Liquid Coolant Jet Breakup with Application to Grinding

Sheikh Ahmad Sakib[1] and Alex Povitsky[2]
University of Akron, Akron, OH, 44325, United States

Liquid jets in surrounding air face capillary and shear forces which eventually disintegrate the jet into droplets or spray. The instabilities developed in the flow inevitably break down an initial laminar (coherent) jet into a turbulent one. In the manufacturing process called grinding, one of the oldest approaches of shaping metals and other materials, liquid coolant jets are frequently used. A non-coherent or turbulent jet has a reduced flow rate due to cavitation, air entrapment and atomization of the fluid particles. The jet spread does not allow the coolant jet to effectively breach the high-speed rotating air layer, created by entrainment of air along the surface of rapidly rotating grinding wheel. The coherent, nearly columnal jet should be sufficiently long to maintain its initial velocity to penetrate the layer of air rotating with the grinding wheel. Thus, in many critical grinding applications, it is advised to use a coherent jet instead of a spray to eradicate defects of ground surface. In this study, we present simulations of liquid jet flows to see how the jet develops and breaks due to surface tension and shear forces. Creating an accurate model to predict liquid jet characteristics, especially for high-speed applications such as grinding wheel cooling would require well-resolving numerical grids and turbulence model selection. The problem being multi-phased with a density ratio of coolant-to-air being order of 1000 adds to the computational complexity. The presented numerical model and results are different compared to the previous simulations of liquid jets as the characteristics of jet disintegration are explored under conditions that closely resemble a grinding cooling application. Finite volume discretization of the flow domain and calculation of flow field characteristics were done by commercial software ANSYS Mesh and ANSYS Fluent modules, respectively. The numerical calculation and visualization of disintegration of free jet and the jet impinging into grinding wheel will be presented.

I. Nomenclature

A_i	=	Area of finite volume cell face
D	=	Diameter of jet
f_i	=	Body force
L_b	=	breakup length
P_n	=	Pressure at neighbor cells
u_i	=	velocity along different axis (i=index notation)
U	=	Average axial velocity of jet
V	=	Volume of the computational cell
We	=	Weber number
ρ	=	Density of liquid
ϑ	=	Kinematic Viscosity

II. Introduction

Grinding is a process mainly used in the manufacturing industry to prepare surfaces on various workpieces using an abrasive wheel. Grinding originated from primitive ages when contemporary humans rubbed stones together to shape them into various tools. Grinding machines, also known as grinders, play a pivotal role in achieving precise dimensions, surpassing the accuracy of other machining methods. The process extends beyond steel products to include ceramics, bronze, copper, and more [1,2]. Despite its advantages, grinding generates significant heat, and

[1] Master's Degree student, Department of Mechanical Engineering, AIAA Student Member (1603867).
[2] Professor, Department of Mechanical Engineering, AIAA Associate Fellow (186283).

efficient cooling becomes crucial to prevent adverse effects like oxidation and poor surface finish. An efficient cooling system can improve performance by dissipating heat more quickly and delivering coolant more precisely. The coolant also acts as a lubricating agent, and aids in removal of ‘chips’, ground metal powder from the workpiece. Grinding burn or heat related surface defects [3] such as brittle ‘white layer formation’ can be minimized largely with the help of an efficient cooling technique. Applying liquid coolant on the grinding wheel-workpiece interface as the wheel removes metal at a high speed has been a widely used technique in manufacturing. However, the challenge lies in making the coolant jet impinge precisely at the interface because of the high-speed rotation of the grinding wheel. Many researchers have worked on this problem and there have been remarkable developments. Webster et al. [4] showed that improvement in cooling efficiency can be achieved with a more coherent smooth streamed coolant jet. Lopez Arraiza et al. [5] designed nozzles and found optimum cooling conditions. Additionally, the orientation of the nozzle and coolant application was studied in [6,7].

When a liquid jet flows out of a nozzle, given that at the end of the nozzle exit it has kept its laminar property, will interact with surrounding air which due to surface tension and capillary effects, inevitably change the jet homogeneity. Eggers & Villermaux [8] discussed these effects in detail in their review of jet breakup phenomena. The term ‘laminar’ means a steady flow where fluid particles do not cross each other’s path and have zero vorticity. It is visible when the flow is not rough and has a smooth stream. Turbulent flow on the other hand, has layers of fluid particles mixing and crossing each other’s paths creating a visibly rough flow. When a laminar jet faces the capillary and shear forces acting on its boundaries, the laminar or smooth flow will cease to exist, and transition to turbulence takes place. At this point of the jet development, the jet is broken up into droplets of different shapes and sizes. During and after the transition, the flow carries less amount of fluid volume per cross sectional area compared to its upstream location where the flow remains laminar. Turbulence in the flow will make the jet look non-coherent or rough. Studies by Webster [9] found that circular ‘coherent jets’ perform better in dissipating the heat from grinding surfaces. According to Rupe [10], if the jet has a non-uniform velocity profile at the exit of the nozzle, even if it is in laminar regime, it will break down easily; even an initially turbulent jet will go longer distances than an initially laminar jet without breaking up. However, Debler and Yu [11] showed that jets with non-uniform velocity profiles are more stable than jets with uniform velocity profiles, where being stable means an increased jet length until breakup. The present work is a numerical approach of predicting jet behavior where we simulate the flow under conditions resembling grinding cooling.

Prediction of free jet behavior using numerical based tools such as ANSYS FLUENT, OPENFOAM or other commercial codes have been carried out previously by many authors [12-17]. Most of the prior research except Pan and Suga [14] did not accommodate multiphase coolant and surrounding air flows having a density ratio of nearly 1000 in their calculations. Compared to many transition to turbulence problems like a Blasius boundary layer separation at rigid wall, free multi-phase shear flows such as jets have not been modeled computationally to that extent. The major reason for this can be attributed to the large spectra of length and time scales associated with a jet flow which needs to be resolved by the finite volume grid, making the problem computationally expensive. Large-eddy simulation (LES) has been used as a turbulence model by Xiao et al [13] where the variant of LES called Detached Eddy Simulation (DES) has been introduced by Zhao [17]. Many of these authors [14,16,17] have used volume of fluid (VOF) approach coupled with the ‘level set’ function. Further discussions on VOF and level set method are to follow in later sections.

This study will attempt to shed light on the jet breakup lengths in different conditions such as jet speed, velocity profile, and presence of downstream walls. At the first stage of the study, liquid jet breakup spreading in surrounding still air has been considered. The commercial software ANSYS FLUENT has been used to carry out the numerical simulations of the flow of liquid jet. Then we aim at finding out what differences a liquid jet shows in its behavior during breakup when there is a solid surface downstream in which the jet is impinging. Although a simplistic emulation, this is quite pertinent to grinding operations because the coolant jet in a real-world grinding machine will have multiple obstacles standing downstream such as grinding wheel and workpiece. Used computational models were validated with experimental data from literature and compared jet development and breakup in several different set-ups. The jet breakup with a regular jet issued from a circular opening was compared with the popular Rouse nozzle design [18]. We have also investigated the effect of a uniform velocity profile as opposed to a parabolic velocity profile at the nozzle exit. As the research advances, we plan to focus on jet behavior with a coaxial flow of coolant to explore new designs of nozzles.

III. Numerical Methods

A. Finite Volume Solver

The finite volume method (FVM) is a method for solving partial differential equations (PDE) describing conservation laws including the Navier-Stokes equations. The FVM discretizes PDEs in the form of algebraic systems of equations [19]. The physical variables of the flow such as velocity and pressure are approximated at discrete nodes at centers of finite volumes within the problem domain. Volume integrals including divergence term are replaced by surface integrals according to the divergence theorem [20].

$$\frac{\partial \boldsymbol{u_i}}{\partial \boldsymbol{t}}\boldsymbol{V} + \sum_{n=1}^{N}(\boldsymbol{u_i u_j n_j A})_n + \sum_{n=1}^{N}(\boldsymbol{P_{n_i} A}) - \sum_{n=1}^{N}\left(\boldsymbol{\vartheta}\frac{\partial \boldsymbol{u_i}}{\partial \boldsymbol{x_j}}\boldsymbol{n_j A}\right) - \boldsymbol{f_i V} = \boldsymbol{0} \quad (1)$$

$$\sum_{i=1}^{N}\boldsymbol{u_i}\,\boldsymbol{A_{i,nb}} = \boldsymbol{0}, \quad (2)$$

where, u= velocity, i= index notation for different axis, P= Pressure at cell center, V= Volume of cell.

The choice of solver of the above system of equations is important because it defines the computational time and accuracy of solution. The SIMPLE and Coupled approaches [21,22] are two of the options that FLUENT offers. The Semi Implicit Method for Pressure Linked Equation (SIMPLE) involves iterative steps to determine velocity and pressure fields in computational fluid dynamics [21]. It adjusts the pressure field to ensure compliance with the continuity equation, introducing a pressure correction equation. . Considering the computational time and better convergence, the SIMPLE method was chosen as the primary solver for the calculations.

B. Turbulence Models:

Various Reynolds-Averaged Navier-Stokes (RANS) models have been developed as alternatives to Direct Numerical Simulations (DNS) to predict flow behavior efficiently without requiring extensive computational resources. Unlike DNS or Large Eddy Simulation (LES), RANS models focus on mean flow properties and use turbulence closures for unresolved turbulence effects. Common RANS models include Reynolds Stress Model (RSM), k-ε, and k-ω models, known for their computational efficiency and applicability in industrial aerodynamics context. However, RANS models have limitations in capturing highly complex and unsteady turbulent flows, which may be better addressed by LES or DNS methods. In our computations, RANS models including Transition Shear Stress Transport (SST) and k-ε have been used. However, as the study advanced in validating and observing flow visualizations for these different models, Detached Eddy Simulation (DES), with a 2nd order discretization scheme for volume fraction was used because it offered better accuracy. The k-ε turbulence model is a computational approach to address the Reynolds stresses that we find in a Reynolds Averaged Navier Stokes (RANS) equations using two variables (k and ε), where k denotes the turbulent kinetic energy and ε denotes its dissipation rate.

Transition SST model is an extension of the family of two-equations models of turbulence where it has two additional terms with two additional equations for γ and Re_θ. The additional terms help to define the onset of turbulence in the flow field. The term γ denotes the turbulent intermittency where its non-zero value suggests existence of turbulence in that part of the domain.

The DES model is a hybrid approach of both RANS and LES, where a sub-grid calculation model (LES) is activated in finer grids in the separated flow region where RANS calculations are carried out in near wall or boundary layer region [23, 24]. A 'limiter' is used to identify and differentiate the zones where RANS and LES will be computed. Although the present study has grids which are uniform throughout the whole domain, more complicated domain, i.e. with rotating grinding wheels where the freedom of having coarser grids in some areas will be beneficial. To improve the accuracy of DES calculations, several shielding functions have been added to the 'limiter' which enables the code to understand the onset of flow separation from its own solutions which essentially 'delays' the activation of LES to a more accurate point. One of such modified models is called the Delayed DES [24]. The DES saves significant computational time compared to LES while offering better accuracy than typical RANS approach. Details of turbulence models can be found on Ansys FLUENT theory guide [25].

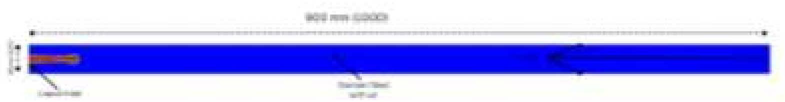
C. Volume of Fluid (VOF) & Level Set Method

Volume of Fluid [26] is a method to solve multiphase flow problems where a continuity equation for volume fractions of participating fluids is solved throughout the whole domain. A primary phase is selected (in this study, it is air) along with the secondary phase (water, as liquid coolant). The fraction of secondary phase is set as initial and boundary conditions. This is a very useful method to predict free shear flows accurately.

The level set method, developed by Osher and Sethian [27] is a front tracking technique useful for multiphase flow simulation when the interphase between phases is not constant and its location is not predictable before simulations. The RANS equations are coupled with a scalar quantity φ that tracks the volume fraction of the secondary fluid in the flow domain [28]. For example, if we are considering water flowing or issuing from a nozzle in an ambient air environment, then the liquid water volume fraction will be tracked throughout the whole computational domain enabling us to visualize the phase interface at every cell. In general, the value of φ is set in such a way that it yields a negative value inside the secondary phase, and a positive value outside of the secondary phase encapsulated cells. Therefore, it would yield a unit value at the interface of the two fluids.

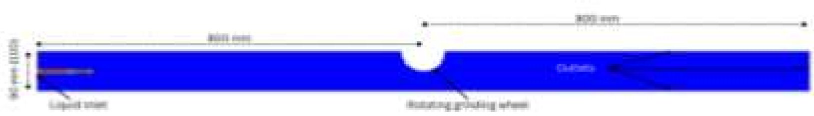
D. Problem Setup

This section will outline the FVM grid generation, geometric domain, and setup of boundary conditions.
Domain and Numerical Grids: Multiple domains were set up, to visualize the liquid jet flow in different conditions. Validation of turbulence model and primary jet breakup length analysis were carried out in an axisymmetric domain of 4D×100D (where D= 9mm is the nozzle exit diameter), see **Figure 1a**. To accommodate the downstream wall impingement of the liquid jet flow, the domain size was changed to 10D×177.78D. (**Figure 1b**). Similarly, a rotating wheel of diameter 10D was set up at 0.8m (nearly 9D) downstream of the nozzle exit to observe the change in jet breakup due to interaction with grinding wheel's surrounding air layer. (**Figure 1c**)



(a)

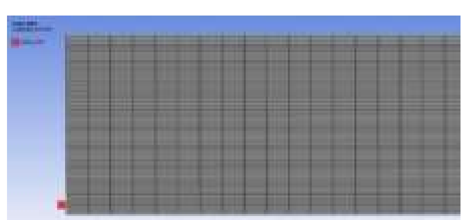
(b)



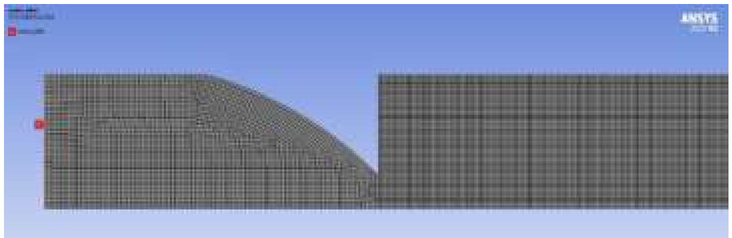
(c)

Figure 1: (a), (b), (c) represents the domains used in the calculations.

The grids were generated using ANSYS Mesh module. To start with, a rectangular grid size was selected having 10 grid layers across the liquid jet inlet (at nozzle exit) to capture the appropriate breakup regimes. We refined the grids to be square shaped in later stages of our numerical modeling.



(a)



(b)

Figure 2: (a) Regular Mesh (Rectangular, 20 grids across liquid inlet) and (b) Refined mesh for Rouse nozzle (Square, 24 grids across nozzle exit)

IV. Results and Discussions

A. Validation of model

The study by Sallam et al. [29] has been adopted to validate our model of predicting jet breakup. The authors ran experiments on circular liquid jets having different Weber numbers based on jet velocity density of jet material and surface tension. They compiled prior results by Chen & Davis [30] and Grant & Middleman [31] where experiments were carried out to define the liquid jet breakup length, developed the following formula to predict the breakup length for liquid jet flow which has a Weber number less than 400:

$$L_b = 5We^{0.5} \quad (3)$$

where, L_b = Breakup length, Weber number, $We = \frac{\rho DU}{\sigma}$, U= Average axial velocity of jet, D = Diameter of jet, ρ = Density of liquid. Our numerical simulations were run with different turbulence models including transition SST, k-ε, and DES with different grids and discretization approaches. The results were compared with both the analytical prediction and experimental results from Pan & Suga [14].

In this research, the jet breakup length is defined as the axial distance from nozzle exit where the density on the jet axis becomes less than 500 kg/m^3, that is ~50% of initial jet density. The density of liquid water at the inlet is ~1000 kg/m^3. The density of water and air mixture along the axis of the jet was plotted in different moments in time and the average breakup length was determined, as the jet breakup occurs in different locations in different time moments. Break-up lengths were observed with the Transition SST (4 equation), k-epsilon (2 equations) and DES (Delayed) models of turbulence where the DES model with square shaped refined grids (similar to **Figure 2b**) gave the highest accuracy compared to literature (**Figure 3**). In further visual investigation of the flow fields, comparing experimental photos from [29] and computational flow field from [14], we observed that the visual shapes of jet breakup resembled the most when the Delayed DES model was used.

B. Circular jet breakup:

Jet issued from a direct circular opening (not a nozzle with curved inside walls) having a range of average axial velocities (0.447m/s to 1.788 m/s) were simulated and the flow-fields (colored by density), were obtained numerically. **Figure 4** shows the breakup of a jet having an average axial velocity 1.095 m/s (We= 150). The surface waves tend to amplify to the point where the wave amplitude is almost equal to the diameter. The surface waves are 'dilatational'[14] in nature and these waves are the primary factor for the jet breakup according to Rayleigh [32] and Pan [14]. These types of instability and breakup observed here have been previously expressed as 'Rayleigh' type and wind induced breakup by authors [8,14]. The jet's movement direction has been set toward the direction of gravity. In the present section, computations for circular jets, all of them with We ≤400, show surface waves and jet breakup in similar manner.

D. Circular vs rectangular jets:

If the domain and boundary conditions are set in such a way that the jet has a rectangular cross section compared to the circular one discussed above, a different breakup phenomenon is observed which is asymmetric unlike the previous circular jet and much more 'violent' in nature (**Figure 5**).

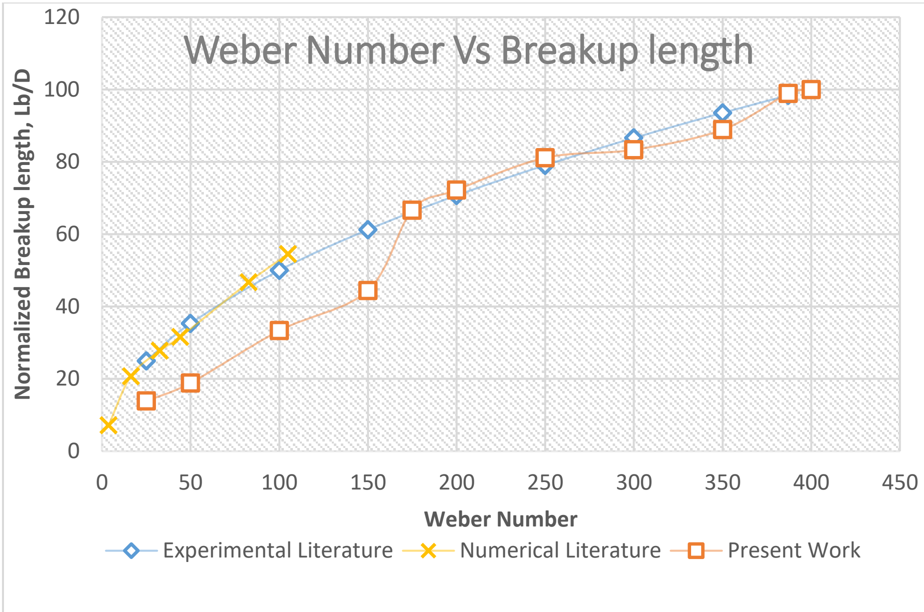


Figure 3: Validation of numerical model, where delayed DES is used.

a)
b)
c)

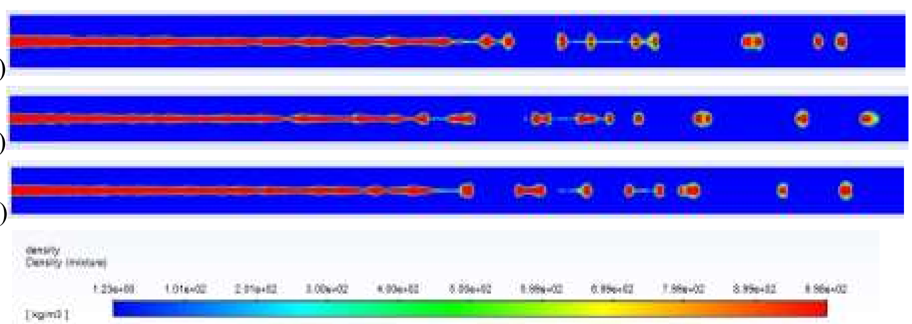
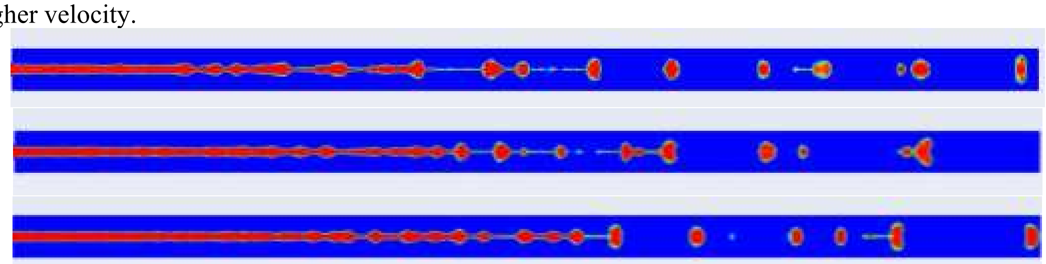
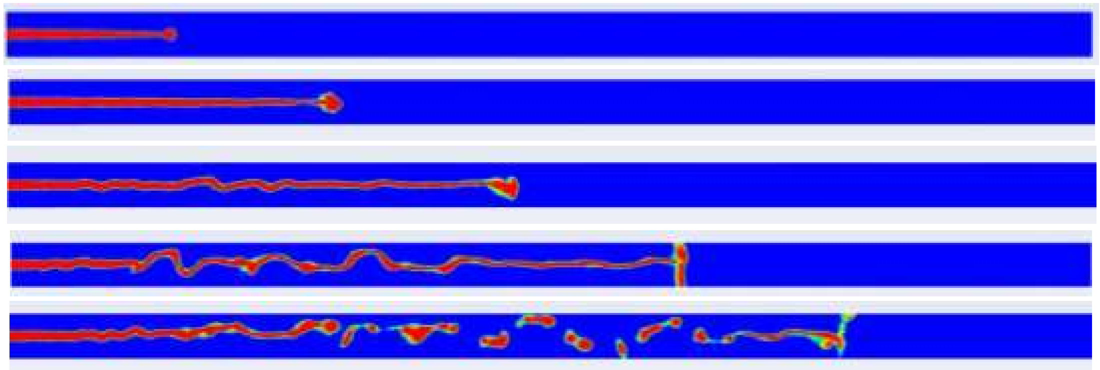


Figure 4: Jet breakup for We=150 (average velocity 1.095 m/s). Snapshots at sub-figures a), b) and c) at t=1,2, and 3 seconds.

Another distinction between the two in **Figure 5(a & b)** is their behavior in time. The circular jets have a proclivity to maintain their maximum reached distance from the nozzle exit while the rectangular jet reduces in its length before breakup by almost 50% and does not reach the length and shape of its first break up again. Therefore, in terms of stability, circular jets are superior to rectangular jets. Pan and Suga [14] discussed multiple breakup regimes where higher velocity jets for We>400, show a second wind induced breakup which is asymmetric and as opposed to the flow fields shown in Figure 4, they are more likely to have shorter diameter of jet at the point of breakup. Therefore, it can be assumed that in the current calculations, the rectangular jets show breakup properties like circular jets having much higher velocity.



a)



b)

Figure 5: Breakup of a) circular jets vs b) rectangular jets in time (average axial speed= 1.788 m/s)

The fact that circular or round shaped jets have higher stability than non-circular jets have been discussed before by Mi et al. [33]. The above results are consistent with their finding.

E. Jet breakup with uniform and non-uniform velocity profiles at nozzle exit

Many authors [9-11] addressed the needed velocity profile suitable for more stable jet development. As discussed earlier, there are contradictory findings at that front. In present calculations, uniform and parabolic velocity profiles were set as boundary conditions and difference in breakup lengths were observed.

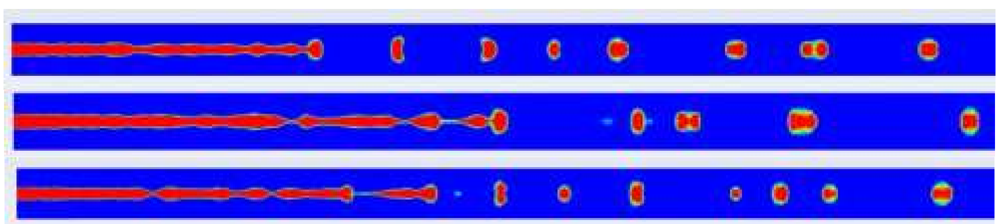


Figure 6: Circular jet breakup with a uniform velocity profile at nozzle exit.

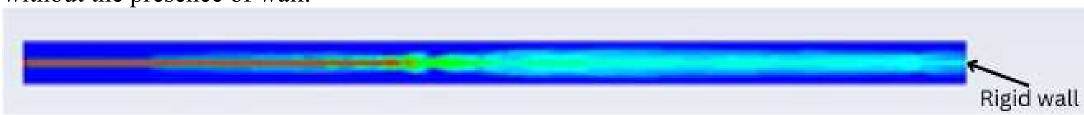
Simulations for circular jets in **Figure 4** were carried out with a parabolic velocity profile at the nozzle exit, that is, the liquid jet inlet of the domain. A fully developed laminar pipe flow yields parabolic type velocity profile which is more resembling of the real-life operations because the liquid coolant is transferred to the nozzles with pipes having significant length to diameter ratio. **Figure 4** pictures are comparable to **Figure 6**, where a 'uniform' velocity profile was set at the nozzle exit. With the same average axial velocity (U=1.095 m/s, We=150) in both cases, reduction in breakup length was observed while using 'uniform' velocity profile. This result is compatible with Debler and Yu's [11] experimental results where they concluded that the non-uniformity of velocity profile at nozzle exit increases the stability of the circular jet.

Table 1: Average breakup length variation with different velocity profiles at nozzle exit (U=1.095 m/s, We=150)

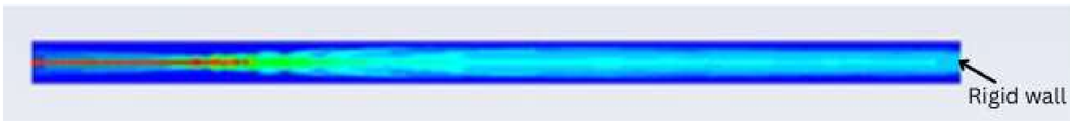
	Non-uniform (parabolic) profile	**Uniform profile**
Average Breakup Length, L_b (m)	0.40	0.30
Non dimensional average breakup Length, $\frac{L_b}{D}$	45	33.33

F. Jet break-up with a rigid surface located downstream.

According to computations using the DES model in **Figure 3**, the average breakup length for a circular jet having We=400 is 0.9m. Boundary conditions were set to observe if a wall, located approximately at the same downstream distance as the breakup location from the nozzle exit, can affect the breakup in any significant way. It was observed that (**Figure 7a and 7b**) the location of the breakup is pushed back by almost 1/3 of its length compared to case of free jet without the presence of wall.

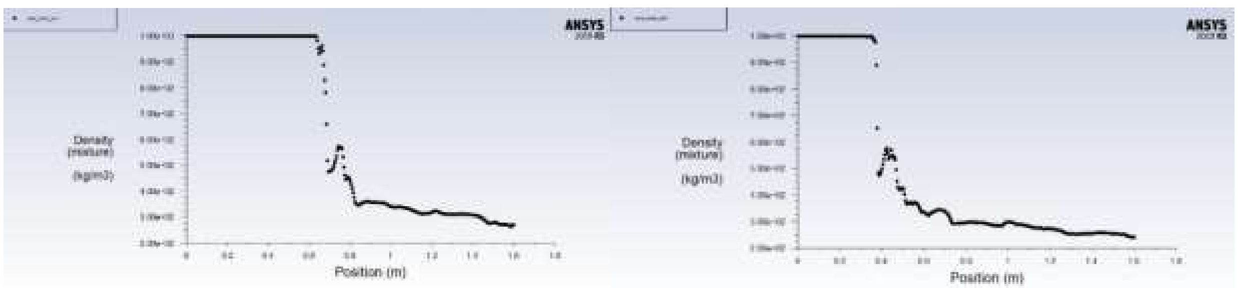


a)



b)

Figure 7: a) Jet flow with the presence of rigid surface a) at t=6s and b) at t=14s



a) b)

Figure 8: Density plot along the axis of jet at a) t=6s, and b) t=14s

G. Jet break-up with a grinding wheel located downstream.

The process of disintegration of jets depends primarily on the interaction of the liquid and the surrounding air. In the discussion above, the effects of jet break-up in still air were observed. In the case of surrounding air which is not still rather moving along with a rotating grinding wheel can be visualized using the same turbulence models, grids, and numerical schemes with some adjustments in the domain. The domain was set up according to Figure 1(c) to accommodate the grinding wheel. Due to limitations of 2D domain, the problem was set up so that the liquid jet has a rectangular cross section. **Figure 9** shows the disintegration of a jet having an average axial velocity of 1.788 m/s (same as **Figure 5a** and **5b**). The wheel velocity is 445 rad/s counterclockwise.

Compared to the rectangular jet without the presence of the rotating wheel, it was observed that the maximum unbroken length of the jet was increased in the presence of the wheel. It can be attributed to the wheel's counterclockwise rotation which drives the air away from the incoming jet and delays the breakup.

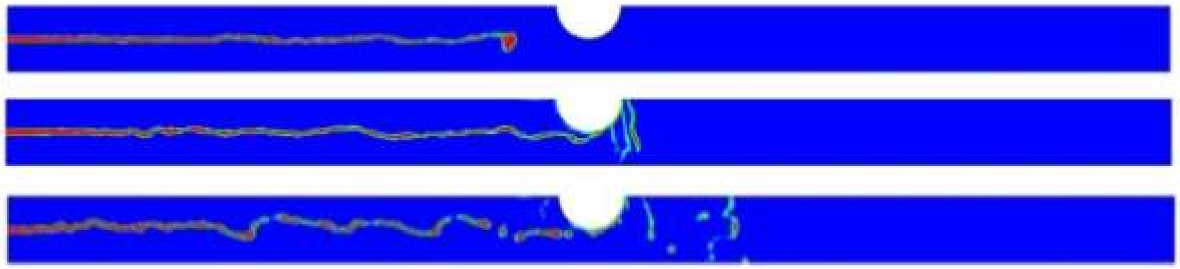
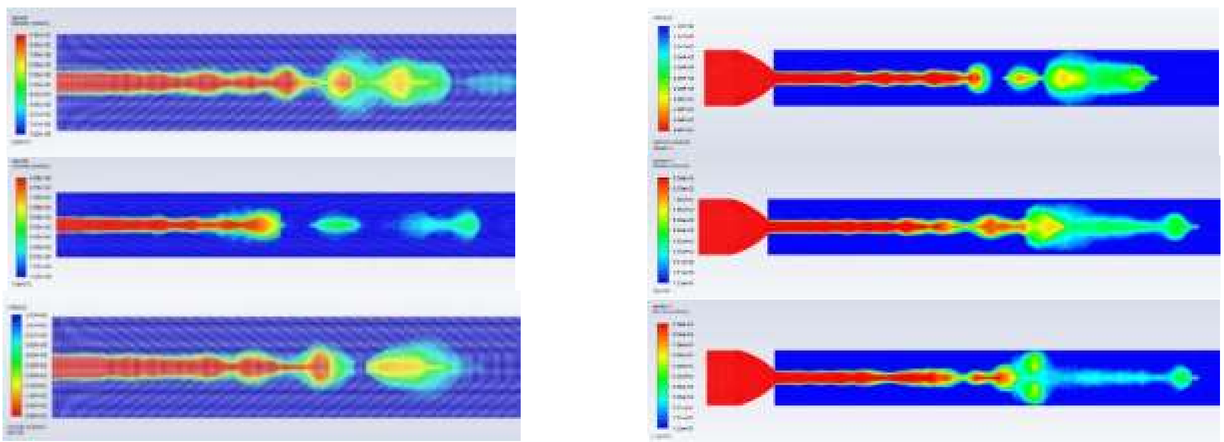


Figure 9: Jet breakup with the presence of rotating grinding wheel

H. Circular jet breakup from 'Rouse' nozzle

Renowned design from Rouse [18] is a popular nozzle which was used by Webster et al. [4] to analyze and develop better nozzles for grinding industry in coolant delivery operations. Using the same domain that we used for validation and analyzing jet breakup length (**Figure 1a**), the development of jet from a Rouse nozzle with same velocity was compared with the jet breakup from simple orifice described in prior sections.



(a) (b)

Figure 10: Jet breakup for We=175 (average exit velocity 1.182 m/s), frames are 1s apart from each other. a) Regular circular opening and b) Rouse [18] nozzle

The snapshots of the jet flow field (**Figure 10**) for the same axial velocity look similar and indicate no discernible change of flow pattern. However, upon analysis of the average breakup length, it was found that for higher average axial speed (1.778 m/s, We=400), the 'Rouse' nozzle produces jet that has lower average breakup length compared to one found in **Figure 3**. Table 2 shows summary of jet properties calculated for different nozzles with same average axial velocity. For lower speeds, i.e. We=175, as presented in **Figure 10,** the length of breakup is approximately the same for both cases.

Table 2: Jet properties for different nozzles with We=400 (U= 1.778m/s)

	Regular circular opening	Rouse nozzle	Rectangular 2-D opening	Rectangular opening with grinding wheel downstream
Average Breakup Length, L_b (m)	0.9	0.6	0.55 (approx.)	0.7
Non dimensional average breakup Length, $\frac{L_b}{D}$	100	66.67	61.12	77.78

V. Conclusion

The numerical results and flow visualizations using ANSYS Fluent accurately predicted the jet breakup, especially for lower speed applications where the instability criteria are dictated by Rayleigh instability laws. The liquid jet impingement on a rotating grinding wheel and the early breakup of jet with a downstream stationary rigid surface was modeled. Distinctions between the breakup nature of round and rectangular jets have been captured by the validated turbulence model. The key takeaway from this study includes the accurate prediction of circular jet breakup lengths as functions of the Weber number and axial velocity. The effects of a non-uniform velocity profile to increase the stability of a circular jet has been investigated by the developed numerical model and findings were confirmed by published experiments.

The numerical analysis of jets, both round and rectangular, suggested that under conditions where surrounding airflow is non-uniform, induced either by the motion of a grinding wheel or by a downstream impediment, the average length of breakup is different (shorter with rigid wall and longer with grinding wheel) compared to that derived in regular conditions of stationary air. The current progress in the computational modeling of jet development and its accurate prediction can be a stepstone to open new opportunities of research where we can quantitively analyze a liquid coolant jet performance impinging on a grinding wheel.

Acknowledgments

The authors are grateful to NSF 2333138 MRI: Track 1: 'Acquisition of Spray Particle & Spray Droplet Size Analyzer for Precision Manufacturing', and NSF 2230411 I-Corps Teams: 'Delivery of coherent coolant jet and nozzle manufacturing for grinding technology' for their partial support of the current research. The authors appreciate the Spark Fund by the State of Ohio and the University of Akron Research Foundation for their award titled 'Coherent coolant jet for grinding technology' for their support of current research. The first author acknowledges support by Teaching Assistantship by the graduate school at the University of Akron.

References

[1] A. C. Shoemaker, M. I. Davies, and H. L. Moore, "Back to the Grindstone? The Archaeological Potential of Grinding-Stone Studies in Africa with Reference to Contemporary Grinding Practices in Marakwet, Northwest Kenya," African Archaeological Review, vol. 34, no. 3, pp. 415–435, Sep. 2017. URL: http://link.springer.com/10.1007/s10437-017-9264-0

[2] J. C. Aurich, C. Effgen, and B. Kirsch, "Grinding," in CIRP Encyclopedia of Production Engineering, S. Chatti, L. Laperri`ere, G. Reinhart, and T. Tolio, Eds. Berlin, Heidelberg: Springer Berlin Heidelberg, 2019, pp. 795–799. URL: https://doi.org/10.1007/978-3-662-53120-4 6427

[3] J. Badger and A. Torrance, "Burn Awareness," Cutting Tool Engineering, vol. 52, no. 12, pp. 16–19, Dec. 2000. URL: https://www.ctemag. com/sites/www.ctemag.com/files/archive pdf/0012-burnawareness.pdf

[4] J. Webster, C. Cui, R. Mindek, and R. Lindsay, "Grinding Fluid Application System Design," CIRP Annals, vol. 44, no. 1, pp. 333–338, 1995. URL: http://linkinghub.elsevier.com/retrieve/pii/S0007850607623373

[5] A. Lopez-Arraiza, G. Castillo, H. N. Dhakal, and R. Alberdi, "High performance composite nozzle for the improvement of cooling in grinding machine tools," Composites Part B: Engineering, vol. 54, pp. 313–318, Nov. 2013. URL: http://linkinghub.elsevier.com/retrieve/pii/S1359836813002837

[6] S. Ebbrell, N. Woolley, Y. Tridimas, D. Allanson, and W. Rowe, "The effects of cutting fluid application methods on the grinding process," International Journal of Machine Tools and Manufacture, vol. 40, no. 2, pp. 209–223, Jan. 2000. URL: http://linkinghub.elsevier.com/retrieve/pii/ S089069559900060

[7] G. Trmal and H. Kaliszer, "Delivery of Cutting Fluid in Grinding," Chartered Mechanical Engineering, vol. 23, no. 8, pp. 95–100, Sep. 1976.

[8] Jens Eggers and Emmanuel Villermaux, "Physics of liquid jets", Reports on Progress in Physics, 71 (2008) URL: t stacks.iop.org/RoPP/71/036601

[9] M.E. Goldstein and S.J Leib, "The role of instability waves in predicting jet noise", URL: The role of instability waves in predicting jet noise | Journal of Fluid Mechanics | Cambridge Core

10] Jack H. Rupe, "On the Dynamic Characteristics of Free-Liquid Jets and a Partial Correlation with Orifice Geometry". Jet Propulsion Laboratory, California Institute of Technology. URL: https://ntrs.nasa.gov/api/citations/19620000813/downloads/19620000813.pdf

[11] W. Debler and D. Yu, "The break-up of laminar liquid jets". Published:08 January 1988
URL: https://doi.org/10.1098/rspa.1988.0005

[12] Junkai Huang, Xin Zhao, "Numerical simulations of atomization and evaporation in liquid jet flows". URL: Numerical simulations of atomization and evaporation in liquid jet flows - ScienceDirect

[13] F. Xiao, M. Dianat, J.J. McGuirk, "LES of turbulent liquid jet primary breakup in turbulent coaxial air flow". URL: LES of turbulent liquid jet primary breakup in turbulent coaxial air flow - ScienceDirect

[14] Yu Pan, Kazuhiko Suga, "A numerical study on the breakup process of laminar liquid jets into a gas". Physics of Fluids, Volume 18, Issue 5. URL: A numerical study on the breakup process of laminar liquid jets into a gas | Physics of Fluids | AIP Publishing

[15] Junkai Huang, Xin Zhao, "Numerical simulations of atomization and evaporation in liquid jet flows". URL: Numerical simulations of atomization and evaporation in liquid jet flows - ScienceDirect

[16] F. Xiao, M. Dianat, J.J. McGuirk, "LES of turbulent liquid jet primary breakup in turbulent coaxial air flow". URL: LES of turbulent liquid jet primary breakup in turbulent coaxial air flow - ScienceDirect

[17] Jiafeng Zhao, Chenglong Yan, Liyin Wu, Wei Lin, Yiheng Tong, Wansheng Nie, "Numerical simulation of single/double liquid jets in supersonic crossflows", Aerospace Science and Technology, Volume 120, 2022,107289, ISSN 1270-9638, DOI: https://doi.org/10.1016/j.ast.2021.107289. URL: https://www.sciencedirect.com/science/article/pii/S1270963821007999

[18] H. Rouse, "Experimental investigation of fire monitors and nozzles," vol. 77. ASCE, 1951, pp. 1–29.

[19] Marcus Herrmann, Marco Arienti, Marios Soteriou, "The Impact of Density Ratio on the Liquid Core Dynamics of a Turbulent Liquid Jet Injected into a Crossflow", 'Journal of Engineering for Gas Turbines and Power', Volume 133, Issue 6. URL: The Impact of Density Ratio on the Liquid Core Dynamics of a Turbulent Liquid Jet Injected Into a Crossflow | J. Eng. Gas Turbines Power | ASME Digital Collection

[20] Weisstein, Eric W. "Divergence Theorem." From *MathWorld*--A Wolfram Web Resource. URL: https://mathworld.wolfram.com/DivergenceTheorem.html

[21] S. V. Patankar and d. B. Spalding, "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows", Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion, Pergamon, 1983, Pages 54-73, ISBN 9780080309378, https://doi.org/10.1016/B978-0-08-030937-8.50013-1. URL: https://www.sciencedirect.com/science/article/pii/B9780080309378500131

[22] Z.J. Chen, A.J. Przekwas, "A coupled pressure-based computational method for incompressible/compressible flows". URL: A coupled pressure-based computational method for incompressible/compressible flows - ScienceDirect

[23] Travin, A., Shur, M., Strelets, M. *et al.* "Detached-Eddy Simulations Past a Circular Cylinder". *Flow, Turbulence and Combustion* **63**, 293–313 (2000). https://doi.org/10.1023/A:1009901401183

[24] Philipe R. Spalart, "Detached-Eddy Simulation", **Annual Review of Fluid Mechanics**, Vol. 41:181-202 (Volume publication date January 2009). URL: https://doi.org/10.1146/annurev.fluid.010908.165130

[25] "Turbulence Models", ANSYS FLUENT Theory Guide, 14.5.18. Turbulence Models (ansys.com)

[26] "VOF and Coupled Level-Set", ANSYS FLUENT Theory Guide, 14.3.12. Coupled Level-Set and VOF Model (ansys.com)

[27] Stanley Osher, James A Sethian, "Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations", Journal of Computational Physics, Volume 79, Issue 1, 1988, Pages 12-49, ISSN 0021-9991, https://doi.org/10.1016/0021-9991(88)90002-2. URL: https://www.sciencedirect.com/science/article/pii/0021999188900022)

[28] Sussman M, Almgren A S, Bell J B, Colella P, Howell L H and Welcome M L 1999 An adaptive level set approach for incompressible two-phase flows. J. Computational. Phys. 148: 81–124

[29] K.A. Sallam, Z. Dai 1 , G.M. Faeth, "Liquid breakup at the surface of turbulent round liquid jets in still gases", International Journal of Multiphase Flow, Volume 28, Issue 3, 2002, Pages 427-449, ISSN 0301-9322, https://doi.org/10.1016/S0301-9322(01)00067-2. URL: https://www.sciencedirect.com/science/article/pii/S0301932201000672

[30] Chen, T.-F., Davis, J.R., 1964. "Disintegration of a turbulent water jet". J. Hyd. Div. 1, 175–206.

[31] Grant, R.P., Middleman, S., 1966. "Newtonian jet stability". AIChE J. 12, 669–678.

[32] Rayleigh, L. (1878), "On the Instability of Jets. Proceedings of the London Mathematical Society", s1-10: 4-13. https://doi.org/10.1112/plms/s1-10.1.4

[33] Mi, J., Nathan, G. & Luxton, R. "Centreline mixing characteristics of jets from nine differently shaped nozzles", *Experiments in Fluids* **28**, 93–94 (2000). https://doi.org/10.1007/s003480050012