The stress and strain analysis double-layer and eight-leaf backward curved impeller disc agitator

C C Cheng¹, S P Wei¹, A D Lin², C H Hsieh¹ and C M Hsu¹*

¹Department of Mechanical Engineering, National Kaohsiung University of Science and Technology, Kaohsiung, 807, Taiwan
²Department of Mechanical Engineering, Cheng Shiu University, Kaohsiung, 833, Taiwan
*Email: jammy@nkust.edu.tw

Abstract. The mixing wing’s stress and strain value of the structure in the mixing equipment needs to be studied and discussed. The research is uses the FLUENT module to simulate the flow field state in the rotating operation of the mixing tank, the stress and strain of the mixing blade, and the modal analysis of the predicted resonance frequency. First, Fluent is uses simulate the shape of the flow field inside the stirred tank. Situation of different speeds, the deformation state of the stress, strain and total displacement of the stirring wing assembly is obtained to achieve the ANSYS fluid-solid coupling effect analysis. Finally, predict the modal appearance and failure points of the simulated stirring wing assembly at different resonance frequencies. After analysis, the rotation speed of the stirring wing is increases, it must be subjected to the centrifugal force of fluid distribution, vortex gravitation, and up and down convection agitation. The total displacement and deformation of the rigid structure of the stirring wing can be obtained. The equivalent stress and strain also adopt a progressive linear growth phenomenon.

1. Introduction

There is a lot work to undertake in terms of stirrer design so as to induce perfect mixing in a horizontal configuration that is much proper to plug flow. The same holds true for the LIW feeders for which the smoothing of the mass flow fluctuations due to the feeding screw is a key issue. The location of these feeders along the mixer needs to be studied into details because, as we pointed out in the present work, segregation by percolation takes place close to the inlet wall [1]. For a stirred process, global mixing time and power consumption are the main performance indicators [2]. As Reynolds number increases, the heat transfer capacity has been significantly improved [3]. Ibrahim study influence of the geometry, parameters of the stirring blade and the stirring Reynolds number on the flow pattern by schlieren method [4]. Low-viscosity fluids can flow at will, and it’s produce larger flow and velocity easily, so that the flow field can reach a turbulent state, forming turbulent vortex or shearing motion to help stirring and mixing. Yianneskis and Whitelaw use LDA experimental method to measure and combine the positioning method of the optical coupler, and the speed change measurement is performed synchronously with the rotation of the stirring shaft, so as to study and analysis the vortex structure dragged by the rotation of the stirring wing blade, and its maximum turbulent flow can be close to the central of the drag vortex and around the blade [5]. Yianneskis tested the velocity of the flow field and the gap between the different blades and the bottom of the tank mixing tank. It was found that the drag vortex of the mixing wing blade will expand with the increase in the distance of the mixing blade. It’s also the place where the turbulence of the mixing tank fluid is the strongest [6]. In addition, the velocity efficiency of the flow field is used to measure the evaluation of the dissipation rate of turbulent energy. Closer to the area to the impeller is the higher dissipation rate and the rate of turbulence are. However,
as the area between the impeller and the baffle is farther away, the energy rate and the dissipation rate also decreases, and closer to the mixing impeller and the partition area, the less isotropic the flow field [7]. For high-viscosity fluids, the flow speed is slow which makes it hard to form turbulent flow. It can only flow in a laminar flow pattern, and it can’t form a vortex to help the mixing effect. It can only rely on the rotation of the stirring wing, there is a high shear force around it to smoothly cut the fluid mass.

Stoots and Calabrese use the LDA experimental method to measure and combine the positioning of the optical coupler to rotate with the stirring shaft wing, and synchronously implement the speed change measurement of the flat impeller disk turbine agitator, and explore the closer to the stirring wing is more cyclical and unequal. It’s inferred that the fluid mass is introduced by the vortex and then ejected in a tangential flow along the axis of the vortex. The gradient of the average velocity is used to form the deformation rate for discussion vortex structural influence [8]. When the fluid moves away from the stirring wing, the shearing force weakens and the fluid viscosity returns to initial. When the phenomenon of high turbulence occurs, the fluid is affected by the vortex viscosity formed by the turbulent flow that causes the fluid mass in the tank sheared by the stirring blade, reduces the fluid viscosity effect and improves the fluid flow mixing instantly. The low steady flow contrary which the phenomenon occurs, the viscosity effect between fluid molecules becomes larger, and increases the difficulty of mixing and stirring. This phenomenon is like Galindo called the cave effect [9]. According to Nienow, Conti et al. and Armenante et al., the research results of different stirring wings are consistent. When adjusting the position of the stirring wings, it’s necessary to pay attention to the distance between the stirring wings and the bottom of the tank to minimize complete suspension and need high speed to achieve the desired goal [10-12]. Therefore, the mixing and stirring circulation rate and changing the turbulence intensity of the flow field are the main controlling factors. The volume circulation rate is the most important factor. As discussed by Kato et al., it’s explored that adding a baffle plate at the bottom of the stirring tank can increase the downward pump after the fluid hits the baffle, the relative speed of the upward pumping fluid can improve the effect of particle suspension, and reach the minimum speed of particle suspension [13].

2. Methods

2.1. Mass conservation equation

Any fluid problem must satisfy the law of conservation, which can be expressed as: the mass increase in fluid micro-component in unit time is equal to the net mass flowing into the micro-component in the same time interval. According to this law, the mass conservation equation can be obtained:

$$\frac{\partial \rho}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial w}{\partial z} = 0$$  \hspace{1cm} (1)

Among them, time is $t$, density is $\rho$, $u$, $v$, $w$, are velocity vectors, and the vectors are in $x$, $y$, and $z$ directions. For incompressible homogeneous fluids, where the density is constant, the formula is as follows:

$$\frac{\partial \rho}{\partial x} + \frac{\partial \rho}{\partial y} + \frac{\partial \rho}{\partial z} = 0$$  \hspace{1cm} (2)

2.2. Momentum conservation equation

It’s also a basic law that must be satisfied by any flow system. The law can be expressed as: the rate of change of momentum in a micro volume with respect to time is equal to the sum of various forces acting on the micro volume. This law is subordinate to Newton’s second law. According to this law, the momentum conservation equation can be obtained:

$$\rho \frac{dv}{dt} = \rho F - \nabla \rho$$  \hspace{1cm} (3)

The conservation of momentum in the three directions of $x$, $y$, and $z$ is obtained as

$$\frac{\partial (\rho u)}{\partial t} + \text{div}(\rho u) = - \frac{\partial (\rho)}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + F_x$$
\[
\frac{\partial (\rho v)}{\partial t} + \text{div}(\rho vu) = - \frac{\partial (\rho)}{\partial y} + \frac{\partial (\tau_{xy})}{\partial x} + \frac{\partial (\tau_{yy})}{\partial y} + \frac{\partial (\tau_{zy})}{\partial z} + F_y \tag{4}
\]

\[
\frac{\partial (\rho w)}{\partial t} + \text{div}(\rho wu) = - \frac{\partial (\rho)}{\partial z} + \frac{\partial (\tau_{xz})}{\partial x} + \frac{\partial (\tau_{yz})}{\partial y} + \frac{\partial (\tau_{zx})}{\partial z} + F_z
\]

The pressure on the fluid tiny volume, \(\tau_{xx}, \tau_{xy}, \tau_{xz}\) are the components of the viscous shear stress acting on the surface of the fluid microvolume due to molecular viscosity, and \(F_x, F_y, F_z\) are the volume forces on the microvolume.

### 2.3. Energy conservation equation

The law of conservation of energy is the basic law that must satisfy the heat exchange flow system. The law is expressed as: the energy increase rate of small volume of fluid is equal the net heat flow into the small volume, and plus the work by the volume force and surface force on the small volume. This law is actually the first law of thermodynamics.

\[
\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} [\rho u_i (\rho E + p)] = \frac{\partial}{\partial x_i} \left[ k_{\text{eff}} \frac{\partial T}{\partial x_i} - \sum_{j'} h_{j'j} J_{j'} + u_j (\tau_{ij})_{\text{eff}} \right] + S_h \tag{5}
\]

The energy \(E\) of the fluid is usually the internal energy \(i\), kinetic energy \(K = \frac{1}{2} (u^2 + v^2 + w^2)\) and the sum of the three terms of potential energy \(p\), there is a certain relationship between internal energy \(i\) and temperature \(T\), that \(i = C_p T\), \(C_p\) is SHC. The energy conservation equation with temperature \(T\) as a variable is:

\[
\frac{\partial (\rho T)}{\partial t} + \text{div}(\rho u T) = \text{div} \left( \frac{K}{C_p} \nabla T \right) + S_T \tag{6}
\]

Therefore, in order to analysis the governing equations, the general variable \(\Phi\) is used to express:

\[
\frac{\partial (\rho \Phi)}{\partial t} + \text{div}(\rho u \Phi) = \text{div} \left( \frac{K}{C_p} \nabla \Phi \right) + S
\]

It’s only consider general methods to solve different types of fluid flow and heat transfer problems, and for different \(\Phi\), only need to repeat the program and give appropriate initial programs and boundary conditions to solve.

### 3. Results

#### 3.1. Flow field pressure analysis

When the stirring and mixing approach the steady state, the static pressure distribution of the shaft section in the stirring tank can be understood from the figure that the mechanical energy exerted by the agitator impeller increases the fluid pressure energy around the bottom of the impeller. The radial diffusion and kinetic energy is converted to displacement energy, and the static pressure value also increases with increases as the radius of the tank, and low pressure zone is also generated at the axis of the agitator.

The agitation of the fluid around the upper impeller is affected by the fluid energy agitated by the lower impeller, causing the upper impeller have higher pressure value and a larger range of pressure disturbance frequency than the lower impeller; While the lower impeller is affected by the stirring barrel, the distance from the bottom of the groove makes the fluid velocity that the lower impeller shows higher upside and under disturbance frequencies.
3.2. Flow field velocity analysis
Stirring and mixing are carried out in a turbulent fluid environment, and the flow field in the mixing tank is affected by the fluctuation of the turbulent flow rate. Regardless of the magnitude or direction of the fluctuation, it’s continuous mixing and stirring. It’s expressed as the sum of the average value and the fluctuating flow velocity in this direction. However, the fluctuating flow velocity is instantaneous, with different flow patterns. The change cycle in the tank varies from half a second to several minutes, and alternately and randomly.

The high-speed rotation of the agitator, a strong circumferential velocity is formed, which causes the fluid to be affected by centrifugal force, forming a strong radial velocity from the outer edge of the blade, bringing the fluid at the upper and lower edges of the blade into the groove to flow field, after colliding with the inner wall of the stirring tank, then moved upwards and downwards into two larger fluid clusters to be carried everywhere in the stirring tank. Therefore, when the blades of the impeller of the agitator are more angled, they will produce stronger radial injection speed under the blades. Therefore, the flow field outside the radius of the impeller blade of the agitator that is inward flow and above the blade. The far away from the blade, the effect of the flow field will gradually decrease with the rotation of the blade that making the flow field, the velocity of the flow velocity in the circumferential direction also becomes smaller.

3.3. Stress and strain analysis of stirring wing
When the fluid machinery impeller operating normally in the stirring tank, the agitator impeller is often subjected to stable or unstable fluid distribution, vortex gravity, and centrifugal force caused by various factors in the rotating operation, which indirectly affects the impeller, blades, shaft and other parts of the agitator to produce internal stress concentration phenomenon (Figure 3). Resulting in high-speed operation of the agitator, the impeller, blades, and shaft are prone to structural rigid deformation and fracture. In order to understand agitator running at different speeds in the fluid that simulations software will be used to explore the internal flow field pattern of the stirring tank, and obtain the stress, strain and total displacement of the rigid structure of the stirring wing assembly.

In this study, the agitator is used 250RPM and the blade thickness is 0.5mm. The total deformation position of the agitator assembly is roughly total displacement and deformation of the rigid structure of the blade reaches 0.195mm, and the agitator assembly positions is the maximum equivalent stress and strain generated are "the root of the impeller of the agitator", the maximum stress is 24.596 MPa, and the maximum strain is 0.00013 mm/mm (Figures 1 to 2).

After investigating this study, the rigid structure of the agitator assembly is changed by the fluid agitation, the impeller blades have undergone slight deformation, and the high-speed operation of the agitator will affect deformation of the impeller blades more deeply severely and the largest. The effect force and strain are both at the root of the impeller blade; Therefore, the "thickness value" of the stirring blade will affect one of the variable factors of the stirrer rotation, because the thickness of the blade must be able to withstand the waves generated by the fluid that the power without damage or deformation. Of course, the thickness of the blade must be as thin as possible to reduce the weight of the mixing wing assembly and achieve the purpose of saving the production cost of the mixing system.

It’s also discussed that as the rotation speed of the stirring wing increases, the agitator assembly must withstand the centrifugal force of stable or unstable fluid distribution, vortex gravitation, and vertical convection agitation. The rigid structure of the stirring wing assembly the displacement and deformation, the maximum equivalent stress and strain also adopt the phenomenon of gradual linear growth (Figure 4).
Figure 1. Schematic diagram of stress and strain of agitator impeller.

Figure 2. A detailed enlarged view of the stress and strain profile of the impeller of the agitator.

Figure 3. The distribution of the fluid disturbance force around the impeller of the agitator.

Figure 4. Relation diagram of total displacement, equivalent stress and equivalent strain of agitator impeller rotation.
4. Conclusion

The agitator in the mixing tank rotates at 250RPM, and affected by the flow rate and pressure of the flow field, resulting in unidirectional fluid-solid coupling pre-stress. The total deformation position is approximately at the tip of the agitator impeller, and the maximum total displacement is about 0.195mm, but the maximum equivalent stress position is in the middle of the impeller blade root of the agitator (maximum stress is 24.6MPa). It seen that the “thickness value” of the agitator blade will affect the rotation of the agitator. To withstand the wave power generated by the fluid without being damaged or deformed and bent. Of course, the thickness of the blade must be as thin as possible to reduce the weight of the stirring wing assembly and achieve the purpose of saving the production cost of the stirring system.

Also, in the condition of the steady or unstable fluid stirring by the fierce turbulence receive the forces by vortex and the centrifugal force originating from up-and-down circulation, hence the agitator increases with the increase of the stirring wings speed and thus can acquires the linear increasing results of the total displacements of stirring wings' rigid structure and the maximum Von-Mises.

With different purposes of stirrings, through the computer simulation of various fluid field and intensity analysis to make sure that agitating vessels and agitators meet the structure load in the qualified vane shape, structure, amounts.

References

[1] Ammarcha C, Gatumel C, Dirion J L, Cabassud M, Mizonov V and Berthiaux H 2019 Powder flow and mixing in a continuous mixer operating in either transitory or steady-state regimes: Mesoscopic Markov chain models, Contents lists available at ScienceDirect, Powder Technology 346 (2019) 116-136

[2] Zhang P, Chen G, Duan J and Wang W 2018 Mixing characteristics in a vessel equipped with cylindrical stirrer, Contents lists available at ScienceDirect, Results in Physics 10 (2018) 699-705

[3] Li H W, Gao Y F, Du C H and Hong W P 2021 Numerical study on swirl cooling flow, heat transfer and stress characteristics based on fluid-structure coupling method under different swirl chamber heights and Reynolds numbers, Contents lists available at ScienceDirect, International Journal of Heat and Mass Transfer 173 (2021) 121228

[4] Ibrahim S and Kienow A W 1995 Power curves and flow patterns for a range of impellers in Newtonian fluids: 40 \( < \) Re \( < \) 5 \( \times \) 105 [J]. Chemical Engineering Research and Design, 73(A5): pp 485-491

[5] Yianneskis M. and Joshi J B, 1993 On the structure of the trailing vortices around rushton turbning blades, Transactions of the Institution of Chemical Engineers vol 71 Part A, pp 543-550

[6] Yianneskis M, Popiolek Z and Whitelaw J H, 1987 An experimental study of the steady and unsteady flow characteristics of reactors, Journal of Fluid Mechanics vol 175 pp 537-555

[7] Lee K C and Yianneskis M, 1998 Turbulence properties of the Impeller Stream of a Rushton Turbine, Aiche Journal vol 44 no 1 pp 13-24

[8] Stoots C M and Calabrese R V Mean Velocity Field Relative to a Rushton Turbine Blade, Aiche Journal vol 41 no 1 pp 1-11

[9] Galindo E M, Arguello A, Velasco D, Albiter V and Martinez A 1996 A Comparison of Cavern Development in Mixing a Yield Stress Fluid by Rushton and Intermig Impellers, Chem. Eng. Technol. pp 315-323

[10] Nienow A W, 1968 Suspension of Solid Particles in Turbine Agitated Baffled Vessels Vessels, Chem. Eng. Sci. vol. 23 pp 1453-1459

[11] Conti R, Sicardi S and Specchia V 1981 Short Communication, the Chem. Eng. J. vol 22 pp 247-249

[12] Armenante M and Piero E U, Nagamine and Susanto J, 1998 Detremination of Correlation to Predict the Minimum Agitation Speed for Complete Solid Suspension in Agitated Vessels, the Can. J. of Chem. Eng. vol 76 pp 413-419

[13] Kato Y, Hiraoka S, Tada Y, Ohnishi M, Naganawa T, Yamauchi R and Shiobara K 2002 Mixing Performance of an Agitated Vessel with Bottom Baffles, J. Chem. Eng. Japan, 35(2), pp 208-210