# Tutorial 12. Cold Flow Simulation Inside an SI Engine

#### Introduction

The purpose of this tutorial is to illustrate the case setup and solution of the two dimensional, four stroke spark ignition (SI) engine with port injection.

SI engines are of extreme importance to the auto industry. The efficiency of an SI engine depends on several complicated processes including induction, mixture preparation, combustion, and exhaust flow. CFD analysis has been used extensively to improve each of these processes. This tutorial simulates the intake, compression, expansion, and exhaust processes without fuel combustion. Port injection is modeled and evaporation of fuel droplets is simulated. The interaction of the fuel spray with the intake valve is modeled through the wall film modeling features available in FLUENT.

This tutorial demonstrates how to do the following:

- Use of the In-Cylinder model for simulating reciprocating engines.
- Use general strategies for modeling valve opening and closing.
- Use of the Discrete Phase Model (DPM) for simulating port injection.
- Carry out solver setup and perform iterations.
- Examine the results.
- Display and create animation for droplet injection.

#### **Prerequisites**

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

#### **Problem Description**

The IC engine simulation is probably one of the most interesting engineering problems in the field of computational fluid dynamics. Port injection is used for efficient air/fuel mixing and fuel distribution in multi-cylinder engines.

In this tutorial, you will consider a two dimensional engine with inlet and exit valves. The engine is running at 2000 rpm. The intake, compression, expansion and exhaust processes

are simulated without considering fuel combustion. The port injection is modeled and evaporation of fuel droplets is included. The interaction of the fuel spray with the intake valve is modeled through the wall film modeling features available in FLUENT.



Figure 12.1: Problem Schematic

## Preparation

- 1. Copy the mesh file, In\_Cylinder.msh and the profile file, valve.prof to your working folder.
- 2. Start the 2D double (2ddp) precision version of FLUENT.

## **Setup and Solution**

## Step 1: Grid

1. Read the mesh file, In\_Cylinder.msh.

 $\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Case...}$ 

FLUENT reads the mesh file and reports the progress in the console window.

2. Check the grid.

Grid —→Check

This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Check the scale of the grid.

$$Grid \longrightarrow Scale...$$

💶 Scale Grid			×
Scale Factors	Unit Co	nversion	
× 1	Grid ¥	∕as Created In 🛒 👻	
Y 1	Chan	ge Length Units	
Domain Extents			
Xmin (m) -0.042	4267	Xmax (m) 0.04550841	
Ymin (m) -0.009	273255	Ymax (m) 0.04549225	
Scale	Jnscale	Close Help	

Check the domain extents to see if they correspond to the actual physical dimensions. Otherwise the grid has to be scaled with proper units.

4. Display the grid (Figure 12.2).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$ 

💶 Grid Display			X
Options Nodes Geges Faces Partitions Shrink Factor Fe	Edge Type • All • Feature • Outline eature Angle •	Surfaces intake-ob intake-seat intake-seat-ib intake-seat-ob intake-stem intake-valve-top intake_port_wall piston	
Surface Name P	Match	Surface Types axis clip-surf exhaust-fan fan	
Display	Colors	Close Help	

(a) Click Colors....

The Grid Colors panel opens.

💶 Grid Colors			
Options	Types		Colors
C Color by Type Color by ID Sample	far-field inlet interior outlet periodic symmetry axis wall free-surface internal	<	light red ight yellow magenta maroon orange pink red tan white yellow Y
Reset Colors	Close		Help

i. Select Color by ID in the Options list.

- ii. Close the  $\mathsf{Grid}\xspace$  panel.
- (b) Click  $\mathsf{Display}$  and close the  $\mathsf{Grid}$   $\mathsf{Display}$  panel.



Figure 12.2: Grid Display

It can be observed that the domain is divided into several fluid zones. A few zones are meshed with quadrilateral elements and the remaining zones are meshed with triangular elements. Further, the area above the valve has non-conformal interfaces. The purpose of such meshing and domain decomposition is to maximize the use of the layering method with the moving and deforming mesh (MDM) model.

# Step 2: Models

Define

The problem is to be solved as unsteady with turbulence effects considered.

→Solver…

1. Enable the unsteady time formulation.

Models

Solver	Formulation
<ul> <li>Pressure Based</li> <li>Density Based</li> </ul>	<ul> <li>Implicit</li> <li>Explicit</li> </ul>
Space	Time
<ul> <li>€ 2D</li> <li>○ Axisymmetric</li> </ul>	⊂ Steady ☞ Unsteady
C Axisymmetric Swirl	Transient Controls
	<ul> <li>Non-Iterative Time Advanceme</li> <li>Frozen Flux Formulation</li> </ul>
Velocity Formulation	Unsteady Formulation
<ul><li>Ĝ Absolute</li><li>Ĝ Relative</li></ul>	C Explicit C 1st-Order Implicit C 2nd-Order Implicit
Gradient Option	Porous Formulation
<ul> <li>Green-Gauss Cell Based</li> <li>Green-Gauss Node Based</li> <li>Least Squares Cell Based</li> </ul>	Superficial Velocity     Physical Velocity

- (a) Select Unsteady in the Time list.
- (b) Click  $\mathsf{OK}$  to close the  $\mathsf{Solver}$  panel.

2. Enable the k- $\epsilon$  turbulence model.

$Define \longrightarrow$	Models	→Viscous…
--------------------------	--------	-----------

Viscous Model	
Model C Inviscid Laminar Spalart-Allmaras (1 eqn) K-epsilon (2 eqn) Reynolds Stress (5 eqn) k-epsilon Model Standard RNG Realizable	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1
Near-Wall Treatment  Standard Wall Functions  Non-Equilibrium Wall Functions  Enhanced Wall Treatment  User-Defined Wall Functions	User-Defined Functions  Turbulent Viscosity  Prandtl Numbers  TKE Prandtl Number  TDR Prandtl Number  none  v v v v v v v v v v v v v v v v v v
OK	ancel Help

- (a) Select k-epsilon (2 eqn) in the Model list.
- (b) Retain the default settings for other parameters.
- (c) Click OK to close the Viscous Model panel.
- 3. Enable Energy Equation.

Define	$\longrightarrow$	Models	$\longrightarrow$ Energy
--------	-------------------	--------	--------------------------



- (a) Enable Energy Equation in the Energy list.
- (b) Click  $\mathsf{OK}$  to close the  $\mathsf{Energy}$  panel.

4. Enable chemical species transport.

$\square$	$\boxed{\text{Define}} \longrightarrow \boxed{\text{Models}} \longrightarrow \boxed{\text{Species}} \longrightarrow \boxed{\text{Transport & Reactions}} $	on
---	--	----

Model	Mixture Properties
Off     Species Transport	Mixture Material
© Non-Premixed Combustion © Premixed Combustion	Number of Volumetric Species 3
Composition PDF Transport	
Reactions	
🗆 Volumetric	
Options	-
✓ Inlet Diffusion	
Diffusion Energy Source	
Full Multicomponent Diffusion	
Thermal Diffusion	

- (a) Enable Species Transport in the Model list.
- (b) Retain the default settings for other parameters.
- (c) Click OK to close the Species Model panel.

An Information dialog box opens with the message 'Available material properties or methods have changed. Please confirm the property values before continuing'. As the species transport is enabled, mixture composition will be required. Mixture composition will be set in Step 3.

(d) Click OK to close the Information dialog box.

5. Define the discrete phase modeling parameter.

|--|

Discrete Phase Model	
Interaction	Particle Treatment
<ul> <li>✓ Interaction with Continuous Phase</li> <li>✓ Update DPM Sources Every Flow Iteration</li> <li>Number of Continuous Phase 5</li> <li>✓</li> </ul>	✓ Unsteady Particle Tracking Track with Fluid Flow Time Step Inject Particles at
	C Particle Time Step Fluid Flow Time Step Particle Time Step Size (s) 0.001 Number of Time Steps 1 ↓ Clear Particles
Options Thermophoretic Force Brownian Motion Saffman Lift Force Erosion/Accretion Saffware	ion .up Breakup Constants y0 Breakup Parcels 2
OK Injections	Cancel Help

- (a) Define the interphase interaction.
  - i. Enable Interaction with Continuous Phase in the Interaction list.

This will include the effects of the discrete phase trajectories on the continuous phase.

- ii. Enable Update DPM Sources Every Flow Iteration in the Interaction list.
- iii. Set the Number of Continuous Phase Iterations per DPM Iteration to 5.
- (b) Click the Physical Models tab.
  - i. Enable Droplet Collision and Droplet Breakup in the Spray Model list.
  - ii. Enable TAB in the Breakup Model group box.
  - iii. Retain the default value of 0 for y0 in the Breakup Constants group box. This parameter is the dimensionless droplet distortion at t = 0.
- (c) Click OK to close the Discrete Phase Model panel.

# **Step 3: Materials**

Define
--------

→Materials...

Name	Material Type		Order Materials By
mixture-template	mixture		▼ 🖲 Name
Chemical Formula	Fluent Mixture M	aterials	Chemical Formula
	mixture-template	•	▼ Fluent Database
	Mixture		User-Defined Database
	none		-
Properties			
Mi×ture Species	names	Edit	
Density (kg/m3)	ideal-gas	Edit	
Cp (j/kg-k)	mixing-la <del>w</del>	Edit	
Thermal Conductivity (w/m-k)	constant 8. 8454	Edit	

- 1. Select ideal-gas from the Density drop-down list in the Properties list.
- 2. Click Change/Create.
- 3. Copy the evaporating species properties.

n-heptane-liquid droplets will evaporate to c7h16 vapors. But this species is not available in the present mixture.

(a) Click Fluent Database... in the Materials panel.

Fluent Database Materials	
Fluent Fluid Materials n-butane (c4h10) n-heptane-liquid (c7h16 <l>) n-heptyl-radical (c7h15) n-heptyl-radical (c7h15) n-hexane-liquid (c6h14<l>) n-hexane-vapor (c6h14) Copy Materials from Case Dele</l></l>	Material Type fluid order Materials By Name Chemical Formula
Properties Density (kg/m3) Cp (j/kg-k)	constant View   4.25 constant View   2471
Thermal Conductivity (w/m-k) Viscosity (kg/m-s)	constant v View 0.0178 constant v View 7e-06
New Edit	Save Copy Close Help

- i. Select fluid from the Material Type drop-down list in the Fluent Database Materials panel.
- ii. Select n-heptane-vapor (c7h16) from the Fluent Fluid Materials list.
- iii. Click Copy and close the Fluent Database Materials panel.
- 4. Set the mixture composition.
  - (a) Select mixture from the Material Type drop-down list.
  - (b) Click Edit... next to the Mixture Species in the Properties list.

Species	
Mixture mixture-template	
Available Materials	Selected Species
nitrogen (n2) oxygen (o2) water-vapor (h2o)	c7h16 air
	Add Remove
Selected Site Species	Selected Solid Species
Add Remove	Add Remove
OK Ca	ncel Help

i. Select c7h16 in the Available Materials list and click Add in the Selected Species list.

- ii. Select the species one by one except c7h16 in the Selected Species list and click Remove.
- iii. Select air in the Available Materials list and click  $\mathsf{Add}.$
- iv. Click OK to close the Species panel.
- (c) Click Change/Create and close the Materials panel.

For cold flow simulation, fuel is injected in the air and vaporized. This does not change the concentration of species like  $O_2$  which constitute air. Therefore, you need not model the species constituting air. However, if you are interested in modeling fuel combustion, then you will have to include the species constituting air.

**Note:** The species should appear in the same order as shown in the Species panel.

# Step 4: Injection

In this step, you will define the characteristics of the fuel injection.

Define  $\longrightarrow$  Injections...

1. Click Create.

The Set Injection Properties panel opens.

Set Injection Properties			
Injection Name injection-0			
Injection Type Num group v 4	iber of Particle Streams		
Particle Type			Laws
🔿 Inert 📀 Droplet	C Combusting C	Multicomponent	Custom
Material Diameter	r Distribution Oxidi	izing Species	-
c7h16	Inzing species Prod	uct species	<b>_</b>
Point Properties Turbulent Disper	sion   Wet Combustion   C	Components UDF	Multiple Reactions
Stochastic Tracking	Cloud Tracking		
<ul> <li>✓ Discrete Random Walk Model</li> <li>✓ Random Eddy Lifetime</li> <li>Number of Tries</li> <li>1</li> <li>1</li> <li>✓</li> <li>Time Scale Constant</li> <li>Ø.15</li> </ul>	Cloud Model Min, Cloud Diameter (m 8 Max. Cloud Diameter (n 199999	<u>1</u> <u>m</u> ]	
	K File Cancel	Неір	

- (a) Select group in the Injection Type drop-down list.
- (b) Set the Number of Particle Streams to 4.

This option controls how many droplet parcels are introduced into the domain at every time step.

- (c) Enable Droplet in the Particle Type list.
- (d) Select n-heptane-liquid from the Material drop-down list.
- (e) Select rosin-rammler from the Diameter Distribution drop-down list.
- (f) Set the Point Properties for the injection.
- (g) Specify the following for each of the properties:

Parameter	Value of	Value of
	First Point	Last Point
X-Position (m)	0.0112	0.0113
Y-Position (m)	0.0394	0.0394
X-Velocity (m/s)	0.5	2
Y-Velocity (m/s)	-20	-20
Temperature (k)	310	310
Start Time (s)	0.005	-
Stop Time (s)	0.0111	-
Total Flow Rate (kg/s)	0.001958	-
Min. Diameter (m)	2e-5	-
Max. Diameter (m)	5e-5	-
Mean Diameter (m)	4e-5	-
Spread Parameter	4.5	-

In this problem, the injection begins at 0.005 s and stops at 0.0111 s. While all the other events like piston motion, valve opening and closing are defined in terms of the crank angle, FLUENT will repeat these events after every 720 degrees i.e., crank period. However, the injection event cannot be defined in terms of crank angle and hence, will not repeat periodically.

(h) Click the Turbulent Dispersion tab.

The lower half of the panel will change to show options for the turbulent dispersion model. These models will account for the turbulent dispersion of the droplets.

- i. Enable the Discrete Random Walk Model.
- ii. Retain the default value for Time Scale Constant.
- iii. Click OK to close the Set Injection Parameters panel.
- (i) Close the Injections panel.

## **Step 5: Boundary Conditions**

Define  $\longrightarrow$  Boundary Conditions...

1. Set the boundary condition for pressure inlet (intake).



- (a) Select intake from the Zone list.
- (b) Click Set....

Pressure Inlet	X
Zone Name Intake	
Momentum       Thermal       Radiation       Species       DPM       Multiphase       UDS         Gauge Total Pressure (pascal)       Image: Constant       Constant         Supersonic/Initial Gauge Pressure (pascal)       Image: Constant       Constant         Direction Specification Method       Normal to Boundary         Turbulence       Specification Method       Intensity and Hydraulic Diameter	- - - -
Turbulent Intensity (%) 1 Hydraulic Diameter (m) 9.96 OK Cancel Help	

- i. Retain the default values for Gauge Total Pressure and Supersonic/Initial Gauge Pressure.
- ii. Select Intensity and Hydraulic Diameter from the Specification Method drop down list.
- iii. Enter 1% for the Turbulence Intensity.
- iv. Enter 0.06 m for the Hydraulic Diameter.

- v. Click the Thermal tab.
- vi. Enter  $318~\mathrm{K}$  for the Total Temperature.
- vii. Click OK to close the Pressure Inlet panel.
- 2. Set the following conditions for the pressure-outlet (exhaust).

Pressure Outlet
Zone Name
exhaust
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) g constant
Backflow Direction Specification Method Normal to Boundary
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%) 1
Backflow Hydraulic Diameter (m) 0.072
OK Cancel Help

- (a) Select Intensity and Hydraulic Diameter from the Turbulence Specification Method drop down list.
- (b) Enter 1 % for Backflow Turbulent Intensity.
- (c) Enter 0.072 m for Backflow Hydraulic Diameter.
- (d) Click the Thermal tab.
- (e) Enter 318 K for Backflow Total Temperature.
- (f) Click OK to close the Pressure Outlet panel.
- 3. Set the following conditions for the wall (exhaust-ib).

💶 Wall	
Zone Name	
exnaust-10	
ex-ib	
Momentum Ther	nal Radiation Species DPM Multiphase UDS
Thermal Condition	s Temperature (k) 369 constant
• Temperature • Convection	Wall Thickness (m) g
C Radiation C Mixed	Heat Generation Rate (w/m3) g constant
Material Name aluminum	Edit
	OK Cancel Help

- (a) Click the Thermal tab.
- (b) Select Temperature in the Thermal Conditions group box.
- (c) Enter 360 K for Temperature.
- (d) Click  $\mathsf{OK}$  to close the Wall panel.
- 4. Copy exhaust-ib boundary conditions to all the walls.
  - (a) Click Copy... in the Boundary Conditions panel.



- (b) Select exhaust-ib in the From Zone list.
- (c) Select all the zones from the To Zones list.
- (d) Click Copy.

This will display a warning message, click OK to confirm the changes.

- (e) Close the Copy BCs panel.
- 5. Set the following conditions for wall (intake-ib).

💶 Wall	X
Zone Name	
intake-ib	
Adjacent Cell Zone	
jin-ib	
Momentum Thermal Radiation Species DPM	Multiphase UDS
Discrete Phase Model Conditions	
Boundary Cond. Type wall-film	
Film Model Parameters	
Number Of Splashed Drops 4	
ОК	ancel Help

- (a) Click the DPM tab.
- (b) Select wall-film from the Boundary Cond. Type drop-down list.
- (c) Retain the Number Of Splashed Drops at 4 in the Film Model Parameters group box.
- (d) Click OK to close the Wall panel.
- 6. Similarly define the boundary conditions for intake-ob wall.
- 7. Close the Boundary Conditions panel.

# Step 6: Grid Interfaces

In this step, you will create the grid interfaces between the cell zones.

 $\mathsf{Grid} \longrightarrow \mathsf{Interfaces...}$ 

Grid Interfaces		X
Grid Interface ex-inter	Interface Zone 1	Interface Zone 2
ex-inter	E = exhaust-interface-ib exhaust-seat-ib exhaust-seat-ob intake-interface-ib Boundary Zone 1	<pre>     = =     exhaust-interface-ib     exhaust-interface-ob     exhaust-seat-ob     intake-interface-ib     v Interface Wall Zone 1</pre>
Periodic     Coupled	Boundary Zone 2	Interface Wall Zone 2
Creat	e Delete List Close	Help

- 1. Select exhaust-seat-ob in the Interface Zone 1 list.
- 2. Select exhaust-seat-ib in the Interface Zone 2 list.
- 3. Enter ex-inter for the Grid Interface.
- 4. Click Create.
- 5. Similarly create the following interfaces:

Interface Zone 1	Interface Zone 2	Grid Interface
exhaust-interface-ob	exhaust-interface-ib	exhaust-ib
intake-seat-ob	intake-seat-ib	in-inter
intake-interface-ob	intake-interface-ib	intake-ib

6. Close the Grid Interfaces panel.

#### Step 7: Mesh Motion Setup

1. Enable dynamic mesh model and specify the associated parameter.

Define  $\longrightarrow$  Dynamic Mesh  $\longrightarrow$  Parameters...

- (a) Enable Dynamic Mesh in the Models list.
- (b) Enable In-Cylinder in the Models list.

Enabling the In-Cylinder option allows input for IC-specific needs, including valve and piston motion.

- (c) Enable Smoothing, Layering, and Remeshing in the Mesh Methods group box.
- (d) Click the Smoothing tab.

💶 Dynamic Mesh Parar	neters 🛛 🗙
Models          Image: Dynamic Mesh         Image: In-Cylinder         125D         Six DOF Solver         Mesh Methods         Image: Smoothing         Image: Layering         Image: Remeshing	Smoothing Layering Remeshing In-Cylinder Six DOF Solver Spring Constant Factor 0.9 Boundary Node Relaxation 0.2 Convergence Tolerance 0.001 Number of Iterations 20
	OK Cancel Help

(e) Specify the following parameters :

Parameter	Value
Spring Constant Factor	0.9
Boundary Node Relaxation	0.2

*Retain the* Convergence Tolerance *and* Number of Iterations *at* 0.001 *and* 20 *respectively.* 

- (f) Click the Layering tab.
  - i. Select Constant Ratio in the Options list.
  - ii. Specify the following properties:

Parameter	Value
Split Factor	0.4
Collapse Factor	0.4

- (g) Click the Remeshing tab.
  - i. Retain the default Must Improve Skewness option.

By default, the Size Function option is disabled and the Must Improve Skewness option is enabled.

ii. Specify the following properties:

Parameter	Value
Minimum Length Scale (m)	0.0008
Maximum Length Scale (m)	0.0012
Maximum Cell Skewness	0.7
Size Remesh Interval	1

If a cell exceeds Minimum Length Scale or Maximum Length Scale limits, the cell is marked for remeshing. Hence, you need to specify problem-specific values for these remeshing parameters.

The Mesh Scale Info panel displays the values for minimum length scale, maximum length scale and maximum cell skewness, obtained from the initial mesh.

A value of 0.6 to 0.7 is recommended for Maximum Cell Skewness for 2D problems. Smaller values of maximum skewness results in improved grid quality at increased computational cost.

- (h) Click the In-Cylinder tab.
  - i. Specify the following properties:

Parameter	Value
Crank Shaft Speed (rpm)	2000
Starting Crank Angle (deg)	360
Crank Period (deg)	720
Crank Angle Step Size (deg)	0.5
Piston Stroke (m)	0.09
Connecting Rod Length (m)	0.15
Piston Stroke Cutoff (m)	0
Minimum Valve Lift (m)	0

(i) Click OK to close the Dynamic Mesh paramters panel.

The In-Cylinder model is specifically used for modeling Internal Combustion Engines. It facilitates the modeling of the dynamic mesh motion of piston and valves, in terms of crank shaft angle, crank speed, piston stroke, and connecting rod length. Further, the solution is advanced in terms of crank angle, specified against crank angle step size.

The piston is currently at the top dead center (TDC). The TDC position is defined by 0, 360, 720... degree crank angles, while the bottom dead center (BDC) position is defined by 180, 540, 900... degree crank angles.

A value of 720 degrees is used for four-stroke engines, while a value of 360 degrees is used for two-stroke engines. This governs the periodicity associated with valve events and valve lift profiles.

2. Read the profile file to be used for valve motion specification.



- (a) Select valve.prof and click OK.
- (b) Plot the piston motion profile using text commands:

You may need to press the <Enter> key to get the > prompt.





Figure 12.3: Piston Motion Profile

3. Specify the motion of piston, valves and other moving zones.

Define → [	Dynamic Mesh	—→Zone…
------------	--------------	---------

Dynamic Mesh Zones			
Zone Names Dynamic Zones cylinder-tri			
Type C Stationary C Rigid Body C Deforming C User-Defined			
Motion Attributes Geometry Definition Meshing Options			
Methods       Image: Smoothing       Image: Remeshing   Zone Parameters			
Minimum Length Scale (m) 0.0009			
Maximum Length Scale (m) 0.0011 Maximum Skewness 0.6			
Zone Scale Info			
Create Draw Delete Update Close Help			

- (a) Specify the motion and other parameters for cylinder-tri zone.
  - i. Select cylinder-tri from the Zone Names drop-down list.
  - ii. Select Deforming in the Type list.
  - iii. Click the Meshing Options tab.
    - A. Enable Smoothing and Remeshing in the Methods list.
    - B. Enter 0.0009 m for Minimum Length Scale, 0.0011 m for Maximum Length Scale and 0.6 for Maximum Cell Skewness in the Zone Parameters group box.
  - iv. Click Create.

(b) Specify the motion and other parameters for exhaust-seat-ib zone.

🗳 Dynamic Mesh Zones 🛛 🔀
Zone Names Dynamic Zones
exhaust-seat-ib  v cylinder-tri
Туре
C Stationary
C Rigid Body
© User-Defined
Motion Attributes Geometry Definition Meshing Ontions
Definition
Cylinder -
Cylinder Hadius (m)
Cylinder Origin Cylinder Axis
X (m) -0.02154253 X -0.2756375
Y (m) 0.009024297 Y 0.9612616
Create Draw Delete Update Close Help

- i. Select exhaust-seat-ib from the Zone Names drop-down list.
- ii. Select Deforming in the Type list.
- iii. Click the Geometry Definition tab.
  - A. Select cylinder from the Definition drop-down list.
  - B. Enter  $0.015~\mathrm{m}$  for the Cylinder Radius.
  - C. Enter  $-0.02154253~{\rm m}$  for X and  $0.009024297~{\rm m}$  for Y in the Cylinder Origin group box.
  - D. Enter -0.2756375 for X and  $\ 0.9612616$  for Y in the Cylinder Axis group box.
- iv. Click the Meshing Options tab.

🗳 Dynamic Mesh Zones 🛛 🛛 🔀			
Zone Names	Dynamic Zones		
exhaust-seat-ib	✓ cylinder-tri		
Type C Stationary Rigid Body Deforming User-Defined			
Motion Attributes	Geometry Definition Meshing Options		
Methods ↓ Smoothing ↓ Remeshing Zone Parameters Minimum Lengt Maximum Lengt Maximum	Smoothing Methods <sup>©</sup> Spring <sup>©</sup> Laplace Remeshing Methods <sup>™</sup> Region <sup>™</sup> Local Skewness Ø.6 Zone Scale Info		
Create Draw Delete Update Close Help			

- A. Enable Smoothing and Remeshing in the Methods list.
- B. Enable Spring in the Smoothing Methods group box.
- C. Select Region in the Remeshing Methods list.
- D. Enter 0.0005 m for Minimum Length Scale, 0.0009 m for Maximum Length Scale and 0.6 for Maximum Cell Skewness in the Zone Parameters list.
- v. Click Create.
- (c) Specify the motion and other parameters for intake-seat-ib zone.
  - i. Select intake-seat-ib from the Zone Names drop-down list.
  - ii. Select Deforming in the Type list.
  - iii. Click the Geometry Definition tab.
    - A. Select Cylinder from the Definition drop-down list.
    - B. Enter 0.018 m for the Cylinder Radius.
    - C. Enter 0.02065343 for X and 0.008345345 for Y in the Cylinder Origin list.
    - D. Enter 0.273957 for X and 0.961714 for Y in the Cylinder Axis list.
  - iv. Click the Meshing Options tab.
    - A. Enable Smoothing and Remeshing in the Methods list.
    - B. Enable Spring in the Smoothing Methods group box.

- C. Enable Region in the Remeshing Methods group box.
- D. Enter 0.0005 m for Minimum Length Scale, 0.0009 m for Maximum Length Scale and 0.6 for Maximum Cell Skewness in the Zone Parameters group box.
- v. Click Create.

The declaration of the deforming boundary zones is necessary only for boundary zones adjacent to the cell zones that need remeshing.

When you specify the cylinder geometry definition, the nodes on the zone selected will be projected onto the cylindrical wall with a specified radius and axis. In this case, the nodes lying on the interfaces, which connect the cylinder to the (intake or exhaust) port, will be projected onto the cylindrical wall generated by sweeping the valve area along the valve axis

- 4. Specify the motion of the Rigid Body zones.
  - (a) Specify the motion for the piston zone.

Dynamic Mesh Zones	X
Zone Names piston 🗸	Dynamic Zones cylinder-tri
Type Stationary Rigid Body Deforming User-Defined	exhaust-seat-ib intake-seat-ib
Motion Attributes Geometry Definiti	on Meshing Options
Motion UDF/Profile **piston-full**	Lift/Stroke (m) Ø
Valve/Piston Axis	
Y 1	
Create Draw Delete	Update Close Help

- i. Select piston from the Zone Names drop-down list.
- ii. Select Rigid Body from the Type list.
- iii. Click the Motion Attributes tab.
  - A. Select **\*\*piston-full\*\*** from the Motion UDF/Profile drop-down list.
  - B. Enter 0 for X and 1 for Y in the Valve/Piston Axis group box.
- iv. Click the Meshing Options tab.
  - A. Enter 0.001 m for Cell Height.
- v. Click Create.

Zone Names	Туре	Motion Attributes			Meshing op- tions (m)
		Motion	Motion Valve/Piston Axis		
		UDF/Profile			
			Х	Y	
ex-ib	Rigid Body	ex-valve	-0.275637	0.9612616	-
exhaust-ob	Rigid Body	ex-valve	-0.275637	0.9612616	0.0005
exhaust-	Rigid Body	ex-valve	-0.275637	0.9612616	0.001
valve-top					
in-ib	Rigid Body	in-valve	0.273959	0.961741	-
intake-ob	Rigid Body	in-valve	0.273959	0.961741	0.0005
intake-	Rigid Body	in-valve	0.273959	0.961741	0.001
valve-top					

(b) Similarly, create the following rigid body zones:

- 5. Specify the motion for the stationary zones.
  - (a) Specify the motion of the exhaust-interior-ib zone.
    - i. Select exhaust-interior-ib in the Zone Names drop-down list.
    - ii. Select Stationary in the Type list.
    - iii. Click the Meshing Options tab.
      - A. Enter 0.001 m for Cell Height in the ex-ib adjacent zone group box.
      - $B. \ Click \ Create.$
    - iv. Similarly create the following stationary zones:

Zone Names	Type	Meshing Options		
		For in-port Zone	For in-ib Zone Cell	
		Cell Height (m)	Height (m)	
intake-interior-ib	Stationary	0	0.001	

6. Close the Dynamic Mesh Zones panel.

By default, if no motion (moving or deforming) attributes are assigned to a face or cell zone, then the zone is not considered when updating the mesh to the next time step. However, in this case an explicit declaration of a stationary zone is required. Because interior adjacent cell zone (ex-ib and in-ib) are assigned solid body motion, the positions of all nodes belonging to these cell zones will be updated even though the nodes associated with the interiors are part of a non-moving boundary zone. An explicit declaration of a stationary zone excludes the nodes on these zones when updating the node positions.

7. Set the dynamic events such as valve opening and closing.

🗖 Dy	namic Mesh Events		X
Num	ber of Events 8		
On	Name	At Crank Angle (deg)	<u>^</u>
	ex-valve-open	120	Define
	in-valve-open	340	Define
	ex-valveclose	380	Define
	in-valveclose	600	Define
	activate-exhaust-port	119	Define
	deactivate-exhaust-port	381	Define
	activate-inlet-port	339	Define
	deactivate-inlet-port	601	Define
1	,		
	Apply Read Write Previ	ew Close	Help

- (a) Set the Number of Events to 8.
- (b) Enter ex-valve-open as the first name in the Name list.
- (c) Enable On for ex-valve-open.
- (d) Enter 120 deg for ex-valve-open in the At Crank Angle list.
- (e) Click the Define... button to open the Define Event panel.

Define Event		
Name		
ex-valve-open		
Туре		
Create Sliding Interface	-	
Definition		
Interface Name		
ex-inter		
Interface Zone 1	≡ = Interface Zone 2	II
intake-seat-ib	intake-seat-ib	~
exhaust-seat-ib	exhaust-seat-ib	
Intake-valve-top	Intake-valve-top	
cylinder side guad	cydinder side guad	
cylinder_side_tri	cylinder side tri	
cylinder_top	cylinder_top	~
Wall 1 Motion	Wall 2 Motion	
none	▼ none	•
ОК	Cancel Help	

i. Select Create Sliding Interface from the Type drop-down list.

- ii. Enter ex-inter as the Interface Name in the Definition group box.
- iii. Select exhaust-seat-ob in the Interface Zone 1 selection list.
- iv. Select exhaust-seat-ib in the Interface Zone 2 selection list
- v. Retain the default selection of none in the Wall 1 Motion and Wall 2 Motion drop-down lists.
- vi. Click  $\mathsf{OK}$  to close the  $\mathsf{Define}\xspace$  panel.
- (f) Similarly, create the following Dynamic Events:

Name	Crank	Setup description
	Angle	
in-valve-open	340 deg	1. Select Create Sliding Interface from the Type drop-down
		list.
		2. Enter in-inter as Interface Name in the Definition group
		box.
		3. Select intake-seat-ob in the Interface Zone 1 selection list.
		4. Select intake-seat-ib in the Interface Zone 2 selection list.
		5. Click OK.
ex-valve-	$380 \deg$	1. Select Delete Sliding Interface from the Type drop-down
close		list.
		2. Enter ex-inter as Interface Name in the Definition list.
		3. Click OK.
in-valve-close	$600 \deg$	1. Select Delete Sliding Interface from the Type drop-down
		list.
		2. Enter in-inter as the Interface Name in the Definition
		group box.
		3. Click OK.
activate-	$119 \deg$	1. Select Activate Cell Zone from the Type drop-down list.
exhaust-port		
		2. Select ex-ib and ex-port in the Definition list.
		3. Click OK.
deactivate-	$381 \deg$	1. Select Deactivate Cell Zone from the Type drop-down list.
exhaust-port		
		2. Select ex-ib and ex-port in the Definition list.
		3. Click OK.
activate-	$339 \deg$	1. Select Activate Cell Zone from the Type drop-down list.
inlet-port		
		2. Select in-ib and in-port in the Definition list.
		3. Click OK.
deactivate-	$601 \deg$	1. Select Deactivate Cell Zone from the Type drop-down list.
inlet-port		
		2. Select in-ib and in-port in the Definition list.
		3. Click OK.

- (g) Click Apply to save the changes.
- $(h)\ Close the Dynamic Mesh Events panel.$

Dynamic events are used to control the timing of specific events during the course of the simulation. With in-cylinder flows for example, you may want to open the exhaust valve (represented by a pair of deforming sliding interfaces) by creating an event to create the sliding interfaces at some crank angle. For the in-cylinder model, the dynamic events are crank angle-based, whereas by default, they are flow time-based.

When the inlet and exhaust values are closed, the flow and thermal conditions inside the inlet and exhaust port are not of our interest. During this period, these zones are deactivated to speed up the solution. Deactivated zones are not available for post-processing and hence, will not be displayed while creating the animations.

#### Step 8: Mesh Preview

1. Save the case file (In\_Cylinder.cas.gz).

 $\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case...}$ 

Since the mesh changes during the mesh preview, ensure that you save the case before displaying the mesh preview.

2. Display the grid.

Display  $\longrightarrow$  Grid...

- (a) Select all the surfaces in the Surfaces list.
- (b) Click Display.
- (c) Close the Grid Display panel.
- 3. Set up the mesh preview.

Solve  $\longrightarrow$  Mesh Motion...

🖴 Mesh Motion	X
Time	Display Options
Current Mesh Time (s) 0.06000001 Time Step Size (s) 4.166667e-05	<ul> <li>✓ Display Grid</li> <li>✓ Save Hardcopy</li> <li>✓ Enable Autosave</li> <li>✓ Display Frequency 1</li> </ul>
Preview Apply Cl	ose Help

The Time Step Size displayed in the read-only text field corresponds to 0.5 degree crank angle and is based on the crankshaft speed and crank angle increment parameters defined earlier.

(a) Enter 1440 for the Number of Time Steps.

This corresponds to four full revolutions of the crankshaft.

(b) Click Preview to preview the mesh motion.

As the mesh is updated by FLUENT, messages appear in the console window reporting the progress of the update.

(c) Close the Mesh Motion panel.

# **Step 9: Solution Setup**

1. Read the case file back into FLUENT (In\_Cylinder.cas.gz).

 $\fbox{File} \longrightarrow \fbox{Read} \longrightarrow \fbox{Case}...$ 

An Information dialog box opens with the message "Available material properties or methods have changed. Please confirm the property values before continuing". Click OK to close it.

2. Retain the default solution controls.

Solve $\longrightarrow$ Co	$ntrols \longrightarrow Solution$
----------------------------	-----------------------------------

Solution Controls		
Equations 📃 🗐	Under-Relaxation Factors	
Flow Turbulence	Pressure	0.3
c7h16 Energy	Density	1
	Body Forces	1
	Momentum	0.7 •
Pressure-Velocity Coupling	Discretization	_
SIMPLE	Pressure	Standard 🖌
	Density	First Order Upwind 🗸
	Momentum	First Order Upwind 👻
	Turbulent Kinetic Energy	First Order Upwind
0	Cancel	telp

(a) Click OK to close Solution Controls panel.

3. Initialize the flow field.

Solve $\longrightarrow$ Initialize $\longrightarrow$ Initialize.	
--	--

Solution Initialization	
Compute From Referen	ice Frame
▼	ative to Cell Zone solute
Initial Values	
Turbulent Kinetic Energy (m2/s2) 0.01	
Turbulent Dissipation Rate (m2/s3) Ø. 01	
c7h16 g	
Temperature (k) 318	
Init Reset Apply Close	Help

- The Gauge Pressure value is zero.
- (a) Enter 0 pascal for the Gauge Pressure.
- (b) Enter  $0~{\rm m/s}$  for X Velocity and Y Velocity.
- (c) Enter 0.01  $\mathrm{m}^2/\mathrm{s}^2$  for Turbulent Kinetic Energy.
- (d) Enter 0.01  $\mathrm{m^2/s^3}$  for Turbulent Dissipation Rate.
- (e) Enter 0 for c7h16.
- (f) Enter 318 K for Temperature.
- (g) Click Init and close the Solution Initialization panel.

4. Enable the plotting of residuals during the calculation.

Solve $\longrightarrow$	Monitors	—→Residual…
-------------------------	----------	-------------

🖳 Residual Mo	nitors -				×
Options	Storage		F	Plotting	
✓ Print ✓ Plot	Iterati	ions 1000		Wind	ow 0
	Normalization	ı		Iterations	100 🔶
	🗆 Nor	malize 🔽	Scale	Axes	Curves
	Convergence	Criterion			
	absolute		-		
Residual	Cheo Monitor Conv	ck / vergence(	Absolute Criteria	-	
continuity		▼	0.001	_	
x-velocity	V	▼	0.001		
y-velocity	V	✓	0.001		
energy		<b>v</b>	1e-06		
k		<b>v</b>	0.001	•	
OK Plot Renorm Cancel Help					

- (a) Enable Plot in the Option list.
- (b) Enter 100 for the lterations in the Plotting group box.

To avoid a cluttered residual plot in transient simulations, it is useful to display only the most recent iterations.

(c) Click OK to close the Residual Monitors panel.

5. Enable the writing of averaged pressure and temperature in the domain during the calculation by defining volume monitors.

Solve  $\longrightarrow$  Monitors  $\longrightarrow$  Volume...

💶 Volume Monit	ors						X
Volume Monitor	rs 2		▲ ▼				
Name	Plo	t Prin	it Write	e Eve	ry When		
vol-mon-1			◄	1	Time Step	▼ Define	
vol-mon-2				1	Time Step	✓ Define	
vol-mon-3			Γ	1	tteration	▼ Define	
vol-mon-4		Γ		1	▲ Iteration	✓ Define	•
			ОК	Can	cel Help		

- (a) Set the Volume Monitors to 2.
- (b) Enable Write for the first monitor (vol-mon-1).

When the Write option is enabled, the volume-averaged pressure history is written to a file. If you do not select the Write option, the history information will be lost when you exit FLUENT.

- (c) Select Time Step from the Every drop-down list.
- (d) Click Define... to define the monitor.

💶 Define Volume Monit	or 🛛 🛛
Name	Field Variable
Report Type	Pressure
Volume-Average 👻	Cell Zones
X Axis Flow Time 🔻	in-port in-ib ex-port
Plot Window	ex-ib cylinder-qurd cylinder-tri
File Name	
pressure.out	
OK CI	urves Axes Cancel Help

- i. Enter **pressure** in the Name field.
- ii. Select Volume-Average from the Report Type drop-down list.
- iii. Select Flow Time in the X Axis drop-down list.
- iv. Select Pressure... and Static Pressure from the Field Variable drop-down lists.
- v. Select cylinder-qurd and cylinder-tri in the Cell Zones list.
- vi. Enter pressure.out for the File Name.
- vii. Click OK to close the Define Volume Monitor panel.
- (e) Similarly, define the mass-averaged temperature monitor.
  - i. Select Time Step from the Every drop-down list.
  - ii. Click Define... to define the monitor.
    - A. Enter temperature in the Name field.
    - B. Select Mass-Average from the Report Type drop-down list.
    - C. Select Flow Time from the X Axis drop-down list.
    - D. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
    - E. Select cylinder-qurd and cylinder-tri in the Cell Zones list.
    - F. Enter temperature.out in the File Name.
    - G. Click  $\mathsf{OK}$  in the Define Volume Monitors panel.
- (f) Click  $\mathsf{OK}$  to close the  $\mathsf{Volume}$  Monitors panel.
- 6. Set up an animation for velocity,  $C_7H_{16}$  mole fraction and DPM injection.
  - (a) Display filled contours of velocity magnitude.

 $\mathsf{Display} \longrightarrow \mathsf{Contours}...$ 

- i. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- ii. Enable Filled in the Options list.
- iii. Click Display.
- iv. Use the mouse button to reposition the geometry as shown in the Figure 12.4.
  - **Note:** The piston is at TDC and during the solution; the computational domain will expand up to the BDC. Therefore leave sufficient space for domain expansion.
- v. Close the Contours panel.



Figure 12.4: Velocity Contours for Animation Setup

(b) Save the current view.

$$\boxed{\mathsf{Display}} \longrightarrow \mathsf{Views...}$$

Views	Actions	Mirror Planes 🔳
back front	Default	
	Auto Scale	
	Previous	
	Save	Define Plane
	Delete	Periodic Repeats
Sa∨e Name	Read	Define
view-0	Write	
,		1

- i. Click Save to save the current view as view-0.
- ii. Close the Views panel.
- (c) Set hardcopy settings.

 $\mathsf{File} \longrightarrow \mathsf{Hardcopy....}$ 

- i. Select TIFF in the Format group box.
- ii. Select Color in the Coloring group box.
- iii. Click Apply and close the  $\mathsf{Graphics}$  Hardcopy panel.

- (d) Specify the commands for animation.
  - Solve  $\longrightarrow$  Execute Commands...

💶 Exe	ecute Commands				X
Defined Commands 12					
	command-7	4	Time Step	▪ disp view res-view view-0	
	command-8	4	Time Step	▼ disp hard-copy "velocity-%t.tif"	
	command-9	4	Time Step	▼ disp set-window 3	
	command-10	4	Time Step	🚽 ack part-track part-dia , , 0.1e-6 50e-6	
	command-11	4	Time Step	▪ disp view res-view view-0	
	command-12	4	Time Step	▼ disp hard-copy "injection-%t.tif"	_
OK Define Macro Cancel Help					

- i. Set Defined Commands to 12.
- ii. Enable On for command-1.
- iii. Enter 4 for Every.
- iv. Select Time Step from the When drop-down list.
- v. Enter disp set-window 1 for the Command.
- vi. Repeat the steps ii. through v. and enter the following commands sequentially:

Name	Command
command-2	disp cont molef-c7h16 0 1e-3
command-3	disp view res-view view-0
command-4	disp hard-copy "species-%t.tif"
command-5	disp set-window 2
command-6	disp cont velo-mag 0 100
command-7	disp view res-view view-0
command-8	disp hard-copy "velocity-%t.tif"
command-9	disp set-window 3
command-10	disp part-track part-track part-dia , , 0.1e-6 50e-6
command-11	disp view res-view view-0
command-12	disp hard-copy "injection-%t.tif"

vii. Click OK to close the Execute Commands panel.

The above commands will first activate 'window n', restore the saved view 'view-0', display contours of velocity magnitude,  $C_7H_{16}$  mole fraction, DPM Injection and then make a hardcopy of the resulting image.

The '%t' appended to the file name instructs FLUENT to append the timestep index to the filename.

The TIFF files saved can then be used to create a movie. For the information on converting TIFF file to an animation file, refer to http://www.bakker.org/cfm/graphics01.htm.

7. Enable autosaving of case and data files.

For detailed postprocessing, save the case and data files after every 180 degree crank angle.

 $\boxed{\mathsf{File}} \longrightarrow \boxed{\mathsf{Write}} \longrightarrow \mathsf{Autosave}...$ 



- (a) Enter 360 for Autosave Case File Frequency.
- (b) Enter 360 for Autosave Data File Frequency.

Since the mesh changes during the simulation, you must save both the case and data files.

(c) Click OK.

When FLUENT saves a file, it appends the time step value to the file name prefix (In\_Cylinder). The standard extensions (.cas and .dat) are also appended.

8. Save the case and data file (In\_Cylinder.cas.gz).

$$\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$$

Click OK to overwrite the previously saved case file.

#### Step 10: Solution

1. Start the calculation.

Solve  $\longrightarrow$  Iterate...

💶 Iterate	×					
Time						
Time Step Size (s) 4.1667e-05						
Number of Time Steps 1449						
Time Stepping Method						
© Fixed © Adaptive © Variable						
Options	Options					
Data Sampling for Time Statistics						
Iteration						
Max Iterations per Time Step 40						
Reporting Interval 1						
UDF Profile Update Interval 1						
Iterate Apply Close Help						

- (a) Set the Number of Time Steps to 1440.
- (b) Set the Max Iterations per Time Step to 40.
- (c) Click Iterate.

During the solution, FLUENT will write the averaged pressure and temperature in the pressure.out and temperature.out files. These files can be read back in FLUENT for plotting.

2. Write the case and data files.

 $\boxed{\mathsf{File}} \longrightarrow \mathsf{Write}} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$ 

## Step 11: Postprocessing

1. Display static pressure and temperature variation.

 $\mathsf{Plot} \longrightarrow \mathsf{File...}$ 

File XY Plot		×		
Plot Title	Legend Title			
Files	legend Entries			
D:\Users\In-Cylinder\pressure.out	Convergence history of Sta			
		Add		
		Delete		
D:\Users\In-Cylinder\pressure.o	rd etc. (in SI units)	Change Legend Entry		
Plot Axes Curves Close Help				

- (a) Click Add... to add the file.
- (b) Select the pressure.out file and click OK.
- (c) Click Plot (Figure 12.5).



Figure 12.5: Convergence History of Static Pressure

- (d) Click Delete to remove the added file.
- (e) Similarly plot the file temperature.out for static temperature variation (Figure 12.6).



Figure 12.6: Convergence History of Static Temperature

- 2. Display filled contours of  $C_7H_{16}$  mass fraction at the 540 degree crank angle position (Figure 12.7).
  - (a) Read the files In\_Cylinder0360.cas.gz and In\_Cylinder0360.dat.gz back into FLUENT.

 $\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$ 

(b) Display filled contours of  $C_7H_{16}$  mass fraction (Figure 12.7).

Display  $\longrightarrow$  Contours...

- i. Select Species... and Mass fraction of c7h16 in the Contours of drop-down lists.
- ii. Click Display.



Figure 12.7: Predicted  $C_7H_{16}$  Mass fraction Distribution

- 3. Display filled contours static temperature at 720 degree crank position.
  - (a) Read the In\_Cylinder0720.cas.gz case and data files back into FLUENT.

$$\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Case} \And \mathsf{Data.}.$$

(b) Display filled contours static temperature (Figure 12.8).

Display  $\longrightarrow$  Contours...

- i. Select Temperature... and Static Temperature in the Contours of drop-down lists.
- ii. Click Display and close the Contours panel.



Figure 12.8: Predicted Static Temperature Distribution

## Summary

Use of In-Cylinder model capabilities has been illustrated for cold flow simulation inside the SI engine. All, suction, compression, expansion and exhaust strokes are simulated. The Discrete Phase Model is used for simulating fuel injection, evaporation, and droplet boiling.

## References

FLUENT 6.3 User's Guide:

 $http://www.fluentusers.com/fluent6326/doc/ori/html/ug/main_pre.htm$ 

## **Exercises/Discussions**

- 1. What will be the effect on fuel vaporization in each of the following situations:
  - (a) The inlet pressure is increased.
  - (b) The exhaust pressure is increased.
  - (c) The crank speed is increased.
  - (d) Valve timing diagram is changed.
- 2. What will be the effect on volumetric efficiency in each of the following situations:
  - (a) The inlet pressure is increased.
  - (b) The exhaust pressure is increased.
  - (c) The crank speed is increased.
  - (d) Valve timing diagram is changed.

# Links for Further Reading

- http://www.nasg.com/index-e.html
- http://www.aae.uiuc.edu/m-selig/
- http://airtrafficcont rol.no-ip.org:8080/airfoil.htm
- http://www.vzlu.cz/htmfile /HSaerodynamics.htm