MODELLING FLOW IN A MIXING TANK

A. GENERAL

This tutorial shows the way of modelling the flow of a liquid (single-phase) inside a mixing tank. This process, usually referred to as **"batch" mixing**, is very frequently used in many industry sectors: chemicals, waste treatment, food, pharmaceutical, etc. Only the flow will be solved in the present tutorial, with emphasis given in simulating the rotating parts which drive the flow (impellers), as well as the stationary parts (baffles, tank walls).

The application will be solved initially in steady conditions, using the Multiple Reference Frame model of FLUENT. Then, unsteady calculations will be conducted in order to simulate the mixing time, where an artificial tracer liquid will be released at time t=0 [s] and then it's concentration will be monitored inside the tank.

B. DESCRIPTION OF THE CASE

The tank is a cylindrical vessel of diameter D=2.8 [m] and height L_v =2.22 [m] with a spherical bottom of height L_b =0.5 [m]. The vertical axis y has its origin at the point where the cylindrical and spherical walls are joined. The tank has one central axis of diameter D_a =0.075 [m], upon which two impellers are mounted; (a) an axial 3–blade one of diameter D_t =1.30 [m] at the top of the tank (y=1.66 [m]) and (b) a radial 6–blade one of diameter D_b =1.0 [m] at the bottom of the tank (y=0.2 [m]). The blades of the axial impeller are inclined by 45°, whereas the blades of the radial impeller are perpendicular.



Figure 1: Geometry of the Mixing Tank.

The combined effect of this configuration enhances recirculation inside the tank and aims at inducing high velocities in all its volume. The axial impeller drives the liquid in a downward direction towards the radial impeller, which, in turn pushes the liquid radially towards the tank side walls. The liquid then goes up along the tank side wall and then it converges towards the tank axis and above the top impeller to close the flow loop. Fig. 2 shows the flow field created by each impeller type, as well as their combination.



) Radial (b) Axial (c) Combination Figure 2: Induced Flow Field as a Function of Impeller Type.

The two impellers are mounted in the shaft, hence they are rotating with the same speed of N=200 [rpm] in clockwise manner, creating a flow field described in the previous paragraph.

The tank is considered closed, hence the top boundary is a wall. In case the tank is open and the distortion of the free surface is small, the boundary can be approximated as a symmetry plane, otherwise a multiphase simulation has to be conducted, taken into consideration a portion of the air above the free surface (VOF model).

The tank also exhibits three baffles of width W=20 [cm] and thickness s=3.5 [cm], covering the whole length of the cylindrical part of the tank. Baffles enhance mixing, however numerically are the reason that impose the simultaneous employment of at least two reference frames: (a) a rotating and (b) a stationary one. In case that the two impellers are rotating at different speeds, then more reference frames should be used.

The operating liquid is supposed to be a chemical substance (paint), lighter than water (density=950 [kg/m³]) and Newtonian with a dynamic viscosity of 0.005 [kg/m–s].

Finally, the total liquid volume of the tank can be easily calculated by FLUENT to be 15.21 [m³] (see 6th step).

C. PREPARATION

Copy the file "mixing-tank_tet.msh" to the working directory folder (e.g. C:\simtec\FLUENT) in your PC. Start the 3d version of FLUENT by double-clicking the FLUENT6.3.26 icon on your desktop:



FLUENT Version	×
Versions	
2d	
2ddp	
3d	
3ddp	
Selection	
3d	
Mode Full Sin	nulation 💌

1st STEP: MESH

Read the mesh file: File > Read > Case...

and select the file "mixing-tank_tet.msh". The mesh file does not require grid scaling, as it was created in [m].

Check the mesh for errors: Grid > Check

In addition to checking for error messages and warnings, check that the minimum volume is positive.

Reorder the mesh: Grid > Reorder > Zones and Grid > Reorder > Domain

These actions will reorder the indexing of the elements and zones of the mesh, minimising the bandwidth of the solution matrix, hence minimizing CPU required for each iteration.

Display the mesh: Display > Grid...

and keeping the defaults press button **Display**. The result is the tetrahedral mesh:

💶 Grid Display	X	
Options Edge Type Nodes C All Edges Feature Faces Outline Partitions Shrink Factor Feature Angle 20	Surfaces default-interior:020 default-interior:021 default-interior:022 default-interior:023 per-bottom per-main per-top	
Surface Name Pattern Match	Surface Types axis clip-surf exhaust-fan fan Outline Interior	
Display Colors	Close Help	

Figure 3: Display of the Mesh ("Edges" Mode).

Familiarize yourself with the names of the various surfaces. It is clear that only 1/3 of the geometry is modelled. Since there are, equally spaced along the perimeter, 3 baffles, 3 blades of the axial and 6 blades of the axial impeller, 1/3 is the minimum "slice" that can be modelled with rotationally periodic boundary conditions (periodic boundaries are shown in cyan).



If we display the mesh in "Faces" mode and activate the lights: Display > Options...

and then by changing face colour and transparency in **Display** > **Scene...** (for more details please examine the FLUENT files themselves) the following result appears.



Figure 4: Display of the Mesh ("Faces" Mode).

2nd STEP: Models

> Activate node based gradient calculation in the steady solver:

Define > Models > Solver...

This method increases accuracy in the calculation of the gradient of the variables in tet meshes, like the present one.

Solver	Formulation
 Pressure Based Density Based 	C Implicit
Space	Time
C 2D C Axisymmetric C Axisymmetric Swirl C 3D	SteadyUnsteady
Velocity Formulation	
Gradient Option	Porous Formulation
 Green-Gauss Cell Base Green-Gauss Node Bas Least Squares Cell Bas 	ed O Physical Velocity

Activate "k–epsilon (2 eqn)" category in "Model", then "RNG" in "k–epsilon model" and finally "Standard Wall Functions" in the "Near–Wall Treatment":

Define > Models > Viscous...

<u>3rd STEP: Materials</u>

> Define the operating liquid:

Define > Materials...

In "Name" delete "air" and type "paint". Then change the values of Density [kg/m3]" to "950" and "Viscosity [kg/m–s]" to "0.005".

💶 Materials					X
Name		Material Ty	De		Order Materials By
paint		fluid		•	• Name
Chemical Formula		Fluent Fluid	Materials		O Chemical Formula
		paint		•	Fluent Database
		Mixture			User-Defined Database
		none		~	
Properties					
Density (kg/m3)	constant		▼ Edit		
	950				
Viscosity (kg/m-s)	constant		▼ Edit		
	0.005				
			-		
	Change/Create	Delete	Close	Hel	n
		201010			<u> </u>

Press button "Change/Create" and answer "Yes" in the following dialogue panel in order to store the above changes.

Question	×
2	Change/Create mixture and Overwrite air?
	Yes No

4th STEP: Periodic Conditions

> Keep the defaults in the periodic conditions panel:

Define > Periodic Conditions...

This panel is designed for input when the simulation includes streamwise/translational (and not rotational) periodic conditions.

5th STEP: Boundary Conditions

> In the boundary conditions we must input data at all domain boundaries.

Define > **Boundary Conditions...** Choose the boundary zones at "Zone" on the left and then press button **Set** to edit/modify the values of the boundary conditions.

(i) "fluid-rotating-bottom". In this fluid zone we will prescribe the rotation of the bottom impeller. This zone can be clearly seen in the GAMBIT file ("mixing-

tank_tet.dbs"); it's the 1/3 of a cylindrical volume that has an axis identical to the axis of rotation and totally surrounds the bottom impeller. It's boundaries lie between the impeller tips and the tank side wall. The Multiple Reference Frame (MRF) model in FLUENT is based on the time-average approach, i.e. the actual flow is assumed to be faithfully represented by a time-averaged solution. This is true when there are no rotor-stator interactions (as in this case where the rotating and stationary parts are at a sufficient distance). Also, this model implies that the effect of the relative angle between different parts (e.g. between the baffles and the blades) is smoothed out due to time-averaging.

Now, regarding the setup of this boundary zone: first set the correct components in "Rotation–Axis Direction" (0,–1,0) and select "Multiple Reference Frame" in "Motion Type".

E Fluid	Fluid
Zone Name	Zone Name
fluid-rotating-bottom	fluid-rotating-bottom
Material Name paint	Material Name paint Edit
C Porous Zone	🗖 Porous Zone
🗖 Laminar Zone	🗖 Laminar Zone
Source Terms	🗖 Source Terms
🗖 Fixed Values	🗖 Fixed Values
Motion Porous Zone Reaction Source Terms Fixed Values	Motion Porous Zone Reaction Source Terms Fixed Values
Rotation-Axis Origin Rotation-Axis Direction	Motion Type Moving Reference Frame
	Rotational Velocity Translational Velocity
Y (m) 0 Y -1	Speed (rpm) 288 × (m/s) 8
	Υ (m/s) 🔒
Motion Type Moving Reference Frame	Z (m/s) 👩
OK Cancel Help	OK Cancel Help

Now, scrolling down there is an entry termed "Rotational Velocity" where the input is requested in [rad/s]. This can be changed in: **Define** > **Units...**

Juantities	Units	Set All to
acceleration angle	▲ rad/s deg/s	default
angular-velocity	rem	si
area area-inverse		british
ignition-energy concentration	Factor 0.1047198	cgs
contact-resistance crank-angle	Offset 0	
crank-angular-velocity	<u>•</u>	

where we select "angular velocity" in "Quantities" and then we set "rpm" in "Units" and press button **Close**. Automatically, the units back in the boundary conditions panels change to "rpm". Put value "200" for this rotating zone.

(ii) "fluid-rotating-bottom". Copy the values used in "fluid-rotating-bottom" by pressing button Copy in the "Boundary Conditions" panel. Then select "fluidrotating-bottom" at "From Zone" and "fluid-rotating-top" at the "To Zones".

🔁 Copy BCs	×
From Zone	To Zones 📃 😑
default-interior default-interior:020 default-interior:021 default-interior:022 default-interior:023 fluid-rotating-bottom	<mark>fluid-rotating-top</mark> fluid-stationary
fluid-rotating-top fluid-stationary	
Copy Close	Help

(iii) "fluid-stationary". Keep the defaults, except the components for the "Rotation-Axis Direction".

E Fluid
Zone Name
fluid-stationary
Material Name paint Edit
C Porous Zone
🗖 Laminar Zone
C Source Terms
Fixed Values
Motion Porous Zone Reaction Source Terms Fixed Values
Rotation-Axis Origin Rotation-Axis Direction X (m) g y Y (m) g y Z (m) g y Motion Type Stationary
OK Cancel Help

(iv) "per-bottom", "per-main" & "per-top": Select "Rotational" in "Periodic Type".

Periodic	×
Zone Name	
per-bottom	
	Periodic Type C Translational C Rotational
OK Cancel	Help

(v) Rotating walls ("wall-impeller-axial", "wall-impeller-radial", "wall-impeller-radialshadow", "wall-shaft:001" & wall-shaft:018"). Start with "wall-impeller-axial": Select "Moving Wall" in "Wall Motion", keep "Relative to Adjacent Cell Zone" and select "Rotational" in "Motion" and finally change the components of the "Rotation-Axis Direction".

Then copy the above settings to the rest of the list of rotating walls.

💶 Wall			×
Zone Name wall-impeller-axial			
Adjacent Cell Zone Fluid-rotating-top Momentum Thermal Radi Wall Motion C Stationary Wall		Speed (rpm)	
<u>(</u> •)	Fransiational Rotational Components	Rotation-Axis Origin × (m) 0 Y (m) 0 Z (m) 0	Rotation-Axis Direction
Shear Condition			
 No Slip Specified Shear Specularity Coefficient Marangoni Stress 			
Wall Roughness			
Roughness Height (m) 0 Roughness Constant 0.5	constant constant	• •	
	ОК	Cancel Help	

Boundary "wall-shaft", which also rotates, is setup differently, because its adjacent fluid zone is stationary. The difference lies in that the "Absolute" is selected in "Motion" and "200" in the "Speed [rpm]".

Wall			
one Name vall-shaft			
djacent Cell Zone Tuid-stationary			
Momentum Thermal Radiation Wall Motion Motion	Species DPM Multi	phase UDS	
C Stationary Wall C Relative Moving Wall	lational onal	Speed from Seed from Rotation-Axis Origin X (m) 0 Y (m) 0 Z (m) 0	Rotation-Axis Direction Y 0 Y -1 Z
Wall Roughness			
Roughness Height (m) Roughness Constant 0.5	constant constant	• •	
	OK Can	cel Help	

(vi) Stationary walls ("wall-baffle", "wall-tank-bottom", "wall-tank-side" & wall-tanktop").

Start with "wall-baffle": Select "Moving Wall" in "Wall Motion", change to "Absolute" and select "Rotational" in "Motion" and finally change the components of the "Rotation-Axis Direction".

Then copy the above settings to the rest of the list of stationary walls.

2 ₩all			
Zone Name wall-baffle			
djacent Cell Zone			
Fluid-stationary			
Momentum Thermal	Radiation Species DPM Multi	ohase UDS	
Wall Motion	Motion		
© Stationary Wall © Moving Wall	C Rolative to Adjacent Cell Zone	Speed (rpm) Ø	
		Rotation-Axis Origin	Rotation-Axis Direction
	© Rotational	× (m) 👔	- ×8
	C Components	Y (m) [g	
		Z (m) 👩	
Shear Condition			
 No Slip Specified Shear Specularity Coeffi Marangoni Stress 			
Wall Roughness			
Roughness Height (n	n) Ø constant	-	
Roughness Consta	nt Ø.5		
	OK Can	cel Help	

*** Now that the correct settings for the rotation are given, we must check again the

mesh: **Grid** > **Check**. The following listing should appear:

Grid Check				
Domain Extents: x–coordinate: min (m) = –1.400000e+000, max (m) = –4.072205e–017				
y–coordinate: min (m) = –5.000000e–001, max (m) = 2.220000e+000				
z–coordinate: min (m) = –1.212436e+000, max (m) = 1.212436e+000				
Volume statistics: minimum volume (m3): 3.073308e–008				
maximum volume (m3): 4.480489e–008				
total volume (m3): 5.069112e+000				
Checking periodic boundaries.				
Zone 6: average rotation angle (deg) = 120.000 (120.000 to 120.000) stored zone rotation angle (deg) = 120.000				
stored axis , (0.000000e+000, -1.000000e+000, 0.000000e+000)				
stored origin, (0.000000e+000, 0.000000e+000, 0.000000e+000)				
Zone 8: average rotation angle (deg) = 120.002 (120.000 to 120.000)				
stored zone rotation angle (deg) = 120.000				
stored axis , (0.000000e+000, -1.000000e+000, 0.000000e+000)				
stored origin, (0.000000e+000, 0.000000e+000, 0.000000e+000)				
Zone 10: average rotation angle (deg) = 120.002 (120.000 to 120.000)				
stored zone rotation angle (deg) = 120.000 stored axis , (0.000000e+000, –1.000000e+000, 0.000000e+000)				
stored origin, (0.000000e+000, 0.000000e+000, 0.000000e+000)				
Checking node count.				
Checking nosolve cell count.				
Checking nosolve face count.				
Checking face children.				
Checking cell children.				
Checking storage.				
Done. In the grey area, messages concerning periodic boundaries appear. We see that for all				

In the grey area, messages concerning periodic boundaries appear. We see that for all

three periodic boundaries, the correct rotationally periodic angle is reported (120°).

6th STEP: Solution

Solution Initialization: First, the solution must be initialised. Since we are solving in steady solver, the initial solution is merely a guess for the iteration process to start. However, the more realistic the guess, the less the required iterations until convergence.

Solve > Initialize > Initialize...

Solution Initialization	×
Compute From	Reference Frame
·	 Relative to Cell Zone Absolute
Initial Values	
Y Velocity (m/s)	0
Z Velocity (m/s)	0
Furbulent Kinetic Energy (m2/s2)	0.0001
Turbulent Dissipation Rate (m2/s3)	0.0001 •
Init Reset Apply	Close Help

Keep the defaults, except for "Turbulent Kinetic Energy [m2/s2]" and "Turbulent Dissipation Rate [m3/s2]", where low values is better to be used (see figure above). Press button **Init** and then button **Close**.

FMG Initialization: Next, the initial solution can be very much improved by using the Full Multi–Grid (FMG) initialization. This tool actually solves very fast on a coarse mesh the Euler (non–viscous) flow equations and then interpolates the results to the actual mesh, creating an excellent initial guess in many applications. The tool is activated through the Text User Interface (TUI). In the console window press <Enter> key and the following menu will appear:

>		
adapt/	file/	report/
define/	grid/	solve/
display/	parallel/	surface/
exit	plot/	view/

Then type:

> solve/initialize/fmg-initialization

and then answer yes in the following question:

Enable FMG initialization? [no] yes

The following listing will appear:

```
>> Reordering domain using Reverse Cuthill–McKee method:
   zones, cells, faces, done.
 Bandwidth reduction = 973/973 = 1.00
 Done.
Creating multigrid levels...
Grid Level 0: 102478 cells, 209484 faces, 20752 nodes; 3 clusters
Grid Level 1: 3017 cells, 65520 faces,
                                          0 nodes; 3 clusters
Grid Level 1: 3017 cells, 18712 faces,
                                          0 nodes
Grid Level 2:
               553 cells, 48035 faces,
                                         0 nodes; 3 clusters
Grid Level 2:
               553 cells, 4598 faces,
                                         0 nodes
               134 cells, 37938 faces,
Grid Level 3:
                                         0 nodes; 3 clusters
Grid Level 3:
               134 cells, 1244 faces,
                                         0 nodes
Grid Level 4:
                33 cells, 29376 faces,
                                         0 nodes; 3 clusters
Grid Level 4:
                33 cells, 286 faces,
                                        0 nodes
Grid Level 5:
                9 cells, 21730 faces,
                                        0 nodes; 3 clusters
Grid Level 5:
                9 cells,
                          66 faces.
                                      0 nodes
Done.
FMG: Converge FAS on level 5
FMG: Converge FAS on level 4
FMG: Converge FAS on level 3
FMG: Converge FAS on level 2
FMG: Converge FAS on level 1
0.->1.->2.->3.->4.->5.<<<<
FMG: Initialize flow for Segregated solution... end
>
```

View the initial solution: Display > Contours...



Select "Pressure" and "Static Pressure" in "Contours Of" and then select "wall" in "Surface Types" (automatically all wall boundaries will be selected in "Surfaces"). Manually deselect the surfaces "wall-tank-side" and "wall-tank-top" and de-

activate "Global Range" tick-box. Then check "Filled" check-box and increase to "50" the "Levels". Now if you press button **Display** the following graph will be shown. The flow field is not uniform as with standard initialization.



Figure 5: Display of the Initial Solution (Static Pressure Contours).

Now by visiting **Display** > Views... and by pressing button **Define...** in "Periodic Repeats" a new panel is activated. There if you press button **Set**

Views		X	💶 Graphics Periodi	icity	
/iews back bottom front isometric	Actions Default Auto Scale	Mirror Planes <u>≡</u> =	Cell Zones fluid-rotating-bo fluid-rotating-to fluid-stationary	ttom	Associated Surfaces
left right top	Previous Save Delete	Define Plane Periodic Repeats	Periodic Type C Translationa C Rotational	Axis Direction X (m) 0 Y (m) -1	Axis Origin X (m) 8 Y (m) 8
ave Name view-0	Read Write	Define	Angle (deg) 120 Number of Repe	Z (m) 🕫	Z (m) 🔋
Apply	Camera Clos	se Help	Set	Reset Defa	ault Close Helj

The display will change to the following:



Figure 6: Display of the Initial Solution in Periodic Repeat Mode.

> Set the residual monitoring: Plot > Residuals...

💶 Residual Mo	nitors				X
Options	Storage			Plotting	
Print	lte	rations 10	000	Wind	
	Normaliza	tion		lerations	100 🍦
		Normalize	🗹 Scale	Axes	Curves
	Convergen	ce Criterio	n		
	absolute		•		
Residual		heck onvergen	Absolute ce Criteria	-	
continuity			0.001		
x-velocity			0.001		
y-velocity			0.001		
z-velocity			0.001		
k			0.001	_	
OK Plot Renorm Cancel Help					

Activate "Plot" in "Options" and reduce to "100" the "Iterations" in "Plotting". Also de–activate "Check Convergence" tick–boxes.

Set solutions monitors:

(a) <mark>Solve</mark> >	Monitors > Surface
--------------------------	--------------------

Surface Monitors	Cefine Surface Monitor
Surface Monitors 1	Name Report of p-axial Pressure
Name Plot Print Write Every When	Report Type Static Pressure
p-axial V V 1 teration v Define	Area-Weighted Average Surface III
monitor-2	Iteration Plot Window Plot Plot Plot Plot Plot Plot Plot Plot
monitor-3	1 wall-impeller-axial wall-impeller-radial
monitor-4	File Name p-axial.out
OK Cancel Help	OK Curves Axes Cancel Help

Increase to 1 the number of "Surface Monitors" and rename it to "p-axial" in the "Name" entry. Then activate "Plot", "Print" and "Write" check–boxes and press button **Define**. This will open another panel where you should select "Area-Weighted Average", in "Report Type", "Pressure" and "Static Pressure" in "Report Of" and finally, "wall-impeller-axial" in "Surfaces".

(b) Solve > Monitors > Forces...

Force Monitors	Wall Zones wall-baffle	II A	Moment Center	Plot Window
✓ Plot ✓ Write ✓ Per Zone Coefficient Moment ✓	wall-impeller-axial wall-impeller-radial wall-impeller-radial-shac wall-shaft wall-shaft:001 wall-shaft:018 wall-tank-bottom	low T	Y (m) g Z (m) g About Y-Axis	Axes Curves
File Name				
cm-history				
Apply Plot Clear Close Help				

Select "Moment" in "Coefficient" and then activate "Plot", "Print" and "Write" check-boxes in "Options". Then select all rotating surfaces (see above figure) in "Wall Zones" and select "Y-Axis" in "About". This will monitor the moment coefficient on all rotating surfaces about the Y-axis, which is the axis of rotation.

Calculate the fluid volume: Report > Volume Integrals...

💶 Yolume Integrals		×
Report Type	Field Variable	Cell Zonec 🛛 🔳 🖃
• Volume	Pressure 🔻	fluid-stationary
O Sum O Maximum	Static Pressure	fluid-rotating-bottom fluid-rotating-top
O Minimum	Total Volume (m3)	
O Volume Integral	5.069112	1
O Volume-Average	1	
 Mass Integral Mass-Average 		
	Compute Write Close Help	

Select all three fluid regions in "Cell Zones" and press button **Compute** and the result 5.069112 [m3] is displayed. Note that this corresponds to the 1/3 of the geometry.

Save the case & data file as "mixing-tank_tet_stage0".

Solve for 100 iterations: Solve > Iterate...



This will indicate whether the initial solution was appropriate and there were no numerical stability problems. Indeed the solutions proceeds without problems, as shown by the dropping residuals.

Save the case & data file as "mixing-tank_tet_stage1".

> Set the solution controls to more appropriate settings:

Solve > **Controls** > **Solution...** and change to 0.6, 0.3, 0.5 & 0.5, the "Under-Relaxation Factors" of "Pressure", "Momentum", "Turbulent Kinetic Energy" and "Turbulent Dissipation Rate", respectively. These values (lower for momentum and higher for pressure) are well–suited for MRF, which takes a lot of iterations to converge. And also change to "PRESTO!" the "Discretization" for Pressure. This scheme is recommended for flows with swirl.

Solution Controls	Solution Controls
Equations I I Under-Relaxation Factors Flow Pressure 0.6 Density 1 Body Forces Momentum 0.3	Equations I I Under-Relaxation Factors Flow Turbulence Under-Relaxation Factors Turbulence Turbulent Kinetic Energy D Turbulent Dissipation Rate 0.5 Turbulent Viscosity 1
Pressure-Velocity Coupling SIMPLE Discretization Pressure PRESTO! Momentum First Order Upwind Turbulent Kinetic Energy First Order Upwind Turbulent Dissipation Rate First Order Upwind	Pressure-Velocity Coupling SIMPLE
OK Default Cancel Help	OK Default Cancel Help

Request another 500 iterations: **Solve > Iterate...**

🖳 Iterate	×
Iteration	
Number of Iterations 500	
Reporting Interval 1	
UDF Profile Update Interval 1	
Iterate Apply Close Help	

Save the case & data file as "mixing-tank_tet_stage2".

Now that the flow field is more-or-less settled near the solution, set the "Under-Relaxation Factor" of "Momentum" to 0.1 and solve for 1000 iterations. This may be considered as a final fine-tuning of the solution procedure. Previously with the higher value (0.3) FLUENT was able to reach quickly in a few iterations the solution. Now with the lower value (0.1) FLUENT will be able to actually get it (convergence).

Save the case & data file as "mixing-tank_tet_stage3".

Fig. 7 show the residual history, whereas Figs. 8 & 9, present the solution history of the surface monitor and Moment Coefficient, respectively.



Figure 7: Residual History After 1600 Iterations.



Figure 8 & 9: History of Surface Monitor and Moment Coefficient After 1600 Iterations.

It is evident that the solution may be considered converged. The picture would have been much better if we were solving the problem on a hex mesh.

7th STEP: Post-Processing The Results

<u>Pressure Contours</u>: Figs 10-13 show the pressure contours on axial impeller, radial impeller, tank walls and baffles, respectively: **Display** > Contours...



Figure 10 & 11: Static Pressure Contours on the Axial and Radial Impeller Walls.



Figure 12 & 13: Static Pressure Contours on the Tank Walls and Baffles.

Note that for each graph the "Global Range" in "Options" was de-activated.

Velocity Vectors: First create isosurfaces of y-value of 0.25 and 1.50 [m]. This will create two horizontal slices of the domain that correspond roughly at the heights of the radial and axial impeller, respectively. Surface > Iso-Surfaces...

	💶 Iso-Surface	×
	Surface of Constant	From Surface 📃 📃
	Grid	default-interior
4		default-interior:020
		default-interior:022
	Min (m) Max (m)	default-interior:023
	-0.5 2.22	fluid-rotating-bottom
	lso-Values (m)	From Zones
	0.25	fluid-rotating-bottom fluid-rotating-top
		fluid-stationary
	New Surface Name	· ·
	y=0.25m	
		1
	Create Compute Manag	ge Close Help

Select "Grid" and "Y-Coordinate" in "Surfaces Of Constant", press button **Compute**, set value 0.25 in "Iso-Values" and rename the "New Surface Name" to "y=0.25m". Repeat the same for y=1.50m. **Display** > **Vectors...**

Options	Vectors of		
C Hede Values	Velocity		
Global Pange	Color by		-
Clip to Range	Velocity Velocity Magnitude		-
Auto Scale Draw Grid			
I* Draw Grid	Min [m/s]	Masc [m/s]	
Style filled arrow	0.06354816	12.65583	
Scale 188	Surfaces		≡.
Skip 0	wall-shaft:018 wall-tank-botto wall-tank-side wall-tank-top	m	Î
Custom Vectors	y=0.25m y=1.50m		
Surface Name Pattern	Surface Types		
Match	axis clip-surf exhaust-fan fan		- 11v

Figs 14 shows the velocity vectors of absolute velocity in these two horizontal planes. It is clear that the predicted flow field corresponds to the configuration of Fig. 2.



Figure 14: Velocity Vectors at y=0.25 and 1.50 [m] Planes.

Velocity Contours: Figs 15 and 16 show the velocity contours for the radial and axial velocity components, respectively, again at the horizontal planes y=0.25 and 1.50 [m].



Figure 15 & 16: Contours of Radial and Axial Velocities at y=0.25 and 1.50 [m] Planes.

Path Lines: Figs 17 and 18 show the pathlines, initiated from planes y=0.25 and 1.50 [m], coloured with time (from release of the lath line) and twisted with turbulence intensity. For clarity reasons, not all path lines are shown and also they are tracked for a small number of steps. Display > Pathlines...



Figure 17 & 18: Pathlines Released from y=0.25 and 1.50 [m] Planes.

Motor Power: FLUENT can directly calculate the required motor power for the operation of the impellers. This power output refers only to the hydrodynamic (both pressure & viscous) losses. Mechanical and other losses should be added before the total power is calculated. Report > Forces...

Force Reports			X
Options • Forces • Moments • Center of Pressure Wall Name Pattern	Moment Center X (m) 0 Y (m) 0 Z (m) 0 Match	Wall Zones wall-baffle wall-impeller-axial wall-impeller-radial- wall-shaft wall-shaft:001 wall-shaft:018 wall-tank-bettom wall-tank-side wall-tank-side	shadow
Print	Write Clo	se Help	

Select "Moments" in "Options", set the "Moment Center" to (0,0,0) and select all rotating surfaces. Then press button **Print** and the following listing will appear in the TUI:

FLUENT [3d, pbns, rngke]			
<u>File Grid Define Solve Adapt Surface</u>	<u>D</u> isplay <u>P</u> lot <u>R</u> eport Parallel <u>H</u> elp		
wall-shaft:018	(-0.0045798309 0.00032025267 -74.751671)	(-0.0016063936 -0.00019482963 0.00051058445)	(-0.0061862245 0.00012542304 -74
net	(4037.8842 2716.9787 -1874.2672)	(7.93566 3.0621523 10.461347)	(4045.8199 2720.0409 -186
Moment Center: (0 0 0)			
zone name	pressure moment	viscous moment	total
	n-m	n-m	
wall-impeller-axial	(3238.3289 1030.7421 249.4359)	(7.5007539 2.4818172 7.2465267)	(3245.8296 1033.2239 256
wall-impeller-radial	(170.73857 322.45703 -161.91559)	(0.19638184 0.24819542 1.7291306)	(170.93495 322.70523 -160
wall-impeller-radial-shadow	(621.10028 1363.9073 525.90753	(0.27241266 0.33191228 1.49547)	(621.37269 1364.2393
wall-shaft	(7.7150536 -0.12831965 -2046.5723)	(-0.053853389 -0.00019852033 -0.0097654294)	(7.6612002 -0.12851817 -20
wall-shaft:001	(0.0060274508 0.00025280379 -366.37109)	(0.02157145 0.00062067853 -0.00052551838)	(0.027598901 0.00087348232 -366
wall-shaft:018	(-0.0045798309 0.00032025267 -74.751671)	(-0.0016063936 -0.00019482963 0.00051058445)	(-0.0061862245 0.00012542304 -74
net	(4037.8842 2716.9787 -1874.2672)	(7.93566 3.0621523 10.461347)	(4045.8199 2720.0409 -1)6
(Þ

All moments (along x-, y- and z-directions) for all wall zones and for both viscous and pressure effects, are displayed. The total Torque in the rotation axis is calculate to 2720 [Nm], hence the required power is $P=(T)x(\omega)=(2720)x(200/60)=9.07$ [kW].

Save (again) the case & data file as "mixing-tank_tet_stage3".

8th STEP: Calculating Mixing Time

The idea is to create a second, but identical to the first, liquid material that will be used as a tracer. We will put set a small volume inside the tank (a sphere of 0.10 [m]) fully filled with the tracer and we will monitor it's spreading inside the tank, hence we will be able to calculate the blending (mixing) time.

The mixing time, t_{99} [s], is defined as the time required for the concentration of the tracer to reach the 99% of it's infinite (corresponding to complete mixing) value, i.e.: $U=C(t)/C_{\infty}$. So, if the local concentration values are employed for C(t), we may evaluate

the blending time locally, in certain regions of the tank. If the average value of many locations is used for C(t), then the overall blending time can be calculated as t_{99} .

Activate the Unsteady Solver: Define > Models > Solver...



Select "Unsteady" in "Time" and keep the default "1st-Order Implicit" in "Unsteady Formulation".

> Activate the Species Transport Model:

Define > Models > Models > Transport & Reaction...

Model	Mixture Properties
 Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport 	Mixture Material mixture-template Number of Volumetric Species 2
Reactions C Volumetric	
Options Inlet Diffusion Diffusion Energy Source Full Multicomponent Diffusion Thermal Diffusion	

Select "Species Transport" in "Model". The panel will expand and will look like above. Keep the defaults close the panel. You will see some warning in the TUI, regarding the materials and the following message:

Informat	ion	x
į)	Available material properties or methods have changed Please confirm the property values before continuing.	

> Modify the existing liquid and add a new one: Define > Materials...

Choose "mixture" in the category "Material Type" and this will lead you to the following panel. Change the name in "Name" to "liquids", choose "volume-weighted-mixing-law" in "Density [kg/m3]" and leave "constant" in "Viscosity [kg/m-s]" and put a value of "0.005". Do not change Thermal Conductivity [w/m-k]" and "Cp [j/kg-k]", as in the next step energy will be de-activated (it was activated when we turned on the Species Transport model, because by default it uses Incompressible Ideal Gas Law for the density estimation).

Materials			×
Name liquids	Material Type mixture	•	Order Materials By Name
Chemical Formula	Fluent Mixture Materials		C Chemical Formula
	liquids	•	Fluent Database
	Mixture		User-Defined Database
Properties	none	<u>v</u>	
Mixture Species names	▼ Edit hted-mixing-law ▼ 5 dit		
Viscosity (kg/m-s) constant	Edit		
Mass Diffusivity (m2/s) constant-dilu 2.88e-05	ite-appx <u>v</u> Edit		
Change/Create	Delete Close	Hel	p

Press button **Change/Create** and press **Yes** in the following dialogue panel in order to store the above changes.



> De-activate energy calculation: **Define** > **Models Energy**...

💶 Energy		×		
Energy				
Evergy Equation				
I	·····	 1		
ОК	Cancel	Help		

Uncheck the check-box and close this panel.

Return to Materials panel: Define > Materials...

Change "Material Type" to "fluid", select "paint", change "Molecular Weight [kg/kgmol]" to "1" and press button Change/Create. Then change it's name to

"tracer" and press button Change/Create. FLUENT will ask you the following questions, to which you should press button No.

Y	inge/Create mixiure a	nd Overwrite paint?	
Materials			×
Name	Material Type		Order Materials By
paint	fluid	•	Name
Chemical Formula	Fluent Fluid Materials		Chemical Formula
	paint	~	Fluent Database
	Mixture		User-Defined Database
	none	7	
Properties			
Density (kg/m3) consta 950 Viscosity (kg/m-s) consta		Edit	
0.005 Molecular Weight (kg/kgmol) constant 1	ıt	▼ Edit	
Change/Create	Delete	Close He	p

We have created the tracer liquid, but now we must define these two ("paint" and "tracer") as the only two components of the mixture.

Select in "Material Type" the category "mixture" and this time press button **Edit** next to the entry "Mixture Species" that has the value of "names". This will lead you to the following panel:

Name Material Type Order Materials By [Liquids mixture © Name Chemical Formula Fliquids Chemical Formula	Sciected Species
C Chemical Formula	Sciected Species
Chemical Formula Stread Line Material International Control Co	Sciected Species
liquids - Fluent Database water-vapor [h20]	tracer
Mixtore: User-Defined Database (oxygen [o2]	paint
none nitrogen (n2)	
Properties	
Mixture Species names	
naures Cult	
	Add Remove
Density (kg/m3) volume-weighted-mixing-law Editor Editor Editor	Selected Solid Species
	5000000 0000 00000
Viscosity (kg/m-s) constant	
0.005	
Mass Diffusivity (m2/s) constant dilute appx Edit	
2.88e-05 Add Remove	Add Remove
Change/Create Delete Close Help OK C	ancel Help

Select from the list on the left "tracer" and press button "Add" and then do the same for "paint". Then select from the right list (one at a time) "h2o", "o2" and "n2" and press button "Remove". The final result should be as above (make sure that "tracer" is on top. Press button **OK** to return to materials panel. Select again in "Material Type" the category "fluid" and select successively "h2o", "n2" and "o2" and press the button "Delete". These are the default mixture gases, which are not needed in this case.

> Patch a mass fraction of unity for the tracer in a small volume.

Adapt > Region...

Select "Sphere" in "Shapes" and then put the following data for the sphere centre and radius. Then press button Mark and FLUENT will report that 27 cells were marked.

Region Adaption				
Options	Input Coordinates			
InsideOutside	X Center (m) X Max (m)	-		
Shapes O Hex	Y Center (m) Y Max (m) Ø Ø			
Cylinder	Z Center [m] Z Max [m] Ø Ø			
Manage Controls	Radius (n) 9.1			
	Select Points with Mous	e		
Adapt	Mark Close Help			

Now with **Solve** > **Initialize** > **Patch...**

💶 Patch		X
Reference Frame C Relative to Cell Zone C Absolute Variable X Velocity Y Velocity Z Velocity Turbulent Kinetic Energy Turbulent Dissipation Rat tracer	Value 1 Use Field Function Field Function	Zones to Patch = = fluid-rotating-bottom fluid-rotating-top fluid-stationary Registers to Patch = = Sphere-r1
	Patch Close Help	

Select "tracer" in "variable", set "1" in "Value", select "sphere-r1" in "Registers to Patch" and press button **Patch**. The above actions result in a value of 1 for the mass fraction of tracer in this bunch (register) of 27 elements. In the rest of the domain the mass fraction of the tracer is zero. Note that this register of elements does not form a perfect sphere, rather it is an edgy shape (by defining the sphere FLUENT selects those elements that lie partly or totally inside this spherical volume). Therefore, register's volume cannot be calculated by geometric formulae. In order to find out how much tracer mass is contained inside the domain, we use volume integrals: **Report > Volume Integrals...**

💶 Volume Integrals			×
Report Type Volume Maximum Minimum Volume Integral Volume-Average Mass Integral Mass-Average	Field Variable Species Mass fraction of tracer Volume-Weighted Average Ø. 0008274153	<u> </u>	ell Zones E = iuid-stationary luid-rotating-bottor luid-rotating-top
	Compute Write Close Help	•	

Select "Volume-Average" in "Report Type", "Species..." and "Mass fraction of tracer" in "Field Variable", select all "Cell Zones" and press button **Compute**. The result is **0.0008274153 [kg_tracer/kg_mixture]**. The total mixture mass (actually the 1/3 of the whole tank) is equal to the product of tank volume (1/3 of the tank) and mixture density (the latter being 950 [kg/m³], since both of its components have the same density), i.e.

M_{tank}=(5.069112 [m³])x(950 [kg_mixture/m³])=4815.656 [kg_mixture].

So the total tracer mass can be indirectly calculated by the product of tankaverage mass fraction and total mass,

M_{tracer}=(0.0008274153[kg_tracer/kg_mixture])x(4815.656[kg_mixture])=3.9845[kg_tracer]. Hence, the infinite value of concentration is equal to:

```
C<sub>∞</sub>=(3.9845 [kg_tracer]) / (5.069112 [m<sup>3</sup>])=0.786045 [kg_tracer/m<sup>3</sup>]
```

and this is the reference value to estimate mixing time, t_{99} .

> Create a Custom Field Function for U variable:

Define > Custom Field Functions...

💶 Custo	m Field F	unction C	alculator				X
Definiti	DN						
concer	ntratio	n-trac	er / .7	86045			
+	-	X	1	y'x	ABS	Select Operand Field Functions from	
INV	sin	cos	tan	In	log10	Field Functions	
0	1	2	3	4	SQRT	Species	
5	6	7	8	9	CE/C	Molar Concentration of tracer	
()	PI	е		DEL	Select	
New Fu	nction N	lame 🚺	value		>		
			Define	Ma	nage	Close Help	

Select "Species..." and "Molar Concentration of tracer" (FLUENT doesn't store mass concentrations, but as we put unity for the molecular weight, then it's identical to molecular concentration) and press button Select, then press button and then the relevant buttons to type the number 0.786045. When you are finished, it should look like the above figure. Now type "u-value" in "New Function Name"

and press button **Define**. This new variable will be now available to all postprocessing operations.

Create the sampling points: Surface > Point...

Point Surface	X
Options Point Tool Reset	Coordinates ×0 (m) -1 y0 (n) 8 z0 (m) 8
Se	elect Point with Mouse
New Surface Na	me
point-tracer	
Create	anage Close Help

Create the following points, by giving their coordinates and typing their names in "New Surface Name" before pressing button **Create**. See with **Display** > **Grid...** where these point are located.

Name	x0 [m]	y0 [m]	z0 [m]
point-tracer	-1.0	0.0	0.0
point-bottom	-0.000001	-0.25	0.0
point-baffle	-1.2	1.0	0.0
point-middle	-0.5	1.0	0.0
point-top	-0.1	2.15	0.0

Activate the monitoring of u-value at all points, including their average value.
 Solve > Monitors > Surface ...

Surface Monitors 6	Name Report of u-all Custom Field Functions
u-baffle I I Image: Step Image: St	Report Type urvalue Facet Average Surfaces X Axis Sourfaces Flow Time point-battle Plat Window point-battle 2 y File Name u-all.out

You will change the settings for the first monitor. Also do not forget to de-activate the Force monitor for moment coefficient that was set in page 14; both monitors are not required anymore. Increase to 6 the number of "Surface Monitors" and rename them to "u-tracer", "u-bottom", "u-baffle", "u-middle", "u-top" and "u-all". Activate "Print" and "Write" check-boxes for all six monitors and "Plot" only for "u-all" and select "Time Step" in "When". Then press button **Define** sequentially for each monitor and there select "Facet Average" in "Report Type", "Flow Time" in "X Axis", "Custom Field Functions..." and "u-value" in "Report Of" and finally select the appropriate surfaces in the "Surfaces" list.

Activate convergence check for all the residuals and decrease the convergence criterion for species to 10⁻⁶. Plot > Residuals ...

💶 Residual Mo	nitors			X
Options	Storage			Plotting
Print Plot	Ite	rations	1000 单	Window 🛛 🍝
	Normaliza	tion		Iterations 100
		Normaliz	e 🗹 Scale	Axes Curves
	Converger	ice Criter	ion	
	absolute		•	
x-velocity			0.001	-
y-velocity			0.001	
z-velocity		•	0.001	
k	9	R	0.001	
epsilon	v		0.001	
tracer	N		0.0000	1)-
01	< рі	ot R	enorm Ca	ncel Help

- > Save the case & data file as "mixing-tank_tet_stage4_0.0s".
- > Deactivate the solution of all the flow & turbulence equations:

Solve > Controls > Solution...

Solution Controls	×
Equations 📃 🖃	Under-Relaxation Factors
Flow	Turbulent Kinetic Energy 0.5
Torbulence tracer	Turbulent Dissipation Rate 8.5
	Turbulent Viscosity 1
	tracer 1
Pressure-Velocity Coupling	Discretization
SIMPLE	Pressure PRESTO!
	Momentum First Order Upwind 👻
	Turbulent Kinetic Energy First Order Upwind 👻
	Turbulent Dissipation Rate First Order Upwind
0K	Default Cancel Help

Flow and turbulence field are converged and only the tracer concentration needs to be tracked in time.

> Set the time step size to 0.01 [s] and request 20 time steps. Solve > Iterate...

💶 Iterate 🛛 🔀
Time
Time Step Size (s) 0.01
Number of Time Steps 20
Time Stepping Method
• Fixed
C Adaptive
C Variable
Options
Data Sampling for Time Statistics
Iteration
Max Iterations per Time Step 20
Reporting Interval 1
UDF Profile Update Interval
Iterate Apply Close Help

- > Save the case & data file as "mixing-tank_tet_stage4_0.2s".
- > Set the time step size to 0.05 [s] and request 6 time steps. Solve > Iterate...

<u> </u>		
Time		
Time Step Size (s) 0.05		
Number of Time Steps 6		
Time Stepping Method		
Fixed		
C Adaptive		
C Variable		
Options		
Data Sampling for Time Statistics		
Iteration		
Max Iterations per Time Step 20		
Reporting Interval 1		
UDF Profile Update Interval 1		
Iterate Apply Close Help		

- > Save the case & data file as "mixing-tank_tet_stage4_0.5s".
- Set the time step size to 0.1 [s] and request 105 time steps. Solve > Iterate...

-		
Time		
Time Step Size (s) 0.1		
Number of Time Steps 5		
Time Otensies Mathed		
Time Stepping Method		
• Fixed		
C Adaptive		
C Variable		
Options		
Data Sampling for Time Statistics		
r Data Sampling for thire Statistics		
Iteration		
Max Iterations per Time Step 20		
Reporting Interval 1		
UDF Profile Update Interval 1		
,		
Iterate Apply Close Help		

Save the case & data file as "mixing-tank_tet_stage4_11.0s".

Figure 19 shows the time history of U for the 5 sampling locations. Depending on the location, the blending time is different, however more or less in all point blending is achieved after 10 [s].

Note that if the tracer was released from a location with lower velocity magnitude, e.g. below the radial impeller, near the top wall, etc., the blending time will be higher. This is a good exercise for the user who wants to further practice with mixing applications. So, if a substance is indeed being added to the bulk liquid, simulations with different release locations will indicate the best position (obviously a location with high velocity).



Figure 19: Time History of U Variable at the Sampling Points.