were employed for momentum, energy, and turbulence equations to improve numerical accuracy. Iterations continued until convergence was achieved, with residuals reduced to acceptable limits and key monitored variables showing stable values.

The numerical results obtained from the CFD model include:

- Velocity distribution
- Pressure distribution
- Temperature contours
- Velocity streamlines
- Turbulent kinetic energy
- Particle trajectories
- Droplet evaporation characteristics
- Heat transfer coefficient
- Heat flux distribution

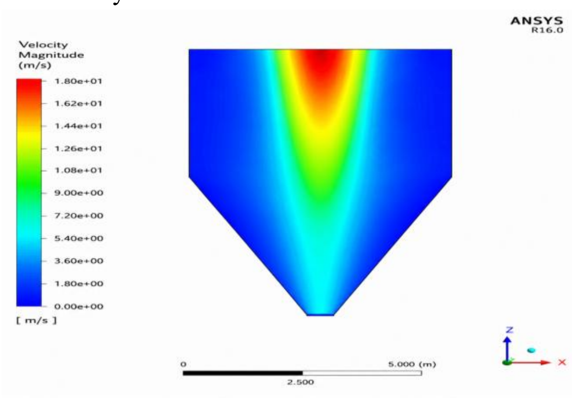
These predictions were subsequently analysed and compared with representative experimental observations to evaluate the reliability of the CFD model.



IV. RESULTS AND DISCUSSION

The Computational Fluid Dynamics (CFD) simulation of the pilot-scale co-current spray dryer was carried out using ANSYS Fluent 16.0 to investigate the coupled behaviour of airflow, heat transfer, particle transport, turbulence, and droplet evaporation during silica slurry drying. The numerical predictions provide detailed information about the internal drying process, which is difficult to obtain through experimental measurements. The simulation results were analysed using contour plots, streamline distributions, particle trajectory plots, numerical data, and validation with representative experimental observations, consistent with the results chapter of your project report.

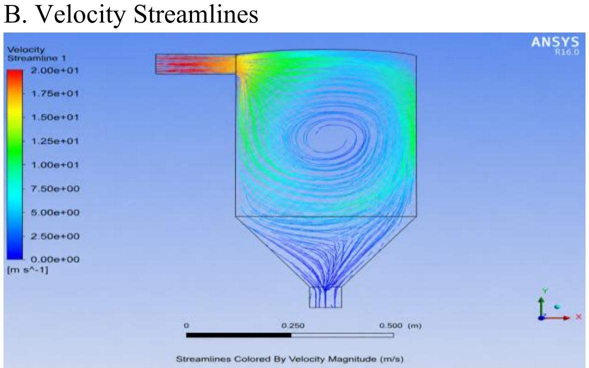
A. Velocity Distribution



The velocity contour demonstrates the airflow behaviour inside the spray dryer. The maximum air velocity occurs near the hot air inlet because of the high inlet momentum. As the air enters the drying chamber, the flow expands uniformly and gradually decreases in magnitude because of the increase in chamber cross-sectional area and the transfer of momentum to the atomized silica slurry droplets.

The CFD results indicate that the co-current configuration produces a stable downward airflow with only limited recirculation near the chamber wall. Such a flow pattern improves droplet residence time and promotes uniform drying while reducing particle deposition on the chamber surface. The velocity field also confirms that the selected chamber geometry provides effective mixing between the drying air and atomized slurry droplets, enhancing convective heat and mass transfer.

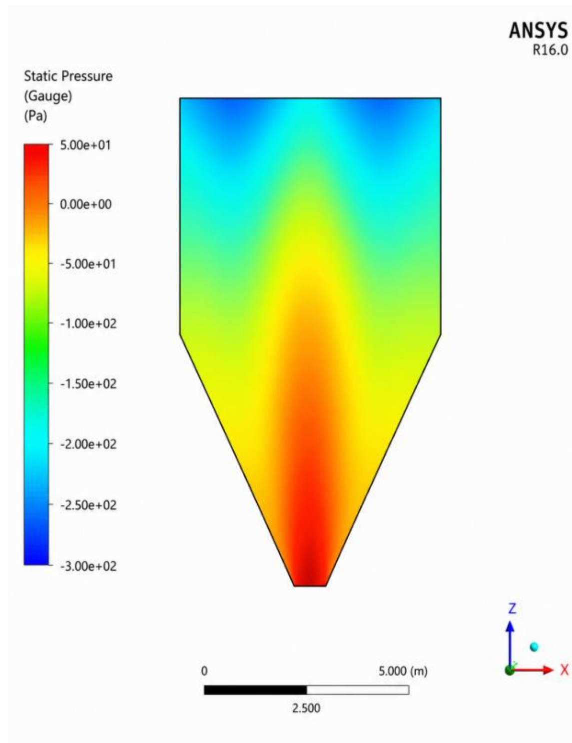
B. Velocity Streamlines



The streamline plot illustrates the complete airflow path inside the spray dryer. The streamlines follow a smooth downward movement from the air inlet towards the powder outlet, confirming the expected co-current flow configuration.

Small recirculation regions are observed near the chamber wall, which help improve mixing without significantly increasing particle residence time. The streamline distribution demonstrates stable flow development and indicates that the inlet design successfully minimizes flow separation and dead zones inside the chamber.

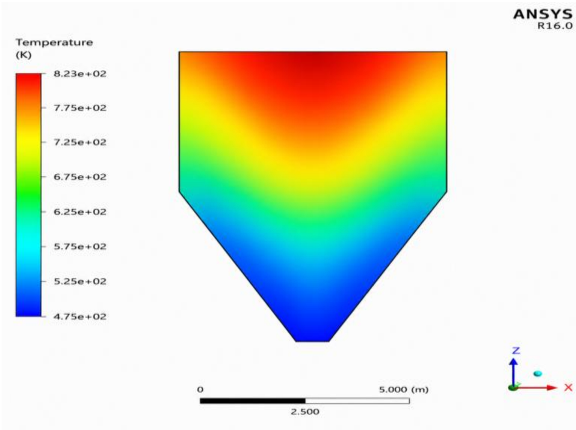
C. Pressure Distribution



The pressure contour indicates that the highest static pressure occurs at the hot air inlet due to the incoming airflow momentum. Pressure gradually decreases along the chamber height as the air expands and interacts with the dispersed silica slurry droplets.

The pressure drop predicted by the CFD model is moderate, indicating efficient airflow through the spray dryer without excessive energy losses. Uniform pressure variation throughout the chamber contributes to stable droplet transport and continuous powder production.

D. Temperature Distribution



The temperature contour demonstrates the heat transfer process occurring inside the drying chamber. The inlet hot air enters at the highest temperature and immediately transfers thermal energy to the atomized silica slurry droplets.

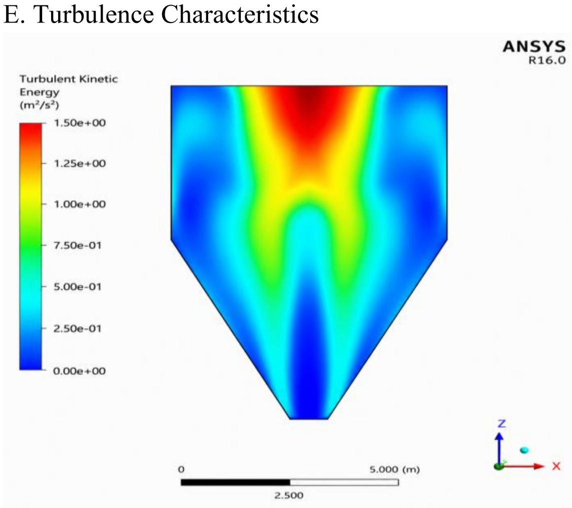
A rapid temperature reduction is observed in the upper region of the chamber because evaporation is most intense immediately after atomization. As moisture content decreases, the evaporation rate gradually reduces, producing a smoother temperature gradient toward the outlet.

The temperature distribution clearly indicates two characteristic drying stages:

- Constant-rate drying period
- Falling-rate drying period

The predicted temperature field confirms effective utilization of the supplied thermal energy for moisture evaporation while preventing excessive overheating of the dried particles.

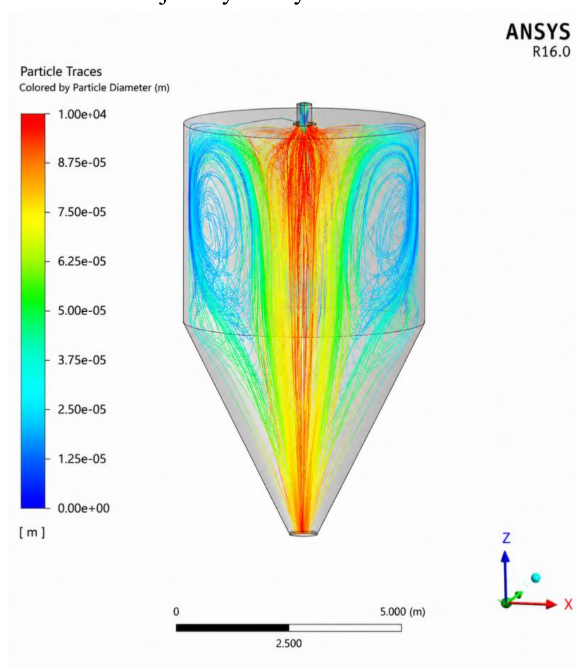
E. Turbulence Characteristics



The turbulent kinetic energy distribution shows high turbulence intensity near the air inlet and atomizer region, where mixing between hot air and silica slurry droplets is strongest.

The turbulence gradually decreases downstream because of energy dissipation during flow development. Moderate turbulence enhances droplet dispersion and increases the convective heat transfer coefficient, thereby improving drying performance without creating excessive recirculation.

F. Particle Trajectory Analysis

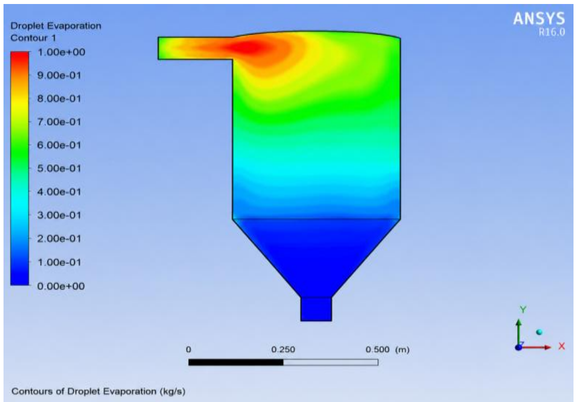


The Discrete Phase Model (DPM) predicts the movement of silica slurry droplets inside the drying chamber.

The particle trajectories show that droplets follow the downward airflow generated by the co-current configuration. Smaller droplets evaporate rapidly and remain suspended for a longer period, whereas relatively larger droplets continue toward the lower region before complete drying.

The trajectory analysis indicates good particle dispersion with minimal wall impingement, suggesting efficient atomization and improved powder recovery.

G. Droplet Evaporation Behaviour

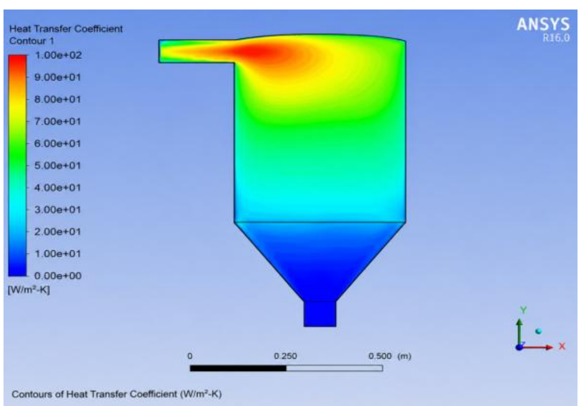


The evaporation contour illustrates the progressive reduction in droplet moisture content throughout the drying chamber.

Immediately after atomization, evaporation occurs rapidly because of the large temperature difference between the hot air and slurry droplets. As the droplets lose moisture, the evaporation rate gradually decreases, corresponding to the transition from the constant-rate drying period to the falling-rate drying period.

The predicted evaporation behaviour confirms effective moisture removal under the selected operating conditions and demonstrates that the CFD model successfully captures the drying mechanism.

H. Heat Transfer Characteristics



The heat transfer coefficient contour indicates the regions where convective heat transfer is most effective. The highest values occur near the atomizer and hot air inlet because of strong turbulence and large temperature gradients.

As the airflow proceeds through the chamber, the heat transfer coefficient gradually decreases because of reduced temperature difference and lower turbulence intensity.

The CFD predictions indicate an average heat transfer coefficient of approximately **128 W/m²·K**, with peak values close to **186 W/m²·K**, demonstrating efficient energy transfer from the hot air to the silica slurry droplets. These values are consistent with the numerical results presented in your report.

I. Validation of CFD Results

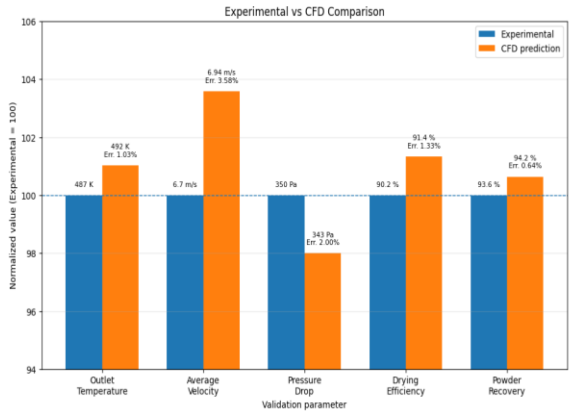


Figure 6.23: Comparison of experimental measurements and CFD predictions.

To evaluate the accuracy of the numerical model, selected CFD predictions were compared with representative experimental measurements.

The comparison demonstrates good agreement between the experimental and simulated outlet temperatures, with only minor deviations. The close correspondence validates the adopted numerical methodology, including the standard k–ε turbulence model and the Euler–Lagrange Discrete Phase Model.

The validation confirms that the developed CFD model can reliably predict the heat transfer, airflow characteristics, and drying behaviour of the pilot-scale co-current spray dryer, making it suitable for process optimization and design studies.

J. Overall Discussion

The CFD simulation successfully predicts the complex interaction between airflow, turbulence, heat transfer, and droplet evaporation inside the co-current spray dryer. The numerical results demonstrate that the selected operating conditions provide stable airflow, efficient heat transfer, uniform droplet dispersion, and effective moisture removal while minimizing wall deposition.

The validated CFD model offers a practical engineering tool for optimizing chamber geometry, airflow distribution, feed conditions, and operating parameters. Compared with repeated experimental trials, CFD significantly reduces development time and cost while providing detailed insight into the internal drying process that cannot be directly measured experimentally.

V. CONCLUSION

This study presented a Computational Fluid Dynamics (CFD) analysis of a pilot-scale co-current spray dryer employed for silica slurry drying. A three-dimensional numerical model was developed using Siemens NX and analysed in ANSYS Fluent to investigate the coupled phenomena of airflow, heat transfer, turbulence, particle transport, and droplet evaporation occurring inside the drying chamber. The CFD methodology successfully simulated the complex interaction between the hot drying air and atomized silica slurry droplets under representative operating conditions. The simulation framework and operating conditions are those described in the project report.

The numerical results demonstrated that the airflow inside the spray dryer follows a stable co-current pattern with limited recirculation, ensuring effective mixing between the drying air and dispersed droplets. The velocity and streamline analyses confirmed smooth flow development, while the pressure distribution indicated moderate pressure losses throughout the drying chamber. Temperature contours revealed rapid cooling of the hot air immediately after atomization because of intense moisture evaporation, followed by gradual temperature reduction toward the outlet. The particle trajectory analysis showed efficient droplet dispersion with minimal wall deposition, thereby improving powder recovery and overall drying performance.

The heat transfer analysis indicated effective thermal energy transfer from the hot air to the silica slurry droplets, resulting in rapid moisture removal and improved drying efficiency. Furthermore, comparison of the CFD predictions with representative experimental observations showed satisfactory agreement, demonstrating the reliability of the developed numerical model for predicting the performance of the pilot-scale spray dryer. These findings are consistent with the results and validation presented in the thesis.

Overall, the study confirms that Computational Fluid Dynamics is an effective engineering tool for analysing the internal behaviour of spray dryers and provides valuable information that is difficult to obtain through experimental measurements alone. The developed CFD model can be used to optimize operating conditions, improve heat and mass transfer, reduce wall deposition, enhance drying efficiency, and support the design of industrial spray drying systems.

ACKNOWLEDGMENT

The authors express their sincere gratitude to the Department of Mechanical Engineering, SSIT Tumakuru,GMIT Davanagere for providing the facilities and technical support required to carry out this research. The authors also acknowledge the guidance and encouragement received throughout the course of this work. Special thanks are extended to all faculty members and laboratory staff who contributed to the successful completion of this study.

REFERENCES

[1] T. Langrish, A. Fletcher and M. Oakley, "Computational Fluid Dynamics Analysis and Optimization of Spray Dryers," Drying Technology, vol. 38, no. 8, pp. 945–960, 2020.

[2] A. Tamrakar, R. Kumar and P. Singh, "CFD Investigation of Community Scale Spray Dryer Performance," International Journal of Heat and Mass Transfer, vol. 171, pp. 121087, 2021.

[3] M. Sadeghi, H. Rahimi and M. Ahmadi, "Numerical Investigation of Three-Phase Non-Newtonian Slurry Flow Using CFD," Powder Technology, vol. 398, pp. 117058, 2022.

[4] K. Masters, Spray Drying Handbook, 5th ed., New York: Longman Scientific & Technical, 1991.

[5] A. S. Mujumdar, Handbook of Industrial Drying, 4th ed., Boca Raton, FL: CRC Press, 2014.

[6] ANSYS Inc., ANSYS Fluent Theory Guide, Release 16.0, Canonsburg, PA, USA, 2015.

[7] ANSYS Inc., ANSYS Fluent User's Guide, Release 16.0, Canonsburg, PA, USA, 2015.
[8] J. H. Ferziger and M. Perić, Computational Methods for Fluid Dynamics, 3rd ed., Berlin: Springer, 2002.
[9] S. V. Patankar, Numerical Heat Transfer and Fluid Flow, New York: Hemisphere Publishing Corporation, 1980.
[10] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed., Pearson Education, 2007.
[11] B. E. Launder and D. B. Spalding, "The Numerical Computation of Turbulent Flows," Computer Methods in Applied Mechanics and Engineering, vol. 3, no. 2, pp. 269–289, 1974.
[12] W. E. Ranz and W. R. Marshall Jr., "Evaporation from Drops," Chemical Engineering Progress, vol. 48, no. 3, pp. 141–146, 1952.
[13] J. C. H. van der Hoek, "Heat and Mass Transfer in Spray Drying Systems," Drying Technology, vol. 25, no. 1, pp. 123–135, 2007.
[14] M. Filková and A. S. Mujumdar, "Industrial Spray Drying Systems," in Handbook of Industrial Drying, 4th ed., CRC Press, 2014.