

Workshop 1

Mixing T-Junction

Introduction to CFX



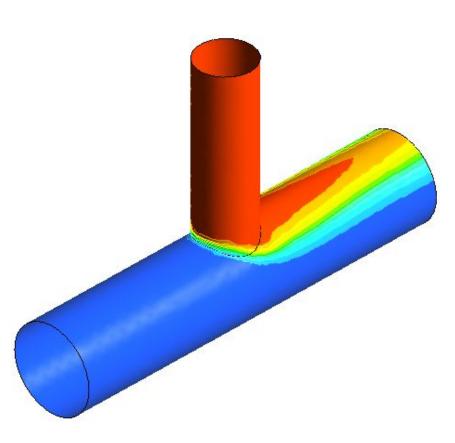
ANSYS, Inc. Proprietary © 2009 ANSYS, Inc. All rights reserved.

WS1: Mixing T-Junction Welcome!

- This introductory tutorial models mixing of hot and cold water streams
- The workshop starts from an existing mesh and applies boundary conditions to model a cold main inlet and a hot side inlet
- Analysis goals for this type of problem could be to determine:
 - how well do the fluids mix?
 - what are the pressure drops?

Note: It's a good idea to identify the quantities of interest from the start. You can use these to monitor the progress of the solution





WS1: Mixing T-Junction Pre-processing Goals

- Launch CFX-Pre from Workbench
- Use pre-defined materials
- Define the fluid models in a domain
- Create and edit objects in CFX-Pre
- Define boundary conditions
- Set up monitor points using simple expressions
- Launch the CFX Solver Manager from Workbench
- Monitor convergence

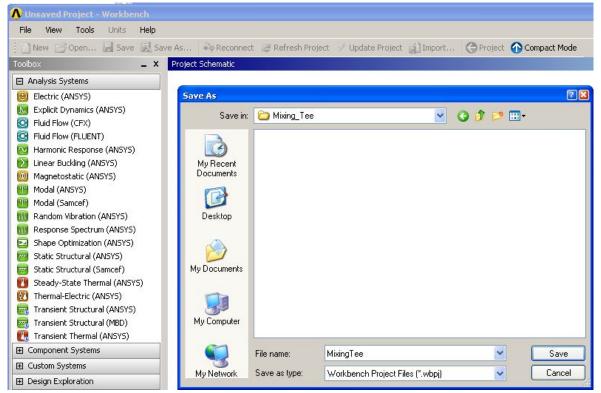
- Launch CFD-Post from an existing CFX simulation in Workbench
- Rotate, zoom and pan the view
- Create contour plots
- Create a plane for use as a locator
- Create a velocity vector plot
- Use pre-defined views
- Create streamlines of velocity
- Create an isosurface, coloured by a separate variable





The first step is to start Workbench:

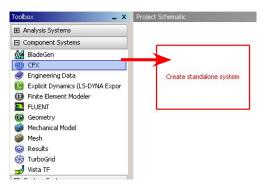
- From the windows Start menu, select Programs > Ansys 12.0 > Workbench
- When Workbench opens, select *File > Save* and save the project as *MixingTee.wbprj*



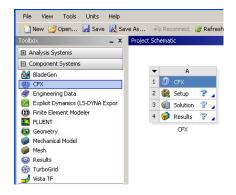
WS1: Mixing T-Junction Start a CFX case

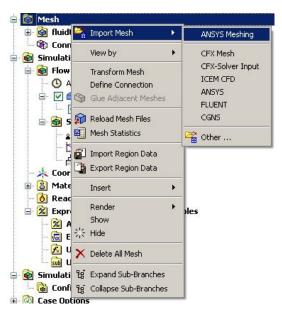


3. Next, expand the *Component Systems* toolbox and drag a *CFX* analysis into the top left area of the *Project Schematic*



- 4. Double-click on *Setup* to launch CFX
- When CFX-Pre opens, right-click on Mesh in the Outline tree and select Import Mesh > ANSYS Meshing
- 6. Select the file *fluidtee.cmdb* and click *Open*





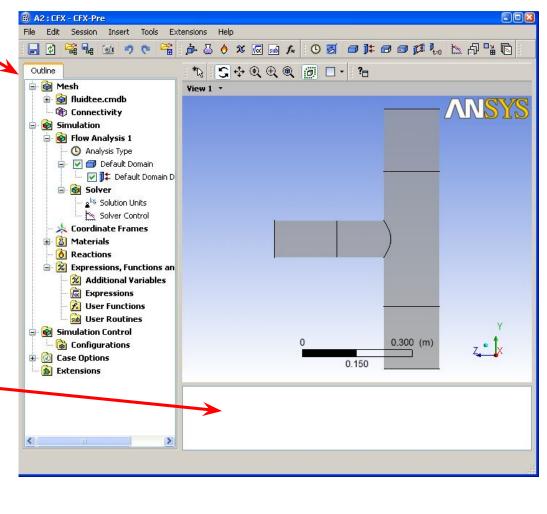
WS1: Mixing T-Junction CFX-Pre GUI Overview

Outline Tree

- New objects appear here as they are created
- Double-click to edit existing object
- New objects are often inserted by right-clicking in the Outline tree

Message Window

 Warnings, errors and messages appear here

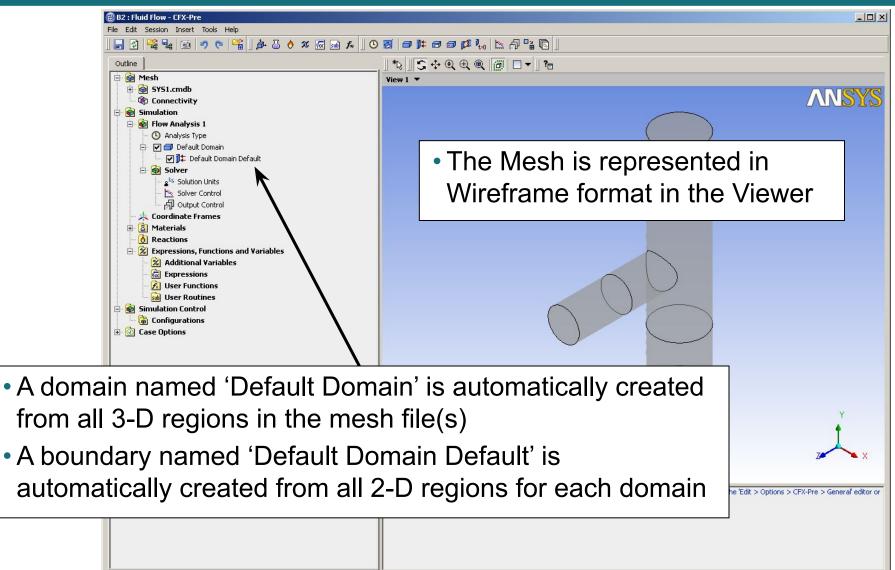




WS1: Mixing T-Junction CFX-Pre Mesh and Regions



Workshop Supplement



WS1: Mixing T-Junction CFX-Pre – Domain settings

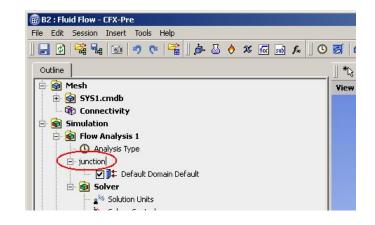




The Default Domain contains all 3D mesh regions that are imported. If you create new domains, those regions are automatically removed from the Default Domain. The Default Domain is automatically deleted if no unassigned 3D regions remain.

The first step is to change the domain name to something more meaningful.

- 1. Right-click on *Default Domain* in the *Outline* tree
- 2. Select Rename
 - The domain name can now be edited
- 3. Change the domain name to *junction*



WS1: Mixing T-Junction CFX-Pre – Domain settings (continued)

- 4. Double-click on the renamed domain *junction*
- Flow Analysis 1
 Analysis Type
 Analysis Type
 O I innotion
 O I I innotion
 O I I innotion
 O I I innotion
 O I I innotion



The Domain panel contains three tabs named *Basic Settings*, *Fluid Models* and *Initialisation*. For more complex simulations additional tabs may appear.

- 5. Set the *Material* to *Water*.
 - The available materials can be found in the drop-down menu

Note that CFX has a comprehensive library of materials. These can be accessed by using the icon and then selecting the *Import Library Data* icon.

ails of junction in Flo	w Analysis 1	
Basic Settings Fluid	Models Initialisation	
Location and Type		
ocation	B491	.
Domain Type	Fluid Domain	~
Coordinate Frame	Coord 0	~
Fluid and Particle Defi	nitions	Ξ
Fluid 1		
		X
Fluid 1		
Option	Material Library	*
Material	Air at 25 C	Image: Market
Morphology	Air Ideal Gas Air at 25 C	
Option	Water	
C Minimum Volu	me Fraction	•••••
Domain Models		
Pressure		
Reference Pressure	1 [atm]	
Buoyancy		Ξ
Option	Non Buoyant	~
Domain Motion		Ξ
Option	Stationary	~
		F
Mesh Deformation		

Workshop Supplement

CFX-Pre – **Domain settings (continued)**

6. Click the Fluid Models tab

WS1: Mixing T-Junction

- 7. In the *Heat Transfer* section, change *Option* to *Thermal Energy*
 - Heat Transfer will be modelled. This model is suitable for incompressible flows
- 8. Leave all other settings as they are
 - The k-Epsilon turbulence model will be used, which is the default
- 9. Click *OK* to apply the new settings and close the domain form



Basic Settings	Fluid Models Initialisation Solve	r Control
Heat Transfer —		
Option	Thermal Energy	•
📕 Incl. Viscous D	issipation	
Turbulence		
Option	(k-Epsilon)	.
Wall Function	Scalable	•
-Advanced Turbu	lence Control	
Combustion		
Option	None	•
-Thermal Radiation		
Option	None	-
	ic Model	

Close



The next step is to create the boundary conditions. You will create a cold inlet, a hot inlet and an outlet. The remaining faces will be set to adiabatic walls. Currently all external 2D regions are assigned to the *junction Default* boundary condition.



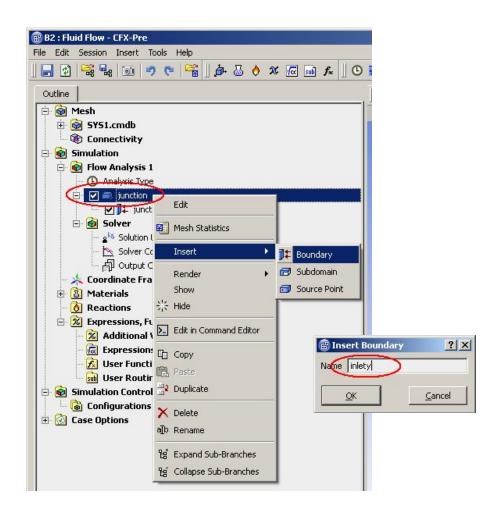
Each domain has an automatic default boundary condition for *external* surfaces. The default boundary condition is a No Slip, Smooth, Adiabatic wall. As you create new boundary conditions, those regions are automatically removed from the default boundary condition.

WS1: Mixing T-Junction CFX-Pre – Inlet boundary conditions



Now that the domain exists, boundary conditions can be added

- 1. Right-click on the *junction* domain
- 2. Select *Insert* > *Boundary*
- 3. Set the Name to inlety
- 4. Click OK

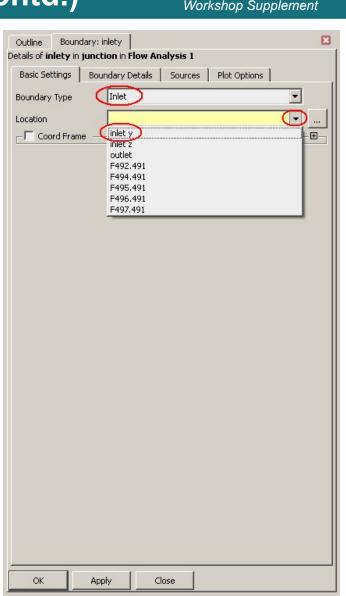


CFX-Pre – Inlet boundary conditions (contd.)

- 5. Leave the Boundary Type field set to Inlet
- 6. Set Location to inlet y

WS1: Mixing T-Junction

 The available locations can be found in the drop-down menu of the extended "..." menu



Workshon Supplement

CFX-Pre – Inlet boundary conditions (contd.)

This inlet will have a normal speed of 5 m/s and temperature of 10°C.

7. Click the Boundary Details tab

WS1: Mixing T-Junction

- 8. Enter a value of 5 for *Normal Speed*. The default units are [*m* s[^]-1]
- 9. Enter a value of *10* for *Static Temperature*. Use the drop-down menu to the right of the field to change the units to *C* (Celcius)
- 10. Click *OK* to apply the boundary and close the form

(10	<u> </u>	
pply	Close	



Boundary: inlety

Details of inlety in junction in Flow Analysis 1

Basic Settings (Boundary Details) Sources Plot Options

Subsonic

Normal Speed

Medium (Intensity = 5%)

Static Temperature

5 [m s^-1]

Outline

Flow Regime

Mass And Momentum

Option

Option

Normal Speed

Turbulence

Option Heat Transfer

Option

Static Temperature

Ξ

Ξ

-

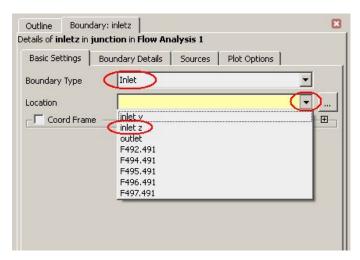
-

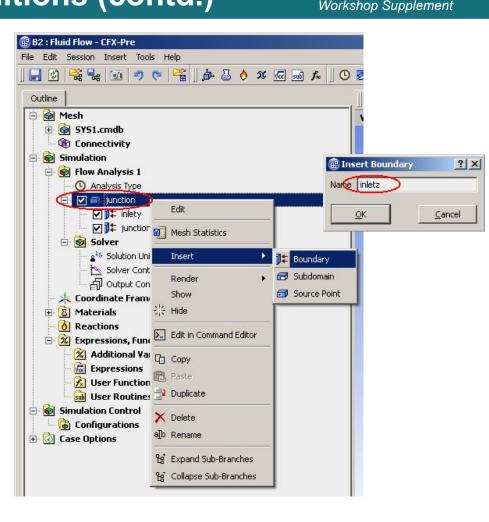
-

-

WS1: Mixing T-Junction CFX-Pre – Inlet boundary conditions (contd.)

- Right-click on the *junction* domain and select *Insert* > *Boundary*
- 2. Set the *Name* to *inletz* and click *OK*
- 3. Leave the *Boundary Type* field set to *Inlet*
- 4. Set *Location* to *inlet z*

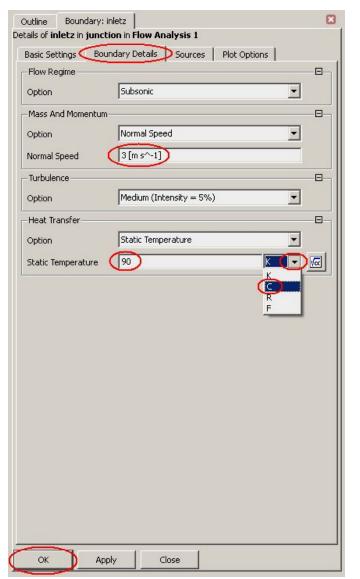




WS1: Mixing T-Junction CFX-Pre – Inlet boundary conditions (contd.)

This inlet will have an inlet speed of 3 m/s and temperature of 90°C.

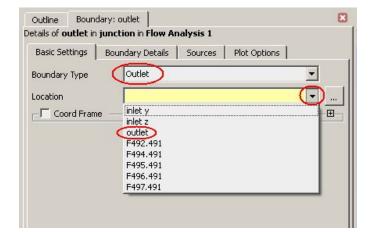
- 5. Click the Boundary Details tab
- 6. Enter a Normal Speed of 3 [m s^-1]
- 7. Set the *Static Temperature* to 90 [C] (make sure the units are correct!)
- 8. Click OK





WS1: Mixing T-Junction CFX-Pre – Outlet boundary conditions

- 1. Insert a boundary named *outlet*
- 2. Set the Boundary Type to Outlet
- 3. Set Location to outlet
- 4. Click the Boundary Details tab
- 5. Set Relative Pressure to 0 [Pa]
 - This is relative to the domain Reference Pressure, which is 1 [atm]
- 6. Leave all other settings at their default values
- The Average Static Pressure boundary condition allows pressure to float locally on the boundary while preserving an specified average pressure. If "Pressure" had been chosen a fixed Pressure would be applied at every nodal location on the outlet boundary
- 7. Click OK



lasic Settings Bou	Indary Details Sources Plot Options	
Flow Regime		
Option	Subsonic	•
Mass And Momentum		
Option	Average Static Pressure	•
Relative Pressure (Pa	•
Pres. Profile Blend	0.05	
🗖 Implicit Pressure /	Averaging	
Pressure Averaging-	28	Ξ
Option	Average Over Whole Outlet	•



WS1: Mixing T-Junction CFX-Pre – Wall boundary conditions

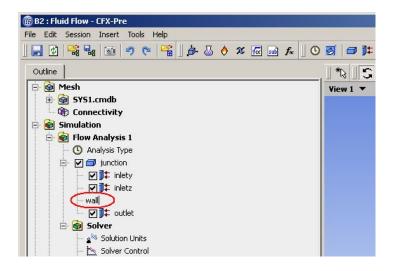


The default boundary condition (*junction Default* in this case) comprises of all the 2-D regions not yet assigned a boundary condition.

1. Right-click *junction Default,* select *Rename* and change the boundary name to *wall*

B2 : Fluid Flow - CFX-Pre	
File Edit Session Insert Tools Help	
0 🖈 📾 🔉 🚸 🕹 🍓 🗊 🤊 🍋 🗃	<u>s</u> =]‡ =
Outline	_* \ _S ↔
🖻 🖗 Mesh	View 1 V
🗄 🚱 SYS1.cmdb	
🔹 🌆 Connectivity	
🚊 🔞 Simulation	
📄 👰 Flow Analysis 1	
- 🕚 Analysis Type	
📄 🗹 🗇 junction	
- 🗹 🕽 🏝 inlety	
🛛 🗹 🕅 🚺 junction Default	
Edit	
🖻 🗑 Solver	
📲 📲 Solution Units	
Solver Control Render +	
에 Output Control Show	
Coordinate Frames	
A derials	
Reactions Edit in Command Editor	
Expressions, Functions and X Additional Variables Copy	
Expressions Duplicate Solutions	
user Routines	
Simulation Control	
Configurations	
⊕ 🐼 Case Options	

• The default boundary type is an adiabatic wall and is appropriate here

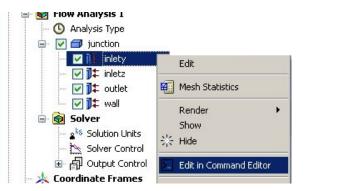


WS1: Mixing T-Junction CCL at a Glance

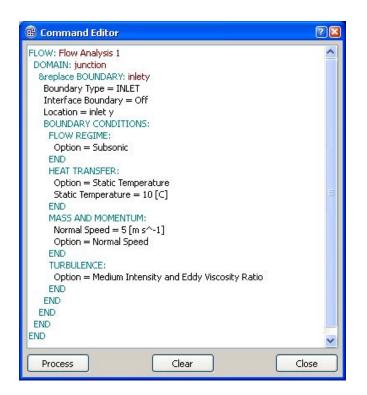


Before proceeding you will now take a quick look at CCL (CFX Command Language). CCL describes objects in a command language format. You will come across CCL in all CFX modules. Among other things, CCL allows you to perform batch processing and scripting.

1. Right-click on *inlety* and select *Edit in Command Editor*



2. Close the *Command Editor* after taking a quick look at the CCL definition of the Inlet boundary condition





Initial values must be provided for all solved variables. This gives the solver a starting point for the solution. There are two options when setting an initial value for a variable:

- Automatic: This will use a previous solution if provided, otherwise the solver will generate an initial guess based on the boundary conditions
- Automatic with Value: This will use a previous solution if provided, otherwise the value you specify will be used



The solver generated initial conditions are often good enough as a starting point. However, in some cases you will need to provide a better starting point to avoid solver failure

Initial conditions can be set on a per-domain basis, or on a global basis.

1.Since you will use Automatic Initial Conditions, there is no need to set any values, but click the *Initialisation* icon settings, and then close the form



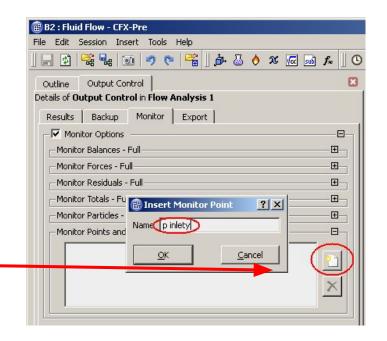
The Solver Control options set various parameters that are used by the solver and can affect the accuracy of the results. The default settings are reasonable, but will not be correct for all simulations. In this case the default settings will be used, but you will still look at what those defaults are.

- 1. Double-click on Solver Control from the Outline tree
 - The solver will stop after *Max. Iterations* regardless of the convergence level
 - Advection Scheme and Timescale Control will be discussed later
 - Residuals are a measure of how well the posed equations have been solved. In this case the solver will stop when the RMS (Root Mean Squared) residuals have reached 1.E-4. Tighter convergence is achieved with lower residuals.
- 2. Click Close

WS1: Mixing T-Junction CFX-Pre – Monitor points

In all engineering flows, there are specific variables or quantities of interest. Sometimes, these establish themselves in a different way from other variables and do not reach a satisfactory value at the same time as the overall solution converges, so it is always a good idea to monitor them as the solution progresses. In this simulation, pressure will be monitored at both inlets.

- 1. Double-click *Output Control* from the Outline tree
- 2. On the *Output Control* form, select the *Monitor* tab
- 3. Check the *Monitor Options* box
- 4. Click the *New* icon
- 5. Set the Name to *p* inlety and click *OK*





WS1: Mixing T-Junction

CFX-Pre – Monitor points (continued)

An expression will be used to define the monitor point.

- 7. Set Option to Expression
- 8. Enter the expression: **areaAve (Pressure)**@inlety in the *Expression Value* field

The expression calculates the area weighted average of pressure at the boundary *inlety*.

Note that expressions and expression language will be covered in more detail elsewhere.



Outline	Output Cor	ntrol			×
Details of O	utput Contr	ol in Flow	Analysis 1		
Results	Backup	Monitor	Export		
	itor Options	â	10 0		
Monito	or Balances - I	Full			
Monito	or Forces - Fu	II			
Monito	or Residuals -	Full			
Monito	or Totals - Full				
Monito	or Particles - F	-ull			
Monito	or Points and	Expression	s		-8-
	on ession Value	area	ression BAve(PressL	ıre)@inlety>	
ОК	Coord Frame		Close		



A second monitor point will be used to monitor the pressure at the second inlet, *inletz*.

- 9. Click the New icon
- 10. Set the *Name* to *p* inletz and click *OK*

😰 B2 : Fluid Flow - CFX-Pre
File Edit Session Insert Tools Help
] 🖬 🕸 🚟 🔩 💿 🦻 降 🚟 🛛 🏚 🐷 🔶 🕸 🖾 🖈 🖉
Outline Output Control
Details of Output Control in Flow Analysis 1
Results Backup Monitor Export
Monitor Options
Monitor Balances - Full
Monitor Forces - Full
Monitor Residuals - Full
-Monitor Totals - Full
- Monitor Particles - Full
Monitor Points and Expressions
p inlety
🔐 Insert Monitor Point 🔗 🗙
Name pinletz
QK <u>C</u> ancel

WS1: Mixing T-Junction

CFX-Pre – Monitor points (continued)

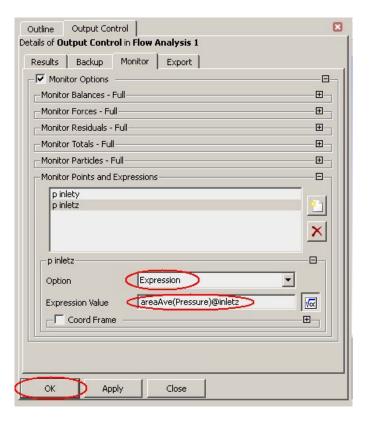
Workshop Supplement

An expression will be used to define the monitor point:

- 12. Set Option to Expression
- 13. Enter the expression areaAve (Pressure) @inletz in the Expression Value field
- 14. Click *OK* to apply the settings and close the Output Control form

The expression calculates the area weighted average of pressure at the boundary 'inletz'.

These monitor points will be utilised during the solution process in a later part of this tutorial.



WS1: Mixing T-Junction Solution Goals

Workshop Supplement

- Launch CFX-Pre from Workbench.
- Use pre-defined materials.
- Define the fluid models in a domain.
- Create and edit objects in CFX-Pre.
- Define boundary conditions.
- Set up monitor points using simple expressions.
- Launch the CFX Solver Manager from Workbench.
- Monitor convergence.

- Launch CFD-Post from an existing CFX Simulation in Workbench.
- Rotate, zoom and pan the view.
- Create contour plots.
- Create a plane for use as a locator.
- Create a velocity vector plot.
- Use pre-defined views.
- Create streamlines of velocity.
- Create an isosurface, coloured by a separate variable.

WS1: Mixing T-Junction Obtaining a solution

- 1. Exit CFX-Pre
 - When running in WB the CFX-Pre case will be saved automatically
- 2. Save the Workbench project
- 3. In Workbench, double-click *Solution* to launch the CFX Solver Manager

File View Tools Units Help	
📄 New 📝 Open 📕 Save 📓 Save A	s 🗟 Reconnect 🖉 Refres
foolbox 🗕 🗙 Pre	oject Schematic
Analysis Systems	
Component Systems	1221
🔏 BladeGen	▼ A
CFX	1 🕖 CFX
Engineering Data	2 🍓 Setup 🗸 🖌
🔝 Explicit Dynamics (LS-DYNA Expor	3 🕼 Solution 🏾 🖉 🔒
🔞 Finite Element Modeler	4 🔗 Results 💡
FLUENT	
🕅 Geometry	CFX
🌍 Mechanical Model	
🍘 Mesh	
💿 Results	
😚 TurboGrid	
🚽 Vista TF	
Custom Systems	
Design Exploration	



WS1: Mixing T-Junction Obtaining a solution (continued)

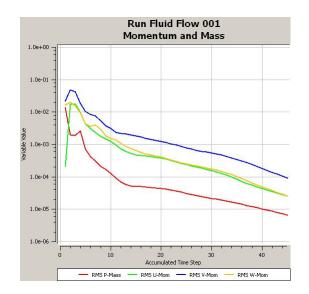


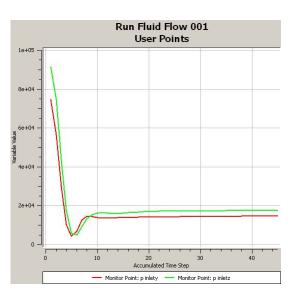
The CFX Solver Manager will start with the simulation ready to run.

3. Click Start Run to begin the solution process

Define Run		?
iolver Input File - Global Run Setting	_files\dp0\CFX\CFX\Fluid Flow.do	ef 🕜 😰
Run Definition	Partitioner Solver Interpo	lator
Initialization Optio	On Current Solution Data (if pos	sible) 💌
Type of Run Parallel Environ	Full	
Run Mode	Serial	-
Partition Weight	ing mode is set to Automatic.	
-Run Environme Working Directo	- nt	- 3
I Show Advanc	ed Controls	

- 45 iterations are required to reduce the RMS residuals to below the target of 1.0x10⁻⁴
- The pressure monitor points approach steady values





WS1: Mixing T-Junction Post-processing Goals

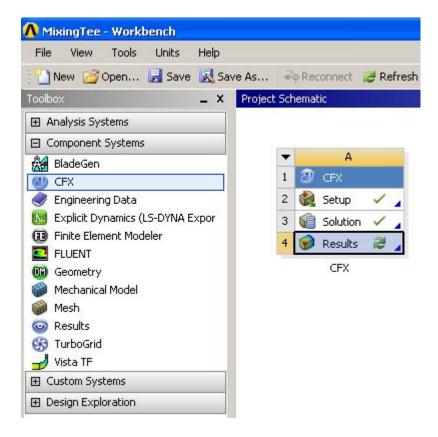


- Launch CFX-Pre from Workbench.
- Use pre-defined materials.
- Define the fluid models in a domain.
- Create and edit objects in CFX-Pre.
- Define boundary conditions.
- Set up monitor points using simple expressions.
- Launch the CFX solver manager from Workbench.
- Monitor convergence.

- Launch CFD-Post from an existing CFX Simulation in Workbench.
- Rotate, zoom and pan the view.
- Create contour plots.
- Create a plane for use as a locator.
- Create a velocity vector plot.
- Use pre-defined views.
- Create streamlines of velocity.
- Create an isosurface, coloured by a separate variable.

WS1: Mixing T-Junction Launching CFD-Post

- 1. Exit the CFX Solver Manager
- 2. Save the project
- 3. Double click *Results* to launch CFD-Post





Workshop Supplement

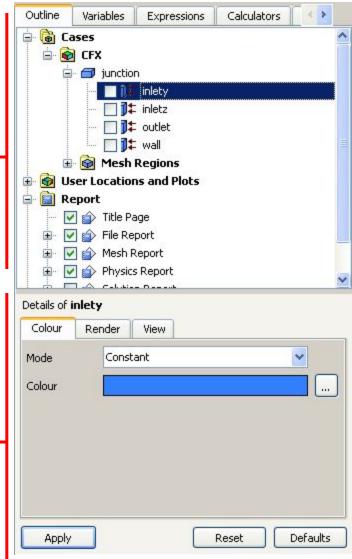
When CFD-Post opens, you will see that the layout is similar to CFX-Pre

There are two windows on the left side:

- Selector Window
 - Lists currently defined graphics objects.
 Object for each boundary condition are created automatically
 - Object are edited by double-clicking or right-clicking on the object
 - The check boxes next to each object turn the visibility on or off in the Viewer

Details Window

 When you edit an object the Details window shows the current object status



Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down Ctrl and the middle mouse button Adjusts the zoom level as you drag with the mouse vertically. Alternatively, you can zoom the view by holding down Shift and the middle mouse button. Zooms to the area enclosed in a box that you create by dragging with the mouse. Alternatively, you can drag and zoom the view by holding down the right mouse button. 0.150 0.300 (m) 0 075 0.225 3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer WS1-32

- When the results are loaded, CFD-Post displays the outline (wireframe) of the model
- The icons on the viewer toolbar control how the mouse manipulates the view
- Outline Variables Expressions Calculators Turbo 🖻 🗟 Cases -View 1 🔻 🖻 📦 CFX 🖻 🗇 junction TI 1 inlety □ 1 inletz 1 outlet 🔲 🚺 🗱 wall Mesh Regions

🔞 Location - 🤹 👩 📚 🔊 🍻 📑 🔛 🕶 💿 🕱 🚾 🏢 🖄 🖸

CFD-Post – Manipulating the view



- 0 ×

ANSYS

ANSYS, Inc. Proprietary © 2009 ANSYS, Inc. All rights reserved.

S

+‡+

•

Ð

WS1: Mixing T-Junction

B4 : Fluid Flow - CFD-Post

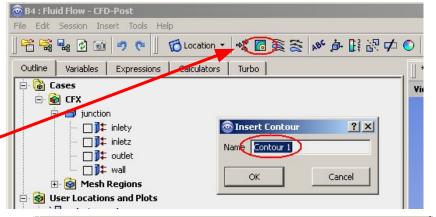
File Edit Session Insert Tools Help 뚵 🕰 🔩 🙆 🗿 🤊 (?)

CFD-Post – Temperature contour plot

In the first step, you will plot contours of temperature on the exterior walls of the model

- Click the Contour icon from the toolbar
- 2. Click OK to accept the default name Contour 1
- 3. Set Locations to wall

WS1: Mixing T-Junction



Details of Cont	our 1	
Geometry	Labels Render View	
Domains	All Domains	·
Locations	wall	
Variable	inlety inletz outlet	
Range 🤇	wal	
Min		283.096 [K]
Max		363.777 [K]
Boundary Dal	ta 💽 Hybrid	C Conservative
Colour Scale	Linear	•
Colour Map	Default Colour Map	
# of Contours	11	* *
📕 Clip to Ran	ge	



WS1: Mixing T-Junction CFD-Post – Temperature contour plot (contd.)

- 4. Set the Variable to Temperature
 - The drop-down menu provides a list of common variables. Use the "..." icon to access a full list
- 5. Leave the other settings unchanged
- 6. Click *Apply* to generate the plot

omains 🖡	All Domains	▼
ocations	vall	
	-	
Variable	Temperature	
Range	Eddy Viscosity	-
Min 🤇	Temperature Contractory Total Pressure	
Max	Total Temperature Turbulence Eddy Dissipation	
Boundary Data	Turbulence Kinetic Energy	
boundary Data	Velocity	_
Colour Scale	Velocity u Velocity v	
Colour Map	Velocity w	
of Contours	11	-
Clip to Range		
Clip to Kange		
Apply		Reset Defaults

etails of Contour 1 Geometry Labels	Render View		
Domains All Do	omains	•	
ocations wall	>	•	
Variable Tem	perature	•	
Range Glot	pal	-	
Min		283.096 [K]	
Max		363.777 [K]	
Boundary Data	Hybrid	C Conservative	
Colour Scale Line	ar	•	
Colour Map Def	ault Colour Map		B
# of Contours 11		*	1
Clip to Range			
Apply		Reset Defa	aults

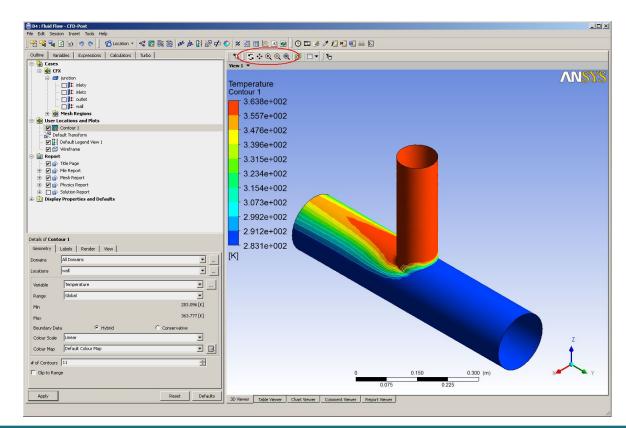


WS1: Mixing T-Junction <u>CFD-Post -</u> Temperature contour plot (contd.)

Workshop Supplement

A temperature contour plot on the walls should now be visible.

- 7. Try changing the view using rotate, zoom and pan. You may find it easier to use the middle mouse button in combination with <Ctrl> and <Shift>
- 8. Also try clicking on the axes in the bottom right corner of the Viewer



WS1: Mixing T-Junction CFD-Post

You can create many different objects in CFD-Post. The *Insert* menu shows a full list, but there are toolbar shortcuts for all items. Some common object are:

- Location: Points, Lines, Planes, Surfaces, Volumes
- Vector Plots
- Contour Plots
- Streamline Plots
- Particle Track (if enabled in CFX-Pre)

For turbo machinery cases there are additional objects available that will be discussed later.



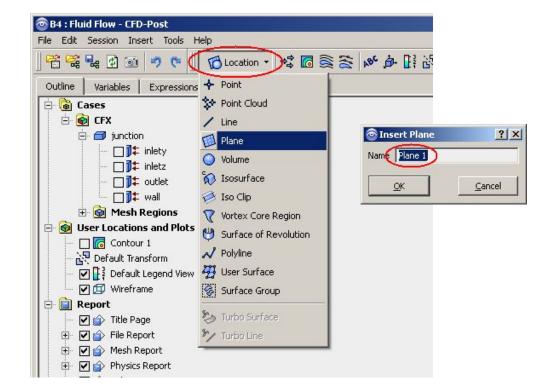


ANSYS, Inc. Proprietary © 2009 ANSYS, Inc. All rights reserved.

WS1: Mixing T-Junction CFD-Post – Creating a plane at x = 0

1. First, hide the previously created contour plot, by un-checking the associated box in the tree view

- 2. Click the *Location* button on the toolbar and select *Plane* from the drop-down menu
- 3. Click *OK*, accepting the default name of *Plane 1*



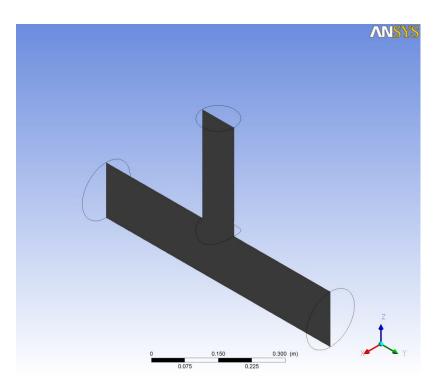




WS1: Mixing T-Junction CFD-Post – Creating a plane at x = 0 (contd.)

- 4. Set Method to YZ Plane
- 5. Leave *X* set to *0* [*m*]
- 6. Click *Apply* to generate the plane

Geometry Colour Render View	•
Definition	
Method YZ Plane	•
x 0.0[m]	
Plane Bounds - None	
Plane Type - Slice	— Đ —
	Defaults





While planes can be coloured by variables, in this case the plane will be used only as a locator for a

1. Hide the plane by un-checking the associated box in the tree view

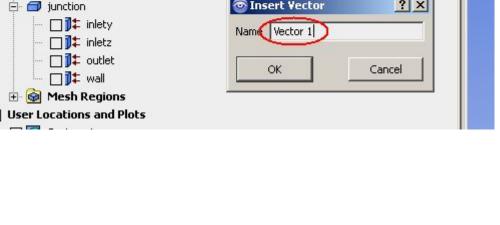
CFD-Post – Creating a velocity vector plot

2. Click the *Vector* icon from the toolbar

WS1: Mixing T-Junction

vector plot.

Click OK, accepting the 3. default name of Vector 1



Turbo

Insert Vector

Calculators





🔞 Location 📢 🤹 📓 📚 📚 🔊 🍻 📑 認 ⊄ 📀

💿 B4 : Fluid Flow - CFD-Post

🔩 🖗

Variables

6

Outline

🖻 🐻 Cases

🖻 🏟 CFX

File Edit Session Insert Tools Help

01

1 12

Expressions

View

? X

WS1: Mixing T-Junction CFD-Post – Velocity vector plot (continued)

- 4. Set *Locations* to *Plane* 1
- 5. Leave the *Variable* field set to *Velocity*
- 6. Click Apply

	Velocity
Details of Vector 1	Velocity Vector 1
Geometry Colour Symbol Render View	7.938e+000
Domains All Domains	
Definition	- 5.954e+000
Locations Plane 1	
Sampling Vertex	3.969e+000
Reduction Factor	
Factor 1.0	1.985e+000
Variable Velocity	
Boundary Data Hybrid C Conservative	0.000e+000
Projection None	[m s^-1]
Apply Reset Defaults	
	0 0.150 0.300 (m) Y Y

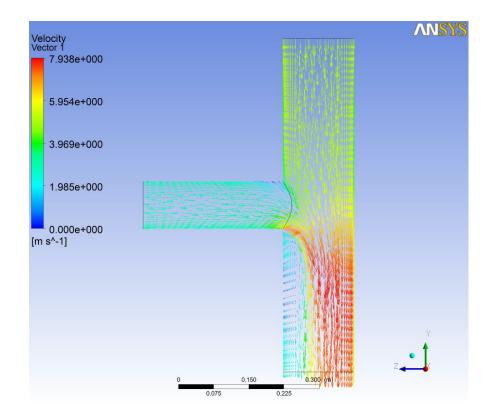
Workshop Supplement

WS1: Mixing T-Junction CFD-Post – Aligning the view



Given that the vector plot is on a 2-D Y-Z plane, you might want to view the plot normal to that axis (i.e. aligned with the X axis).

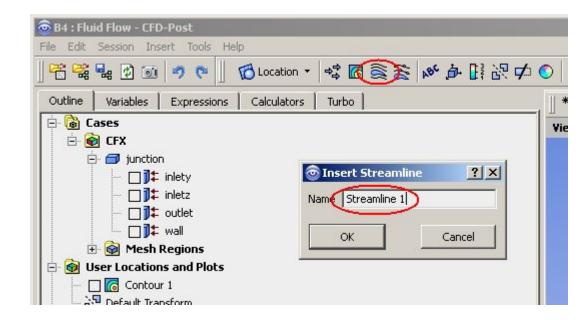
7. Click on the red x-axis in the bottom right corner of the Viewer to orientate the view



WS1: Mixing T-Junction **CFD-Post – Creating velocity streamlines**

Hide the previously created vector plot, by 1. un-checking the associated box in the tree view

- 2. Click the Streamline icon from the toolbar
- Click OK, accepting the 3. default name of Streamline 1







WS1: Mixing T-Junction CFD-Post – Velocity streamlines (continued)



- 4. In the *Start From* field, select both *inlety* <u>and</u> *inletz*. Use the '...' icon to the right of the field and select both locations using the CTRL key.
- 5. Leave the *Variable* field set to *Velocity*

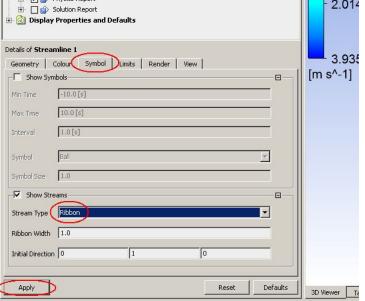
?]:
ots

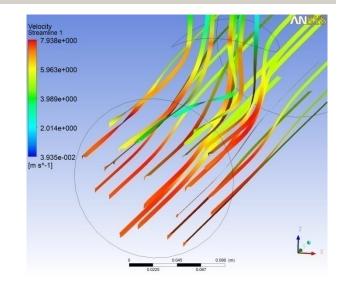
CFD-Post – Velocity streamlines (continued)

6. Click the Symbol tab

WS1: Mixing T-Junction

- 7. Change the Stream Type to Ribbon
- 8. Click Apply
- 9. Examine the streamlines from different views using rotate, zoom and pan
 - The ribbons give a 3-D representation of the flow direction
 - Their colour indicates the velocity magnitude
 - Velocity streamlines may be coloured using other variables e.g. temperature





April 28, 2009

Inventory #002599

ANSYS, Inc. Proprietary © 2009 ANSYS, Inc. All rights reserved.

2.

on the toolbar and select Outline Variables Expressions *Isosurface* from the 🖻 🗋 Cases 🖻 📦 CFX drop-down menu 🖻 🗇 junction 🗂 🚺 inlety

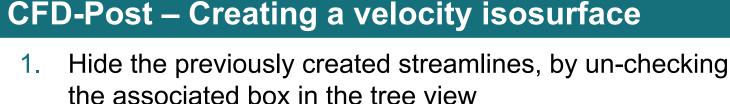
💿 B4 : Fluid Flow - CFD-Post

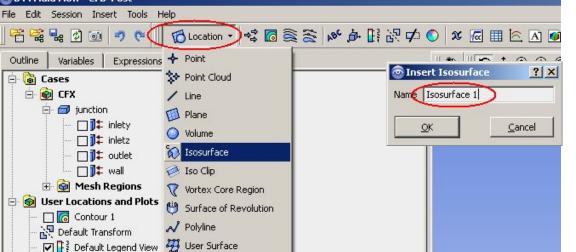
诺 👯 🔩 🚺 🔟

Click OK, accepting the 3. default name of Isosurface 1

Click the *Location* button

WS1: Mixing T-Junction







Inventory #002599

1 1 inletz

T1 toutlet

Surface Group

🏷 Turbo Surface

Y Turbo Line

TI wall

🗄 🚱 Mesh Regions

Contour 1

Plane 1

Vector 1

Report

Default Transform

□ 📚 Streamline 1

Plane 1

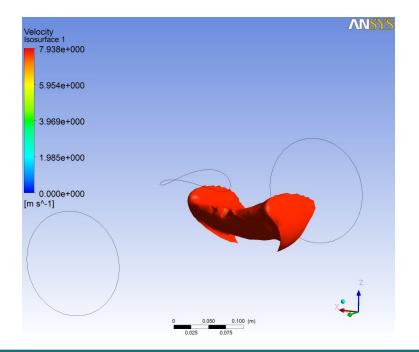
Streamline

Vector 1

WS1: Mixing T-Junction CFD-Post – Velocity isosurface (continued)

- 4. Set the *Variable* to *Velocity* (magnitude used in this context)
- 5. Enter a value of 7.7 [*m* s^-1] in the Value field (note: there is nothing special about this value other values can be tried)
- 6. Click Apply
 - The speed is > 7.7 m/s inside the isosurface and < 7.7 m/s outside.
 Isosurfaces in general are useful for showing pockets of highest velocity, temperature, turbulence, etc.

	▼
	•
nservative	
m s^-1	- 🕢
	iservative ⊨ m s^-1



Workshop Supplement

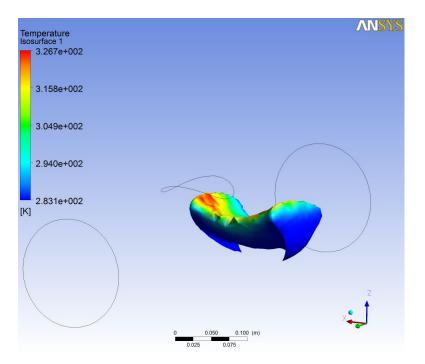
WS1: Mixing T-Junction CFD-Post – Velocity isosurface (continued)



By default, an isosurface is coloured by the variable used to create it (speed in this case), but a different variable can be used.

- 7. Click the Colour tab
- 8.Set the Mode to Temperature
- 9.Set the Range to Local

10.Click Apply



ariable Tempera	ture		<u></u>
ange Local		283.12	<u>с</u> 6[K]
lax		326.	
oundary Data	Hybrid	C Conservative	
olour Scale Linear			•
olour Map Default (Colour Map		·
def. Colour			