ISSN 2278-3091
Volume 10, No.2, March - April 2021
International Journal of Advanced Trends in Computer Science and Engineering
Available Online at http://www.warse.org/IJATCSE/static/pdf/file/ijatcse811022021.pdf
https://doi.org/10.30534/ijatcse/2021/811022021

Finite Element Analysis of Mixing Flow in a Circular Vessel with Concentric Three Blade Agitator

Mir Afzal Ahmed Talpur[1], Ahsanullah Baloch[2], Hameedullah Kazi[3], Farhan Ahmed Shaikh[4]
[1]Asst. Prof., Department of Computer Science, ISRA University Hyderabad – Pakistan
Email: meermessi@gmail.com
[2]Prof. Dr., Department of Computer Science, ISRA University Hyderabad –Pakistan
Email: ahsanullah.baloch@isra.edu.pk
[3]Prof. Dr., Department of Computer Science, ISRA University Hyderabad –Pakistan
h.kazi@isra.edu.pk
[4]Lecturer, Department of Mathematics, Federal Government Degree College Hyderabad –Pakistan
Email: farhankhalil.shaikh@gmail.com

ABSTRACT

This study is to investigate the optimisation of the mixing process in two folds, such as, homogenisation of material and to predict the power consumption. This research work is extension of other studies conducted by different researchers. In all other research investigations, no one had employed agitator. The geometry features a cylindrical vessel fixed with a mechanically revolving stirrer along with fixed agitator. The flow is modelled for incompressible constant viscosity Newtonian fluid with isothermal condition. Simulated numerical predictions are achieved through so called a finite element algorithm. The governing equations considered here are two–dimensional equation continuity and time–dependent Navier–Stokes equation in cylindrical polar coordinates. Employed numerical scheme is constructed in multiple–stages. Where, time derivative is discretised in two step quadratic approximation of Taylor series expansion. Whereas, Galerkin approximation is employed for spatial discretisation. Whilst, at second step pressure correction is adopted through projection method. Whilst, implicitness is applied on only diffusion term to make algorithm in semi–implicit form of TGPC. The influences of inertia will be analysed through fluid inertia using dimensionless Reynolds number. The effects of rotational speed of stirrer with agitator will be explored. The computed results will be illustrated for pressure by isobars and flow structure through streamline contours plots. The keyaim of the numerical study is to estimate the improved possible design of the blenders, that enhance the mixing process.

Key words: Numerical Modelling, Finite Element Scheme, Rotating Mixing Flow, Three Blade Agitator.

1. INTRODUCTION

The pivotal point regarding this research work is the design of mixer, which contain concentric stirrer with agitator. The purpose of this study is to optimise the mixing process for homogenisation of material and to predict the power consumption. The importance of this problem relates to the mixing devices, such as mixers of fluids, which are used in food industries, paint industry, pharmaceutical industry, and other related fluid processing industries [1].

Mixing of substances is a very old need of humans, which is being exercised for thousands of years. By the advancement of time, new ideas and latest techniques are being developed that are refining the process of mixing. For the last century, development of science and technology has given rise to new inventions of tools and machines to improve the mixing and blending practice. As a result, computerised techniques and simulation software's are present in research laboratories throughout the world [2, 3].

Still a lot is to be learned from the past classical methods and techniques which have played a vital role in the art of mixing in developing science [4]. The important role of 'stirring' cannot be ignored in the process of mixing. Every material that has been synthesised has gone through the mixing process [2, 3, 4]. The mixing technology dependent industries are, food, pharmaceutical, paint and rubber industry.

A problem associated with dough mixing is studied by to optimize the product by improving the efficiency of mixture [5, 6]. A case of contra rotating stirrer with rotating container is investigated. It is a two–dimensional cylindrical coordinates study under implementation of Bird Carreau model [2, 7]. The numerical algorithm used in the research study, to obtain the numerical solution, is based on a finite element Taylor–Galerkin/pressure–correction scheme in its semi–implicit form and the system of co–ordinate frame of reference is based on cylindrical polar coordinate system. The scheme is composed of second order time-stepping algorithm in multi-stages. Under the model more shearing and stretching was seen in the fluid. The outcomes of the problem are applicable at industrial level. Results were compared with previously simulated outcomes and observation showed that they are similar to one another.

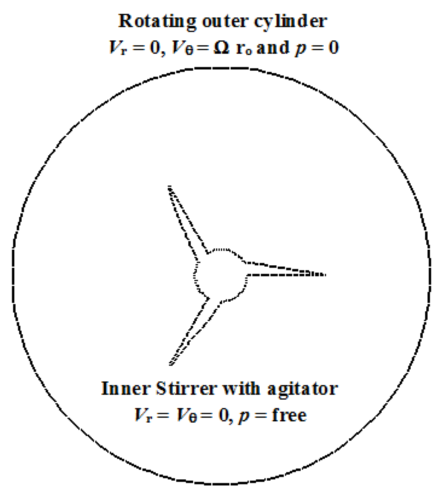
Computational Fluid Dynamics (CFD), principally, is a tool to perform numerical simulations. With the help of powerful computing dynamics, it solves the Navier-Stokes equations applied to a fluid flow situation within the specified boundary conditions. These equations are solved using three acknowledged methods i.e., the finite difference, the finite volume and the finite element methods. Applying such methods one can predict the transfer of heat and mass, and momentum of the fluid. Also estimate the optimal paths and design within specified conditions [5, 6, 7]. In processing industry, the standard methods of partial differential equations based on the Navier-Stokes will continue to lead, in comparison to other mathematical models [8, 9].

The bread making process is an inconsistent process as the composition, physical appearance, properties and structure of the material keeps on changing during the progression of the process. Such multifaceted processes are applicable in CFD modelling. A similar case of heat modelling in electric ovens [8], condensation of starch and scale of browning. The work was the reflection of the work done in the end of nineteenth century and start of the new century, when robust software's of CFD modelling were being developed and compared it with the continuing progress in recent years focusing on the whole process of baking; from baking oven to the transformation process of the bread [5, 8].

In all the above researches, single stirrer in eccentric position or double stirrer without agitator has been investigated widely. In the present study, concentric stirrer with agitator is being investigated, that has never been worked out before.

2. PROBLEM DEFINITION

The role of stirring in mixing process has its unique importance. It is the base of the whole mixing process. Replicated problem investigated in this research study is a mixing of fluid in a cylindrical vessel concentrically placed a single spinning stirrer along with agitator fixed at the top cover lid. The motion of the rotating stirrers along with agitator is analysed against the rotational velocity of vessel. The stirrer rotates in clockwise direction and outer cylindrical container in counter clockwise direction. Industrially, stirrer spins around its axis and external vessel is fixed physically. Because of mathematical complexities, the rotation of external vessel is considered here to be around centre of vessel, instead of the rotation of stirrers, to protect the originality of physical problem.



Figur 1: Two–dimensional computational domain of concentric rotating stirrer with agitator in a cylindrical Vessel.

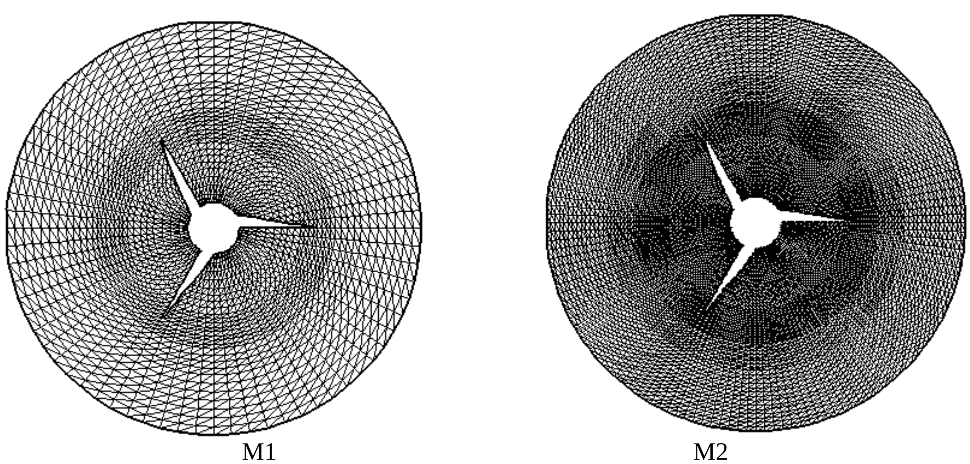


Figure 2: Finite Element Meshes used in computation.

3. GOVERNING EQUATIONS AND NUMERICAL SCHEME

The equations that is being considered here are two–dimensional in cylindrical polar coordinates, i. e., equation of continuity and time–dependent Navier–Stokes equation. A mixed field of primitive variable velocity–pressure formulation is adopted [11]. The incompressible continuity and Navier–Stoke equations are given as:

$$\nabla \cdot \boldsymbol{v} = 0 \qquad in\ \ \Omega\ t \in [0,T] \qquad (1)$$

$$\rho\left(\frac{\partial \boldsymbol{v}}{\partial t} + \boldsymbol{v}\cdot\nabla\boldsymbol{v}\right) = \nabla\cdot\sigma - \nabla p \qquad in\ \Omega\ t \in [0,T] \qquad (2)$$

Where, rate–of–strain tensor '$\underline{\underline{d}}$'is define as: $\underline{\underline{d}} = \frac{1}{2}\left[\nabla \underset{\sim}{u} + (\nabla \underset{\sim}{u})^t\right]$.While, the velocity vector field in cylindrical polar coordinates is $\underset{\sim}{u} \equiv (u_r, u_\theta, 0)$. Whereas, u_θ, u_r are respectively represents azimuthal and radial velocity components, and t is the time. Whilst, ρ and μ are material parameters as fluid density and dynamic viscosity respectively. Per unit density the isotropic fluid pressure is denoted by p .

The flow is modelled for isothermal incompressible Newtonian fluid through so called, a finite element algorithm. Numerical technique is constructed on multi–stages, a Taylor–Galerkin coupled with Pressure–Correction (TGPC) scheme, as introduced by the first explorers of the scheme [6, 7, 8, 10, 11, 12, 13]. Implicitness is employed on only diffusion term to make a semi–implicit form. Predictions are analysed in terms of influences of inertia through dimensionless Reynolds number. Whereas, the effect of dynamics i.e., rotational speed of stirrer and agitator will be explored. The computed numerical results will be illustrated for pressure by isobars and flow structure through contours plots.

Stage–Ia: For given $\underset{\sim}{u}$and p at time–step level n, compute velocity components at half time step:

$$\left(\frac{2}{\Delta t} - \frac{1}{2\,\mathrm{Re}}\nabla^2\right)\left(\underset{\sim}{u}^{n+\frac{1}{2}} - \underset{\sim}{u}^n\right) = \frac{1}{\mathrm{Re}}\nabla^2 \underset{\sim}{u}^n - \left(\underset{\sim}{u}^n \cdot \nabla\right)\underset{\sim}{u}^n - \nabla p^n \tag{3}$$

Stage–Ib:

$$\left(\frac{1}{\Delta t} - \frac{1}{2\,\mathrm{Re}}\nabla^2\right)\left(\underset{\sim}{u}^* - \underset{\sim}{u}^n\right) = \frac{1}{\mathrm{Re}}\nabla^2 \underset{\sim}{u}^n - \nabla p^n - \left(\underset{\sim}{u}^{n+\frac{1}{2}} \cdot \nabla\right)\underset{\sim}{u}^{n+\frac{1}{2}} \tag{4}$$

Stage–2:

$$\theta \nabla^2\left(p^{n+1} - p^n\right) = \frac{1}{\Delta t}\underset{\sim}{u}^* \tag{5}$$

Stage–3:

$$\frac{1}{\Delta t}\left(\underset{\sim}{u}^{n+1} - \underset{\sim}{u}^*\right) + \theta\nabla\left(p^{n+1} - p^n\right) = 0 \tag{6}$$

Where $\underset{\sim}{u}^n, \underset{\sim}{u}^{n+\frac{1}{2}}, \underset{\sim}{u}^*, \underset{\sim}{u}^{n+1}, p^n$ and p^{n+1} are correspondingly nodal velocity and pressure vectors at different time levels (t^n, t^{n+1}).

The vessel has been considered to have infinite height, therefore, two-dimensional flow problem is being investigated here in coordinates (r,θ). Similar problems have been studied with different numbers and shapes of agitators, in two-dimension and three-dimension. However, in three-dimensions the vessel height is assumed to be finite.

4. NUMERICAL RESULTS AND DISCUSSION

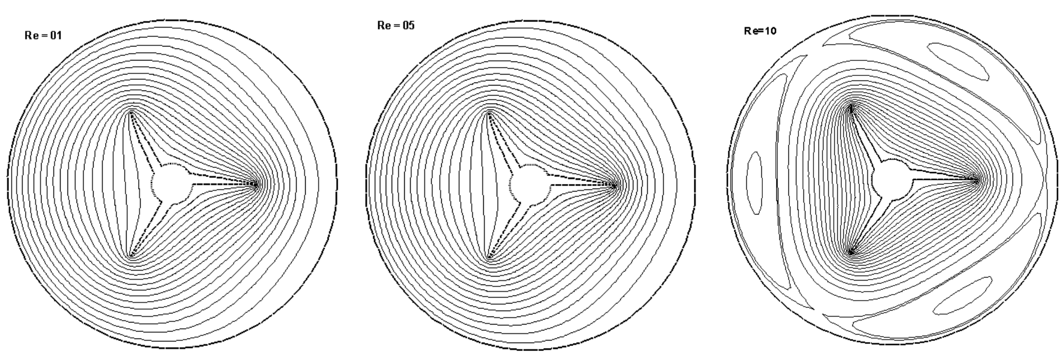
For maximum and minimum values of variables, in terms of contour plots and isobars, predicted numerical solutions are presented. For post processing of the numerical solutions is presented by plotting graphs and contours.

Simulations were considered converged at earlier mentioned order 10^{-6} because of involvement of steady-state. Initial simulations required about 59573 iterations to converge then it reduced to 20 to 30 thousand for next three to four simulations. Later on it took less than 10 thousand iterations to converge after Reynold's number value of 200. Convergence was typically achieved after 4 to 5 hours for initial few Reynold's numbers, later it took less than an hour. Numerical results are demonstrated at both low and relatively high Reynolds number (Re).

4.1 Flow Structure (Ψ) And Pressure Drop (P)

4.1.1 *Influence of Inertia on Flow Structure*

Flow structures of Newtonian fluid, through streamlines are presented in the following figures show themixing process in a cylinder–shaped vessel[14] through single rotating stirrerattached agitator, showing the top view of problem in streamline contours at various values of Reynolds number. It has been identified that the fluid starts to be mixed, at lower Reynolds number values, a weak embryo vortex appears and recirculation starts to mix the material. The vortex becomes strong as the inertial values (Re) increases and mixing improves. At higher values of inertia (Reynolds number), the vortex eye has been observed which shows the good homogenization of material.



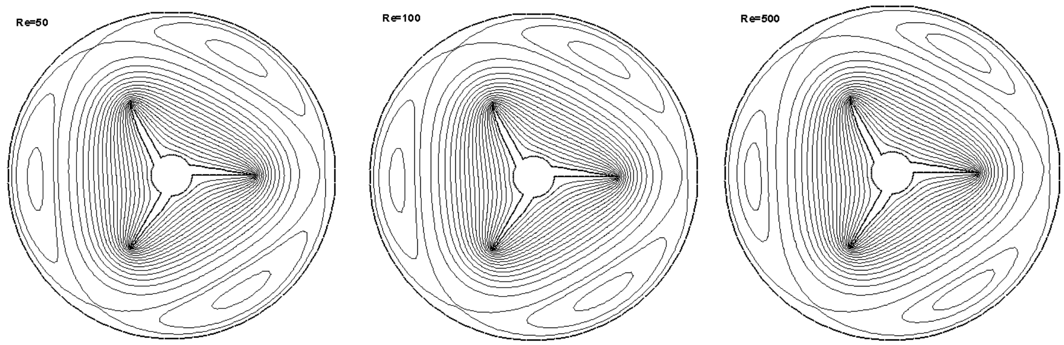


Figure 3: Streamline contours at different Reynolds numbers

4.1.2 *Effect of Inertia on Pressure*

The second objective of the research study is to investigate the pressure gradients. The pressure gradients have been computed by displaying the pressure isobars at various values of Reynolds number. It has been witnessed that the pressure slightly rises at low inertial levels, but rapidly decreases as the value of inertia (Re) increases.

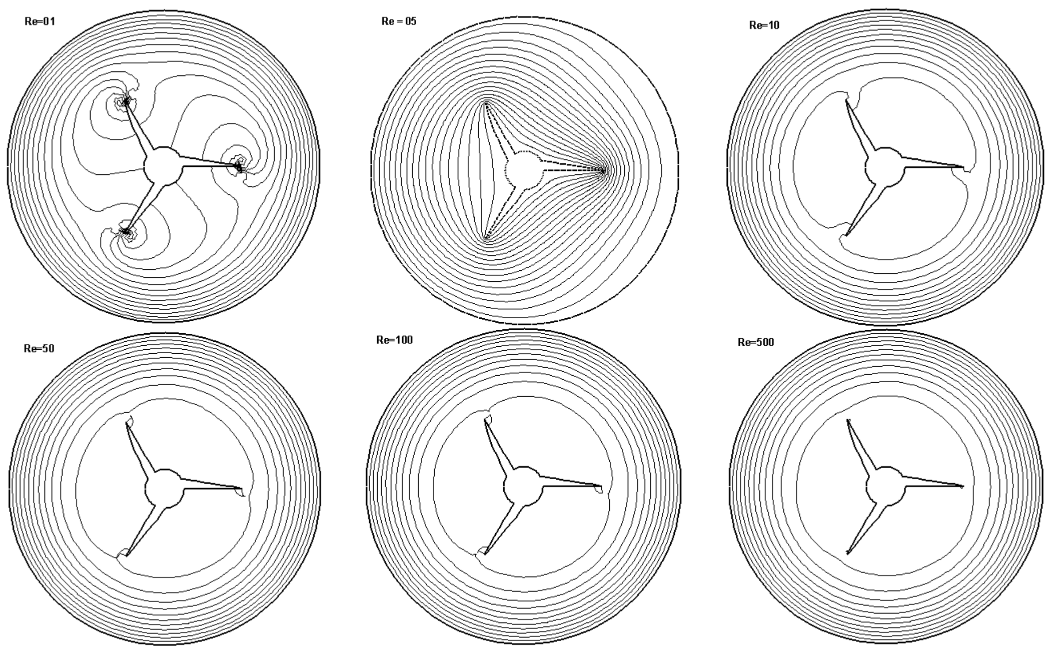


Figure 4: Pressure drop at different Reynolds numbers

4.1.3 *Scaled Pressure Drop and Couette-Correction:*

Couette-Correction ‘C’ as a function depending on Reynolds number and pressure differential, is given as:

$$C = \frac{\left|\nabla P_{\max} - \nabla P_{\min}\right| \times \mathrm{Re}}{\nabla P_{\max}\big|_{\mathrm{Re}=1}} \; .$$

Where, $\nabla P_{\max}$ and $\nabla P_{\min}$ are maximum and minimum pressure differential in domain respectively, while, $\nabla P_{\max}\big|_{\mathrm{Re}=1}$ is maximum pressure differential at Re = 1, and Re is Reynolds number. From computed numerical solution an empirical equation is derived by linear curve fitting as:

C = 0.04754134 + 0.6123122 x Re

It can be observed from Fig.(4) that as the Reynold’s number is increasing, pressure is also increasing. The Couette correction function values are shown in Fig. (5) in round bullets meanwhile the linear curve fitted can be seen in square boxes. It can be observed that Couette correction values are very close to the fitted curve and both the lines in graph are taking a similar path of linearity.

Table 2: Couette Correction and Estimated Linear Curve values

Re	$C = \frac{\lvert \nabla P_{\max} - P_{\min} \rvert \times \mathrm{Re}}{\nabla P_{\max} \rvert_{\mathrm{Re}=1}}$	C = 0.04754134 + 0.6123122 x Re
01	1	0.6598
10	6.4425	6.1706
25	15.4631	15.3553
50	30.6620	30.6631
100	61.2221	61.2787
150	91.7527	91.8943
200	122.4013	122.5099
250	153.0218	153.1255
300	183.6422	183.7412
400	244.8778	244.9724
500	306.1240	306.2036
750	459.2062	459.2816
1000	612.3018	612.3597
1500	918.4929	918.5158
2000	1224.7109	1224.6719
3000	1837.0664	1836.9841

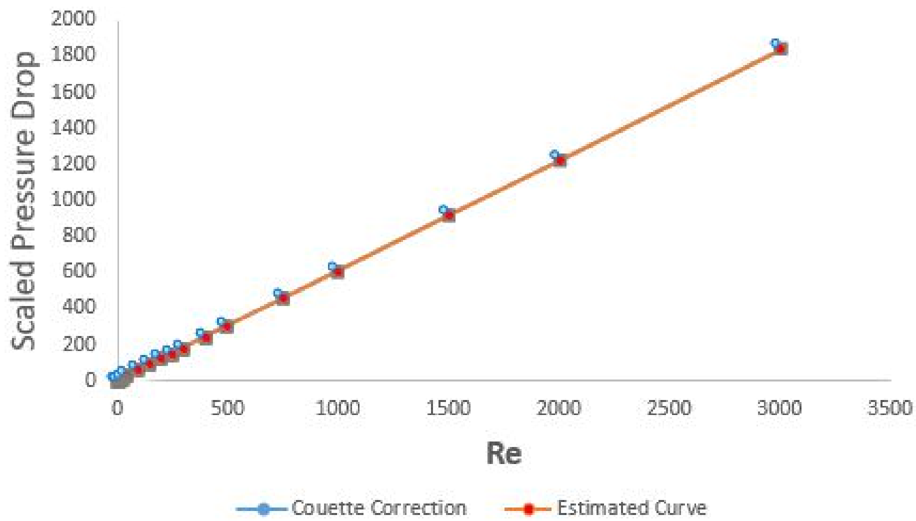


Figure 5: Empirical relationship of Couette-Correction and Pressure drop.

4.2. Work–Done, Local Rate Of Work Done And Power Consumption

Another important objective to be achieved was Work done and Power consumption. These variables are computed using the following scheme:

Work done	$W = \dot{w}(x,t)^n \Delta t$
Local Rate of work done	$\dot{w}(x,t) = \tau : \nabla v$
Total Power consumption	$P_w(t) = \int_{\Omega} \dot{w}(x,t)\, d\Omega$

4.2.1 Work–done

At Reynold number 01 the rate of work done starting from 36.36 dropped to 4 suddenly between 100 and 1000 time–steps that are mentioned in logarithmic scale. Then from 10^4 to 10^5 it keeps maintaining minimum level to 0.307. Initially it has high value so as to start flow of fluid in the vessel.

At Re=10 starting from under 0.308 it increases between 100 and 10^4 time–steps and takes maximum position to 0.37 approaching close to 10^4 time–step.

At Reynolds number 25, similar trend can be observed as in Re = 10. Rate of work done has suddenly increased from 0.315 and reached peak level to 0.41 from time step 100 to 10^4. After that it is fluctuating between that values 0.36 and 0.38.

At Reynolds number 50 and 100 a similar trend can be seen. It can be observed that very small piece of change in work done and work done value is started from the value 0.36 approximate and slowly kept on increasing towards level 0.37 level from time step 1 to time step and 10^4 and after that to 0.38 and maintains the same horizontal path. There is rarely any change in work done values at higher values of Reynolds number difference in work done values is being observed from milli– to micro–level.Table – 03gives you the values of work done at different Reynold's numbers.

4.2.2 Local Rate of Work Done

Local rate–of–work done determines the local power consumption. As work done is multiple of the time interval therefore the values the local rate of work done start at higher position than the work done. But local rate of work done follows exactly similar fashion as the work done, but at higher values. For work–done, local–rate of work–done and power consumption, the minimum and maximum values are tabulated in Table–03 at different level of inertia (Re), whereas, the graphs are illustrated in Fig.(5).

Table 3: Work Done, Local Rate of Work Done and Total Power Consumption:

Reynold's Number	Variable	Minimum Value	Maximum Value
01	Work Done	36.3617	0.3078
	Local Rateof Work Done	153.9129	18182.6034
	Power Consumption	36.3617	41397.6357
05	Work Done	0.3081	0.3299
	Local Rate of Work Done	153.9129	173.7999
	Power Consumption	0.3081	7305.2402
10	Work Done	0.3081	0.3443
	Local Rateof Work Done	153.9129	185.2041
	Power Consumption	0.3081	12137.3132
25	Work Done	0.3081	0.3623
	Local Rate of Work Done	153.9129	204.4060
	Power Consumption	0.3081	27355.1355
50	Work Done	0.3623	0.3797
	Local Rate of Work Done	181.1950	190.1318
	Power Consumption	0.3623	4772.5572
100	Work Done	0.3797	0.4061
	Local Rate of Work Done	189.8679	203.0523
	Power Consumption	0.3797	3575.1074
150	Work Done	0.4061	0.4208
	Local Rate of Work Done	203.0570	213.0159
	Power Consumption	0.4061	10829.5828
200	Work Done	0.4208	0.4327
	Local Rate of Work Done	210.4360	216.3815
	Power Consumption	0.4209	2127.7127
300	Work Done	0.2140	0.2164
	Local Rate of Work Done	216.4060	216.4400
	Power Consumption	0.2164	3.8959
500	Work Done	0.2150	0.2165
	Local Rate of Work Done	216.4980	216.5490
	Power Consumption	0.2164	3.2481
750	Work Done	0.2165	0.2166
	Local Rate of Work Done	216.5540	216.6290
	Power Consumption	0.2165	3.8990
1000	Work Done	0.2163	0.2168
	Local Rate of Work Done	216.6340	216.6820
	Power Consumption	0.2166	2.3834

4.2.3 Total Power Consumption

At the start of first simulation with inertial level at 1, the graph of power consumption is displaying a minimum value of 36.36 and maximum of 41397.63 which is very high. It is high initially to drive the flow in the vessel. At Re = 5 it drops rapidly and minimum is 0.3081 and maximum is 7.305.24, that interprets that the flow is now in circulating and very power requirement has become lesser than the previous condition. Then at inertial levels of 10 and 25 it has again taken a medium figure as compared to inertial levels of 1 and 5. But as we increase the inertial levels further, it is dropping down rapidly and reaching at some steady state at the end of each Reynolds number value. We can observe that after Reynolds number 200 it is at very low level as there is no work done and power consumption has also reached is minimum levels.

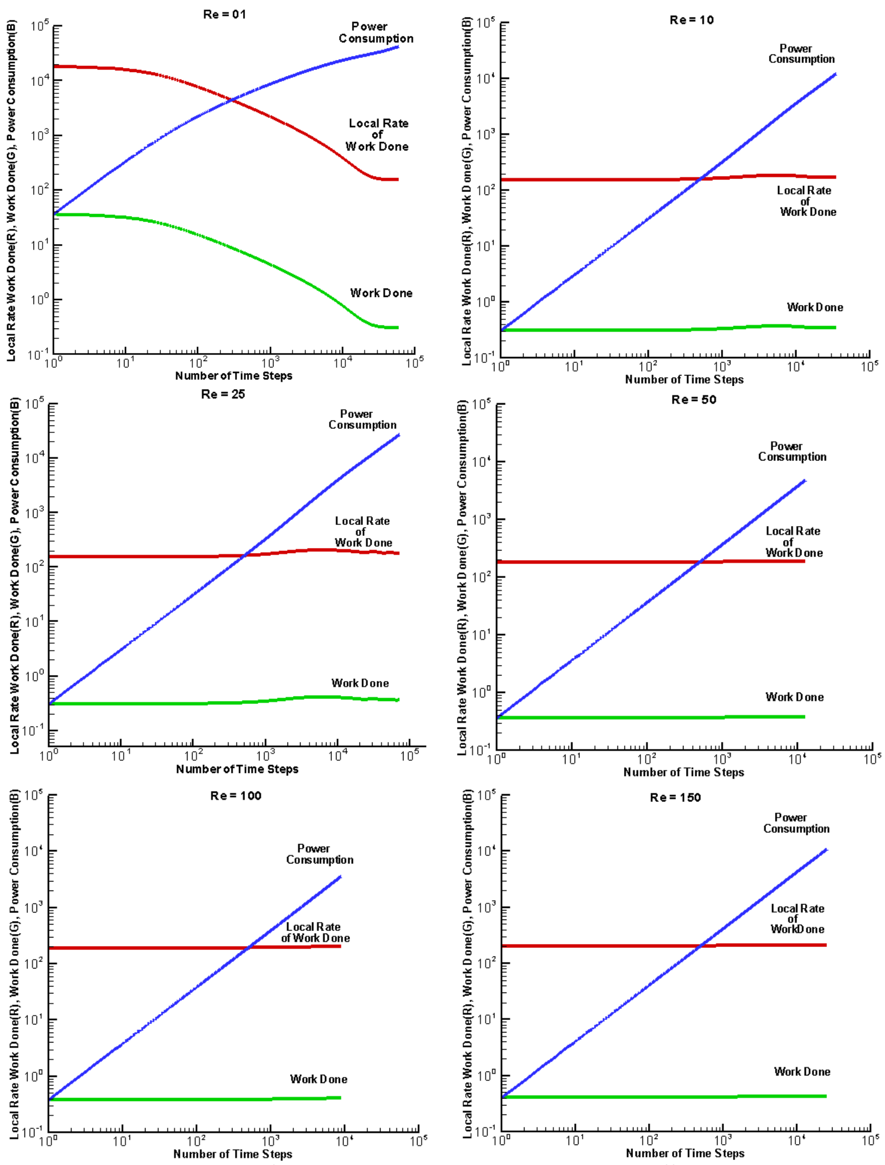


Figure 5: Work Done, Local Rate of Work Done and Power Consumption at different Reynold's Number.

5. CONCLUSION

Finite element model of mixing of fluid in cylindrical vessel by rotating stirrer concentrically fixed with agitator was developed successfully. The goals and objectives that were targeted in this study have been achieved. For a complex mixing process, physically realistic simulations have been provided using Newtonian fluid. Behaviour of fluid at different Reynolds numbers using stream function and pressure gradient is observed. Work done and prediction of power consumption have been presented with numerical results. All the results are in good agreement with the previous available experimental and numerical data in related literature. The capability of Taylor-Galerkin/pressure-correction scheme has been demonstrated successfully and can be mentioned being robust and most promising and reliable in forecasting.

The mixing process using agitator is accomplished at low Reynolds numbers that saves the cost of power consumption. Pressure isobars are observed at low Reynolds numbers and rapidly increasing pressure as inertial value increases. Appearance of vortex starting from low Reynold number, Re = 10 improves homogenisation of material by recirculation.

Simulations were performed for various Reynolds numbers for testing purpose starting from Re=1 to Re = 3000. But it was observed that most of the work done was accomplished at very low values of Reynolds number. Similarly, the power consumption that was seen very high at the start of the process, dropped rapidly as the inertial levels were increased. Hence the presence of agitator improved the mixing process as the presence of vortices supported homogenisation of material and work done reached asymptotic solution at the earliest inertial levels. This design of rotating stirrer with agitator can be used in related domestic and commercial appliances related to dough kneading and other mixing processes. The work has been tested for Newtonian fluid, it can be further tested for non-Newtonian fluids for different inertial levels.

ACKNOWLEDGEMENT

The authors are gratefully acknowledging the Department of Computer Science, ISRA university, Hyderabad, providing computation facilities to accomplish the research work.

Nomenclature					
Symbols	**Description**	**Symbols**	**Description**	**Symbols**	**Description**
r	Radial Axis	t	Time	μ	Dynamic Viscosity
□	Azimuthal Axis	p	Fluid Pressure	Ω	Domain of Interest
$\boldsymbol{v}$	Velocity Vector	n	Time Step Level	$'\underline{\underline{d}}'$	Deformation Tensor
u_r	Radial Velocity	ρ	Fluid Density	σ	Stress Tensor
$u_□$	Azimuthal Velocity	Re	Reynolds Number	∇	Differential Operator

REFERENCES

[1] Memon, R. A., Solangi, M. A., and BALOCH, A., “Stress Analysis of Mixing of Non-Newtonian Flows in Cylindrical Vessel Induced by Co-Rotating Stirrers”, *Mehran University Research Journal of Engineering and Technology*, Vol.*32*, No.2, pp. 283-286, 2013.

[2] Memon, R., “Finite Element Modelling of Complex Rotational Flows”, *Ph. D. Thesis*, Bahauddin Zakarya University, Multan, 2012.

[3] Memon, R., Solangi, M., Khokhar, R., Baloch, A., and Sayed, K., “Computational Analysis of Rotating Mixing of Bird Carreau Model Fluid”, *Sindh University Research Journal-SURJ (Science Series)*, Vol. 47, No. 1, pp. 15-18, 2015.

[4] Galaktionov, O. S., “Optimization of Distributive Mixing: From Prototype Flows to Industrial Devices”, Doctoral dissertation, Eindhoven University of Technology,2002.

[5] Ding, D., and Webster, M. F., “Three-Dimensional Numerical Simulation of Dough Kneading”, *XIII Int. Cong. on Rheol, British Society of Rheology, Cambridge*, UK, Vol. 2,pp. 318-320, 2000.

[6] Baloch, A., Grant, P. W. and Webster, M. F., “Parallel Computation of Two-Dimensional Rotational Flows of Viscoelastic Fluids in Cylindrical Vessels”, *Engineering Computations*, Vol.19, No.7, pp. 820-853, 2002.

[7] Baloch, A., Solangi, M. A., and Memon, G. M., “Simulation of Rotational Flows in Cylindrical Vessel with Rotating Single Stirrer”, *Punjab University Journal of Mathematics*, Vol. 40, pp. 83-96, 2008.

[8] Townsend, P. and Webster, M. F., “An Algorithm for the Three–Dimensional Transient Simulation of Non–Newtonian Fluid Flows”, In Proc. of NUMETA Conf., Numerical Methods in Eng.Theory and Applications, NUMETA 87,Vol2, T12/1–I, 1, 1987.

[9] Chhanwal, N., Tank, A., Raghavarao, K. S. M. S. and Anandharamakrishnan, C.,“Computational Fluid Dynamics (CFD) Modelling for Bread Baking Process—A Review”, *Food and Bioprocess Technology*, Vol. 5, No. 4, pp. 1157-1172, 2012.

[10] Haidari, A. H. and Matthews, B., “Future Trends for Computational Fluid Dynamics in the Process Industry”. *Third International Conference on CFD in the Minerals and Process Industries CSIRO*, Melbourne, Australia”, Vol.1, pp.331-338, 2003.

[11] Bhatti, I., Baloch, A., and Qureshi, K. (2010), “Finite Element Modelling of Rotating Mixing of Toothpaste”, *World Academy of Science, Engineering and Technology, International Journal of Chemical, Molecular, Nuclear, Materials and Metallurgical Engineering*, Vol. 4, No.1, pp. 47-52.

[12] Donea, J., “A Taylor–Galerkin Method for Convective Transport Problems”, *International*

Journal for Numerical Methods in Engineering, Vol. *20*, No.1,pp. 101-119, 1984.

[13] Al-Muslimawi, A. H., and Shakir, S. D., “Numerical Analysis of the Taylor–Galerkin/Pressure–Correction (TGPC) Finite Element Method for Newtonian Fluid”, *Mathematical Theory and Modelling*, Vol. *5*, No. 8, pp. 259-272, 2015.

[14] Sujatha, K. S. and Webster, M. F.,"Modelling Three-Dimensional Rotating Flows in Cylindrical-Shaped Vessels", International Journal for Numerical Methods in Fluids, 2003.