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.

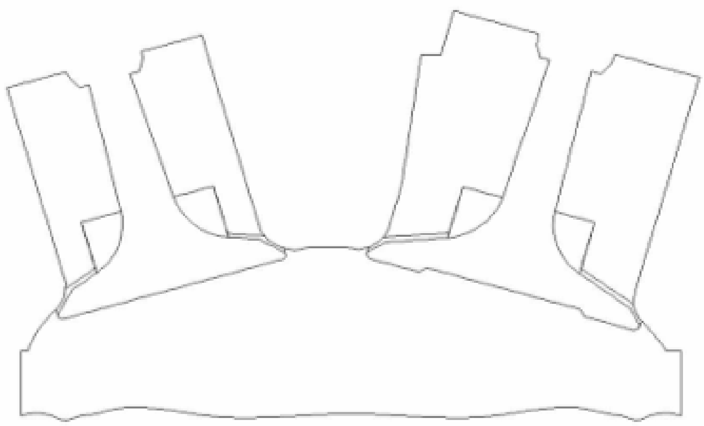


Figure 12.1: Problem Schematic

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`.

   File ⟶ Read ⟶ Case...

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

2. Check the grid.

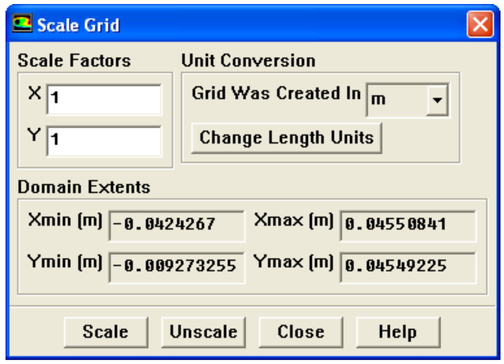
   Grid ⟶ Check

   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 ⟶ Scale...

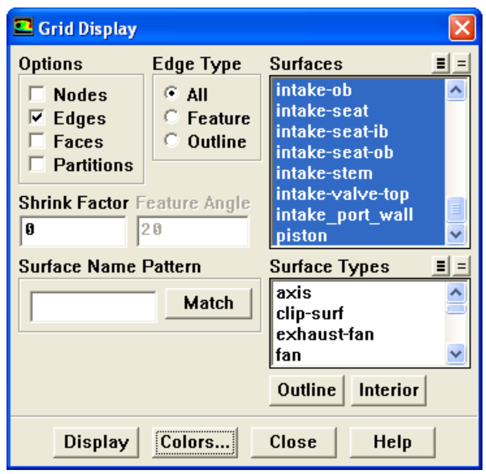
© Fluent Inc. January 17, 2007



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

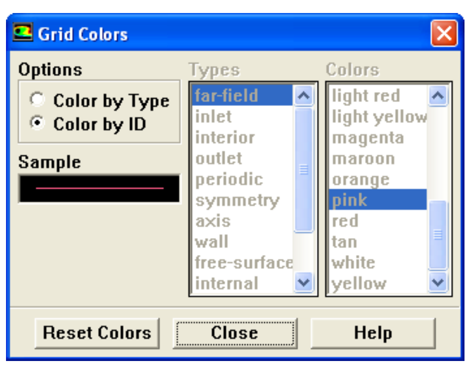
4. Display the grid (Figure 12.2).

   Display ⟶Grid...



   (a) Click Colors....

   The Grid Colors *panel opens.*



   i. Select Color by ID in the Options list.

ii. Close the Grid Colors panel.

(b) Click Display and close the Grid Display panel.

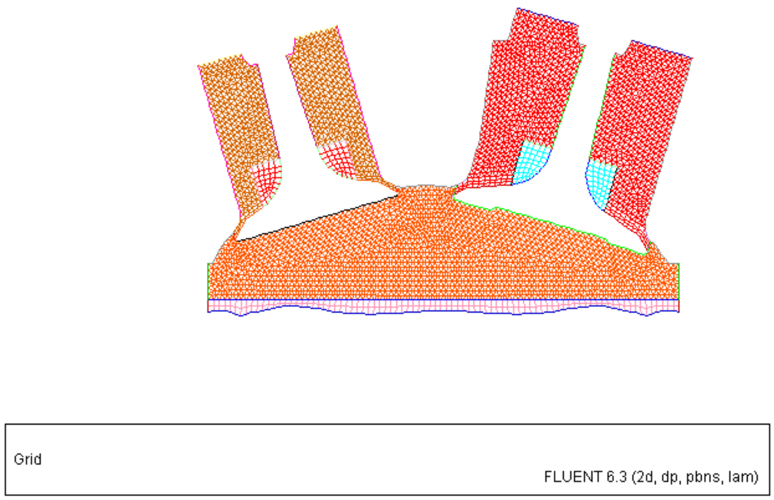


Figure 12.2: Grid Display

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.

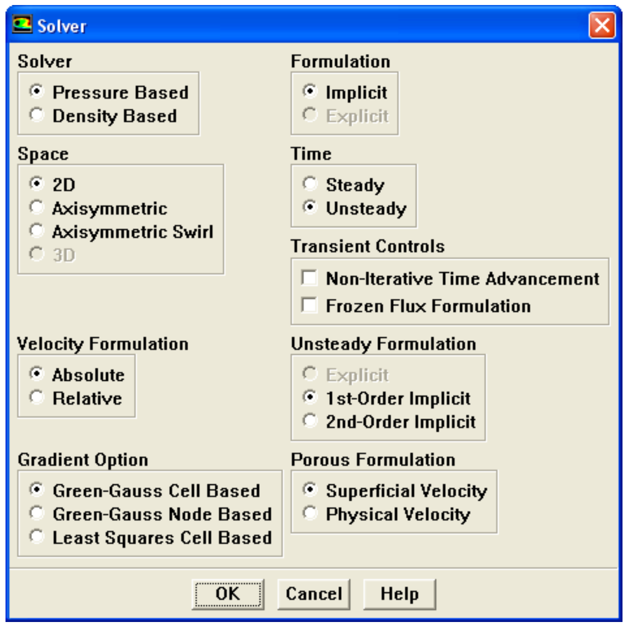
© Fluent Inc. January 17, 2007

Step 2: Models

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

1. Enable the unsteady time formulation.

   Define → Models → Solver...

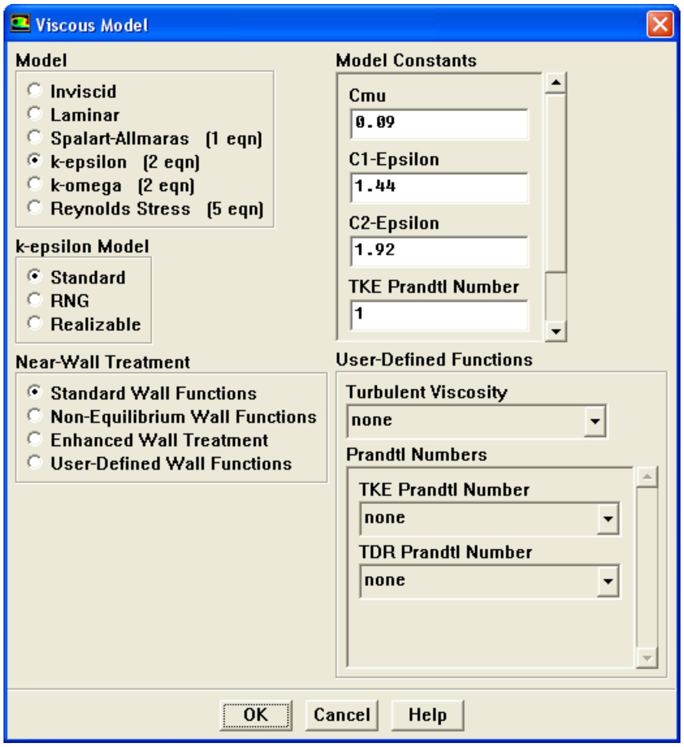


   (a) Select Unsteady in the Time list.

   (b) Click OK to close the Solver panel.

2. Enable the k-ϵ turbulence model.

   Define $\longrightarrow$ Models $\longrightarrow$ Viscous...



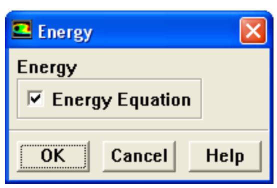
   (a) Select k-epsilon (2 eqn) in the Model list.

   (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...



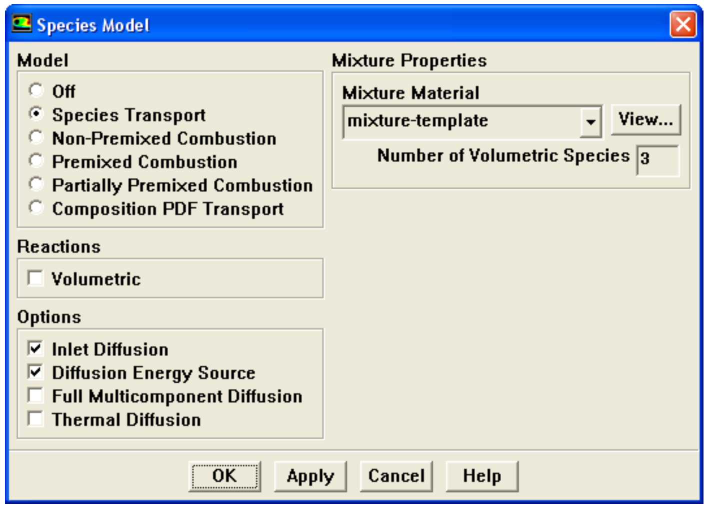
   (a) Enable Energy Equation in the Energy list.

   (b) Click OK to close the Energy panel.

© Fluent Inc. January 17, 2007

4. Enable chemical species transport.

   Define ⟶ Models ⟶ Species ⟶ Transport & Reaction...



   (a) Enable Species Transport in the Model list.

   (b) Retain the default settings for other parameters.

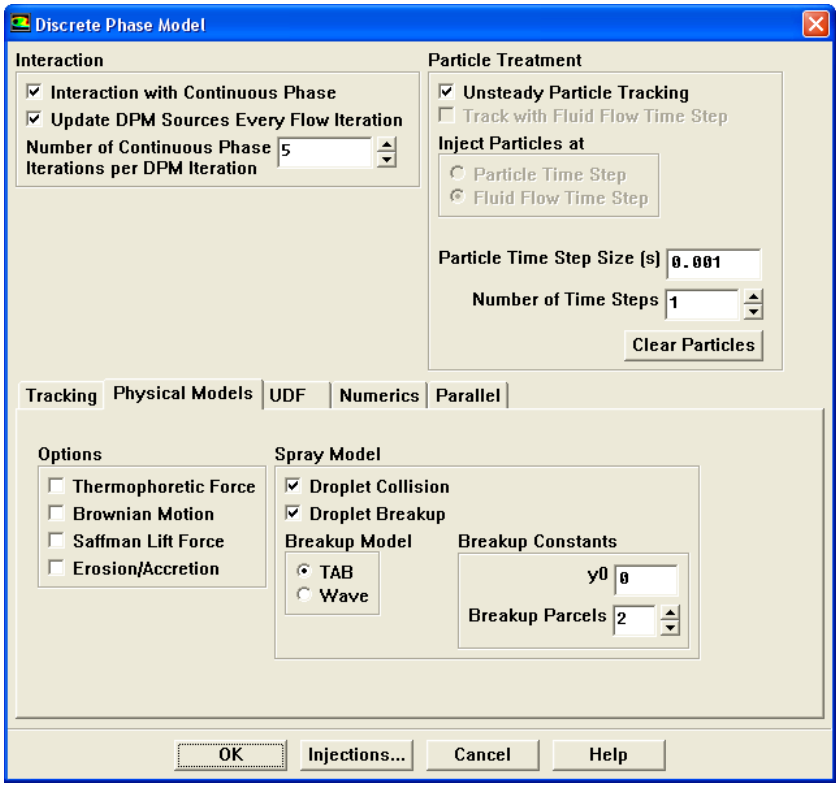
   (c) Click OK to close the Species Model panel.

   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.

   Define ⟶ Models ⟶ Discrete Phase...



(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.

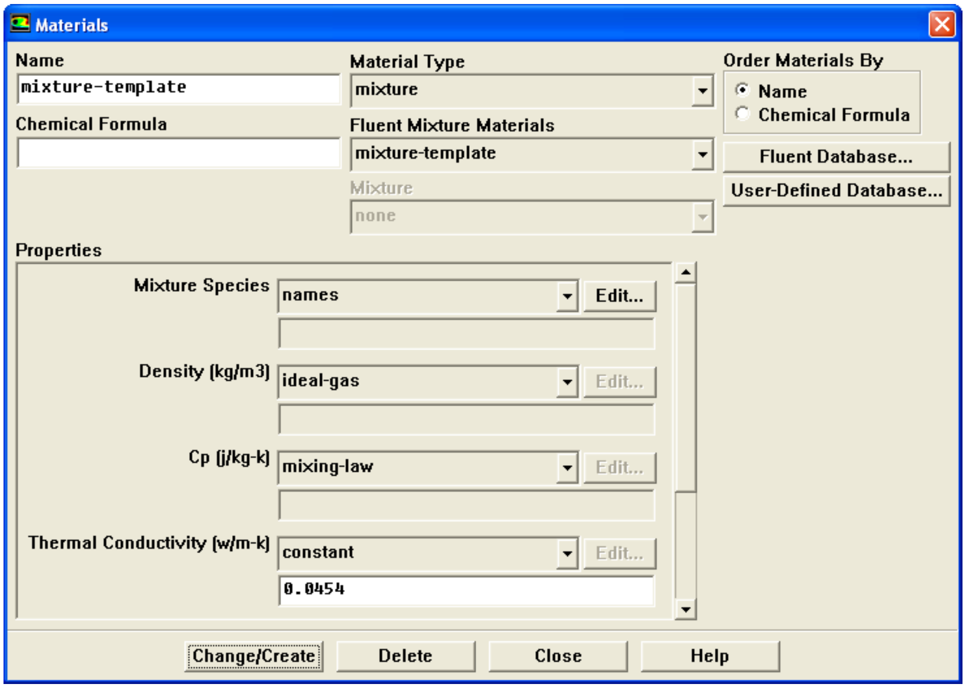
      This parameter is the dimensionless droplet distortion at $t = 0$.

(c) Click OK to close the Discrete Phase Model panel.

 © Fluent Inc. January 17, 2007

Step 3: Materials

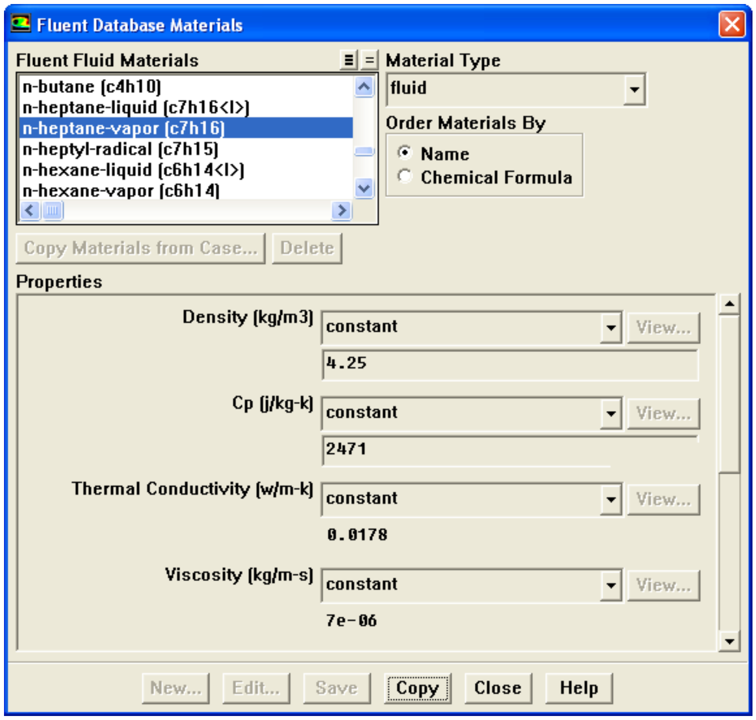
Define ⟶Materials...



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.



i. Select fluid from the Material Type drop-down list in the Fluent Database Materials panel.

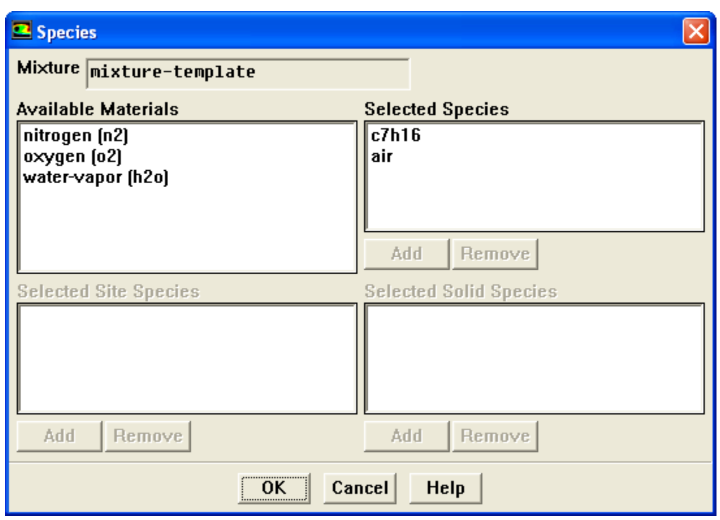
ii. Select n-heptane-vapor (c7h16) from the Fluent Fluid Materials list.

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.



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

© Fluent Inc. January 17, 2007

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 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.*

© Fluent Inc. January 17, 2007

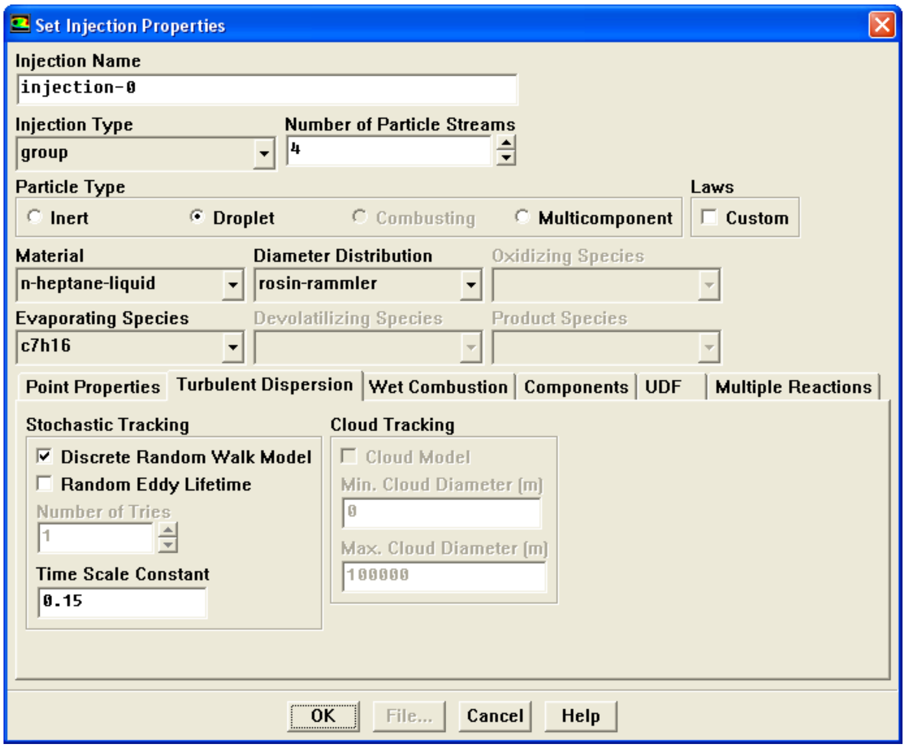
Step 4: Injection

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

Define ⟶Injections...

1. Click Create.

   The Set Injection Properties *panel opens.*



   (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:

© Fluent Inc. January 17, 2007

Parameter	Value of First Point	Value of 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.

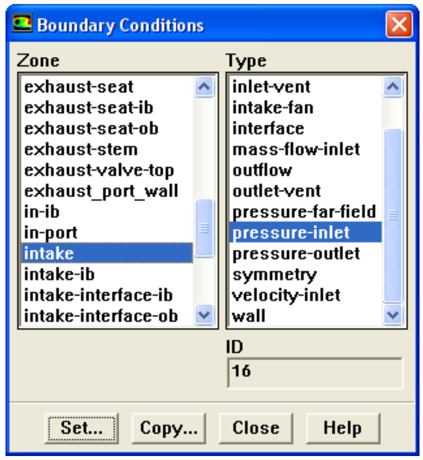
iii. Click OK to close the Set Injection Parameters panel.

(i) Close the Injections panel.

Step 5: Boundary Conditions

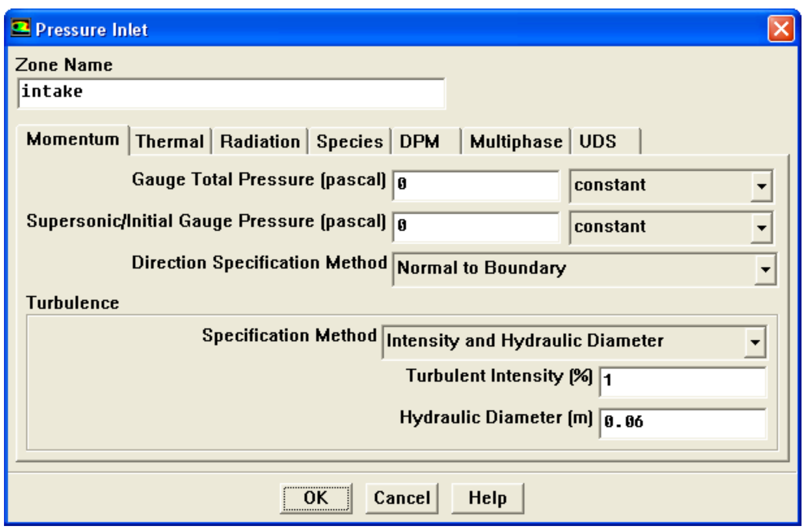
Define ⟶Boundary Conditions...

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



(a) Select intake from the Zone list.

(b) Click Set....



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.

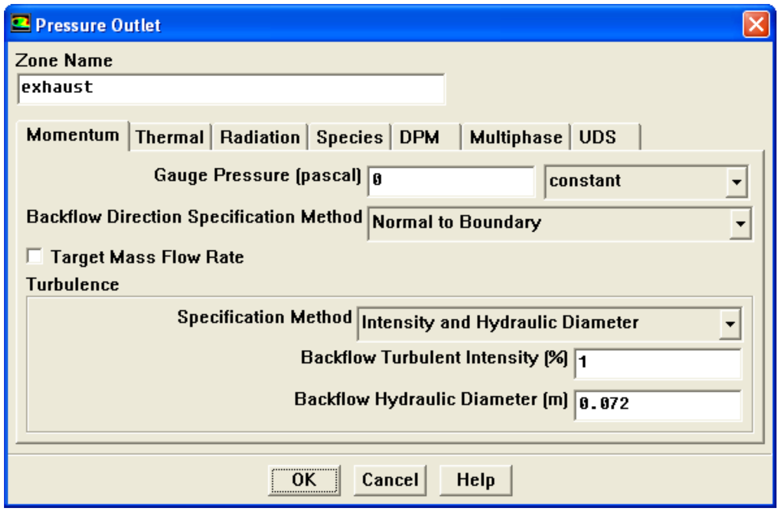
© Fluent Inc. January 17, 2007

v. Click the Thermal tab.

vi. Enter 318 K for the Total Temperature.

vii. Click OK to close the Pressure Inlet panel.

2. Set the following conditions for the pressure-outlet (exhaust).



(a) Select Intensity and Hydraulic Diameter from the Turbulence Specification Method drop down list.

(b) Enter 1 % for Backflow Turbulent Intensity.

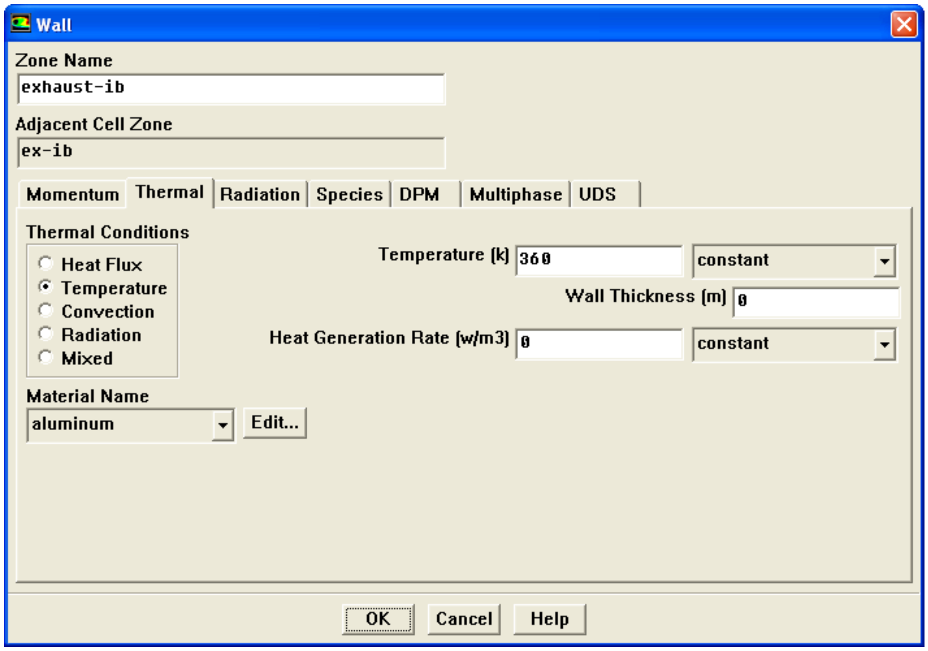
(c) Enter 0.072 m for Backflow Hydraulic Diameter.

(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).



(a) Click the Thermal tab.

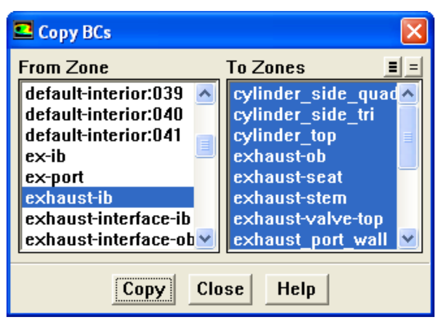
(b) Select Temperature in the Thermal Conditions group box.

(c) Enter 360 K for Temperature.

(d) Click 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.

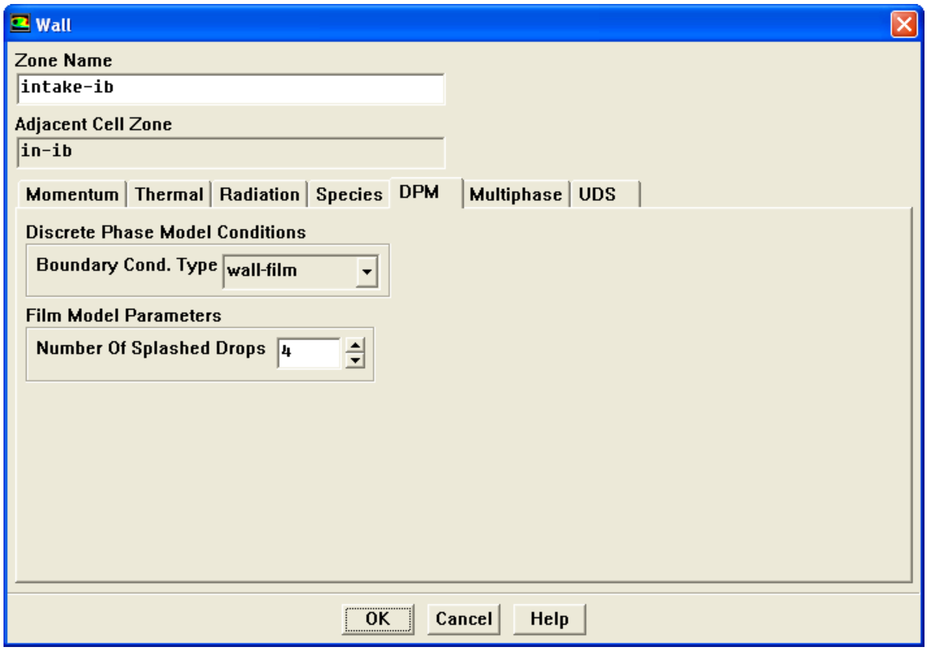
   (d) Click Copy.

   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).

© Fluent Inc. January 17, 2007



(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.

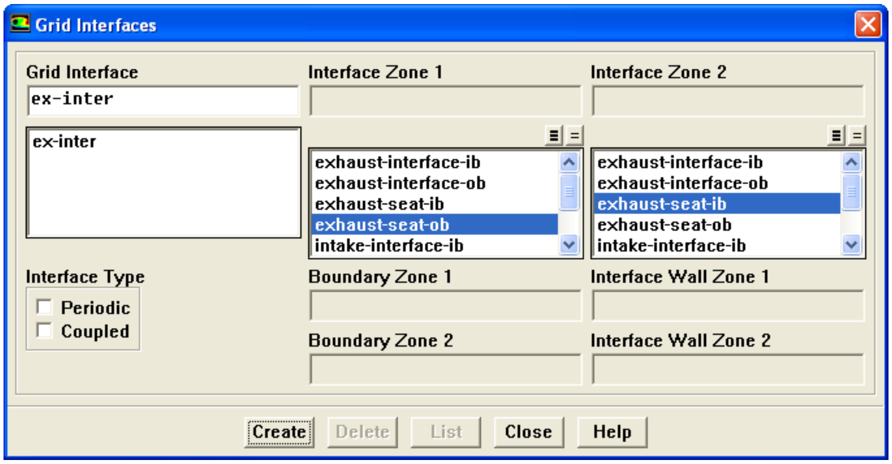
6. Similarly define the boundary conditions for intake-ob wall.

7. Close the Boundary Conditions panel.

Step 6: Grid Interfaces

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

Grid ⟶Interfaces...



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.

© Fluent Inc. January 17, 2007

Step 7: Mesh Motion Setup

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

   Define ⟶ Dynamic Mesh ⟶ Parameters...

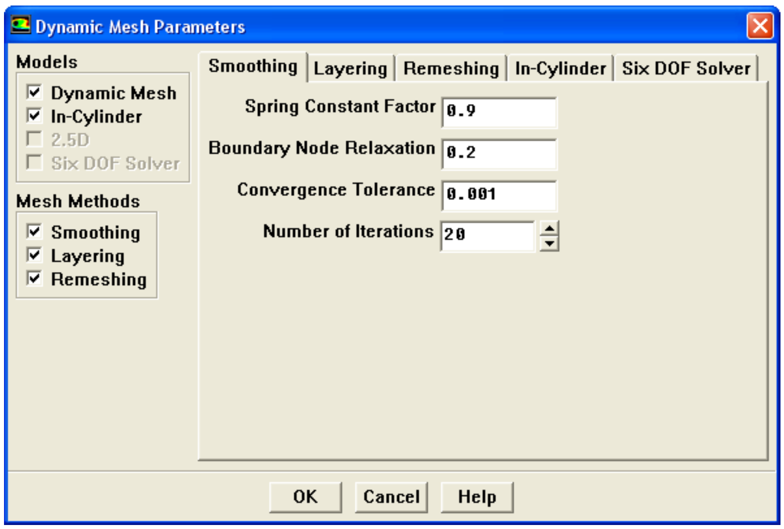
   (a) Enable Dynamic Mesh in the Models list.

   (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.



   (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.

 © Fluent Inc. January 17, 2007

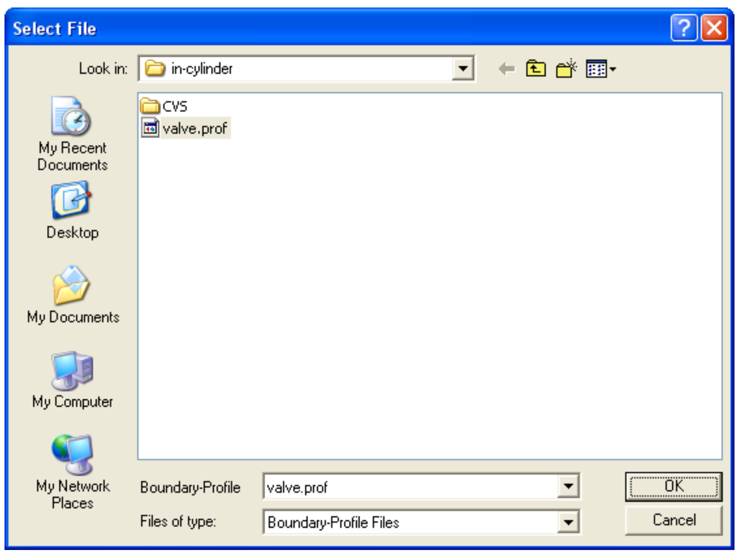
The In-Cylinder model is specifically used for modeling Internal Combustion Engines. It facilitates the modeling of the dynamic mesh motion of piston and valves, in terms of crank shaft angle, crank speed, piston stroke, and connecting rod length. Further, the solution is advanced in terms of crank angle,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

   File ⟶ Read ⟶ Profile...



   (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.*

```
> define/models/dynamic-mesh-controls
/define/models/dynamic-mesh-controls> icp
/define/models/dynamic-mesh-controls/in-cylinder-parameter> ppl
#f
Lift Profile:(1) [()] in-valve
Lift Profile:(2) [()] ex-valve
Lift Profile:(3) [()] <Enter>

Start: [360] <Enter>
End: [1080] <Enter>
Increment: [10] <Enter>
Plot lift? [yes] <Enter>
/define/models/dynamic-mesh-controls/in-cylinder-parameter>
```

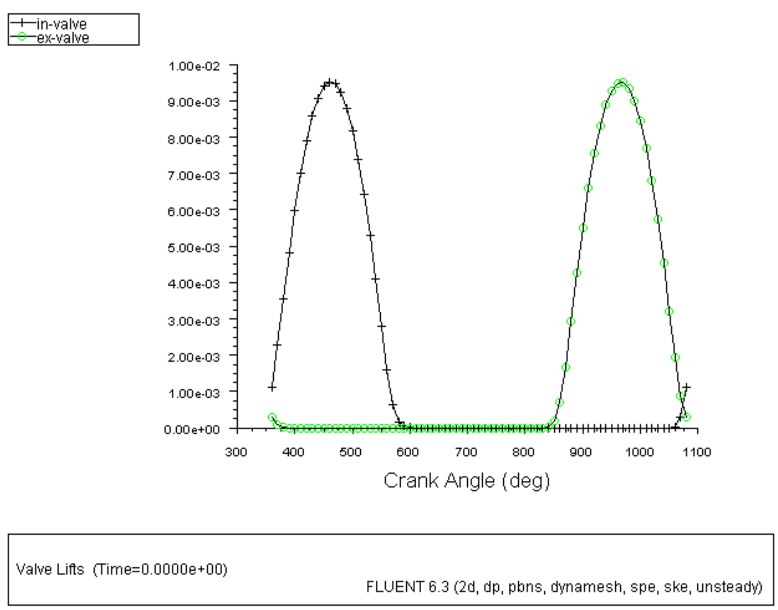
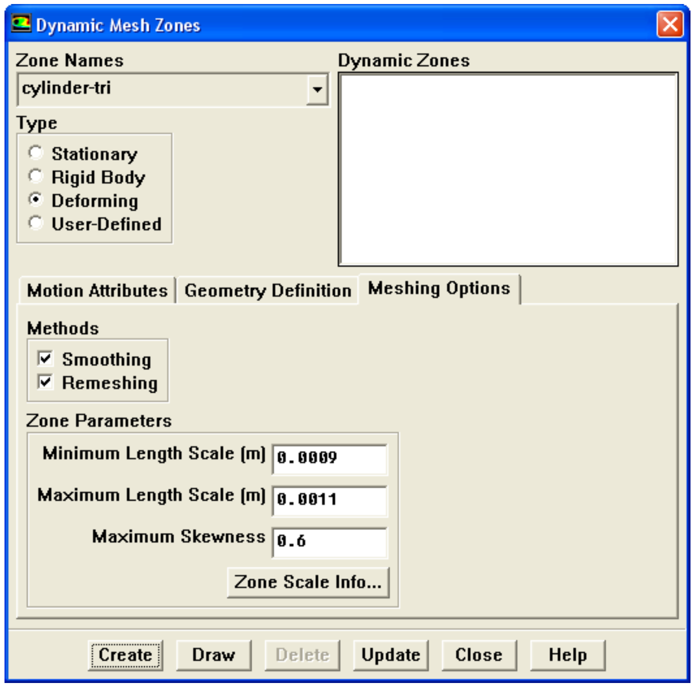


Figure 12.3: Piston Motion Profile

© Fluent Inc. January 17, 2007

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

   Define ⟶ Dynamic Mesh ⟶ Zone...



   (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.

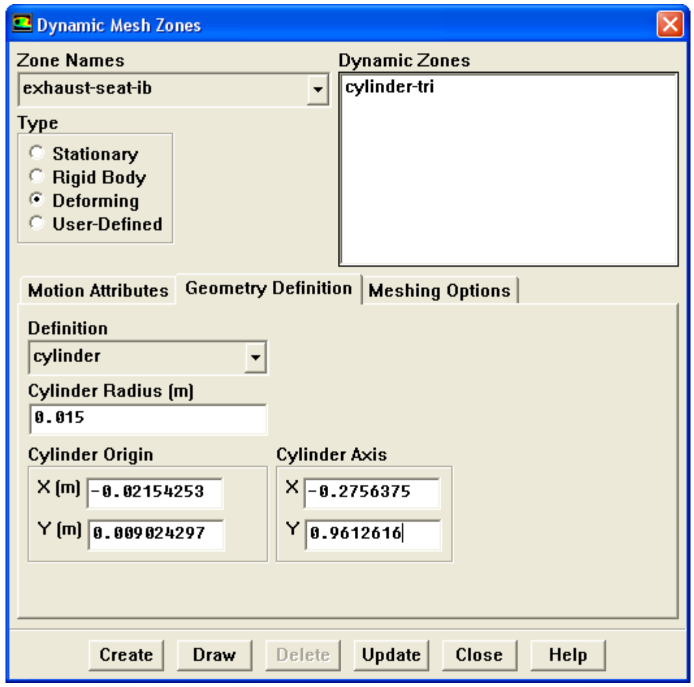
      iii. Click the Meshing Options tab.

         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.



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.

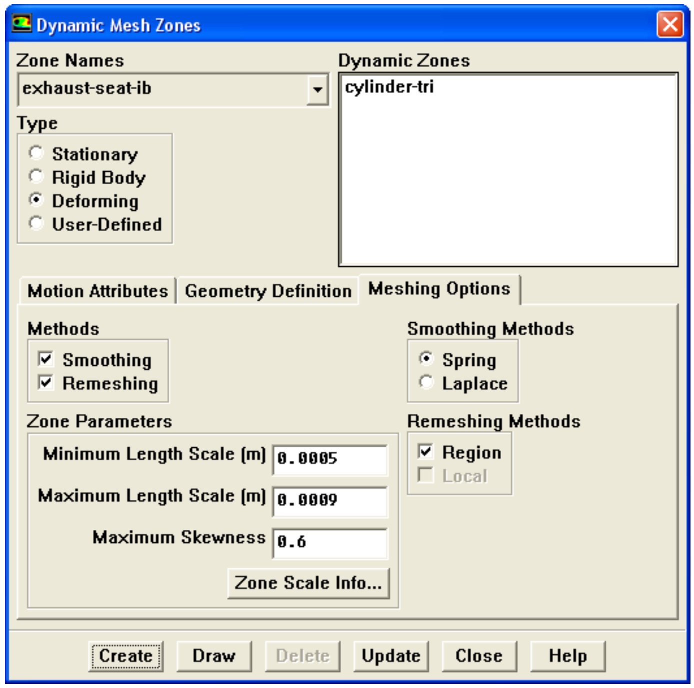
B. Enter 0.015 m for the Cylinder Radius.

C. Enter -0.02154253 m for X and 0.009024297 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.

© Fluent Inc. January 17, 2007



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.

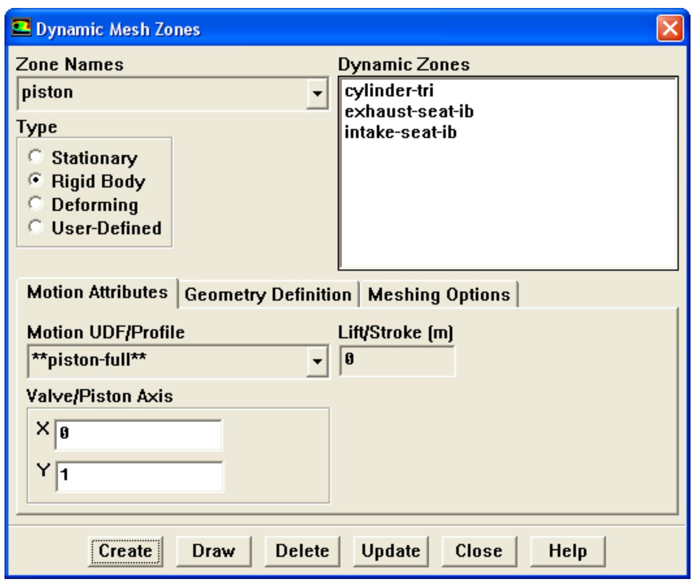
v. Click Create.

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.



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.

© Fluent Inc. January 17, 2007

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

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

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:

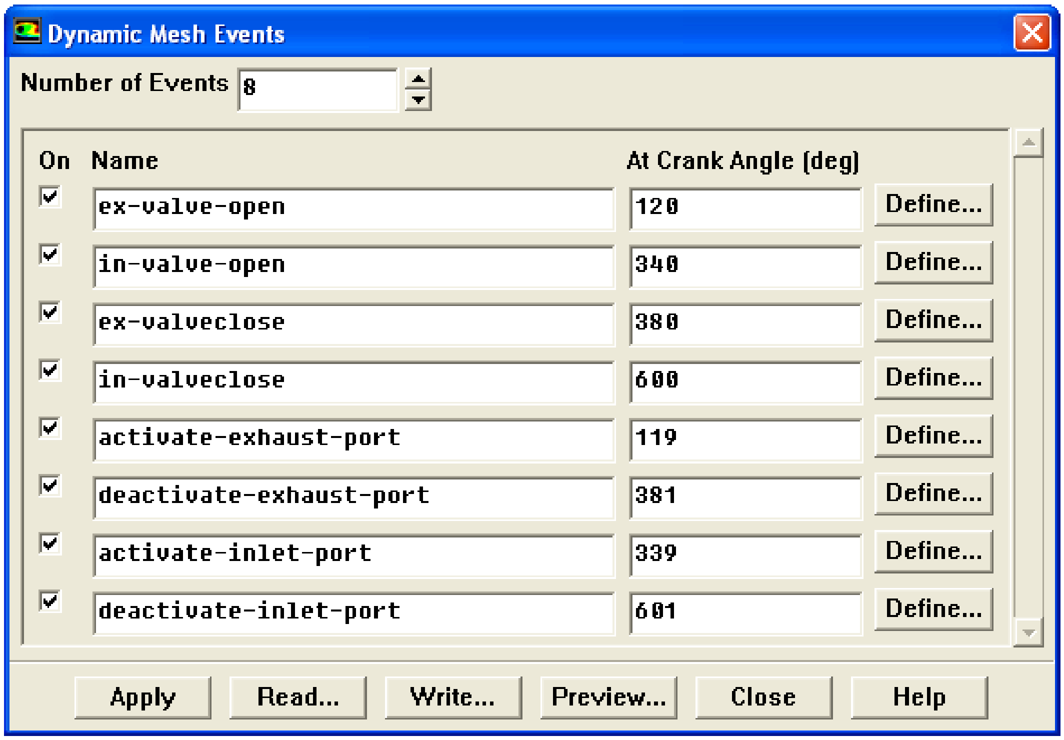
Zone Names	Type	Meshing Options	
		For in-port Zone Cell Height (m)	For in-ib Zone Cell Height (m)
intake-interior-ib	Stationary	0	0.001

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.

   Define ⟶ Dynamic Mesh ⟶ Events...



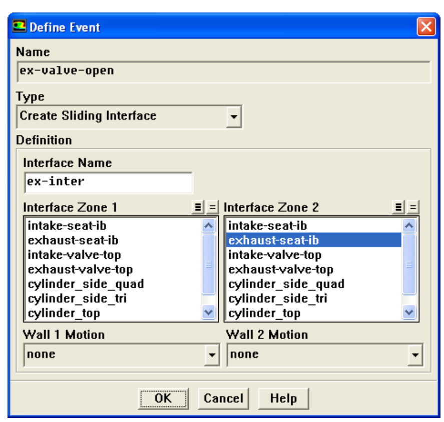
   (a) Set the Number of Events to 8.

   (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.



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

© Fluent Inc. January 17, 2007

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 OK to close the Define Event panel.

(f) Similarly, create the following Dynamic Events:

Name	Crank Angle	Setup description
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-close	380 deg	1. Select Delete Sliding Interface from the Type drop-down 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-exhaust-port	119 deg	1. Select Activate Cell Zone from the Type drop-down list. 2. Select ex-ib and ex-port in the Definition list. 3. Click OK.
deactivate-exhaust-port	381 deg	1. Select Deactivate Cell Zone from the Type drop-down list. 2. Select ex-ib and ex-port in the Definition list. 3. Click OK.
activate-inlet-port	339 deg	1. Select Activate Cell Zone from the Type drop-down list. 2. Select in-ib and in-port in the Definition list. 3. Click OK.
deactivate-inlet-port	601 deg	1. Select Deactivate Cell Zone from the Type drop-down list. 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

 © Fluent Inc. January 17, 2007

model, the dynamic events are crank angle-based, whereas by default, they are flow time-based.

When the inlet and exhaust valves 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`).

   File ⟶ Write ⟶ 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 ⟶ Grid...

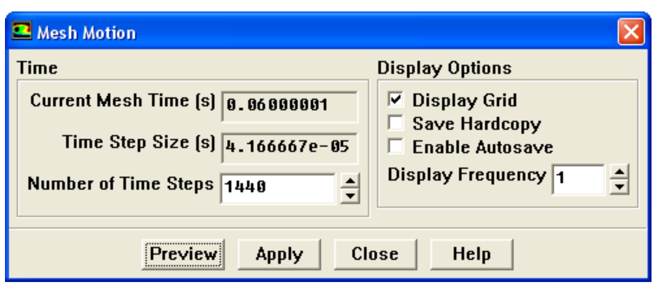
   (a) Select all the surfaces in the Surfaces list.

   (b) Click Display.

   (c) Close the Grid Display panel.

3. Set up the mesh preview.

   Solve ⟶ Mesh Motion...



   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

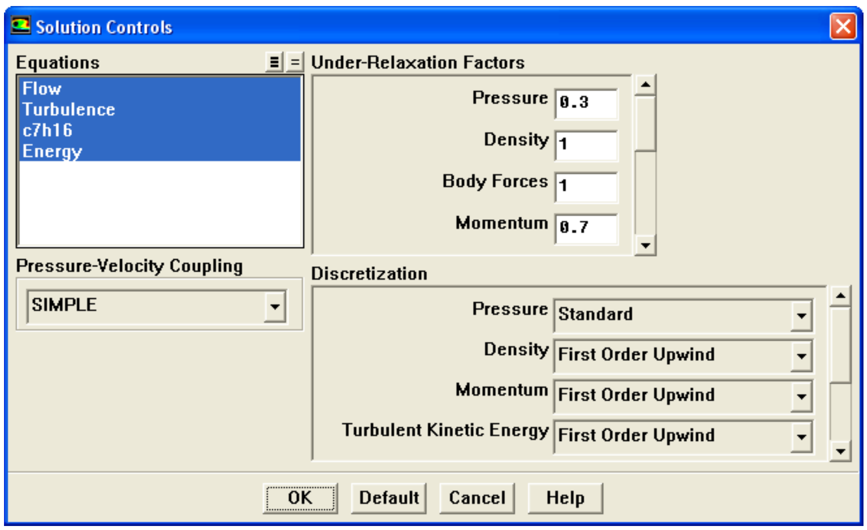
1. Read the case file back into FLUENT (`In_Cylinder.cas.gz`).

   File ⟶ Read ⟶ 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 ⟶ Controls ⟶ Solution...

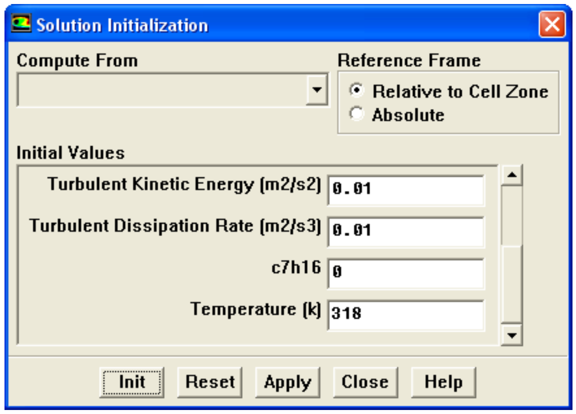
   

   (a) Click OK to close Solution Controls panel.

© Fluent Inc. January 17, 2007

3. Initialize the flow field.

   Solve ⟶ Initialize ⟶ Initialize...



The Gauge Pressure *value is zero.*

(a) Enter 0 pascal for the Gauge Pressure.

(b) Enter 0 m/s for X Velocity and Y Velocity.

(c) Enter 0.01 m^2/s^2 for Turbulent Kinetic Energy.

(d) Enter 0.01 m^2/s^3 for Turbulent Dissipation Rate.

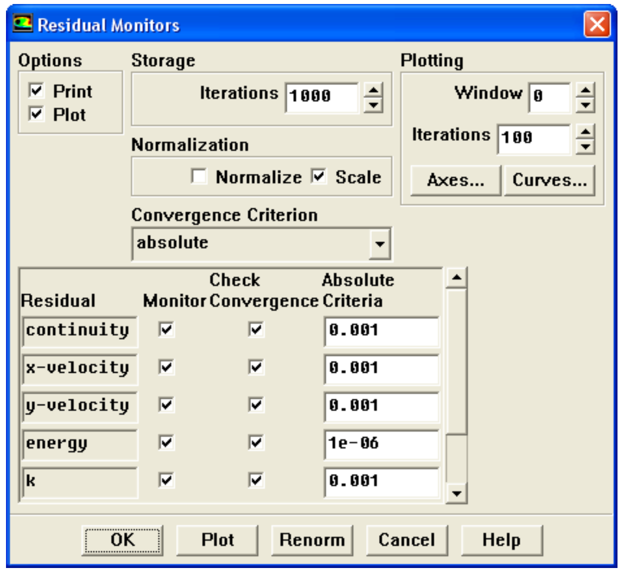
(e) Enter 0 for c7h16.

(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 ⟶ Monitors ⟶ Residual...



(a) Enable Plot in the Option list.

(b) Enter 100 for the Iterations in the Plotting group box.

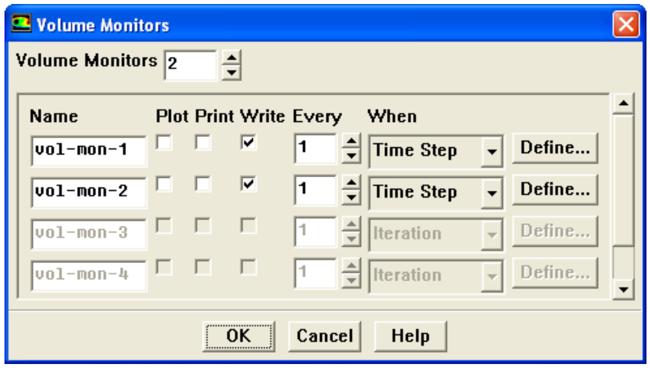
To avoid a cluttered residual plot in transient simulations, it is useful to display only the most recent iterations.

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

© Fluent Inc. January 17, 2007

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

   Solve ⟶ Monitors ⟶Volume...



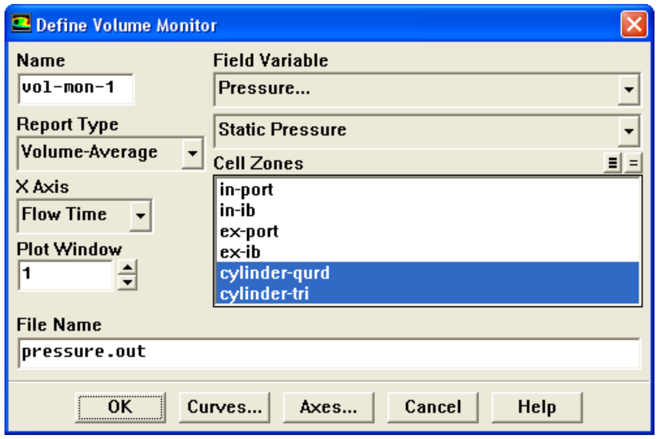
   (a) Set the Volume Monitors to 2.

   (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.



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 OK in the Define Volume Monitors panel.

(f) Click OK to close the 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.

Display ⟶Contours...

 © Fluent Inc. January 17, 2007

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.

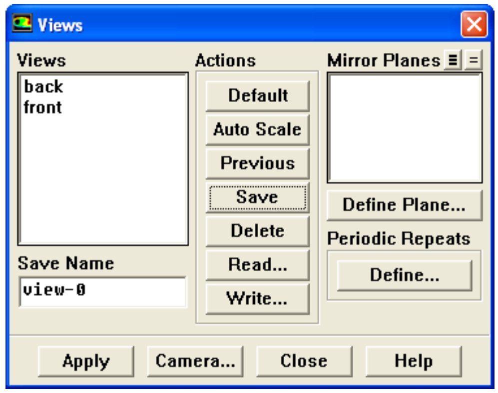
Note: *The piston is at TDC and during the solution; the computational domain will expand up to the BDC. Therefore leave sufficient space for domain expansion.*

v. Close the Contours panel.

Figure 12.4: Velocity Contours for Animation Setup

(b) Save the current view.

Display ⟶Views...



i. Click Save to save the current view as view-0.

ii. Close the Views panel.

(c) Set hardcopy settings.

File ⟶Hardcopy....

i. Select TIFF in the Format group box.

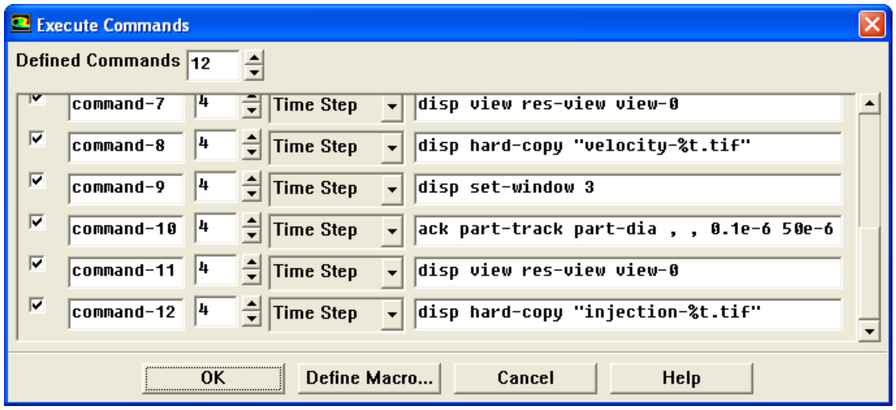
ii. Select Color in the Coloring group box.

iii. Click Apply and close the Graphics Hardcopy panel.

© Fluent Inc. January 17, 2007

(d) Specify the commands for animation.

Solve ⟶Execute Commands...



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.

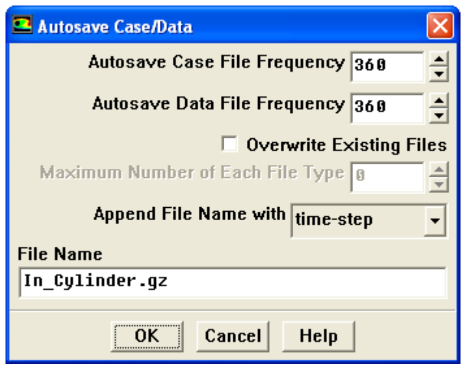
The '%t' appended to the file name instructs FLUENT *to append the timestep index to the filename.*

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.

   File ⟶ Write ⟶ 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`).

   File ⟶ Write ⟶ Case & Data...

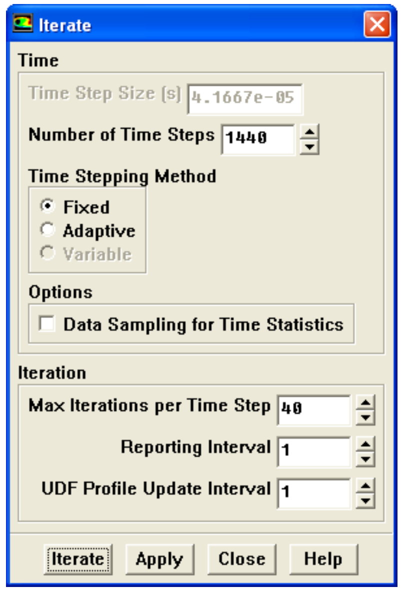
   Click OK *to overwrite the previously saved case file.*

Step 10: Solution

1. Start the calculation.

   Solve ⟶ Iterate...

© Fluent Inc. January 17, 2007



(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.*

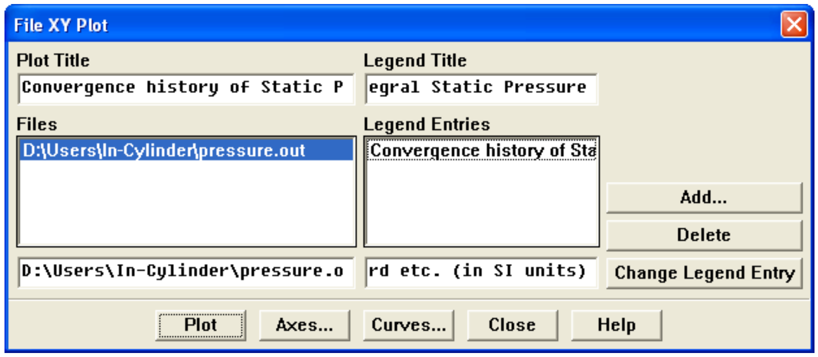
2. Write the case and data files.

File ⟶ Write ⟶ Case & Data...

Step 11: Postprocessing

1. Display static pressure and temperature variation.

   Plot ⟶File...



   (a) Click Add... to add the file.

   (b) Select the `pressure.out` file and click OK.

   (c) Click Plot (Figure 12.5).

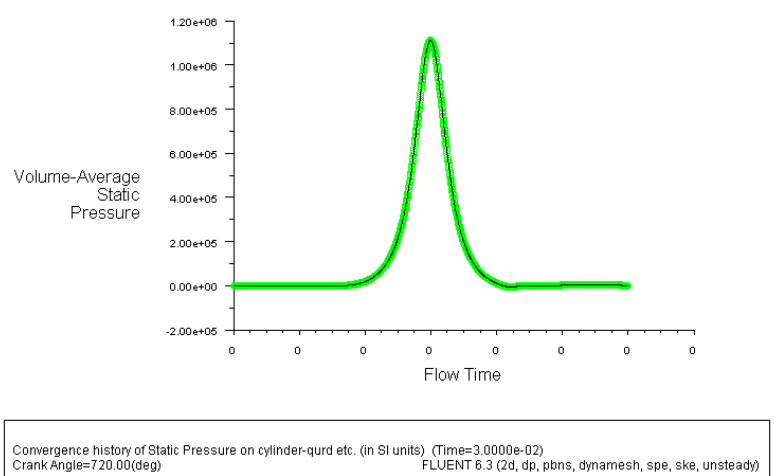
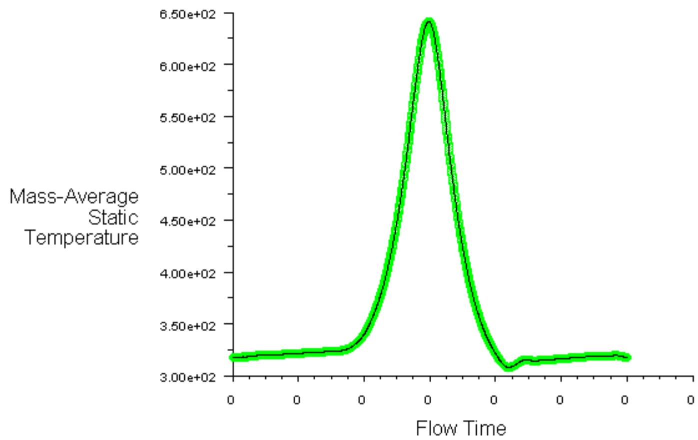


Figure 12.5: Convergence History of Static Pressure

© Fluent Inc. January 17, 2007

(d) Click Delete to remove the added file.

(e) Similarly plot the file `temperature.out` for static temperature variation (Figure 12.6).



Convergence history of Static Temperature on cylinder-qurd etc. (in SI units) (Time=3.0000e-02)
Crank Angle=720.00(deg) FLUENT 6.3 (2d, dp, pbns, dynamesh, spe, ske, unsteady)

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.

   File ⟶ Read ⟶Case & Data...

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

   Display ⟶Contours...

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

   ii. Click Display.

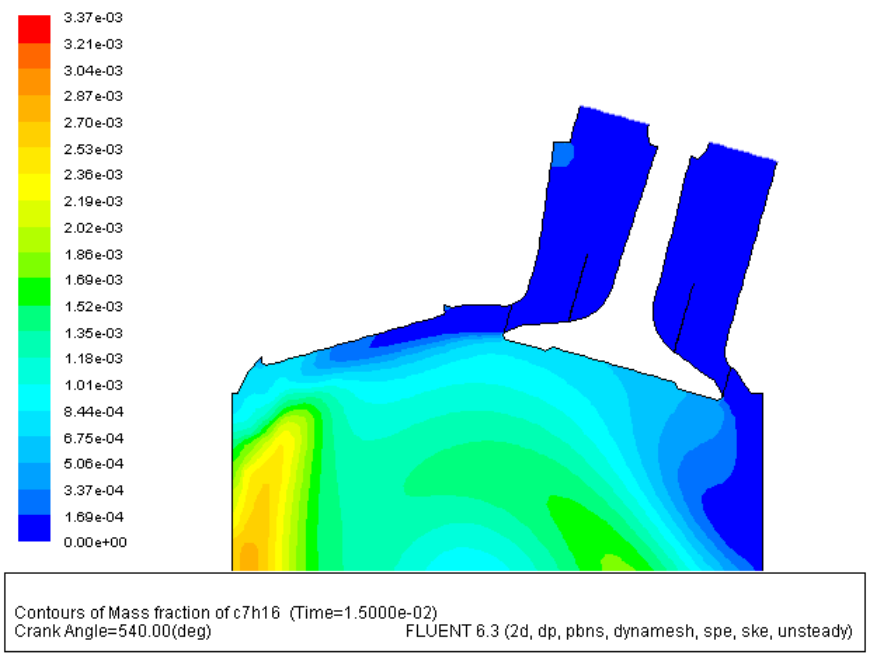


Figure 12.7: Predicted C_7H_{16} Mass fraction Distribution

 © Fluent Inc. January 17, 2007

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.

   File ⟶ Read ⟶ Case & Data...

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

   Display ⟶ Contours...

   i. Select Temperature... and Static Temperature in the Contours of drop-down lists.

   ii. Click Display and close the Contours panel.

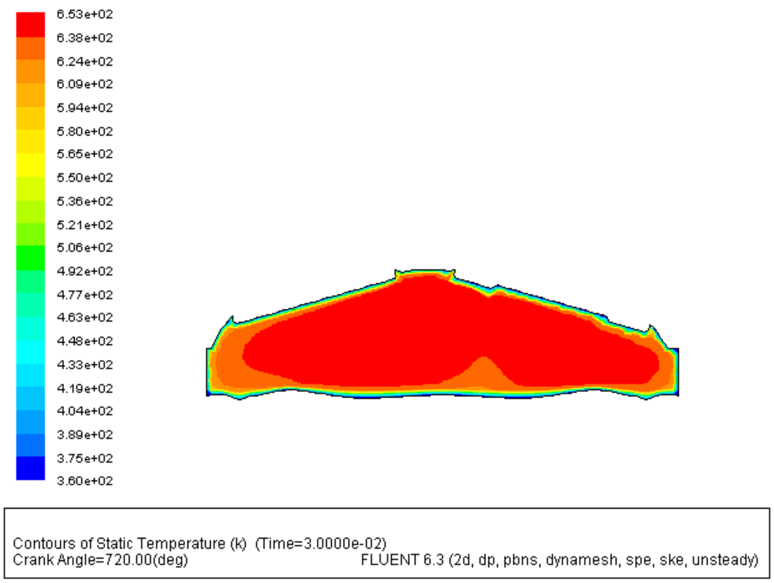


Figure 12.8: Predicted Static Temperature Distribution

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

 © Fluent Inc. January 17, 2007