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^o, whereas the blades of the radial impeller are perpendicular.

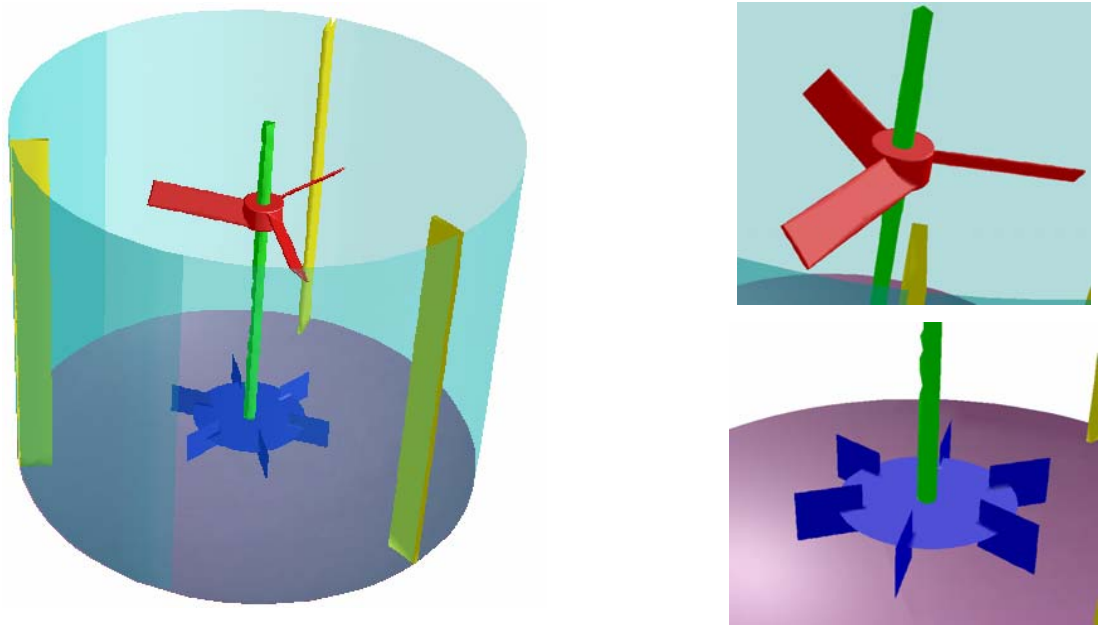
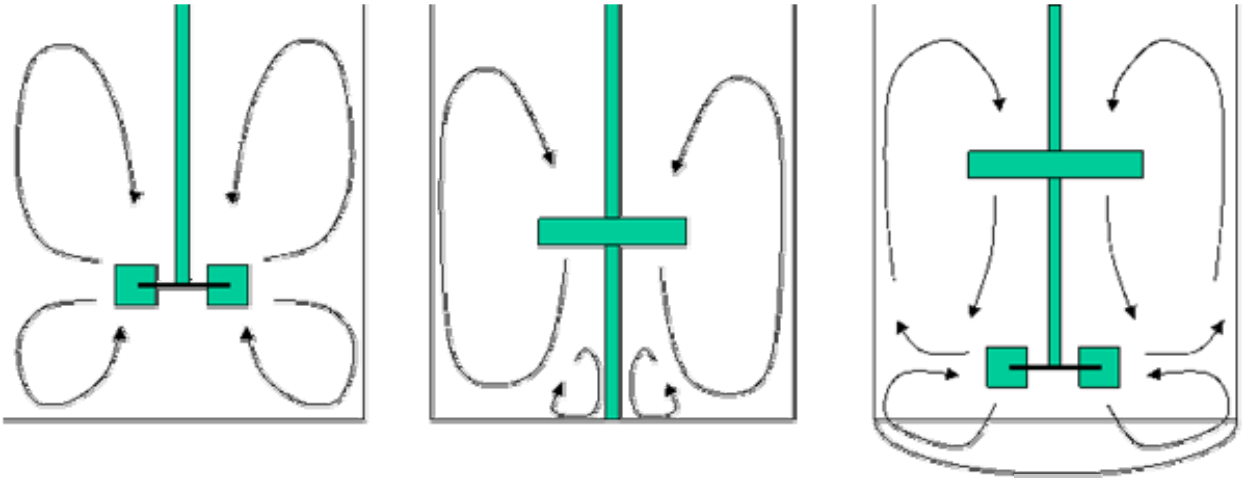


Figure 1: Geometry of the Mixing Tank.

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.



(a) 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).

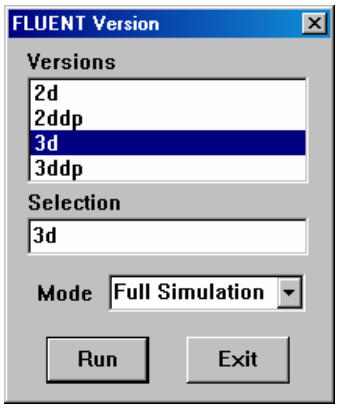
The tank also exhibits three baffles of width W=20 [cm] and thickness s=3.5 [cm], covering the whole length of the cylindrical part of the tank. Baffles enhance mixing, however numerically are the reason that impose the simultaneous employment of at least two reference frames: (a) a rotating and (b) 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/m^3]) 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 [m^3] (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:



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**

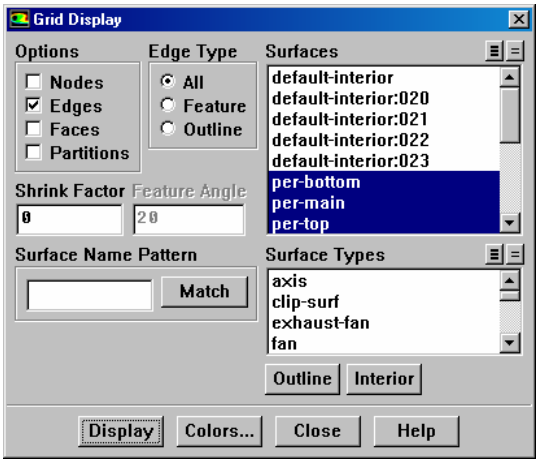
In addition to checking for error messages and warnings, check that the minimum volume is positive.

- 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:



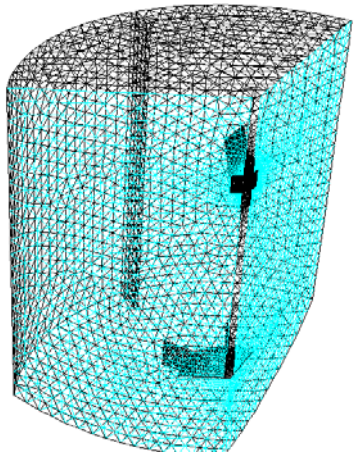
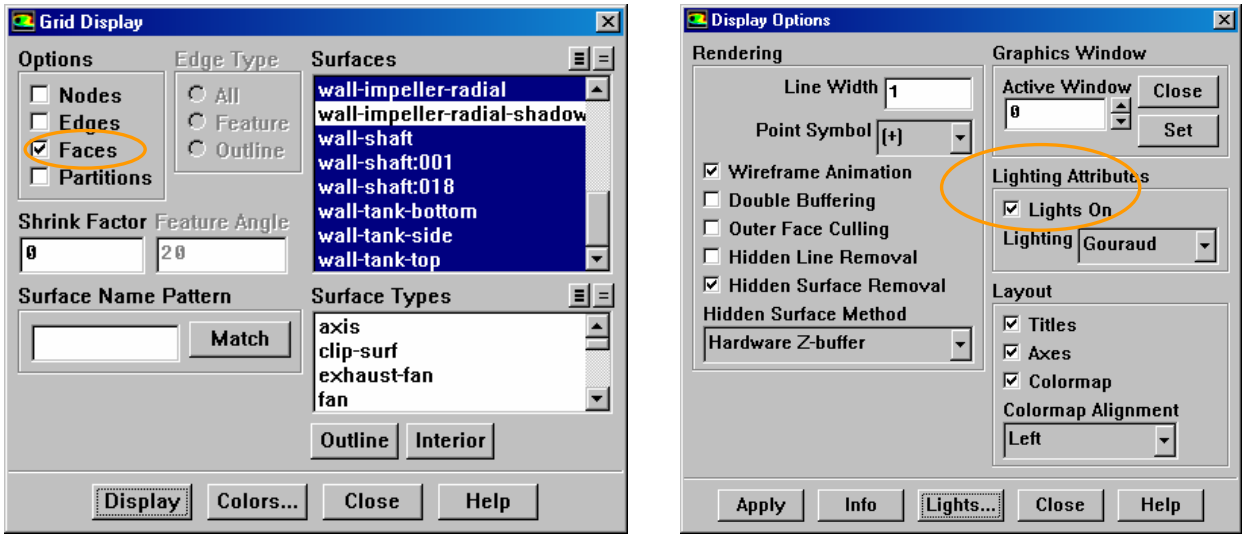


Figure 3: Display of the Mesh ("Edges" Mode).

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.

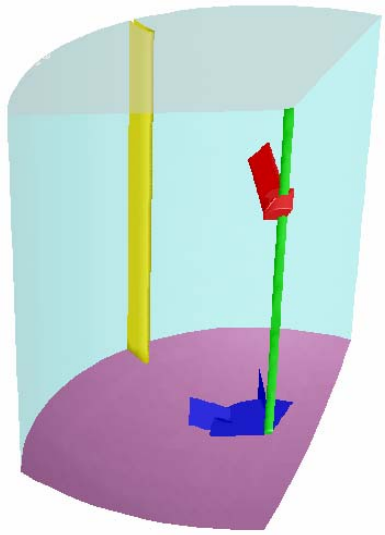


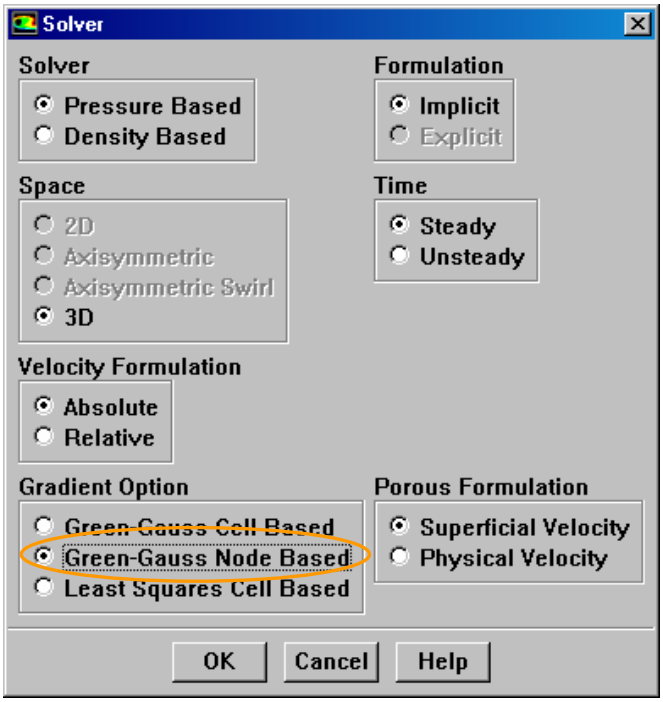
Figure 4: Display of the Mesh ("Faces" Mode).

2nd STEP: Models

- Activate node based gradient calculation in the steady solver:

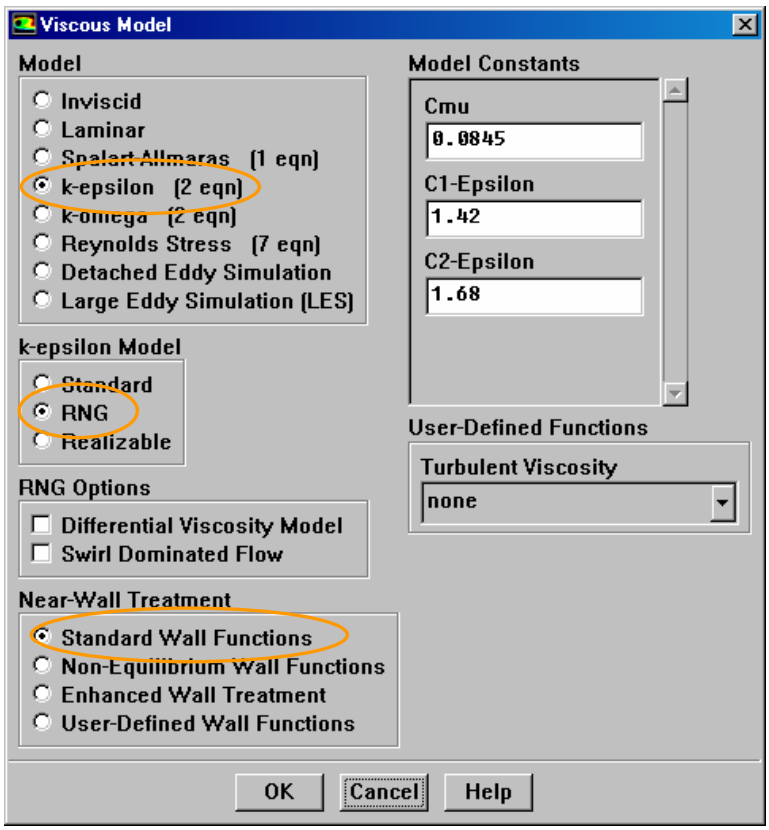
Define > Models > Solver…

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



- 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…

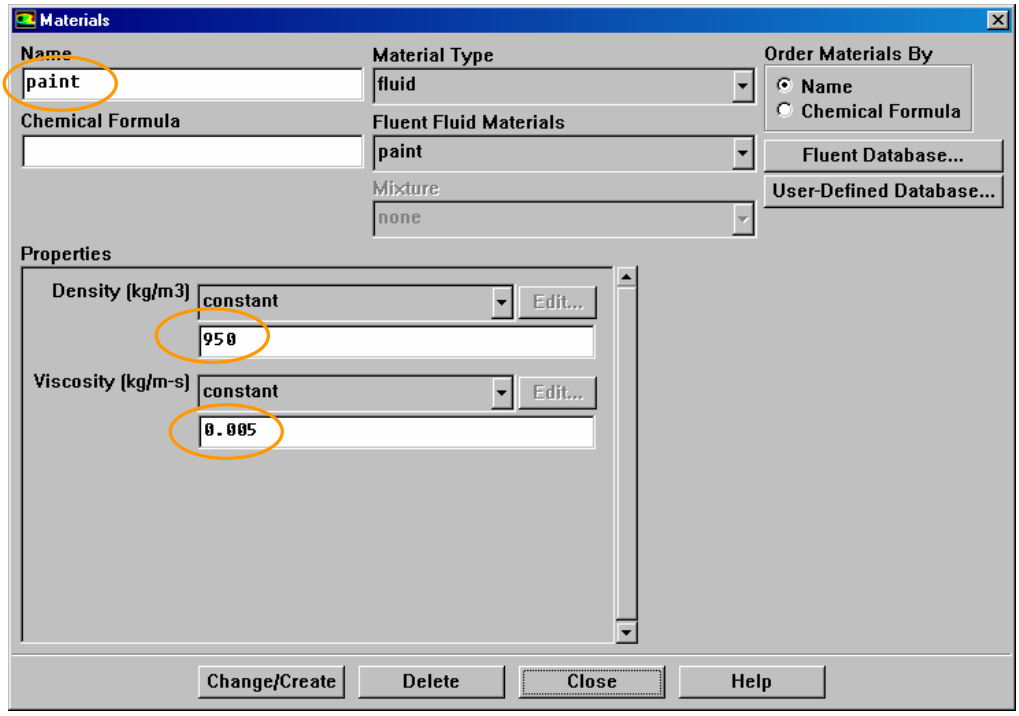


3rd STEP: Materials

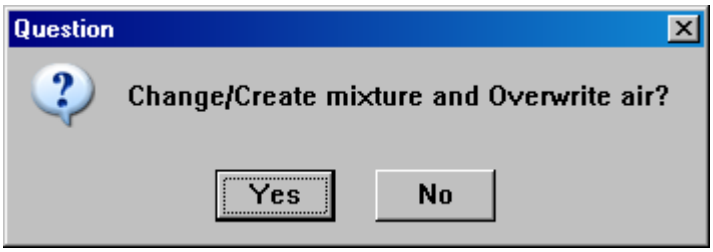
- Define the operating liquid:

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".



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



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

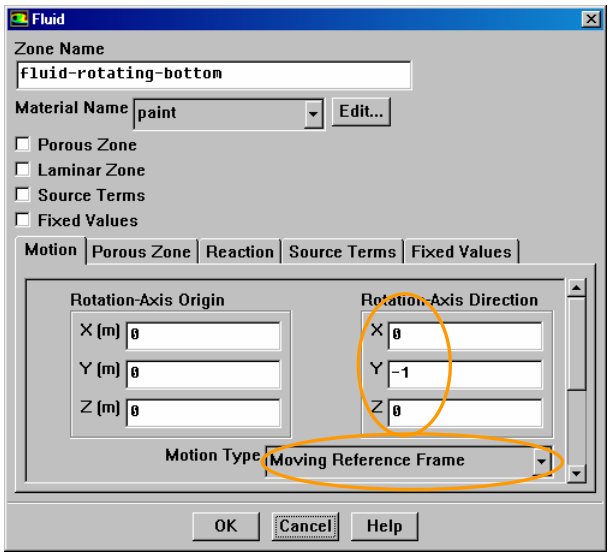
- In the boundary conditions we must input data at all domain boundaries.

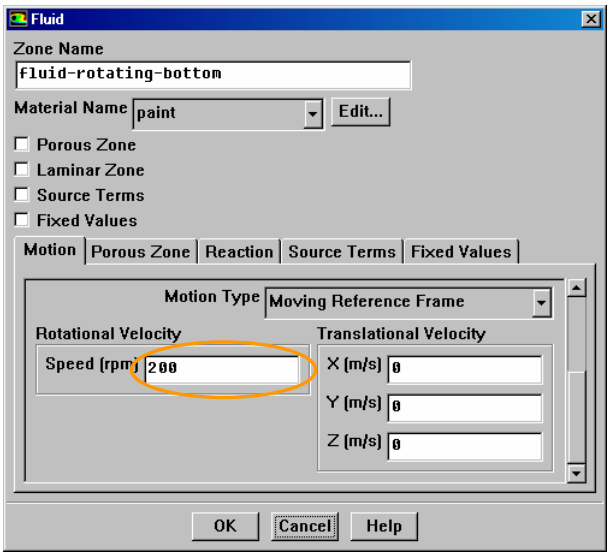
Define > Boundary Conditions… Choose the boundary zones at "Zone" on the left and then press button **Set** to edit/modify the values of the boundary conditions.

(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–

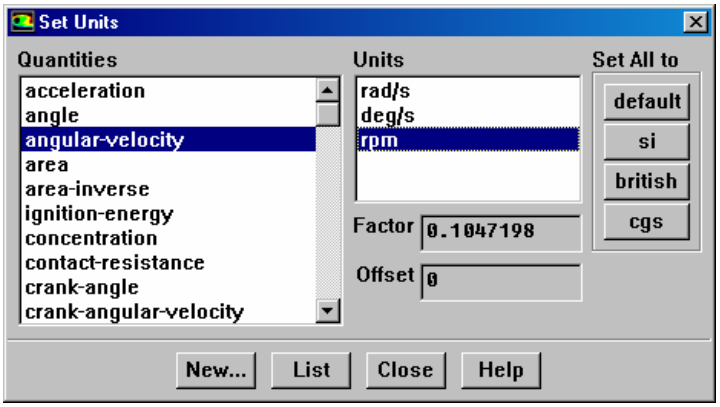
tank_tet.dbs"); it's the 1/3 of a cylindrical volume that has an axis identical to the axis of rotation and totally surrounds the bottom impeller. It's boundaries lie between the impeller tips and the tank side wall. The Multiple Reference Frame (MRF) model in FLUENT is based on the time–average approach, 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".



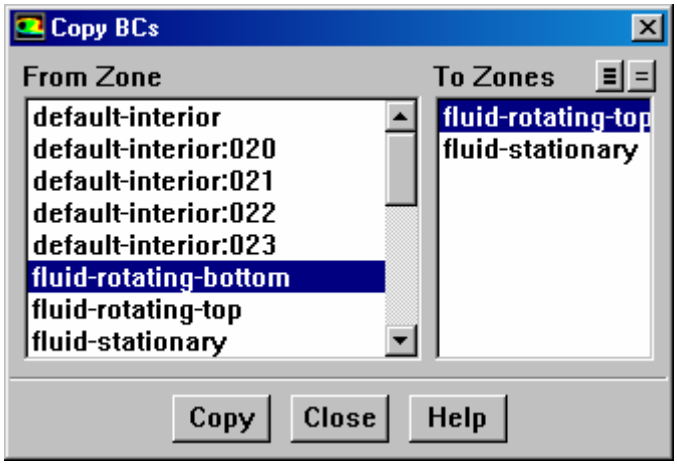


Now, scrolling down there is an entry termed "Rotational Velocity" where the input is requested in [rad/s]. This can be changed in: **Define > Units…**

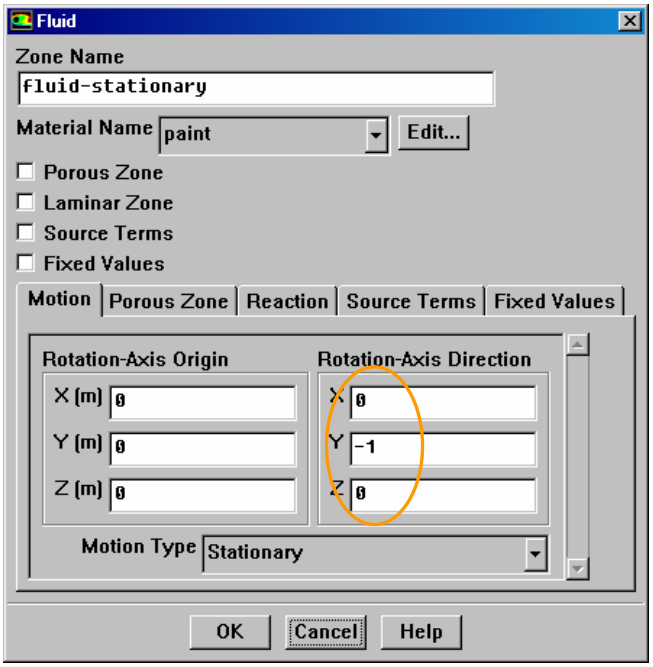


where we select "angular velocity" in "Quantities" and then we set "rpm" in "Units" and press button **Close**. Automatically, the units back in the boundary conditions panels change to "rpm". Put value "200" for this rotating zone.

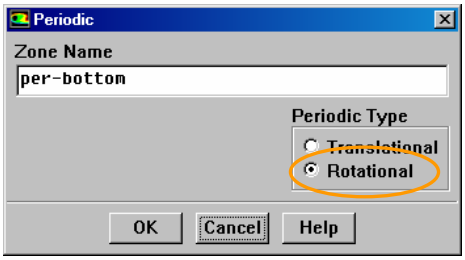
(ii) "fluid–rotating–bottom". Copy the values used in "fluid–rotating–bottom" by pressing button **Copy** in the "Boundary Conditions" panel. Then select "fluid–rotating–bottom" at "From Zone" and "fluid–rotating–top" at the "To Zones".



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



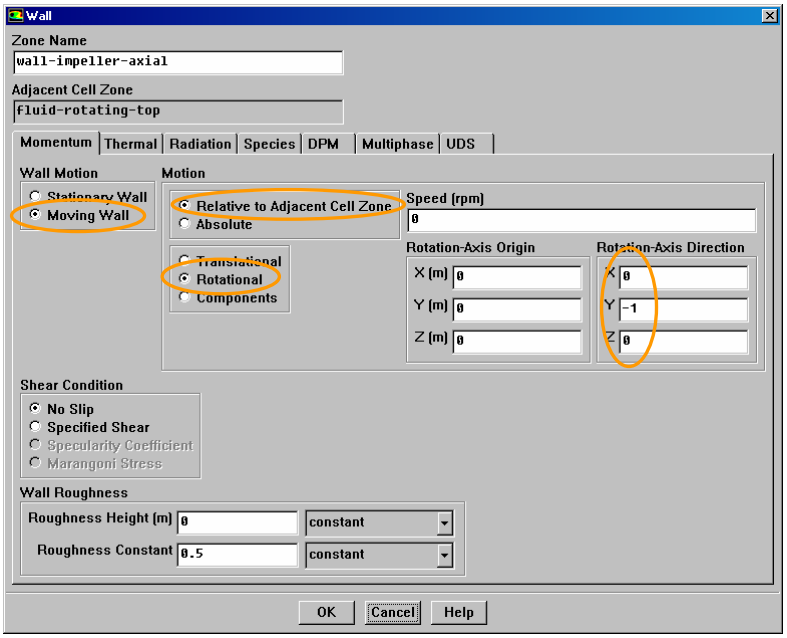
(iv) "per–bottom", "per–main" & "per–top": Select "Rotational" in "Periodic Type".



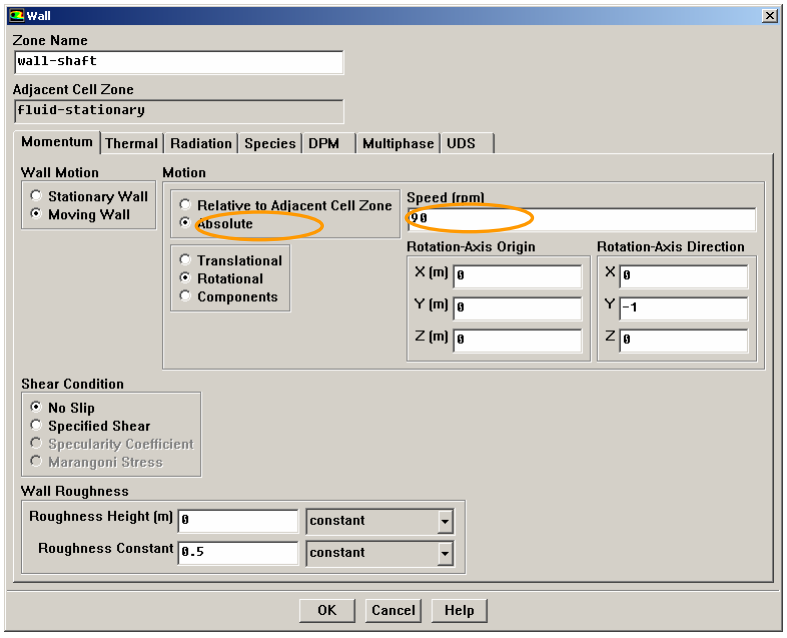
(v) Rotating walls ("wall–impeller–axial", "wall–impeller–radial", "wall–impeller–radial–shadow", "wall–shaft:001" & wall–shaft:018").

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.



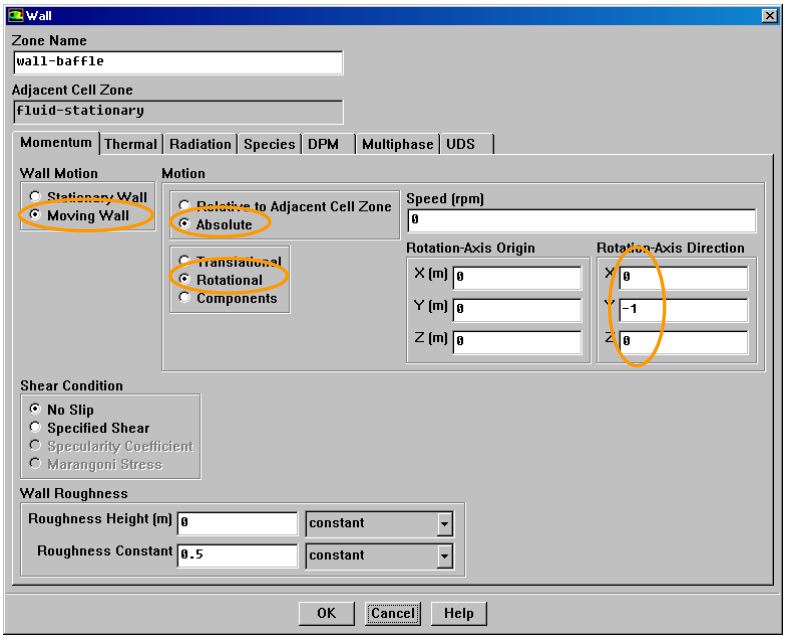
Boundary "wall–shaft", which also rotates, is setup differently, because its adjacent fluid zone is stationary. The difference lies in that the "Absolute" is selected in "Motion" and "200" in the "Speed [rpm]".



(vi) Stationary walls ("wall–baffle", "wall–tank–bottom", "wall–tank–side" & wall–tank–top").

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.



*** Now that the correct settings for the rotation are given, we must check again the mesh: **Grid > Check**. The following listing should appear:

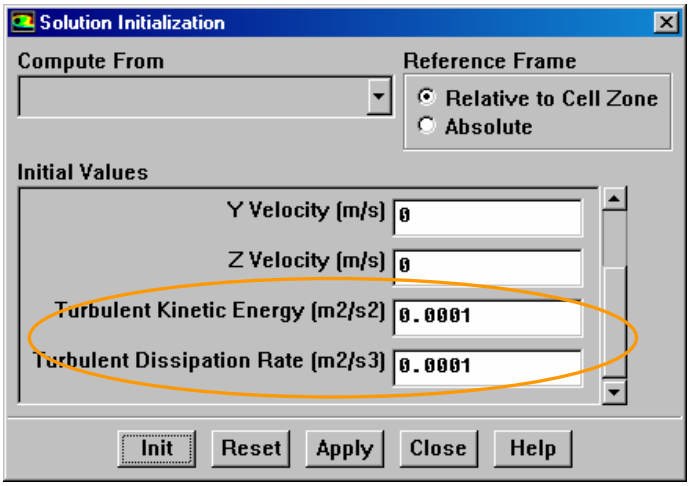
```
Grid Check

 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–004
    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 three periodic boundaries, the correct rotationally periodic angle is reported (120^{o}).

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…



  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

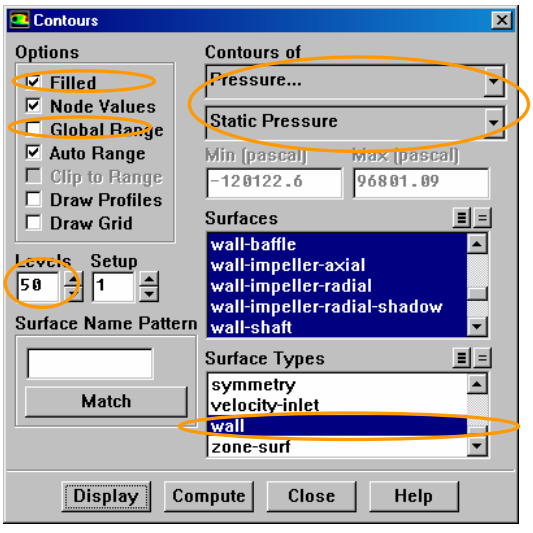
  and then answer yes in the following question:

  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
 Grid Level  3:    134 cells,  37938 faces,      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.

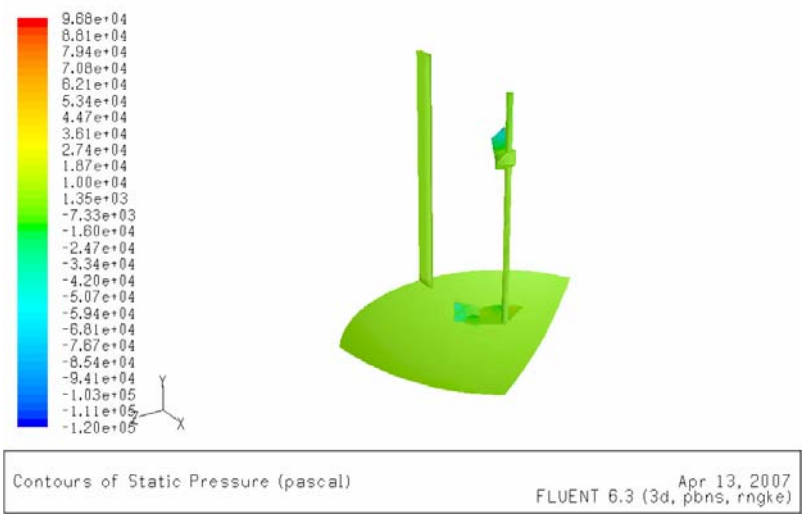
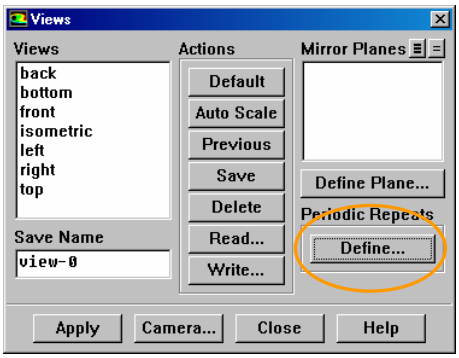
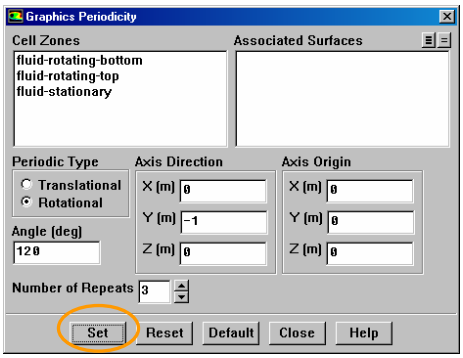


Figure 5: Display of the Initial Solution (Static Pressure Contours).

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





The display will change to the following:

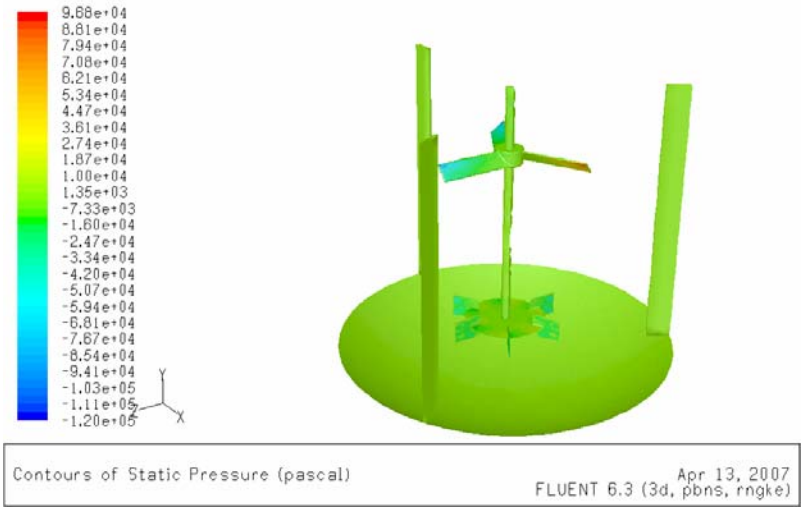
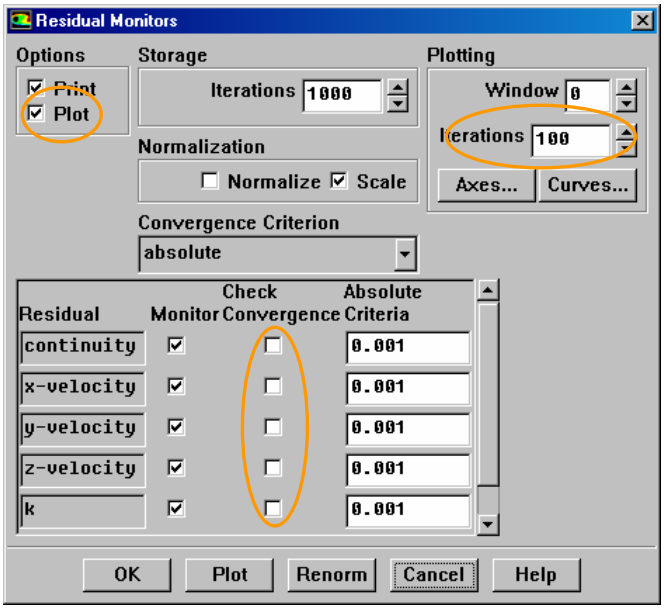


Figure 6: Display of the Initial Solution in Periodic Repeat Mode.

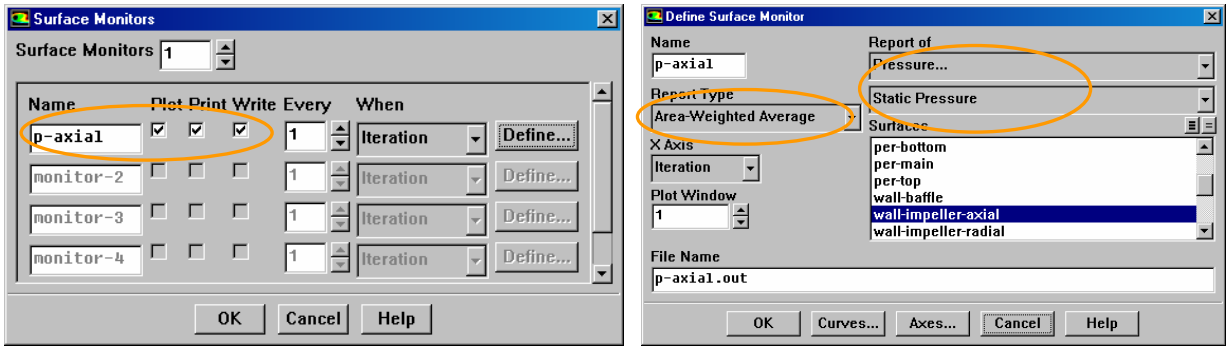
➢ Set the residual monitoring: **Plot > Residuals…**



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

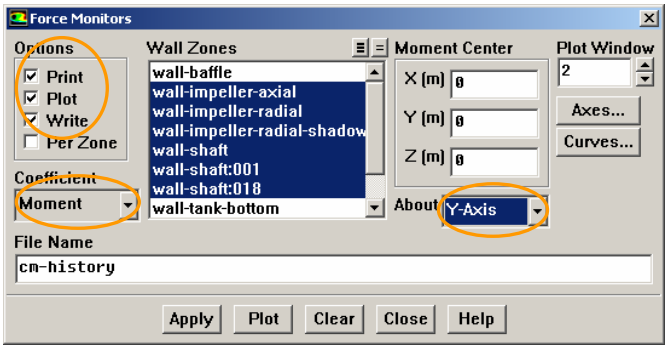
➢ Set solutions monitors:

(a) **Solve > Monitors > Surface …**



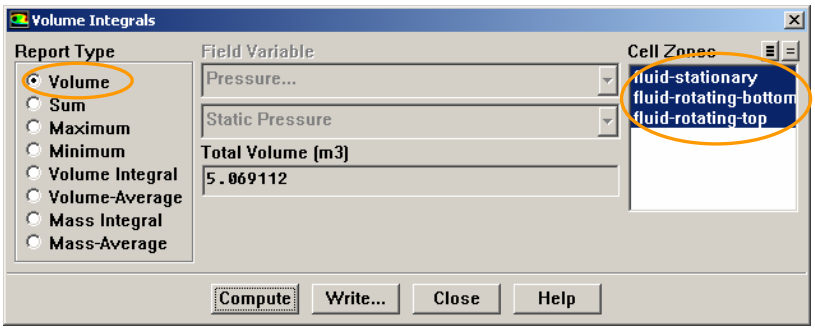
Increase to 1 the number of "Surface Monitors" and rename it to "p-axial" in the "Name" entry. Then activate "Plot", "Print" and "Write" check–boxes and press button **Define**. This will open another panel where you should select "Area-Weighted Average", in "Report Type", "Pressure" and "Static Pressure"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

(b) **Solve > Monitors > Forces…**



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.

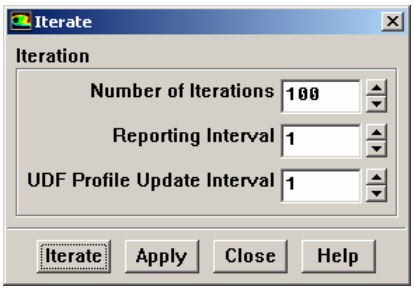
- Calculate the fluid volume: **Report > Volume Integrals…**



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…**

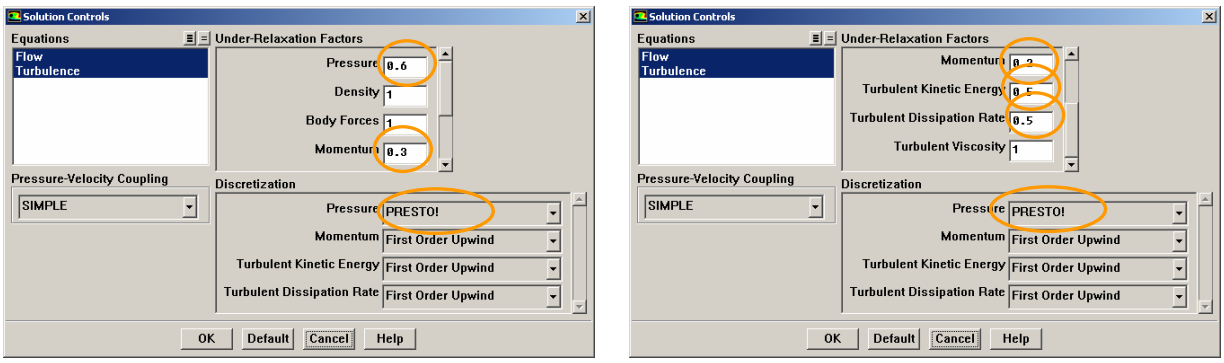


This will indicate whether the initial solution was appropriate and there were no numerical stability problems. Indeed the solutions proceeds without problems, as shown by the dropping residuals.

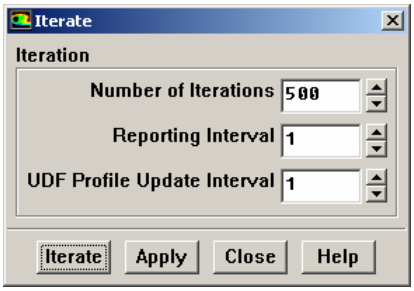
Save the case & data file as "mixing-tank_tet_stage1".

- 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.



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



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.

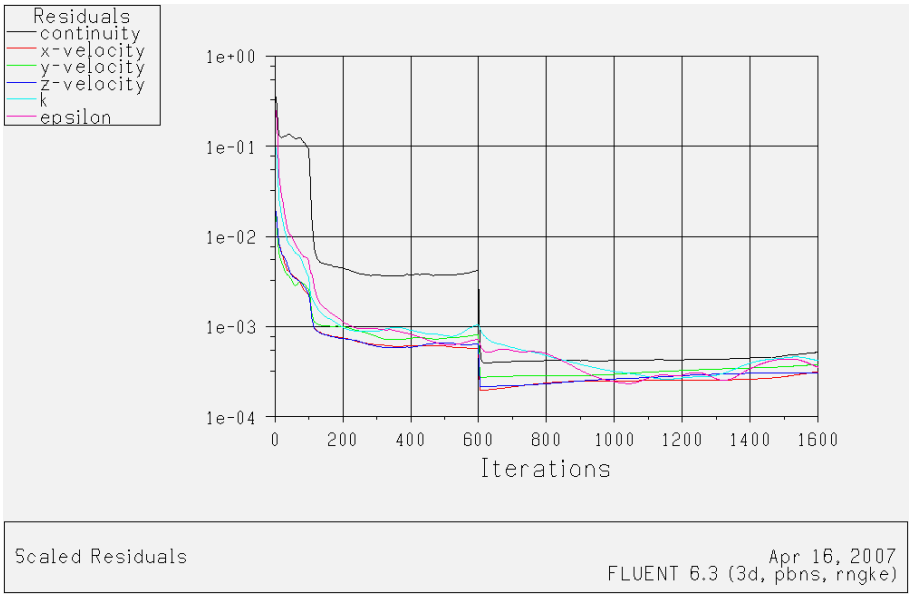


Figure 7: Residual History After 1600 Iterations.

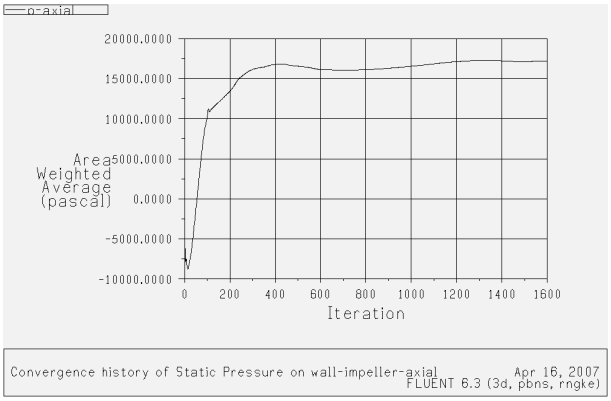


Figure 8 & 9: History of Surface Monitor and Moment Coefficient After 1600 Iterations.

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

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

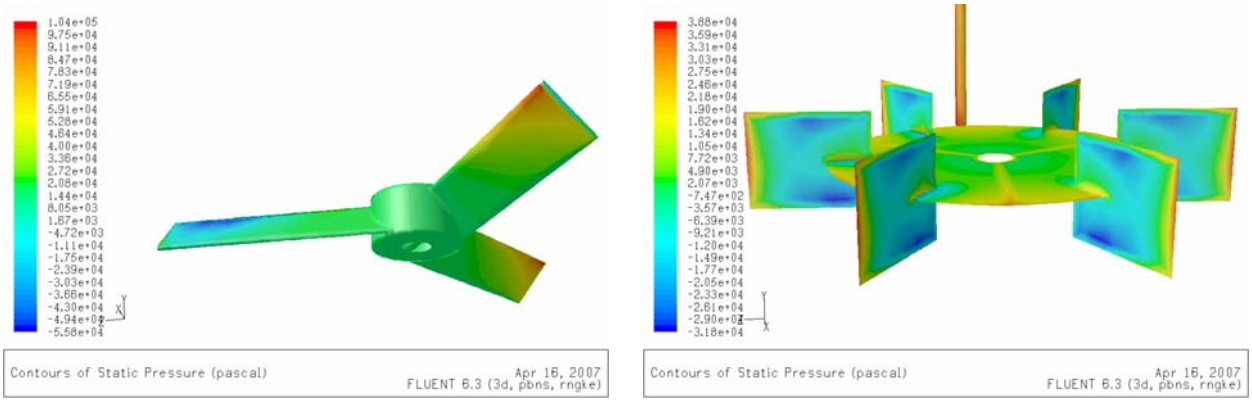


Figure 10 & 11: Static Pressure Contours on the Axial and Radial Impeller Walls.

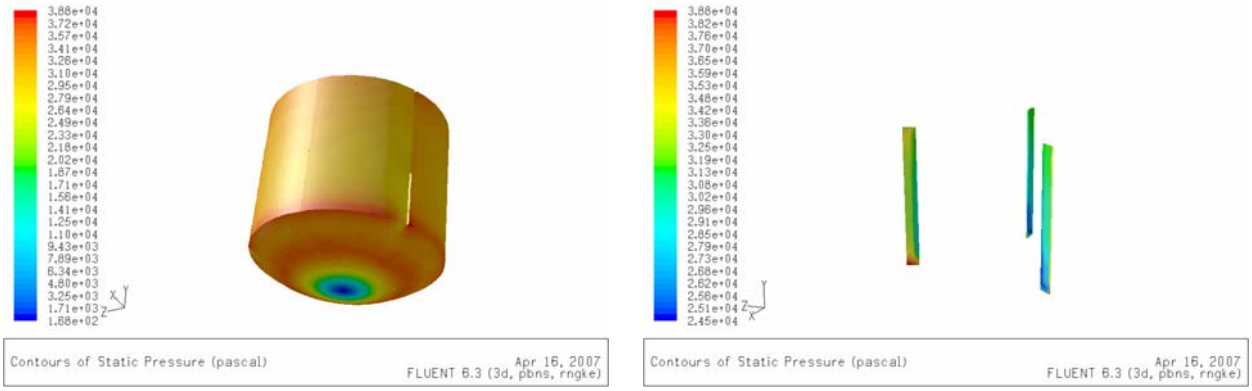
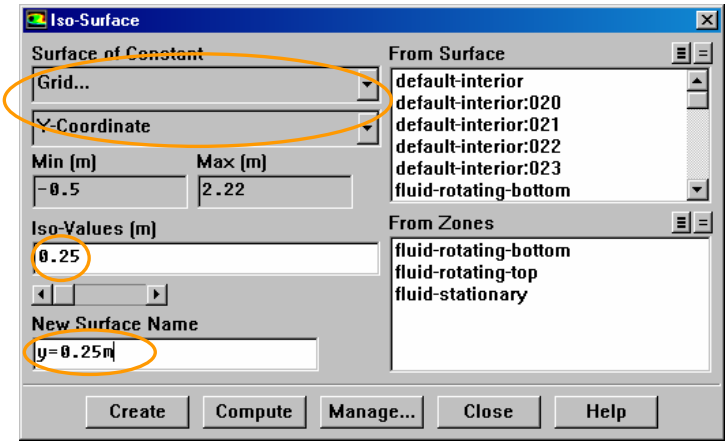


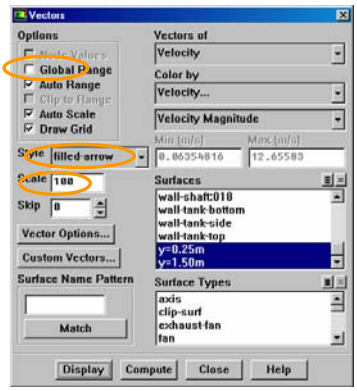
Figure 12 & 13: Static Pressure Contours on the Tank Walls and Baffles.

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…**



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…**



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.

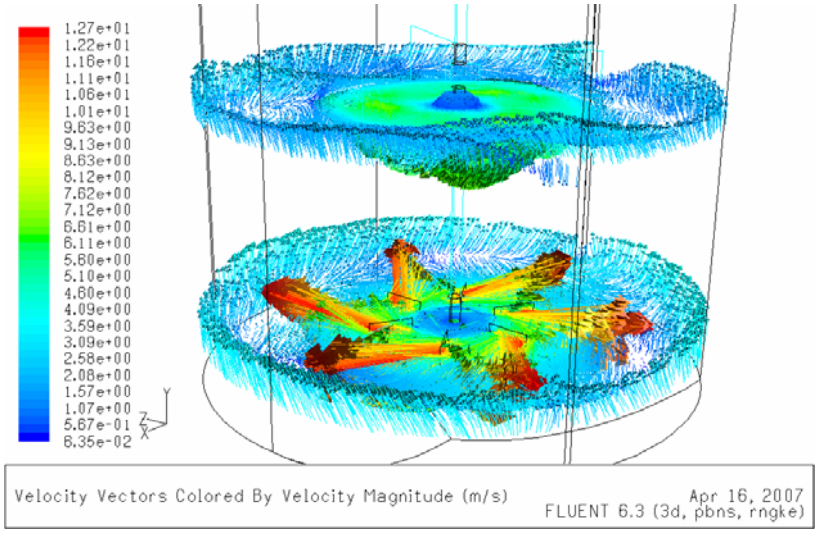


Figure 14: Velocity Vectors at y=0.25 and 1.50 [m] Planes.

- 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].

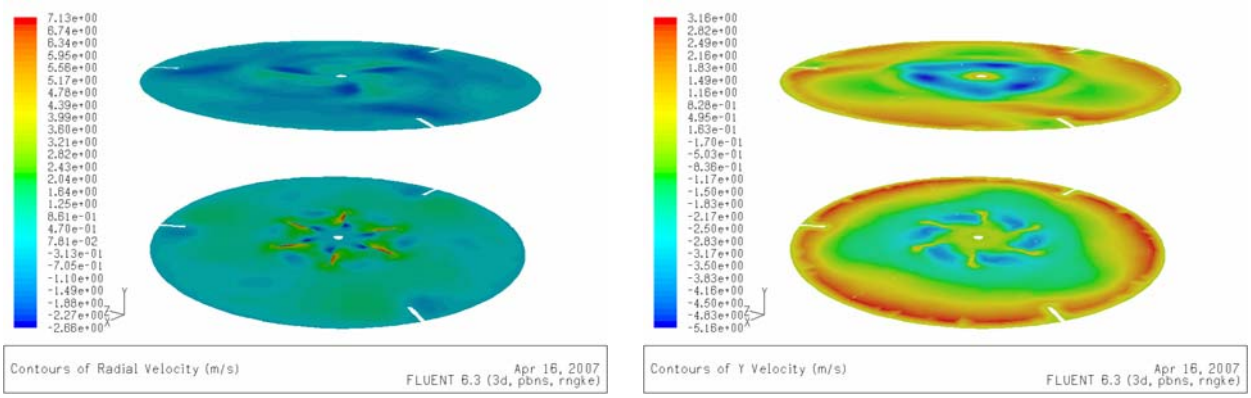
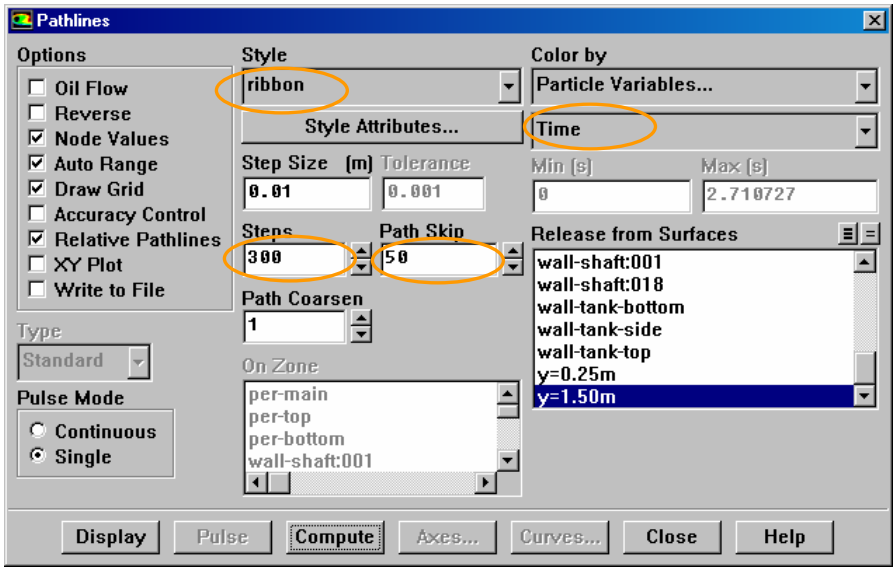


Figure 15 & 16: Contours of Radial and Axial Velocities at y=0.25 and 1.50 [m] Planes.

- 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…**



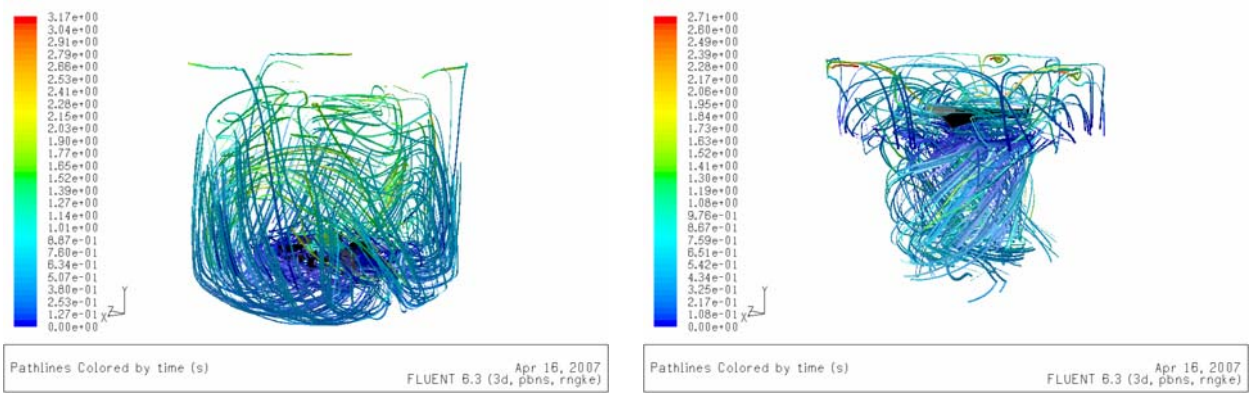
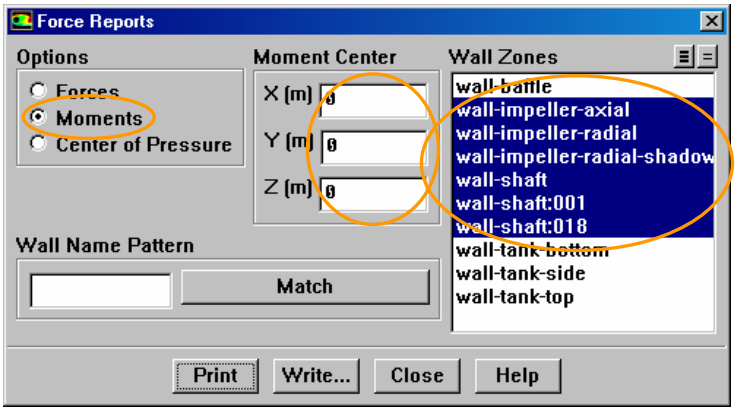
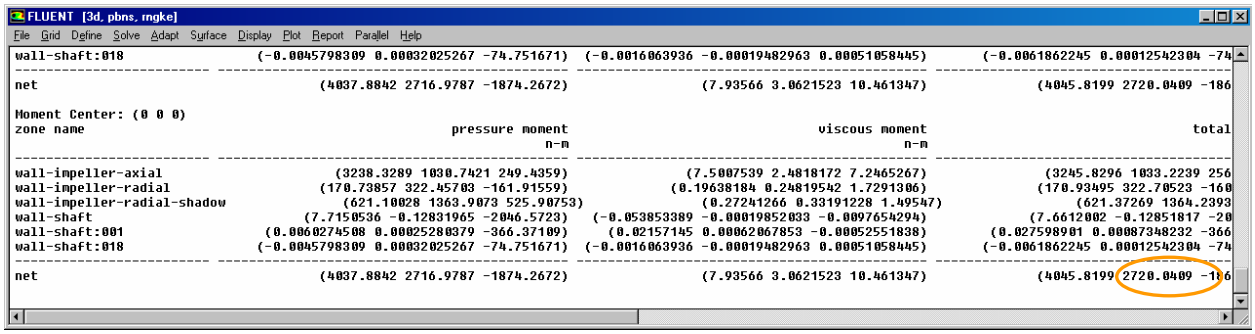


Figure 17 & 18: Pathlines Released from y=0.25 and 1.50 [m] Planes.

- 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…**



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:



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(ω)=(2720)x(200/60)=9.07 [kW].

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

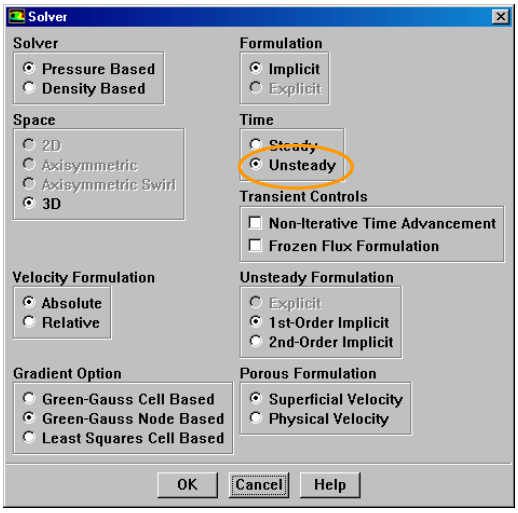
8th STEP: Calculating Mixing Time

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, $\mathbf{t_{99}}$ **[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.: $\mathbf{U=C(t)/C_{\infty}}$. 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}.

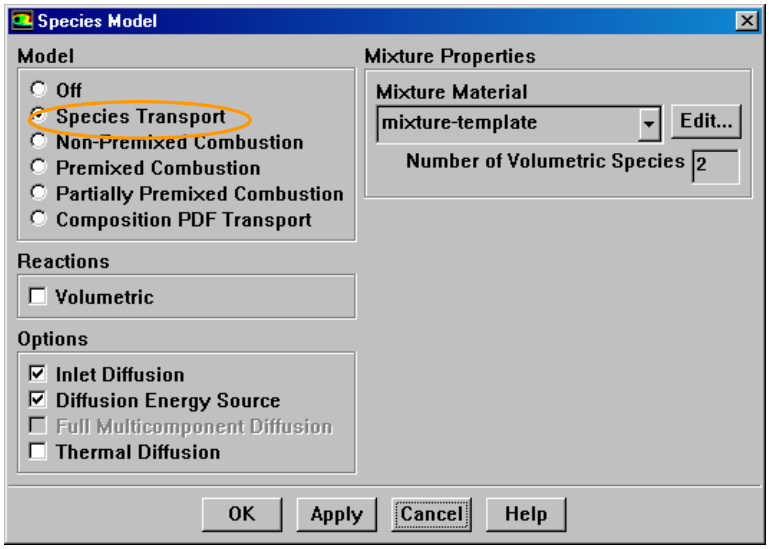
- Activate the Unsteady Solver: **Define > Models > Solver…**



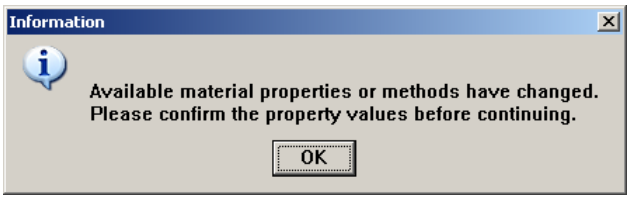
Select "Unsteady" in "Time" and keep the default "1st-Order Implicit" in "Unsteady Formulation".

- Activate the Species Transport Model:

  Define > Models > Models > Transport & Reaction…

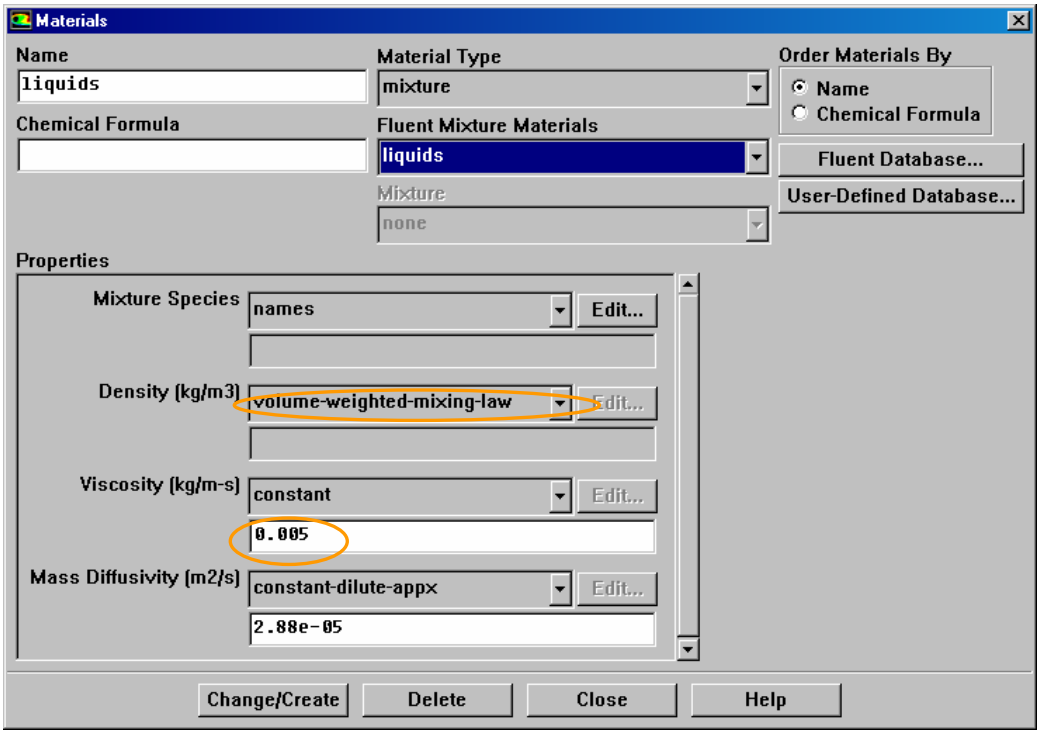


Select "Species Transport" in "Model". The panel will expand and will look like above. Keep the defaults close the panel. You will see some warning in the TUI, regarding the materials and the following message:

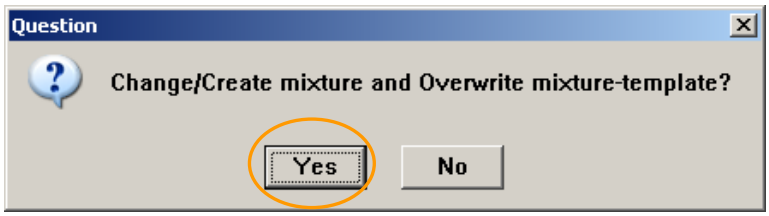


- Modify the existing liquid and add a new one: **Define > Materials…**

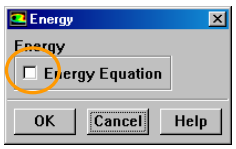
Choose "mixture" in the category "Material Type" and this will lead you to the following panel. Change the name in "Name" to "liquids", choose "volume-weighted-mixing-law" in "Density [kg/m3]" and leave "constant" in "Viscosity [kg/m-s]" and put a value of "0.005". Do not change Thermal Conductivity [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).



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…**

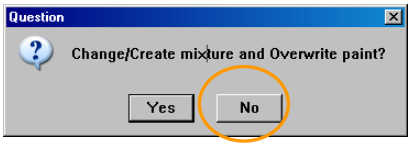


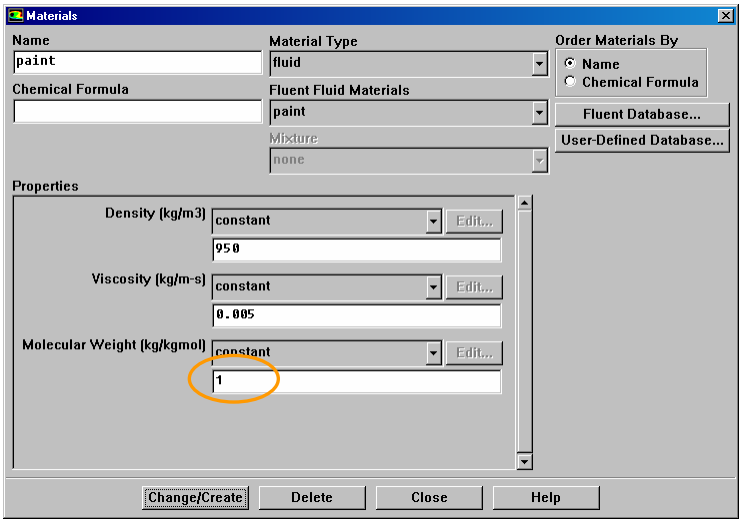
Uncheck the check-box and close this panel.

- 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

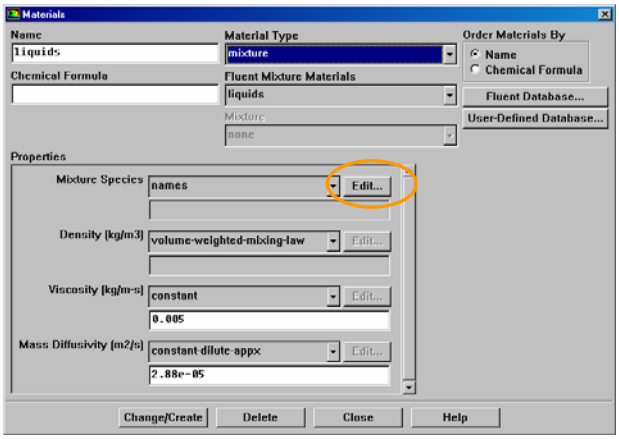
"tracer" and press button **Change/Create**. FLUENT will ask you the following questions, to which you should press button **No**.

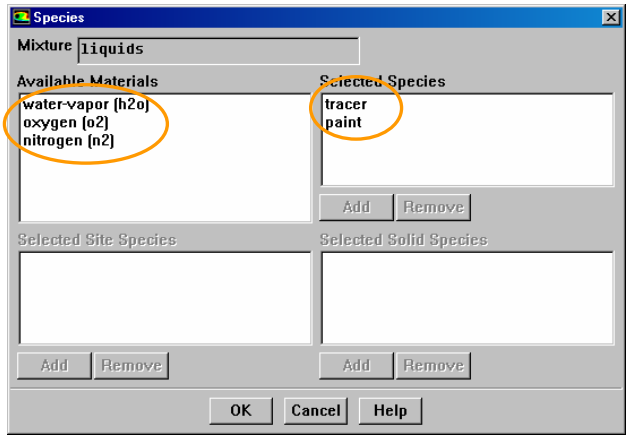




We have created the tracer liquid, but now we must define these two ("paint" and "tracer") as the only two components of the mixture.

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:



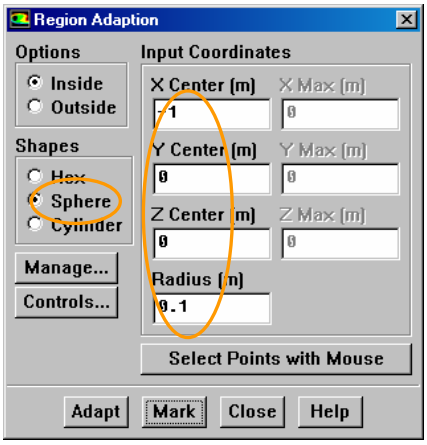


Select from the list on the left "tracer" and press button "Add" and then do the same for "paint". Then select from the right list (one at a time) "h2o", "o2" and "n2" and press button "Remove". The final result should be as above (make sure that "tracer" is on top. Press button **OK** to return to materials 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.

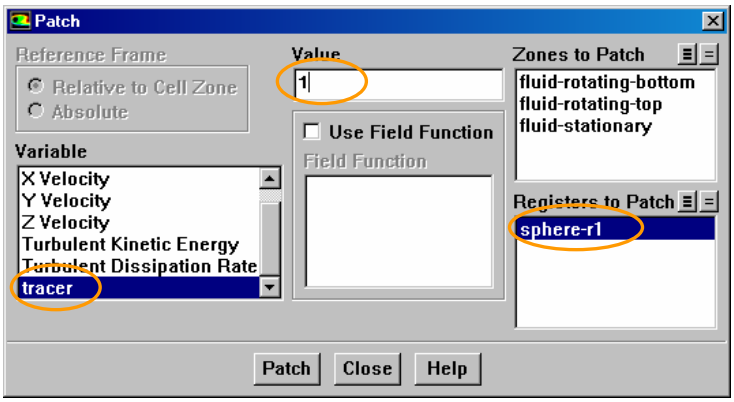
➢ Patch a mass fraction of unity for the tracer in a small volume.

Adapt > Region…

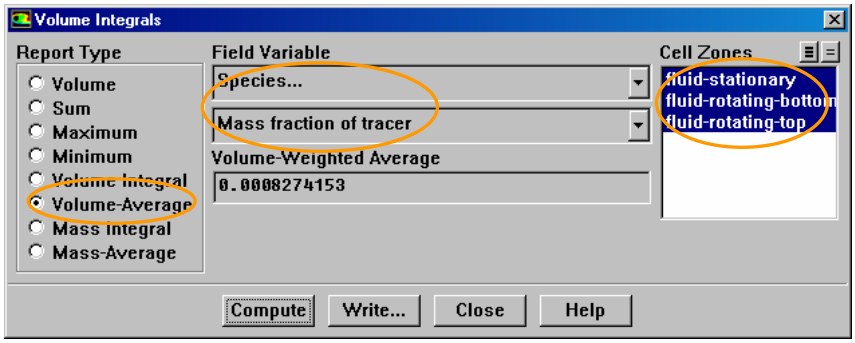
Select "Sphere" in "Shapes" and then put the following data for the sphere centre and radius. Then press button **Mark** and FLUENT will report that 27 cells were marked.



Now with **Solve > Initialize > Patch…**



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…**



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/m^3], since both of its components have the same density), i.e.

M_{tank}=(5.069112 [m^3])x(950 [kg_mixture/m^3])=4815.656 [kg_mixture].

So the total tracer mass can be indirectly calculated by the product of tank-average mass fraction and total mass,

M_{tracer}=(0.0008274153[kg_tracer/kg_mixture])x(4815.656[kg_mixture])=3.9845[kg_tracer].

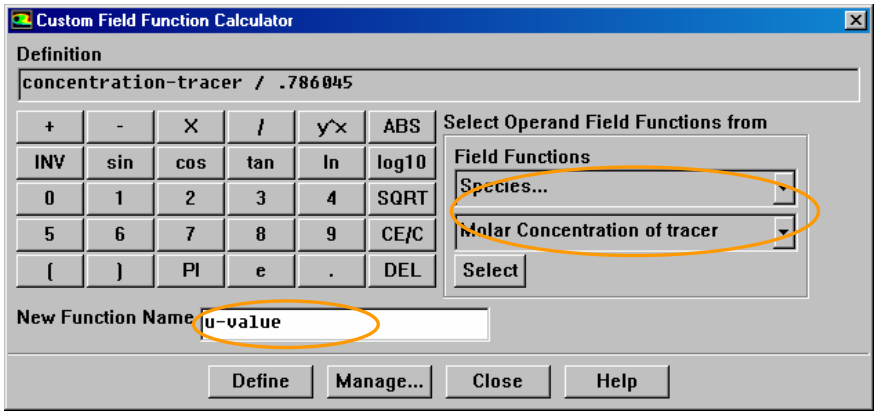
Hence, the infinite value of concentration is equal to:

C_∞=(3.9845 [kg_tracer]) / (5.069112 [m^3])=0.786045 [kg_tracer/m^3]

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

- Create a Custom Field Function for U variable:

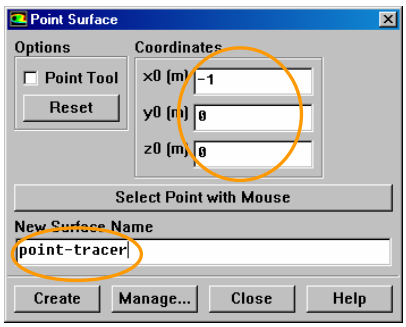
  Define > Custom Field Functions…



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 post-processing operations.

- Create the sampling points: **Surface > Point…**

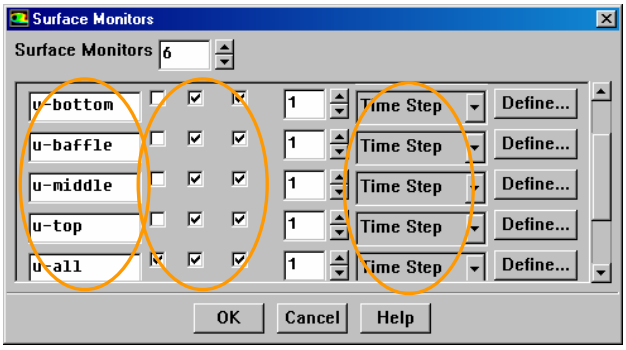


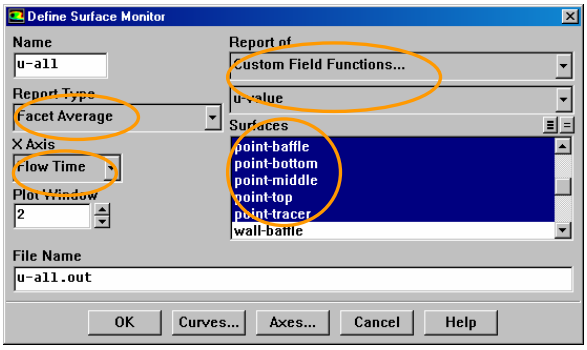
Create the following points, by giving their coordinates and typing their names in "New Surface Name" before pressing button **Create**. See with **Display > Grid…** where these point are located.

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.

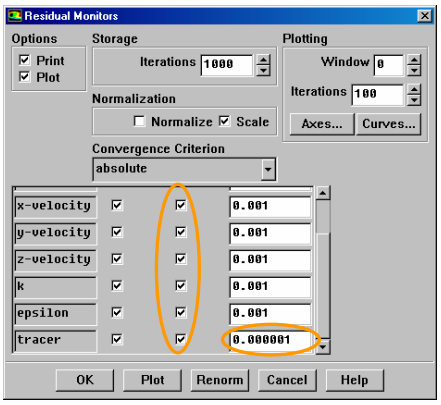
Solve > Monitors > Surface …



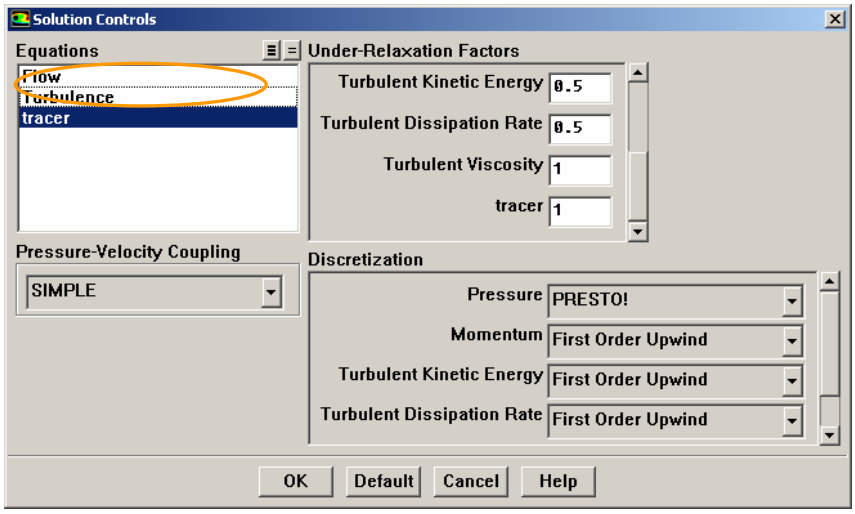


You will change the settings for the first monitor. Also do not forget to de-activate the Force monitor for moment coefficient that was set in page 14; both monitors are not required anymore. Increase to 6 the number of "Surface Monitors" and rename them to "u-tracer", "u-bottom", "u-baffle", "u-middle", "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 …**

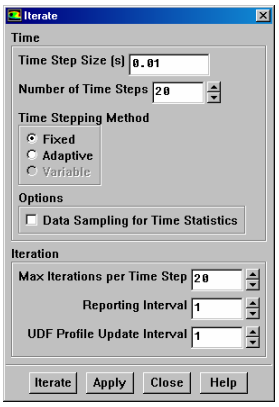


- Save the case & data file as "mixing-tank_tet_stage4_0.0s".
- Deactivate the solution of all the flow & turbulence equations:
  Solve > Controls > Solution…

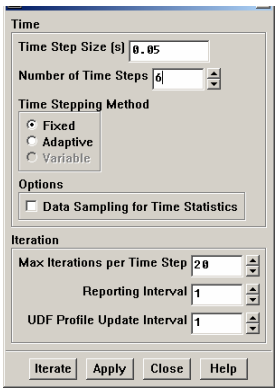


  Flow and turbulence field are converged and only the tracer concentration needs to be tracked in time.

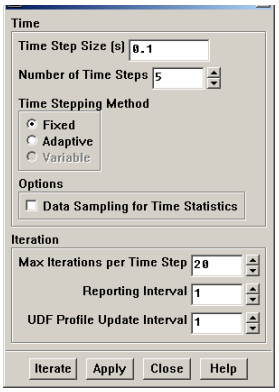
- Set the time step size to 0.01 [s] and request 20 time steps. **Solve > Iterate…**



- 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…**



- 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…**



- 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).

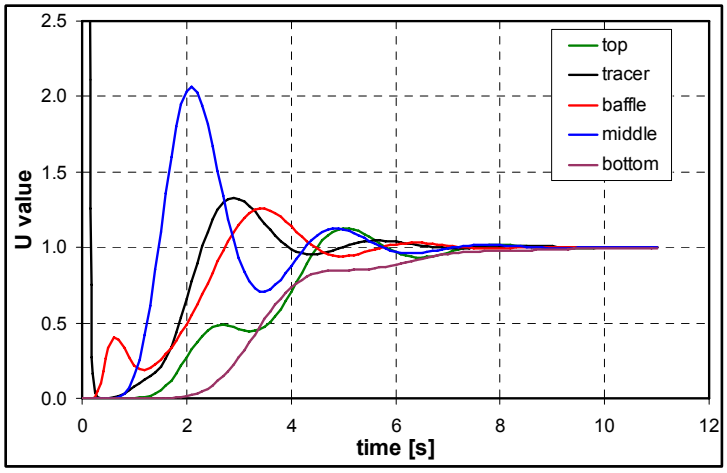


Figure 19: Time History of U Variable at the Sampling Points.