Example 7. Modelling of Multiphase Liquid-Liquid Flow in a Static Mixer

7.1. Introduction and Objectives

Multiphase flows have a wide range of applications in many industrial areas, such as the chemical, pharmaceutical, cosmetic and food industries. One type of multiphase system is a two-phase liquid-liquid dispersion, called emulsion. It is a mixture of two liquids that are usually immiscible or only partially miscible due to the liquid-liquid phase separation. An emulsion is a thermodynamically unstable system because of a natural tendency for a liquid-liquid system to separate and reduce its interfacial area and energy. However, most emulsions are stable over some time, i.e. they demonstrate kinetic stability. There are various ways to obtain emulsions, including mixing that can be performed in a particular emulsification device, namely a static mixer. Static mixers consist of a cylindrical pipe and a series of motionless mixing elements called inserts. There are various shapes of inserts, and their function is to induce complex flow fields by redistributing liquids in directions transverse to the main flow. Static mixers are an attractive alternative to mechanical mixers. Their main advantages are no moving elements, continuous work, small space requirements, and low power requirements. In this example, we will investigate the emulsification process with the classical Kenics helical mixer. The main objectives of this exercise are as follows:

- create 3D geometry with application of Sweep, Pattern and Body Operations options,
- illustrate the setup of the multiphase simulation using Mixture model,
- visualization of results contours of volume fraction of both liquids, vectors of velocity.

7.2. Problem Description

Details of the analysed flow problem are shown in Figure E7-1.



Figure E7-1. Problem description.

The fluids involved in emulsification process in this example are the water as continuous phase and oil as dispersed phase. The density and viscosity of the oil are equal to 903.1 kg/m³ and 0.473 Pas, respectively. The volume ratio of the continuous and dispersed phases is 90:10. The surface tension at the interface of the two phases is equal to 0.02 N/m. The velocity of both liquids at the inlet is equal to 0.1 m/s.

7.3. Geometry

Open *Ansys DesignModeler* and go to the *Sketching* tab. Make sure that *Auto Constraints* options are enabled and setup a proper view by clicking *Look At Face/Plane/Sketch*. Using the *Rectangle* option draw a rectangle in a way that the origin of the coordinate system will be inside this rectangle. Then measure two sides of the rectangle and distances from the *X* axis to the top side of the figure and from the *Y* axis to its right side (Fig. E7-2a). Enter the appropriate values as H2 = 100 mm, V1 = 3 mm, H3 = 50 mm and V1 = 1.5 mm to draw the base of a single insert. Set the isometric view (Fig. E7-2b). Go to the *Modeling* tab and make *Sketch1* always visible (*Always Show Sketch* option). Now come back to the Sketching tab in order to draw a path along which the insert profile will be swept. Note, that the path must be perpendicular to the profile, therefore in the main menu change the plane selecting from the dropdown list *ZXPlane*. The view of the geometry in the *Graphics* window should be like in Fig. E7-2c (pay attention to the axles). Now draw a line along *Z* axis (it is denoted by red colour in the local coordinate system) starting from the origin, measure it (Fig. E7-2d) and set value as 150 mm (the insert length). The profile and path needed to create the insert are ready.



Figure E7-2. Single insert drawing steps.

Now select from the top ribbon (or *Create* menu) the *Sweep* option. In the *Details View* (Fig. E7-3a) select *Sketch1* as *Profile*, *Sketch2* as *Path*, *Add Frozen* as *Operation* and *Turns* as *Twist Specification*. Since insert is twisted by a degree of 180° enter the value of 0.5 in the *FD5*, *Turns* field and click generate. The first insert is ready (Fig. E7-3b)

D	etails View	ф.	
Ξ	Details of Sweep1		
	Sweep	Sweep1	
	Profile	Sketch1	
	Path	Sketch2	
	Operation	Add Frozen	
	Alignment	Path Tangent	
	FD4, Scale (>0)	1	
	Twist Specification	Turns	
	FD5, Turns	0,5	
	As Thin/Surface?	No	
	Merge Topology?	No	
-	Profile: 1	8	
	Sketch	Sketch1	

Figure E7-3. Details View of Sweep option and one insert.

The second insert has to be draw in analogous way with such difference that i) it starts in place where the preceding one ends, ii) it is positioned perpendicular to the previous insert and iii) it is twisted in the opposite direction.

The drawing of the second insert should be started by creating a plane in the space where the start face of the insert will be located. We know the second insert touch the previous one therefore we will create a new plane at the end face of the first insert. In this purpose select from the top ribbon a *New Plane* icon \bigstar . Now you have two possibilities of drawing new plane – starting *From Plane* or *From Face*. Here we will use *From Plane* as a type, therefore set the *Details of Plane4* as it is shown in Fig. E7-4a. If you want to try *From Face* type, then select this method from the *Type* dropdown list, choose the *Base Face* directly from the graph and set the *Subtype* in the details as *Tangent Plane*. Make sure that the origin of a new plane is in the centre of the first insert end face and click generate. The *Plane4* is ready (Fig. E7-4b).

D	etails View		ą
	Details of Plane4		
	Plane	Plane4	
	Sketches	0	
	Туре	From Plane	
	Base Plane	XYPlane	
	Transform 1 (RMB)	Offset Global Z	
	FD1, Value 1	150 mm	
	Transform 2 (RMB)	None	
	Reverse Normal/Z-Axis?	No	
	Flip XY-Axes?	No	
	Export Coordinate System?	No	



Figure E7-4. Details View of Plane4 and a new Plane4.

On plane 4, draw and dimension a rectangle analogous to the one on the XY plane rotated by 90 degrees (Fig. E7-5a). Then create a new plane starting from the *ZXPlane* in order to draw a path along which the profile of second insert will be swept (Fig. E7-5b).



Figure E7-5. Profile and path needed to create the second insert.

Again select the *Sweep* option. Details (Fig. E7-6a) set up analogous as previously with one difference – since the second insert is twisted in opposite direction enter the value of -0.5 in the *FD5*, *Turns* field and click generate. The second insert is ready (Fig. E7-3b)

D	etails View	¢.
-	Details of Sweep2	
	Sweep	Sweep2
	Profile	Sketch3
	Path	Sketch4
1	Operation	Add Frozen
	Alignment	Path Tangent
	FD4, Scale (>0)	1
	Twist Specification	Turns
	FD5, Turns	-0,5
	As Thin/Surface?	No
	Merge Topology?	No
-	Profile: 1	
	Sketch	Sketch3



Figure E7-6. Details View of Sweep2 option and the second insert.

The total number of inserts is 6, but each subsequent insert is the same as the first or second, so we won't draw them. They will be copied. For this we will use the *Pattern* option that can be found in the *Create* menu. In the *Details View* window set up the *Pattern Type* as *Linear*, select two inserts from the *Graphics window* as *Geometry* and *Global Z* axis with "positive" direction

as *Direction*. The axis must be selected directly from the *Graphics window* and the copy direction can be set by clicking on the grey button with black and red arrows that will appear in the bottom left corner of the



graphics window. The copy direction will be indicated as in the Fig. E7-7. To shift the copies to the correct position, set the *FD1*, *Offset* to 300 mm. At the end enter the number of copies in the *FD3*, *Copies* (>=0) field as 2 and click generate to create four new inserts (Fig. E7-8).



Figure E7-8. Six Kenics inserts.

It was earlier discussed that when we draw a geometry of the flow domain our task is to draw not the object but the fluid around this object. Therefore, now we have to draw a circular pipe – cylinder that represents fluid around the elements, and then remove (cut) inserts from the volume of that cylinder. In order to create cylinder, use the *Primitives* options with details presented in Fig. E7-9.

In order to cut the inserts from the cylinder body, select from the *Create* menu the *Body Operation*. In the *Details View* window set the *Type* as *Cut Material* and select as *Bodies* 6 geometries that represent the inserts (Fig. E7-10).

Details of Cylinder1		
Cylinder	Cylinder1	
Base Plane	XYPlane	
Operation	Add Material	
Origin Definition	Coordinates	
FD3, Origin X Coordinate	0 mm	
FD4, Origin Y Coordinate	0 mm	
FD5, Origin Z Coordinate	-300 mm	
Axis Definition	Components	
FD6, Axis X Component	0 mm	
FD7, Axis Y Component	0 mm	
FD8, Axis Z Component	1500 mm	
FD10, Radius (>0)	50 mm	
As Thin/Surface?	No	

Figure E7-9. Details of Cylinder1.

D	etails View	ew		
	Details of BodyO	p1		
	Body Operation	BodyOp1		
	Туре	Cut Material		
	Bodies	6		
	Preserve Bodies?	No		

Figure E7-10. Details of *Body Operation*.

Figure E7-7. Details View of Pattern option.

You can select them from the *Graphics* window or from the *Tree Outline* (remember about pressing *Ctrl* button when selecting more than one body). Confirm by clicking *Generate*. Your geometry is ready now (Fig. E7-11). If everything was done correctly, you should see in the *Tree Outline* 1 Part and 1 Body.



Figure E7-11. Geometry of the static mixer.

7.4. Mesh

Open the Ansys Meshing, rename the geometry as Kenics, and set the Material as Fluid. Next, choose the Mesh from the Tree Outline. Ensure that in the Details of "Mesh" the Physics Preference is set as CFD and Solver Preference as Fluent, and confirm by clicking Generate Mesh. In order to modify the grid from the Top Menu select the Mesh tab and the Method option. In the Details of "Automatic Method" – Method dialog box define the geometry. Then from the Method dropdown list select Layered Tetrahedrons. The Details window will expand. Enter value of the Layer Height as 0.003 m. Click Generate. The mesh is ready (Fig. E7-12). When you examine the mesh quality you should see that it is poor, not enough to perform the reliable simulation. However, do not change any settings of the grid. In this example you will see that sometimes a poor quality of the mesh can be improved in Ansys Fluent. Here, in the last step create two Named Selections – Inlet and Outlet.



Figure E7-12. Part of a numerical mesh generated for the Kenics static mixer.

7.5. Simulations

Open the *Ansys Fluent* with enabled *Double Precision* and *Display Mesh After Reading* options and set the *Solver Processes* as 1. Run down the *Outline View Tree* to edit subsequent parameters and variables:

A. Setup – General

Examine the *Mesh* \rightarrow *Check* and *Mesh* \rightarrow *Report Quality*. After selecting *Report Quality*, you should see in the console warning that the minimum orthogonal quality is below 0.01, what indicate the poor quality of the mesh and can lead to poor results or even to iteration divergence. In order to improve the grid quality, select from the top ribbon *Domain* tab and then *Mesh* \rightarrow *Quality* \rightarrow *Improve Mesh Quality*... as it is shown in Fig. E7-13a. A new dialog box will open (Fig. E7-13b), where we can define the percentage of cells to be improved and number of iterations. Set both these values as 0.1 and 4, respectively and click *Improve*. The results of grid improvement will be reported in the console (Fig. E7-14).



Figure E7-13. Improving the mesh quality in Ansys Fluent.

Wininwum Orthogonal Quality = 1.66402e-03 cell 270768 on zone 3 (ID: 407748 on partition: 0) at location (7.20902e-03, -8.52150e-03, -1.38381e-01)
Warning: minimum Orthogonal Quality below 0.01.
Naximum Aspect Ratio = 3.42917e+01 cell 282559 on zone 3 (ID: 168637 on partition: 0) at location (-8.25914e-03, 9.88666e-03, -2.21626e-01)
Fluent can try to improve the mesh quality via the TUI command
/mesh/repair-improve/improve-quality
Improving poor quality cells.
Considering worst 0.100000% of cells.
Identified 524 cell(s) (out of 501881), 0.104407%, with orthogonal quality below 0.022949.
Done.
Mesh Quality:
Minimum Orthogonal Quality = 1.47119e-02 cell 433316 on zone 3 (ID: 120873 on partition: 0) at location (8.73611e-03, -3.86918e-02, -2.99477e-01)
Maximum Aspect Ratio = 8.74478e+01 cell 206172 on zone 3 (ID: 470445 on partition: 0) at location (1.71585e-02, -6.97948e-03, -2.77391e-01)

Figure E7-14. Report of the numerical grid improvement.

In the General tab set also the Solver Type \rightarrow Pressure-Based, Solver Velocity Formulation \rightarrow Absolute, Time \rightarrow Steady.

B. Setup – Materials

In this example some settings of the models will be connected with a new material that is not available in the *Fluent Database*, therefore in this example before *Models* we will define *Materials* first. As it was described in the *Problem Description* two liquids will be used in simulation, water and oil. Water (*water-liquid* material) can be simply added from the *Fluent Database*. The used oil is not available here, therefore you have to add this material analogically as it was described in the *Example 4*, entering the name as *Oil*, constant value of the density as 903.1 kg/m³ and also constant value of the viscosity as 0.473 Pas. When both materials are added to the *Material List* you can go to the *Models* tab in order to define all needed models.

C. Setup – Models

The fluid flow in this example is turbulent, therefore from the *Viscous Model* list select the *RNG k-epsilon* model together with the *Enhanced Wall Treatment* option. All of the model constants left without change. In order to simulate a two-phase liquid-liquid flow, select the *Multiphase* model from the model list. A new dialog box will open where you can select the specific model and enter additional options. In the *Models* tab (Fig. E7-15a) set the *Model* as *Mixture*, *Number of Eulerian Phases* as 2, *Volume Fraction Parameters* as *Implicit, Interface Modeling Type* as *Dispersed*, and select *Slip Velocity* and *Implicit Body Force*. In the same *Multiphase Model* dialog box go to the *Phases* tab. From the *Phases* list select *phase-1 – Primary Phase*, set the *Phase Material* as *water-liquid* and confirm by clicking *Apply* (Fig. E7-15b). Then from the *Phases* list select *phase-2 – Secondary Phase*, set the *Phase Material* as *oil*, define constant diameter as 0.001 m and also confirm with *Apply* (Fig. E7-15c). Now go to the *Phase Interaction* tab. From the *Phase Pairs* list select the *phase-1 phase-2* and enter the following settings as the *Force Setup: Drag Coefficient*, *Coefficient* → *schiller-naumann*, *Drag Coefficient*, *Modofication* → none, *Slip Velocity* → *manninen-et-al*, *Surface Tension Coefficient* → *constant* with value of 0.02 N/m (Fig. E7-15d).



Figure E7-15. Setup of the Multiphase model.

D. Setup – Cell Zone Conditions

In this example materials were assigned to the flow domain in the previous step, therefore there is no settings that we should enter in the *Cell Zone Conditions* tab.

E. Setup – Boundary Conditions

Open the *Boundary Conditions Task Page* and set the boundary conditions as *velocity-inlet* for *Inlet* zone and *pressure-outlet* for the *Outlet* zone. Then edit the inlet boundary condition entering value of the *Velocity Magnitude* as 0.1 m/s, separately for each phase, as it is shown in Figs. E7-16a, b. When you select phase-2 from the dropdown list then the *Multiphase* tab will be activated (Fig. E7-16c). You can enter here a fraction of the dispersed phase, which is equal to 0.1 (the volume ratio of the continuous and dispersed phases is 90:10).

🥌 Velocity I	inlet >
Zone Name	Phase
inlet	phase-1 💌
Momentum	Thermal Radiation Species DPM Multiphase P phase-1 cture UDS
Ve	elocity Specification Method Magnitude, Normal to Boundary
	Reference Frame Absolute
	Velocity Magnitude [m/s] 0.1
	Apply Close Help
🥌 Velocity I	inlet >
Zone Name	Phase
inlet	phase-2 💌
Momentum	Thermal Radiation Species DPM Multiphase Pr phase-1 cture UDS
Ve	elocity Specification Method Magnitude, Normal to Boundary
	Peferance Frame Abcelute
	Velocity Magnitude [m/s] 0.1
	Apply Close Help
🥌 Velocity I	inlet >
Zone Name	Phase
inlet	phase-2 💌
Momentum	Thermal Radiation Species DPM Multiphase Potential Structure UDS
	Volume Fraction 0.1
	Apply Close Help

Figure E7-16. Details of the Velocity Inlet Boundary Condition.

F. Solution – Methods

Open the Solution Methods Task Page and ensure that the options within this page are set up as: Scheme \rightarrow Coupled, Gradient \rightarrow Least Squares Cell Based, Pressure \rightarrow PRESTO!, Momentum, Turbulent Kinetic Energy, k, and Turbulent Dissipation Rate, ε , \rightarrow Third-Order MUSCL, Volume Fraction \rightarrow QUICK and other options are disabled.

G. Solution – Monitors

Open the *Residual Monitors* dialog box, make sure that all equations are monitored and their convergence is checked. Then set all *Absolute Criteria* as 1e-7.

H. Solution – Initialization

Open the *Solution Initialization Task Page* and initialize the calculation selecting the *Hybrid Initialization*.

I. Solution – Run Calculation

Open the *Run Calculation Task Page* and examine your setup selecting *Check Case...* There should be no recommendations here. You can start the calculation by entering 1000 iterations. During the calculations in the main window you will observe the *Scaled Residuals*.

7.6. Results

In order to present the results, a plane must be created passing through the centre of the static mixer. In this example, the mid-plane will show both the contours and the vectors of the selected parameters.

Contours of volume fraction

Open the *Contours* dialog box (Fig. E7-17). Set the *Contours of* as *Phases*... and *Volume fraction*, and the *Phase* as *phase-1*. From the *Surfaces* list select the created mid-plane (here it is *plane-4*). From the *Options* lists select *Filled*, *Node Values*, *Boundary Values*, *Global Range*, *Auto Range*. The *Coloring* set up as *Smooth*. Click *Save/Display* button.

Figure E7-17. Settings in the Contours dialog box.

Contours		×
contour-1		
Options	Contours of	
✓ Filled	Phases	~
✓ Node Values	Volume fracti	on 👻
Boundary values	Phase	
Clobal Range	phase-1	-
Auto Range	Min	Max
Clip to Range	0.4592192	0.9808314
Draw Profiles	Surfaces Filte	r Text 🔂 🖶 🛃
	inlet outlet	
Coloring	plane-4	
 Banded Smooth 	wall wall-solid	
	Display State	
Colormap Options	None	 Use Active New Surface -
s	ave/Display	Compute Close Help

The obtained contours of the volume fraction of continuous phase (phase-1) will be visible in the main window. The above contours are presented in Fig. E7-18a. In similar way you can create contour of the volume fraction of dispersed phase (phase-2, Fig. E7-18b) just by changing in the *Contours* dialog box *Phase* from *phase-1* to *phase-2*.



Figure E7-18. Contours of the volume fraction of continuous (phase-1) and dispersed (phase-2) phases.

Vectors of velocity

Double-click the *Vectors* in the *Outline View Tree* to open the *Vectors* dialog box (Fig. E7-19). Set the *Vectors of* as *Velocity*, the *Phase* in two fields as *mixture*, and the *Color* as *Velocity*... and *Velocity Magnitude*. From the *Surfaces* list select the created midplane (*plane-4*). From the *Options* lists select *Global Range*, *Auto Range*, *Auto Scale* and *Draw Mesh*. By selecting the last option, you will open a new dialog box – *Mesh Display* (Fig. E7-20). Here you can set the parameters that allow you to visualise the geometry of the flow domain together with the simulation results.

rector nume	
vector-1	
Options	Vectors of
✓ Global Range	Velocity
Auto Range	Phase
Clip to Range	mixture
✓ Auto Scale	Color by
✓ Draw Mesh	Velocity
Style	Valacity Magnituda
3d arrow 💌	
Scale Skip	Phase
6 50	
Vector Ontions	ダ Min [m/s] Max [m/s]
vector options	7.983011e-06 0.2765201
Custom Vectors	Surfaces Filter Text
Colormap Options	inlet
	outlet
	plane-4
	wall
	wall-solid
	Display State
	None Vise Active New Surface

Figure E7-19. Settings in the Contours dialog box.

This tool is particularly useful when displaying vectors because, unlike contours, once the vectors have been adjusted to the right scale for correct analysis, they do not completely fill the

analysed area. The geometry view therefore makes it possible to unambiguously determine the position of vectors in the computational domain. In the *Mesh Display* dialog box set the *Options* as *Edges*, *Edge Type* as *Outline* and from the *Surfaces* list select *inlet*, *outlet* and *wall*. Click *Display* and close this dialog box.

Options	Edge Type	Surfaces Filter Text
✓ Edges	O All	inlet
Eages	Outline	outlet
Destitions	Outime	plane-4
Fartitions		wall
Uverset		wairsolid
Shrink Factor F	eature Angle	
0	20	
Outline	Interior	
Adjacency		New Surface 🚽

Figure E7-20. Settings in the Mesh Display dialog box.

Come back to the settings in the *Vectors* dialog box (Fig. E7-19). From the *Style* dropdown list you can select one of the available styles. In this example select *3d arrow*. Then enter the values of *Scale* and *Skip* as 6 and 50, respectively. With this option, the vectors will be larger, but at the same time the number of vectors displayed will be smaller. In the next step click on the *Vector Options*... field in the *Vectors* dialog box. This will open the *Vector Options* dialog

box (Fig. E7-21), where you should select *In Plane* option since we display the vectors in the plane. You can also change the scale of the vector head here and set a specific colour for the vectors, but leave these settings unchanged. Click *Apply* and close this dialog box. After enter all the above settings Click *Save/Display* button in the *Vectors* dialog box in order to display the vectors. They are presented in Fig. E7-22.

✓ In Plane	Scale Head
Fixed Length	0.3
X Component	t
Y Component	i l
✔ Z Component	í I
	Color

Figure E7-21. Settings in the Vector Options dialog box.



Figure E7-22. Basic and close-up views of vectors of the mixture velocity magnitude.