

ANSYS Internal Combustion Engines Tutorial Guide



ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Release 16.0
January 2015

ANSYS, Inc. is
certified to ISO
9001:2008.

Copyright and Trademark Information

© 2014-2015 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS, AIM and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

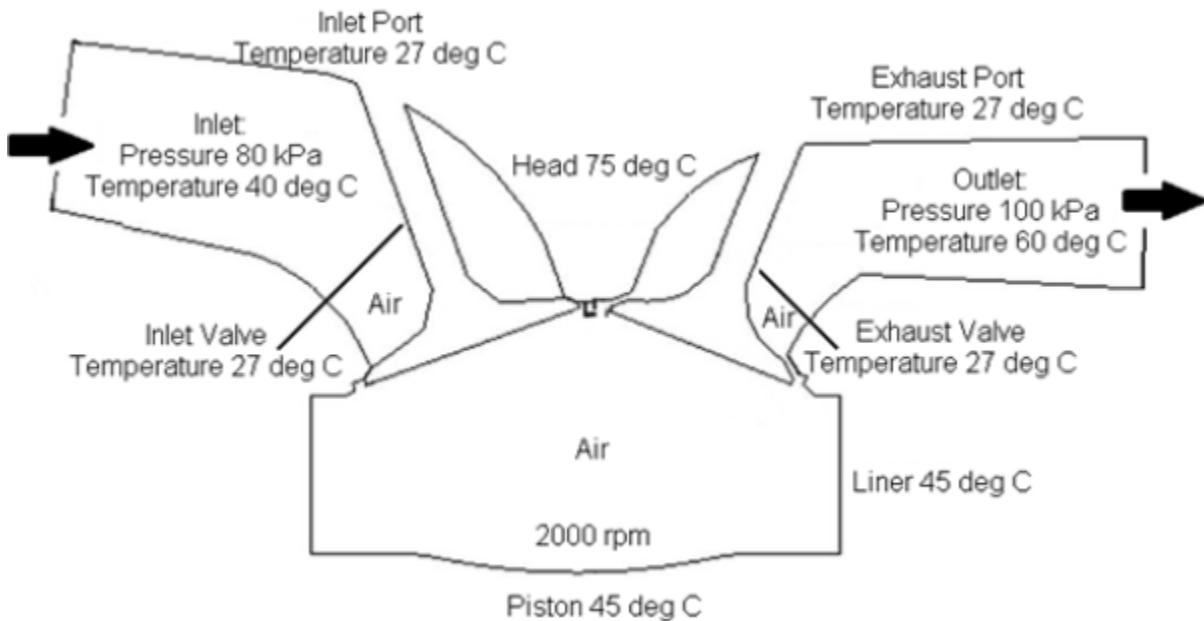
Table of Contents

1. Tutorial: Solving a Cold Flow Simulation	1
1.1. Preparation	2
1.2. Step 1: Setting the Properties	2
1.3. Step 2: Performing the Decomposition	4
1.4. Step 3: Meshing	12
1.5. Step 4: Setting up the Simulation	15
1.6. Step 5: Running the Solution	28
1.7. Step 6: Obtaining the Results	30
1.8. Step 7: Postprocessing	57
1.9. Summary	62
1.10. Further Improvements	62
2. Tutorial: Solving a Port Flow Simulation	63
2.1. Preparation	64
2.2. Step 1: Setting the Properties	64
2.3. Step 2: Performing the Decomposition	65
2.4. Step 3: Meshing	76
2.5. Step 4: Setting up the Simulation	78
2.6. Step 5: Running the Solution	84
2.7. Step 6: Obtaining the Results	92
2.8. Summary	106
2.9. Further Improvements	106
3. Tutorial: Solving a Combustion Simulation for a Sector	107
3.1. Preparation	108
3.2. Step 1: Setting the Properties	108
3.3. Step 2: Performing the Decomposition	110
3.4. Step 3: Meshing	117
3.5. Step 4: Setting up the Simulation	119
3.6. Step 5: Running the Solution	133
3.7. Step 6: Obtaining the Results	136
3.8. Summary	171
3.9. Further Improvements	171
4. Tutorial: Solving a Gasoline Direct Injection Engine Simulation	173
4.1. Preparation	174
4.2. Step 1: Setting the Properties	174
4.3. Step 2: Performing the Decomposition	176
4.4. Step 3: Meshing	189
4.5. Step 4: Setting up the Simulation	190
4.6. Step 5: Running the Solution	208
4.7. Step 6: Obtaining the Results	212
4.8. Summary	252
4.9. Further Improvements	252
Index	253

Chapter 1: Tutorial: Solving a Cold Flow Simulation

A three dimensional single cylinder CFD simulation of a 4-stroke engine is performed under motored conditions (cold flow) in this tutorial. Detailed boundary conditions are shown in [Figure 1.1: Problem Schematic \(p. 1\)](#). Engine simulation is started from Intake valve opening (IVO) followed by air flow during intake stroke. Air is compressed as piston moves towards top dead centre (TDC). This is followed by expansion of air as piston moves towards bottom Dead centre (BDC). This tutorial serves as an introduction in releasing the streamlined workflow between pre-processing, solver and post processing while carrying out simulations with Fluent.

Figure 1.1: Problem Schematic



This tutorial illustrates the following steps:

- Launch IC Engine system.
- Read an existing geometry into IC Engine.
- Decompose the geometry.
- Define the mesh setup and mesh the geometry.
- Run the simulation.
- Examine the results in the report.
- Perform additional postprocessing in CFD-Post.

This tutorial is written with the assumption that you are familiar with the IC Engine system and that you have a good working knowledge of ANSYS Workbench.

- 1.1. Preparation
- 1.2. Step 1: Setting the Properties
- 1.3. Step 2: Performing the Decomposition
- 1.4. Step 3: Meshing
- 1.5. Step 4: Setting up the Simulation
- 1.6. Step 5: Running the Solution
- 1.7. Step 6: Obtaining the Results
- 1.8. Step 7: Postprocessing
- 1.9. Summary
- 1.10. Further Improvements

1.1. Preparation

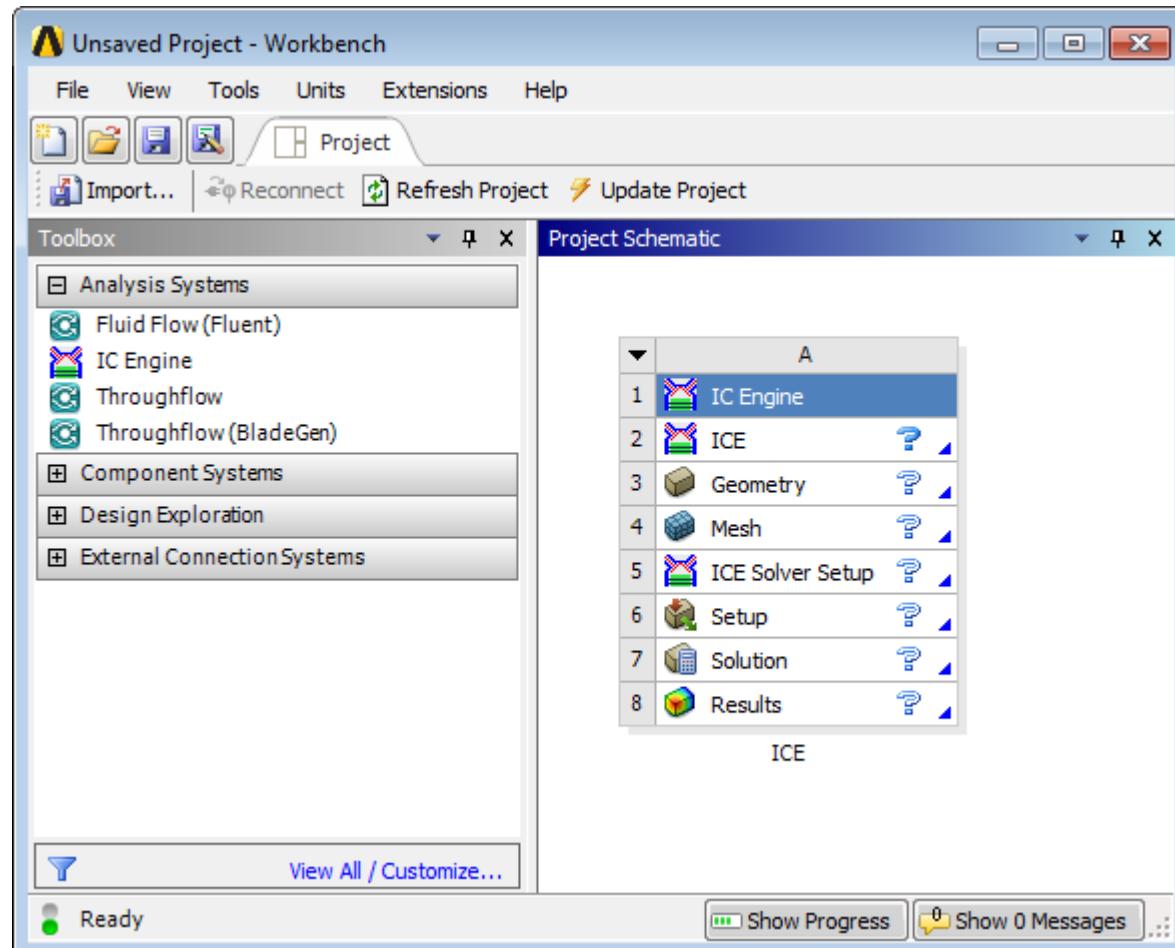
1. Copy the files (demo_eng.x_t and lift.prof) to your working folder.

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

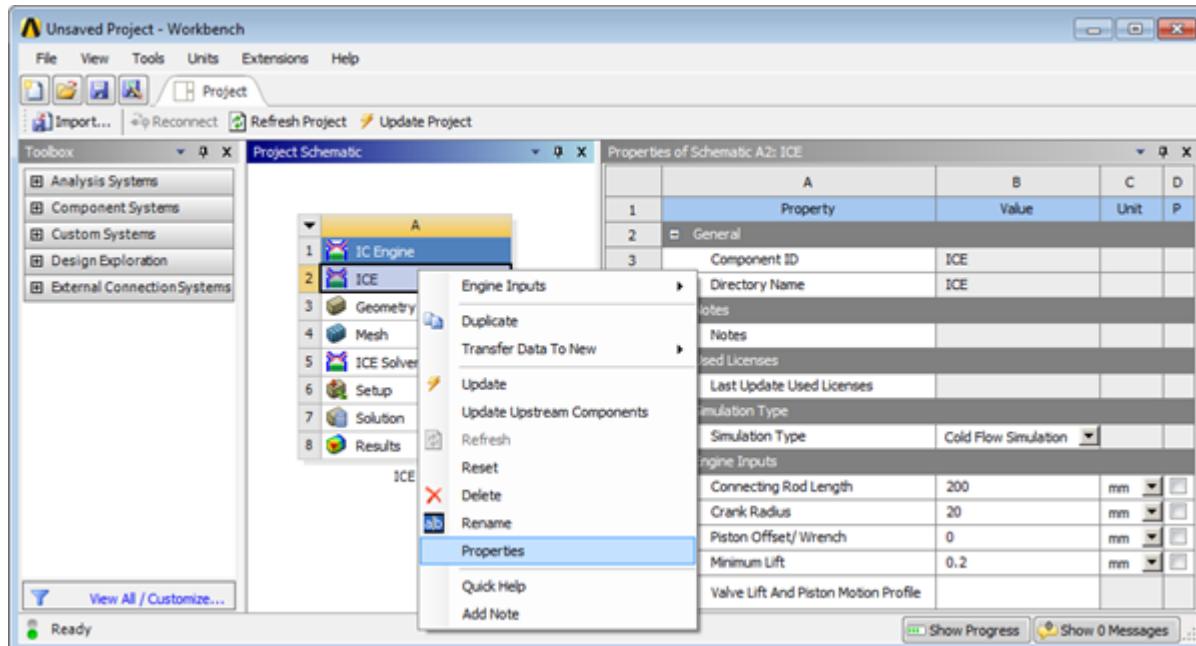
2. Start Workbench.

1.2. Step 1: Setting the Properties

1. Create an IC Engine analysis system in the Workbench interface by dragging or double-clicking **IC Engine** under **Analysis Systems** in the **Toolbox**.



2. If the **Properties** view is not already visible, right-click **ICE**, cell 2, and select **Properties** from the context menu.

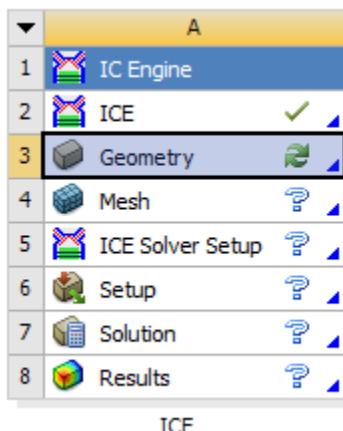


3. Select **Cold Flow Simulation** from the **Simulation Type** drop-down list.
4. In the **Properties** dialog box under **Engine Inputs** enter **144 . 3** for **Connecting Rod Length**.
5. Enter **45** for **Crank Radius**.
6. Retain **0** for **Piston Offset/Wrench**.
7. Enter **0 . 5** for **Minimum Lift**.
8. Click **Browse File** next to **Lift Curve**. The **File Open** dialog box opens. Select the valve profile file **lift.prof** and click **Open**.

	A	B	C	D
1	Property	Value	Unit	P
2	General			
3	Component ID	ICE		
4	Directory Name	ICE		
5	Notes			
6	Notes			
7	Used Licenses			
8	Last Update Used Licenses			
9	Simulation Type			
10	Simulation Type	Cold Flow Simulation	▼	
11	Engine Inputs			
12	Connecting Rod Length	144.3	mm ▼	
13	Crank Radius	45	mm ▼	
14	Piston Offset/ Wrench	0	mm ▼	
15	Minimum Lift	0.5	mm ▼	
16	Valve Lift And Piston Motion Profile	ICE\ICE\lift.prof		

1.3. Step 2: Performing the Decomposition

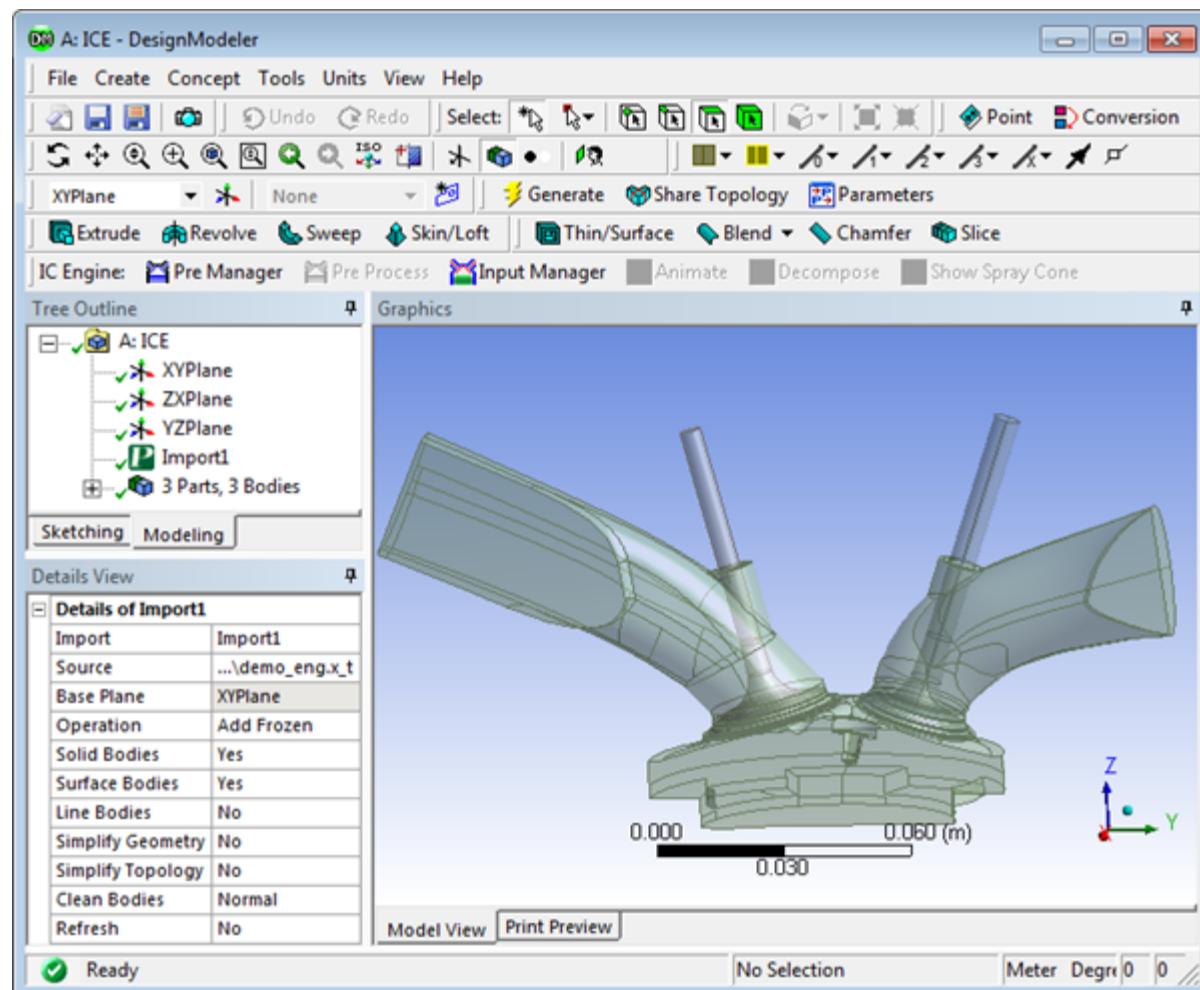
Here you will read the geometry and prepare it for decomposition. Double-click the **Geometry** cell to open the DesignModeler.



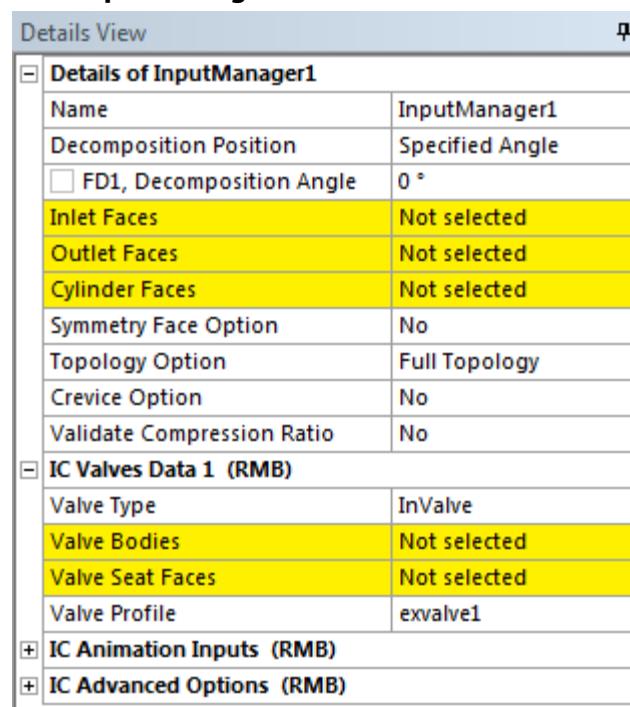
1. Select **Millimeter** from the **Units** menu.
2. Import the geometry file, `demo_eng.x_t`.

File > Import External Geometry File...

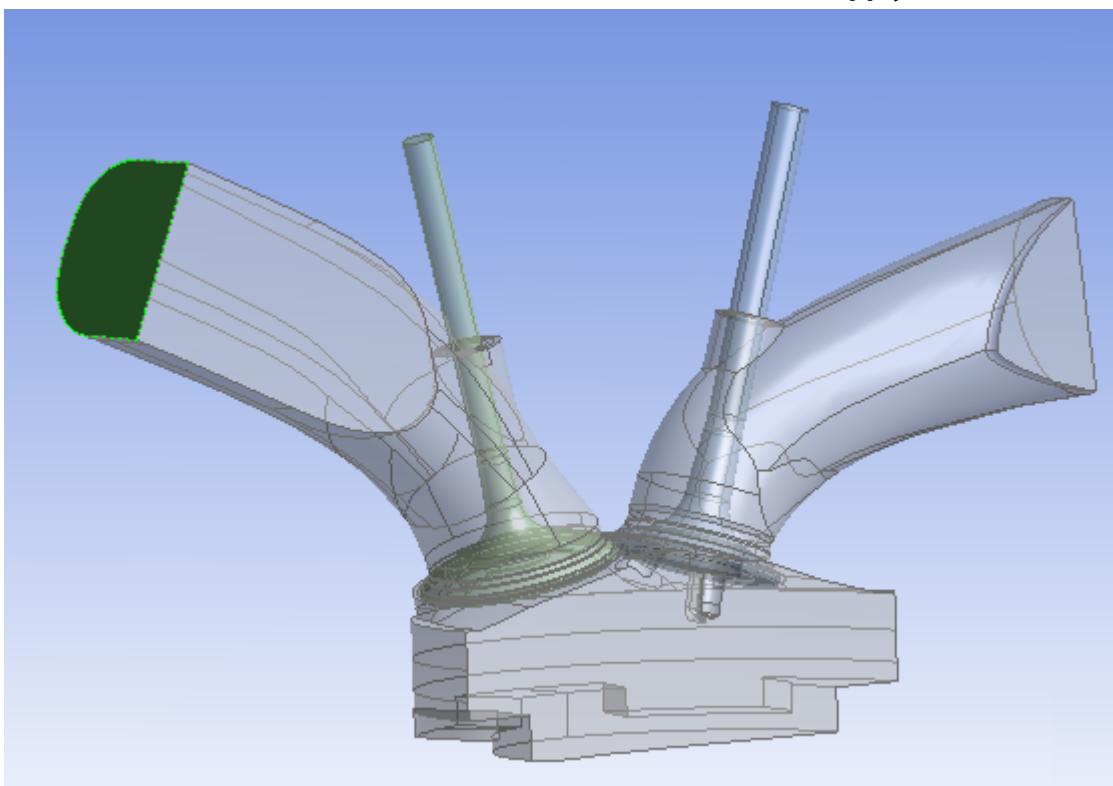
3. Click **Generate** to complete the import feature.



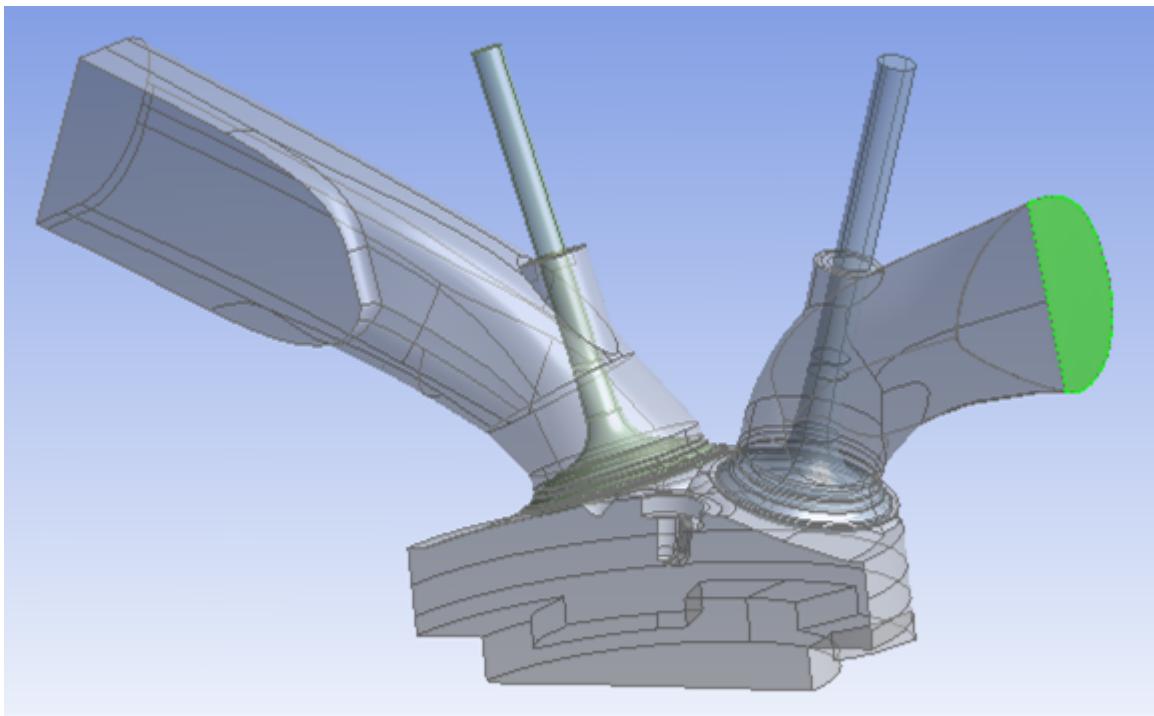
4. Click **Input Manager** located in the **IC Engine** toolbar.



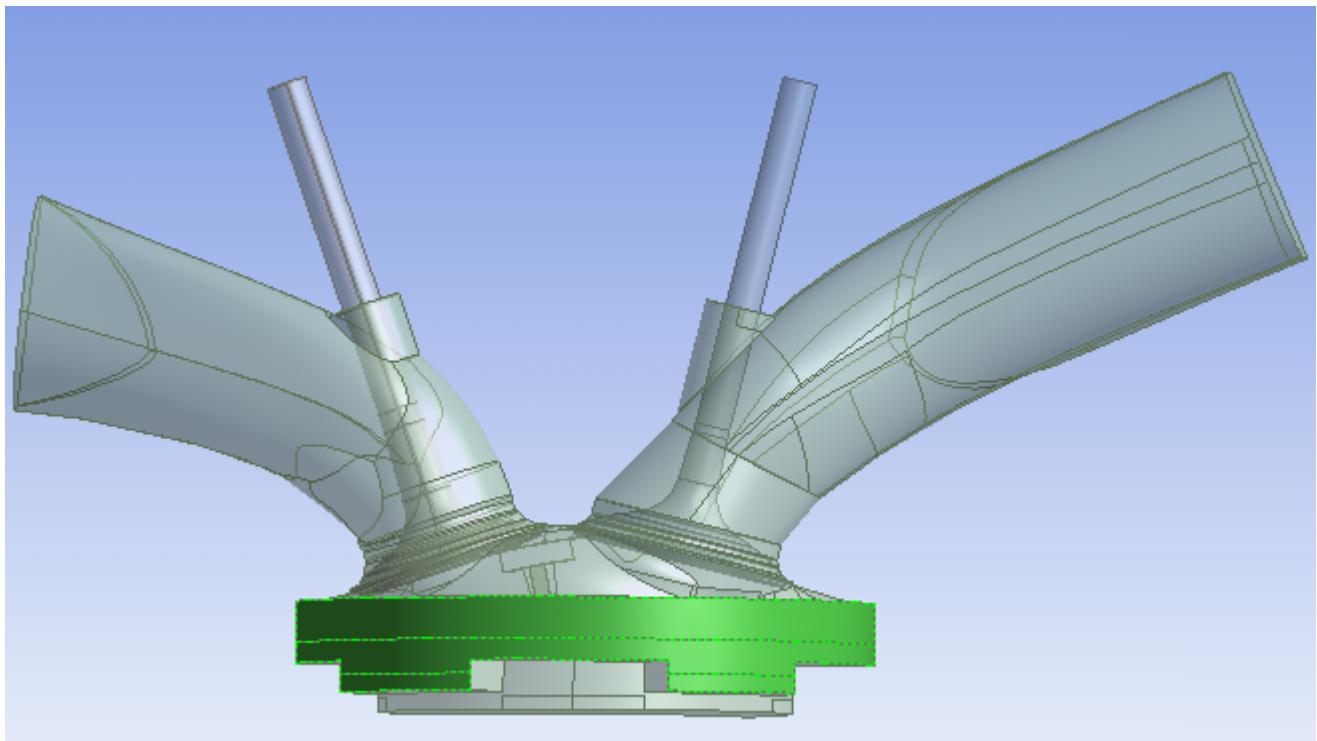
- a. Select **IVO** from **Decomposition Position** drop-down list.
- b. Click next to **Inlet Faces**, select the face of the inlet valve and click **Apply**.



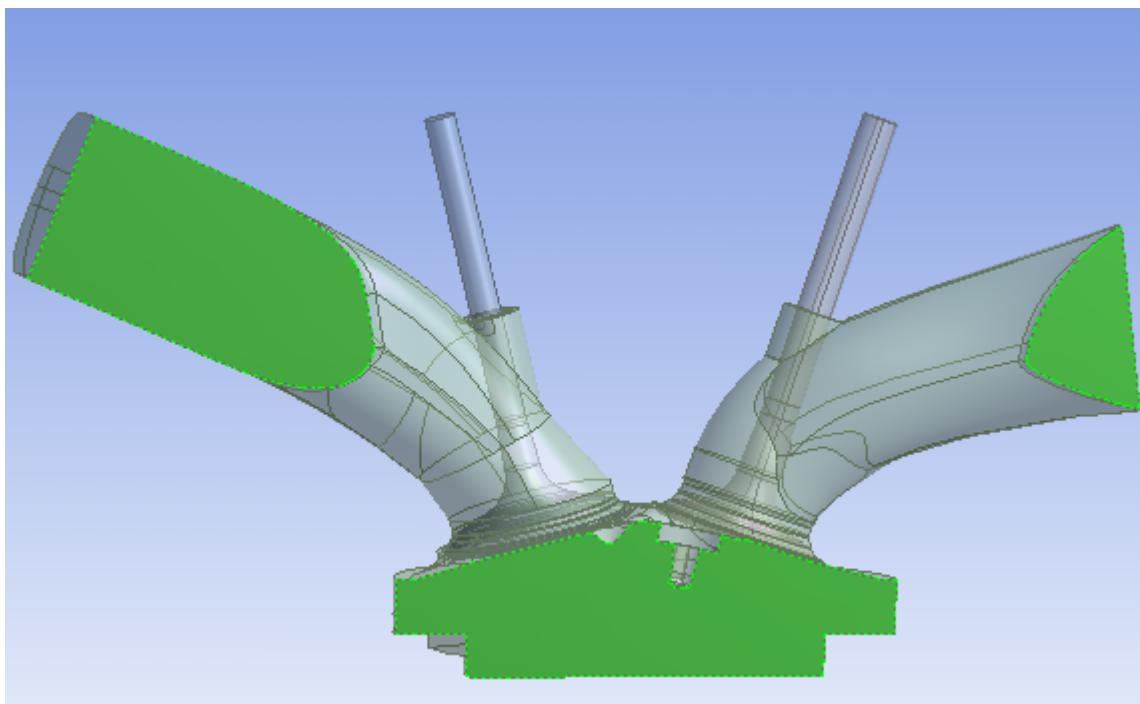
- c. Click next to **Outlet Faces**, select the face of the exhaust valve and click **Apply**.



- d. Select the four faces as shown in Figure 1.2: Cylinder Faces (p. 7) for **Cylinder Faces** and click **Apply**.

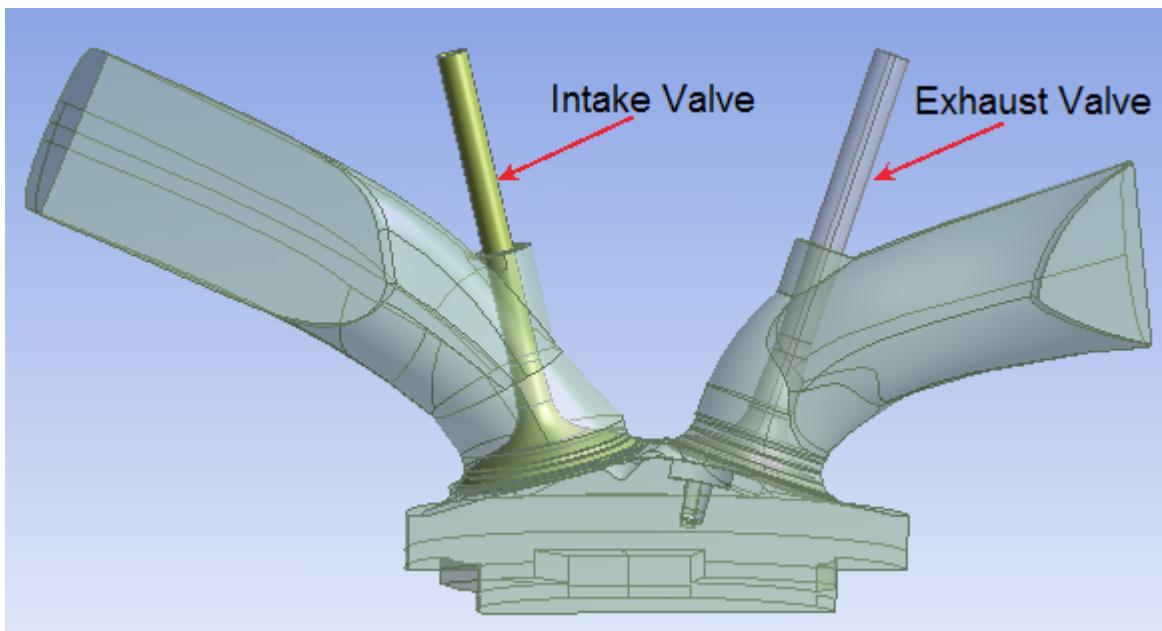
Figure 1.2: Cylinder Faces

- e. Retain selection of **Yes** from the **Symmetry Face Option** drop-down list.
- f. Select the three faces shown in [Figure 1.3: Symmetry Faces \(p. 7\)](#) for **Symmetry Faces** and click **Apply**.

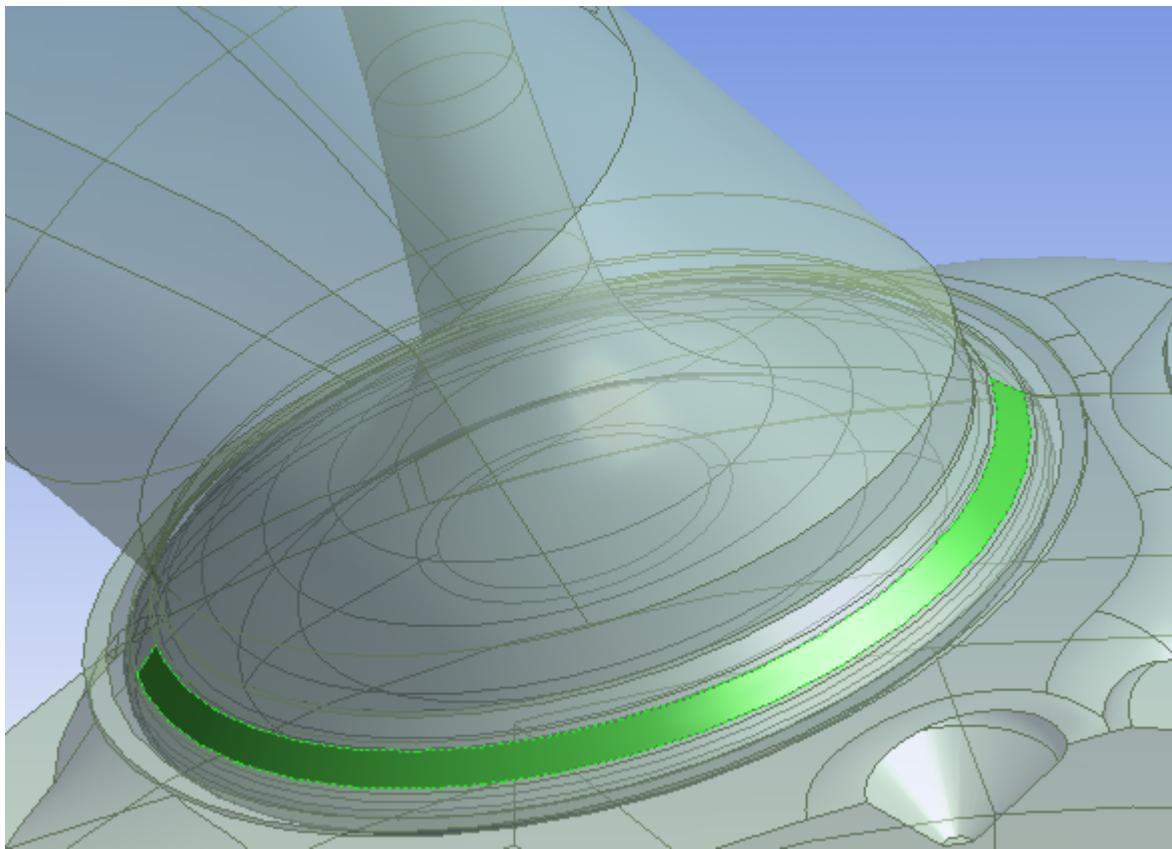
Figure 1.3: Symmetry Faces

- g. Retain the selection of **Full Topology** from the **Topology Option** drop-down list.
- h. Retain selection of **No** for **Crevice Option**.
- i. Retain selection of **No** for **Validate Compression Ratio**.
- j. Retain selection of **InValve** from the **Valve Type** drop-down list.
- k. Select the valve body as shown in [Figure 1.4: Intake Valve \(p. 8\)](#) for **Valve Bodies** and click **Apply**.

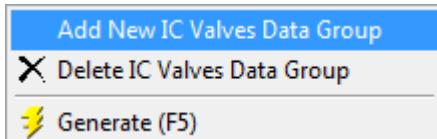
Figure 1.4: Intake Valve



- i. Select the valve seat face as shown in [Figure 1.5: Intake Valve Seat \(p. 9\)](#) for **Valve Seat Faces** and click **Apply**.

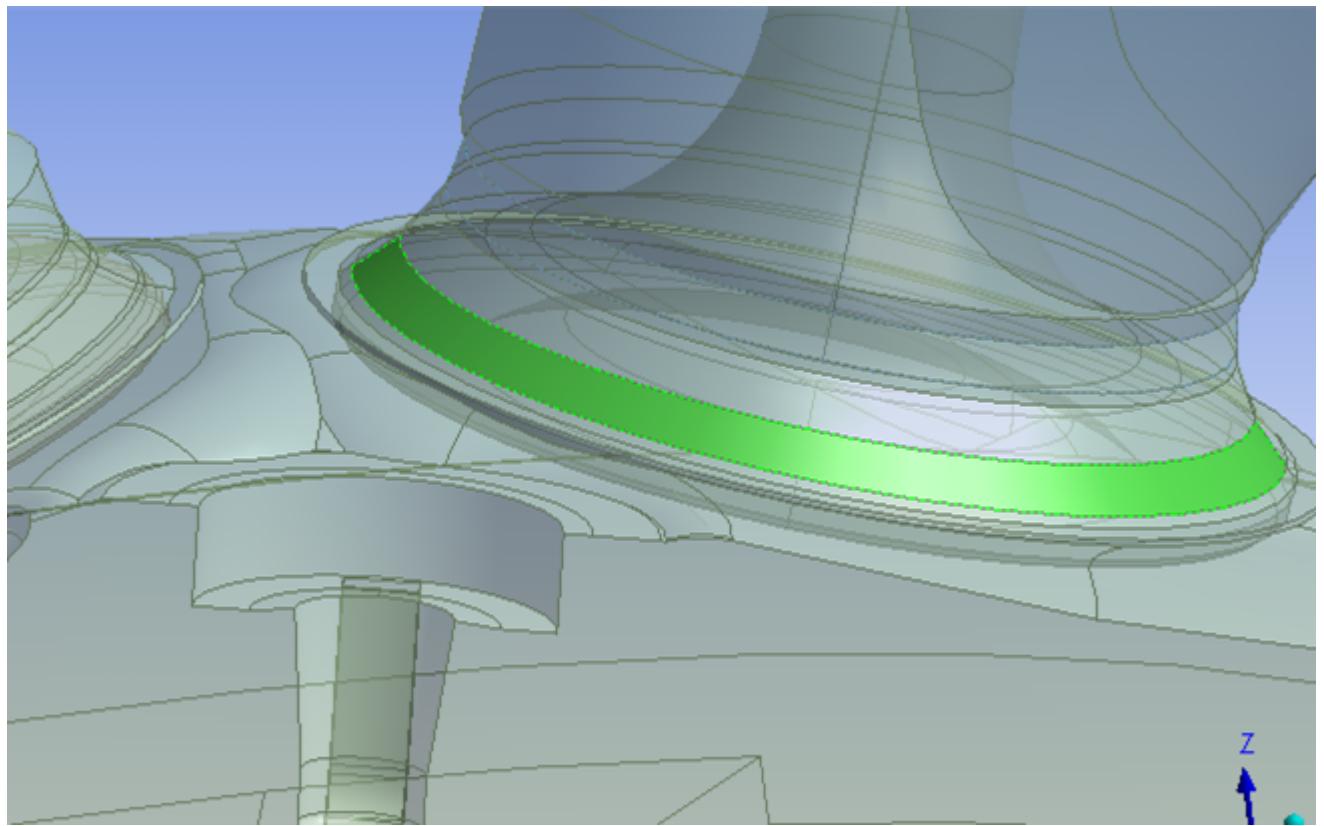
Figure 1.5: Intake Valve Seat

- m. Select **invalve1** from the **Valve Profile** drop-down list.
- n. Right-click on **IC Valves Data** in the **Details of InputManager** and select **Add New IC Valves Data Group** from the context menu.



- o. In this **IC Valves Data** group following the steps for the intake valve, set the other valve body to **ExValve** and set its profile to **exvalve1**. Select the valve seat face of that valve as shown in [Figure 1.6: Exhaust Valve Seat \(p. 10\)](#).

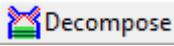
Figure 1.6: Exhaust Valve Seat



- p. Retain the default options under Under **IC Advanced Options**.
- q. After all the settings are done click **Generate** .

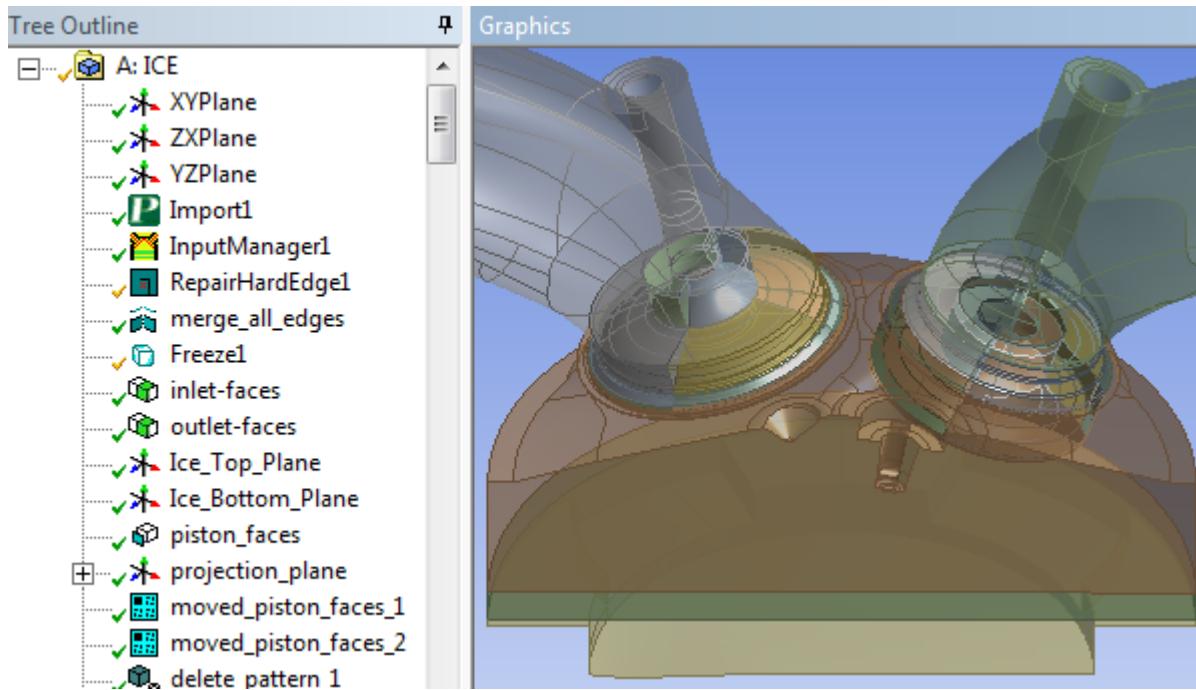
Details View	
Details of InputManager1	
Name	InputManager1
Decomposition Position	IVO
Decomposition Angle	329.6 °
Inlet Faces	1 Face
Outlet Faces	1 Face
Cylinder Faces	4 Faces
Symmetry Face Option	Yes
Symmetry Faces	3 Faces
Topology Option	Full Topology
Crevice Option	No
Validate Compression Ratio	No
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	invalve1
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	exvalve1
IC Animation Inputs (RMB)	
IC Advanced Options (RMB)	
V Layer Slice	Yes
V Layer Slice Angle	15 °
V Layer Approach	4 Layers
Piston Profile Option	No
Decompose Chamber	Yes
Decompose Chamber Inputs	Automatic

You can see that the **Decomposition Angle** is set to **329.6** after clicking on **Generate**.

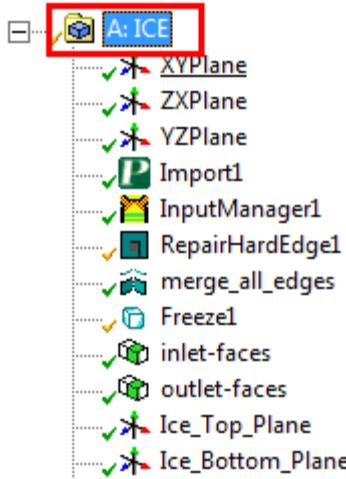
5. Click **Decompose** ( located in the **IC Engine** toolbar).

Note

The decomposition process will take a few minutes.

Figure 1.7: Decomposed Geometry

6. You can close the DesignModeler, after the geometry is decomposed without any errors.



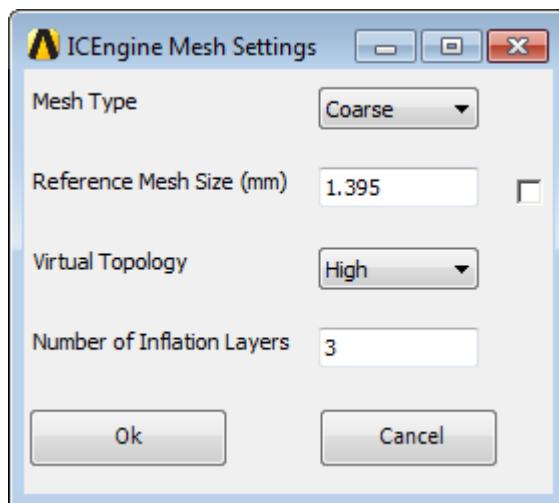
7. Save the project by giving it a proper name (`demo_tut.wbpj`).

File > Save

1.4. Step 3: Meshing

Here you will mesh the decomposed geometry.

1. Click **Edit Mesh Settings** in **Properties of Schematic A4: Mesh** under **IC Engine** to open the **ICEngine Mesh Settings** dialog box.



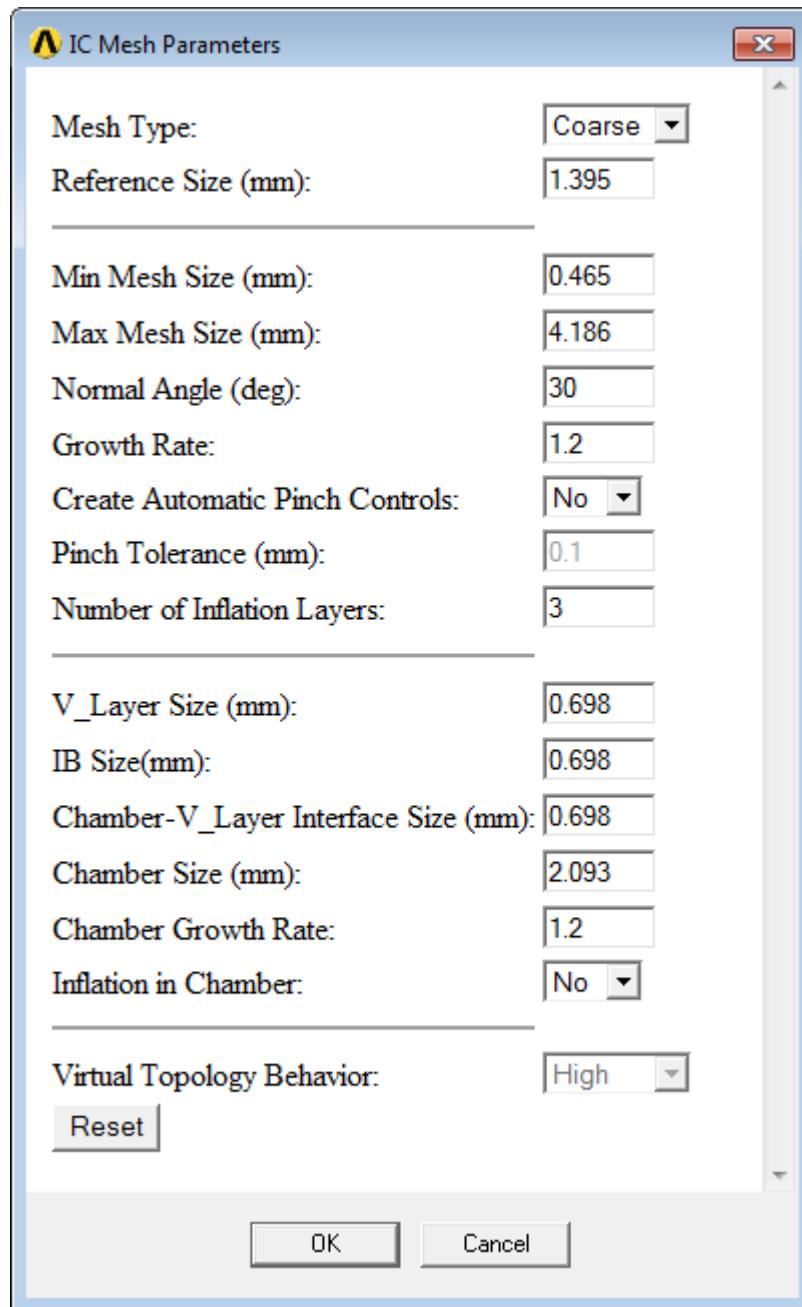
- Select **Coarse** from the **Mesh Type** drop-down list.
- Click **OK** to create mesh controls according to the settings.
- Right-click on **Mesh**, cell 4, and click **Update** from the context menu.

Note

You can also open Meshing by double-clicking on **Mesh** cell in the ICE analysis system.

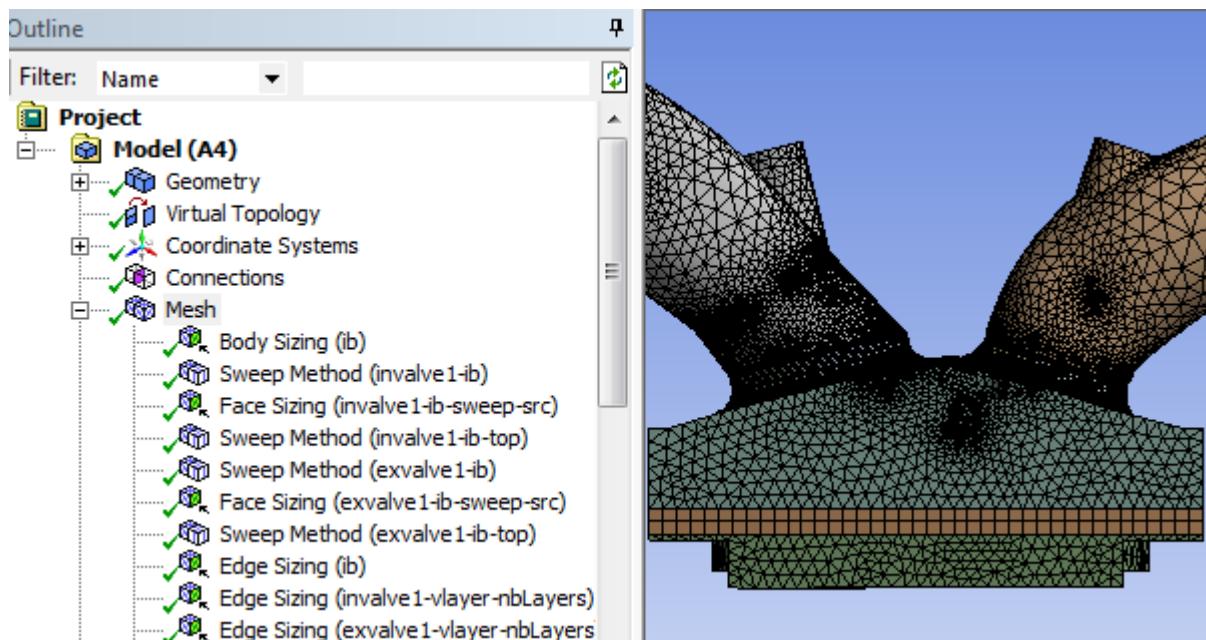


- Click **IC Setup Mesh** (located in the **IC Engine** toolbar).



This shows the mesh controls in details.

2. Retain the default settings in the **IC Mesh Parameters** dialog box and click **OK**
3. Then click **IC Generate Mesh** (located in the IC Engine toolbar) to generate the mesh.

Figure 1.8: Meshed Geometry

4. Close the ANSYS Meshing window once the mesh is generated.
5. Before starting to run the solution, update the **Mesh** cell. You can do this by right-clicking on **Mesh** cell in the Workbench window and selecting **Update** from the context menu.

-
2. Save the project.

File > Save

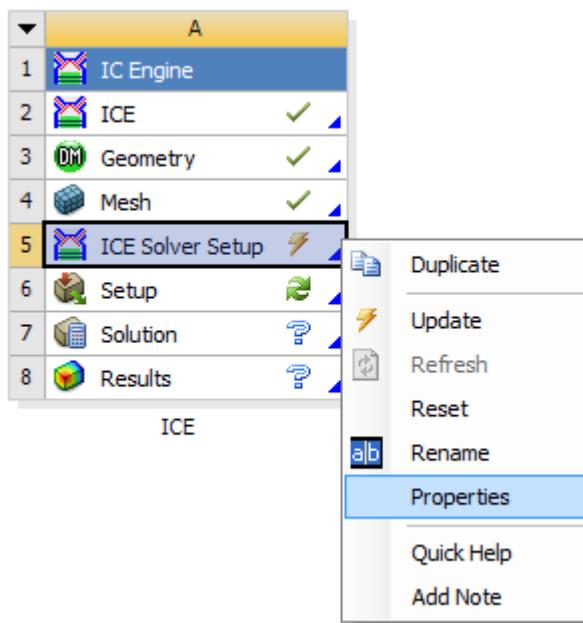
Note

It is a good practice to save the project after each cell update.

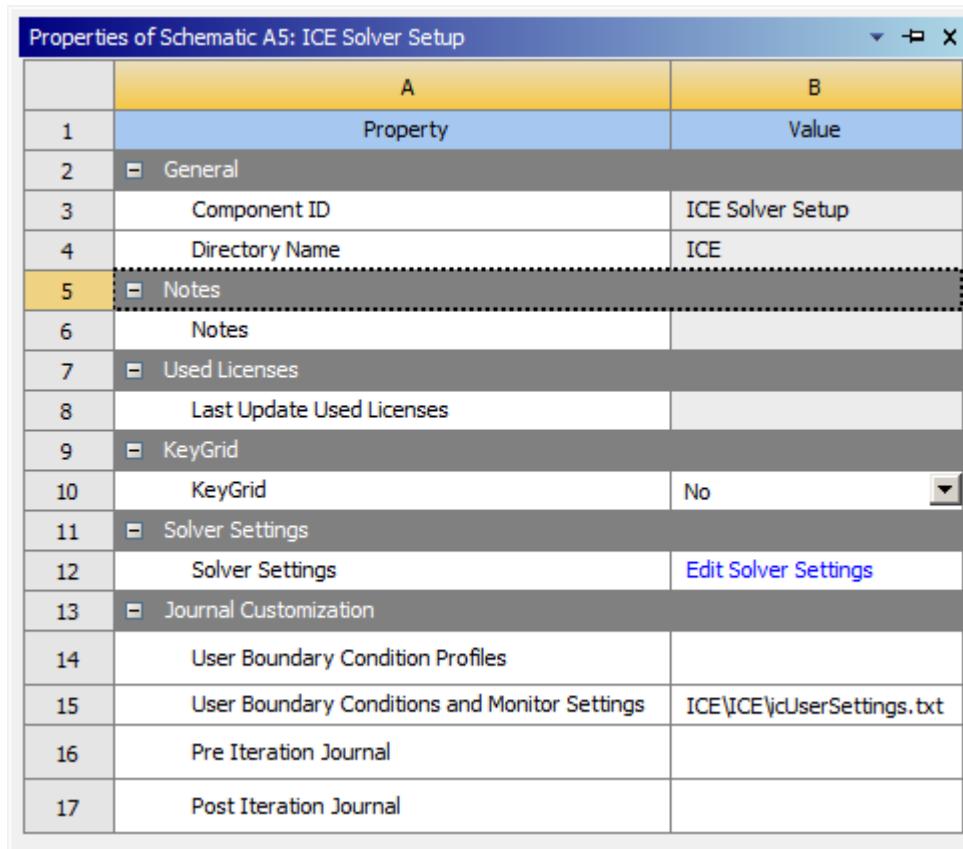
1.5. Step 4: Setting up the Simulation

After the decomposed geometry is meshed properly, you can set boundary conditions, monitors, and postprocessing images or set the Keygrid crank angles. You can also decide which data and images should be included in the report.

1. If the **Properties** view is not already visible, right-click **ICE**, cell 2, and select **Properties** from the context menu.



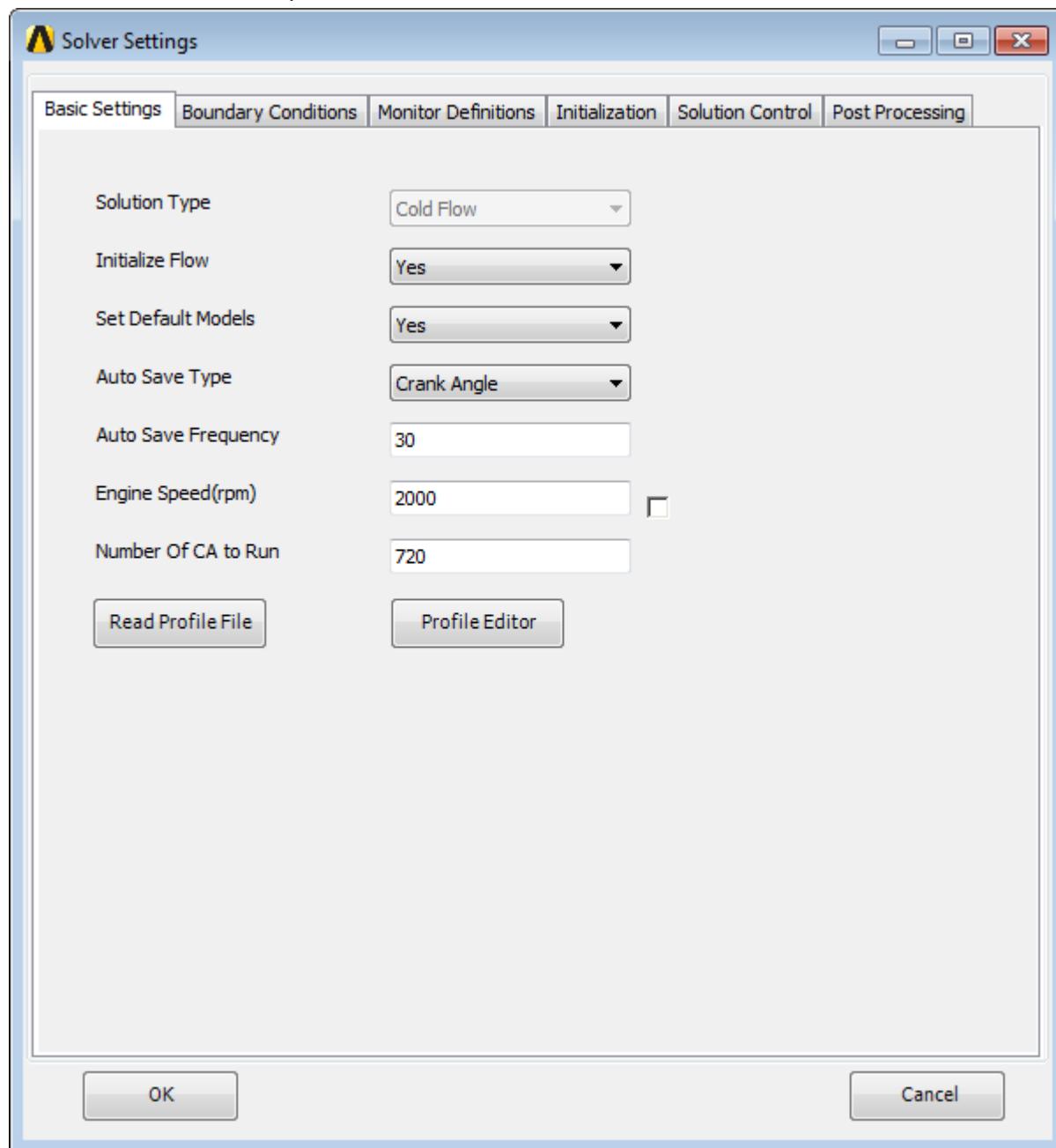
2. Click **Edit Solver Settings** to open the **Solver Settings** dialog box.



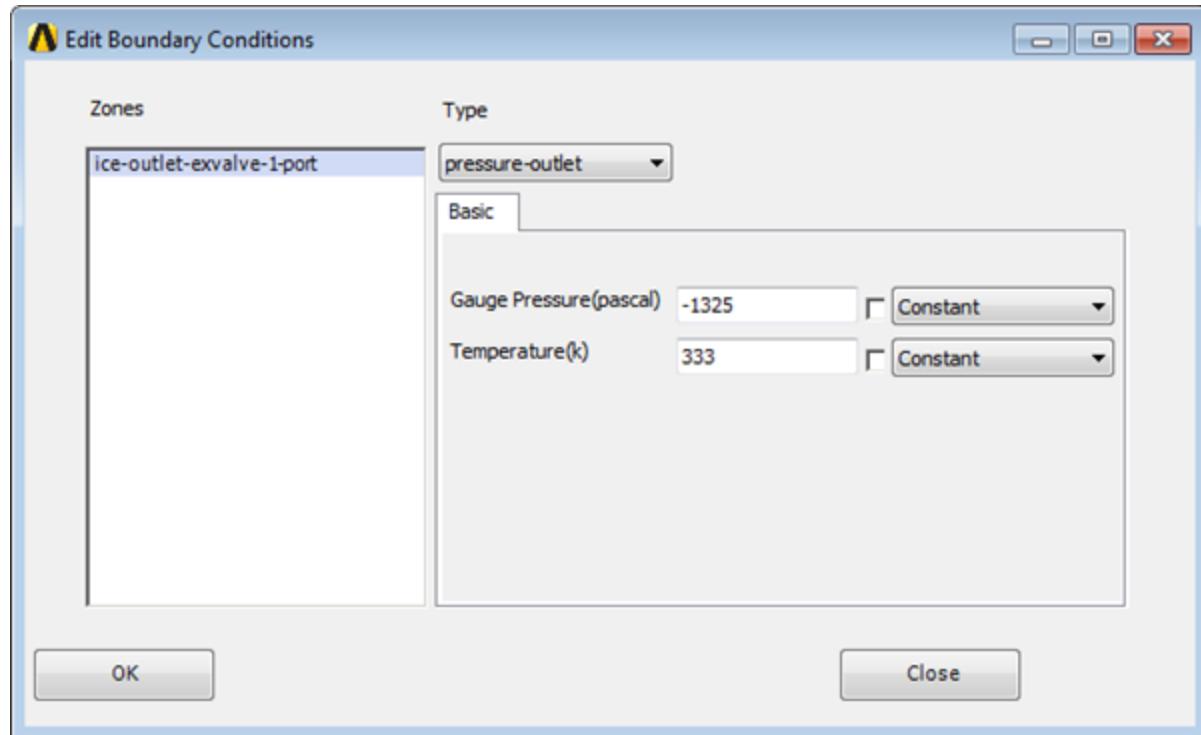
Note

In the **Solver Settings** dialog box you can check the default settings in the various tabs. If required you can change the settings.

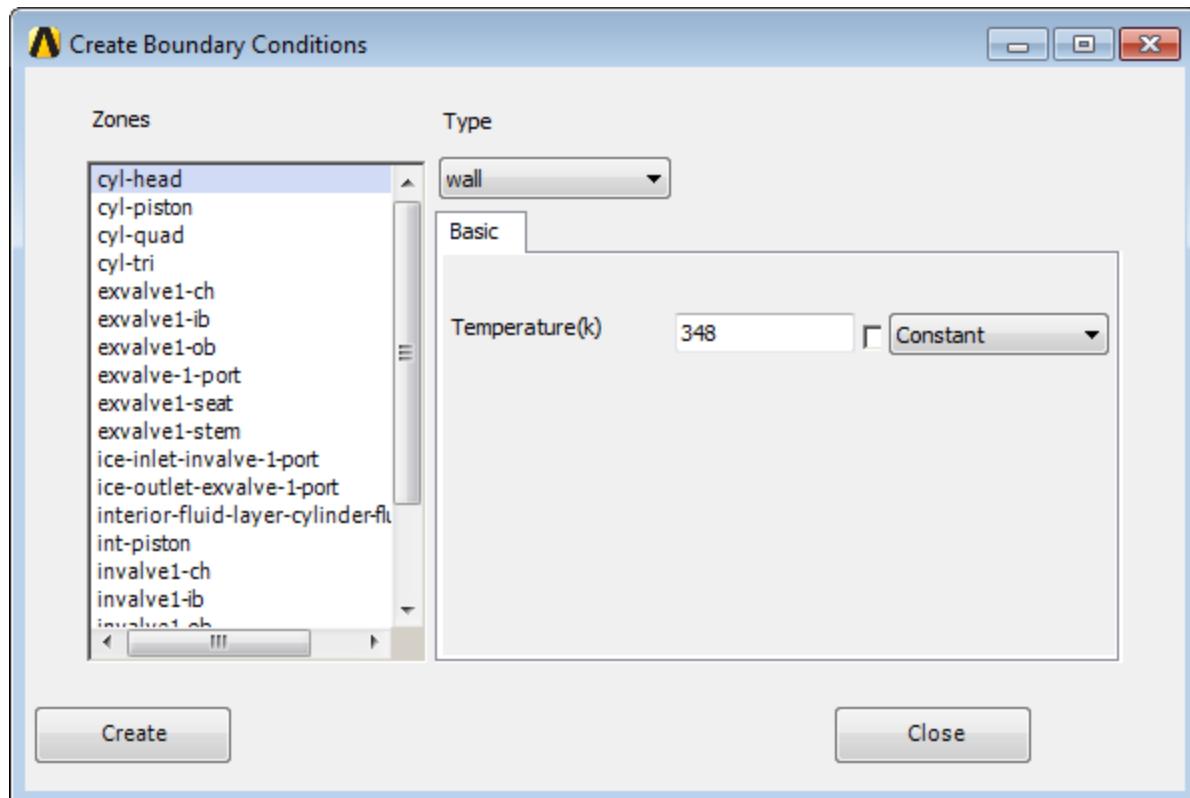
- a. In the **Basic Settings** tab you can see that the under-relaxation factors (URF) and mesh details will be included in the final report. Also the default models are used and the flow is initialized.



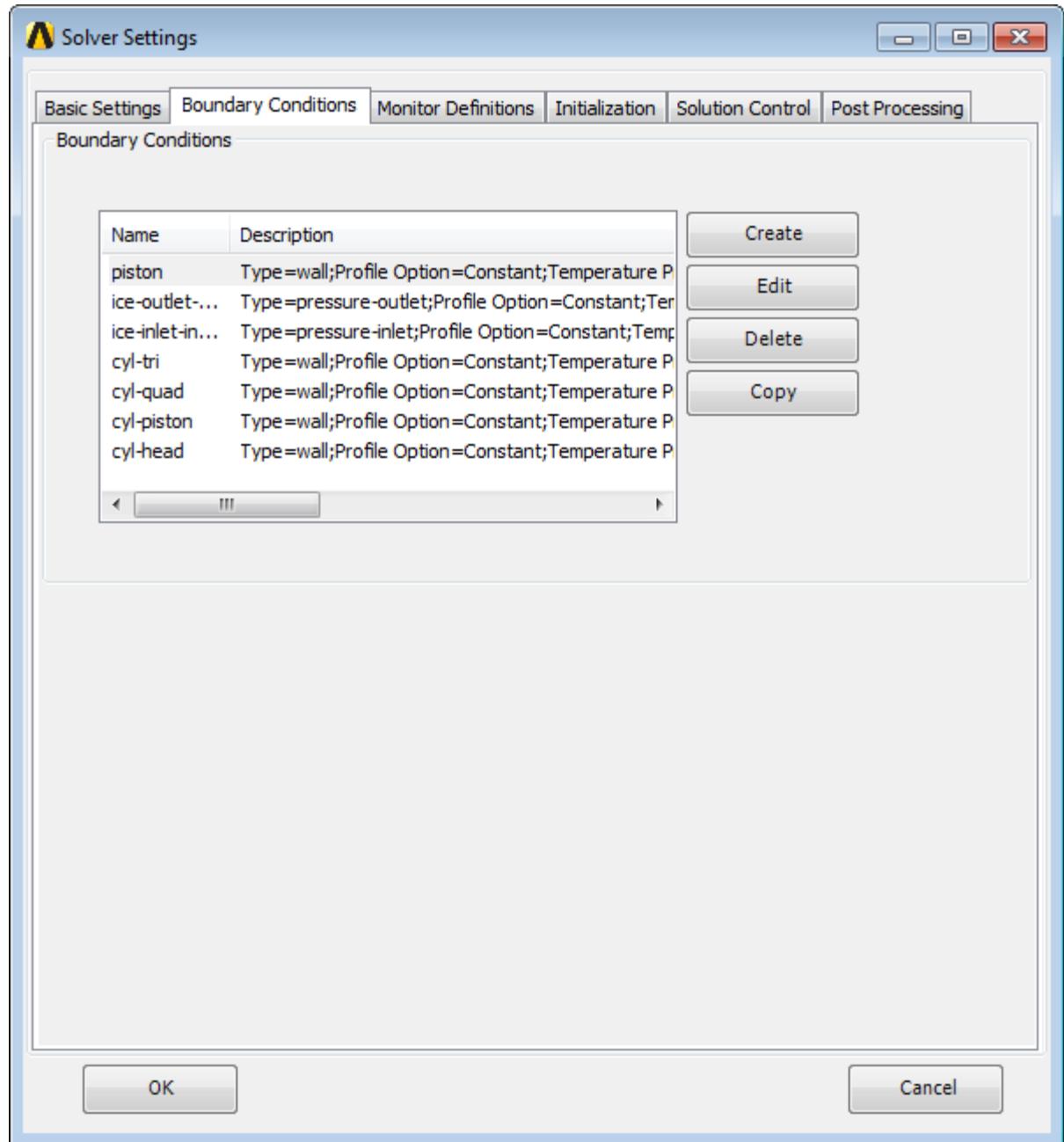
- Enter 2000 for **Engine Speed**.
- b. Click the **Boundary Conditions** tab.
- i. Double-click **ice-outlet-exvalve-1-port** to open the **Edit Boundary Conditions** dialog box.



- A. Enter -1325 pascal for **Gauge Pressure**.
 - B. Enter 333 k for **Temperature**.
 - C. Click **OK** to close the dialog box.
-
- ii. Select **ice-inlet-invalve-1-port** and click **Edit**.
 - A. Enter -21325 pascal for **Gauge Pressure**.
 - B. Enter 313 k for **Temperature**.
 - C. Click **OK** to close the **Edit Boundary Conditions** dialog box.
 - iii. Click **Create** to open the **Create Boundary Conditions** dialog box.



- A. Select **cyl-head** from the list of **Zones**.
- B. Enter 348 k for **Temperature**.
- C. Click **Create**.
- D. Similarly set the liner which includes the three zones —**cyl-piston**, **cyl-quad**, and **cyl-tri** to 318k.
- E. Also set **piston** to 318 k.

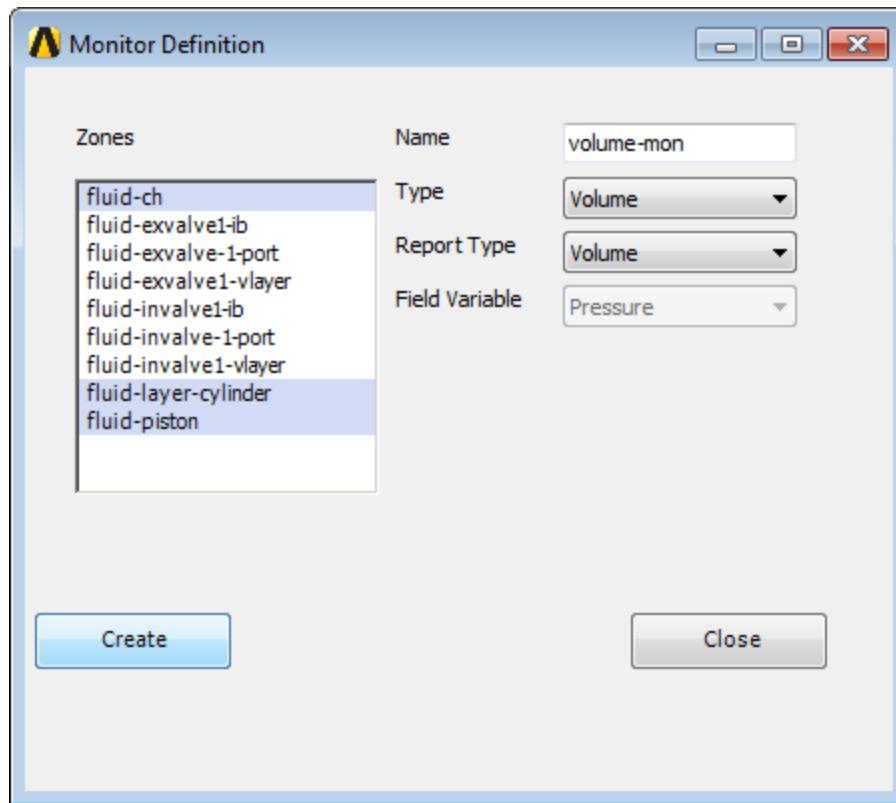


F. Close the **Create Boundary Conditions** dialog box.

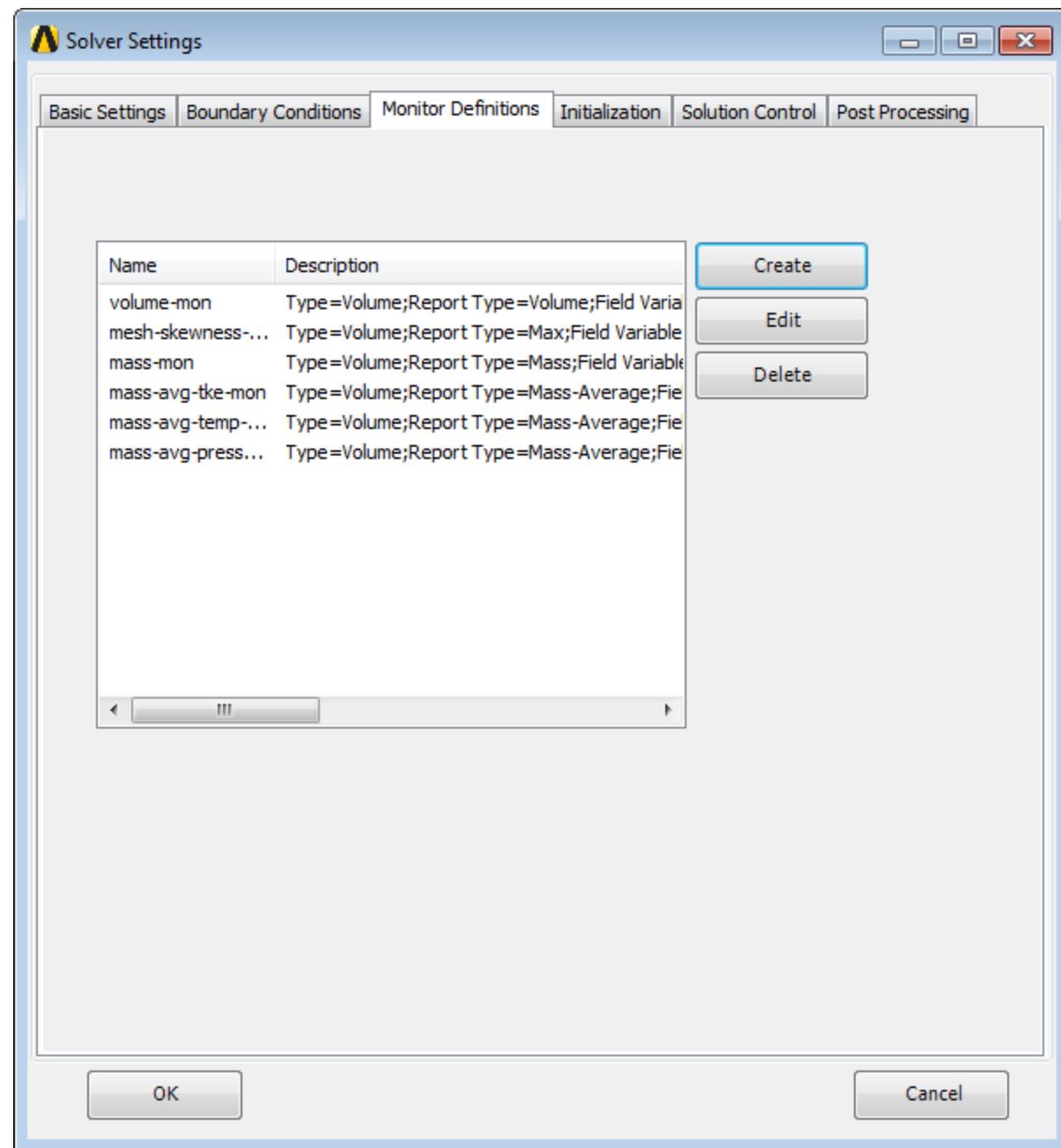
- c. In the **Monitor Definitions** tab you can see that four volume monitors have been set. **Cell Equivolume Skewness, Turbulent Kinetic Energy, Temperature, and Pressure** will be plotted on the zones **fluid-ch, fluid-layer-cylinder, and fluid-piston**.

You will add volume monitors of volume and mass for the chamber zone.

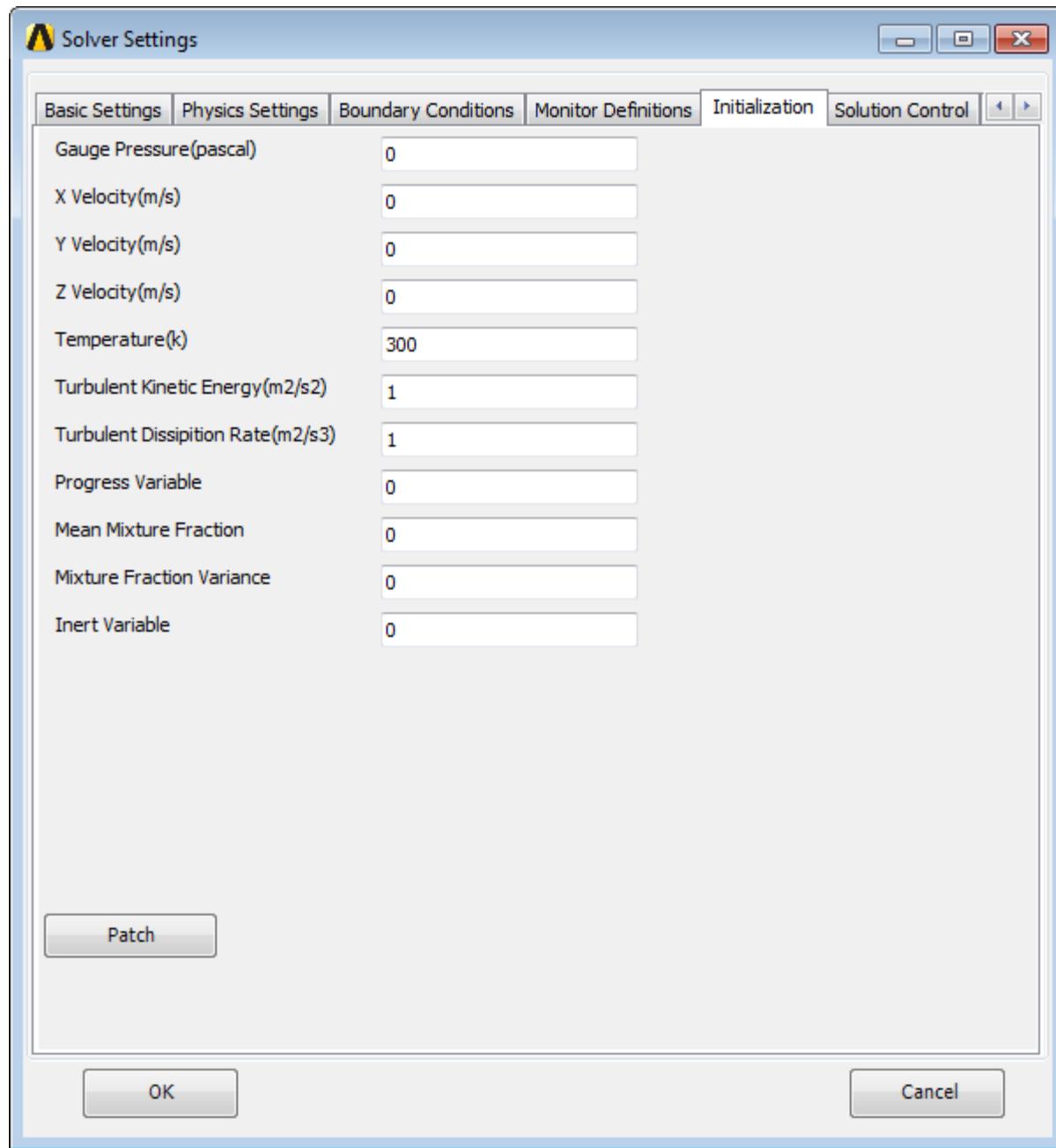
- i. Click **Create** to open the **Monitor Definition** dialog box.



- ii. Select the chambers zones — **fluid-ch**, **fluid-layer-cylinder**, and **fluid-piston** from the list of **Zones**.
- iii. Retain the selection of **Volume** from the **Type** drop-down list.
- iv. Select **Volume** from the **Report Type** drop-down list.
- v. Click **Create**.
- vi. Retaining the selection of the **Zones** and **Type**, select **Mass** from the **Report Type** drop-down list and click **Create**.
- vii. Close the **Monitor Definition** dialog box.

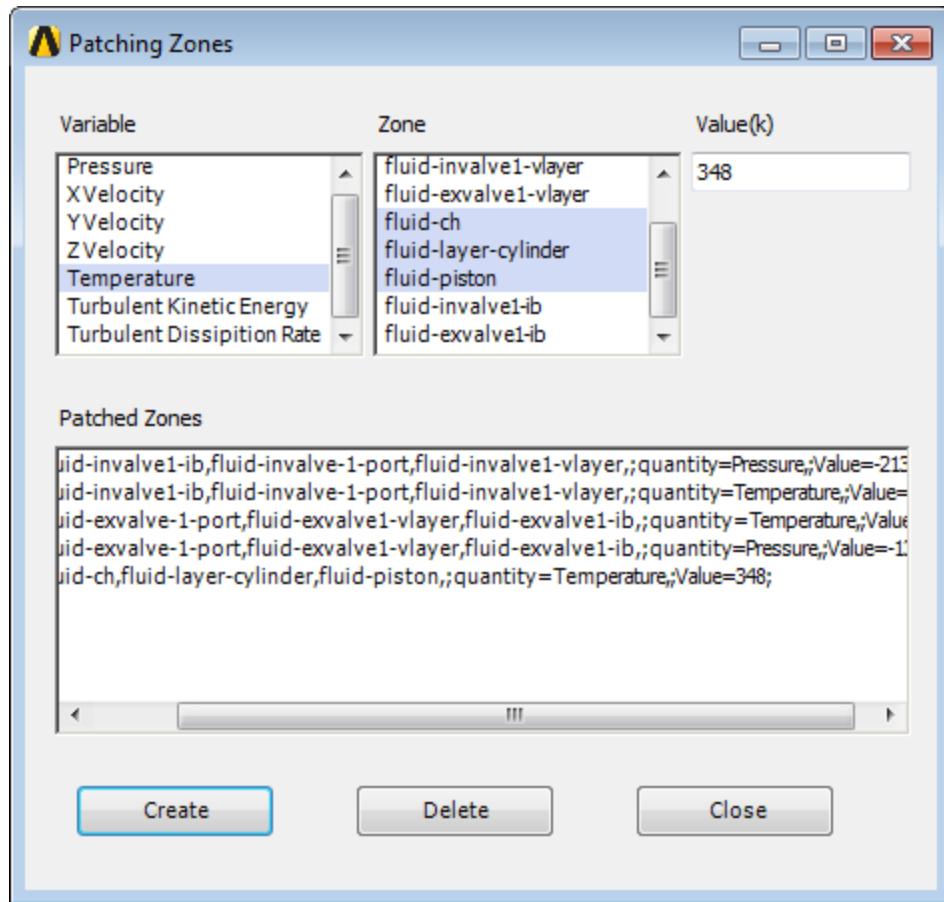


- d. In the **Initialization** tab you can see the default set values for the various parameters. You will be deleting the existing patching conditions and adding new ones.



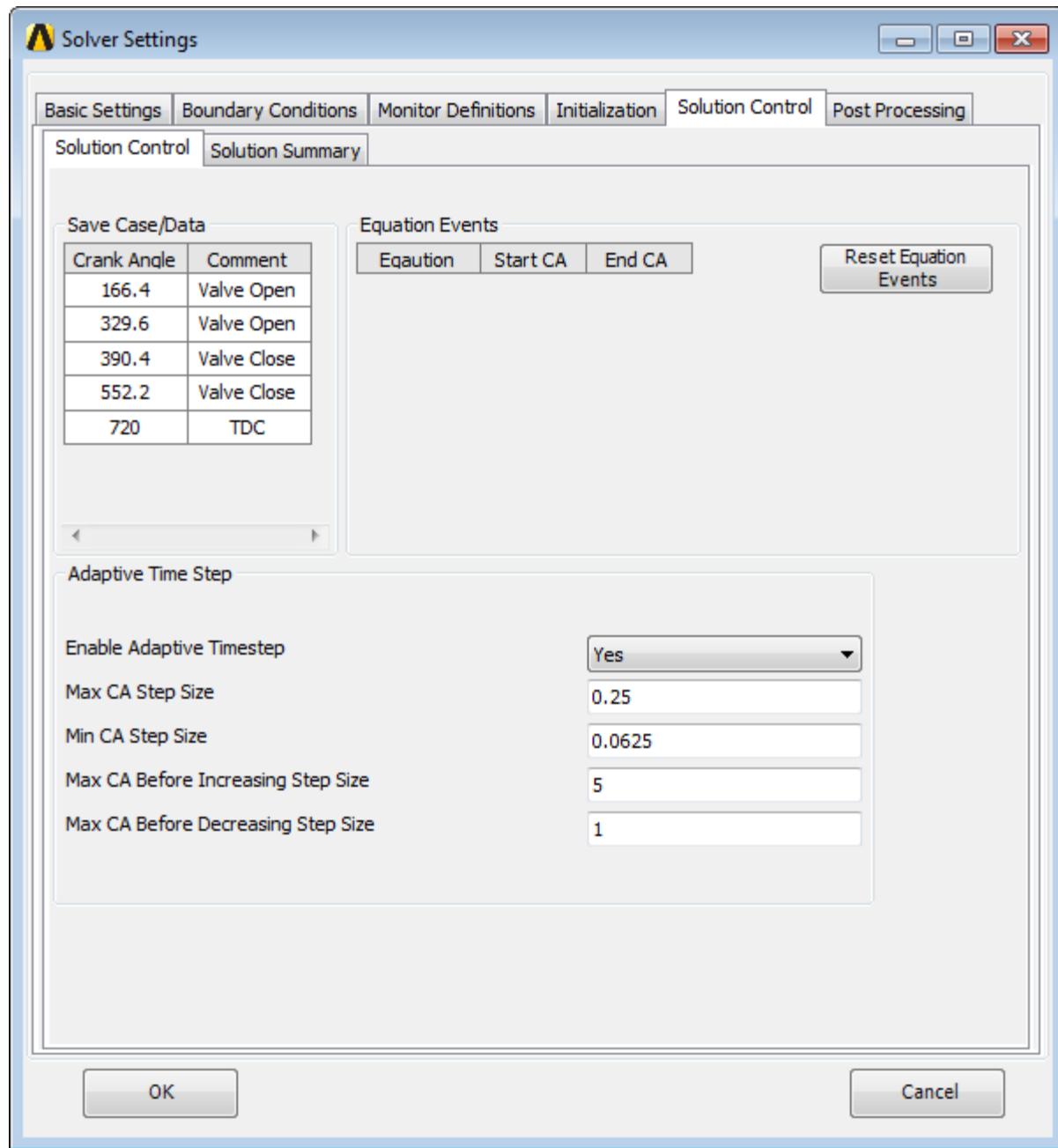
- i. Click **Patch** to open the **Patching Zones** dialog box.
- ii. For the inlet port, select **fluid-invalve-1-port**, **fluid-invalve-1-vlayer**, and **fluid-invalve-1-ib** from the list of **Zone**.
- iii. Select **Pressure** from the list of **Variable**.
- iv. Enter **-21325** for **Value(pascal)** and click **Create**.
- v. Similarly patch the same zones for **Temperature** value **313 k**.
- vi. For outlet port, patch zones **fluid-exvalve-1-port**, **fluid-exvalve-1-vlayer**, and **fluid-exvalve-1-ib** to **Pressure** equal to **-1325 pascal** and **Temperature** equal to **333 k**.

- vii. For chamber select **fluid-chfluid-layer-cylinder**, and **fluid-piston** from the list under **Zone** and patch the **Temperature** to a value of 348 k.

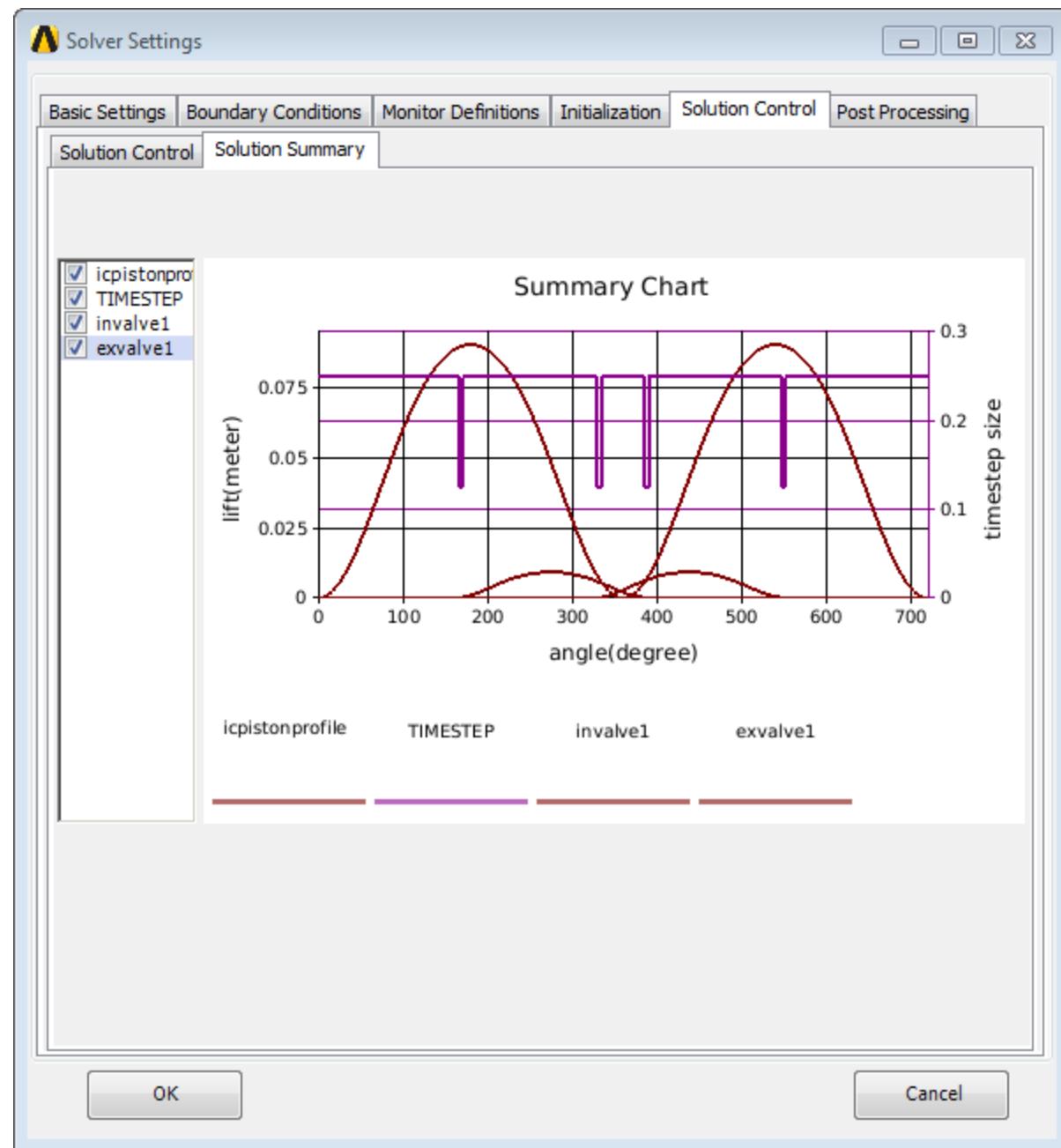


- viii. Close the **Patching Zones** dialog box.

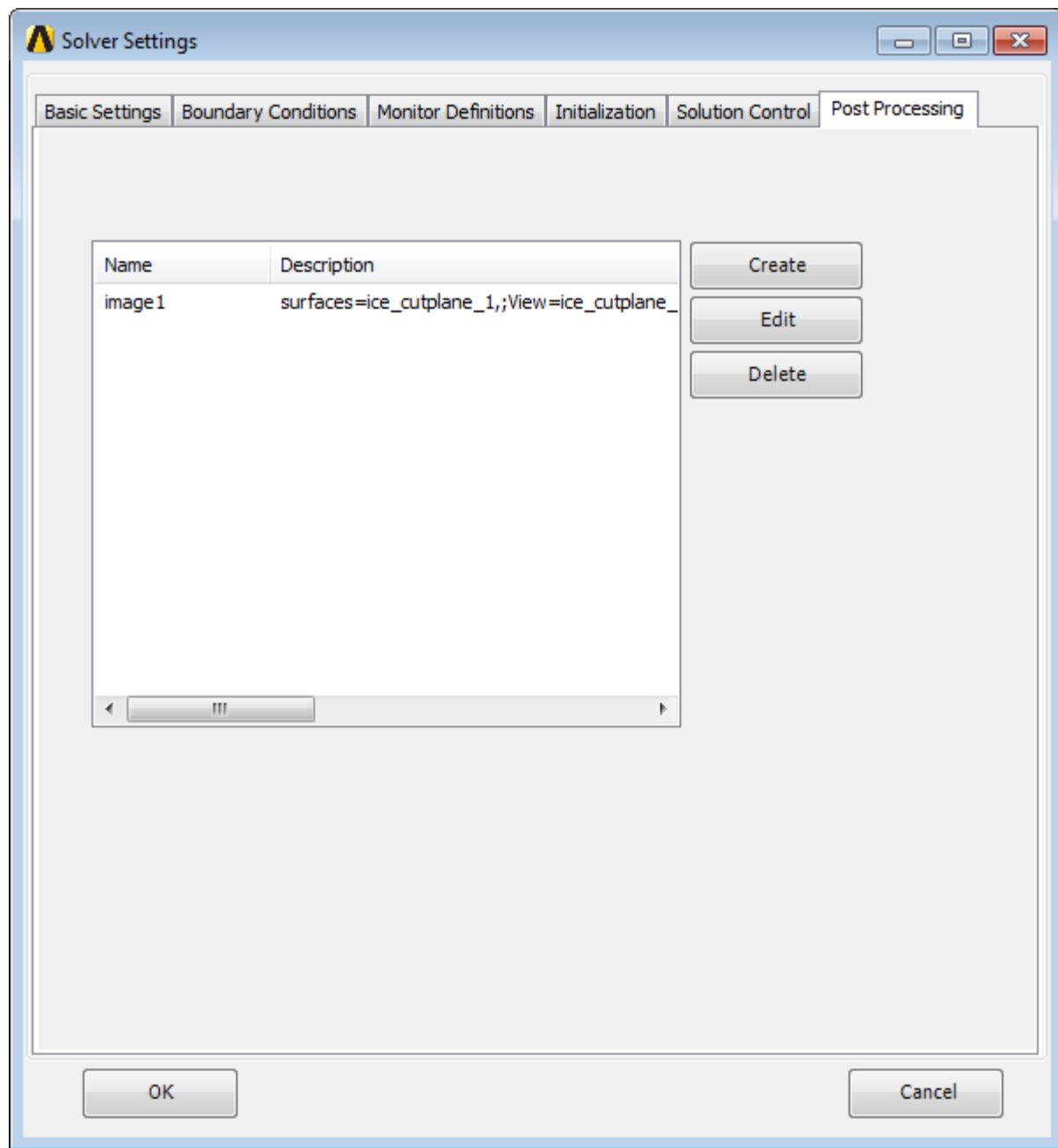
- e. In the **Solution Control** tab select **Yes** from the **Enable Adaptive Timestep** drop-down list. Retain the default settings.



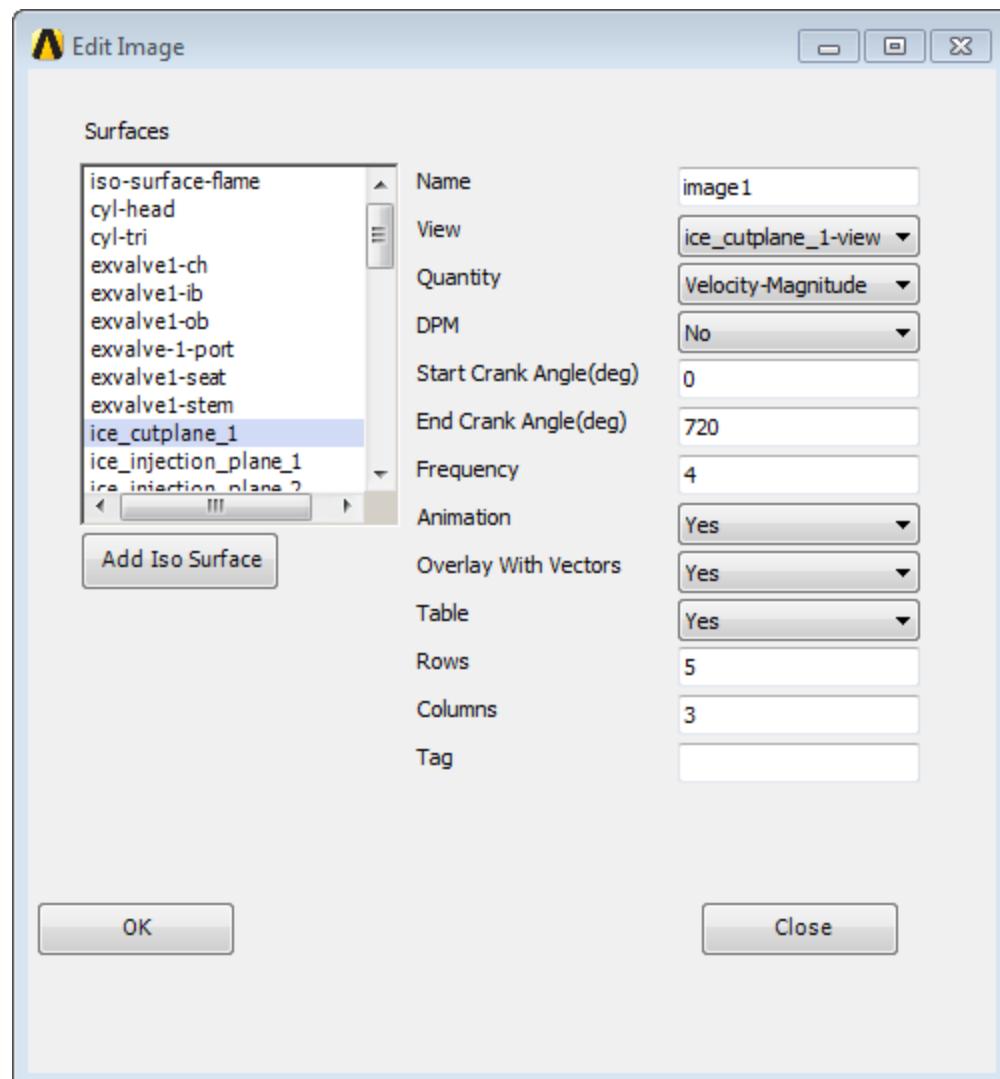
- In the **Solution Summary** tab you can select from the list and check the plots in the **Summary Chart**.



- f. In the **Post Processing** tab you can see that velocity-magnitude contours on the surface of cut-plane will be saved during simulation and displayed in a table format in the report.



- i. Double-click **image-1** to open the **Edit Image** dialog box.



- ii. Select **Yes** from the **Overlay with Vectors** drop-down list.
- iii. Click **OK**.

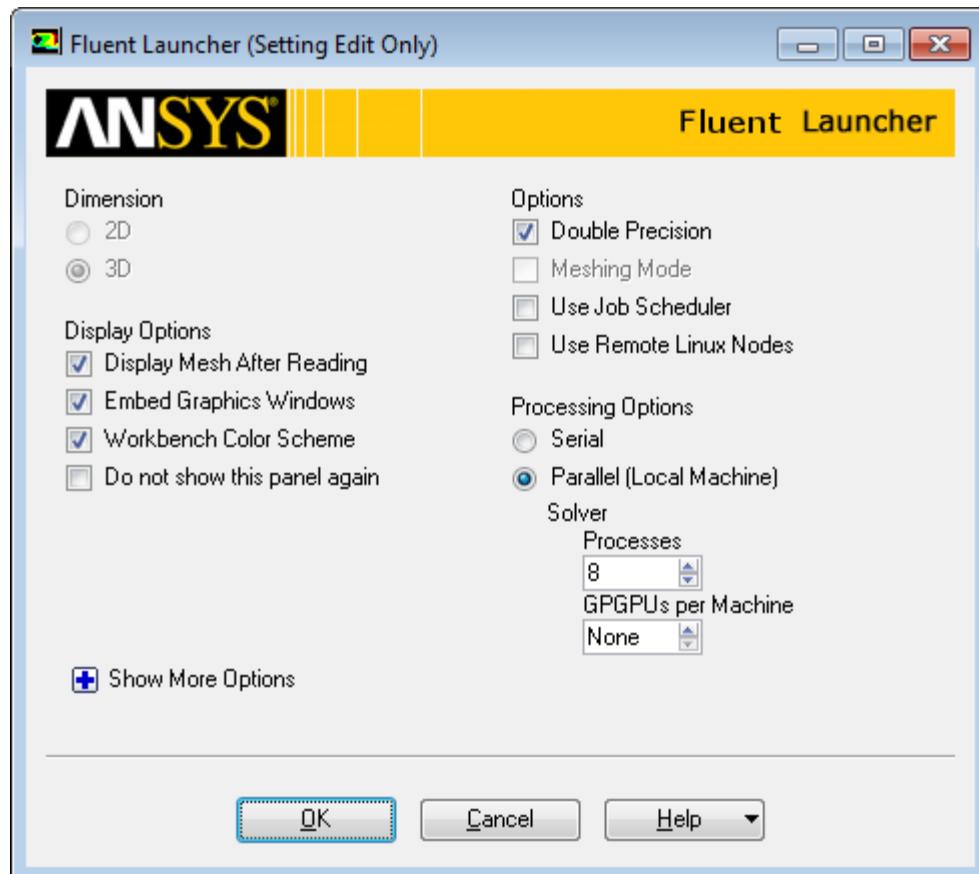
3. Click **OK** to set and close the **Solver Settings** dialog box.
4. Save the project.

File >Save

1.6. Step 5: Running the Solution

In this step you will run the solution.

1. Double-click the **Setup** cell.

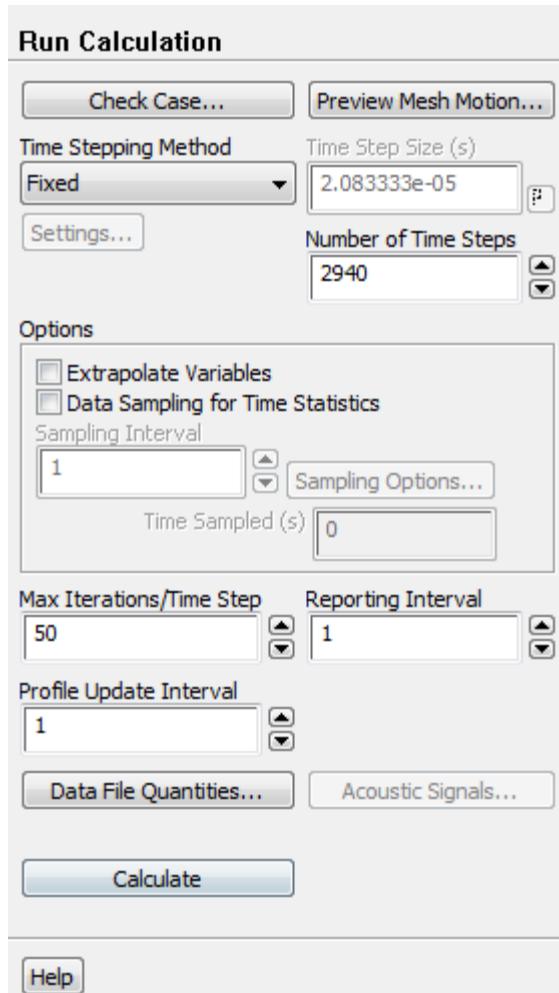


2. Ensure that **Double Precision** is enabled under **Options**.
3. Change the **Processing Options** as required.
4. Click **OK** in the **Fluent Launcher** dialog box.

Note

ANSYS Fluent opens. It will read the mesh file and setup the case till initializing and patching the solution.

5. Click **Run Calculation** in the navigation pane.



6. As you can see **Number of Time Steps** is already set to **2940** which is calculated from the **Number of CA to Run** set in the **Basic Settings** tab of **Solver Settings** dialog box.
7. Click **Calculate**.
8. Close the Fluent window after it finishes running the solution.
9. Save the project.

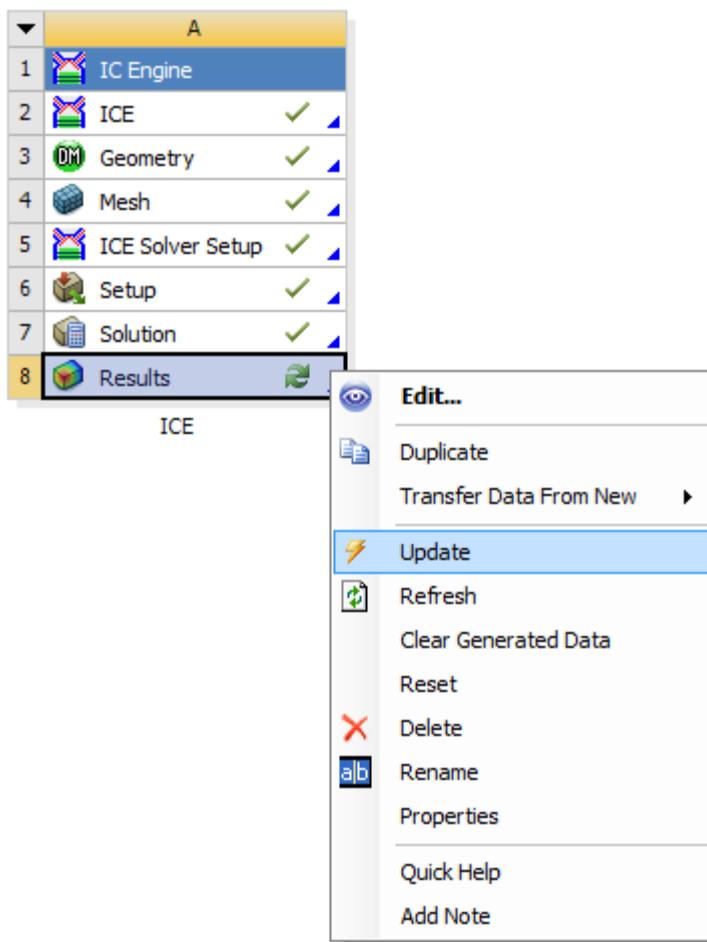
File > Save

Note

Running the solution can take around forty hours for a 8 CPU machine. You can open the project file provided and check the results of intermediate steps in CFD-Post. The intermediate solution is also provided as a separate project. You can open this project and continue the simulation.

1.7. Step 6: Obtaining the Results

1. Go to the ANSYS Workbench window.
2. Right-click on the **Results** cell and click on **Update** from the context menu.



- Once the **Results** cell is updated, view the files by clicking on **Files** from the **View** menu.

View > Files

	A	B	C	D	E	F
1	Name	Cell ID	Size	Type	Date Modified	Location
107	Chart002.png	A7	5 KB	Default File	8/12/2011 3:49:11 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
108	Chart003.png	A7	5 KB	Default File	8/12/2011 3:49:12 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
109	Chart004.png	A7	6 KB	Default File	8/12/2011 3:49:12 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
110	Chart005.png	A7	6 KB	Default File	8/12/2011 3:49:12 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
111	ice-anim-mesh-on-plane1-.gif	A7	0 B	Default File	8/12/2011 3:49:11 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
112	ice-anim-vel-mag-on-plane1-.gif	A7	0 B	Default File	8/12/2011 3:49:11 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
113	ice-profile-plot.jpg	A7	59 KB	Default File	8/12/2011 3:49:05 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
114	ice-profile-plot1.inp	A7	67 KB	Default File	8/12/2011 3:49:05 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
115	Report.html	A7	0 KB	Default File	8/12/2011 3:49:12 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Post\Report\Report
116	designpoint.wbdp			Open Containing Folder	Point File	8/12/2011 11:51:39 AM
117	cortexerror.log			File Type Filter...		8/11/2011 6:22:13 PM
118	ICE-2-00000.cas.gz	A	25 MB	FLUENT Case File	8/1/2011 8:24:07 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
119	ICE-2-00000.dat.gz	A	330 KB	FLUENT Data File	8/1/2011 8:24:30 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
120	ice-anim-mesh-on-plane1-.gif	A	181 MB	.gif	8/11/2011 4:47:36 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
121	ice-anim-mesh-on-plane1-0015.jpg	A	54 KB	.jpg	8/1/2011 5:27:12 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
122	ice-anim-mesh-on-plane1-0030.jpg	A	55 KB	.jpg	8/1/2011 5:54:34 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
123	ice-anim-mesh-on-plane1-0045.jpg	A	55 KB	.jpg	8/1/2011 6:14:44 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
124	ice-anim-mesh-on-plane1-0060.jpg	A	55 KB	.jpg	8/1/2011 6:31:20 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
125	ice-anim-mesh-on-plane1-0075.jpg	A	56 KB	.jpg	8/1/2011 6:48:08 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
126	ice-anim-mesh-on-plane1-0090.jpg	A	56 KB	.jpg	8/1/2011 7:07:27 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
127	ice-anim-mesh-on-plane1-0105.jpg	A	57 KB	.jpg	8/1/2011 7:24:29 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
128	ice-anim-mesh-on-plane1-0120.jpg	A	57 KB	.jpg	8/1/2011 7:42:26 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent
129	ice-anim-mesh-on-plane1-0135.jpg	A	58 KB	.jpg	8/1/2011 7:59:53 PM	E:\tut\tut\demo-tut_files\dp0\ICE\Fluent

4. Right-click on **Report.html** from the list of files, and click **Open Containing Folder** from the context menu.
5. In the **Report** folder double click on **Report.html** to open the report.

ANSYS®

Title
IC Engine Cold Flow Simulation Report

Date
2014/10/02 00:54:01

Contents

- [1. File Report](#)
 - [Table 1 File Information for Cold_flow_demo_tut](#)
- [2. Mesh Report](#)
 - [Table 2 Mesh Information for Cold_flow_demo_tut](#)
 - [Chart 1 Monitor: Max Cell Equivolume Skew \(fluid-piston fluid-layer-cylinder fluid-ch\)](#)
 - [Table 3 Cell count at crank angles](#)
- [3. Setup](#)
 - [3.1. Physics](#)
 - [Table 4 Boundary Conditions](#)
 - [3.2. Piston and Valves Lift profiles](#)
 - [3.3. Valves Lift profiles](#)
 - [3.4. Relaxations](#)
 - [Table 5 Relaxation changes through events](#)
 - [3.5. Dynamic Mesh Setup](#)
 - [Table 6 Dynamic Mesh Events](#)
 - [3.6. IC Engine System Inputs](#)
- [4. Solution Data](#)
 - [4.1. Animation: mesh-on-ice_cutplane_1](#)
 - [4.2. Animation: velocity-magnitude on ice_cutplane_1](#)
 - [4.3. Table: mesh-on-ice_cutplane_1](#)
 - [4.4. Table: velocity-magnitude on ice_cutplane_1](#)
 - [4.5. Table: Residuals](#)
 - [4.6. Charts](#)
 - [Chart 2 Last iteration residual values corresponding to each time step](#)
 - [Chart 3 Swirl Ratio](#)
 - [Chart 4 Tumble Ratio](#)
 - [Chart 5 Cross Tumble Ratio](#)
 - [Chart 6 Adapt-time-step-changes](#)
 - [Chart 7 Number of Iterations per Time Step](#)
 - [Chart 8 Monitor: Mass-Average Static Pressure \(fluid-piston fluid-layer-cylinder fluid-ch\)](#)
 - [Chart 9 Monitor: Mass-Average Static Temperature \(fluid-piston fluid-layer-cylinder fluid-ch\)](#)
 - [Chart 10 Monitor: Mass-Average Turbulent Kinetic Energy \(k\) \(fluid-piston fluid-layer-cylinder fluid-ch\)](#)
 - [Chart 11 Monitor: Mass Static Pressure \(fluid-piston fluid-layer-cylinder fluid-ch\)](#)
 - [Chart 12 Monitor: Volume Static Pressure \(fluid-piston fluid-layer-cylinder fluid-ch\)](#)

- You can check the node count and mesh count of the cell zones in the table, **Mesh Information for ICE**.

1. File Report

Table 1. File Information for ICE

Case	ICE
File Path	E:\ICETutorials16\New folder\demo_tut_files\dp0\ICE\Fluent\ICE-4-02961.dat.gz
File Date	07 July 2014
File Time	11:38:18 AM
File Type	FLUENT
File Version	16.0.0

2. Mesh Report

Table 2. Mesh Information for Cold_flow_demo_tut

Domain	Nodes	Elements
fluid ch	44314	219131
fluid exvalve 1 port	35099	122576
fluid exvalve1 ib	6612	4988
fluid exvalve1 vlayer	72000	64600
fluid invalve 1 port	64615	227128
fluid invalve1 ib	4482	3402
fluid invalve1 vlayer	25000	19200
fluid layer cylinder	21228	38052
fluid piston	3816	17008
All Domains	277166	716085

- You can see the change in volume-average cell skewness with crank angle, in the chart **Monitor: Max Cell Equivolume Skew (fluid-piston fluid-layer-cylinder fluid-ch)**. Also the table **Cell count at crank angles**, shows the mesh count at 0 and 180 degree crank angles.

Chart 1. Monitor: Max Cell Equivolume Skew (fluid-piston fluid-layer-cylinder fluid-ch)

Monitor: Max Cell Equivolume Skew (fluid-piston fluid-layer-cylinder fluid-ch)

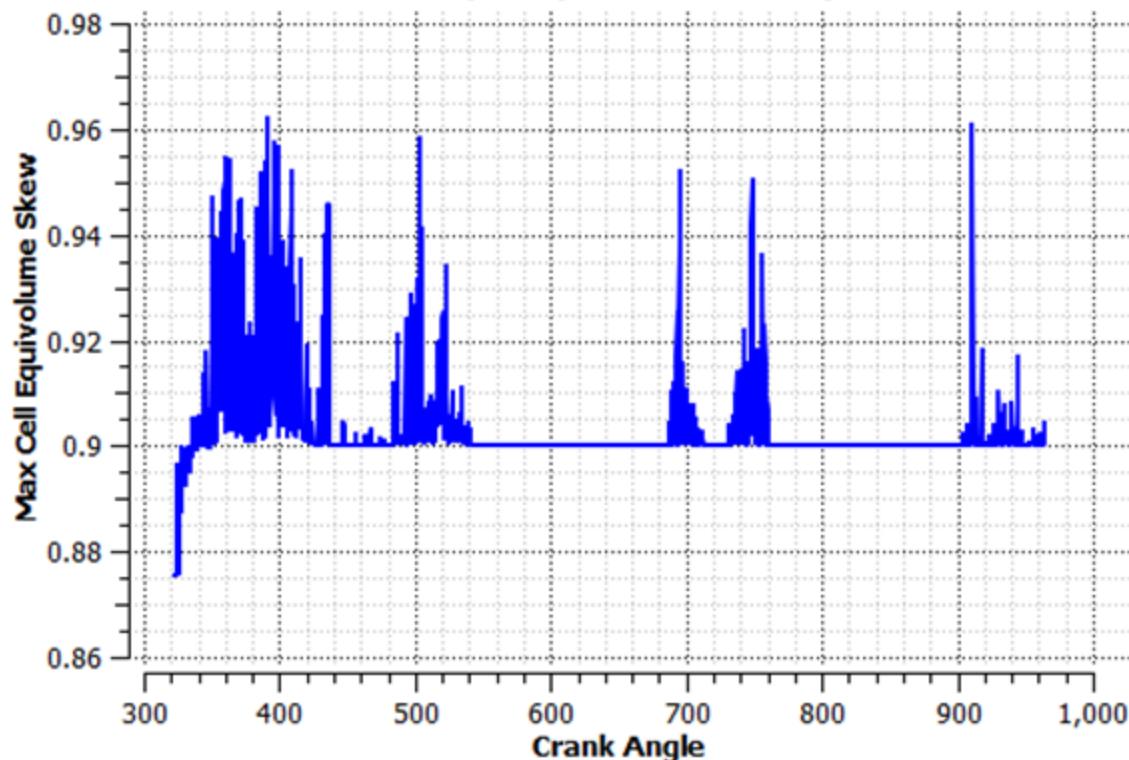


Table 3. Cell count at crank angles

Crank Angle	Cell Count
0.000e+00	5.809e+05
1.800e+02	6.592e+05

- You can see the boundary conditions set, in the table **Boundary Conditions**.

3. Setup

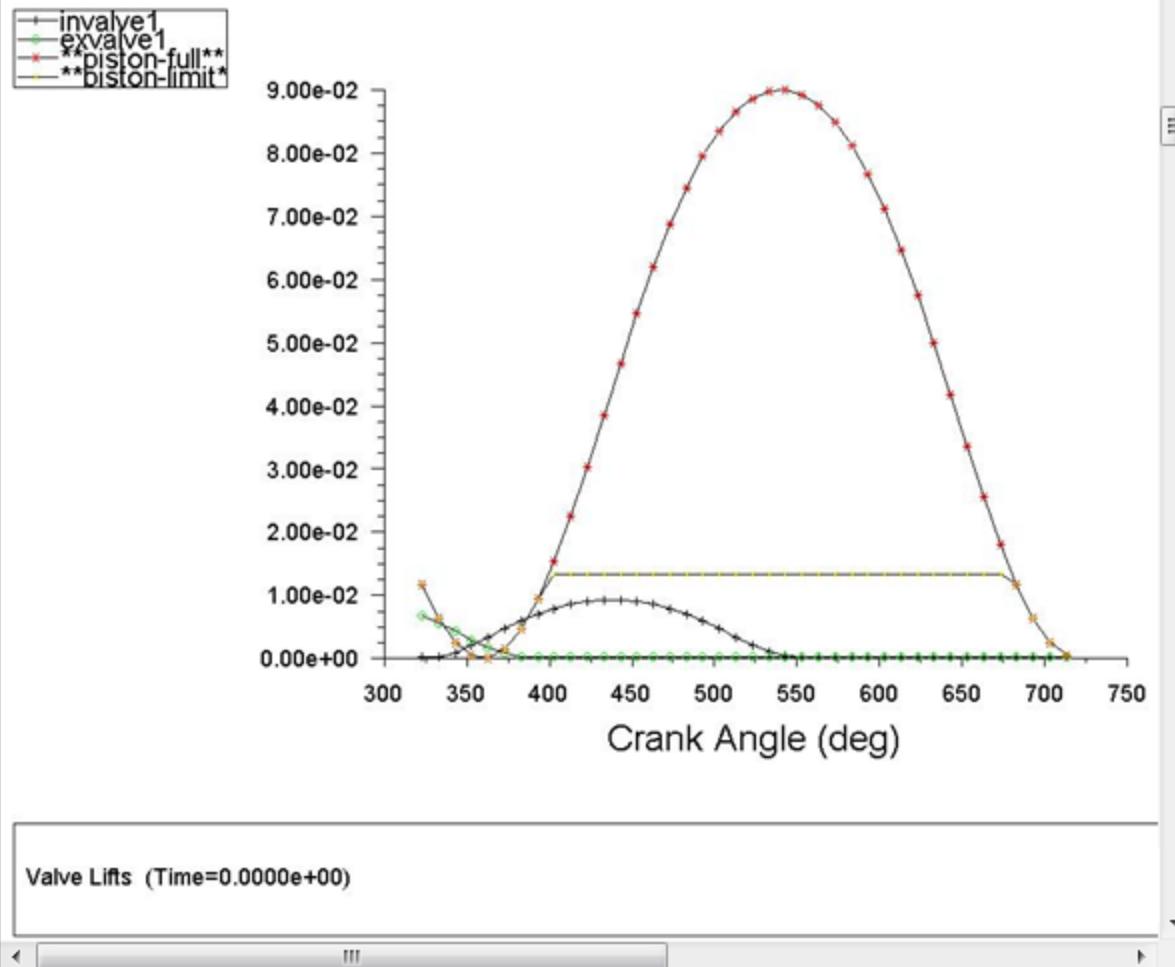
3.1. Physics

Table 4. Boundary Conditions

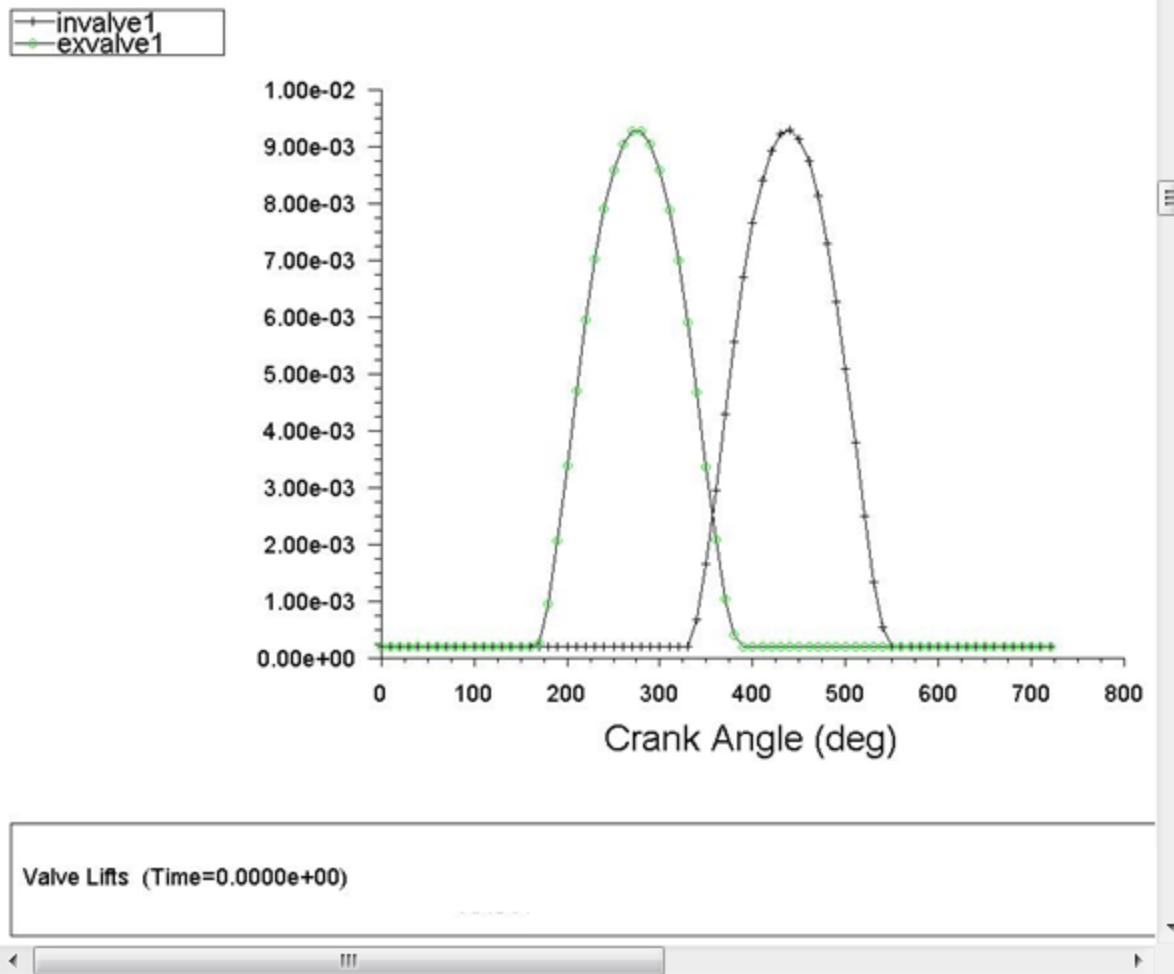
Type	Zones	Values
wall (invalve1)	invalve1-stem, invalve1-ob, invalve1-ch, invalve1-ib	Temperature (k) 300
wall (exvalve1)	exvalve1-stem, exvalve1-ob, exvalve1-ch, exvalve1-ib	Temperature (k) 300
wall (invalve-port)	invalve-1-port	Temperature (k) 300
wall (exvalve-port)	exvalve-1-port	Temperature (k) 300
pressure-inlet	ice-inlet-invalve-1-port	Gauge Total Pressure (pascal) -21325
		Supersonic/Initial Gauge Pressure (pascal) 0
		Total Temperature (k) 313
pressure-outlet	ice-outlet-exvalve-1-port	Gauge Pressure (pascal) -1325
		Backflow Total Temperature (k) 333
wall	invalve1-seat	Temperature (k) 300
wall	exvalve1-seat	Temperature (k) 300
wall	cyl-head	Temperature (k) 348
wall	cyl-piston	Temperature (k) 318
wall	cyl-quad	Temperature (k) 318
wall	cyl-tri	Temperature (k) 318
wall	piston	Temperature (k) 318

- Check the piston and valve lift profiles in the figures displayed below.

3.2. Piston and Valves Lift profiles



3.3. Valves Lift profiles



- The table **Relaxations at crank angles**, shows the under-relaxation factors at different crank angles.

3.4. Relaxations

Table 5. Relaxation changes through events

Crank Angle	Pressure	Density	Body Forces	Momentum	Turbulent Kinetic Energy	Turbulent Dissipation Rate	Turbulent Viscosity	Energy
0.000	0.300	1.000	1.000	0.500	0.400	0.400	1.000	1.000
166.400	0.500	"	"	0.500	0.800	0.800	"	1.000
168.400	0.300	"	"	0.500	0.400	0.400	"	1.000
329.600	0.500	"	"	0.500	0.800	0.800	"	1.000
331.600	0.300	"	"	0.500	0.400	0.400	"	1.000

- The table **Dynamic Mesh Events**, shows the events at different crank angles.

3.5. Dynamic Mesh Setup

Table 6. Dynamic Mesh Events

At Crank Angle (deg)	Name	Description
0.000	dt-event-at-0(0.25)	Changing the Time step size in terms of crank angle.
160.200	write-solution-point-at-ca-160.200	Saves solution files at this point.
166.400	reduce-urf-due-to-open-exvalve1, change-positivity-at-valve-open, open-exvalve1, start-smoothing-at-exvalve1-open, dt-event-at-166.4(0.125)	epsilon=0.8, k=0.8, mom=0.5, pressure=0.5, temperature=1. Reducing URFs 1deg before valve opening for solution stability. Changing the Time step size in terms of crank angle.
168.400	increase-urf-due-to-open-exvalve1, change-positivity-after-valve-open	epsilon=0.4, k=0.4, mom=0.5, pressure=0.3, temperature=1. Increasing URFs for accelerating the solution.
169.400	change-positivity-after-valve-open	
171.400	dt-event-at-171.4(0.25)	Changing the Time step size in terms of crank angle.
180.000	save-residual-plot-180	Saves the residual plot image from last saved iteration to the current iteration.
185.300	stop-smoothing-after-exvalve1-open	Stops smoothing in vlayer and starts layering.
323.300	write-solution-point-at-ca-323.300	Saves solution files at this point.
329.600	reduce-urf-due-to-open-invalve1, change-positivity-at-valve-open, open-invalve1, start-smoothing-at-invalve1-open, dt-event-at-329.6(0.125)	epsilon=0.8, k=0.8, mom=0.5, pressure=0.5, temperature=1. Reducing URFs 1deg before valve opening for solution stability. Changing the Time step size in terms of crank angle.
331.600	increase-urf-due-to-open-invalve1, change-positivity-after-valve-open	epsilon=0.4, k=0.4, mom=0.5, pressure=0.3, temperature=1. Increasing URFs for accelerating the solution.
332.600	change-positivity-after-valve-open	

334.600	dt-event-at-334.6(0.25)	Changing the Time step size in terms of crank angle.
348.400	stop-smoothing-after-invalve1-open	Stops smoothing in vlayer and starts layering.
360.000	save-residual-plot-360	Saves the residual plot image from last saved iteration to the current iteration.
365.400	start-smoothing-before-exvalve1-close	Stops layering in vlayer and starts smoothing.
385.400	change-positivity-at-valve-close, dt-event-at-385.4(0.125)	Changing the Time step size in terms of crank angle.
390.400	close-exvalve1, dt-event-at-390.4(0.25)	Deleting interface between vlayer and chamber for stopping flow. Changing the Time step size in terms of crank angle.
391.400	change-positivity-at-valve-close	
392.400	change-positivity-at-valve-close	
402.300	write-solution-point-at-ca-402.300	Saves solution files at this point.
528.700	start-smoothing-before-invalve1-close	Stops layering in vlayer and starts smoothing.
540.000	save-residual-plot-540	Saves the residual plot image from last saved iteration to the current iteration.
547.200	change-positivity-at-valve-close, dt-event-at-547.200000000001 (0.125)	Changing the Time step size in terms of crank angle.
552.200	close-invalve1, dt-event-at-552.200000000001(0.25)	Deleting interface between vlayer and chamber for stopping flow. Changing the Time step size in terms of crank angle.
553.200	change-positivity-at-valve-close	
554.200	change-positivity-at-valve-close	
563.000	write-solution-point-at-ca-563.000	Saves solution files at this point.
720.000	save-residual-plot-720, dt-event-at-720(0.25), write-solution-point-at-ca-720.000	Saves the residual plot image from last saved iteration to the current iteration. Changing the Time step size in terms of crank angle. Saves solution files at this point.

- In the section **IC Engine System Inputs** check engine inputs you have entered in the Properties dialog box. It also lists the **Journal Customization** files if they are provided.

3.6. IC Engine System Inputs

Engine Inputs

Engine Speed (rev/min) : 2000
Crank Radius (mm) : 45
Piston Pin Offset/Wrench (mm) : 0
Connecting Rod Length (mm) : 144.3
Minimum Lift (mm) : 0.5

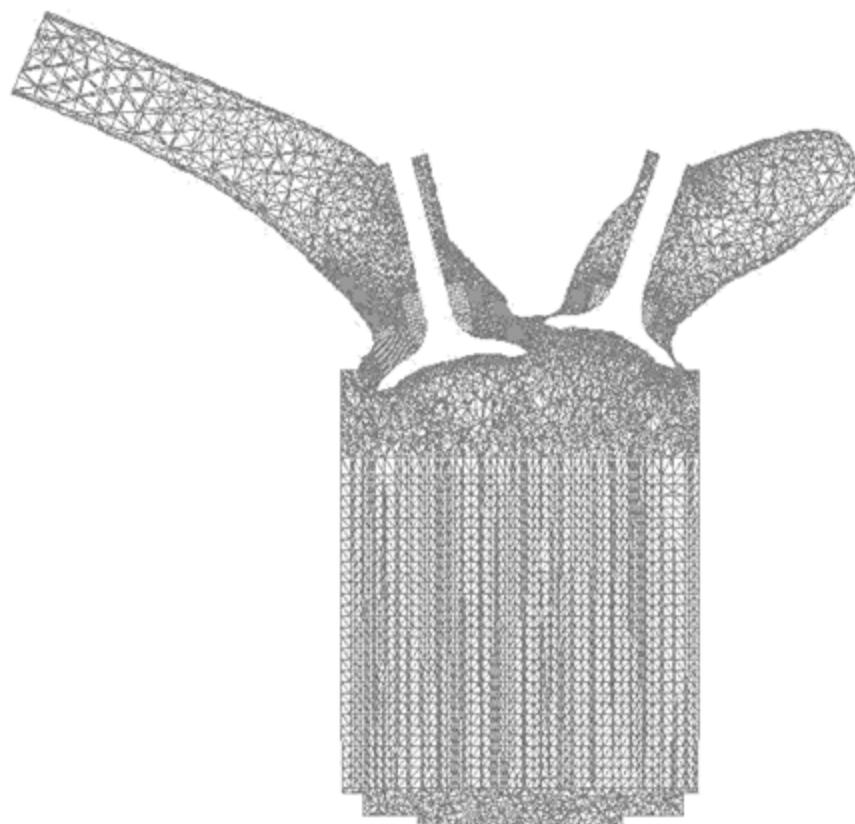
Journal Customization

Pre Iteration Journal File : N/A
Post Iteration Journal File : N/A

- Check the mesh motion in the section **Solution Data**.

4. Solution Data

4.1. Animation: mesh-on-ice_cutplane_1

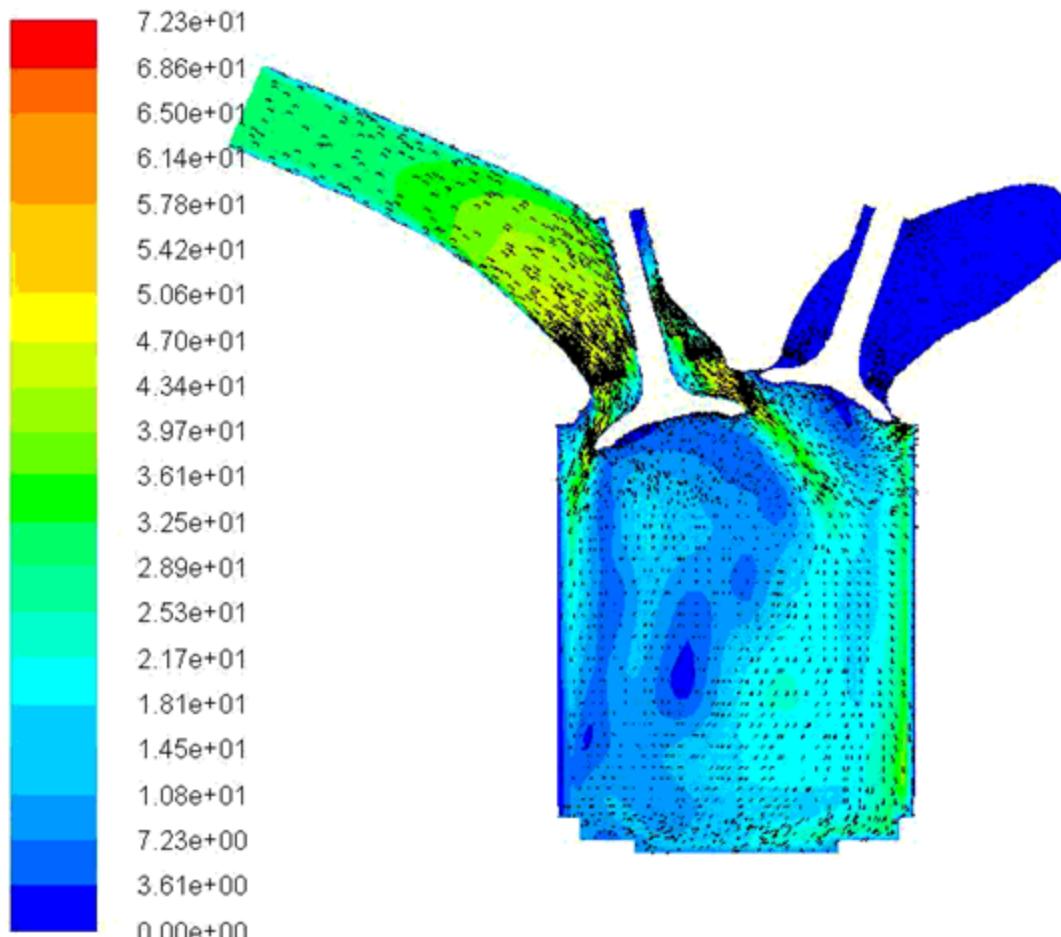


Surface Grid (Time=1.3599e-02)
Crank Angle=486.19(deg)

ANSYS Fluent Release 16.0 (3d, dp, pb)

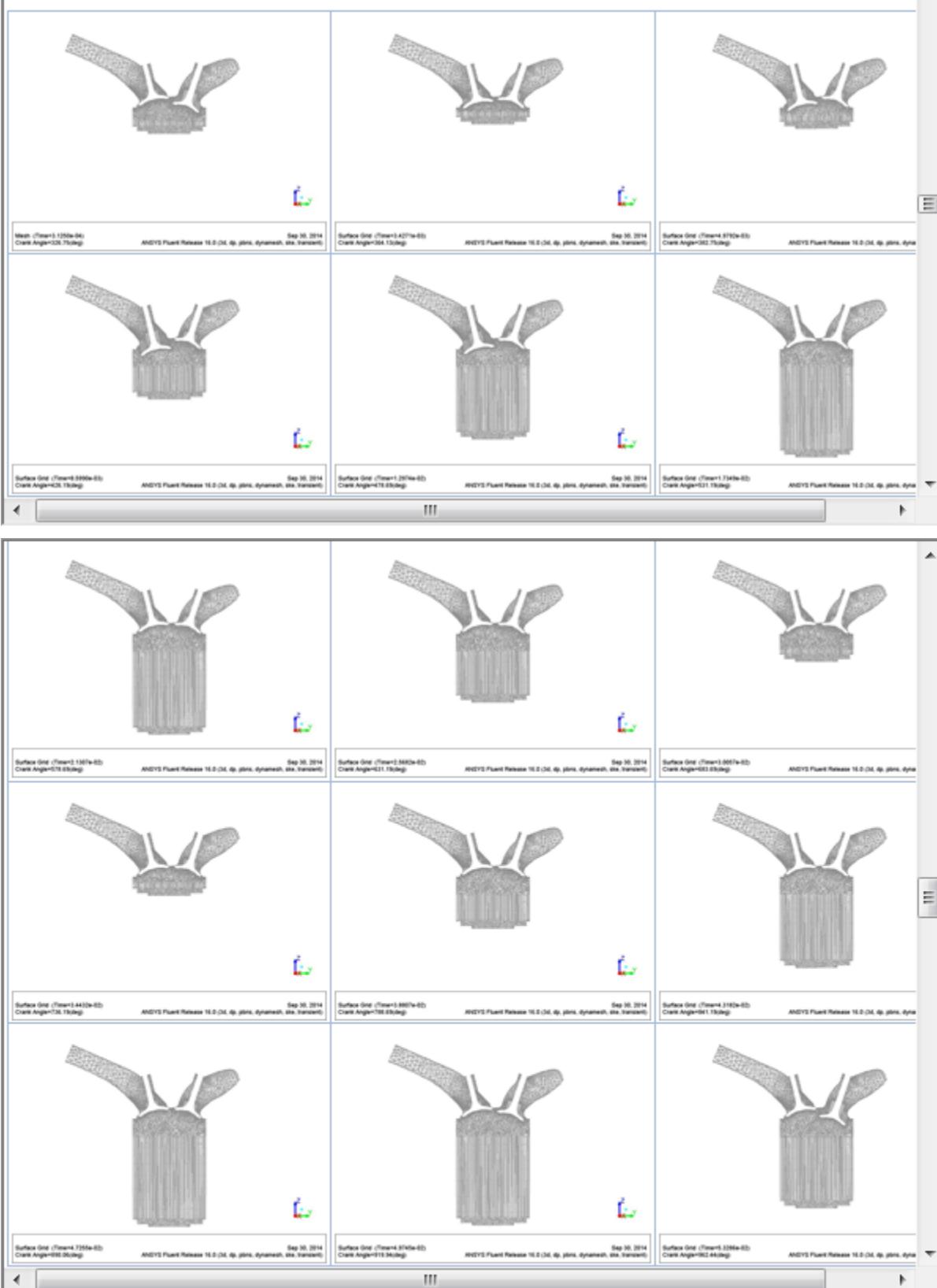
- Check the velocity contour animation under section **Solution Data**.

4.2. Animation: velocity-magnitude on ice_cutplane_1

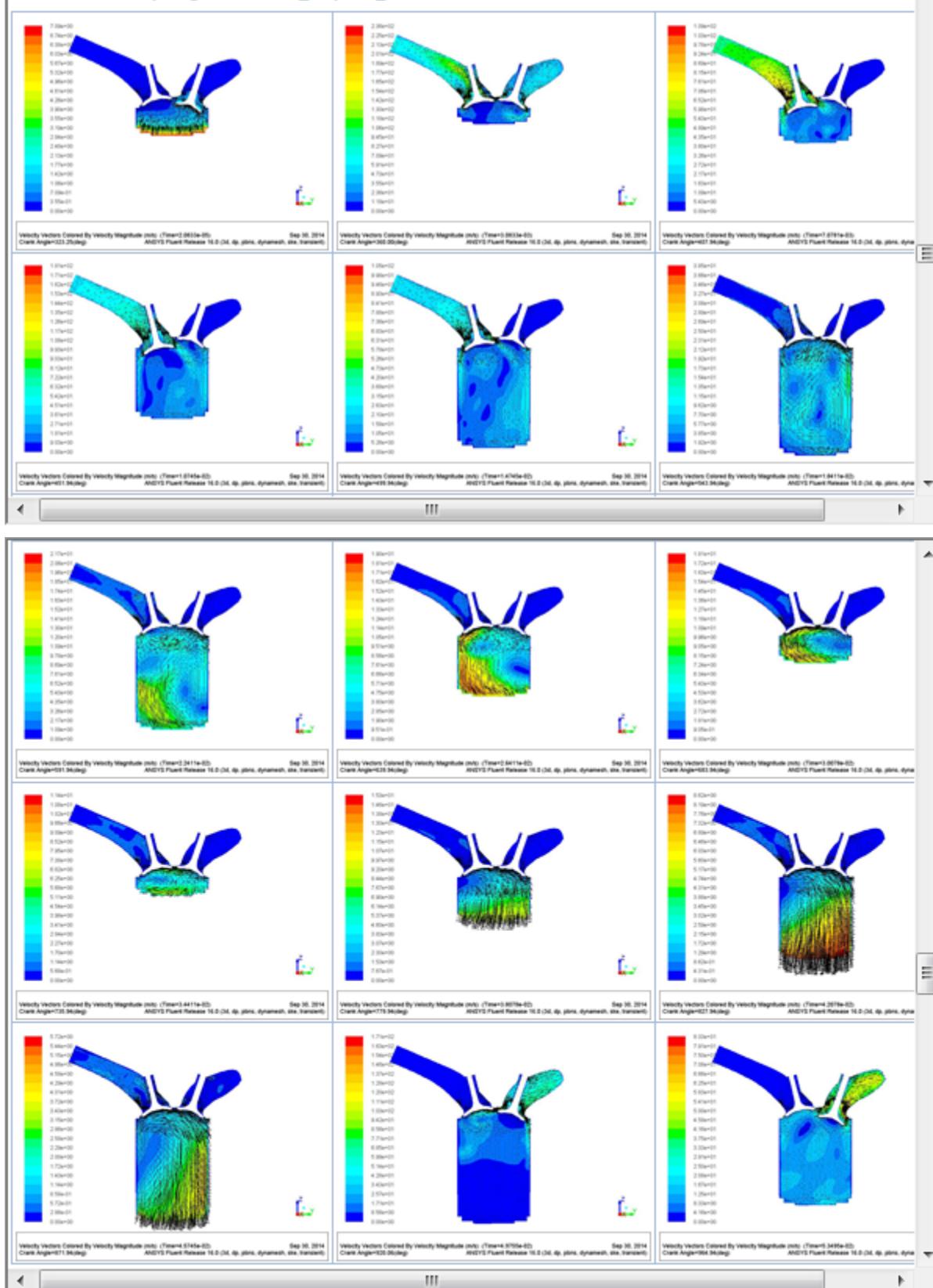


Velocity Vectors Colored By Velocity Magnitude (m/s) (Time=1.2745e-02)
Crank Angle=475.94(deg) ANSYS Fluent Release 16.0 (3d, dp, pb)

- Observe the saved images of the mesh at the cut-plane in the table **mesh-on-ice_cutplane_1**.

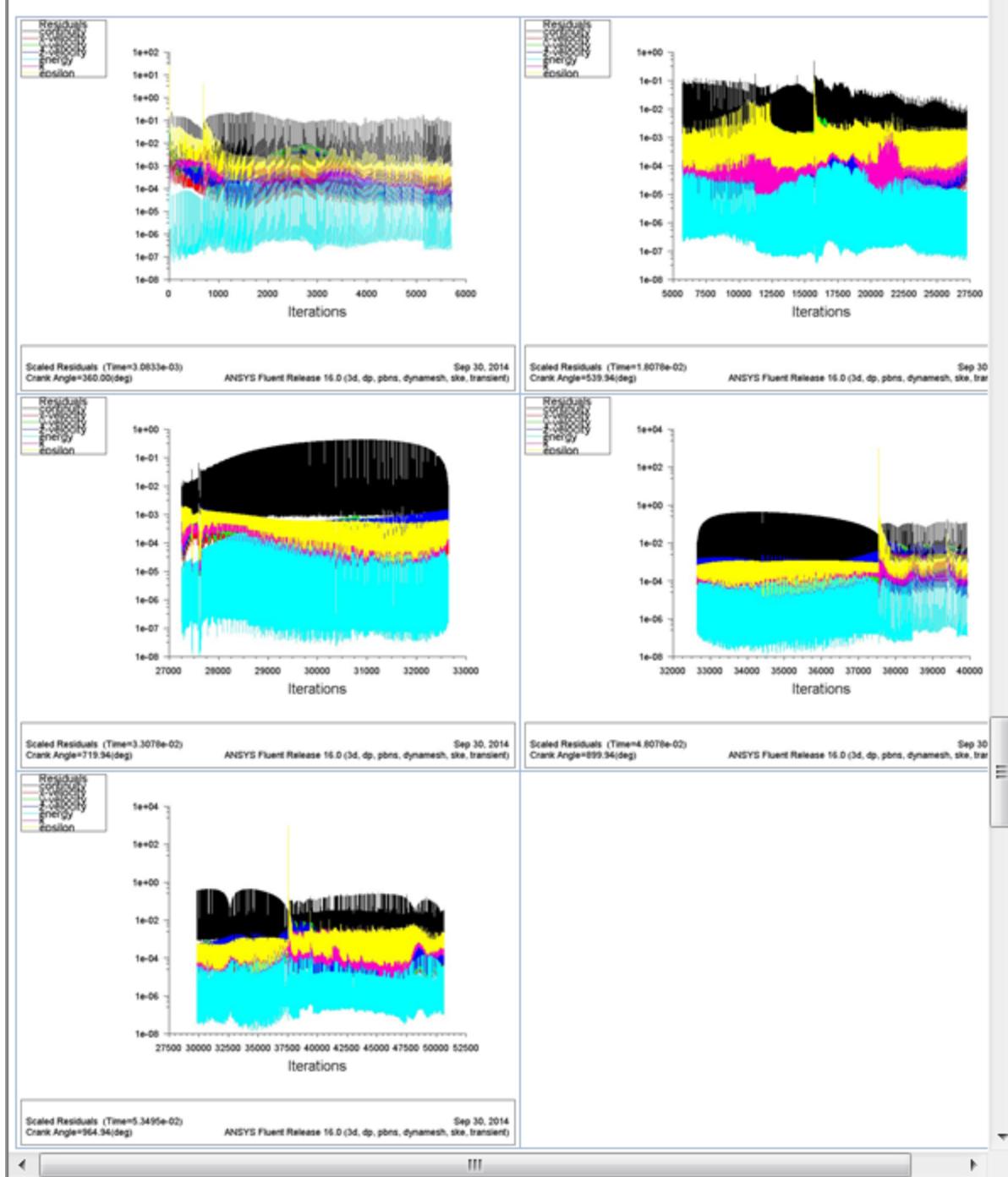
4.3. Table: mesh-on-ice_cutplane_1

- Similarly observe the saved images of the velocity contours on the cut-plane in the table **velocity-magnitude on "ice_cutplane_1"**.

4.4. Table: velocity-magnitude on ice_cutplane_1

- Check the residuals in the **Residuals** table. The residual chart is saved after every 180 degrees.

4.5. Table: Residuals

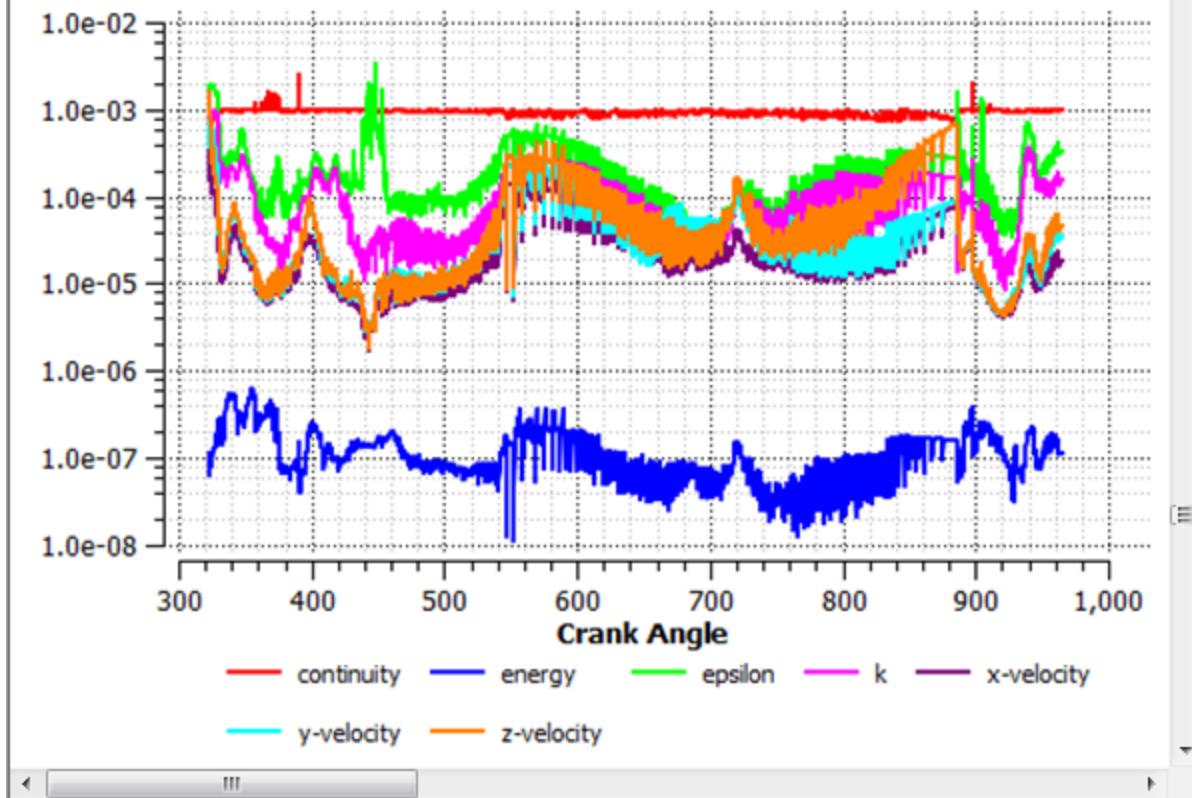


- Check the **Last iteration residual values corresponding to each time step chart.**

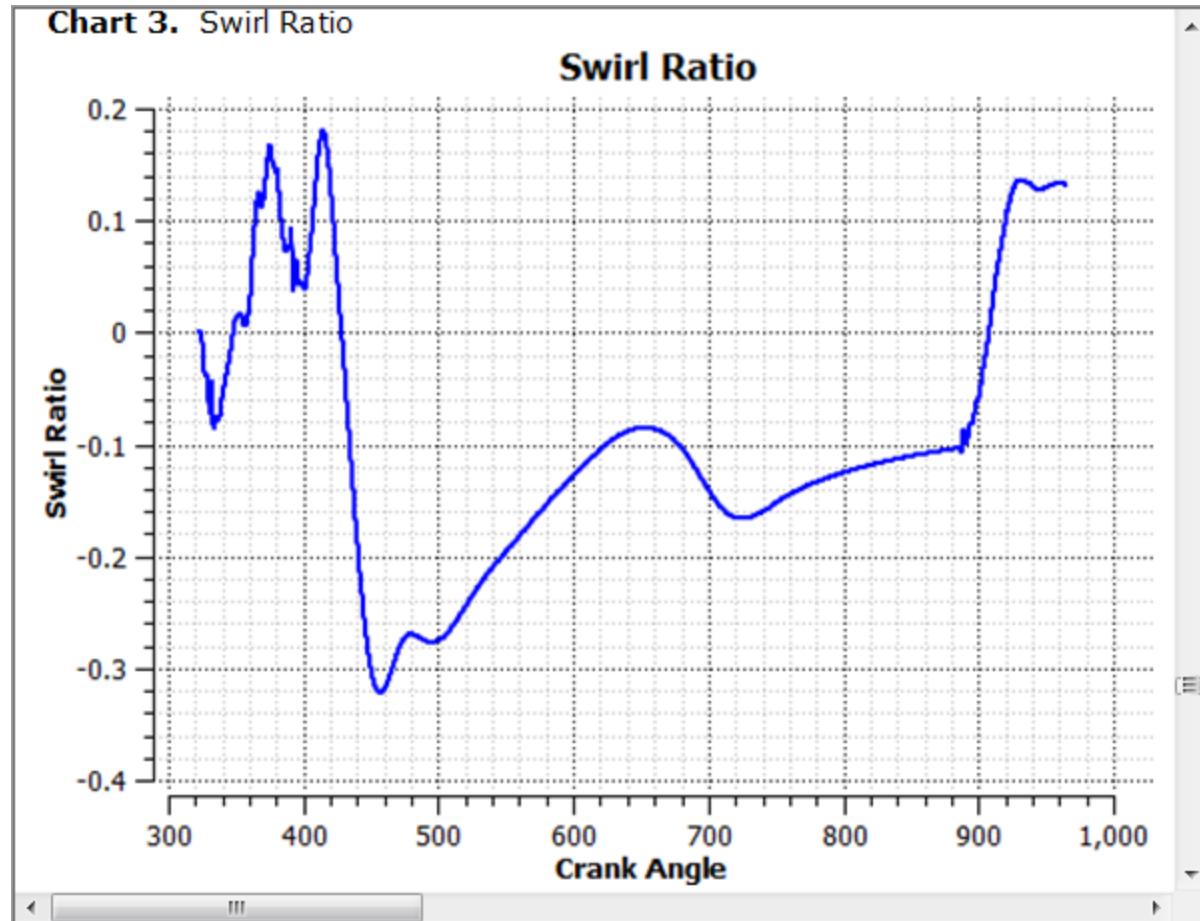
4.6. Charts

Chart 2. Last iteration residual values corresponding to each time step

Last iteration residual values corresponding to each time step



- Observe the chart of swirl ratio at different crank angles in chart **Swirl Ratio**.

Chart 3. Swirl Ratio

- Observe the chart of tumble at different crank angles in charts **Tumble Ratio** and **Cross Tumble Ratio**

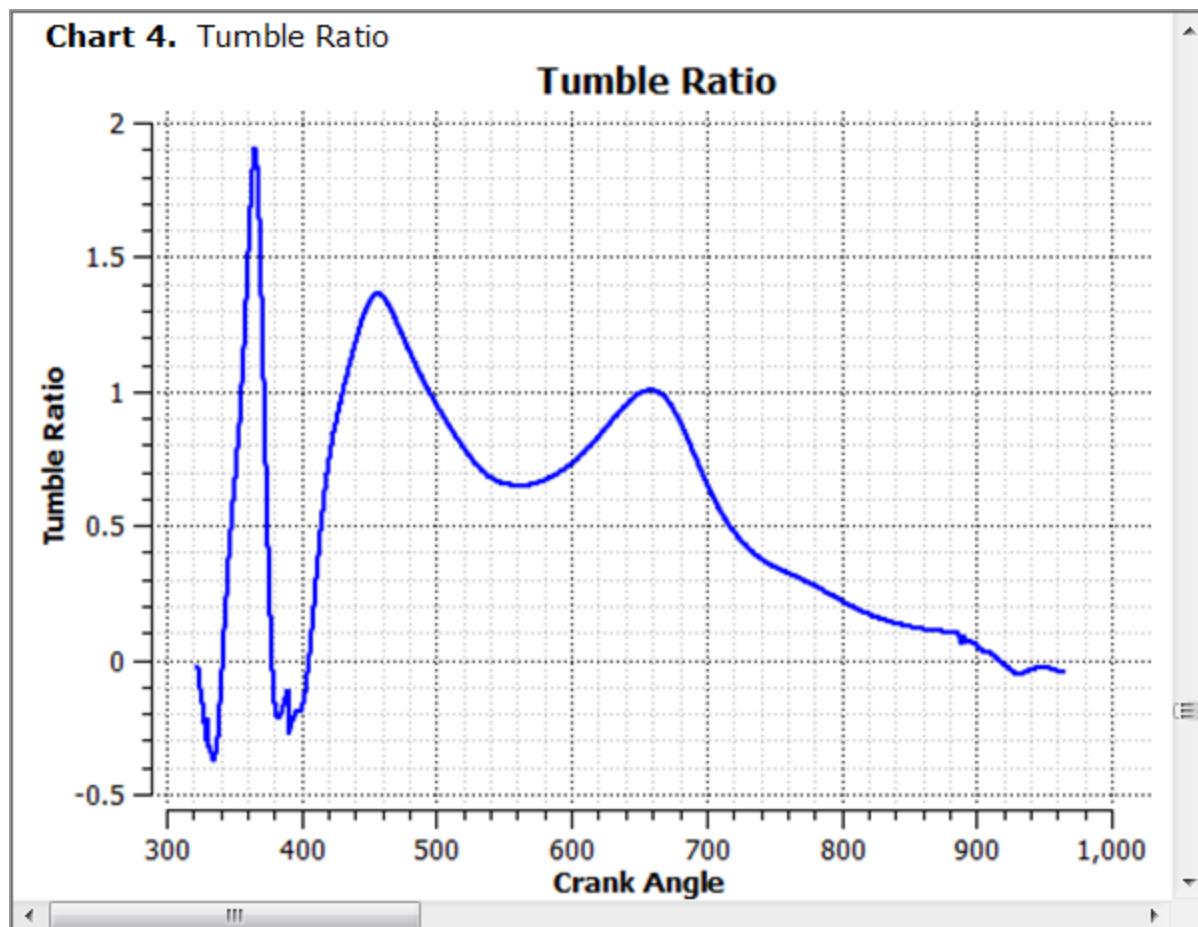
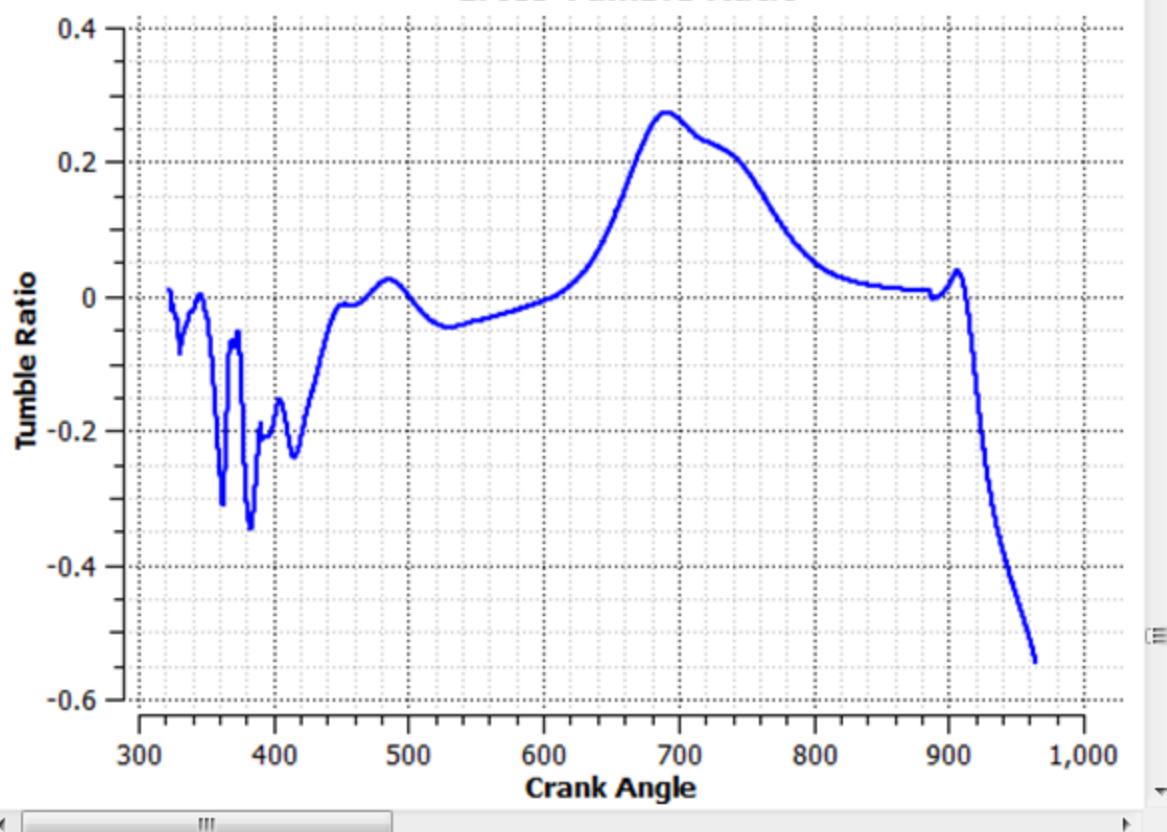
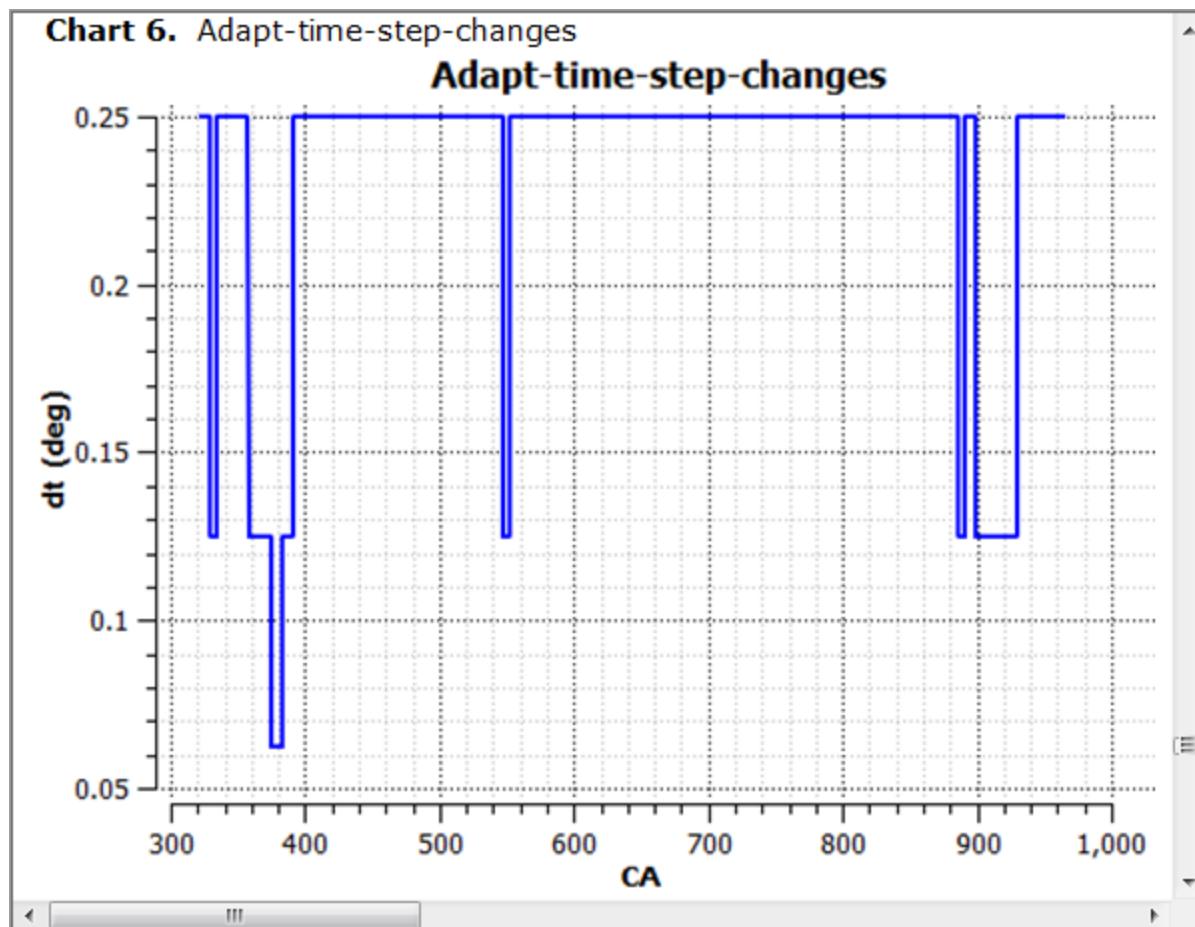
Chart 4. Tumble Ratio

Chart 5. Cross Tumble Ratio

Cross Tumble Ratio



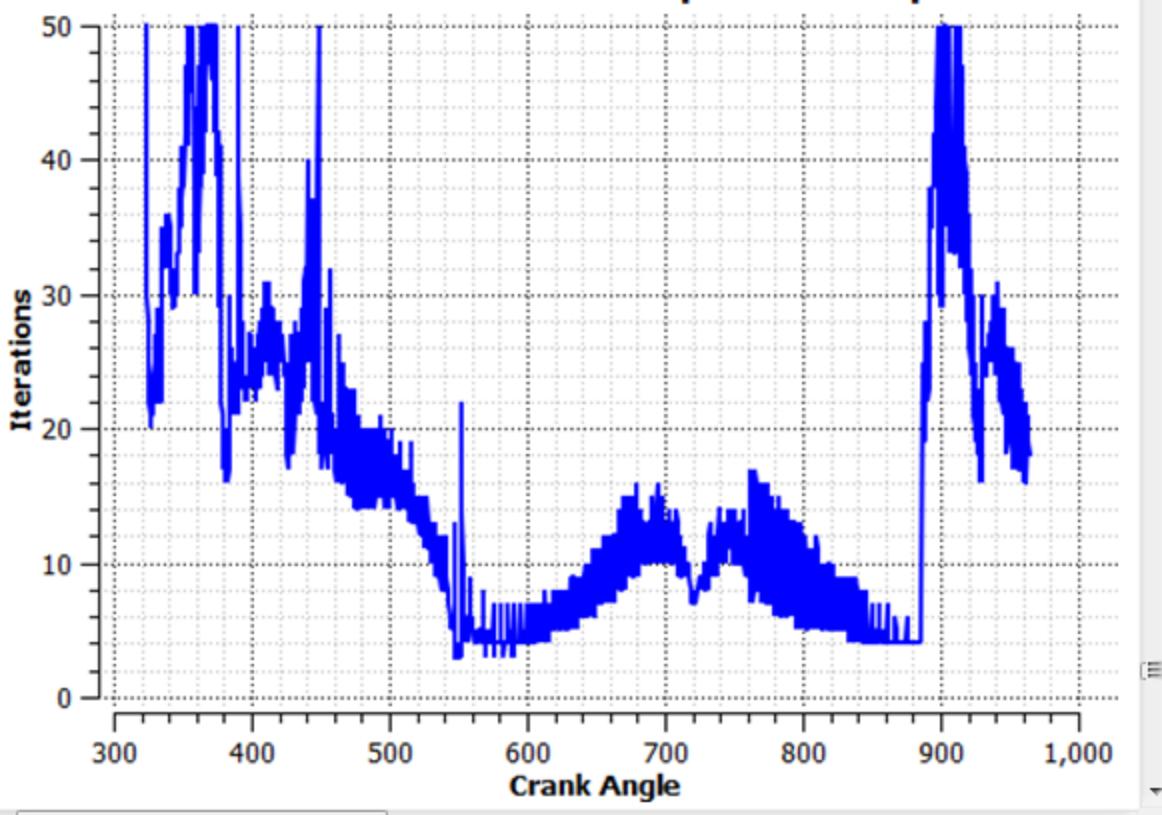
- Check the changes in time steps.

Chart 6. Adapt-time-step-changes

- Check the **Number of Iterations per Time Step**.

Chart 7. Number of Iterations per Time Step

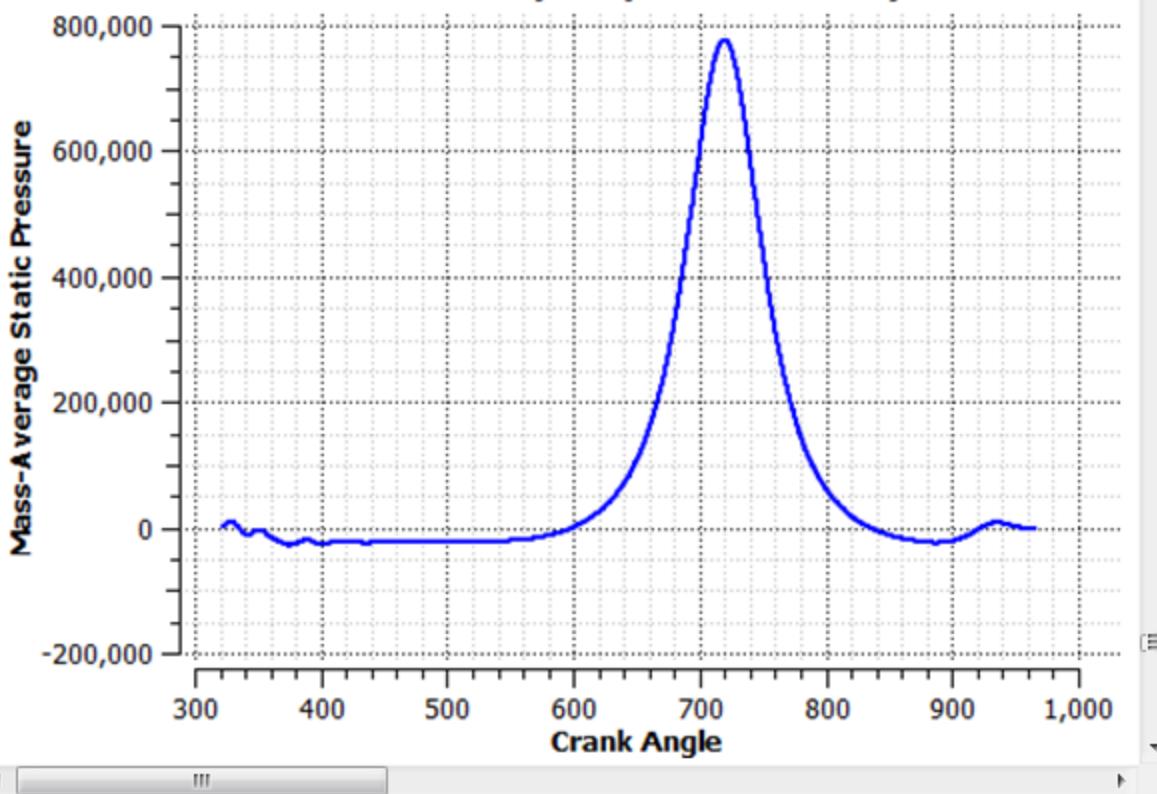
Number of Iterations per Time Step



- Observe the convergence history of static pressure in the figure below.

Chart 8. Monitor: Mass-Average Static Pressure (fluid-piston fluid-layer-cylinder fluid-ch)

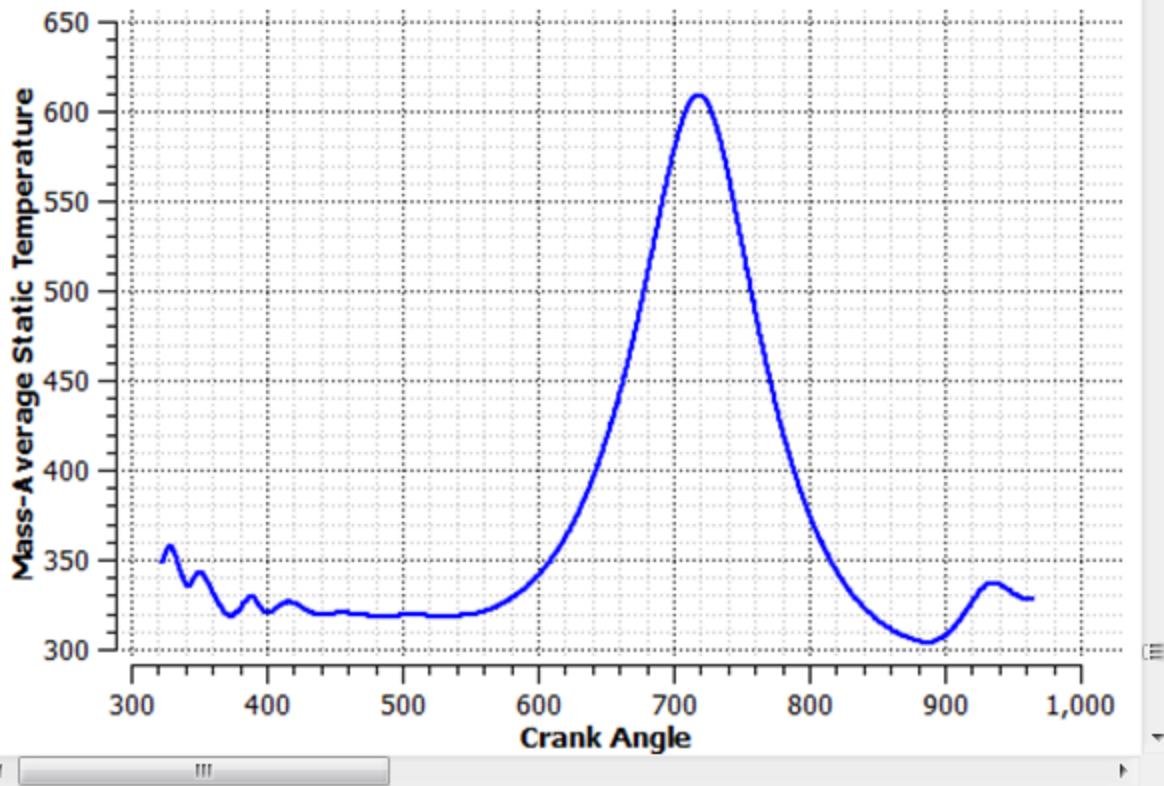
Monitor: Mass-Average Static Pressure (fluid-piston fluid-layer-cylinder fluid-ch)



- Observe the convergence history of static temperature in the figure below.

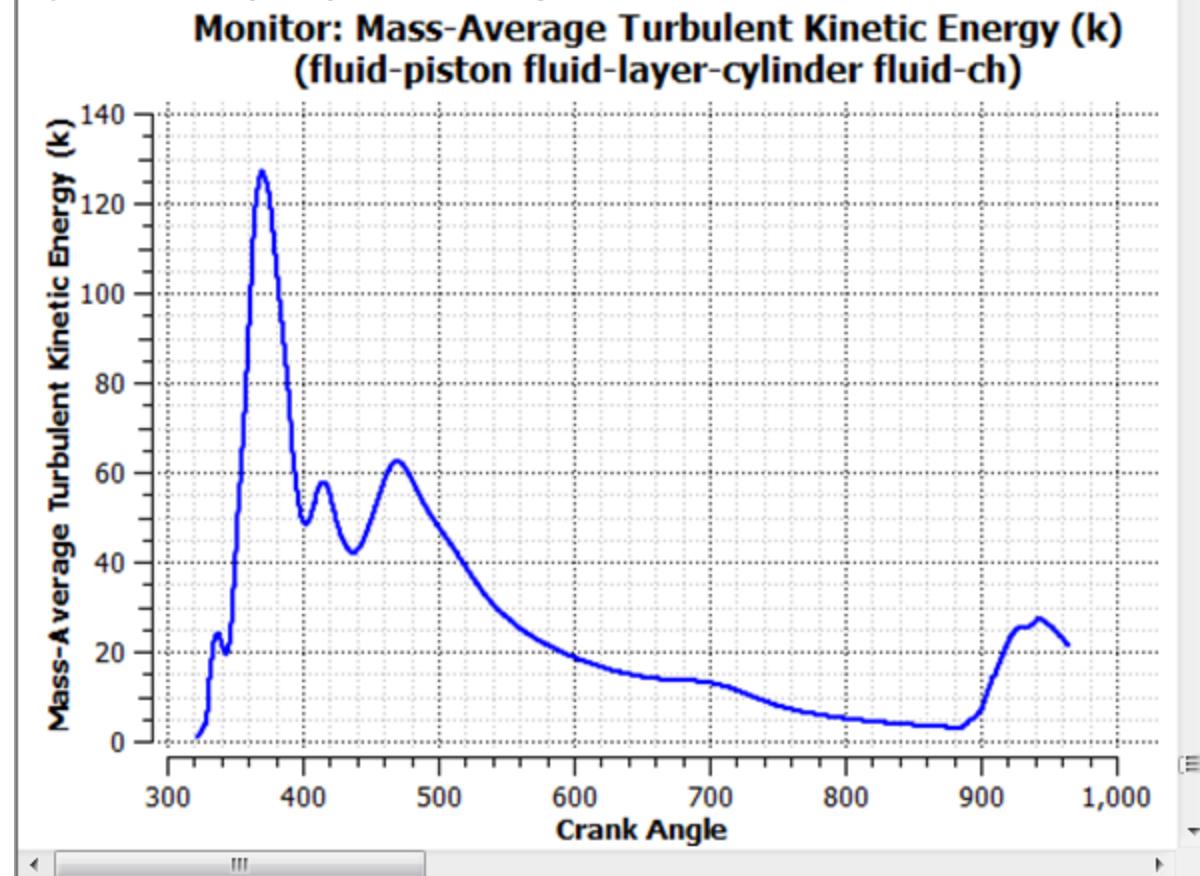
Chart 9. Monitor: Mass-Average Static Temperature (fluid-piston fluid-layer-cylinder fluid-ch)

Monitor: Mass-Average Static Temperature (fluid-piston fluid-layer-cylinder fluid-ch)

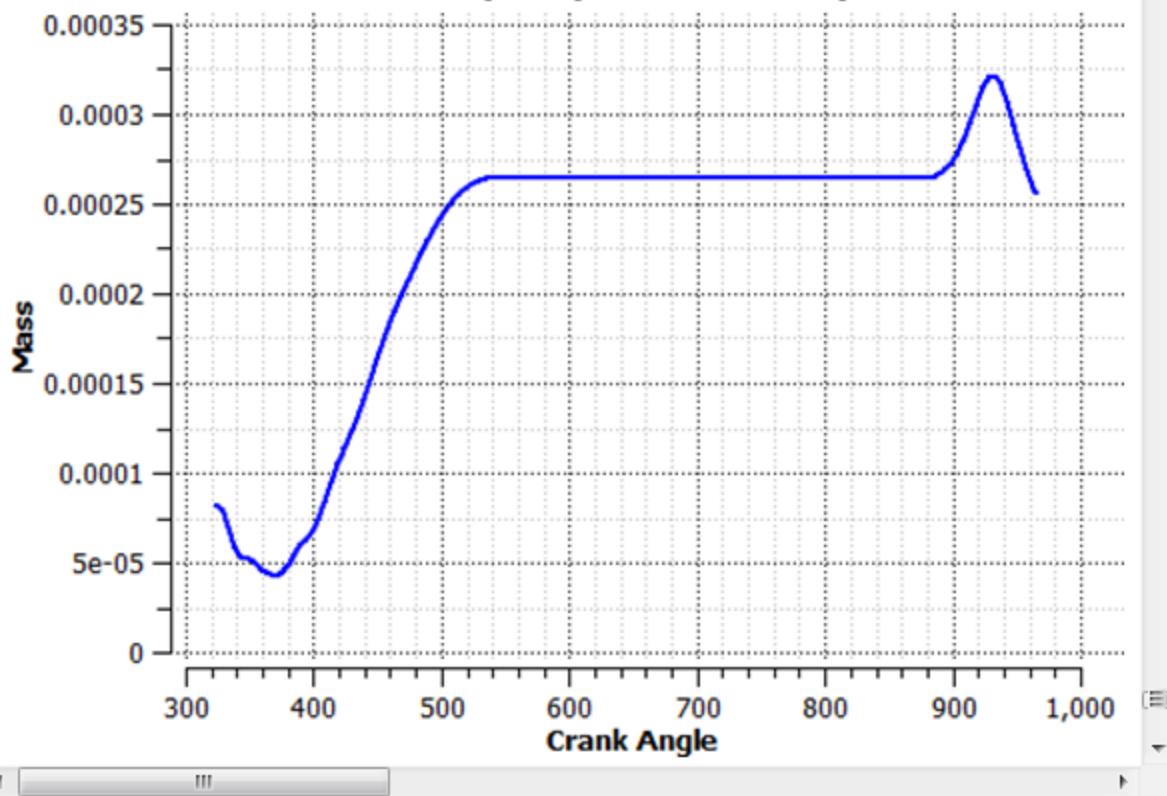


- Observe the convergence history of turbulent kinetic energy in the figure below.

Chart 10. Monitor: Mass-Average Turbulent Kinetic Energy (k) (fluid-piston fluid-layer-cylinder fluid-ch)



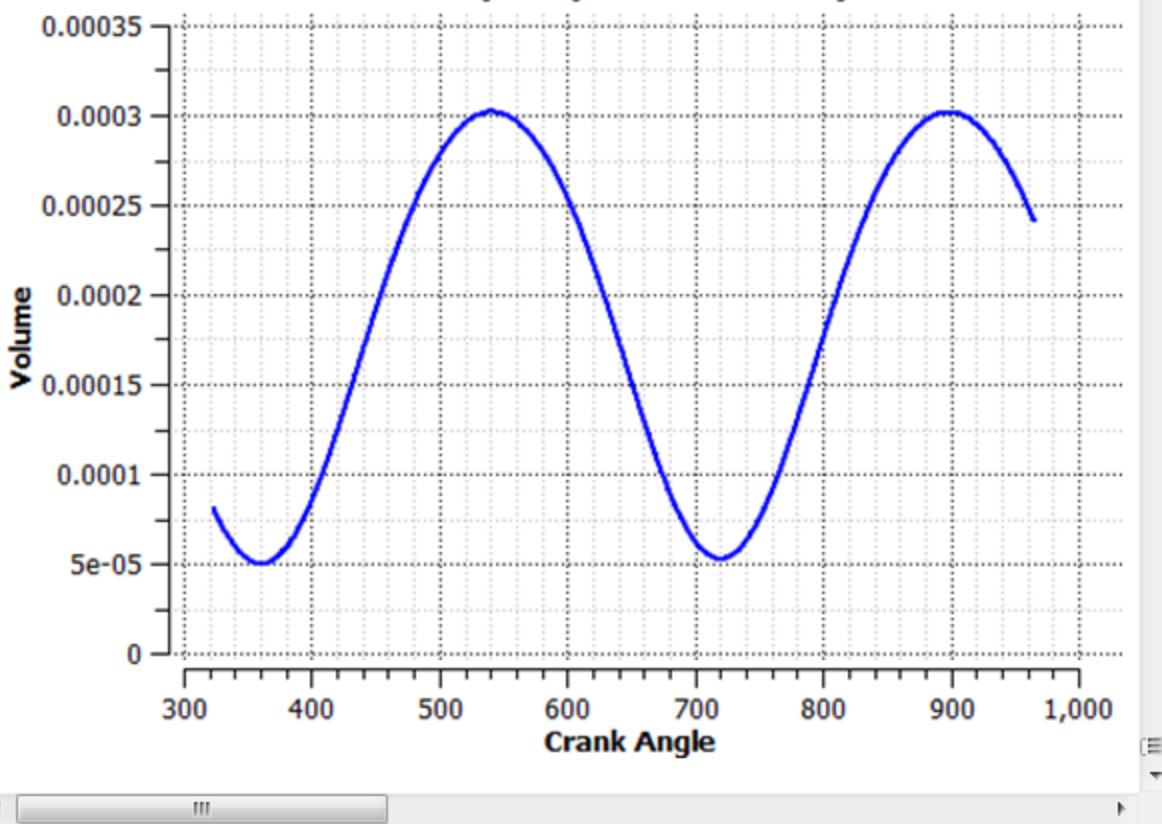
- Observe the convergence history of mass static pressure.

Chart 11. Monitor: Mass Static Pressure (fluid-piston fluid-layer-cylinder fluid-ch)**Monitor: Mass Static Pressure (fluid-piston fluid-layer-cylinder fluid-ch)**

- Observe the convergence history of volume static pressure.

Chart 12. Monitor: Volume Static Pressure (fluid-piston fluid-layer-cylinder fluid-ch)

Monitor: Volume Static Pressure (fluid-piston fluid-layer-cylinder fluid-ch)

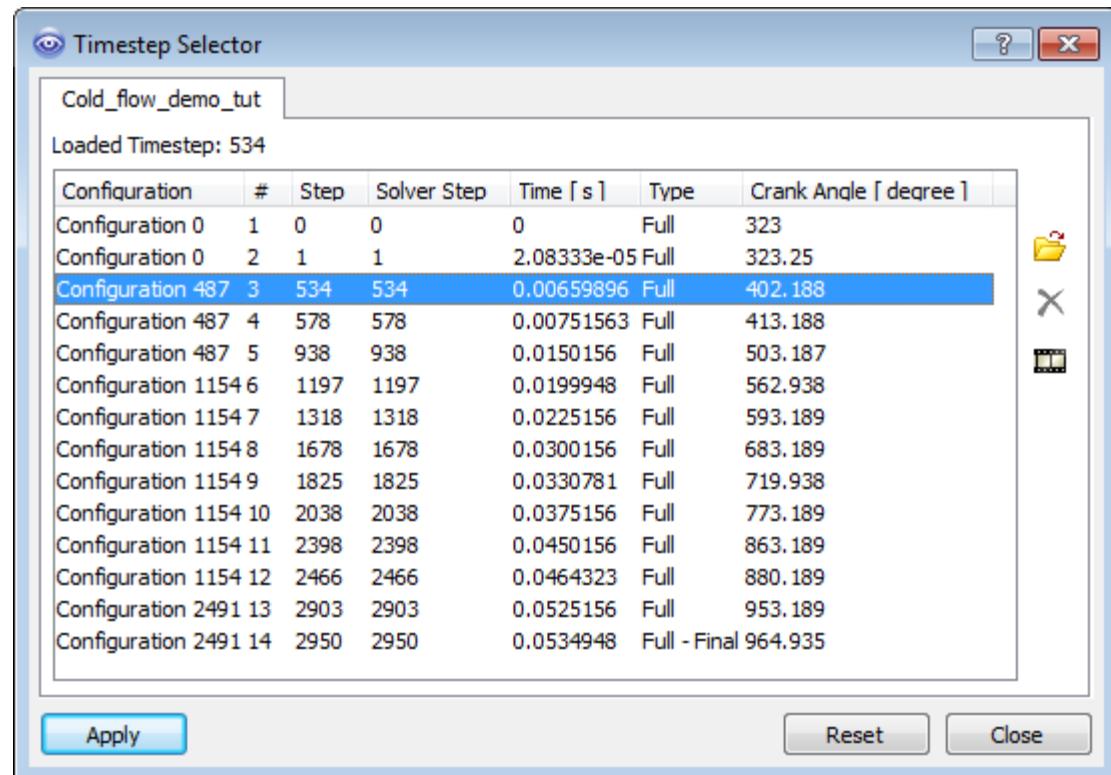


1.8. Step 7: Postprocessing

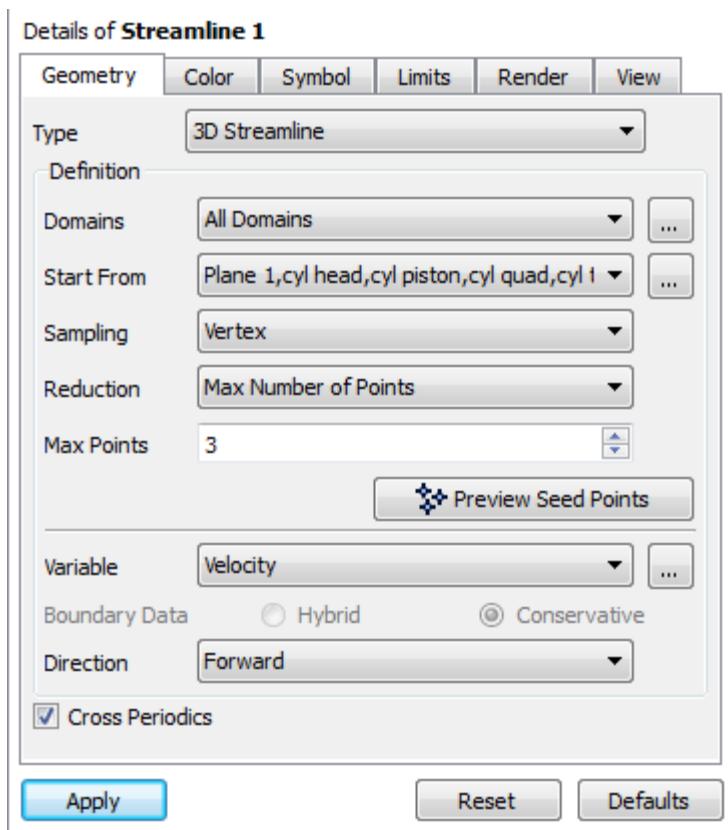
In this step you will display the velocity vectors in CFD-Post.

1. Double-click the **Results** cell to open CFD-Post.
2. You can choose the time step at which you want to display the velocity vectors. Open the **Timestep Selector** dialog box by selecting **Timestep Selector** (⌚) from the **Tools** menu.

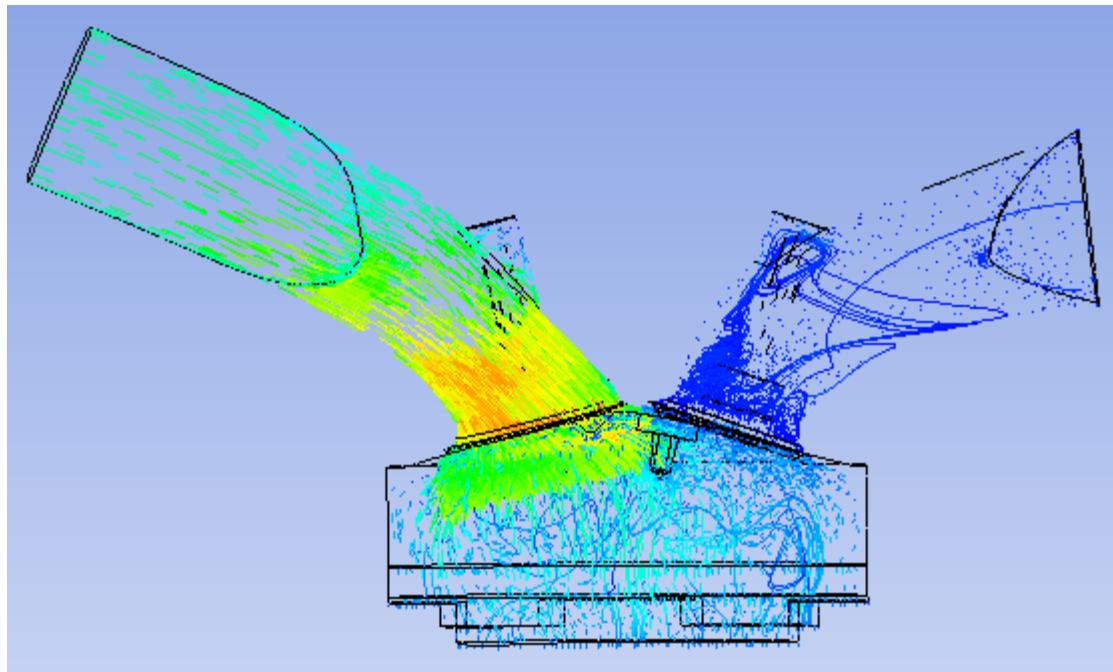
Tools > Timestep Selector



1. Select the **Step** of your choice from the list and click **Apply**.
2. Close the **Timestep Selector** dialog box.
3. Select **Streamline** (from the **Insert** menu.
Insert > Streamline
 - a. Retain the default name and click **OK** in the **Insert Streamline** dialog box.



- b. Click **Location Editor** (next to **Locations**, in the **Geometry** tab.
- c. Select all items under **ICE** in the **Location Selector** dialog box and click **OK**.
- d. Enter 3 for **Max Points**.
- e. Enter 1 for **Line Width** under the **Symbol** tab.
- f. Click **Apply**.



4. Include the image in the report.

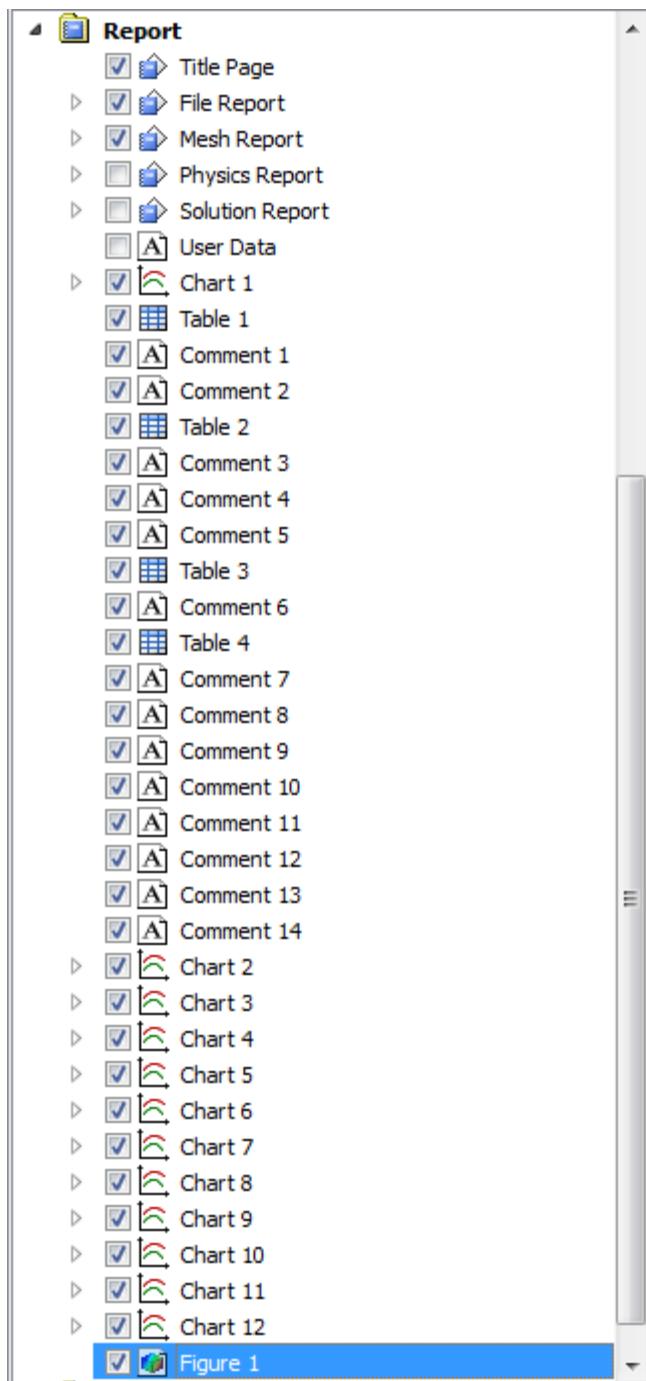
- a. Select **Figure** () from the **Insert** menu

Insert > Figure

- b. Retain the default name and click **OK**.

Note

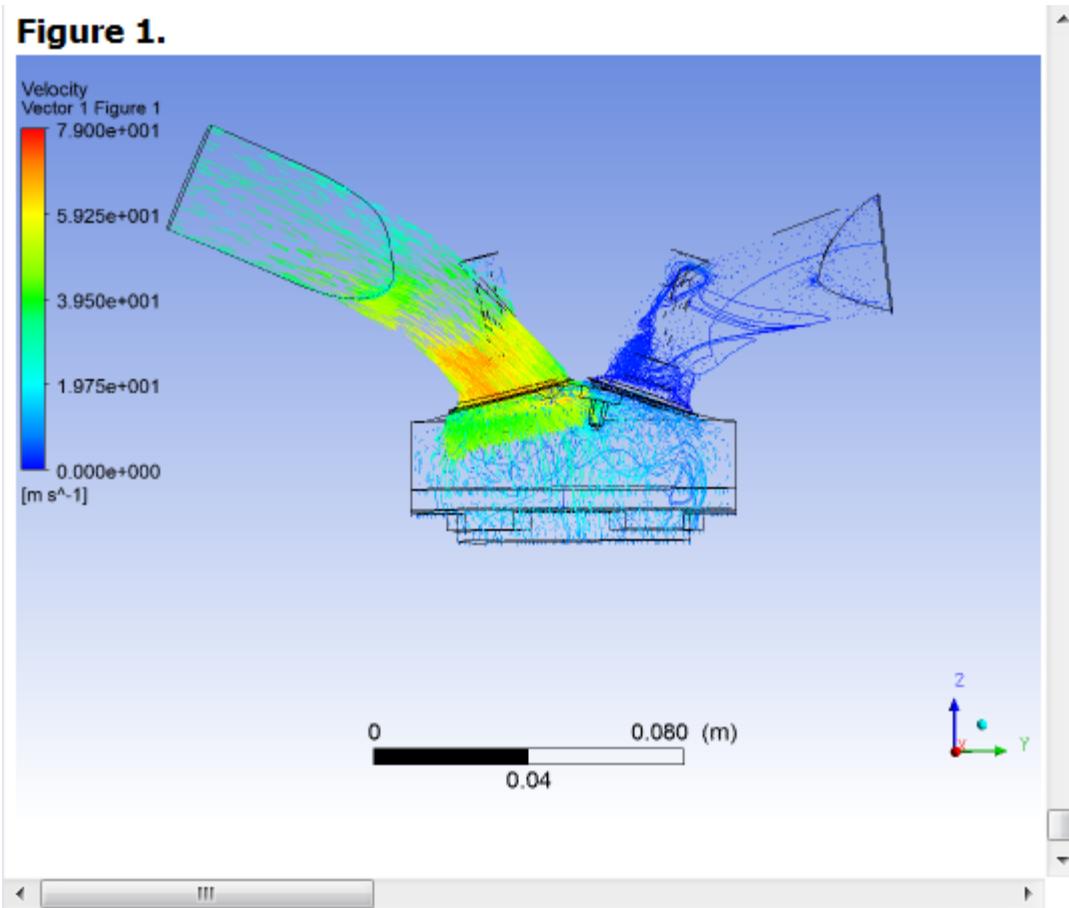
This will copy the image displayed in the **3D Viewer**, to the list under **Report** in the **Outline** tree.



- c. Click the **Report Viewer** tab in the display window and click **Refresh** ().

Note

The vectors figure is displayed in the report.

Figure 1.

5. To save the appended report, click **Publish** ().

This concludes the tutorial which demonstrated the setup and solution for a cold flow simulation of an IC engine.

1.9. Summary

In this tutorial you have learnt how streamlined workflow is achieved in WB-ICE. Motored engine operation was performed using K-epsilon with standard wall treatment turbulence model in Fluent.

1.10. Further Improvements

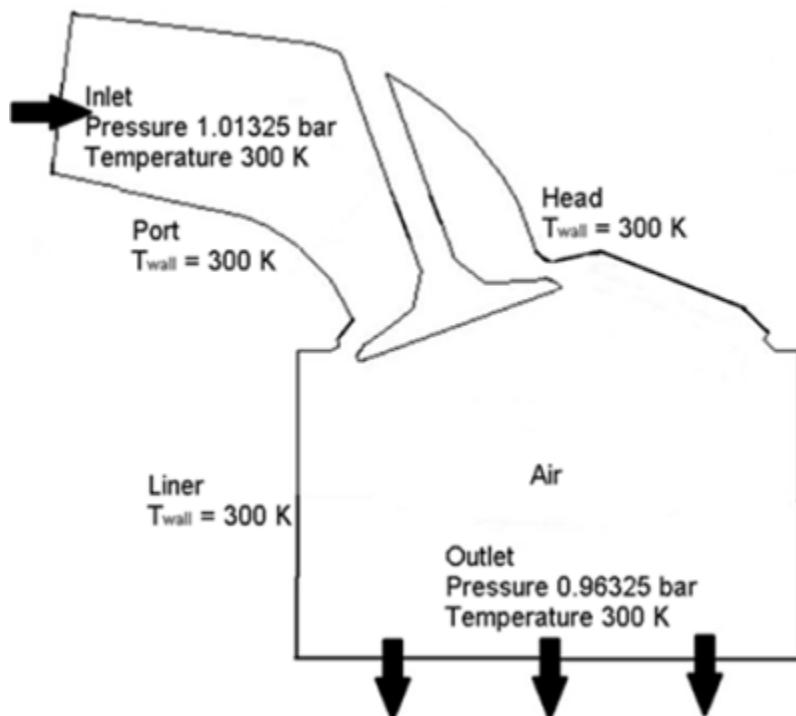
You may use mesh refinement while using keygrids for accurate results. You can refer to Motored validation results on EKM.

Chapter 2: Tutorial: Solving a Port Flow Simulation

In this tutorial of port flow analysis, you will measure mass and angular momentum flux (swirl and tumble) for given cylinder head and intake port design over varying valve lifts of 2mm, 6mm and 10mm. You will create swirl monitor planes at 30 mm, 45 mm, and 60 mm below the cylinder head. The inlet, outlet and wall boundary conditions are as shown in the [Figure 2.1: Problem Schematic \(p. 63\)](#). Initial conditions are pressure 101325 Pa and temperature 300 K. The tutorial illustrates the following steps in setting up and solving a port flow simulation of an IC engine.

- Launch IC Engine system.
- Read an existing geometry into IC Engine.
- Decompose the geometry.
- Define mesh setup and mesh the geometry.
- Add design points to observe the change in results with change in input parameters.
- Run the simulation.
- Examine the results in the report.

Figure 2.1: Problem Schematic



This tutorial is written with the assumption that you are familiar with the IC Engine system and that you have a good working knowledge of ANSYS Workbench.

- 2.1. Preparation
- 2.2. Step 1: Setting the Properties
- 2.3. Step 2: Performing the Decomposition
- 2.4. Step 3: Meshing
- 2.5. Step 4: Setting up the Simulation
- 2.6. Step 5: Running the Solution
- 2.7. Step 6: Obtaining the Results
- 2.8. Summary
- 2.9. Further Improvements

2.1. Preparation

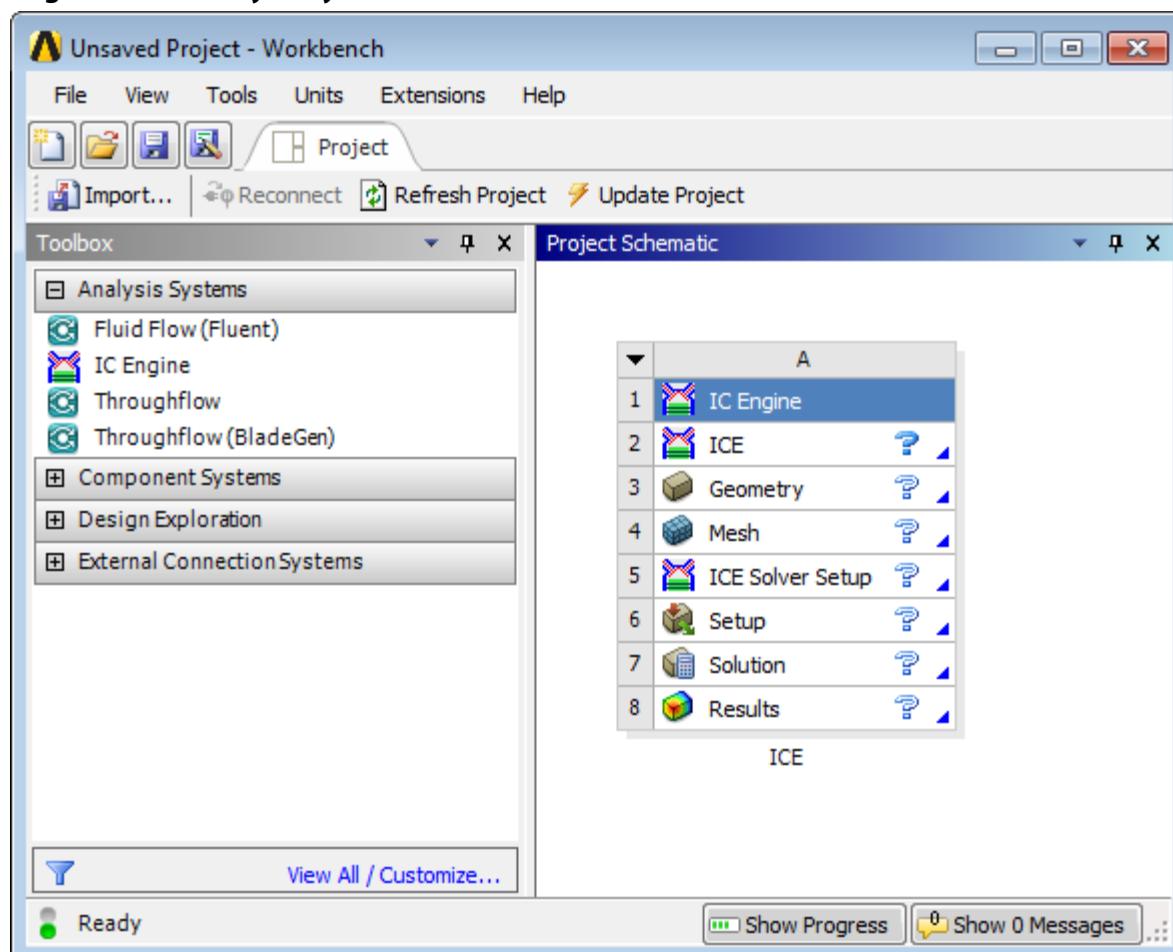
1. Copy the file (`tut_port.x_t`) to your working folder.

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

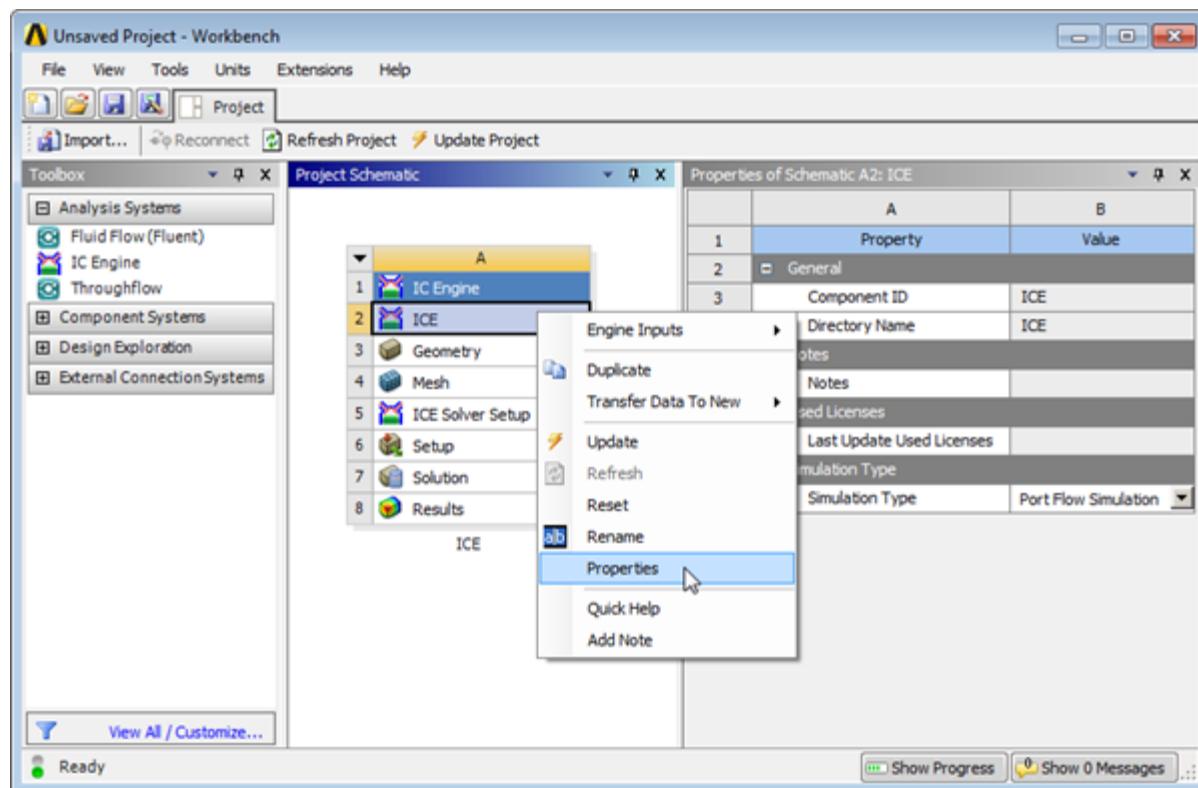
2. Start Workbench.

2.2. Step 1: Setting the Properties

1. Create an IC Engine analysis system in the Workbench interface by dragging or double-clicking on **IC Engine** under **Analysis Systems** in the **Toolbox**.



2. Right-click on **ICE**, cell 2, and click **Properties** (if it is not already visible) from the context menu.



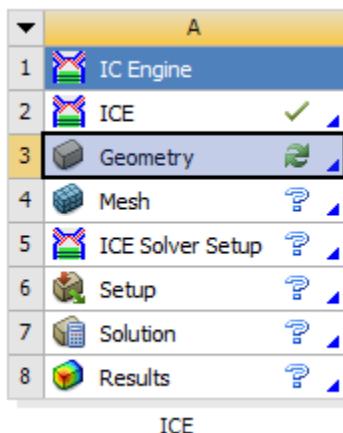
3. Select **Port Flow Simulation** from the **Simulation Type** drop-down list.

Note

The **ICE** cell is updated after you select **Port Flow Simulation**. You can now proceed to decomposition.

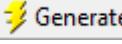
2.3. Step 2: Performing the Decomposition

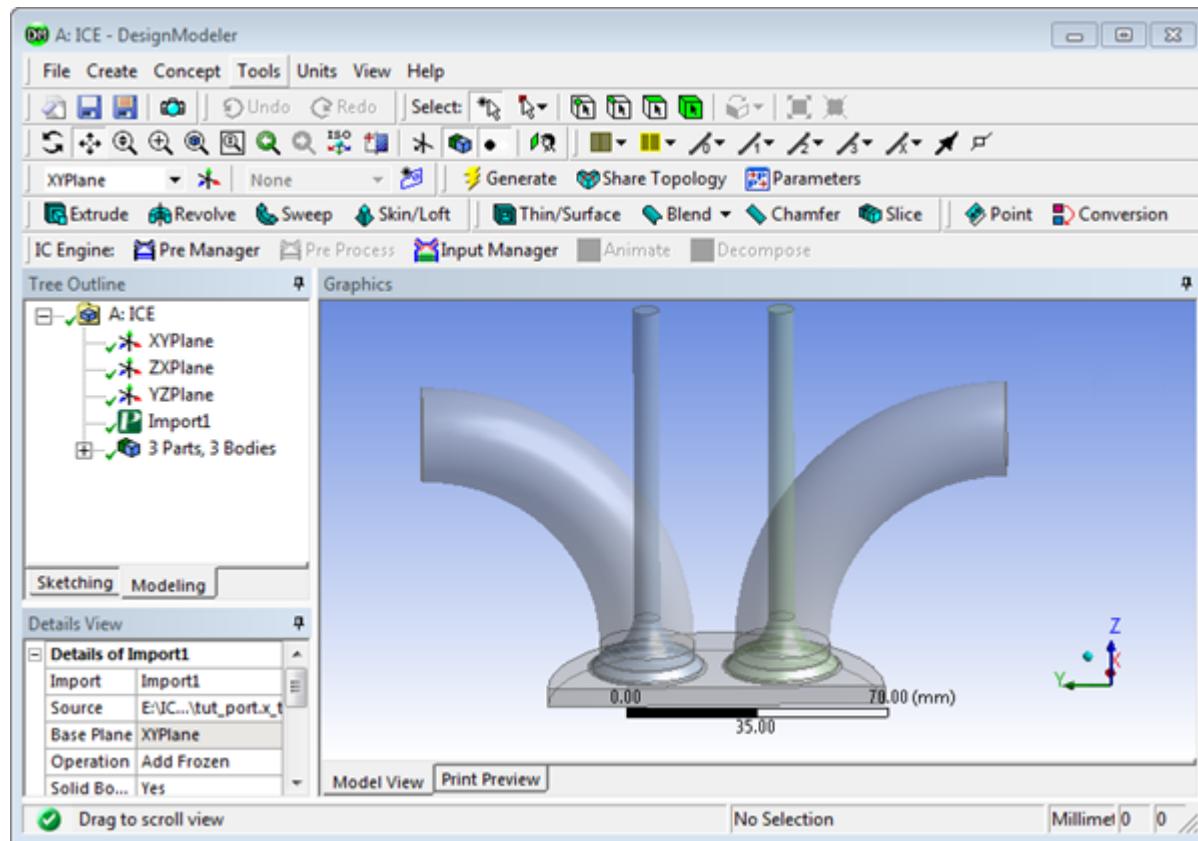
Here you will read the geometry and prepare it for decomposition. Double-click on the **Geometry** cell to open the DesignModeler.

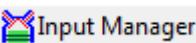


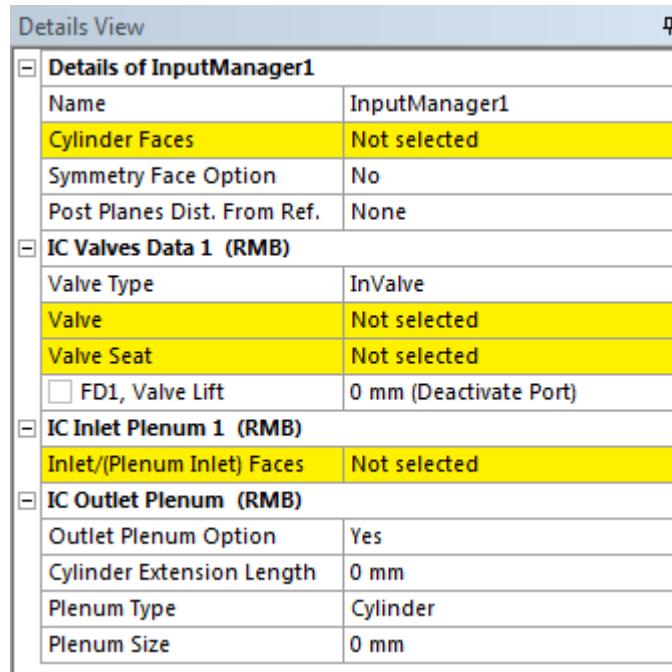
1. Select **Millimeter** from the **Units** menu.
2. Import the geometry file, `tut_port.x_t`.

File > Import External Geometry File...

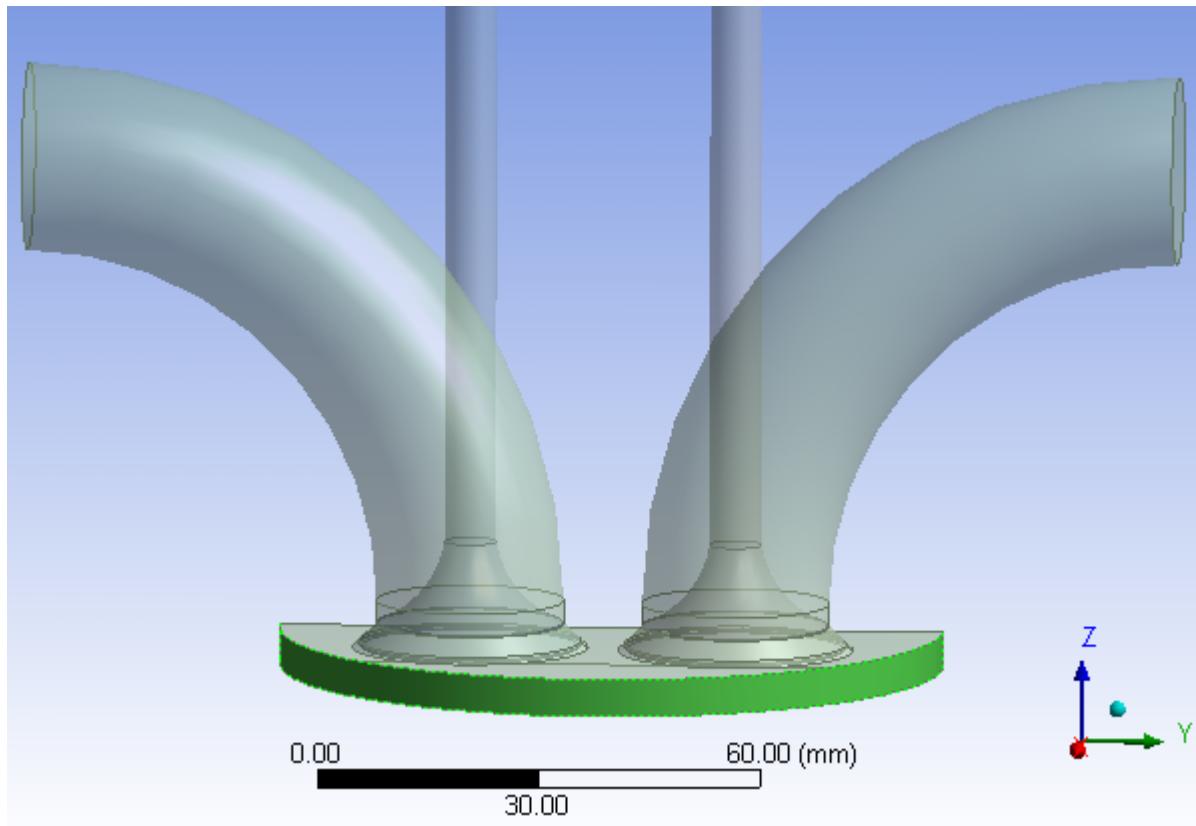
3. Click **Generate**  to complete the import feature.



4. Click **Input Manager**  located in the **IC Engine** toolbar.

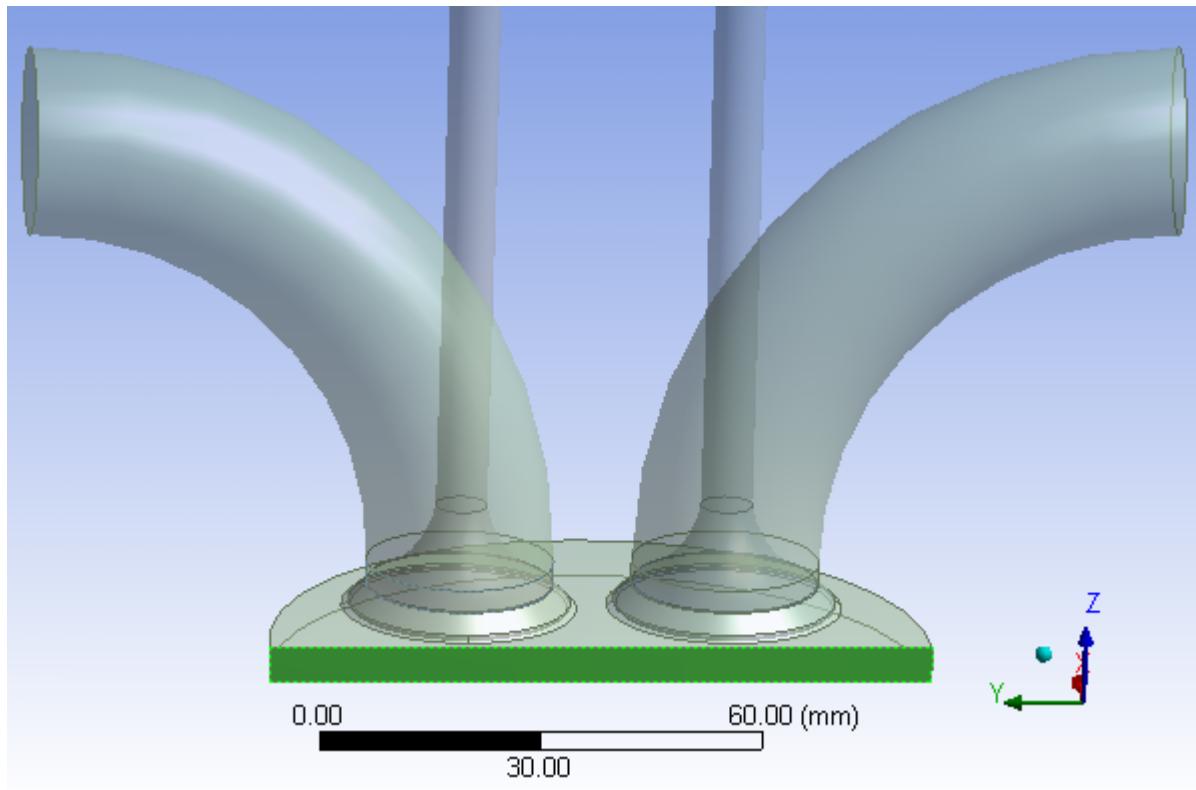


- a. Select the face as shown in [Figure 2.2: Cylinder Faces \(p. 67\)](#) for **Cylinder Faces** and click **Apply**.

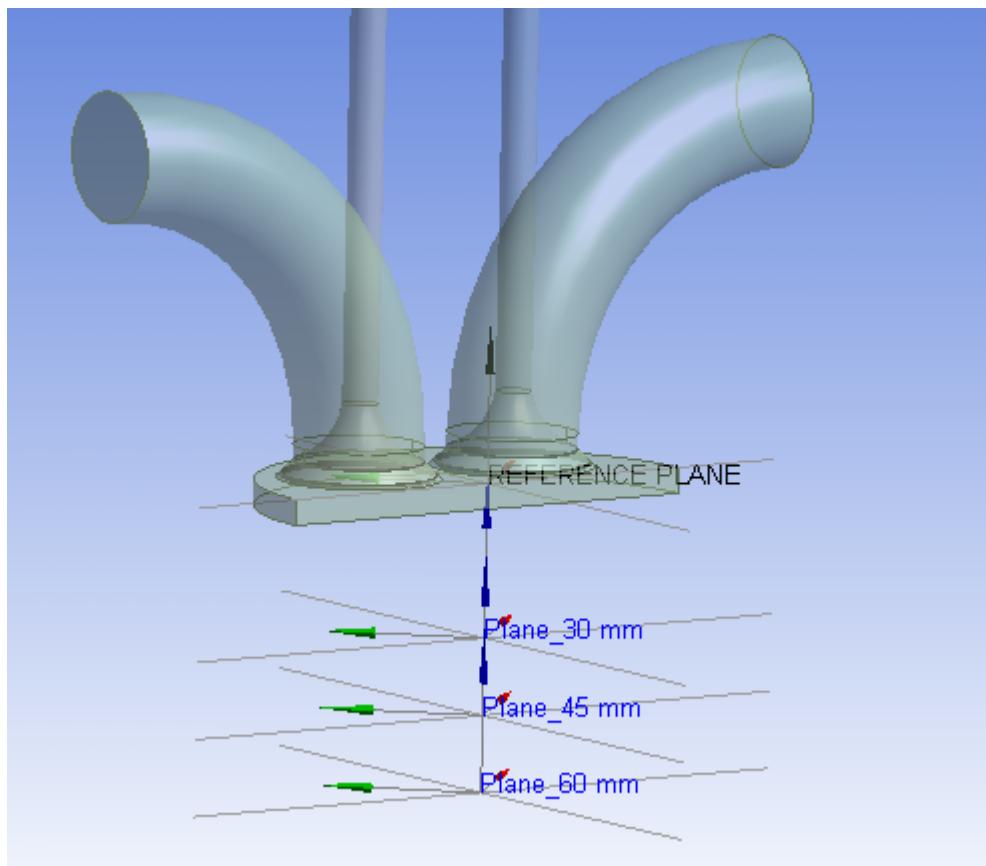
Figure 2.2: Cylinder Faces

- b. Retain **Yes** from the **Symmetry Face Option** drop-down list.
- c. Select the face shown in Figure 2.3: Symmetry Faces (p. 68) for **Symmetry Faces** and click **Apply**.

Figure 2.3: Symmetry Faces



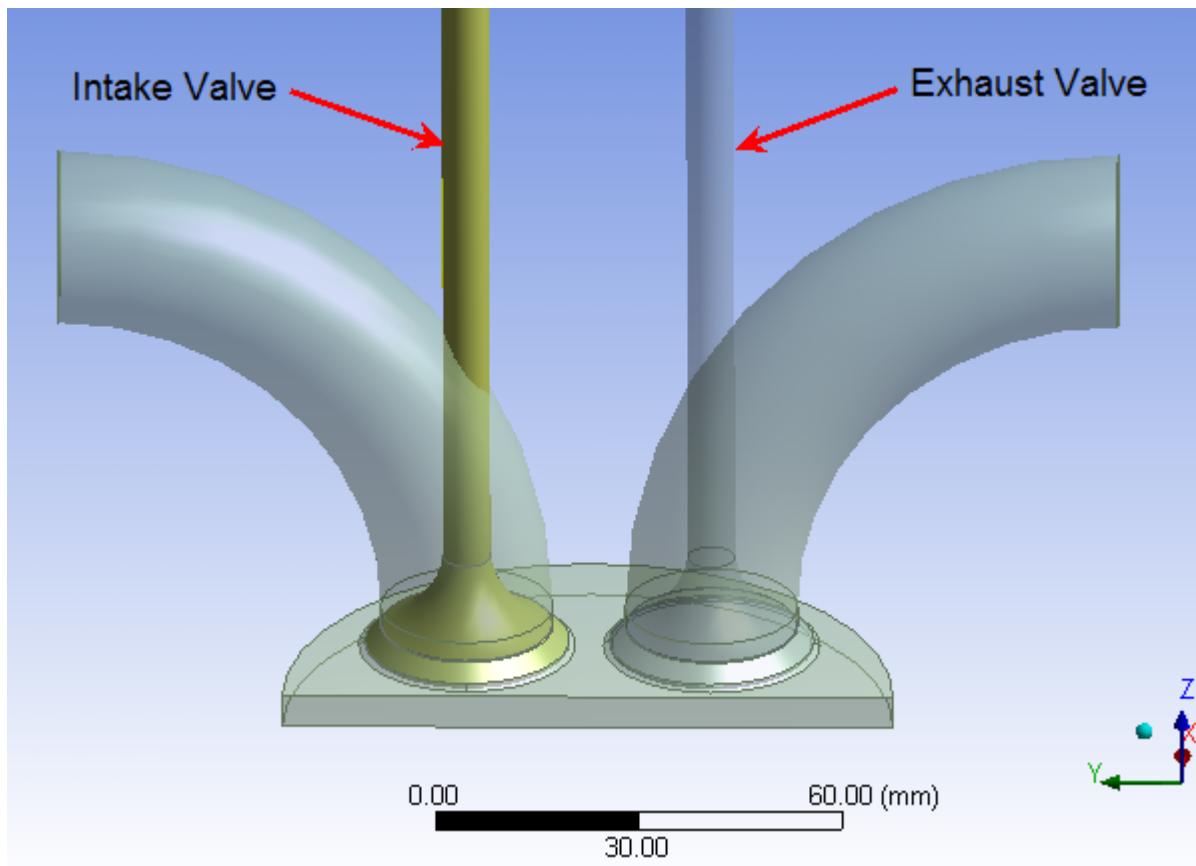
- d. For **Post Planes Dist. From Ref.** you can enter the distance at which you would like to have the postprocessing plane(s). It is a semicolon separated list, for e.g. you can enter 30 ; 45 ; 60.

Figure 2.4: Postprocessing Planes

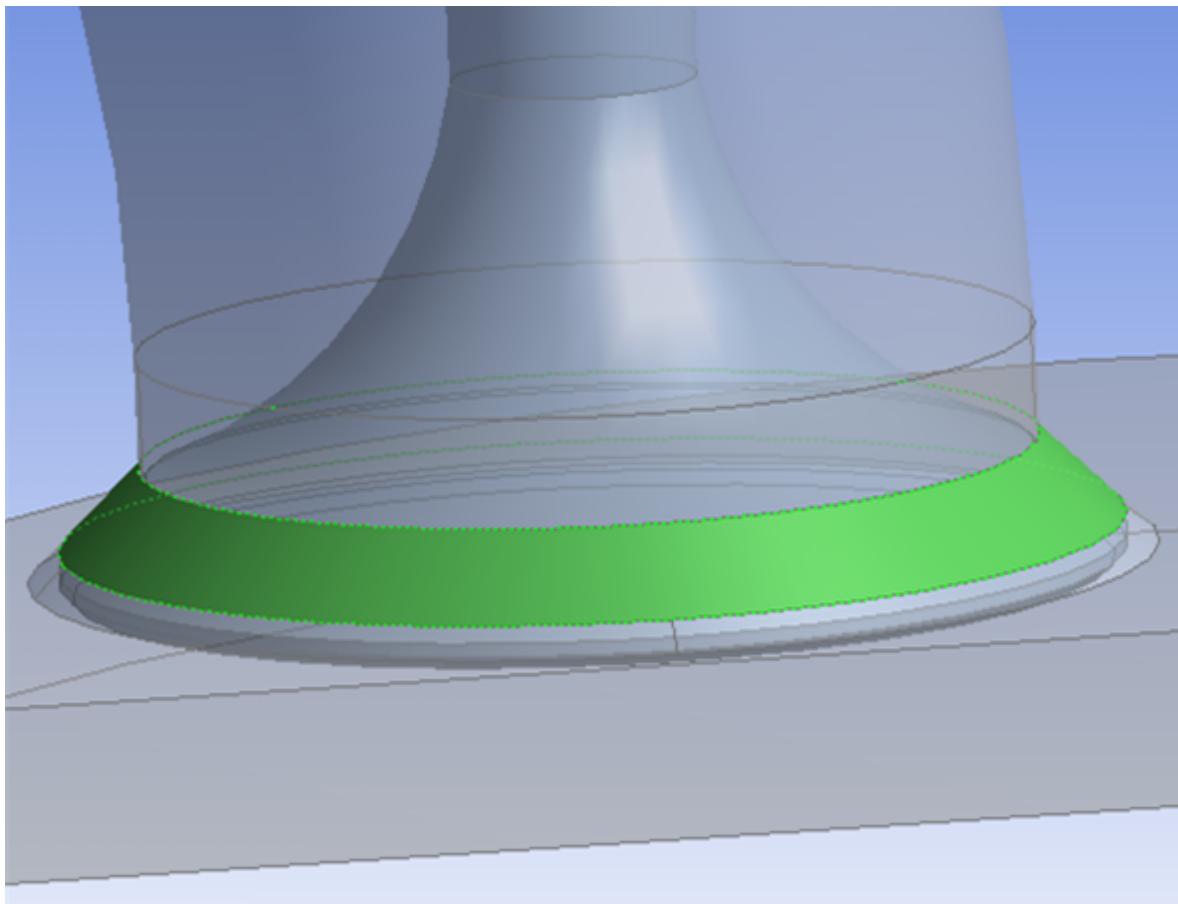
The representation of the reference planes and the postprocessing planes is visible in the geometry after you enter the distances. These planes are required for creating swirl monitors in Fluent.

- e. Retain selection of **InValve** from the **Valve Type** drop-down list.
- f. Select the valve body as shown in Figure 2.5: Intake Valve (p. 70) for **Valve Bodies** and click **Apply**.

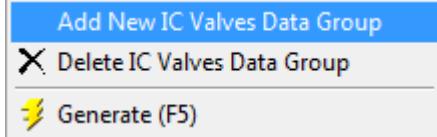
Figure 2.5: Intake Valve



- g. Select the valve seat face as shown in [Figure 2.6: Intake Valve Seat \(p. 71\)](#) for **Valve Seat Faces** and click **Apply**.

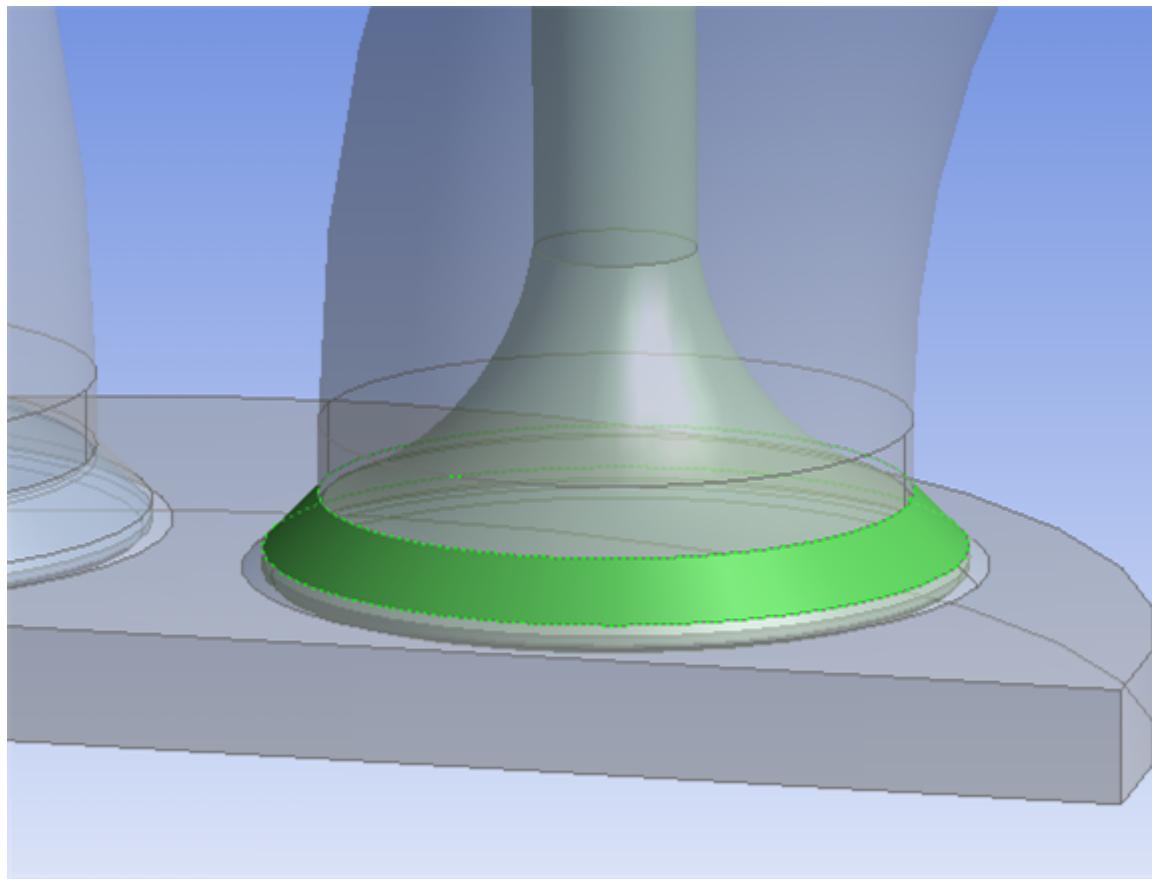
Figure 2.6: Intake Valve Seat

- h. Enter 2 for **Valve Lift**.
- i. Right-click on **IC Valves Data** in the **Details of InputManager** and select **Add New IC Valves Data Group** from the context menu.



- j. In this **IC Valves Data** group following the steps for the intake valve, set the other valve body to **ExValve**. Select the valve seat face of that valve as shown in [Figure 2.7: Exhaust Valve Seat \(p. 72\)](#).

Figure 2.7: Exhaust Valve Seat

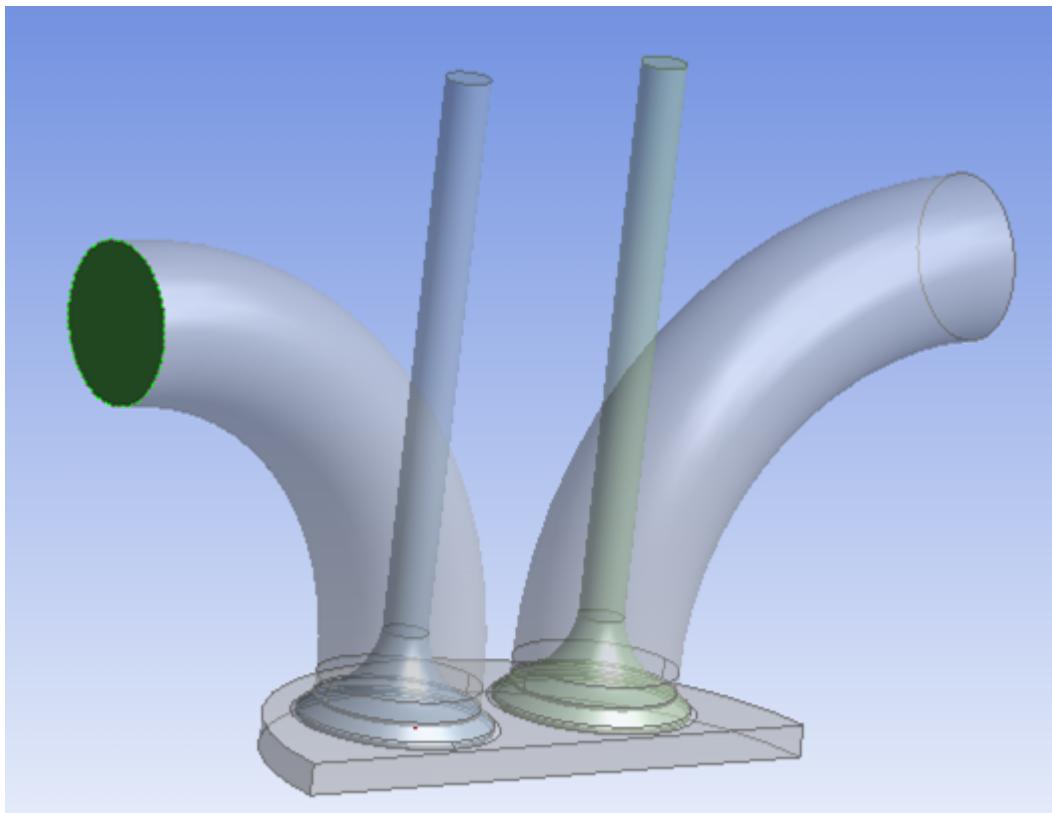


- k. Retain 0 for **Valve Lift**.
-

Note

This port will be automatically deactivated.

- l. Click next to **Inlet/(Plenum Inlet) Faces**, select the face of the inlet valve and click **Apply**.

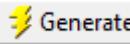
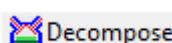
Figure 2.8: Inlet Face

- m. Select **Hemisphere** from the **Plenum Type** drop-down list.
- n. Retain the default value for **Inlet Extension Length**.
- o. Enter 100 for **Plenum Size**.
- p. Retain the default value for **Plenum Blend Rad**.
- q. The **Outlet Plenum Option** is set to **Yes**.
- r. Enter 130 for **Cylinder Extension Length**.
- s. Retain the default selection of **Cylinder** for **Pleum Type**.
- t. Enter 160 for **Plenum Size**.

Note

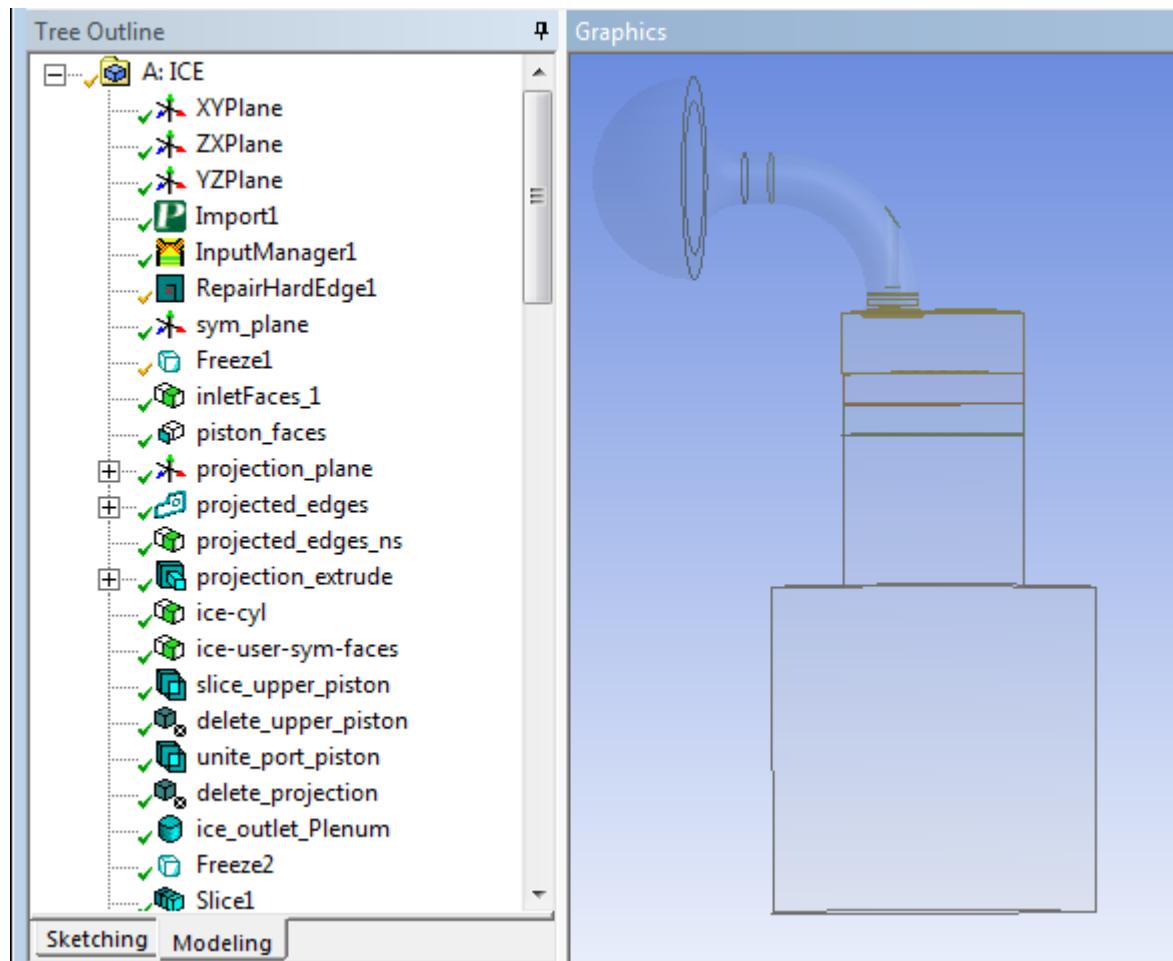
The default values **Plenum Size** and **Cylinder Extension Length** are reduced so that the number of mesh elements generated will be reduced. This will reduce the solution time. This is one way to optimize the solution.

Details of InputManager1	
Name	InputManager1
Cylinder Faces	1 Face
Symmetry Face Option	Yes
Symmetry Faces	1 Face
Post Planes Dist. From Ref.	30.0; 45.0; 60.0 (mm)
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve	1 Body
Valve Seat	1 Face
<input type="checkbox"/> FD1, Valve Lift	2 mm
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve	1 Body
Valve Seat	1 Face
<input type="checkbox"/> FD2, Valve Lift	0 mm (Deactivate Port)
IC Inlet Plenum 1 (RMB)	
Inlet/(Plenum Inlet) Faces	1 Face
Plenum Type	Hemisphere
Inlet Extension Length	38.1 mm
Plenum Size	100 mm
Plenum Blend Rad	25 mm
IC Outlet Plenum (RMB)	
Outlet Plenum Option	Yes
Cylinder Extension Length	130 mm
Plenum Type	Cylinder
Plenum Size	160 mm

- u. After all the settings are done click **Generate** 
5. Click **Decompose** ( located in the **IC Engine** toolbar).

Note

The decomposition process will take a few minutes.

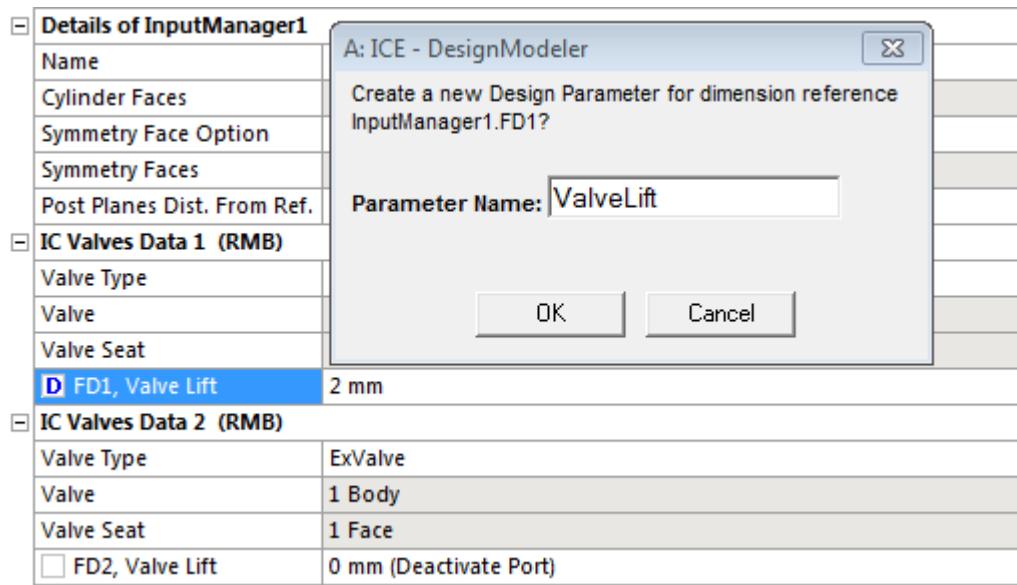
Figure 2.9: Decomposed Geometry

6. Add an input parameter.

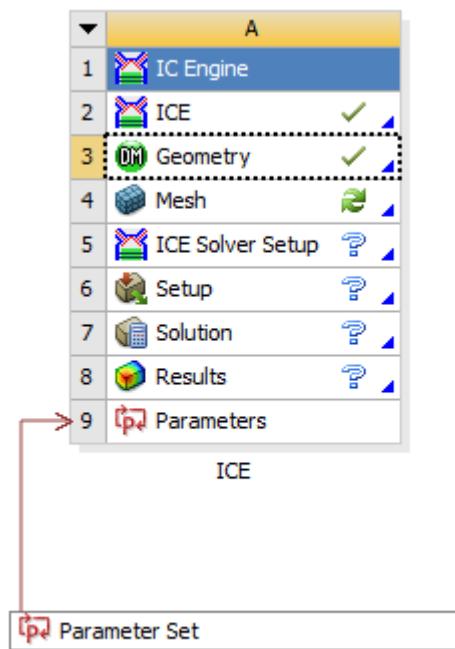
Note

Most of the port flow simulations are done to study the effect of valve lift on the velocity, mass flow rate, and other flow parameters. Here you will add design points. Valve lift is selected as the input parameter for this tutorial.

- a. Select **InputManager1** from the **Tree Outline**.
- b. Enable **FD1** next to **Valve Lift** for the **InValve**.



This will create a parameter for this component. A dialog box opens asking you to name the parameter. Enter **ValveLift** for the **Parameter Name**. Click **OK** to close the dialog box.



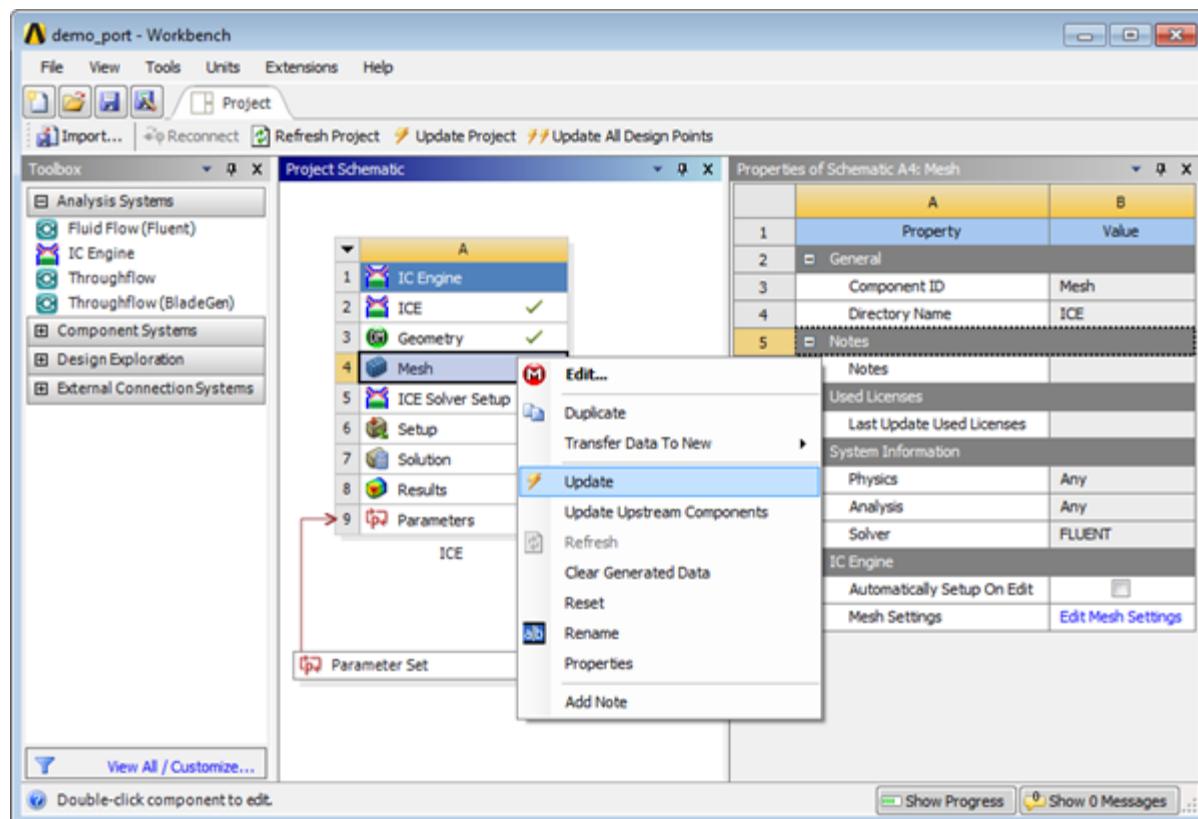
A **Parameters** cell is added to the **ICE** system and the **Parameter Set** is connected to the cell.

7. Close the DesignModeler.
8. Save the project by giving it a proper name (`demo_port.wbpj`).

2.4. Step 3: Meshing

Here you will mesh the decomposed geometry.

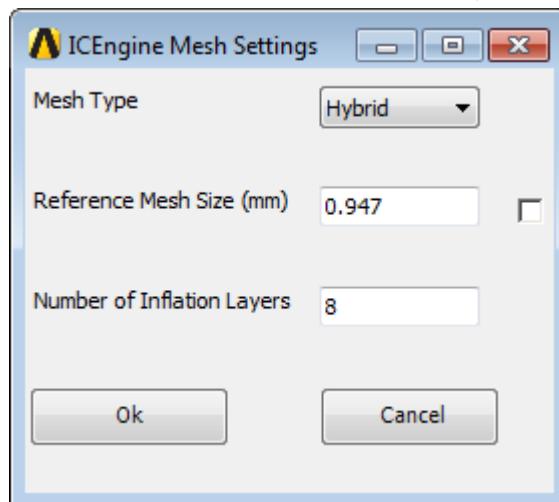
1. Right-click on **Mesh**, cell 4, and click **Update** from the context menu.



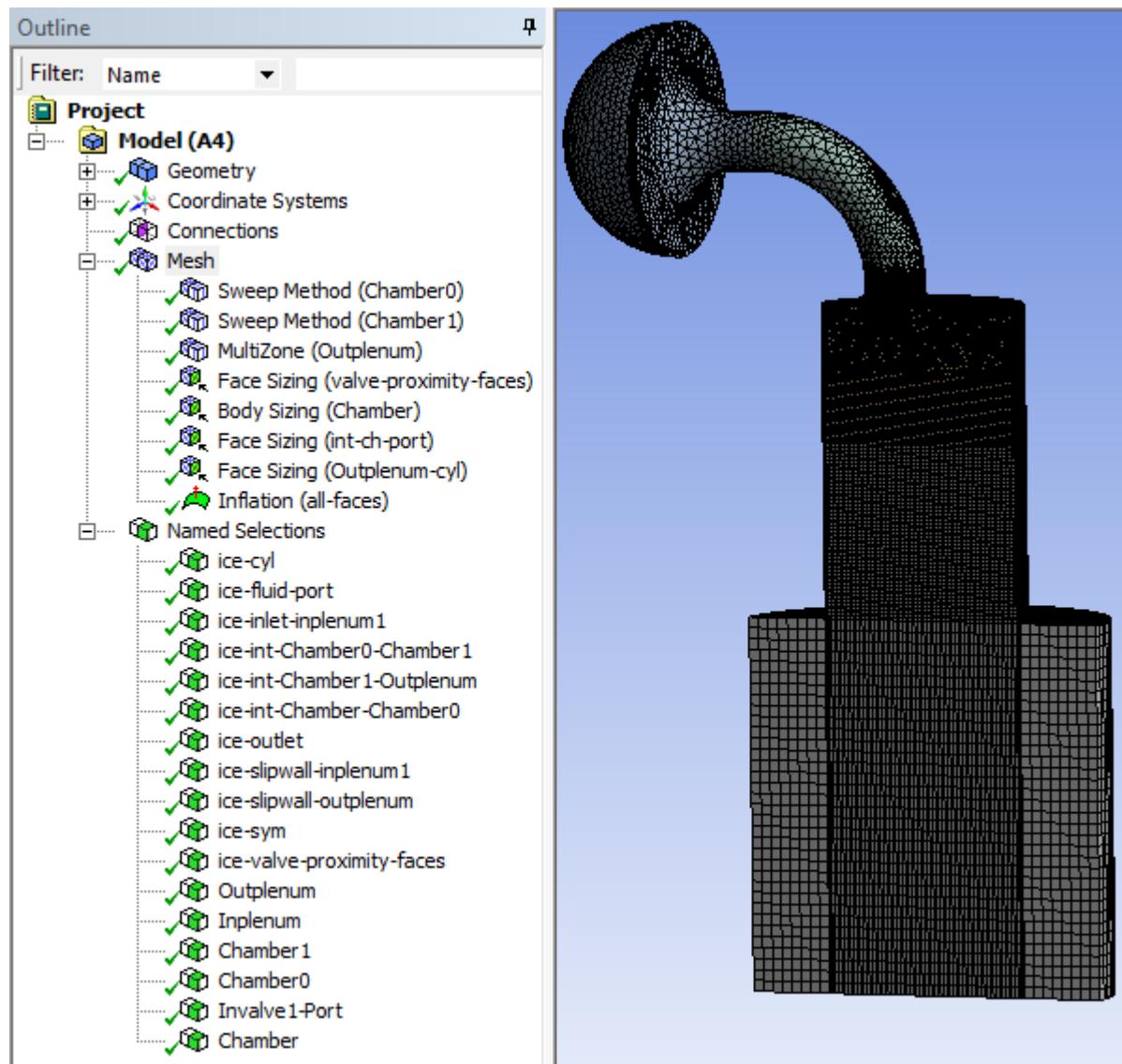
In a single step it will first create the mesh controls, then generate the mesh and finally update the mesh cell.

Note

If you want to check or change the mesh settings click **Edit Mesh Settings** in **Properties of Schematic A4: Mesh** under **IC Engine**.



For this tutorial you are going to retain the default mesh settings.

Figure 2.10: Meshed Geometry

- Save the project.

File > Save

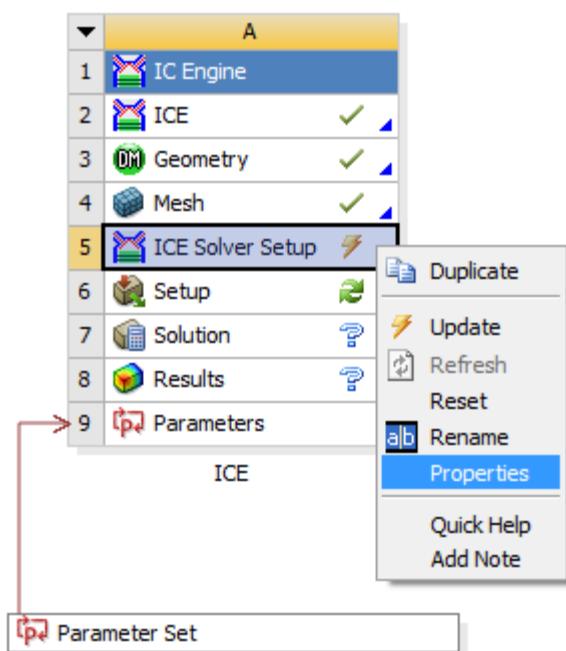
Note

It is a good practice to save the project after each cell update.

2.5. Step 4: Setting up the Simulation

After the decomposed geometry is meshed properly you can set boundary conditions, monitors, and postprocessing images. You can also decide which data and images should be included in the report.

- If the **Properties** view is not already visible, right-click **ICE**, cell 2, and select **Properties** from the context menu.



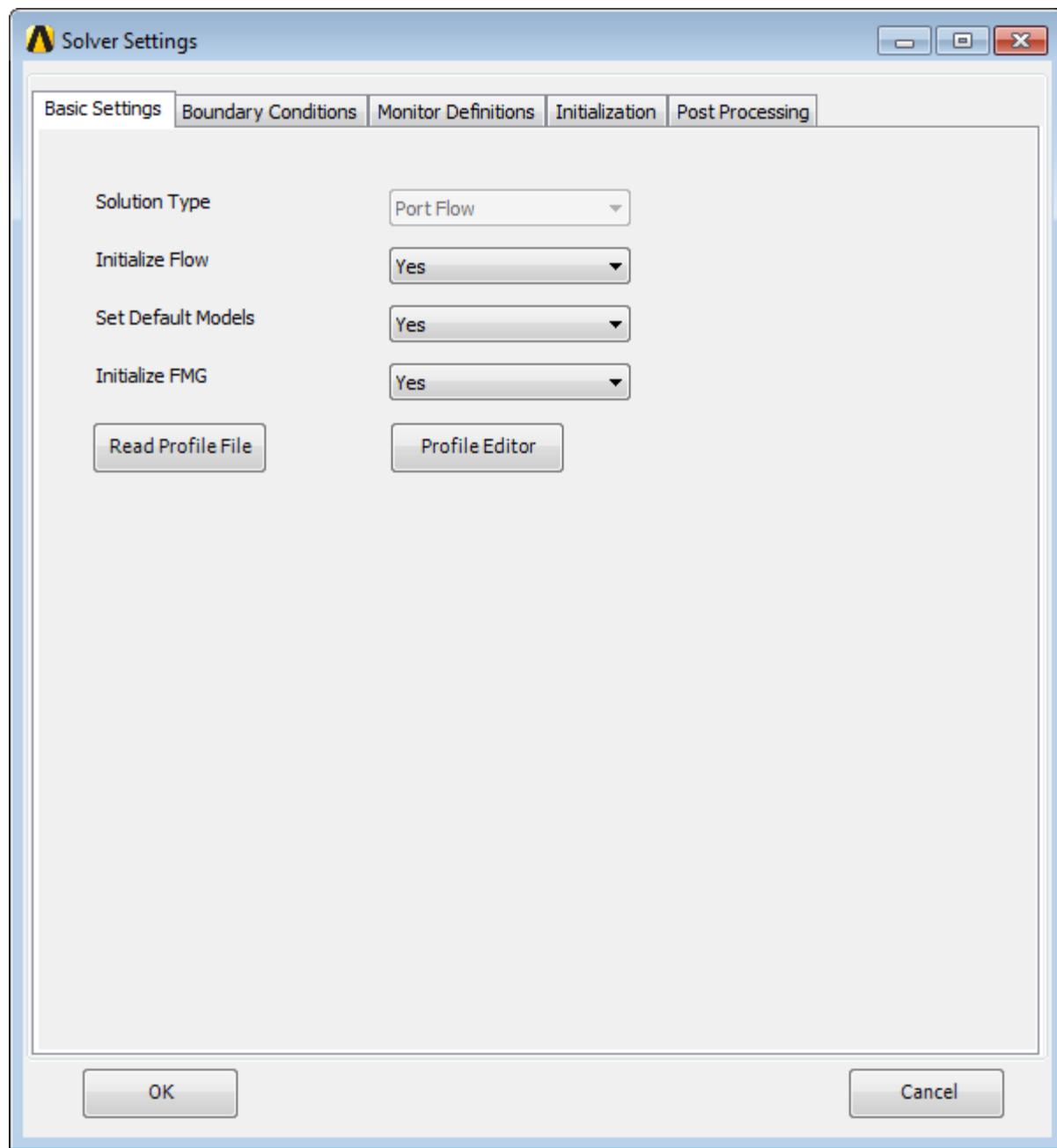
- Click **Edit Solver Settings** to open the **Solver Settings** dialog box.

Properties of Schematic A5: ICE Solver Setup		
	A	B
1	Property	Value
2	General	
3	Component ID	ICE Solver Setup
4	Directory Name	ICE
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Solver Settings	
10	Solver Settings	Edit Solver Settings
11	Journal Customization	
12	User Boundary Condition Profiles	
13	User Boundary Conditions and Monitor Settings	ICE\ICE\jcUserSettings.txt
14	Pre Iteration Journal	
15	Post Iteration Journal	

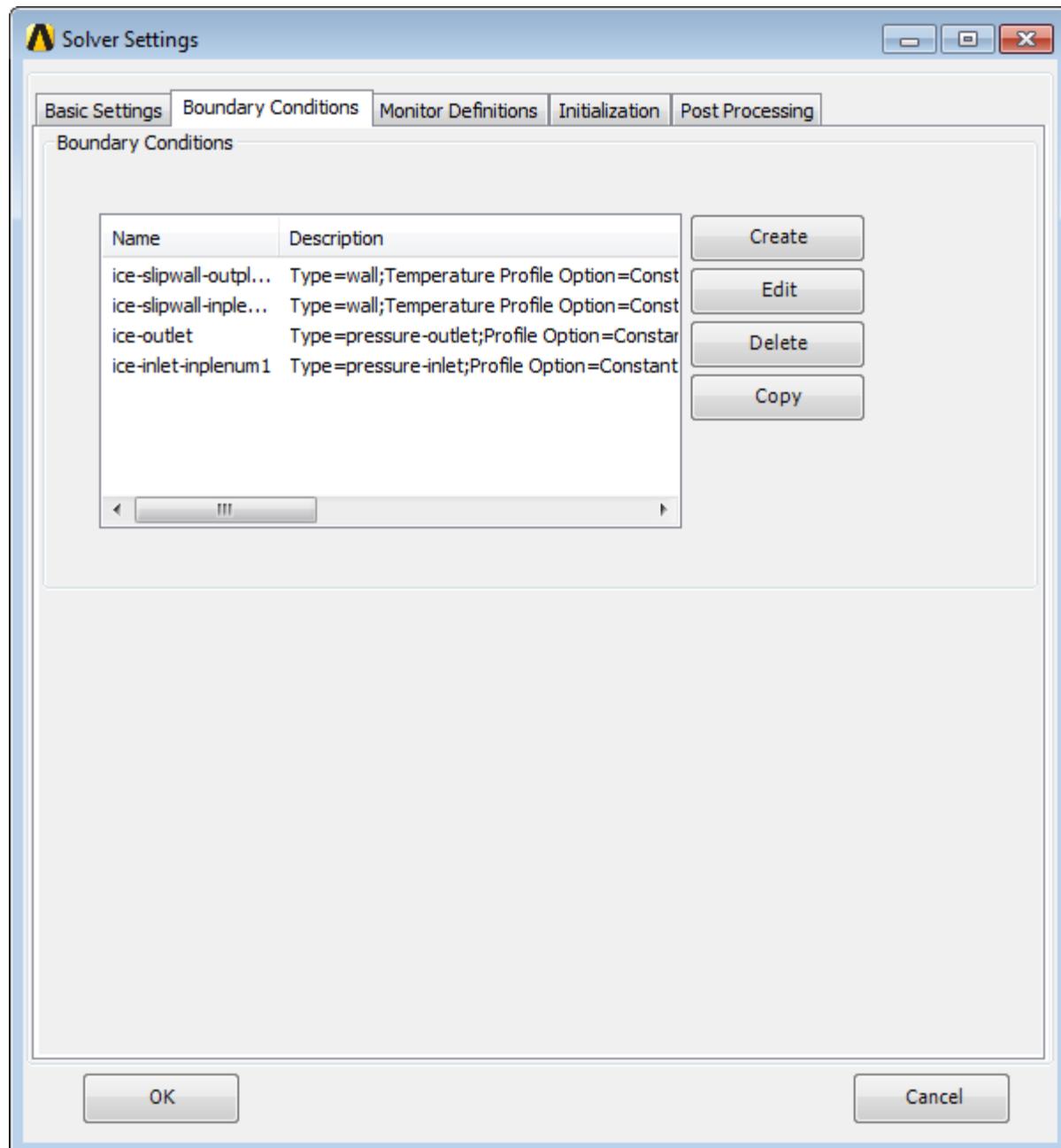
Note

In the **Solver Settings** dialog box you can check the default settings in the various tabs. If required you can change the settings.

- In the **Basic Settings** tab you can see that the default models are used and the flow is initialized using FMG.

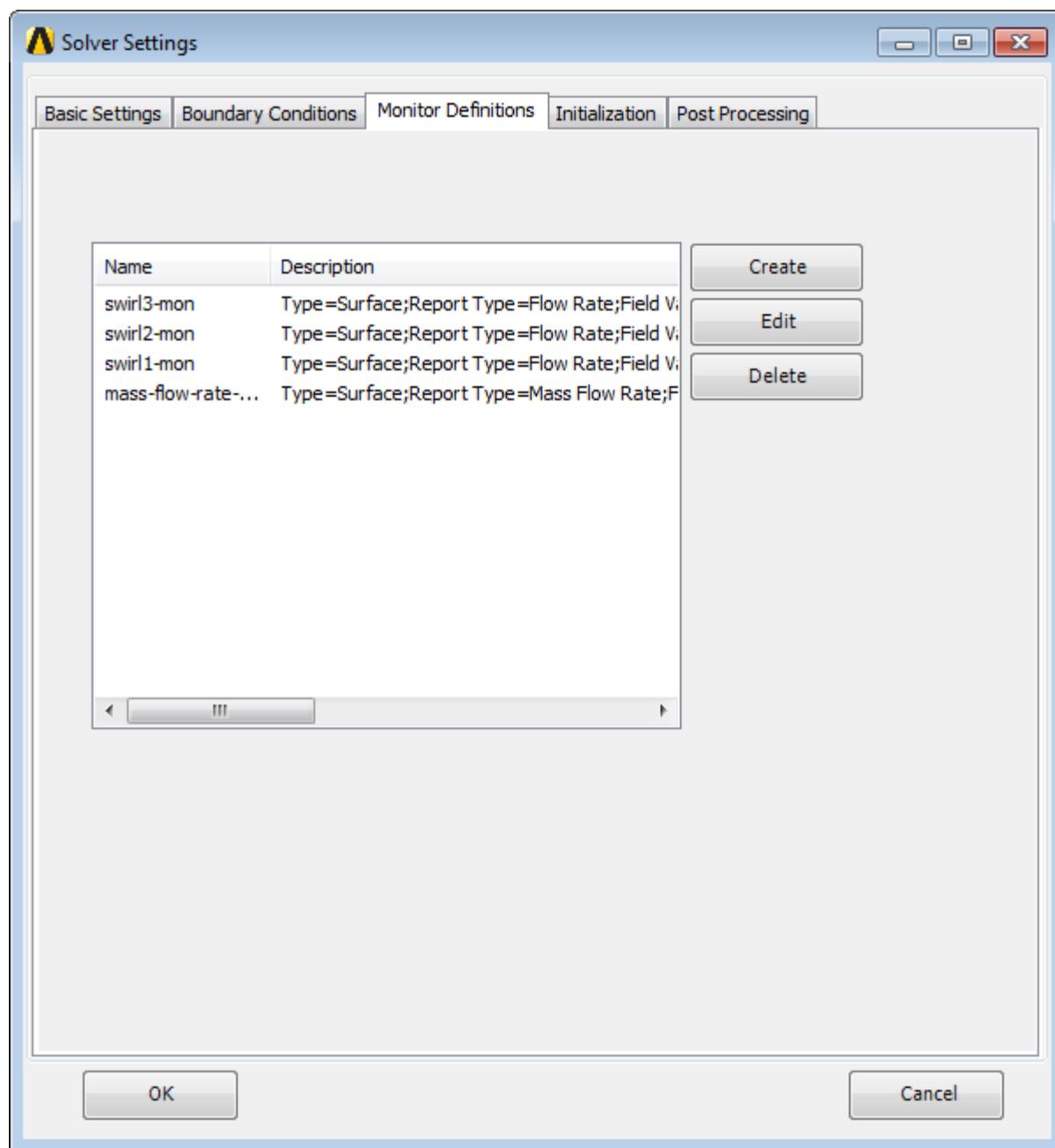


- b. In the **Boundary Conditions** tab you can see that the wall **ice-slipwall-outplenum** and **ice-slipwall-inplenum1** are set to **slipwall** with **Temperature** set to **300**.

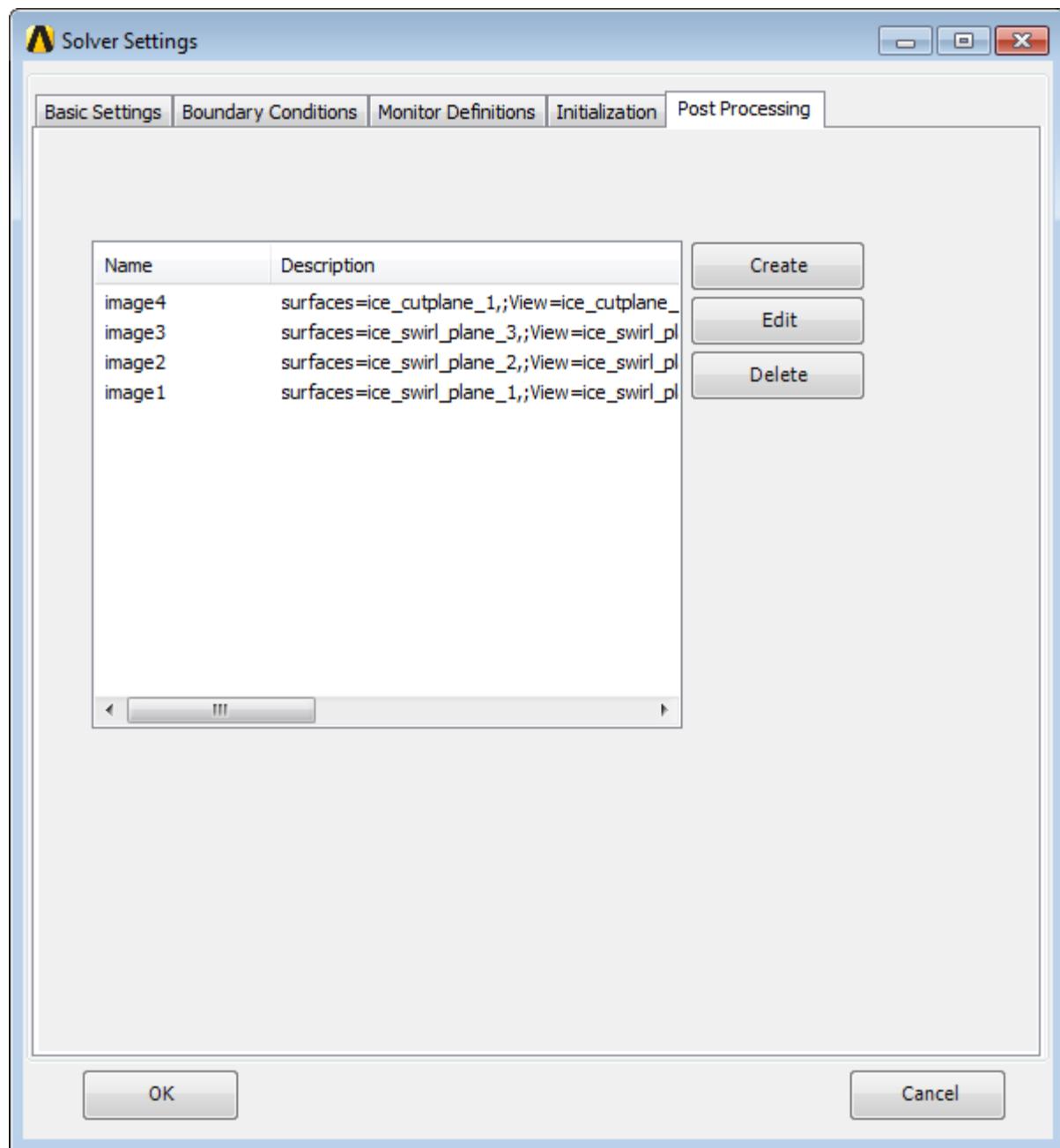


ice-outlet is set as **Pressure Outlet** with **Gauge Pressure** set to **-5000** and **Temperature** set to **300**. Similarly for **ice-inlet-inplenum1** which is set to type **Pressure Inlet**, **Temperature** is set to **300** and **Gauge Pressure** to **0**.

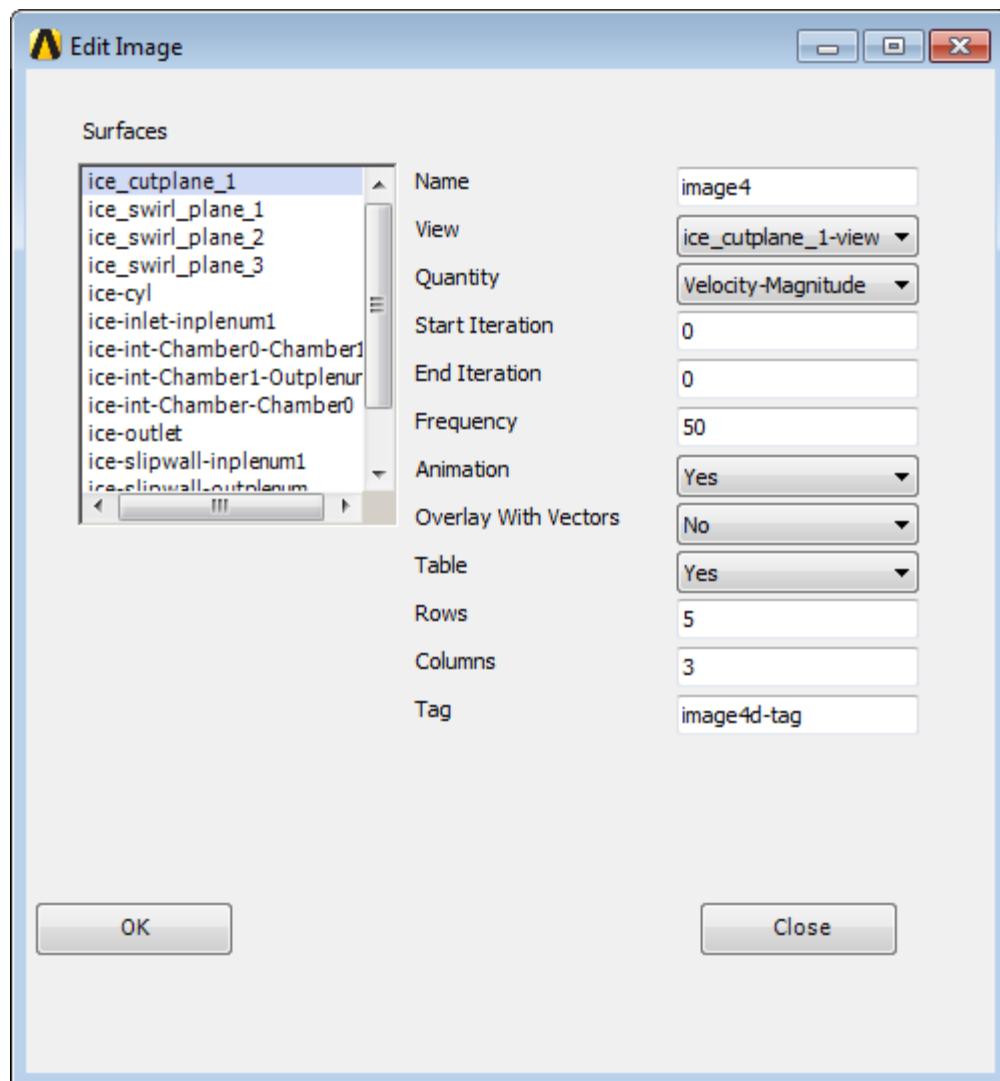
- c. In the **Monitor Definitions** tab you can see that four surface monitors have been set. Three plot the **Flow Rate** of swirl on the three swirl planes you have define in the **Input Manager**. One surface monitor plots the **Mass Flow Rate** on **ice-inlet-inplenum1** and **ice-outlet**.



- d. In the **Post Processing** tab you can see that four images are saved during simulation. Velocity-magnitude contours plotted on the surface of cut-plane and all the swirl planes will be saved during simulation and displayed in a table format in the report.



The details will be displayed after selecting the image name and clicking **Edit**.



Note

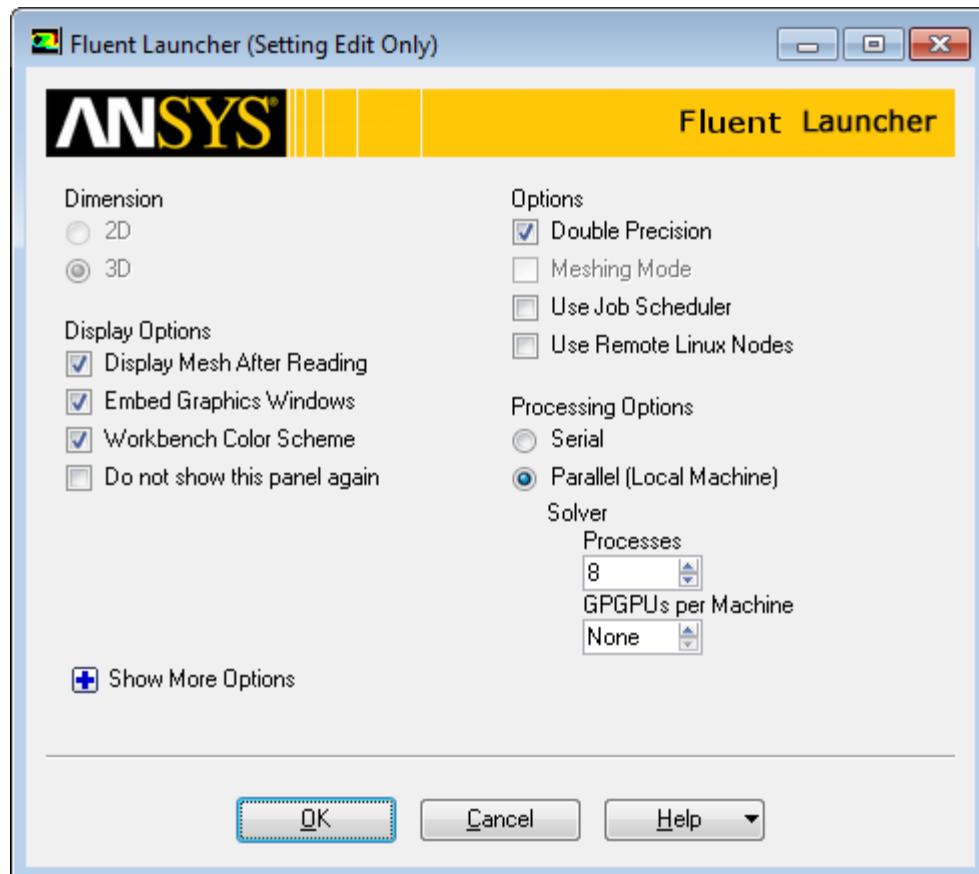
For this tutorial you will be using the default solver settings. You can try changing the settings and observe the difference in the results.

3. After checking the settings close the **Solver Settings** dialog box.
4. Right-click the **ICE Solver Setup** cell and click on **Update** from the context menu.

2.6. Step 5: Running the Solution

In this step you will setup the solution.

1. Double-click on the **Setup** cell.

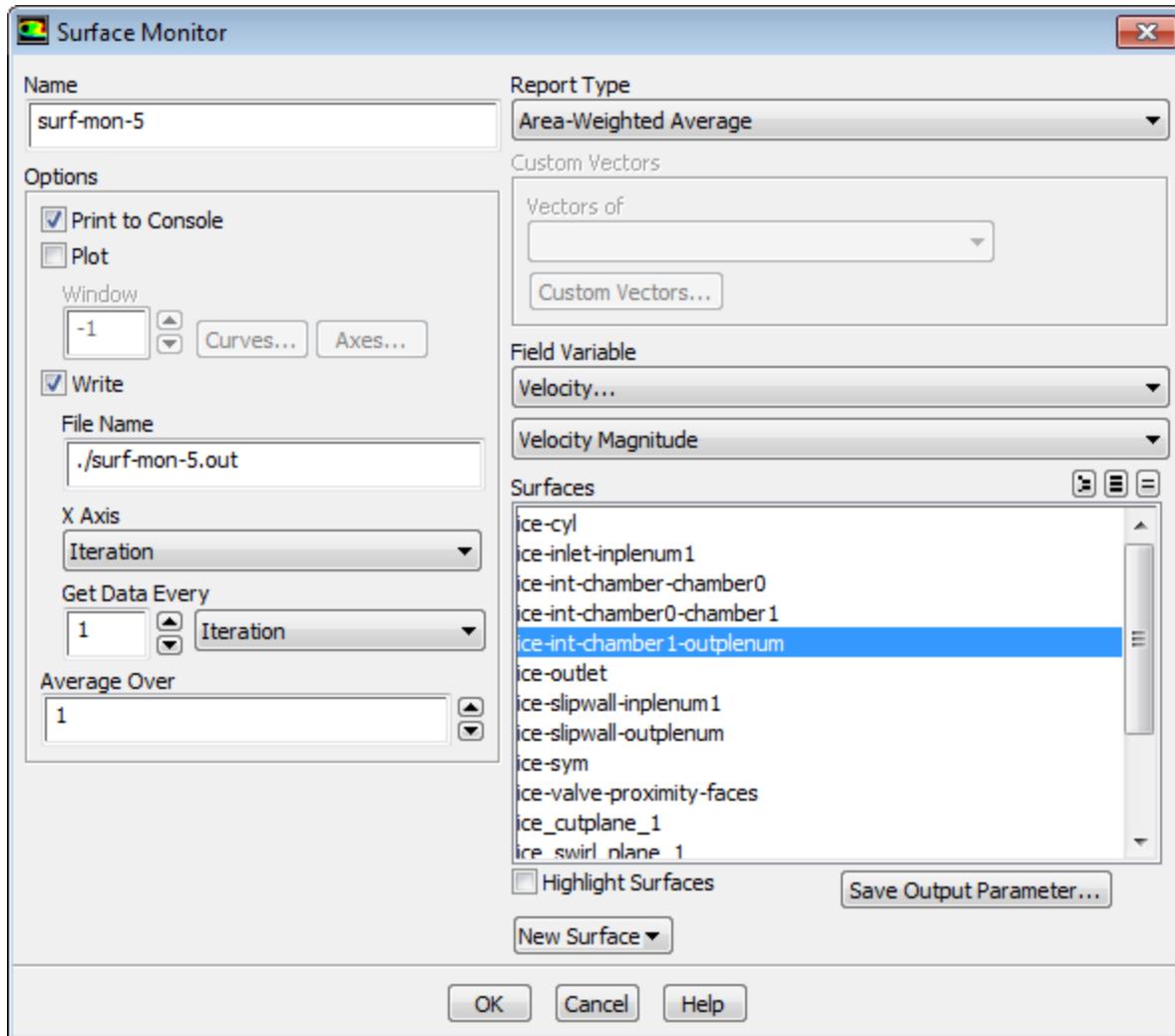


2. Ensure that **Double Precision** is enabled under **Options**.
3. You can run the simulation in parallel with increased number of processors to complete the solution in less time.
4. Click **OK** in the **Fluent Launcher** dialog box.

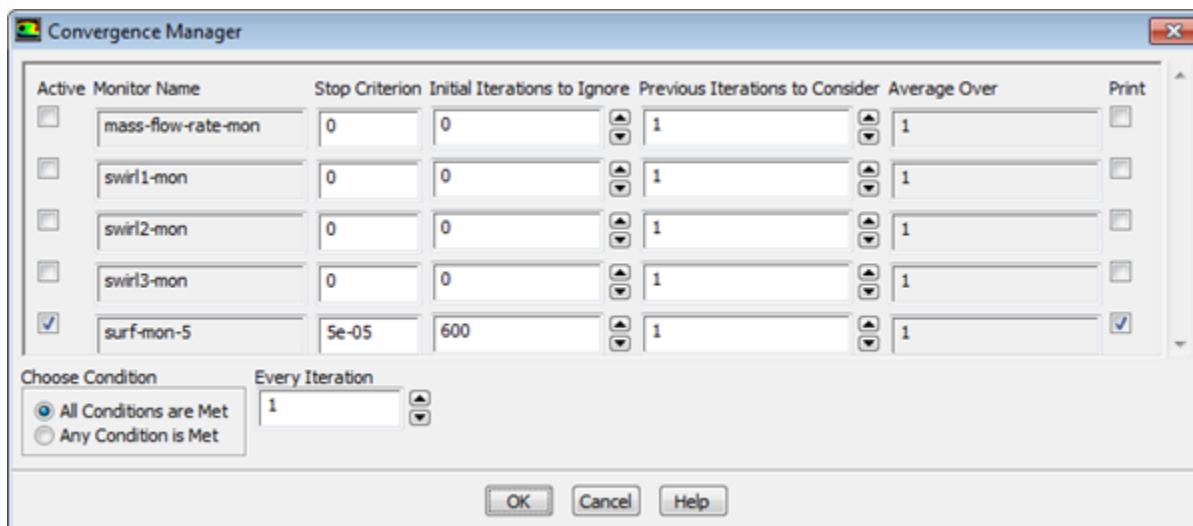
Note

ANSYS Fluent opens. It will read the mesh file and setup the case.

5. For the solution of this tutorial you will use monitor based convergence criteria. To achieve this you will define one velocity-magnitude surface monitor on an interior face zone and then will use this data for defining convergence criteria.
 - Select **Monitors** from the navigation pane and click **Create...** under the **Surface Monitors** group box.



- a. Retain the default name of **surf-mon-5**.
 - b. Enable **Write**.
 - c. Select **Area-Weighted Average** from the **Report Type** drop-down list.
 - d. Select **Velocity...** and **Velocity Magnitude** from the **Field Variable** drop-down lists.
 - e. Select **ice-int-chamber1-outplenum** from the list of **Surfaces**.
 - f. Click **OK** in the **Surface Monitor** dialog box and close it.
6. Add convergence criteria for the **Area-Weighted Average** monitor.
- Click **Convergence Manager...** under the **Convergence Monitors** group box.



- a. In the **Convergence Manager** dialog box enable the monitor **surf-mon-5** which you have just created.

Note

The solution is considered to be converged if the criteria of all of the **Active** monitors are satisfied.

- b. Enter 5e-05 as the **Stop Criterion** for **surf-mon-5**.

Note

Stop Criterion indicates the criterion below which the solution is considered to be converged.

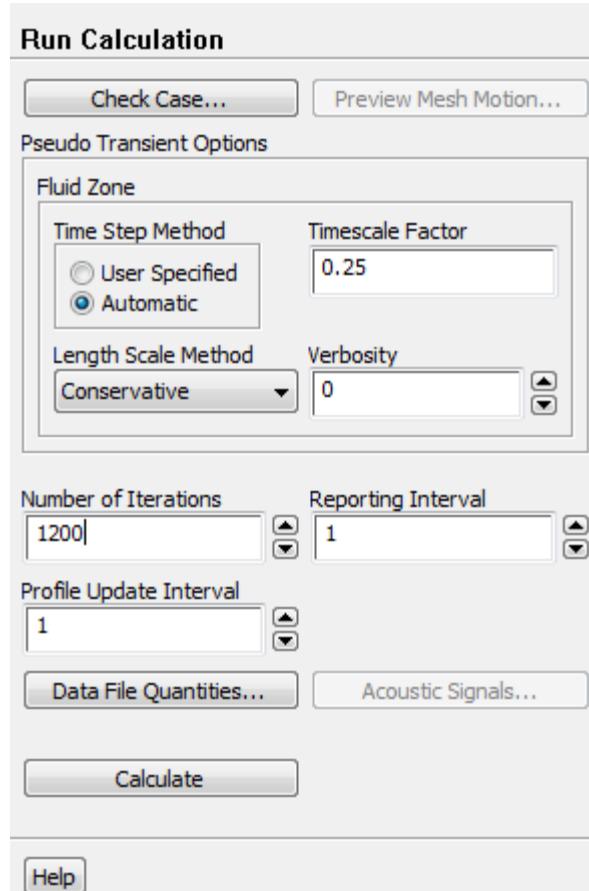
- c. Enter 600 as the **Initial Iterations to Ignore** for **surf-mon-5**.

Note

Enter a value in the **Initial Iterations to Ignore** column if you expect your solution to fluctuate in the initial iterations. Enter a value that represents the number of iterations you anticipate the fluctuations to continue. The convergence monitor calculation will begin after the entered number of iterations have been completed. For more information refer to, [Convergence Manager](#) in the [Fluent User's Guide](#).

- d. Enable **Print** for **surf-mon-5**.
e. Click **OK** to set and close the **Convergence Manager** dialog box.

7. Click **Run Calculation** in the navigation pane.

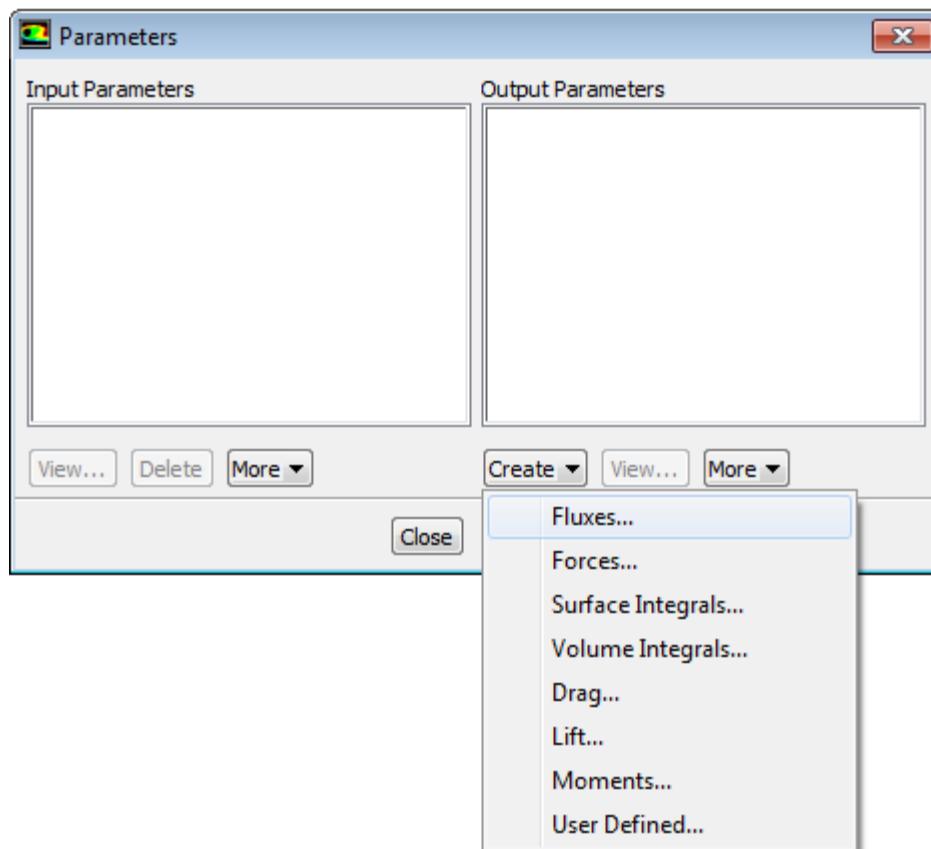


- Enter 1200 for the **Number of Iterations**.
8. Add an output parameter.

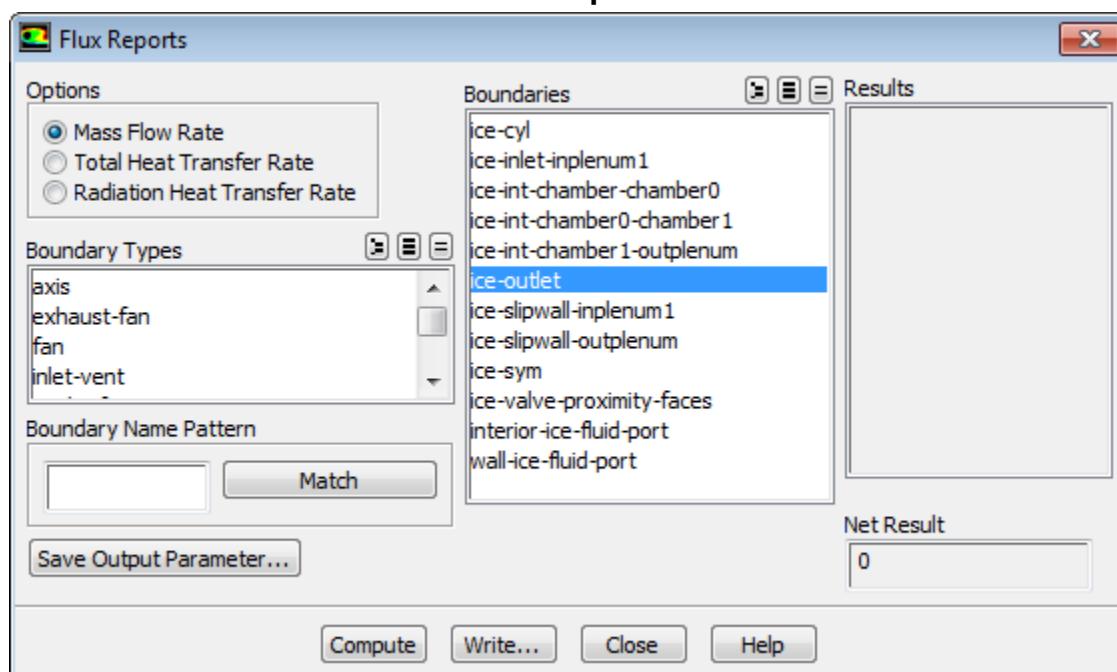
Note

To quantify the output result, mass flow rate is defined as the output parameter. So at the end of this design points study, change in the mass flow rate for the above defined valve lifts can be observed.

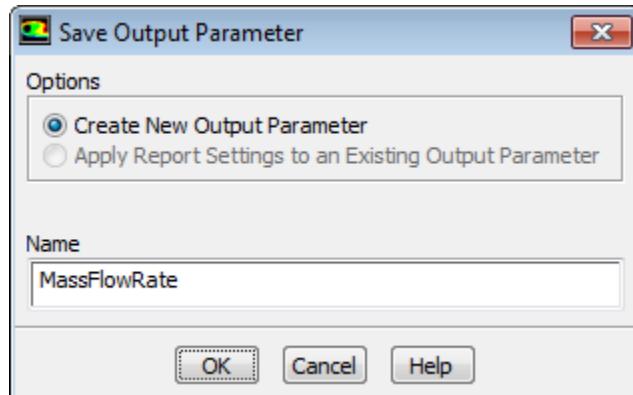
- a. Click **Parameters...** from the **Define** menu.
- b. In the **Parameters** dialog box click **Create** and select **Fluxes...** from the drop-down list.



- i. Retain selection of **Mass Flow Rate** from the **Options** list.

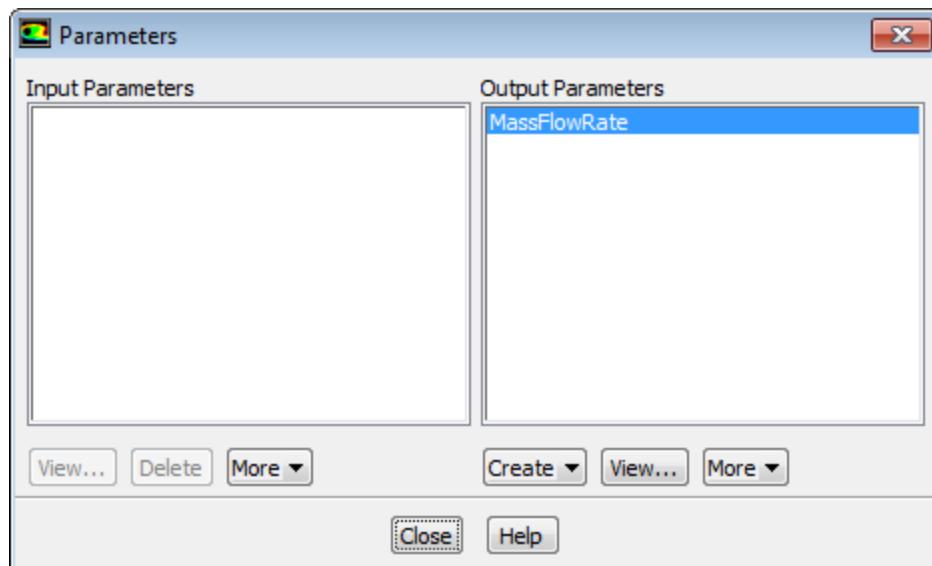


- ii. Select **ice-outlet** from the list of **Boundaries**.
- iii. Click **Save Output Parameter....**

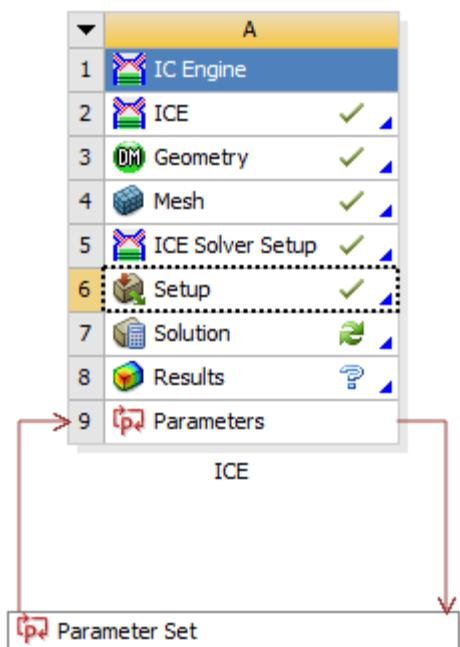


Enter **MassFlowRate** for the **Name** of and click **OK** to close the **Save Output Parameter** dialog box.

- iv. Close the **Flux Reports** dialog box.
- v. The parameter **MassFlowRate** is added under **Output Parameters** in the **Parameters** dialog box. Click **Close**.



9. Go to the ANSYS Workbench window. The parameter loop is now complete.



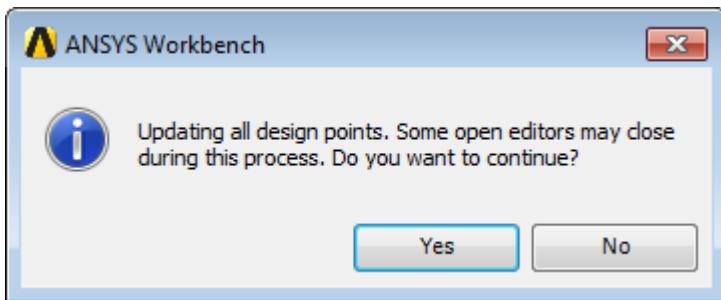
10. In ANSYS Workbench double-click on the parameter bar or right-mouse click and select **Edit...** from the context menu to access the Parameters and Design Points workspace.
11. In the Parameters and Design Points view, you will see the work area of **Table of Design Points**. Enter 6 and 10 in the column of **P1-ValveLift**.
12. Enable the check box next to **Retain** which will enable all check boxes in the **Retain** column for the design points you have added.

	A	B	C	D	E
1	Name ▾	P1 - ValveLift ▾	P2 - MassFlowRate ▾	<input type="checkbox"/> Exported	Note ▾
2	Units	mm	kg s^-1		
3	Current	2	⚡		
4	DP 1	6	⚡	<input checked="" type="checkbox"/>	
5	DP 2	10	⚡	<input checked="" type="checkbox"/>	
*				<input type="checkbox"/>	

13. After adding the desired valve lift values click **Update All Design Points** () from the menu bar.

Note

Click **Yes** in the message dialog box that appears.



Now the simulation will run for each design point. This process will take some time to complete. As solution for each design point is completed its output parameter is updated in the **Table of Design Points** under **MassFlowRate**.

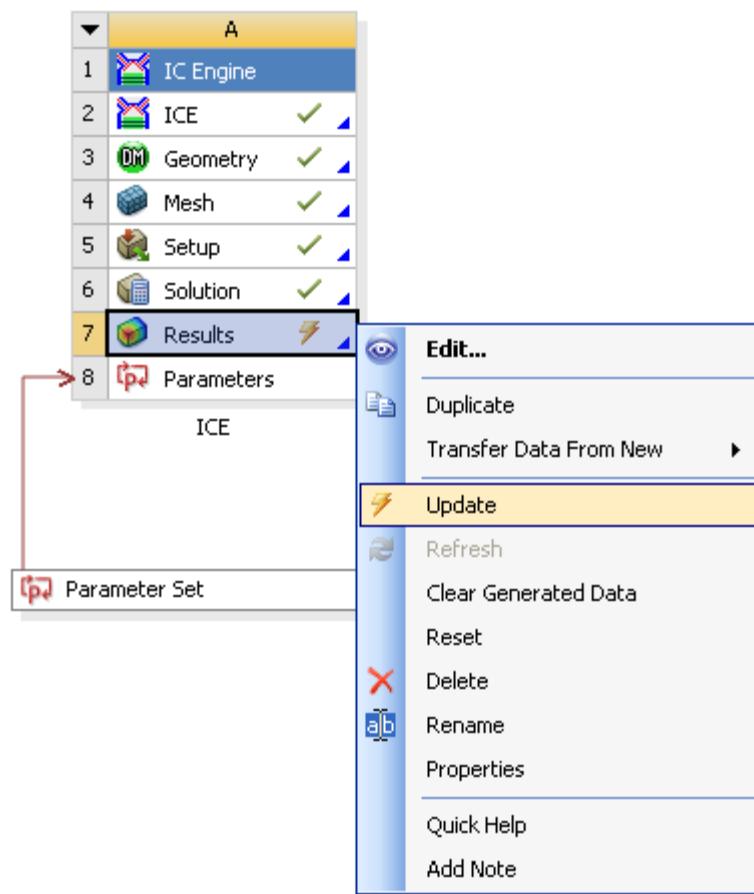
	A	B	C	D	E
1	Name	P1 - ValveLift	P2 - MassFlowRate	<input type="checkbox"/> Exported	Note
2	Units		kg s ⁻¹		
3	Current	2	-0.012873		
4	DP 1	6	-0.029861	<input checked="" type="checkbox"/>	
5	DP 2	10	-0.0357	<input checked="" type="checkbox"/>	
*				<input type="checkbox"/>	

Note

Updating the design points can take around 5 hours on a 8 CPU machine. You can open the project file provided and check the results.

2.7. Step 6: Obtaining the Results

1. Right-click on the **Results** cell and click **Update** from the context menu.



2. Once the **Results** cell is updated, view the files by clicking **Files** from the **View** menu.

3. View > Files

	A	B	C	D	E	F
1	Name	Cell ID	S...	Type	Date Modified	Location
2	tut_port.x_t	A3	47 KB	Geometry File	7/2/2012 12:45:11 PM	E:\ICE
3	demo_port.wbpj		160 KB	ANSYS Project File	7/4/2012 10:53:59 AM	E:\ICE\port
4	ICE.agdb	A3	2 MB	Geometry File	7/3/2012 3:02:45 PM	E:\ICE\port\demo_port_files\dp0\ICE\DM
5	ICE.msddb	A4	27 MB	Mesh Database File	7/3/2012 11:05:59 AM	E:\ICE\port\demo_port_files\dp0\global\MECH
6	ICE.msh	A4,A5	106 MB	Fluent Mesh File	7/3/2012 11:06:41 AM	E:\ICE\port\demo_port_files\dp0\ICE\MECH
7	ice-swirl1-mon.out	A5	127 B	FLUENT Surface Monitors	7/3/2012 6:35:19 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
8	ice-swirl2-mon.out	A5	127 B	FLUENT Surface Monitors	7/3/2012 6:35:25 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
9	ice-swirl3-mon.out	A5	127 B	FLUENT Surface Monitors	7/3/2012 6:35:28 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
10	surf-mon.5.out	A5	167 B	FLUENT Surface Monitors	7/3/2012 6:35:31 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
11	ICE.set	A5	49 KB	FLUENT Model File	7/3/2012 4:39:08 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
12	ice-mass-flow-rate-mon.out	A5	137 B	FLUENT Surface Monitors	7/3/2012 6:30:35 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
13	ICE-1.cas.gz	A1	42 MB	FLUENT Case File	7/3/2012 4:39:23 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
14	ICE-1-00000.dat.gz	A1	160 MB	FLUENT Data File	7/3/2012 4:40:07 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
15	ICE-2.cas.gz	A6	54 MB	FLUENT Case File	7/3/2012 6:47:20 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
16	ICE-2-00001.dat.gz	A6	162 MB	FLUENT Data File	7/3/2012 6:48:17 PM	E:\ICE\port\demo_port_files\dp0\ICE\Fluent
17	Report.html	A7	18 KB	Default File	7/4/2012 10:36:39 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report
18	ice-velocity-eu		54 KB	Default File	7/4/2012 10:36:39 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report
19	ice-velocity-ma		49 KB	Default File	7/4/2012 10:36:39 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report
20	ice-velocity-ma		56 KB	Default File	7/4/2012 10:36:39 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report
21	ice-velocity-magnitude-on-image2-0001.jpg	A7	49 KB	Default File	7/4/2012 10:36:39 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report
22	ice-velocity-magnitude-on-image1-0627.jpg	A7	60 KB	Default File	7/4/2012 10:36:38 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report
23	ice-velocity-magnitude-on-image1-0001.jpg	A7	51 KB	Default File	7/4/2012 10:36:38 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report
24	Chart006.png	A7	13 KB	Default File	7/4/2012 10:36:38 AM	E:\ICE\port\demo_port_files\dp0\ICE\Post\Report

Right-click **Report.html** from the list of files, and click **Open Containing Folder** from the context menu.

- In the **Report** folder double-click **Report.html** to open the report.

ANSYS®

Title
IC Engine Port Flow Simulation Report

Date
2013/08/06 14:59:03

Contents

- [1. File Report](#)
 - [Table 1 File Information for ICE](#)
- [2. Mesh Report](#)
 - [Table 2 Mesh Information for ICE](#)
- [3. Setup](#)
 - [Table 3 Boundary Conditions](#)
 - [Table 4 Models](#)
 - [Table 5 Equations](#)
 - [Table 6 Relaxation](#)
 - [Table 7 Pressure-Velocity Coupling](#)
 - [Table 8 Discretization Scheme](#)
- [4. Solution Data](#)
 - [4.1. Animation: velocity-magnitude on ice_cutplane_1](#)
 - [4.2. Table: velocity-magnitude on ice_swirl_plane_1 ice_swirl_plane_2 ice_swirl_plane_3](#)
 - [4.3. Table: velocity-magnitude on ice_cutplane_1](#)
 - [4.4. Residuals](#)
 - [4.5. Charts](#)
 - [Chart 1 Monitor: Mass Flow Rate \(ice-inlet-inplenum1 ice-outlet\)](#)
 - [Chart 2 Monitor: Flow Rate swirl1 \(ice_swirl_plane_1\)](#)
 - [Chart 3 Monitor: Flow Rate swirl2 \(ice_swirl_plane_2\)](#)
 - [Chart 4 Monitor: Flow Rate swirl3 \(ice_swirl_plane_3\)](#)
 - [Chart 5 Convergence history of Velocity Magnitude on ice-int-chamber1-outplenum \(in SI units\)](#)
- [5. Design Points Report](#)
 - [5.1. Design Points Parameter values Charts](#)
 - [Chart 6 ValveLift \[mm\] vs MassFlowRate \[kg s⁻¹\]](#)
 - [5.2. Table: velocity-magnitude on ice_swirl_plane_1 for Design Points DP 0, DP 1, DP 2](#)
 - [5.3. Table: velocity-magnitude on ice_swirl_plane_2 for Design Points DP 0, DP 1, DP 2](#)
 - [5.4. Table: velocity-magnitude on ice_swirl_plane_3 for Design Points DP 0, DP 1, DP 2](#)
 - [5.5. Table: velocity-magnitude on ice_cutplane_1 for Design Points DP 0, DP 1, DP 2](#)

- You can check the node count and mesh count of the cell zones in the table, **Mesh Information for ICE**.

1. File Report

Table 1. File Information for ICE

Case	ICE
File Path	E:\ICE\ICE15\Port-Flow\demo_port_files\dp0\ICE\Fluent\ICE-1-00660.dat.gz
File Date	02 August 2013
File Time	07:27:56 PM
File Type	FLUENT
File Version	15.0.0

2. Mesh Report

Table 2. Mesh Information for ICE

Domain	Nodes	Elements
ice fluid port	566452	1484953

- You can see the boundary conditions set, in the table **Boundary Conditions**.

3. Setup

Table 3. Boundary Conditions

Type	Zones	Values
pressure-outlet	ice-outlet	Gauge Pressure (pascal) -5000
		Backflow Total Temperature (k) 300
pressure-inlet	ice-inlet-inplenum1	Gauge Total Pressure (pascal) 0
		Supersonic/Initial Gauge Pressure (pascal) 0
		Total Temperature (k) 300
wall	wall-ice-fluid-port	Temperature (k) 300
wall	ice-cyl	Temperature (k) 300
wall	ice-slipwall-inplenum1	Temperature (k) 300
wall	ice-slipwall-outplenum	Temperature (k) 300
wall	ice-valve-proximity-faces	Temperature (k) 300

- The table **Models**, shows the models selected for the simulation.

Table 4. Models

Model	Settings
Space	3D
Time	Steady
Viscous	Standard k-omega turbulence model

Table 5. Equations

Equation	Solved
Flow	yes
Turbulence	yes
Energy	yes

Table 6. Relaxation

Variable	Relaxation Factor
Density	1.000
Body Forces	1.000
Turbulent Kinetic Energy	0.750
Specific Dissipation Rate	0.750
Turbulent Viscosity	1.000
Energy	0.750

In the table **Equations** you can see for which equations the simulation has been solved.

The **Relaxations** table displays the under relaxation factors set for the various variables.

- The **Pressure-Velocity Coupling** table displays the settings.

Table 7. Pressure-Velocity Coupling

Parameter	Value
Type	Coupled
Pseudo Transient	yes
Explicit momentum under-relaxation	0.500
Explicit pressure under-relaxation	0.500

Table 8. Discretization Scheme

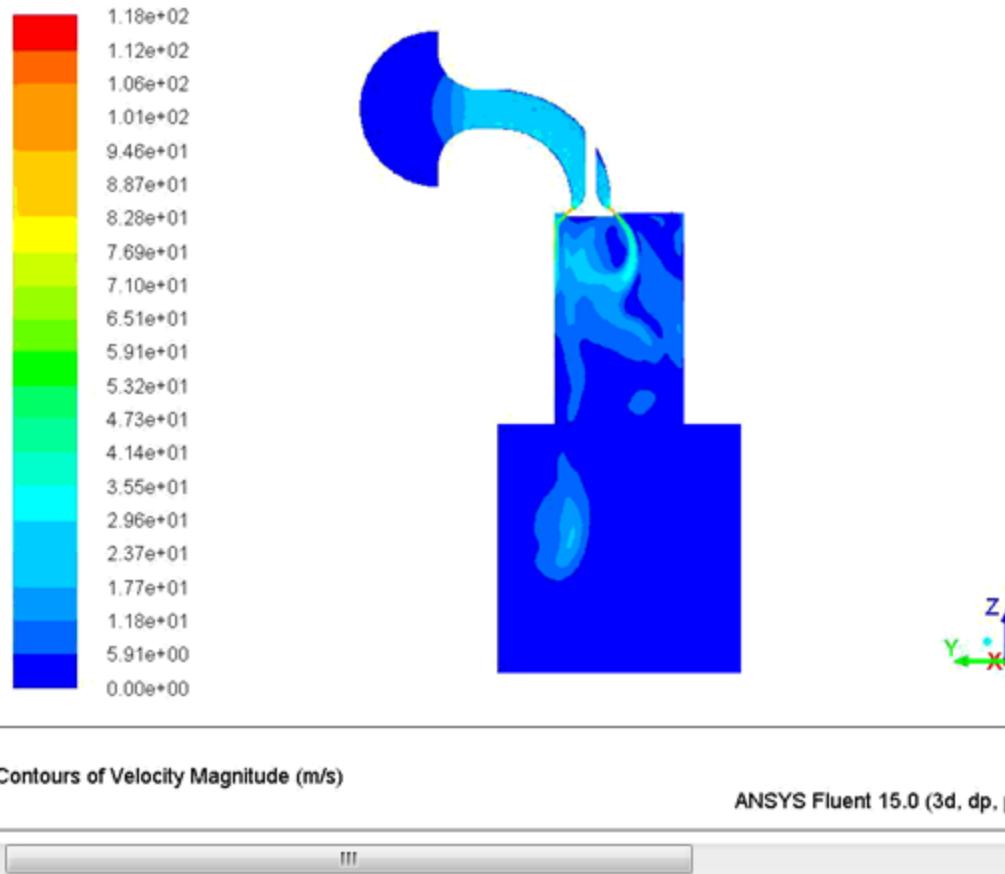
Variable	Scheme
Pressure	Standard
Density	Second Order Upwind
Momentum	Second Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Specific Dissipation Rate	First Order Upwind
Energy	Second Order Upwind

The **Discretization Scheme** table displays the discretization schemes set for the various variables.

- Check the animation of velocity magnitude on the cut-plane in the section **Solution Data**.

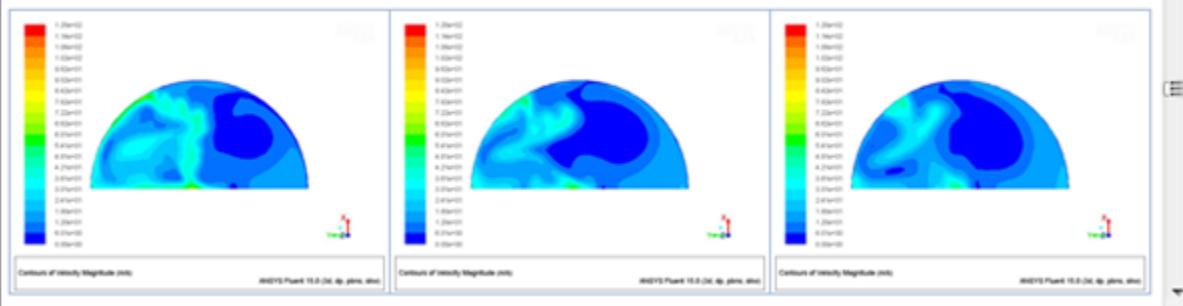
4. Solution Data

4.1. Animation: velocity-magnitude on ice_cutplane_1



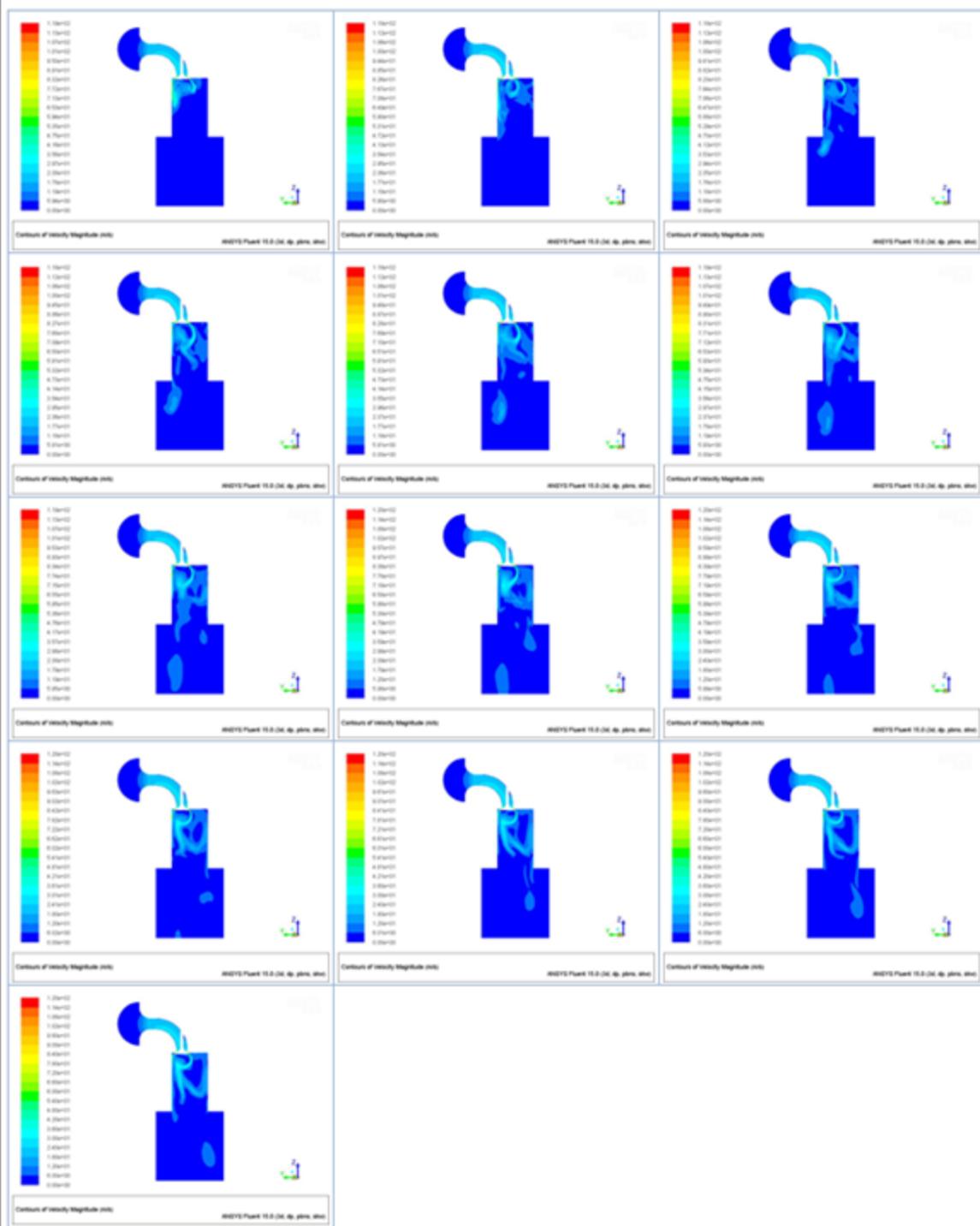
- In a **Table** you can observe the velocity-magnitude contours on the swirl planes which you have created. These images are taken at the end of the simulation.

4.2. Table: velocity-magnitude on ice_swirl_plane_1 ice_swirl_plane_2 ice_swirl_plane_3

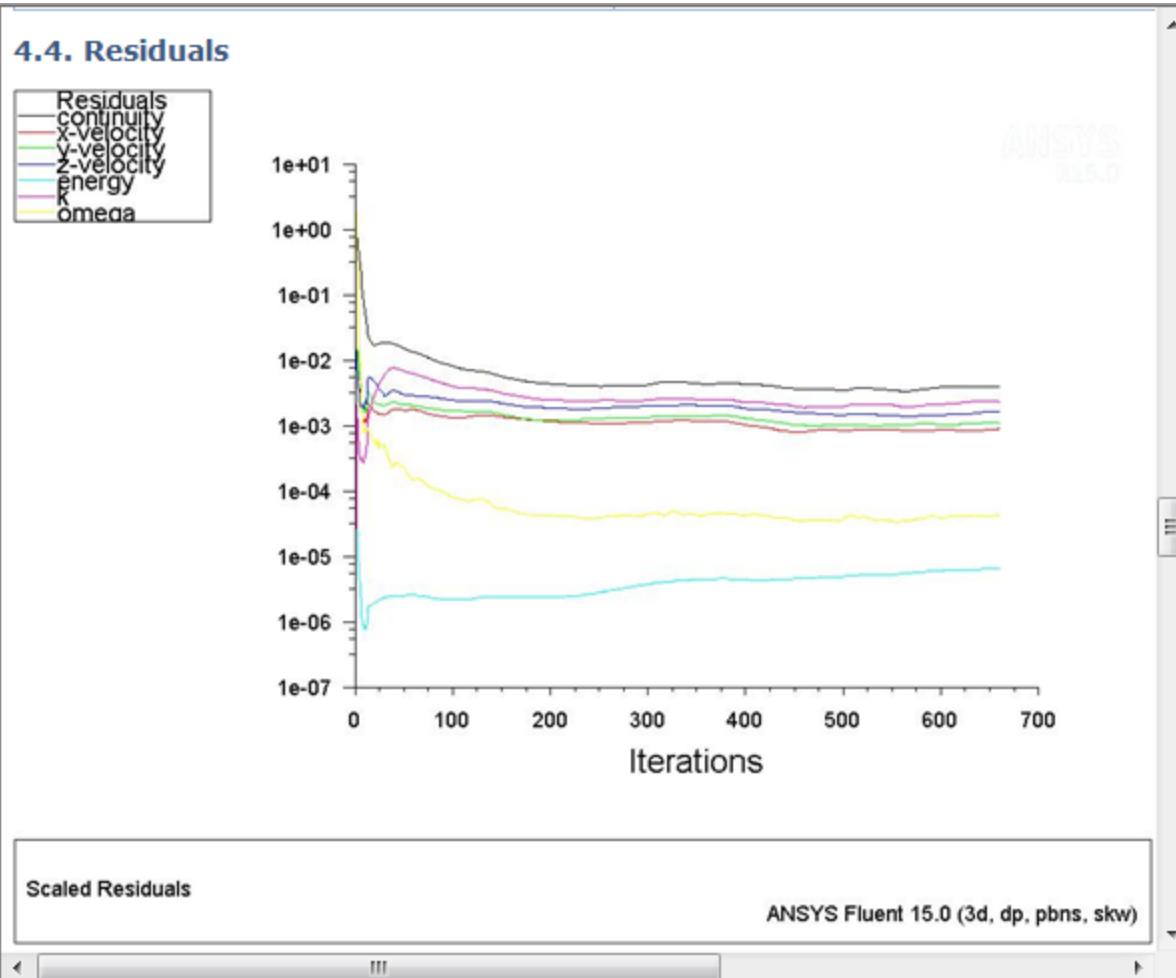


- You can also observe the contours of pressure on the cut-plane taken in intervals, in another table.

4.3. Table: velocity-magnitude on ice_cutplane_1



- Check the residuals.



- Check the mass flow rate surface monitor plot. Also you can check the mass flow rate plots on the swirl planes.

4.5. Charts

Chart 1. Monitor: Mass Flow Rate (ice-inlet-inplenum1 ice-outlet)

Monitor: Mass Flow Rate (ice-inlet-inplenum1 ice-outlet)

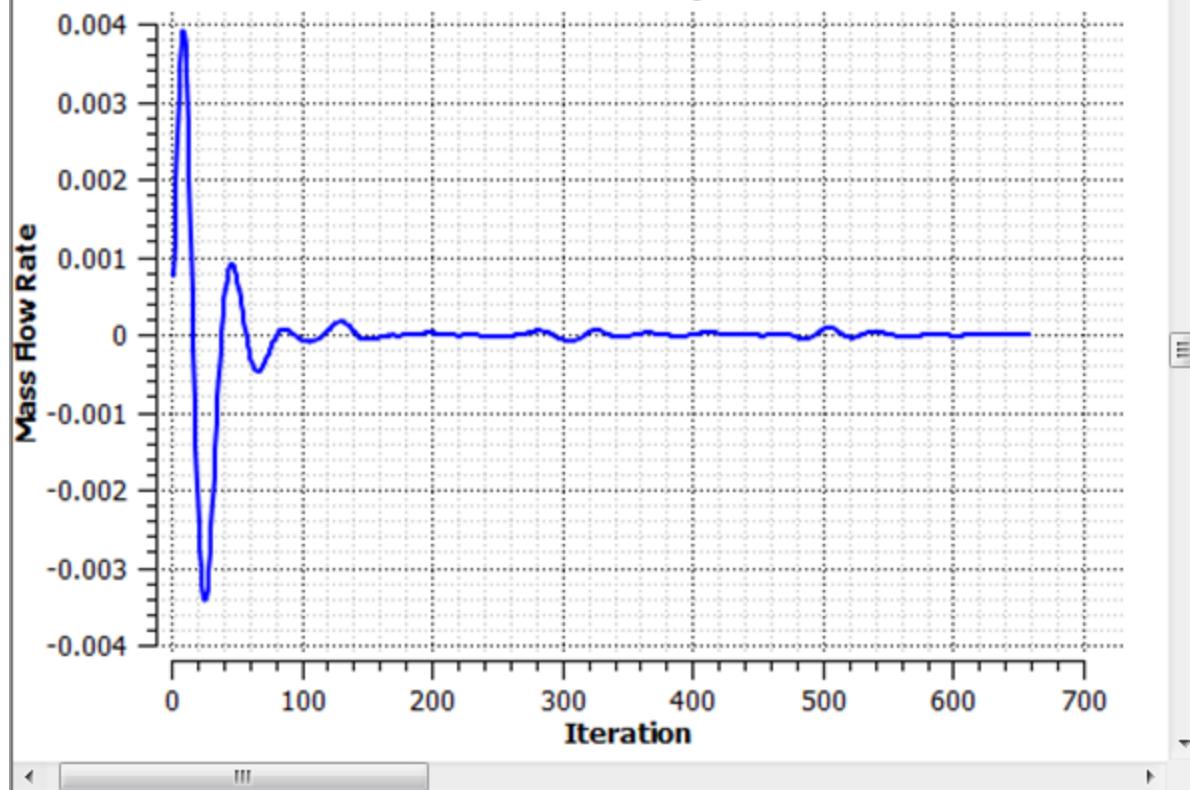


Chart 2. Monitor: Flow Rate swirl1 (ice_swirl_plane_1)

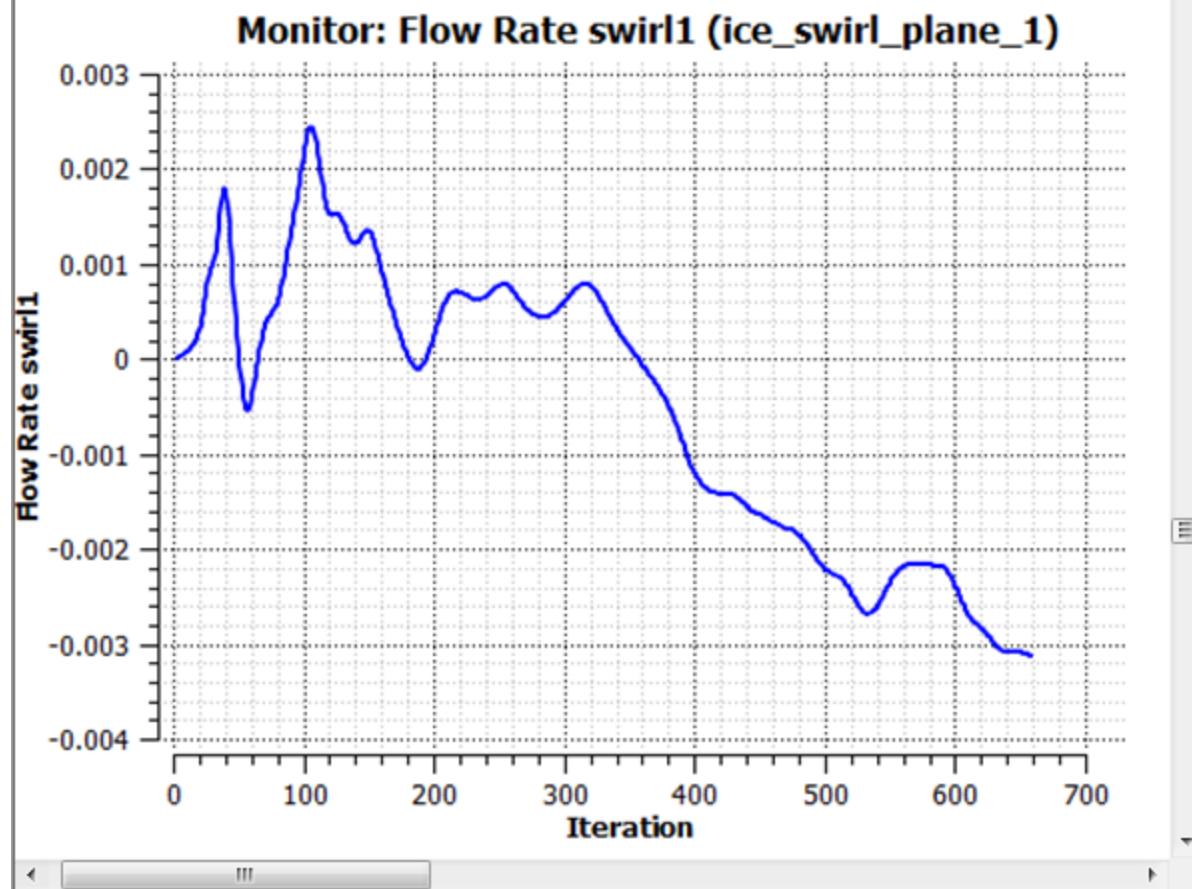


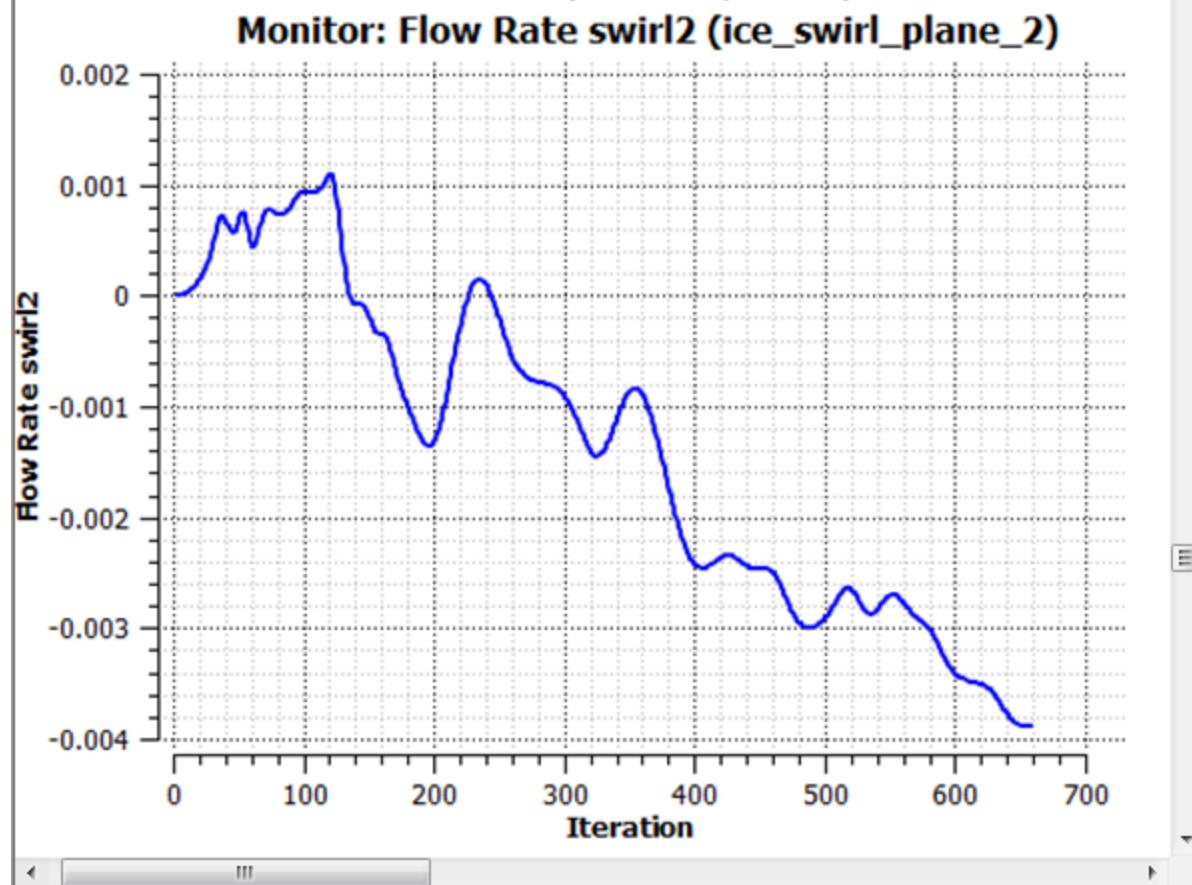
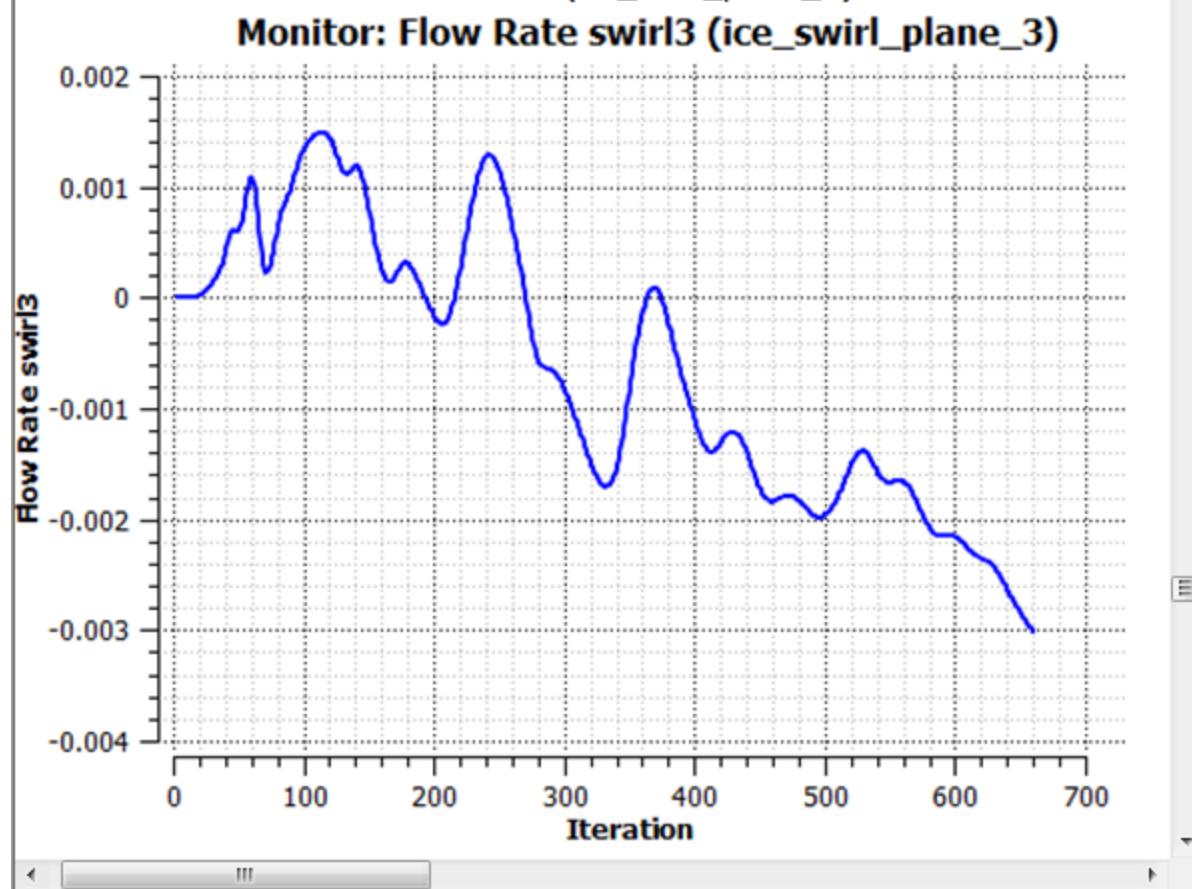
Chart 3. Monitor: Flow Rate swirl2 (ice_swirl_plane_2)

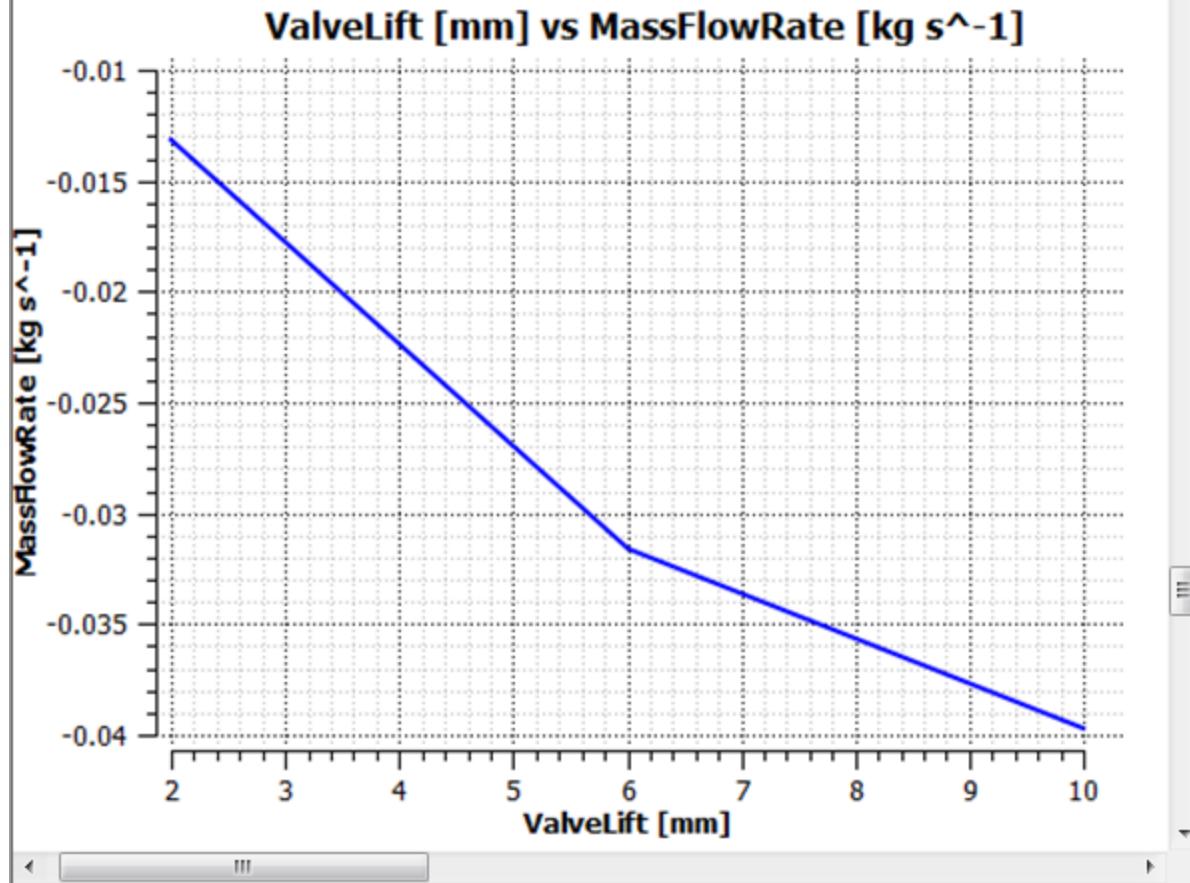
Chart 4. Monitor: Flow Rate swirl3 (ice_swirl_plane_3)

- In a chart under **Design Points report** you can check the values of input parameter against the output parameter.

5. Design Points Report

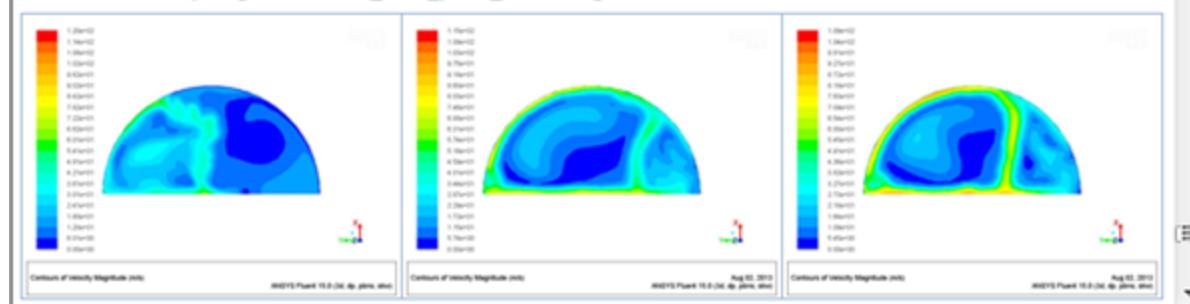
5.1. Design Points Parameter values Charts

Chart 6. ValveLift [mm] vs MassFlowRate [kg s^{-1}]

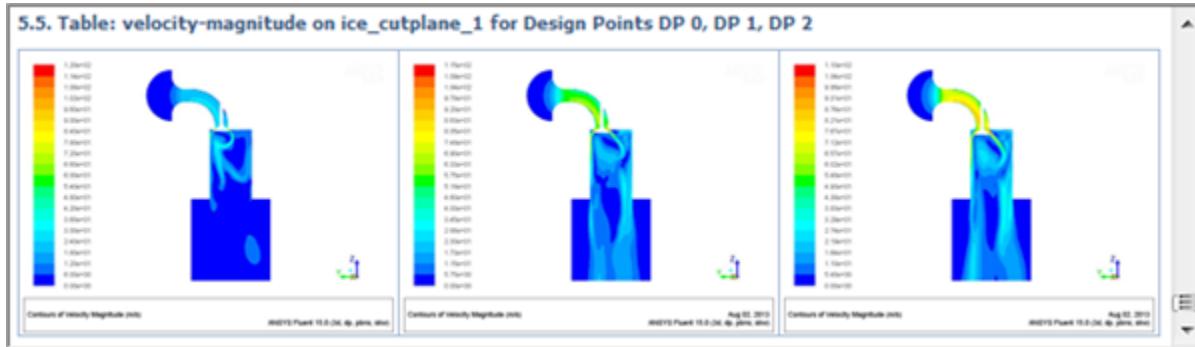


- In different tables you can observe the velocity magnitude contours on the different swirl planes for the design points which you have created. These images are taken at the end of the simulation.

5.2. Table: velocity-magnitude on ice_swirl_plane_1 for Design Points DP 0, DP 1, DP 2



- In another table you can observe the contours of velocity magnitude on the cut-plane for the different design points. These images are taken at the end of the simulation.



This concludes the tutorial which demonstrated the setup and solution for a port flow simulation of an IC engine.

2.8. Summary

In this tutorial, you have learned to set up and solve an IC Engine problem. You have also learned how to use ANSYS Workbench parametric system, which is here used for varying the valve lifts and examining their effect on mass flow rate.

2.9. Further Improvements

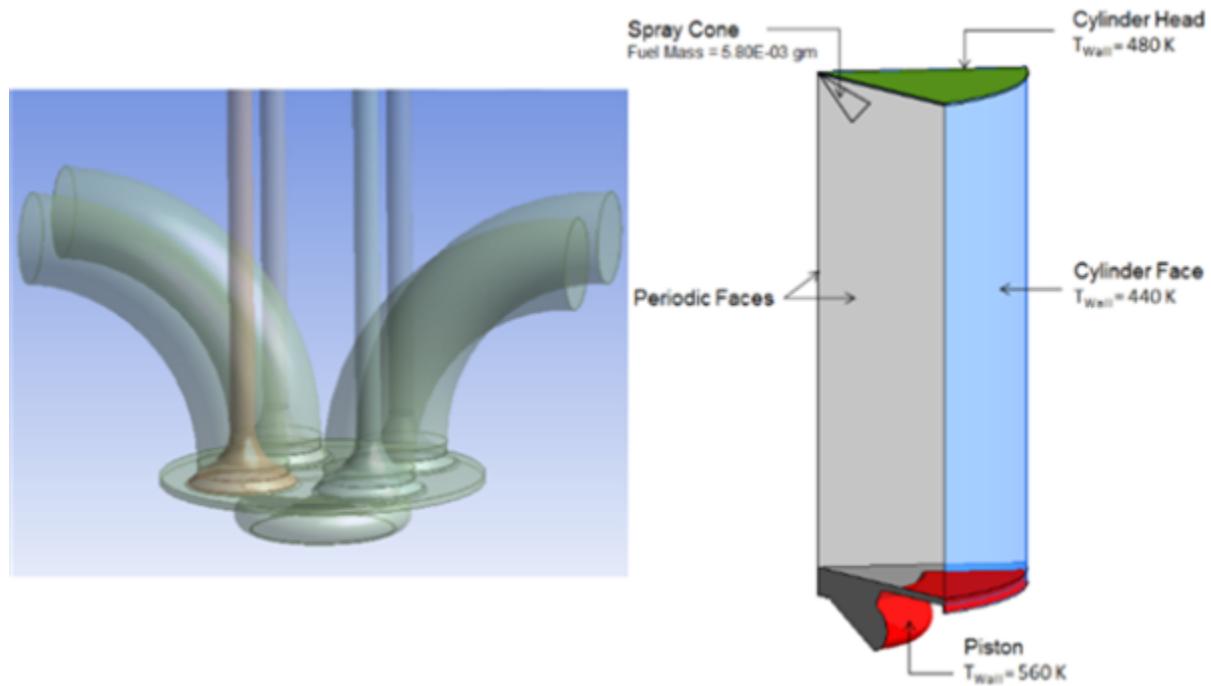
This tutorial presents streamlined workflow between pre-processing, solver and post processing for ease of port flow simulations and performing parametric analysis for varying valve lifts. You may use different meshing strategies for accurate results. You may use parametric system in workbench for analysis of different port angles, valve seat inclinations etc.

Chapter 3: Tutorial: Solving a Combustion Simulation for a Sector

In this tutorial a complete Direct injection (DI) compression ignition (CI) engine geometry is transformed into 60° sector in-order to reduce mesh size and solution time. Detailed boundary conditions are as shown in the [Figure 3.1: Problem Schematic \(p. 107\)](#). Sector simulation is started at intake valve Closing (IVC) with initial conditions as 3.45 bar and 404 K, species mass fraction of O₂=0.1369, N₂=0.7473, CO₂=0.0789, H₂O=0.0369. n-heptane (nc7h16) is used as surrogate for diesel fuel and is injected 8 degrees before compression (Top Dead Centre). Engine rpm is increased from 1500 rpm to 2000 rpm and its effect on unburnt fuel is examined. This tutorial illustrates the following steps in setting up and solving a combustion simulation for a sector.

- Launch IC Engine system.
- Read an existing geometry into IC Engine.
- Decompose the geometry.
- Define mesh setup and mesh the geometry.
- Define the solver setup.
- Run the simulation.
- Examine the results in the report.

Figure 3.1: Problem Schematic



This tutorial is written with the assumption that you are familiar with the IC Engine system and that you have a good working knowledge of ANSYS Workbench.

- 3.1. Preparation
- 3.2. Step 1: Setting the Properties
- 3.3. Step 2: Performing the Decomposition
- 3.4. Step 3: Meshing
- 3.5. Step 4: Setting up the Simulation
- 3.6. Step 5: Running the Solution
- 3.7. Step 6: Obtaining the Results
- 3.8. Summary
- 3.9. Further Improvements

3.1. Preparation

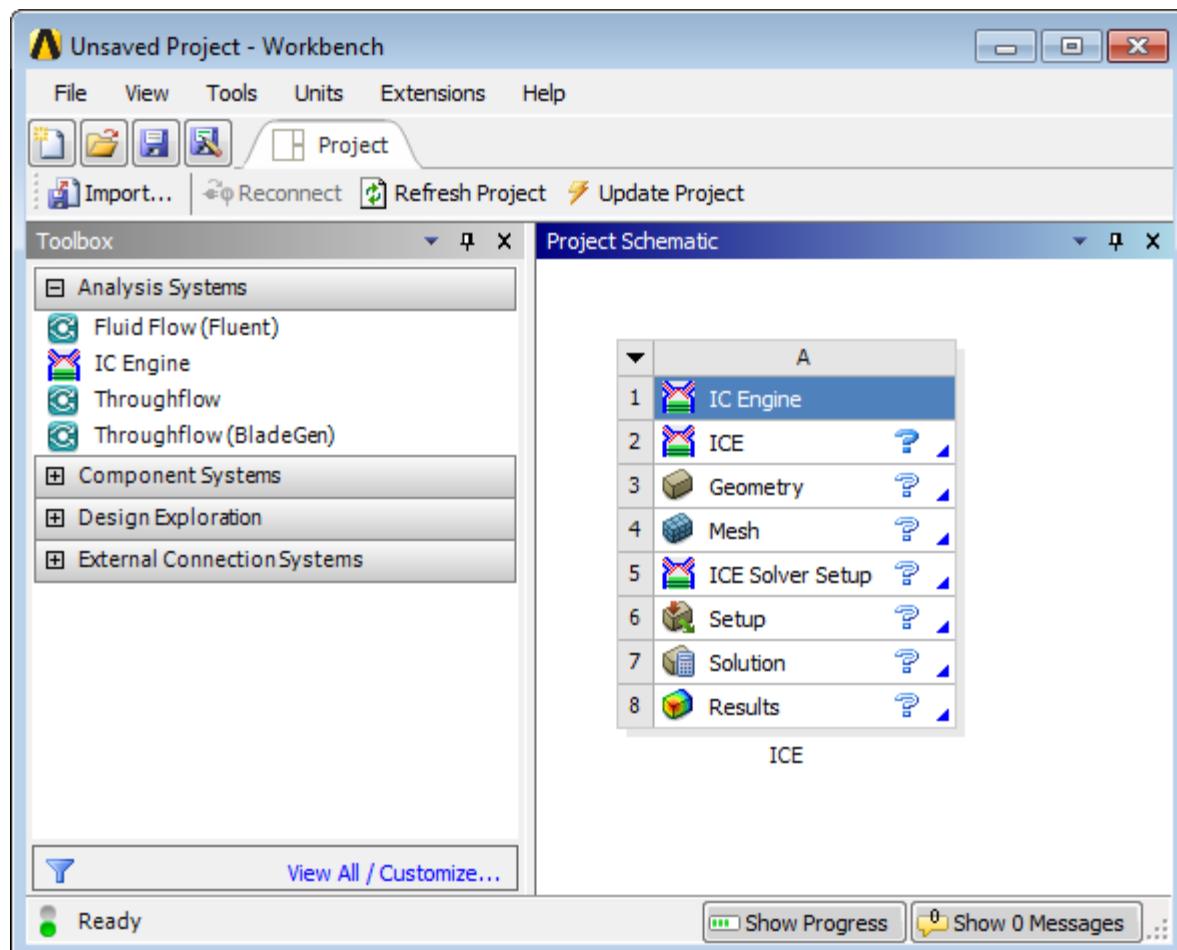
1. Copy the files (`tut_comb_sect.x_t`, `injection-profile`, `Diesel_1comp_35sp_chem.inp`, and `Diesel_1comp_35sp_therm.dat`) to your working folder.

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

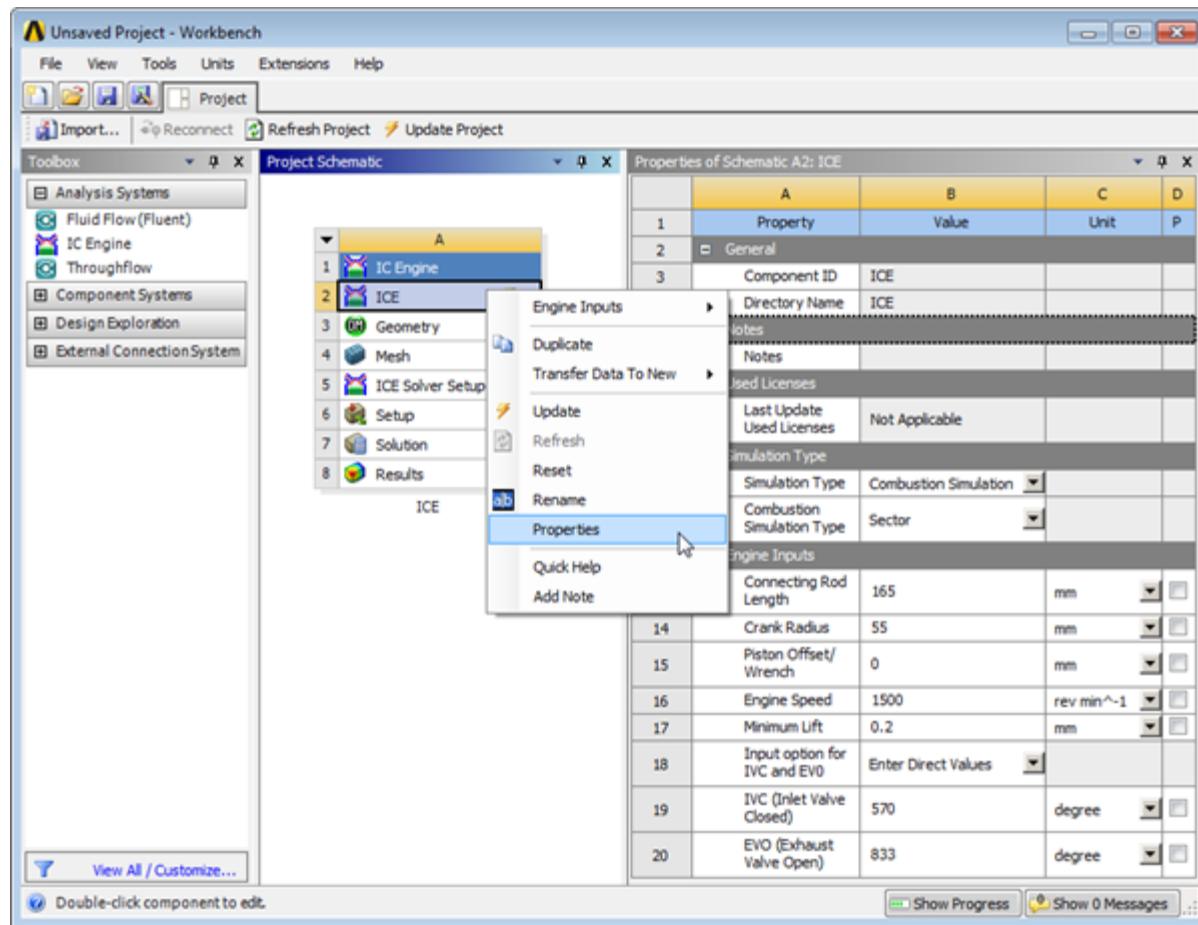
2. Start Workbench.

3.2. Step 1: Setting the Properties

1. Create an IC Engine analysis system in the Workbench interface by dragging or double-clicking on **IC Engine** under **Analysis Systems** in the **Toolbox**.



- Right-click on **ICE**, cell 2, and click **Properties** (if it is not already visible) from the context menu.



3. Select **Combustion Simulation** from the **Simulation Type** drop-down list.
4. Select **Sector** from the **Combustion Simulation Type** drop-down list.
5. Enter 165 for **Connecting Rod Length**.
6. Enter 55 for **Crank Radius**.
7. From the **Input option for IVC and EVO** drop-down list select **Enter Direct Values**.
8. Enter 570 for **IVC (Inlet Valve Closed)**.

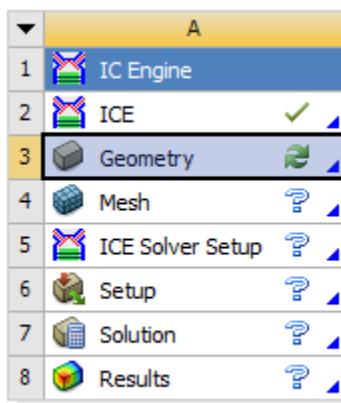
Note

This will be set as the decomposition crank angle.

9. Enter 833 for **EVO (Exhaust Valve Open)**.
10. Right-click **ICE** cell and select **Update** from the context menu.

3.3. Step 2: Performing the Decomposition

Here you will read the geometry and prepare it for decomposition. Double-click on the **Geometry** cell to open the DesignModeler.

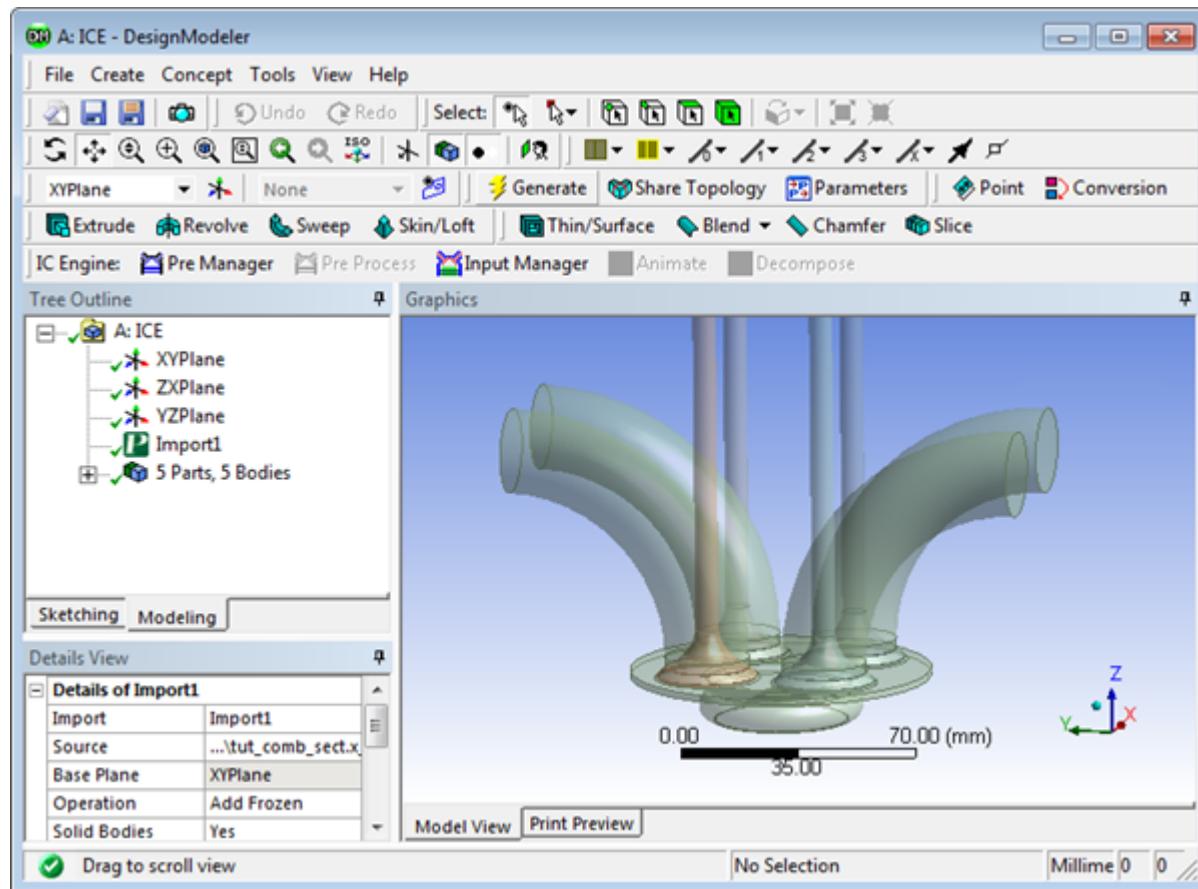


ICE

1. Select **Millimeter** from the **Units** menu.
2. Import the geometry file, `tut_comb_sect.x_t`.

File >Import External Geometry File...

3. Click **Generate** to complete the import feature.



4. Click **Input Manager** located in the **IC Engine** toolbar.

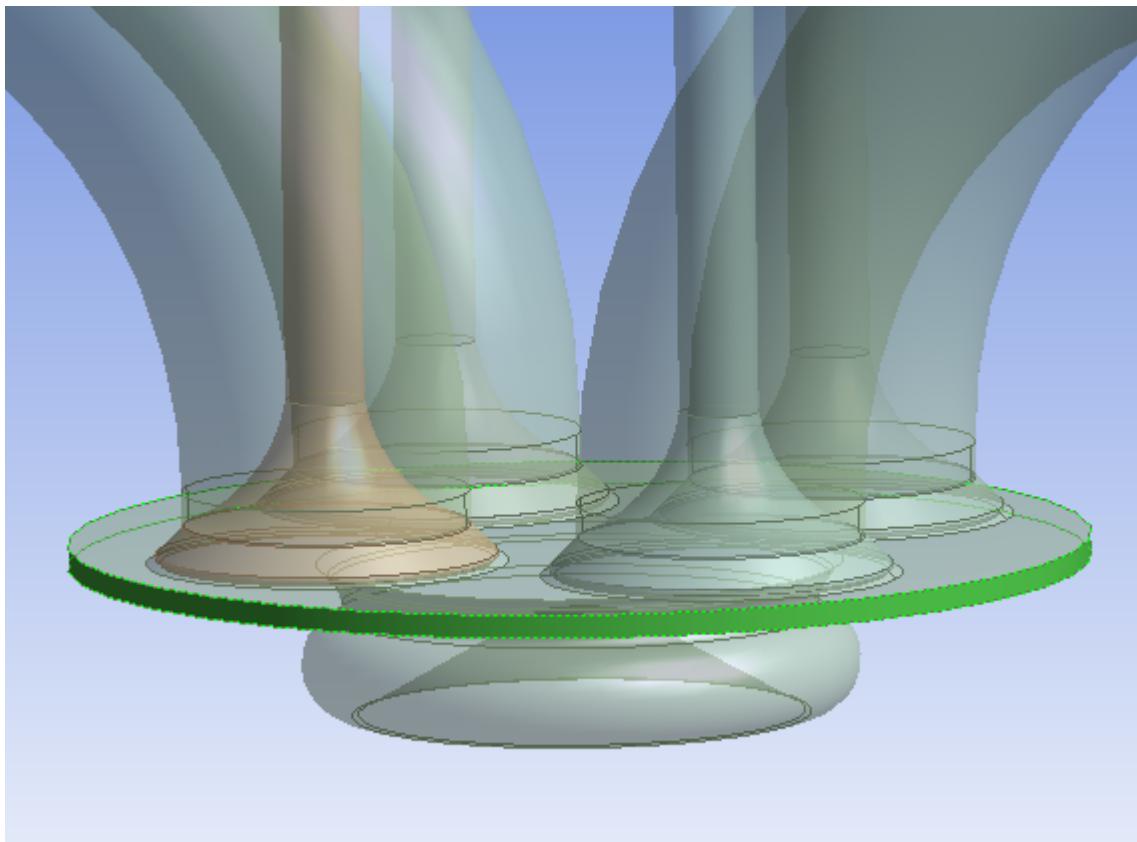
Details View	
Details of InputManager1	
Name	InputManager1
Decomposition Position	IVC
Decomposition Angle	Click on Generate
Sector Decomposition Type	Complete Geometry
Cylinder Faces	Not selected
Sector Angle	60 °
Validate Compression Ratio	No
Crevice H/T Ratio	3
Spark Points	Not selected
IC Valves Data 1 (RMB)	
Valve Bodies	Not selected
Valve Seat Faces	Not selected
IC Injection 1 (RMB)	
Spray Location Option	Height and Radius
Spray Location, Height	0 m
Spray Location, Radius	0 m
Spray Direction Option	Spray Angle
Spray Angle	0 °
IC Advanced Options (RMB)	

- a. Retain the selection **IVC** for **Decomposition Position**.

Note

The inlet valve closing (IVC) angle is chosen as the geometry decomposition angle, since for combustion simulation you are more interested in the power stroke of the engine cycle, starting from closing of valves to the end of the compression stroke.

- b. Retain **Complete Geometry** from the **Sector Decomposition Type** drop-down list as the input geometry you have chosen is a complete geometry.
- c. Select the face as shown in [Figure 3.2: Cylinder Face \(p. 113\)](#) for **Cylinder Faces** and click **Apply**.

Figure 3.2: Cylinder Face

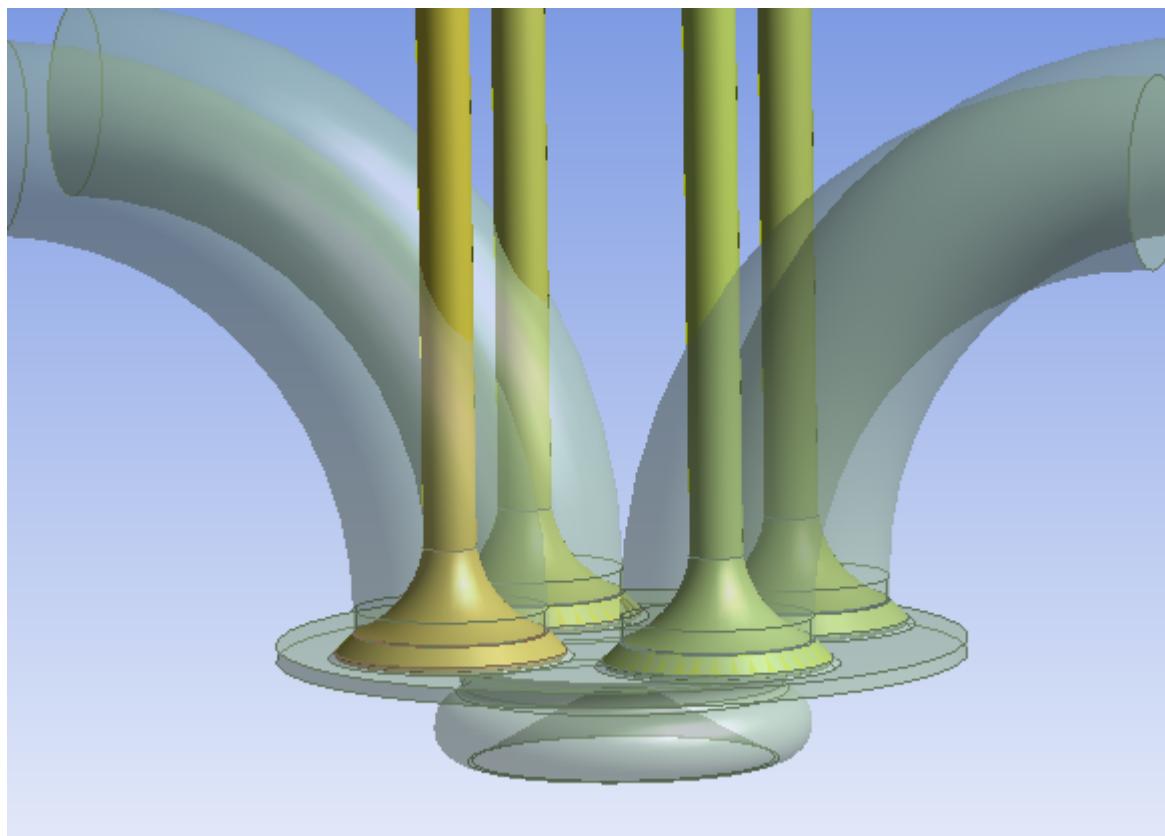
- d. Retain **60 °** for **Sector Angle** drop-down list.

Note

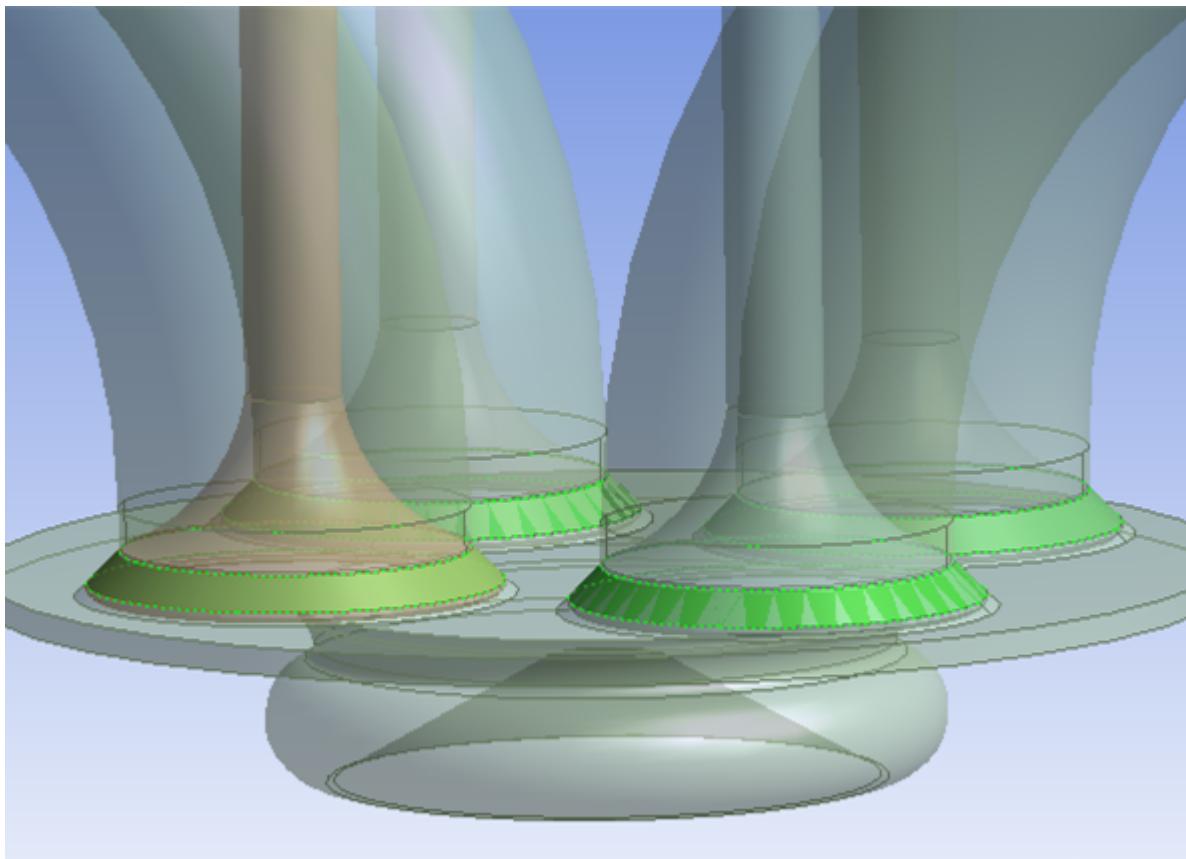
This will slice the geometry into a sector of 60 °.

- e. Select **Yes** for **Validate Compression Ratio**.
- f. Enter **15 . 83** for **Compression Ratio**.
- g. Retain the default setting **3** for **Crevice H/T Ratio**.
- h. Select the valve bodies as shown in [Figure 3.3: Valves \(p. 114\)](#) for **Valve** and click **Apply**.

Figure 3.3: Valves



- i. Select the valve seat faces as shown in [Figure 3.4: Valve Seats \(p. 115\)](#) for **Valve Seat** and click **Apply**.

Figure 3.4: Valve Seats

- j. Retain the selection of **Height and Radius** for **Spray Location Option** under **IC Injection 1**.
- k. Enter **0 . 3 mm** for **Spray Location, Height**.
- l. Enter **0 . 5 mm** for **Spray Location, Radius**.

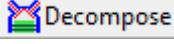
Note

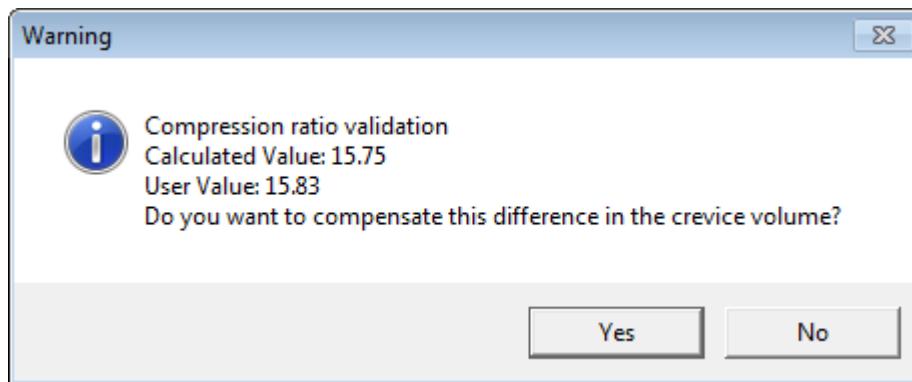
Depending upon the height and radius the spray location is calculated..

- m. Enter **70°** for **Spray Angle**.

- n. After all the settings are done click **Generate**  **Generate**

Details View	
Details of InputManager1	
Name	InputManager1
Decomposition Position	IVC
Decomposition Angle	Click on Generate
Sector Decomposition Type	Complete Geometry
Cylinder Faces	1 Face
Sector Angle	60 °
Validate Compression Ratio	Yes
Compression Ratio	15.83
Crevice H/T Ratio	3
Spark Points	Not selected
IC Valves Data 1 (RMB)	
Valve Bodies	4 Bodies
Valve Seat Faces	4 Faces
IC Injection 1 (RMB)	
Spray Location Option	Height and Radius
Spray Location, Height	0.3 mm
Spray Location, Radius	0.5 mm
Spray Direction Option	Spray Angle
Spray Angle	70 °
IC Advanced Options (RMB)	

5. Click **Decompose** ( located in the **IC Engine** toolbar).
6. During decomposition a warning pops up asking if you would like to compensate for the difference in compression ratio.



Click **Yes**.

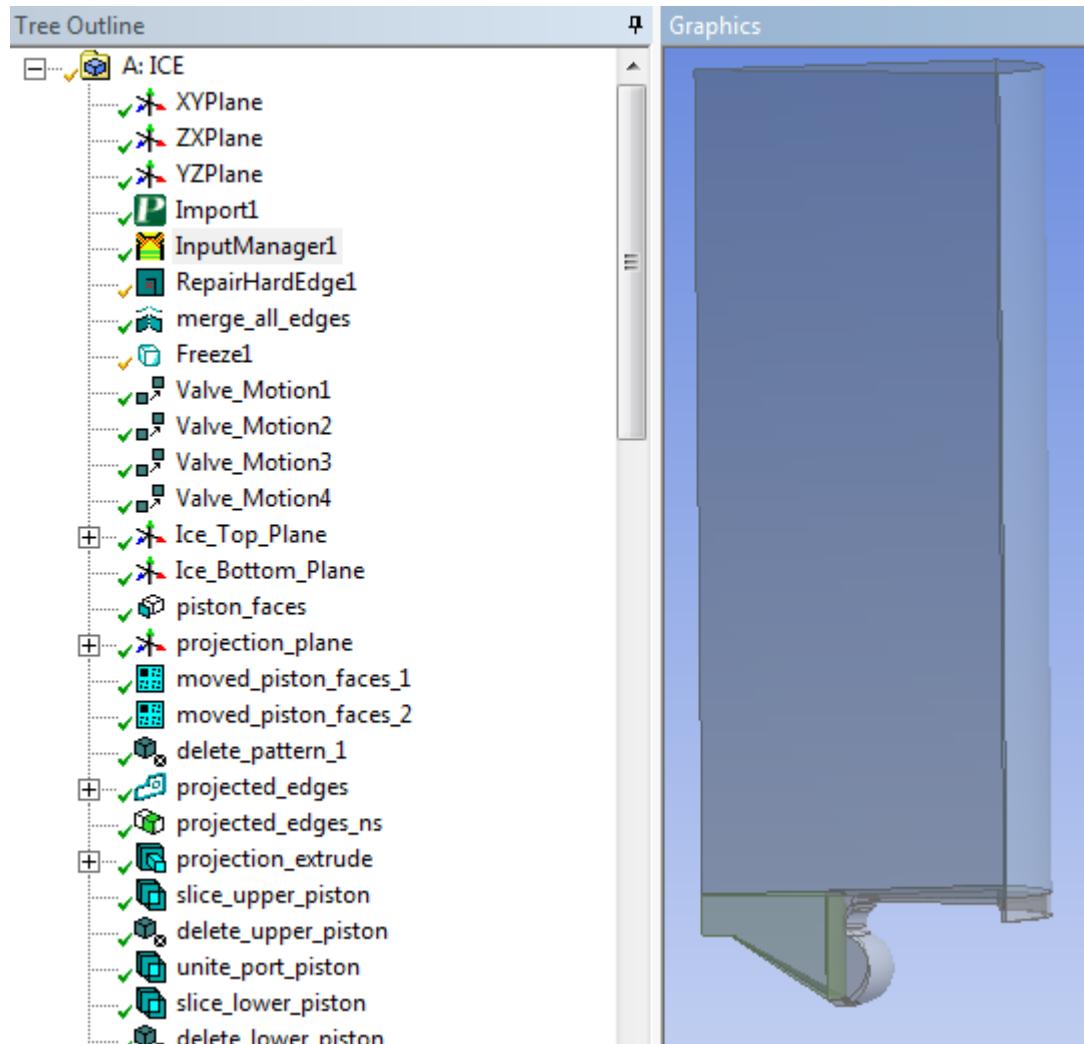
Note

The decomposition process will take a few minutes. During decomposition the following changes take place:

1. The engine port is divided into a sector of the given **Sector Angle**.
2. The valves are removed.

3. The clearance volume is formed into a crevice.
4. The compression ratio difference is adjusted in the crevice.
5. The piston is moved to the appropriate position as per the **Decomposition Crank Angle**.

Figure 3.5: Decomposed Geometry



7. Close the DesignModeler.
8. Save the project by giving it a proper name (`demo_sector.wbpj`).

File >Save

3.4. Step 3: Meshing

Here you will mesh the decomposed geometry.

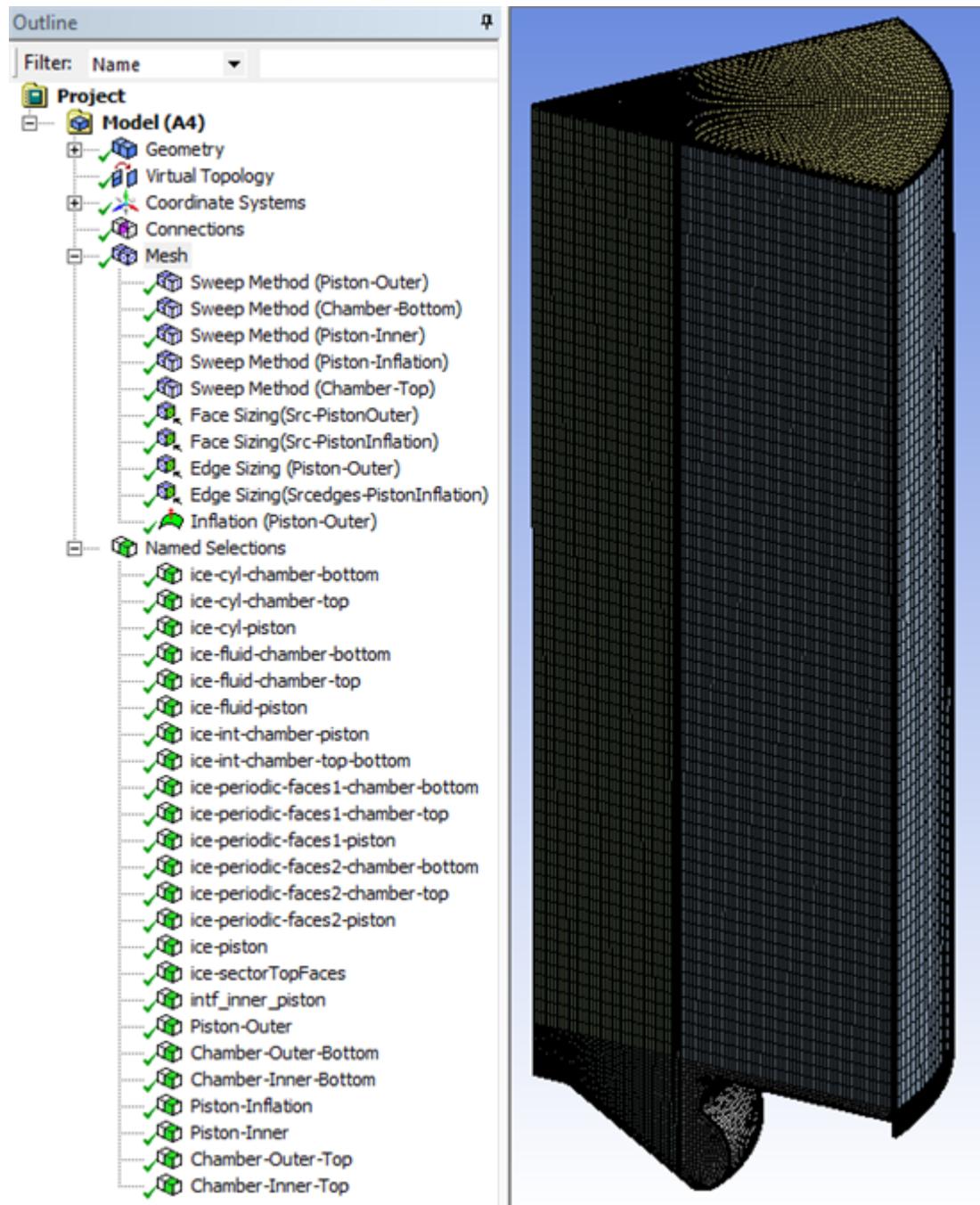
1. Right-click on the **Mesh** cell in the IC Engine analysis system and select **Update** from the context menu.

Note

This meshing process will take a few minutes.

2. You can double-click the **Mesh** cell to check the mesh. See [Figure 3.6: Meshed Geometry \(p. 118\)](#)

Figure 3.6: Meshed Geometry



3. Save the project.

File >Save

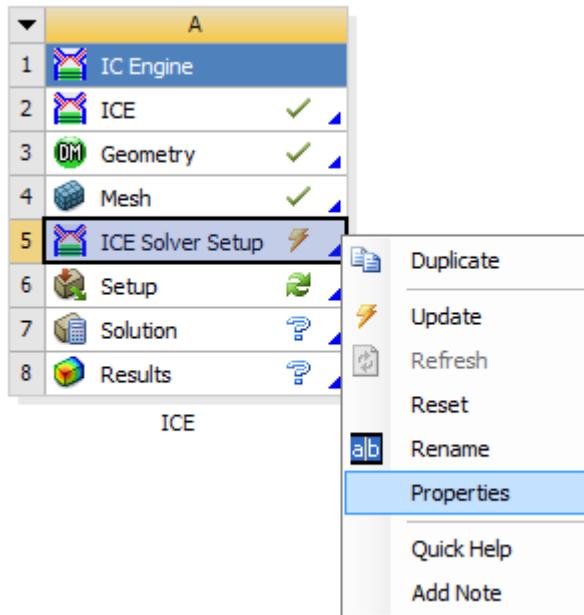
Note

It is a good practice to save the project after each cell update.

3.5. Step 4: Setting up the Simulation

After the decomposed geometry is meshed properly you can set boundary conditions, monitors, and postprocessing images. You can also decide which data and images should be included in the report.

1. If the **Properties** view is not already visible, right-click **ICE Solver Setup**, cell 5, and select **Properties** from the context menu.



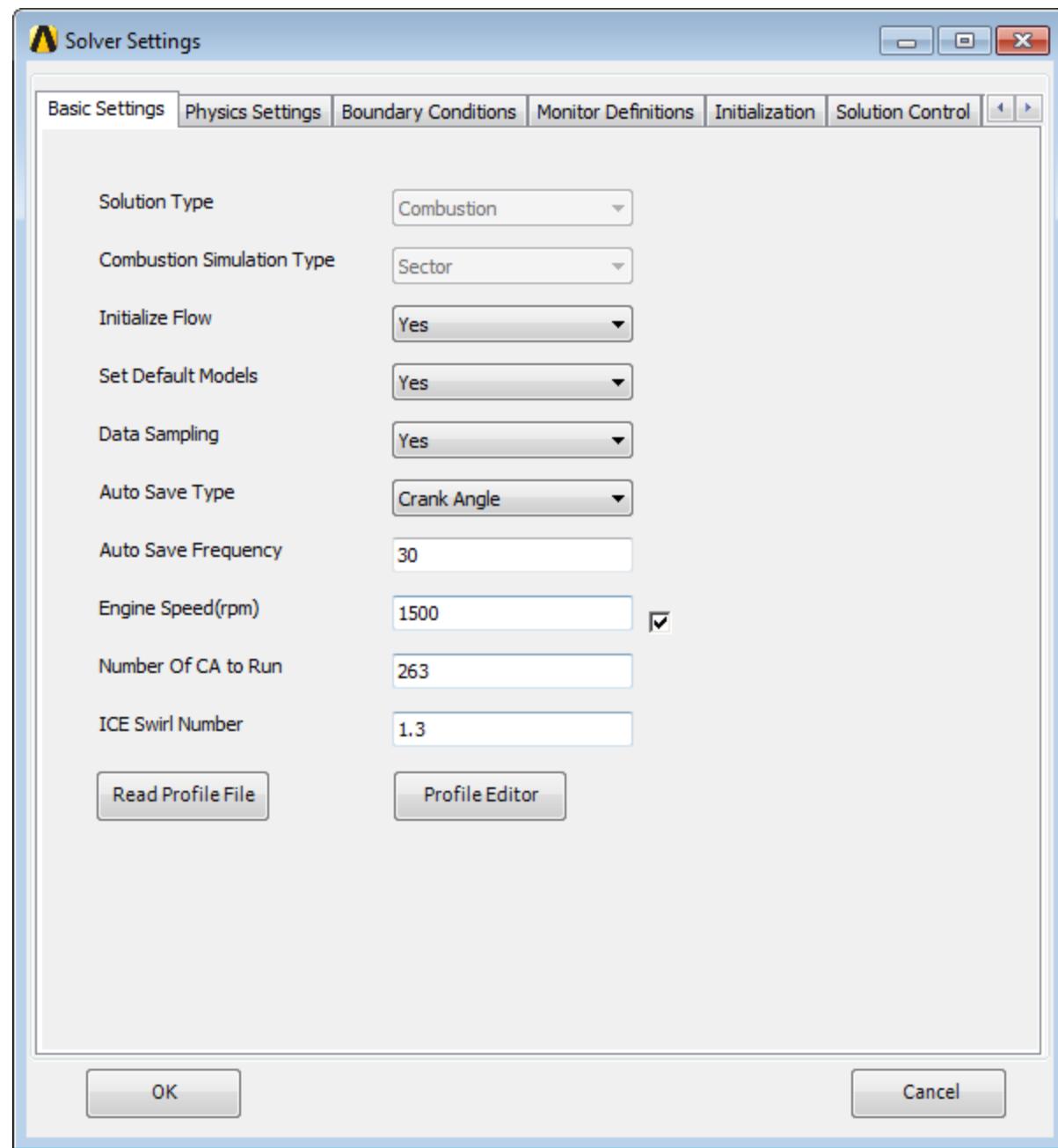
2. Click **Edit Solver Settings** to open the **Solver Settings** dialog box.

Properties of Schematic A5: ICE Solver Setup		
	A	B
1	Property	Value
2	General	
3	Component ID	ICE Solver Setup
4	Directory Name	ICE
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	KeyGrid	
10	KeyGrid	No
11	Solver Settings	
12	Solver Settings	Edit Solver Settings
13	Journal Customization	
14	User Boundary Condition Profiles	
15	User Boundary Conditions and Monitor Settings	ICE\ICE\icUserSettings.txt
16	Pre Iteration Journal	
17	Post Iteration Journal	

Note

In the **Solver Settings** dialog box you can check the default settings in the various tabs. If required you can change the settings.

- a. Click the **Basic Settings** tab.
- i. Enter 1500 for **Engine Speed**.



Enable the check box next to **Engine Speed**. This will add an engine speed input parameter.

- ii. The **Number of CA to Run** is set to **263**.

Note

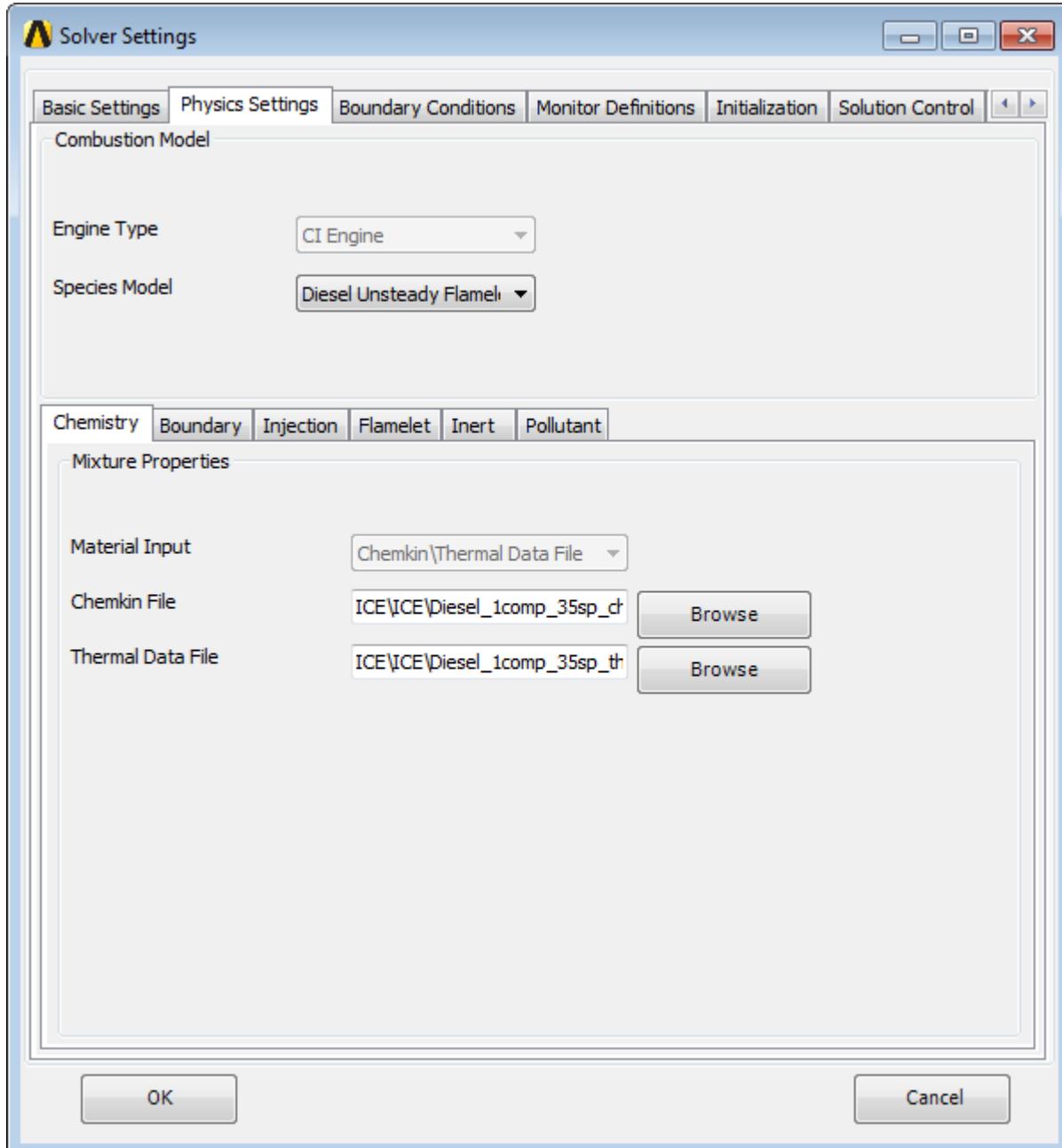
You have entered the IVC and EVO as 570 and 833. You are interested only in the compression and power stroke. So the **Number of CA to Run** is automatically calculated from these values.

- iii. Click **Browse** next to **Profile File** and select `injection-profile.prof` in the **Select Profile File** dialog box.

Note

You will be using this file to set the **Total Flow Rate** and **Velocity Magnitude** in the **Injection Properties** dialog box.

- b. In the **Physics Settings** tab select **Diesel Unsteady Flamelet** from the **Species Model** drop-down list in the **Combustion Model** group box.

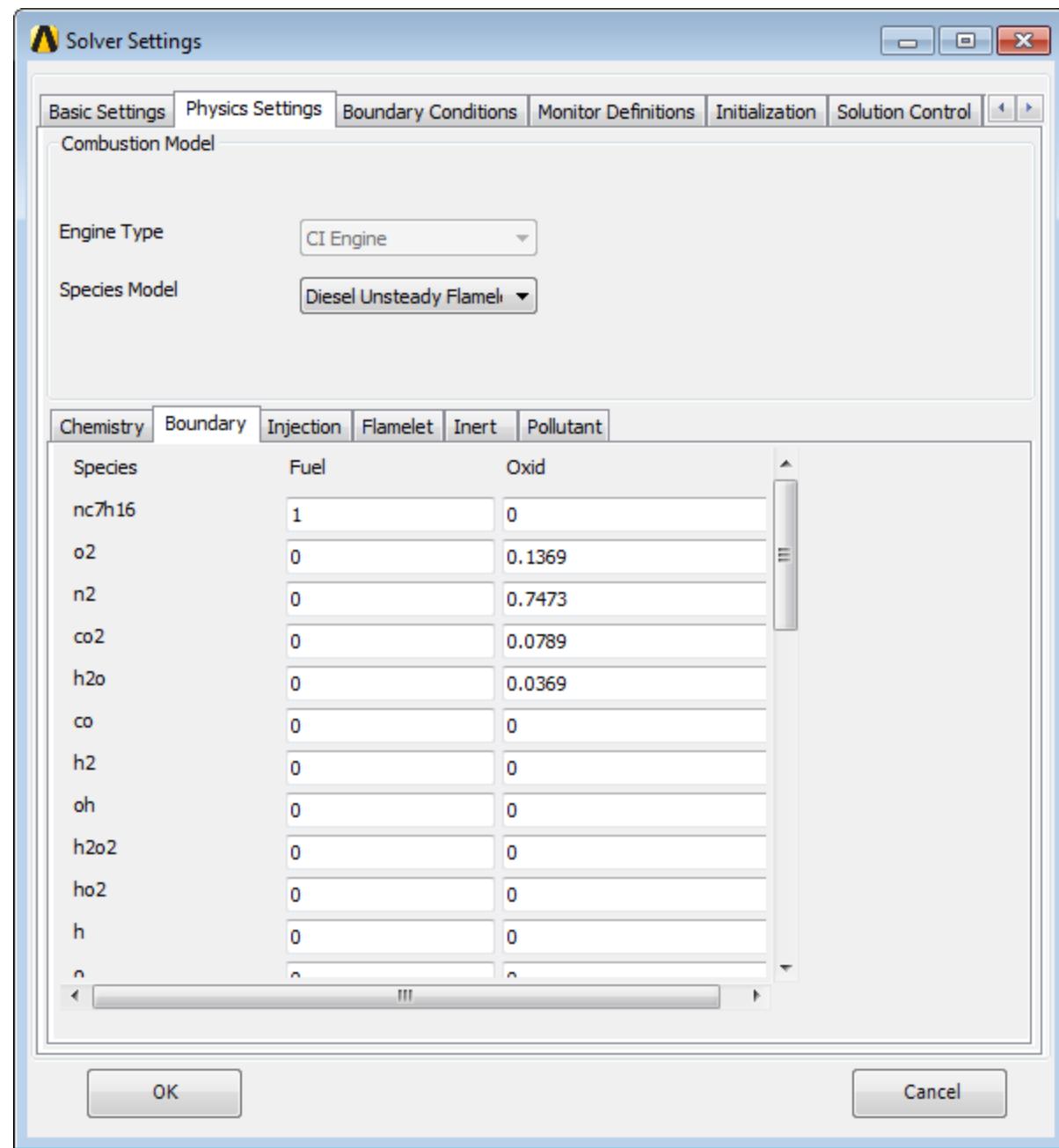


- c. In the **Physics Settings** tab click **Chemistry** tab.

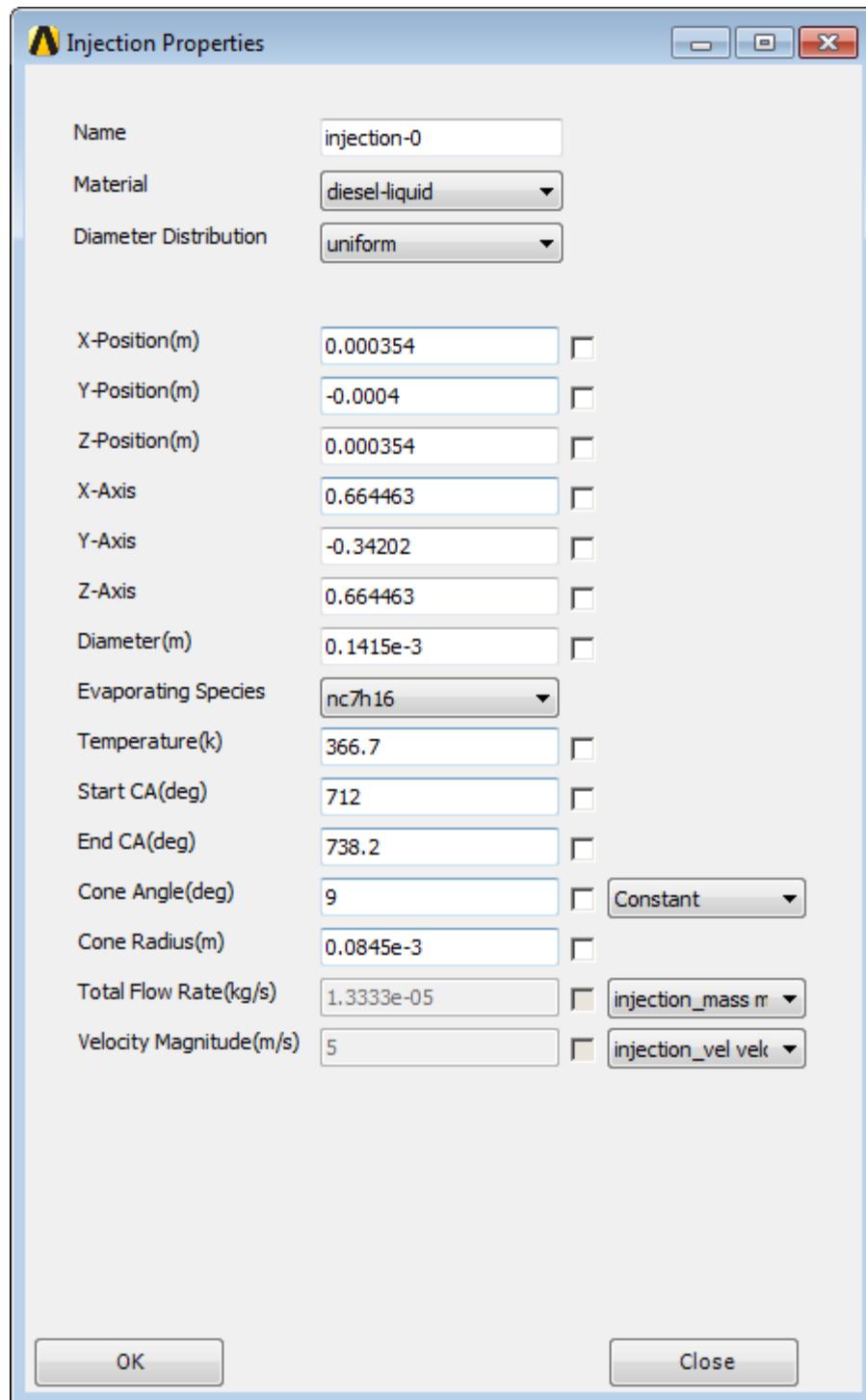
- i. Click **Browse** next to **Chemkin File** and select the file Diesel_1comp_35sp_chem.inp from your working folder.
 - ii. Similarly select the file Diesel_1comp_35sp_therm.dat for **Thermal Data File** from your working folder.
- d. In the **Physics Settings** tab click **Boundary** tab.
- Enter the values shown in [Table 3.1: Species Composition \(p. 123\)](#) for the **Oxid** values for the listed **Species**.

Table 3.1: Species Composition

Species	Oxid
o2	0.1369
n2	0.7473
co2	0.0789
h2o	0.0369

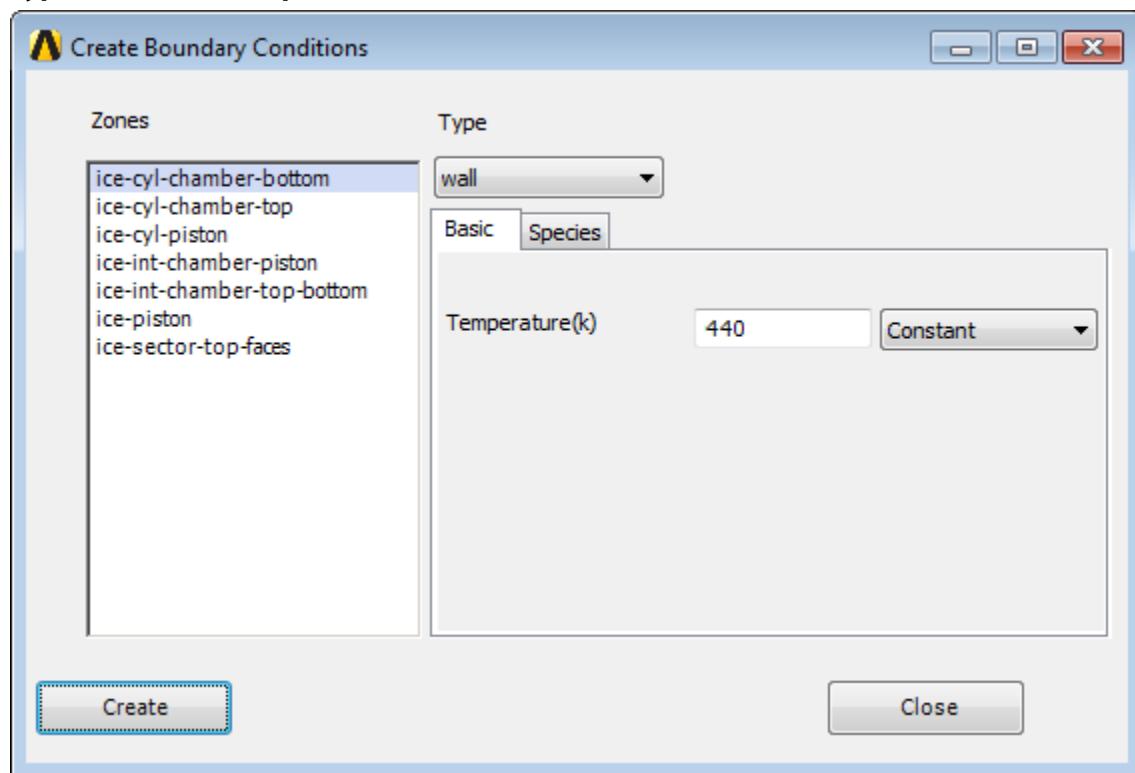


- e. In the **Physics Settings** tab click **Injection** tab. Select **injection-0** from the list and click **Edit** to open the **Injection Properties** dialog box.

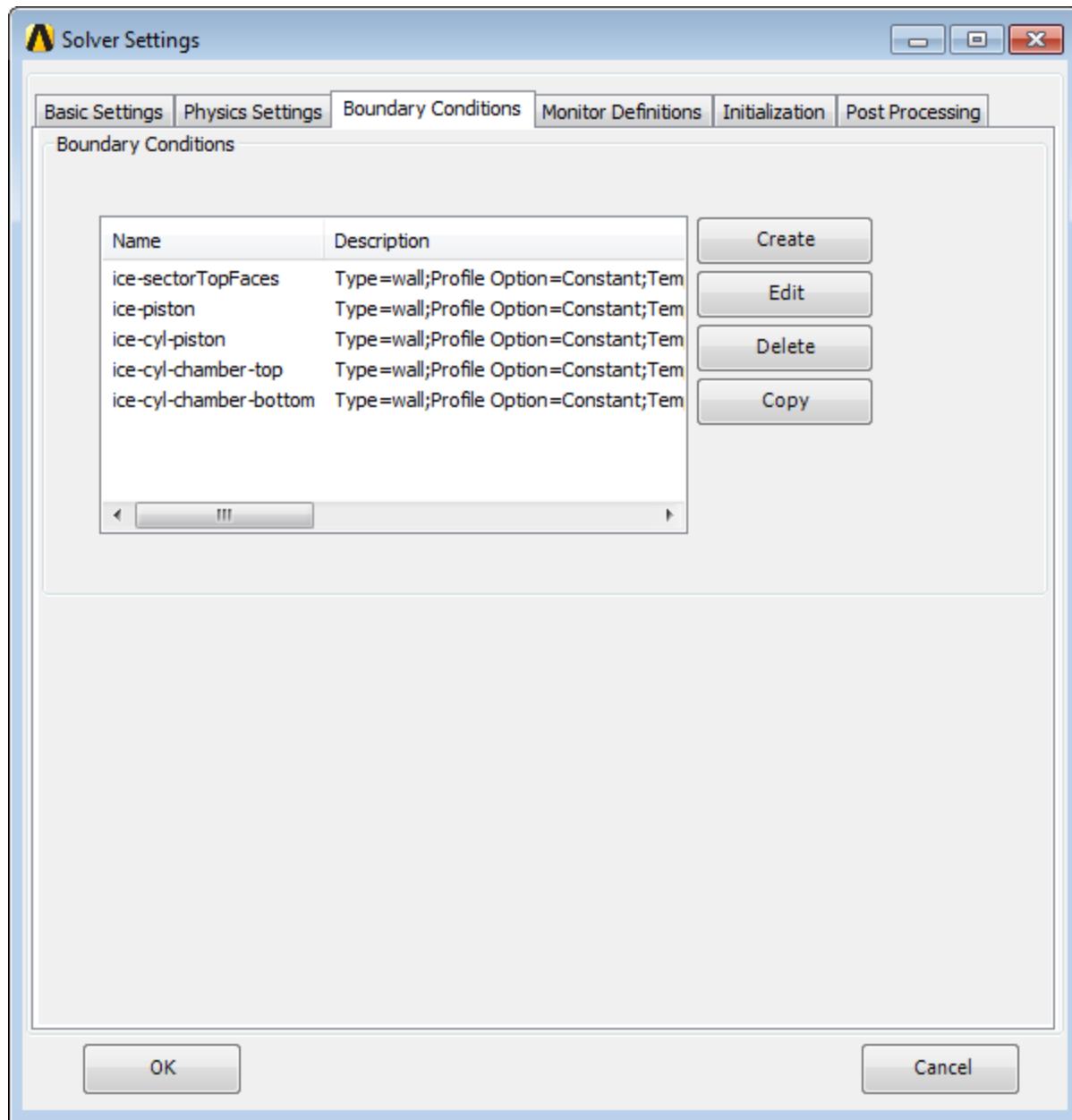


- i. Select **uniform** from the **Diameter Distribution** drop-down list.
- ii. Enter $0.1415e-3$ for **Diameter**.
- iii. Enter 712 for **Start CA**.
- iv. Enter 738.2 for **End CA**.
- v. Enter $0.0845e-3$ for **Cone Radius**.

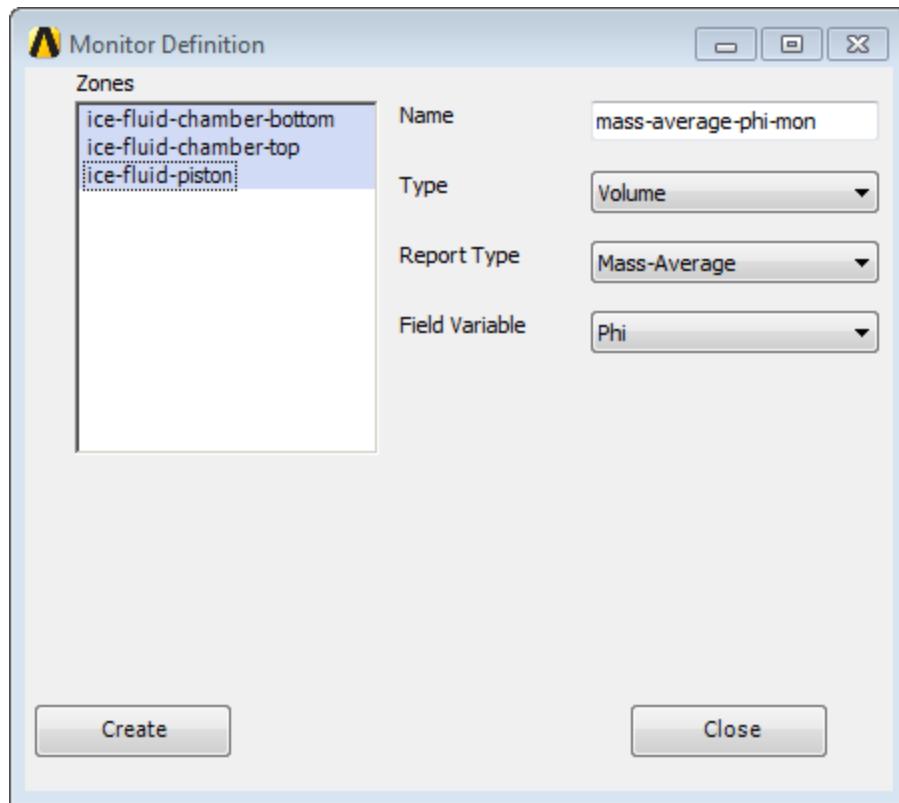
- vi. Click the **Constant** button next to **Total Flow Rate** and select **injection_mass massflowrate** from the drop-down menu.
 - vii. Click the **Constant** button next to **Velocity Magnitude** and select **injection_vel velocity** from the drop-down menu.
 - viii. Click **OK** to apply the settings.
- f. In the **Boundary Conditions** tab you can see that no default settings are present. For this tutorial you will create some wall boundary conditions. Click **Create** to open the **Create Boundary Conditions** dialog box.
- i. Select **ice-cyl-chamber-bottom** from the **Zones** list. Retain the default selection of **wall** as the **Type** and set the **Temperature** to 440. Click **Create**.



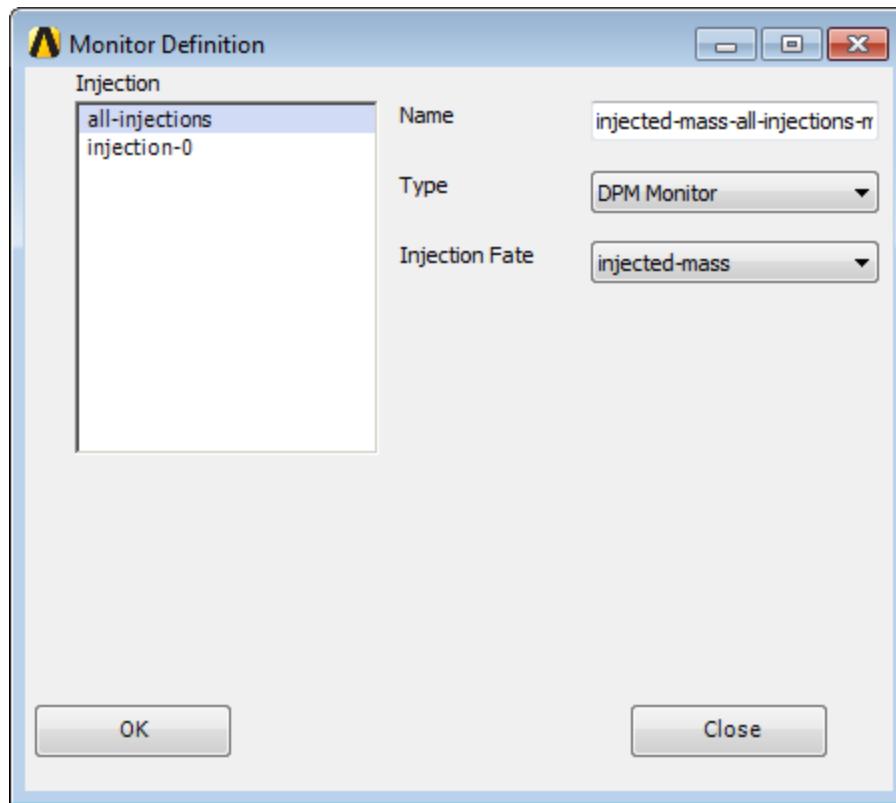
- ii. Make same settings for zones **ice-cyl-chamber-top** and **ice-cyl-piston**.
- iii. For zone **ice-piston** set the **Temperature** to 560.
- iv. Similarly for **ice-sector-top-faces** set the **Temperature** to 480.



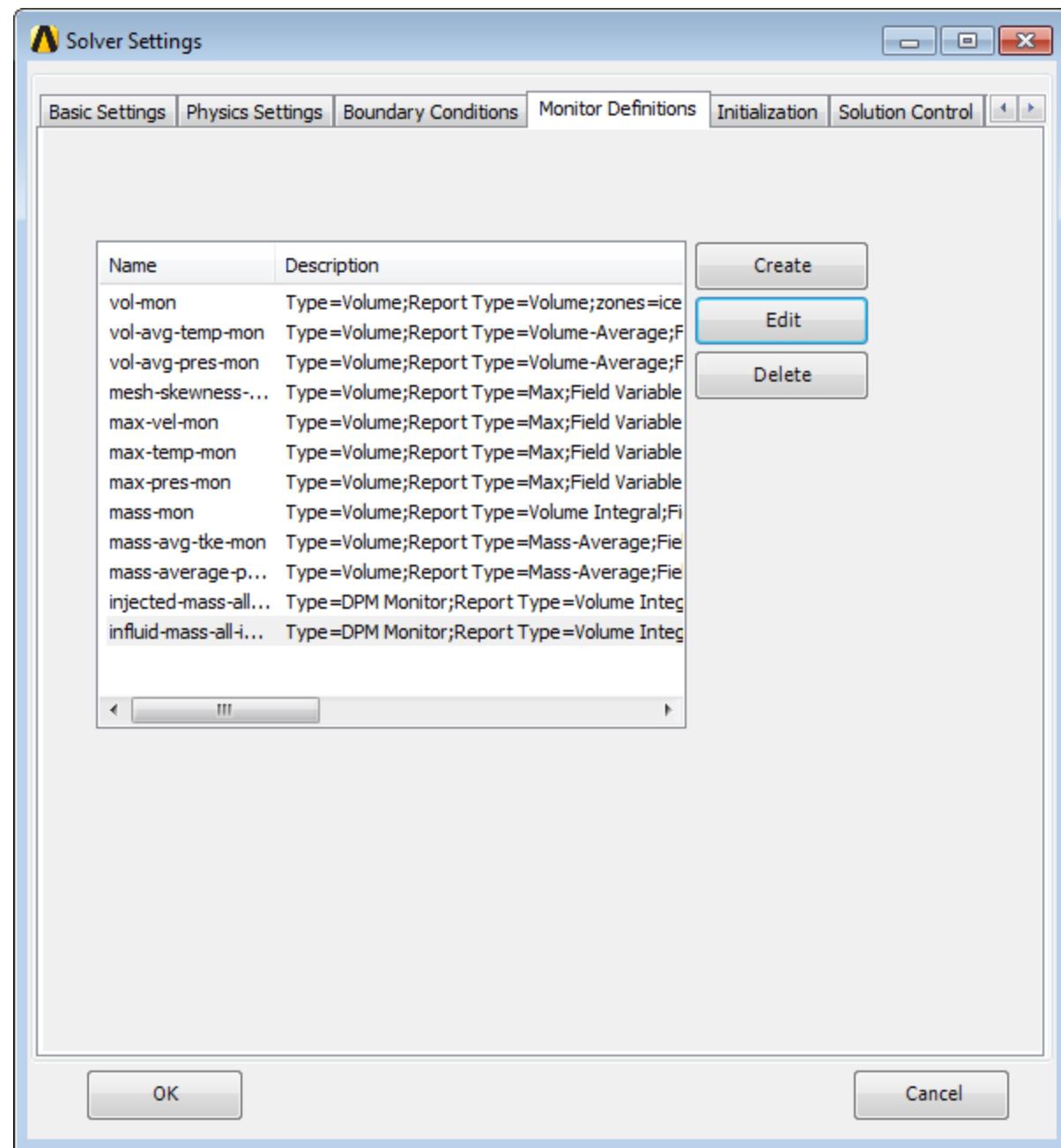
- v. Close the **Create Boundary Conditions** dialog box.
- g. In the **Monitor Definitions** tab you can see that nine volume monitors have been set on the zones **ice-fluid-chamber-bottom**, **ice-fluid-chamber-top**, and **ice-fluid-piston**. You will set additional monitors.
- Click **Create**.
 - In the **Monitor Definition** dialog box select all in the list of **Zones**.
 - Retain selection of **Volume** for **Type**.



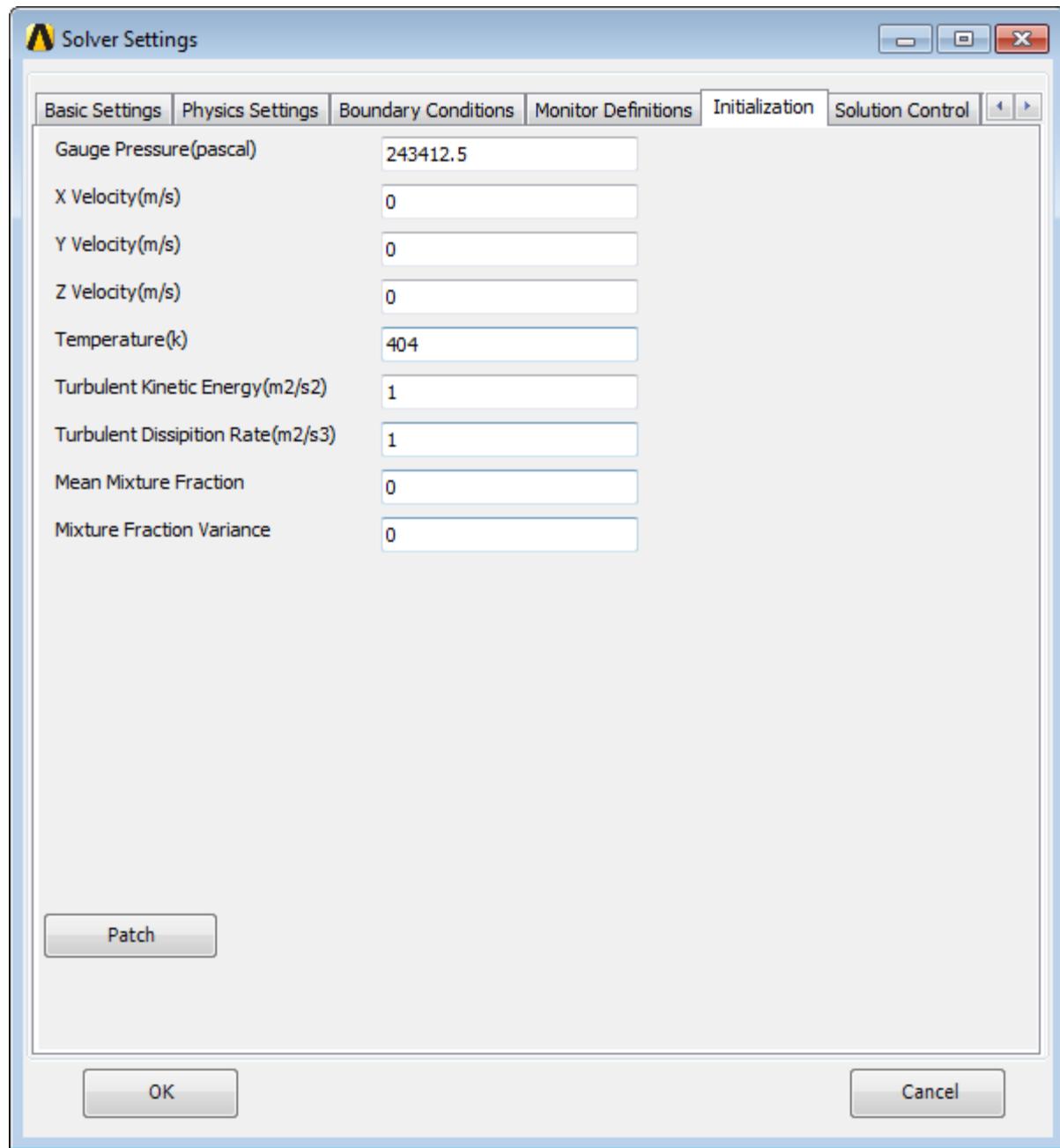
- iv. Select **Mass-Average** from the **Report Type** drop-down list.
- v. Select **Phi** from the **Field Variable** drop-down list.
- vi. Click **Create**. A volume monitor named **mass-average-phi-mon** is created.
- vii. Now select **DPM Monitor** from the **Type**.



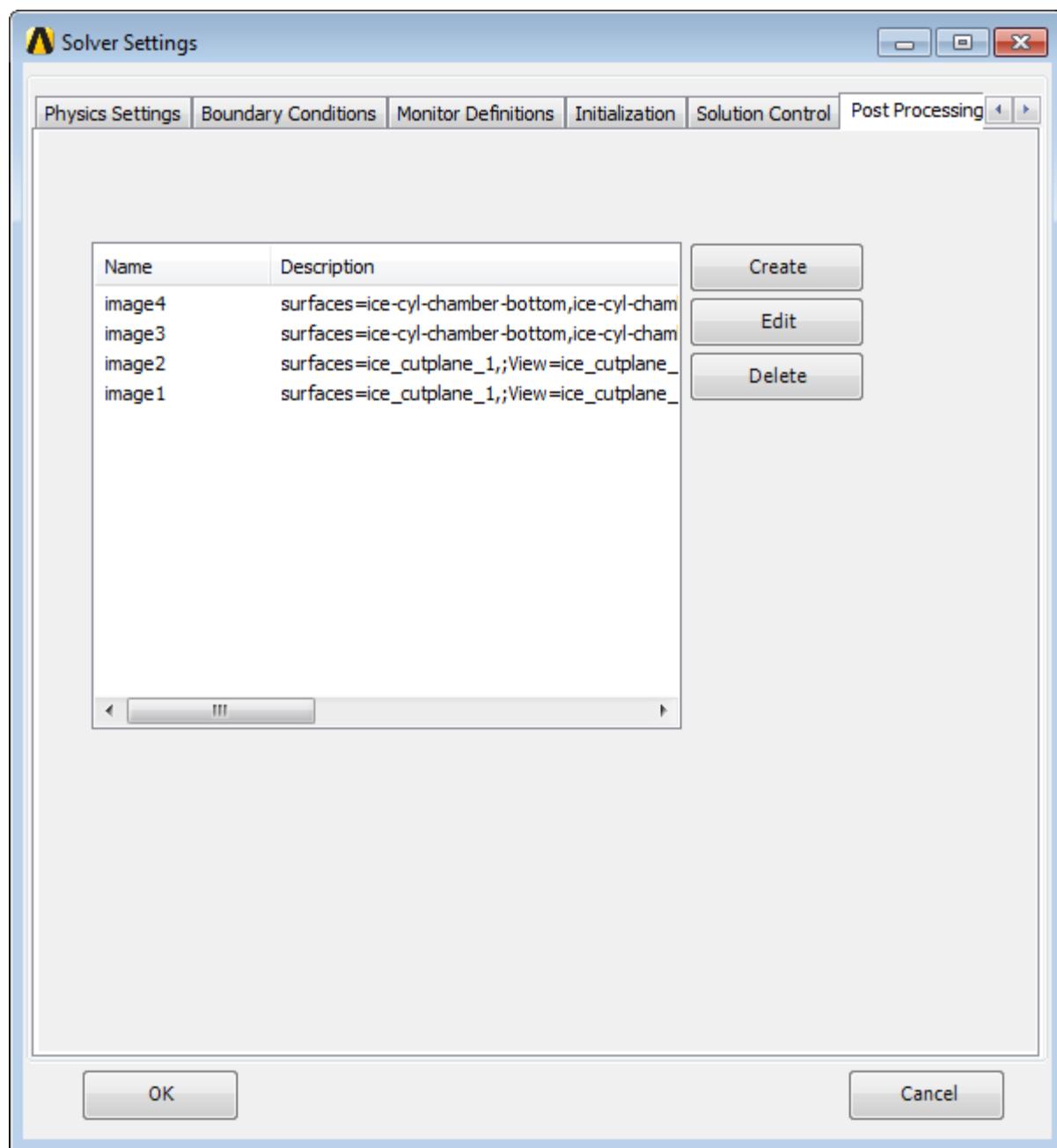
- viii. Select **injected-mass** from the **Injection Fate** drop-down list.
- ix. Select **all-injections** from the list of **Injection**.
- x. Click **Create**. A monitor by the name of **injected-mass-all-injections-mon** is created.
- xi. Select **influid-mass** from the **Injection Fate** drop-down list and click **Create** to create a monitor of **influid-mass-all-injections-mon**.



- h. In the **Initialization** tab enter 243412.5 for **Gauge Pressure** and 404 for **Temperature**.



- i. In the **Post Processing** tab you can see four types of images will be saved during simulation and displayed in a table format in the report. The details will be displayed after selecting the image name and clicking **Edit**.



3. Click **OK** to close the **Solver Settings** dialog box.
4. Right-click on the **Solution** cell and click on **Properties** to open the **Properties of Schematic** window (if not already open).

Properties of Schematic A7: Solution		
	A	B
1	Property	Value
2	General	
3	Component ID	Solution
4	Directory Name	ICE
5	Use Setup Launcher Settings	<input checked="" type="checkbox"/>
6	Precision	Double Precision
7	Show Launcher at Startup	<input checked="" type="checkbox"/>
8	Display Mesh After Reading	<input checked="" type="checkbox"/>
9	Embed Graphics Windows	<input checked="" type="checkbox"/>
10	Use Workbench Color Scheme	<input checked="" type="checkbox"/>
11	Environment Path	
12	Setup Compilation Environment for UDF	<input checked="" type="checkbox"/>
13	Use Job Scheduler	<input type="checkbox"/>
14	Run Parallel Version	<input checked="" type="checkbox"/>
15	UDF Compilation Script Path	<code>\$(FLUENT_ROOT)\\$(ARCH)\udf.bat</code>
16	Initialization Method	Solver Controlled <input type="button" value="▼"/>
17	Use Remote Linux Nodes	<input type="checkbox"/>
18	Solution Monitoring	<input type="checkbox"/>
19	Data Interpolation	<input type="checkbox"/>
20	Notes	
21	Notes	
22	Used Licenses	
23	Last Update Used Licenses	
24	Parallel Run Settings	
25	Number of Processors	4
26	Interconnect	default
27	MPI Type	default
28	Use Shared Memory	<input checked="" type="checkbox"/>
29	Solution Process	
30	Update Option	Run in Foreground <input type="button" value="▼"/>

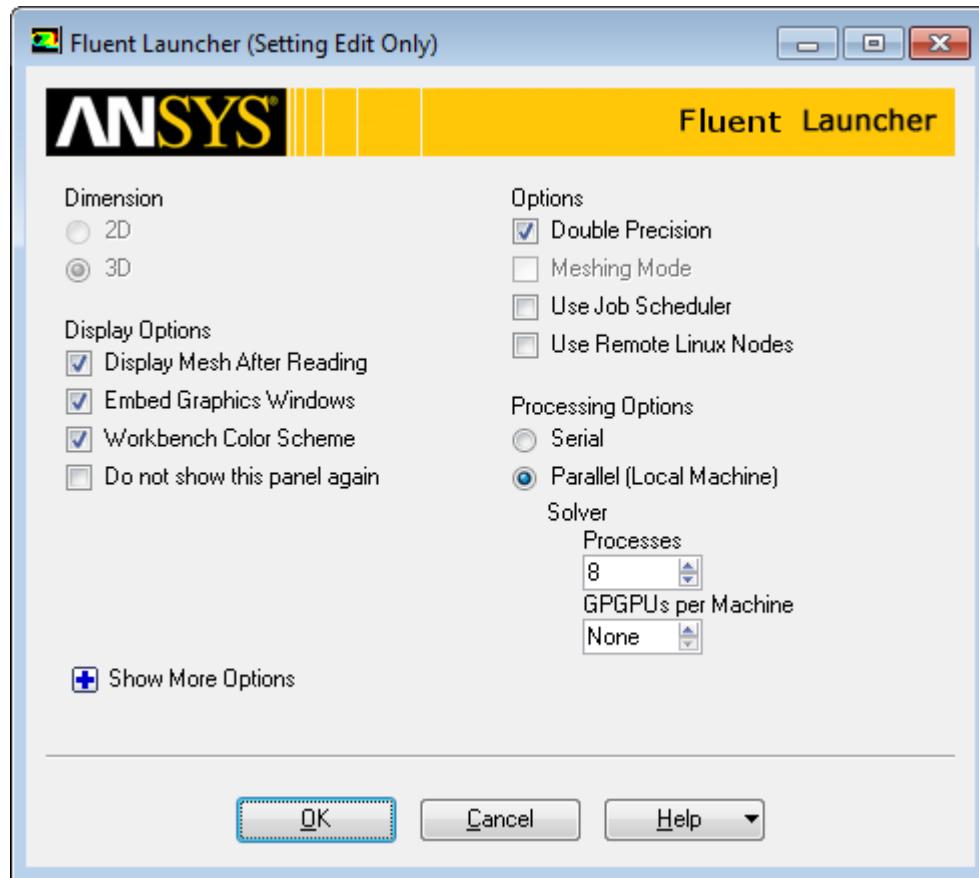
- a. In the **Properties of Schematic** window ensure that the **Initialization Method** is set to **Solver Controlled**.
- b. Disable **Solution Monitoring**.
5. Save the project.

File >Save

3.6. Step 5: Running the Solution

In this step you will start the simulation.

1. Double-click on the **Setup** cell.



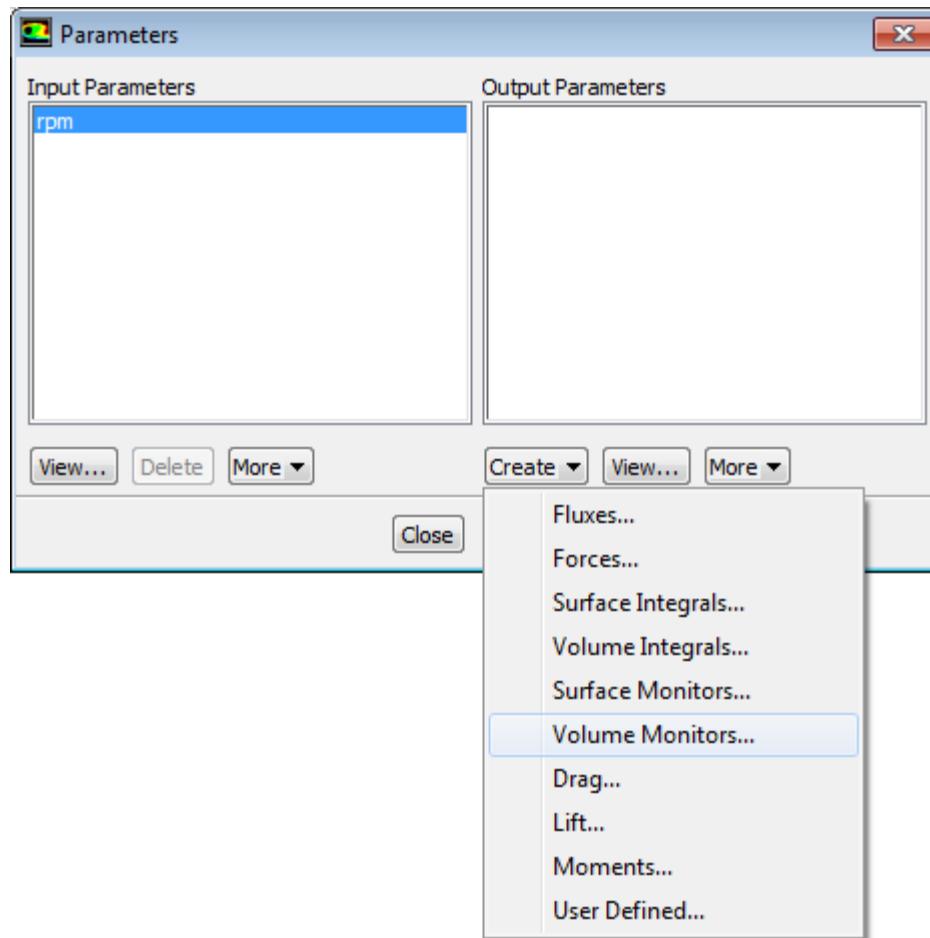
2. You can run the simulation in parallel with increased number of processors to complete the solution in less time.
3. Click **OK** in the **FLUENT Launcher** dialog box.

Note

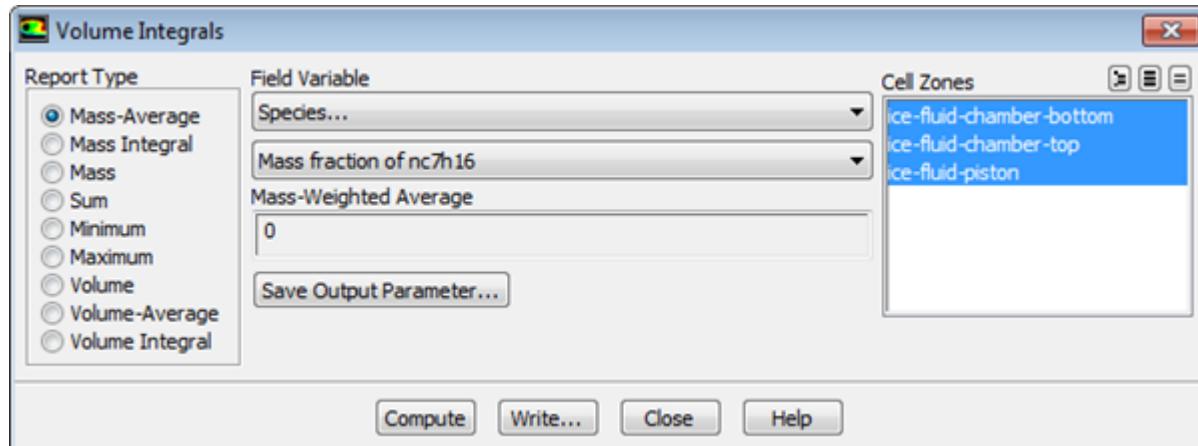
ANSYS Fluent opens. It will read the mesh file and setup the case.

4. To add an output parameter select **Parameters** from the **Define** menu.

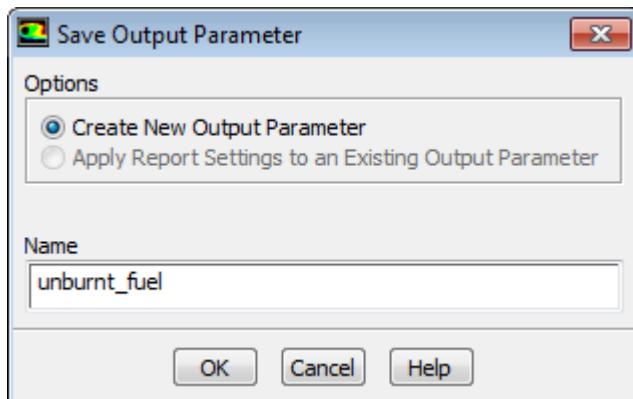
Define >Parameters...



- In the **Parameters** dialog box click **Create** and select **Volume Integrals...** from the drop-down list.



- In the **Volume Integrals** dialog box retain selection of **Mass-Average** from the **Report Type** list.
- Select **Species...** and **Mass fraction of nc7h16** from the **Field Variable** drop-down list.
- Select all zones from the list of **Cell Zones**.
- Click **Save Output Parameter...**.
- In the **Save Output Parameter** dialog box enter `unburnt_fuel` for **Name** and click **OK**.

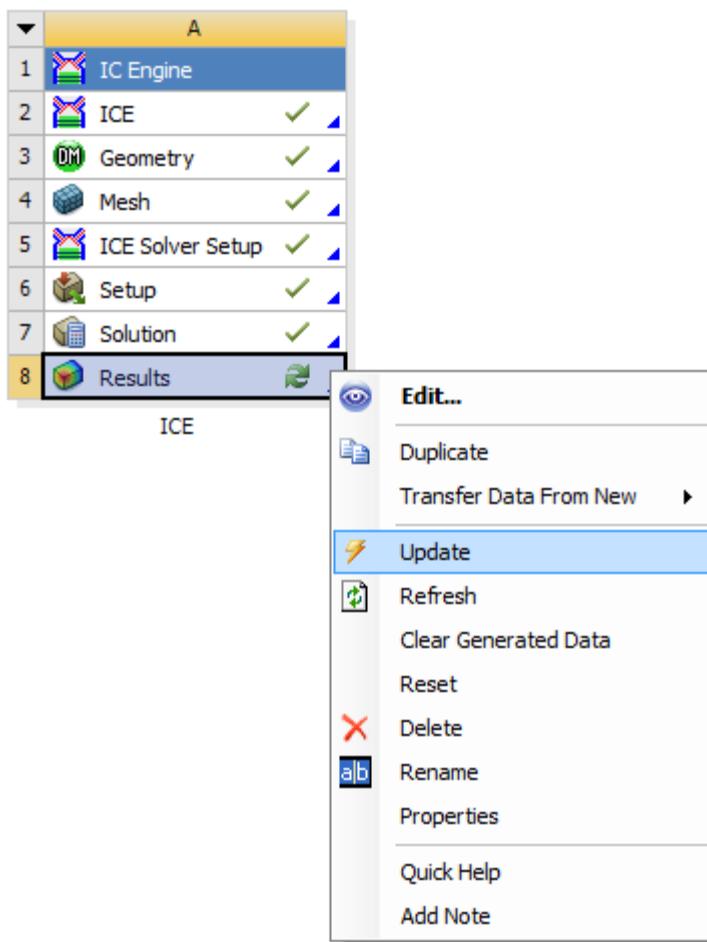


- g. Close the **Volume Integrals** and **Parameters** dialog box.
5. Return to Workbench window.
6. Double-click on **Parameter Set**.
 - a. In the **Table of Design Points** you can see **DP 0**. Under **P1-rpm** enter 2000 for **DP 1**.
 - b. Enable the check box next to **Retain**.
 - c. Click **Update All Design Points**.

Table of Design Points						
	A	B	C	D	E	F
1	Name	P1 - rpm	P2 - unburnt-fuel	<input type="checkbox"/> Retain	Retained Data	Note
2	DP 0 (Current)	1500	3.8406E-05	<input checked="" type="checkbox"/>	✓	
3	DP 1	2000	3.0406E-05	<input checked="" type="checkbox"/>	✓	
*				<input type="checkbox"/>		

3.7. Step 6: Obtaining the Results

1. Right-click on the **Results** cell and click **Update** from the context menu.



- Once the **Results** cell is updated, view the files by clicking **Files** from the **View** menu.

View >Files

- Right-click **Report.html** from the list of files, and click **Open Containing Folder** from the context menu.

	A	B	C	D	E	F
1	Name	Cell ID	Size	Type	Date Modified	Location
51	Report.xml	AB	13 KB	Default File	9/10/2013 5:52:41 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report
52	Report.html	AB	30 KB	Default File	9/10/2013 5:53:11 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report
53	AnsysReport.rpt			Ult File	9/10/2013 5:53:42 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
54	Chart001.png			Ult File	9/10/2013 5:53:06 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
55	Chart002.png			Ult File	9/10/2013 5:53:07 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
56	Chart003.png			Ult File	9/10/2013 5:53:07 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
57	Chart004.png	AB	11 KB	Default File	9/10/2013 5:53:07 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
58	Chart005.png	AB	12 KB	Default File	9/10/2013 5:53:08 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
59	Chart006.png	AB	15 KB	Default File	9/10/2013 5:53:08 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
60	Chart007.png	AB	13 KB	Default File	9/10/2013 5:53:08 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
61	Chart008.png	AB	13 KB	Default File	9/10/2013 5:53:09 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
62	Chart009.png	AB	13 KB	Default File	9/10/2013 5:53:09 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
63	Chart010.png	AB	13 KB	Default File	9/10/2013 5:53:09 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
64	Chart011.png	AB	14 KB	Default File	9/10/2013 5:53:10 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report
65	Chart012.png	AB	14 KB	Default File	9/10/2013 5:53:10 PM	E:\ICE\ICE15\Combustion\Sector-tut\demo_sector_files\dp0\ICE\Post\Report\Report

- In the **Report** folder double-click **Report.html** to open the report.



Title

IC Engine Combustion Simulation Report

Contents

- [1. File Report](#)
 - [Table 1](#) File Information for ICE
- [2. Mesh Report](#)
 - [Table 2](#) Mesh Information for ICE
 - [Chart 1](#) Monitor: Max Cell Equivolume Skew (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Table 3](#) Cell count at crank angles
- [3. Setup](#)
 - [3.1. Physics](#)
 - [Table 4](#) Boundary Conditions
 - [Table 5](#) Relaxation changes through events
 - [3.3. Dynamic Mesh Setup](#)
 - [Table 6](#) Dynamic Mesh Events
 - [3.4. IC Engine System Inputs](#)
- [4. Solution Data](#)
 - [4.1. Animation: mesh-on-ice_cutplane_1](#)
 - [4.2. Animation: pt-temperature on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-piston, ice-sector-top-faces](#)
 - [4.3. Animation: pt-velocity-magnitude on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-piston, ice-sector-top-faces](#)
 - [4.4. Animation: temperature on ice_cutplane_1](#)
 - [4.5. Animation: velocity-magnitude on ice_cutplane_1](#)
 - [4.6. Table: mesh-on-ice_cutplane_1](#)
 - [4.7. Table: pt-temperature on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-piston, ice-sector-top-faces](#)
 - [4.8. Table: pt-velocity-magnitude on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-piston, ice-sector-top-faces](#)
 - [4.9. Table: temperature on ice_cutplane_1](#)
 - [4.10. Table: velocity-magnitude on ice_cutplane_1](#)
 - [4.11. Table: Residuals](#)
 - [4.12. Charts](#)
 - [Chart 2](#) Last iteration residual values corresponding to each time step
 - [Chart 3](#) Swirl Ratio
 - [Chart 4](#) Tumble Ratio
 - [Chart 5](#) Cross Tumble Ratio
 - [Chart 6](#) Apparent Heat Release Rate on (ice-fluid-chamber-bottom ice-fluid-chamber-top ice-fluid-piston)
 - [Chart 7](#) Number of Iterations per Time Step
 - [Chart 8](#) Monitor: Mass-Average phi (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 9](#) Monitor: Mass-Average Turbulent Kinetic Energy (k) (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 10](#) Monitor: Volume Integral Density (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 11](#) Monitor: Max Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 12](#) Monitor: Max Static Temperature (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 13](#) Monitor: Max Velocity Magnitude (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 14](#) Penetration length of injection=0 per Time Step
 - [Chart 15](#) Monitor: Volume-Average Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 16](#) Monitor: Volume-Average Static Temperature (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 17](#) Monitor: Volume Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)
 - [Chart 18](#) Total mass influid for all injections per Time Step
 - [Chart 19](#) Total mass injected for all injections per Time Step
 - [Chart 20](#) Total mass evaporated for all injections per Time Step
- [5. Design Points Report](#)
 - [5.1. Design Points Parameter values Charts](#)

- You can check the node count and mesh count of the cell zones in the table, **Mesh Information for ICE**.

1. File Report

Table 1. File Information for ICE

Case	ICE
File Path	E:\Sector-tutorial-work\SECTOR-TUTORIAL\Solution\run-13-build\parametric\NST-dufl-13-build-parameter_files\dp0\ICE\Fluent\ICE-2-ca832.950.dat.gz
File Date	18 November 2014
File Time	11:24:06 PM
File Type	FLUENT
File Version	16.0.0

2. Mesh Report

Table 2. Mesh Information for ICE

Domain	Nodes	Elements
ice fluid chamber bottom	465252	445264
ice fluid chamber top	13584	9822
ice fluid piston	72492	65683
All Domains	551328	520769

Chart 1. Monitor: Max Cell Equivolume Skew (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

**Monitor: Max Cell Equivolume Skew (ice-fluid-piston
ice-fluid-chamber-top ice-fluid-chamber-bottom)**

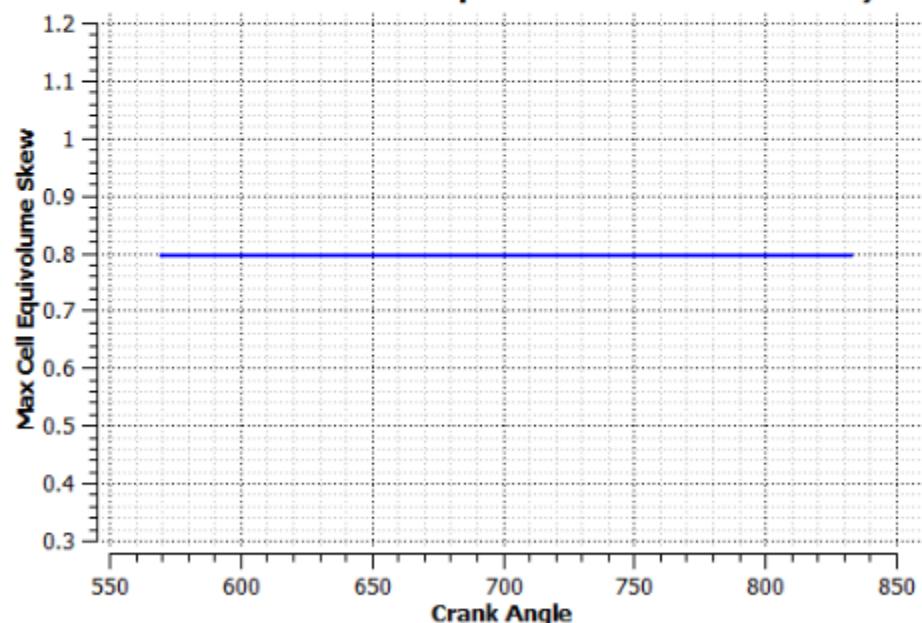


Table 3. Cell count at crank angles

Crank Angle	Cell Count
0.000e+00	8.860e+04

- You can see the boundary conditions, under-relaxations factors, and other setup conditions under **Setup**.

3. Setup

3.1. Physics

Table 4. Boundary Conditions

Type	Zones	Values
wall	ice-cyl-chamber-bottom	Temperature (k) 440
wall	ice-cyl-chamber-top	Temperature (k) 440
wall	ice-cyl-piston	Temperature (k) 440
wall	ice-piston	Temperature (k) 560
wall	ice-sector-top-faces	Temperature (k) 480

3.2. Relaxations

Table 5. Relaxation changes through events

Crank Angle	Pressure	Density	Body Forces	Momentum	Turbulent Kinetic Energy	Turbulent Dissipation Rate	Turbulent Viscosity	Energy	Temperature	Unsteady Flamelet Probability	Mean Mixture Fraction	Mixture Fraction Variance	Discrete Phase Source
0.000	0.300	1.000	1.000	0.500	0.400	0.400	1.000	1.000	1.000	1.000	1.000	0.900	1.000

3.3. Dynamic Mesh Setup

Table 6. Dynamic Mesh Events

At Crank Angle (deg)	Name	Description
570.000	pdf-equation-on-event, dt-bound-start-at-570.00(0.25)	
712.000	enable-pt-cal-act-for-injection-0, enable-writing-dpm-monitors-to-file, dt-injection-0-unsteady-ca-start-at-712.00(0.05)	
720.000	write-solution-point-at-ca-720.000	Saves solution files at this point.
738.200	disable-pt-cal-act-for-injection-0, dt-injection-0-unsteady-ca-end-at-738.20(0.25)	
833.000	dt-bound-end-at-833.00(0.25)	

3.4. IC Engine System Inputs

Engine Inputs

Engine Speed (rev/min) : 1500
Crank Radius (mm) : 55
Piston Pin Offset/Wrench (mm) : 0
Connecting Rod Length (mm) : 165
-

Journal Customization

Pre Iteration Journal File : N/A
Post Iteration Journal File : N/A

- Check the animation of mesh on the cut-plane in the section **Solution Data**.

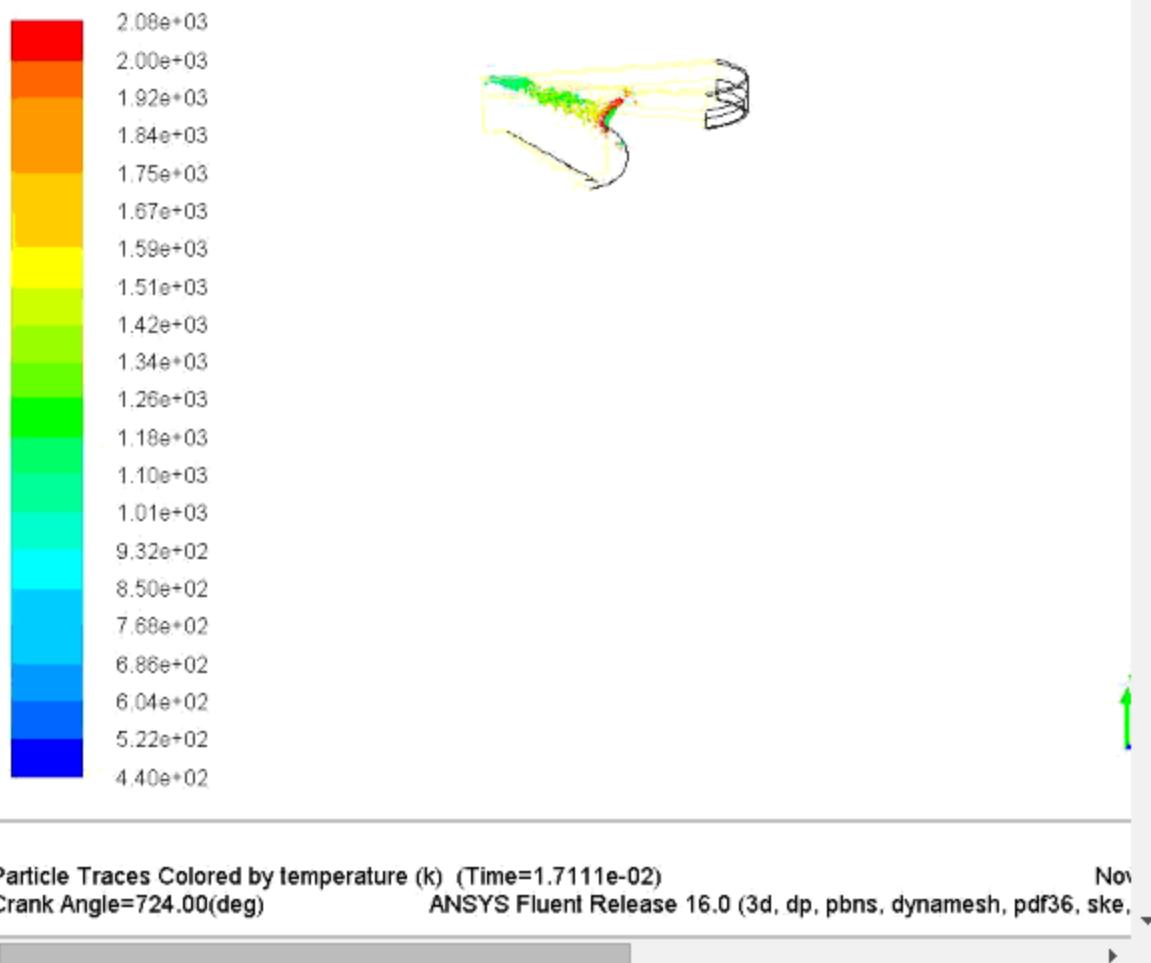
4. Solution Data

4.1. Animation: mesh-on-ice_cutplane_1

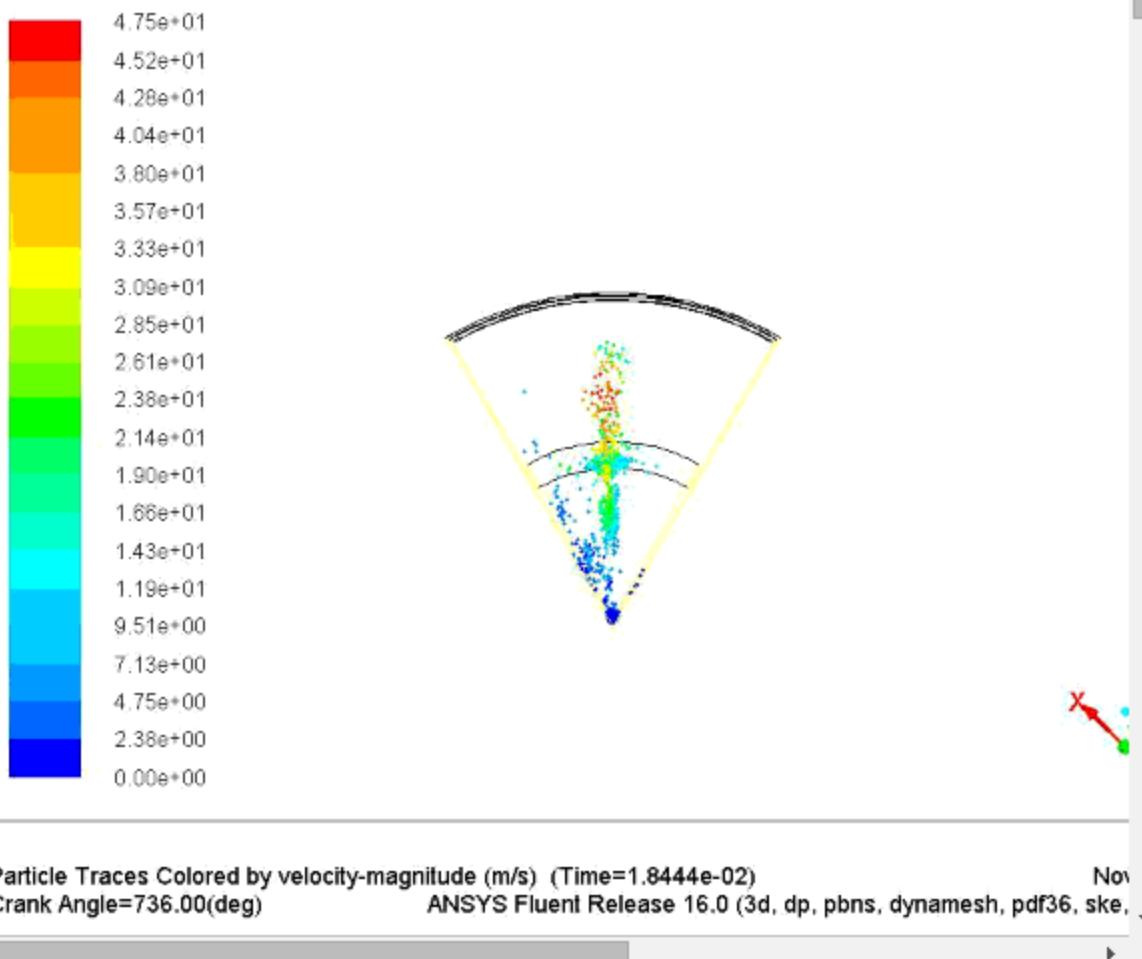


- Check the two animations of particle traces in the section **Solution Data**. Both show the animation from different views.

4.2. Animation: pt-temperature on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-piston, ice-sector-top-faces

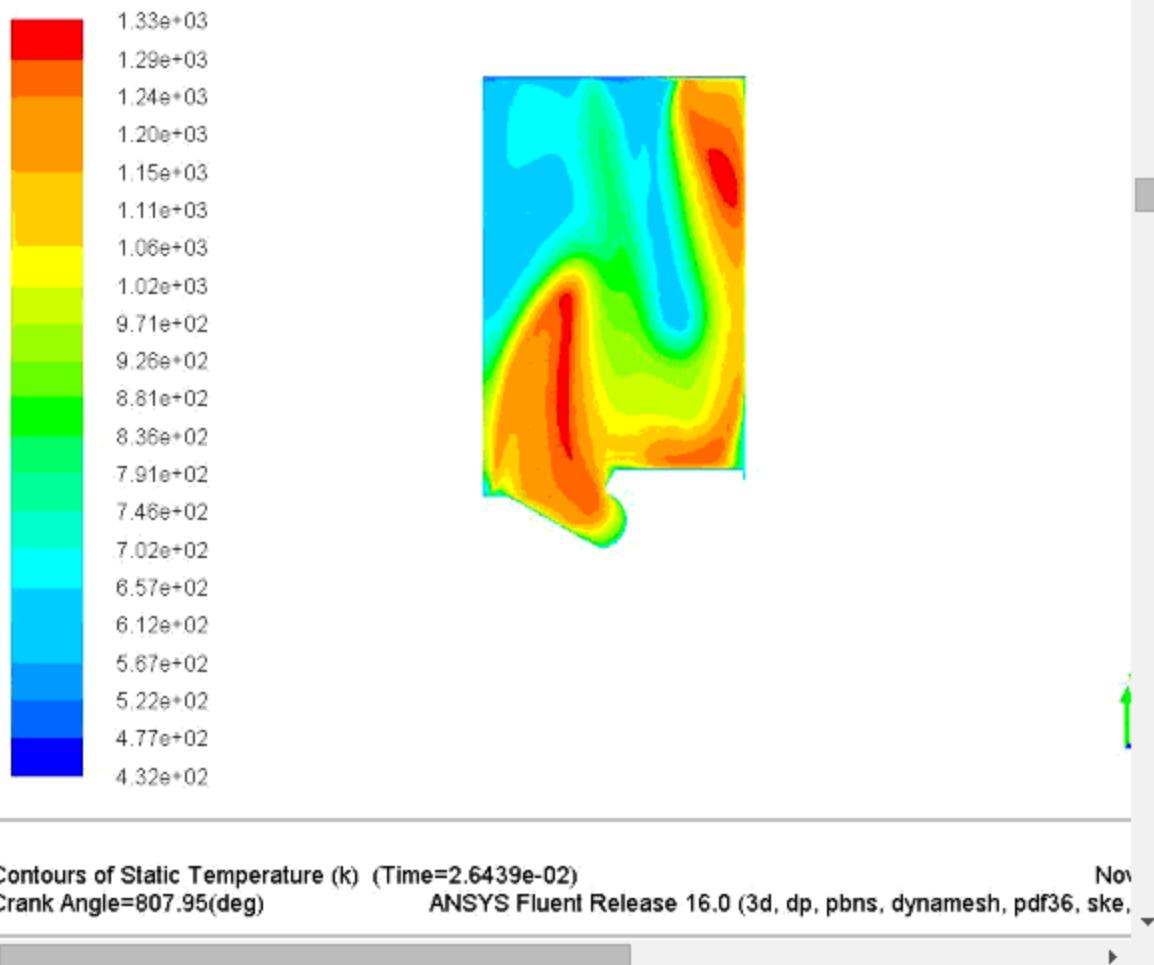


4.3. Animation: pt-velocity-magnitude on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-sector-top-faces



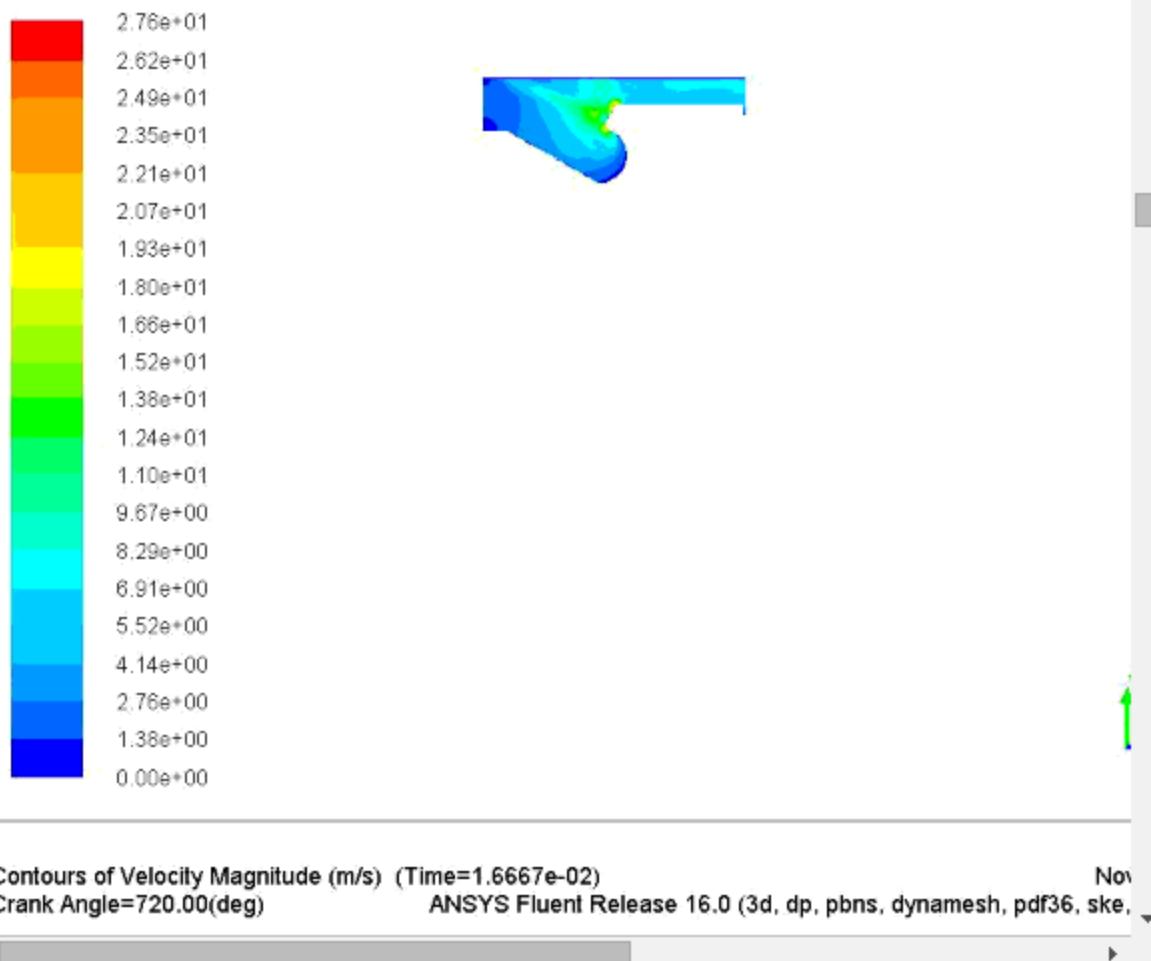
- Check the animation of temperature.

4.4. Animation: temperature on ice_cutplane_1



- Check the animation of velocity magnitude.

4.5. Animation: velocity-magnitude on ice_cutplane_1

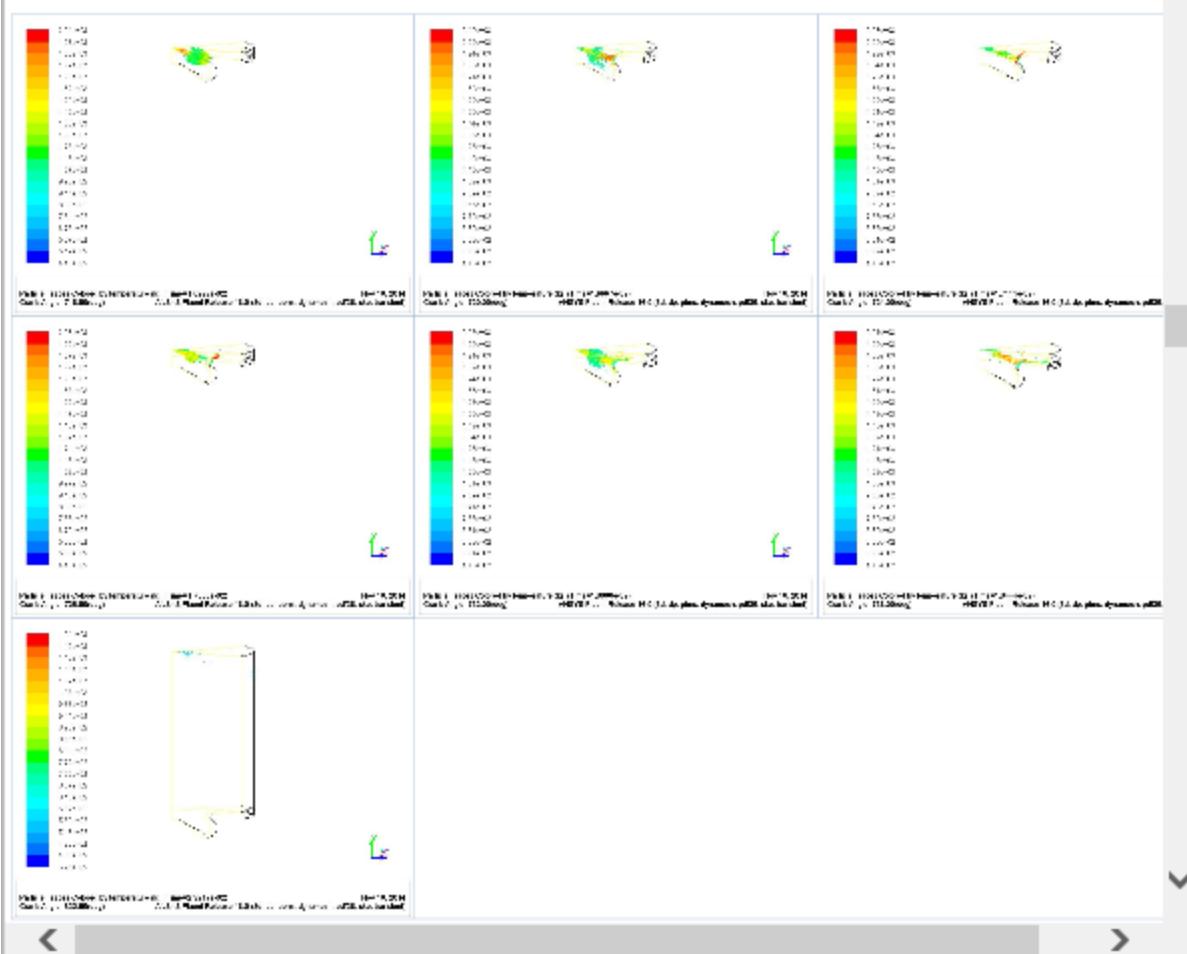


- In a **Table** you can observe the mesh images at various stages of the simulation.

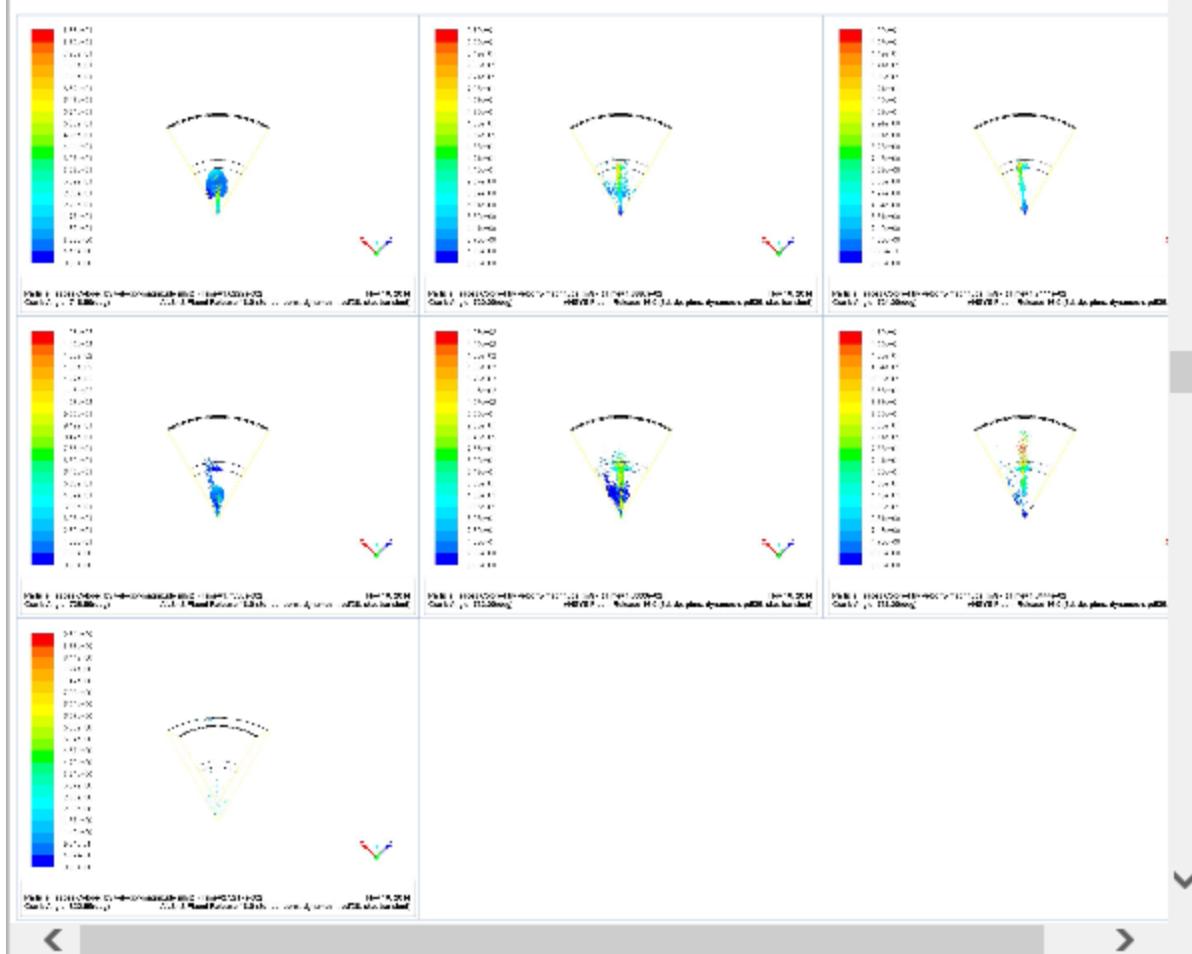
4.6. Table: mesh-on-ice_cutplane_1

- In two tables you can observe the particle traces images at various stages of simulation from two different views.

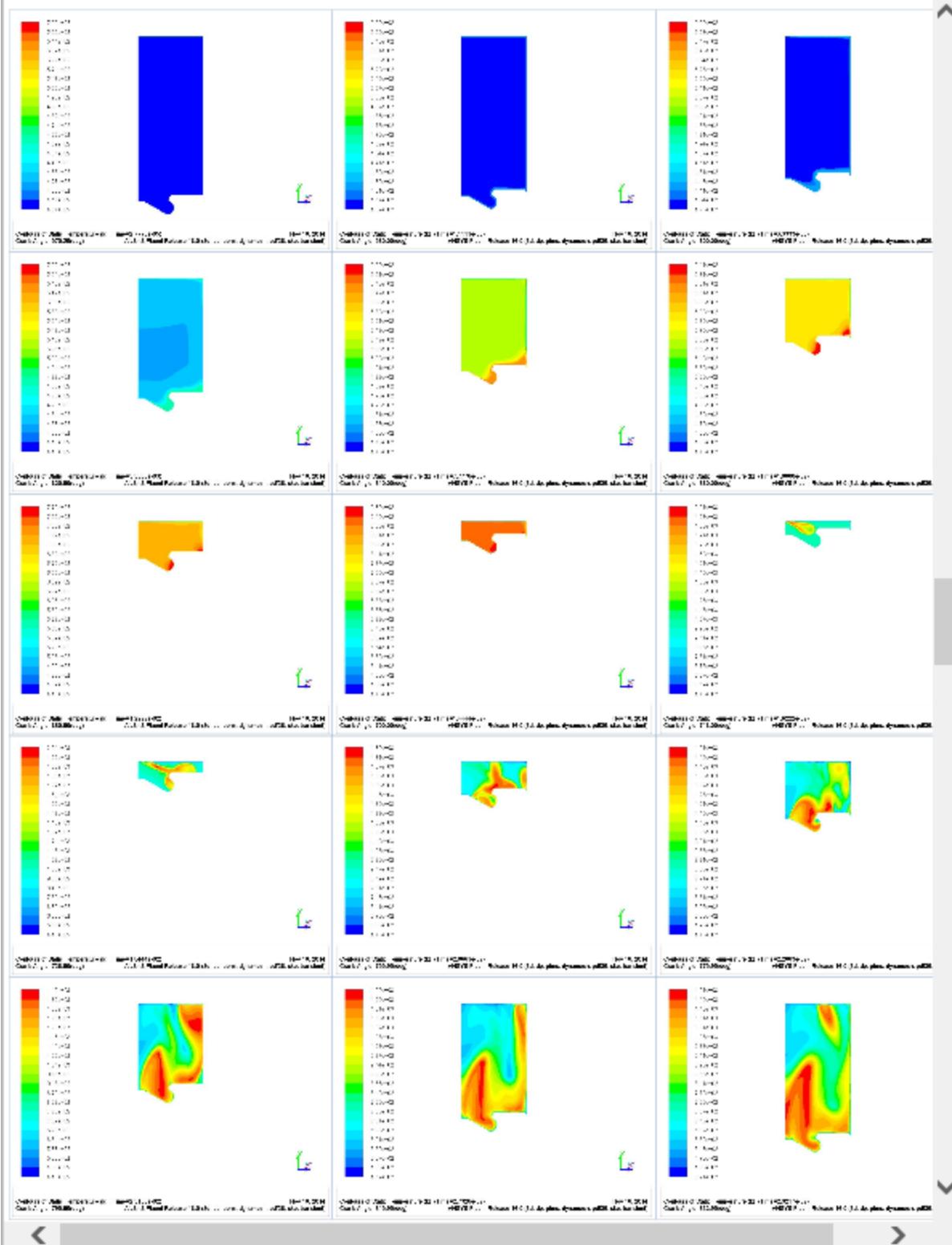
4.7. Table: pt-temperature on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-piston, ice-sector-top-faces



4.8. Table: pt-velocity-magnitude on ice-cyl-chamber-bottom, ice-cyl-chamber-top, ice-cyl-piston, ice-periodic-faces1-chamber-bottom, ice-periodic-faces1-chamber-top, ice-periodic-faces1-piston, ice-periodic-faces2-chamber-bottom, ice-periodic-faces2-chamber-top, ice-periodic-faces2-piston, ice-piston, ice-sector-top-faces

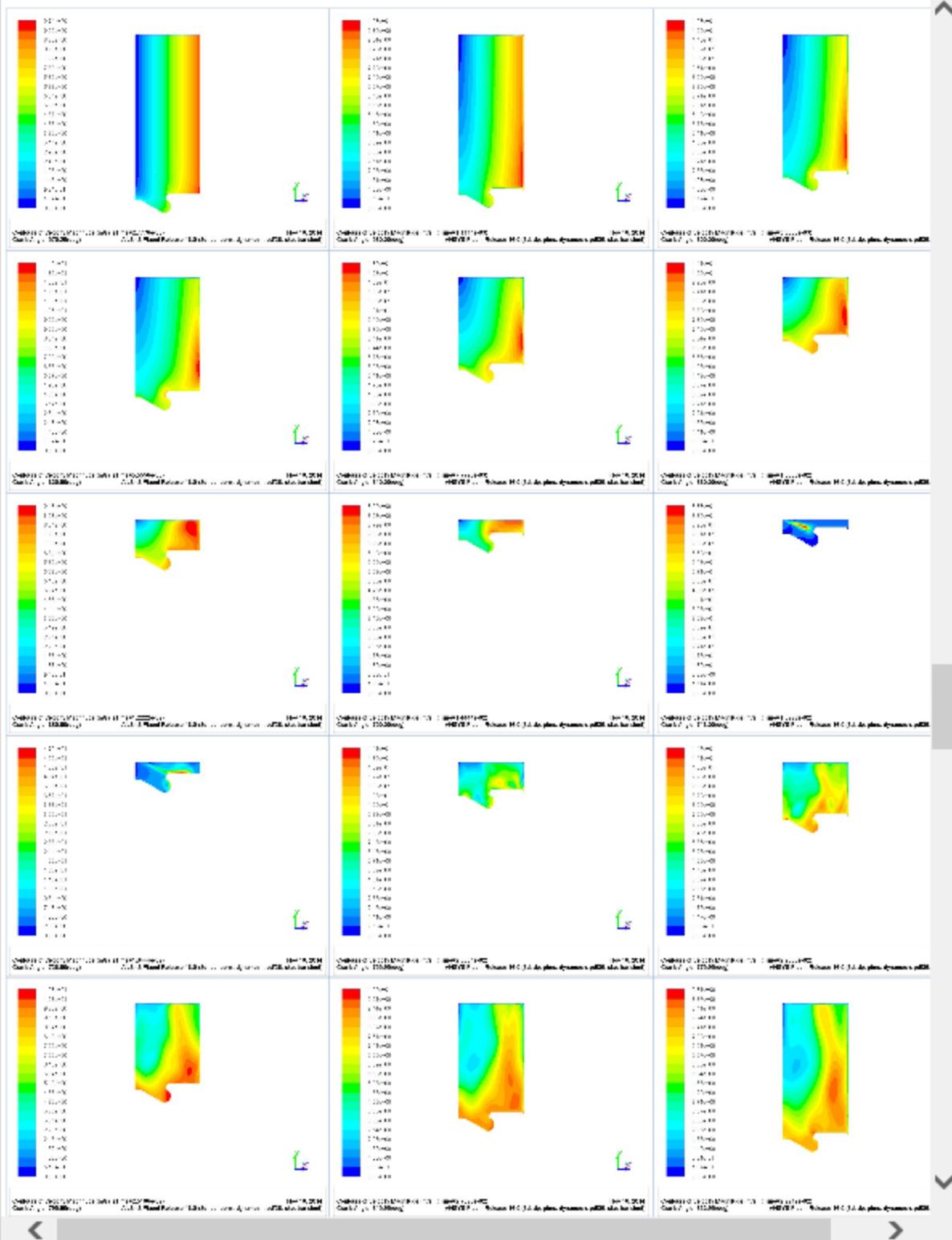


- In another **Table** you can observe the temperature contours on the cut plane. These images are taken at the end of the simulation.

4.9. Table: temperature on ice_cutplane_1

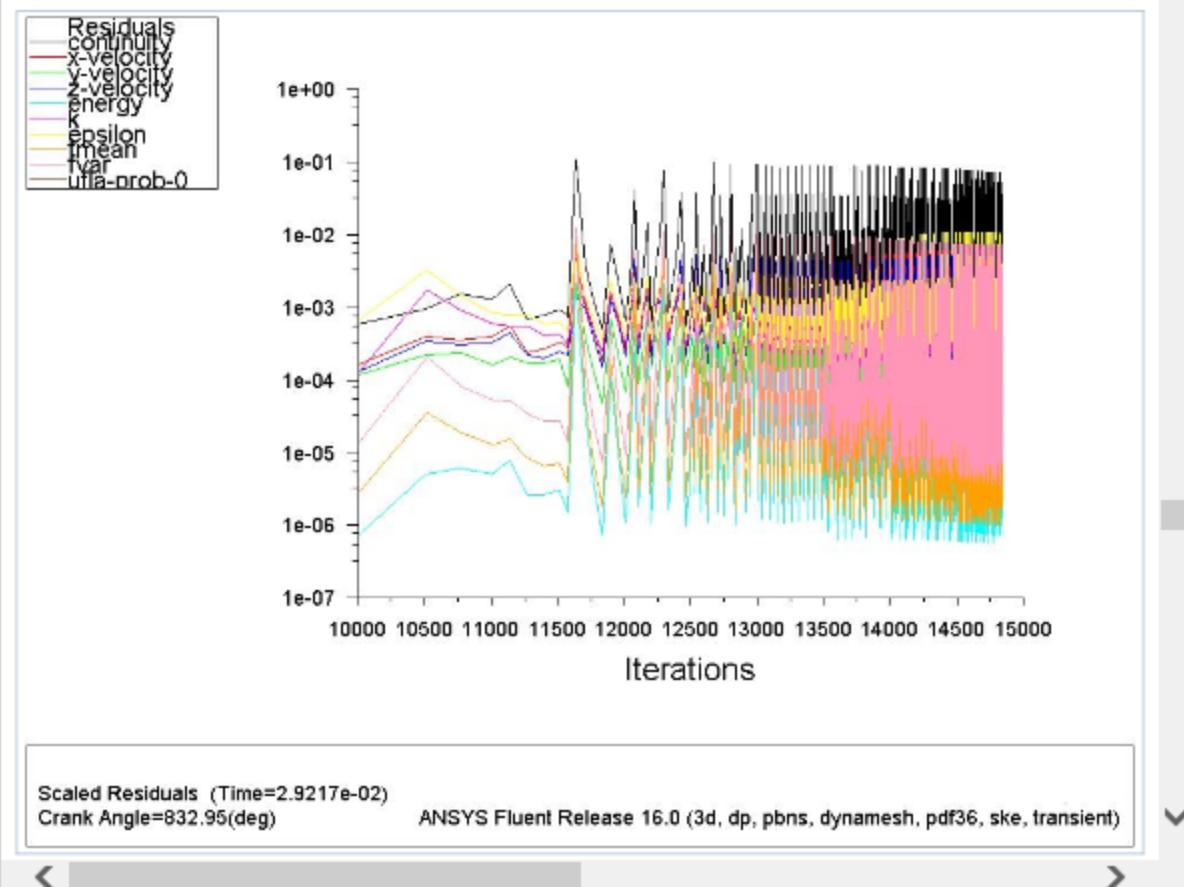
- You can also observe the velocity-magnitude contours on the cut-plane in another **Table**. These images are taken at the end of the simulation.

4.10. Table: velocity-magnitude on ice_cutplane_1



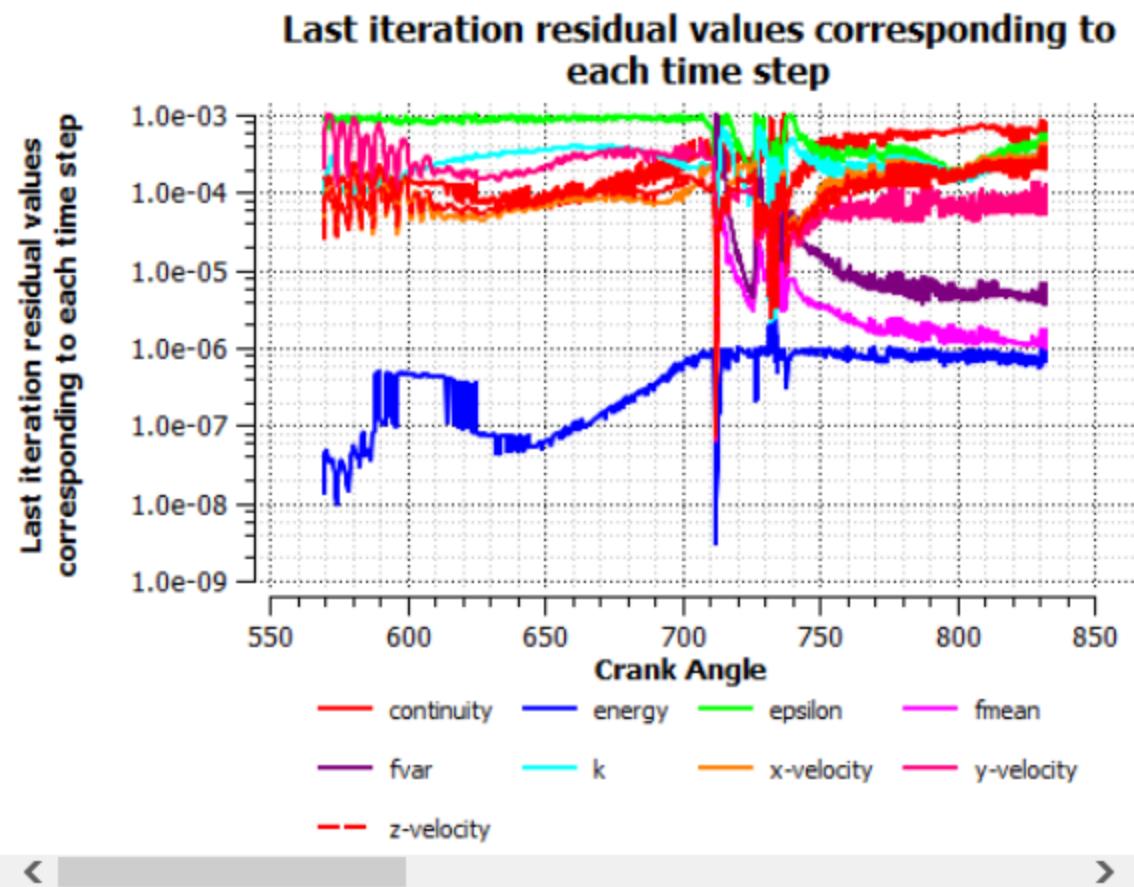
- Under **Charts** you will find plots of **Residuals** and **Last iteration residual corresponding to each time step**

4.11. Table: Residuals



4.12. Charts

Chart 2. Last iteration residual values corresponding to each time step



- You will find plots for **Swirl Ratio**, **Tumble Ratio**, and **Cross Tumble Ratio**.

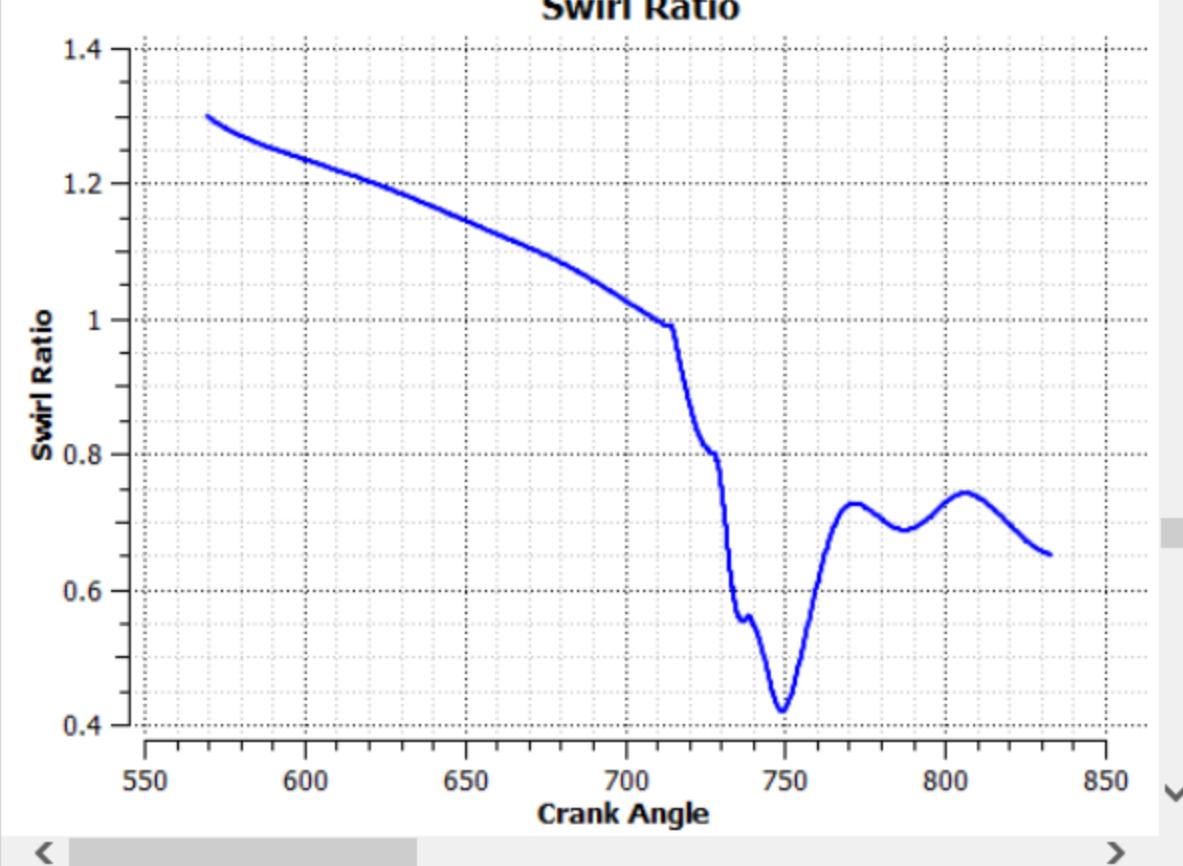
Chart 3. Swirl Ratio

Chart 4. Tumble Ratio

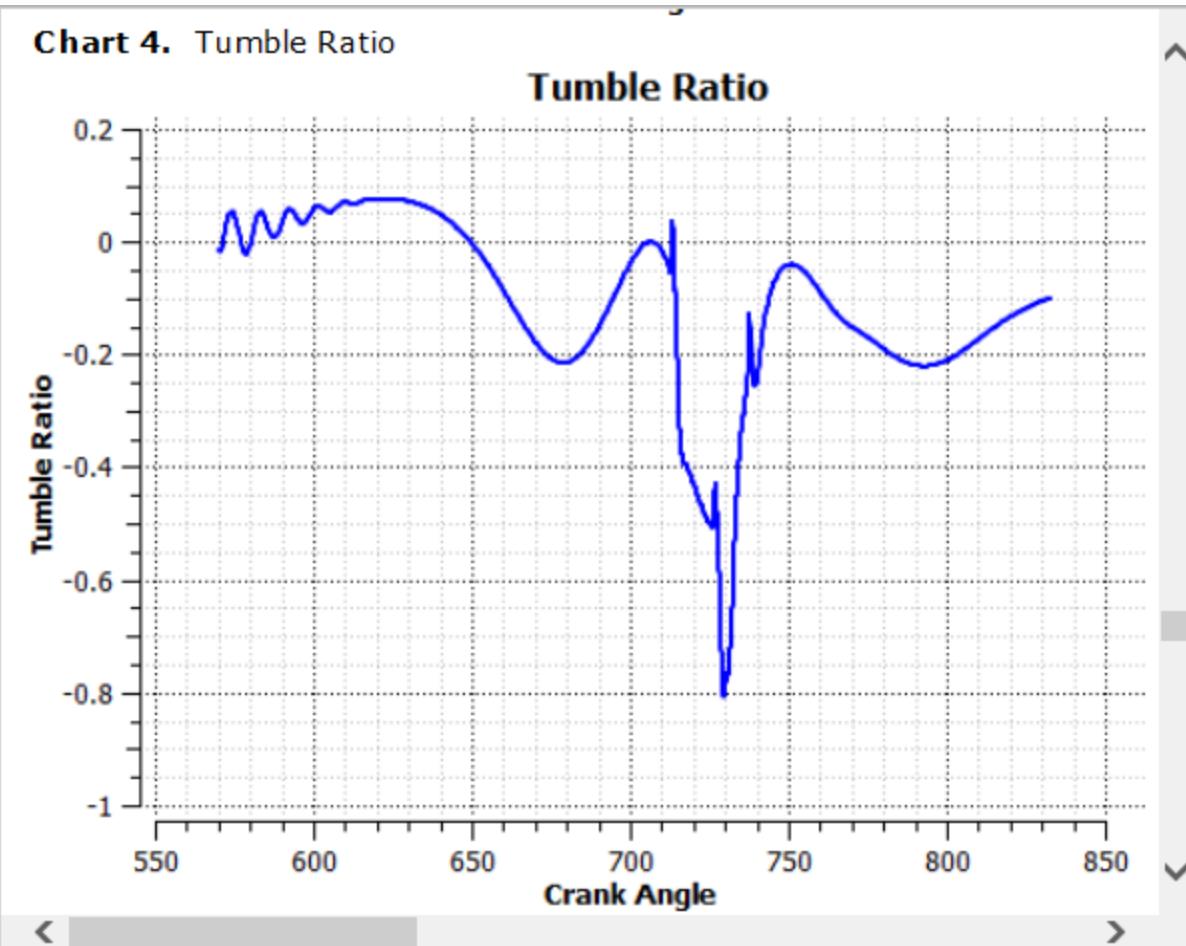
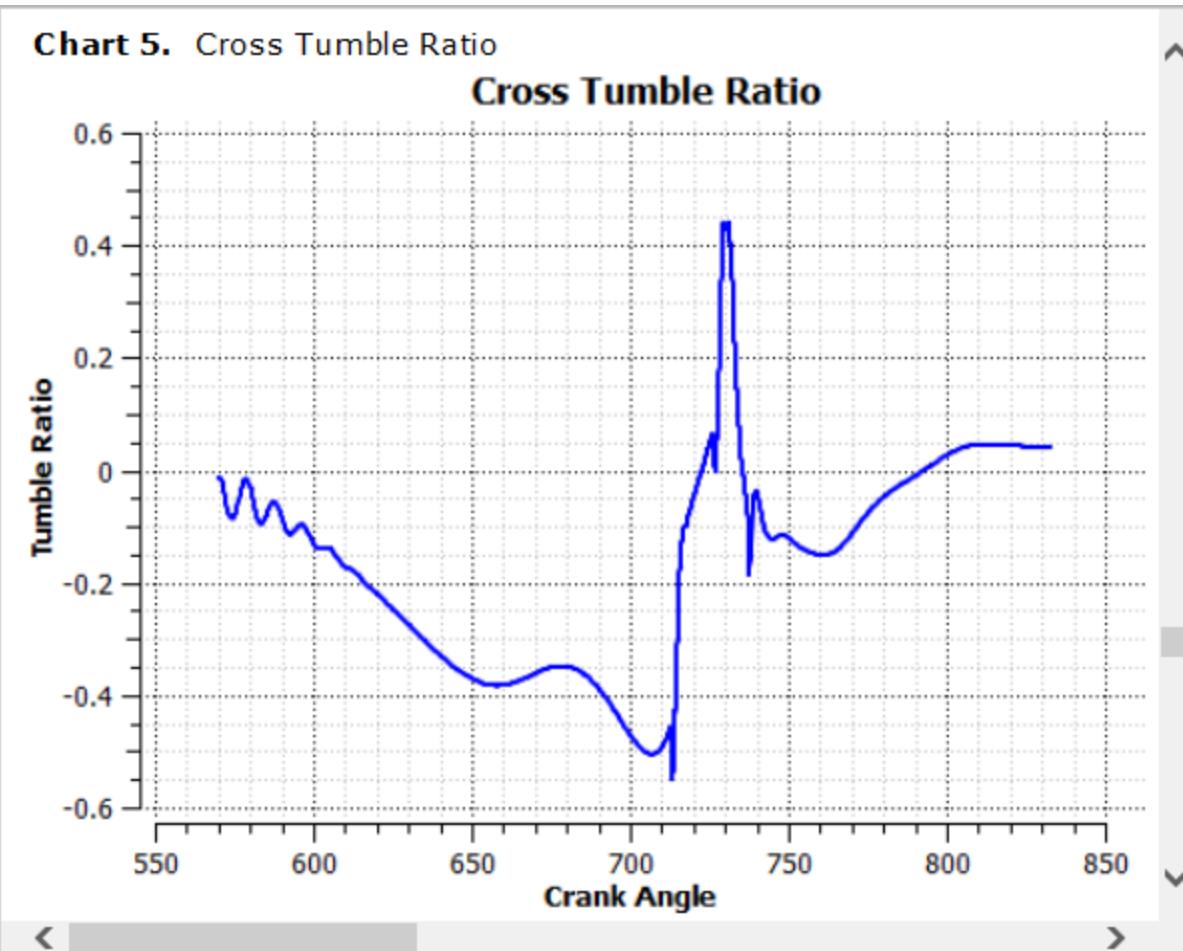


Chart 5. Cross Tumble Ratio

- You will find plots of **AHRR** (Apparent Heat Release Rate) and **Number of Iterations per Time Step**. Apparent heat release rate is defined as:

$$AHRR = \frac{\gamma}{\gamma-1} \times P \times \frac{V_2 - V_1}{A_2 - A_1} + \frac{1}{\gamma-1} \times V \times \frac{P_2 - P_1}{A_2 - A_1}$$

where

γ = 1.35 (can also be computed from Fluent)

V = Volume of sector, m³ X number of sectors

P = Absolute pressure, Pa

A = Crank Angle

Chart 6. Apparent Heat Release Rate on (ice-fluid-chamber-bottom ice-fluid-chamber-top ice-fluid-piston)

Apparent Heat Release Rate on (ice-fluid-chamber-bottom ice-fluid-chamber-top ice-fluid-piston)

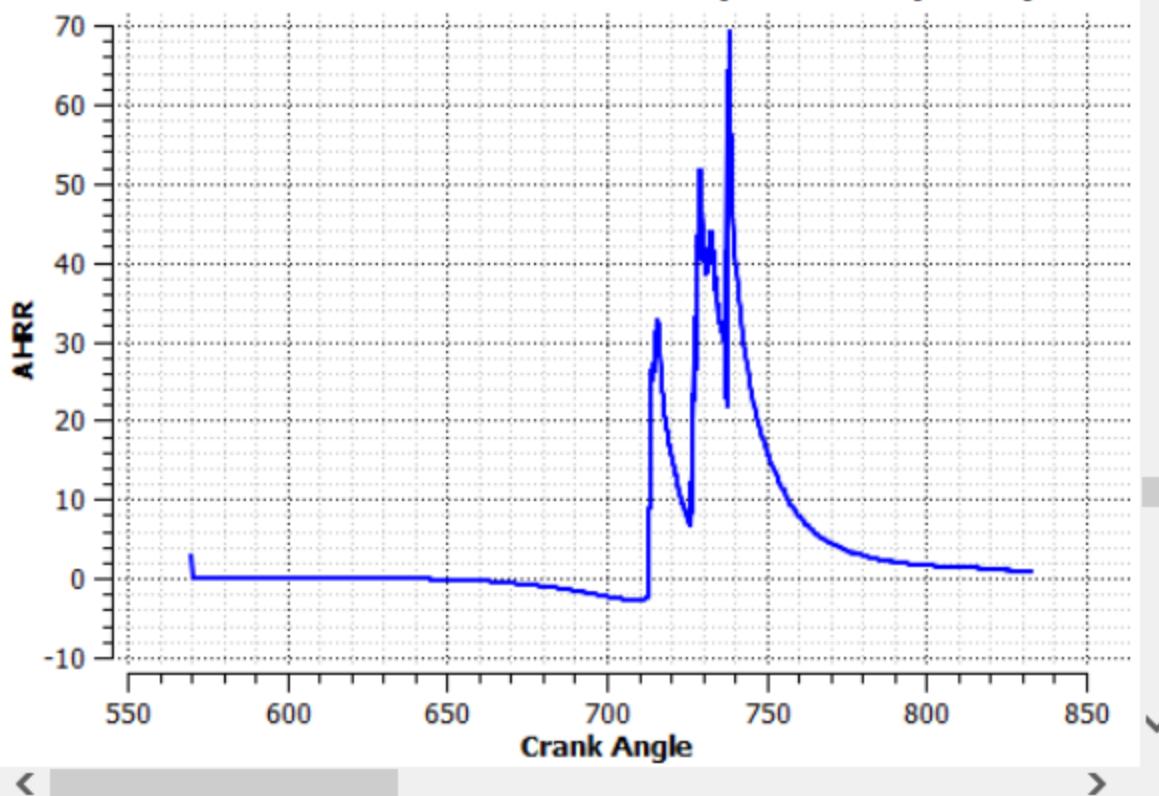
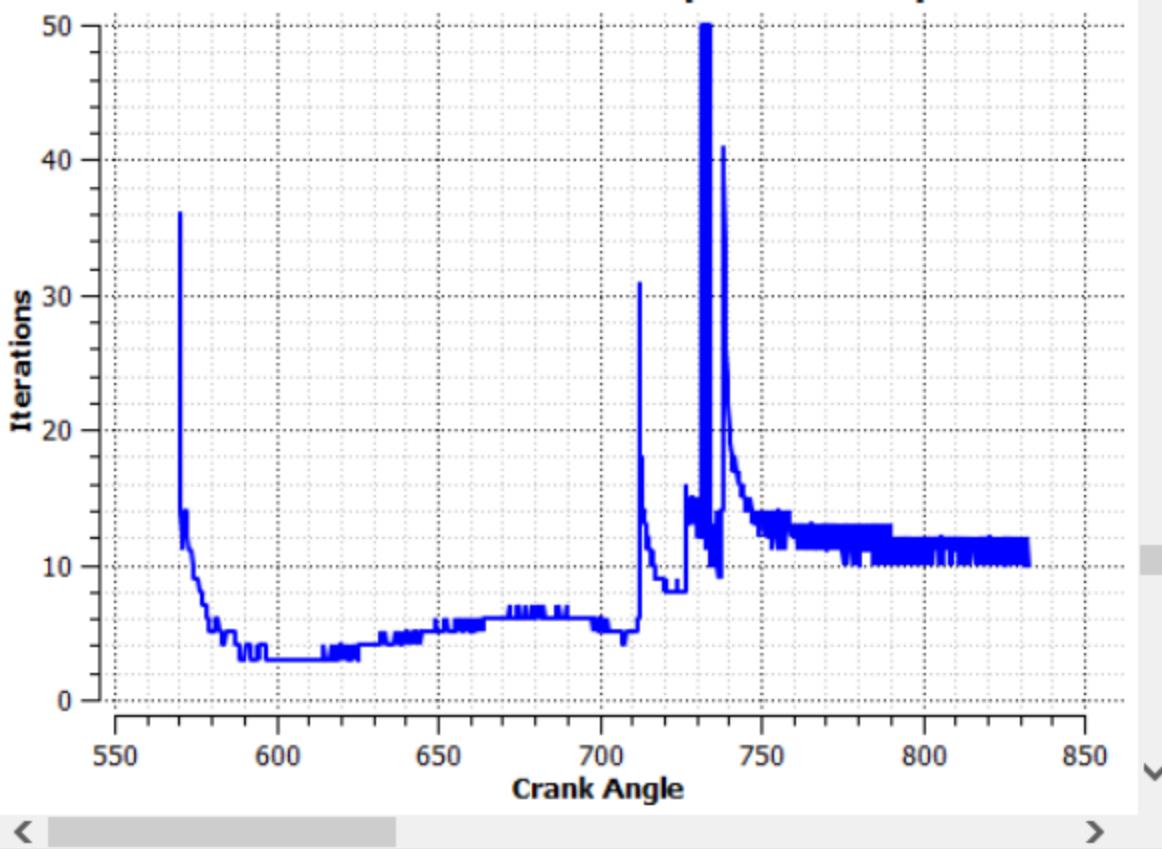


Chart 7. Number of Iterations per Time Step**Number of Iterations per Time Step**

- Monitors of **Mass-Average phi**, **Mass-Average Turbulent Kinetic Energy**, and **Volume Integral Density** are plotted against the crank angle.

Chart 8. Monitor: Mass-Average phi (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

Monitor: Mass-Average phi (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

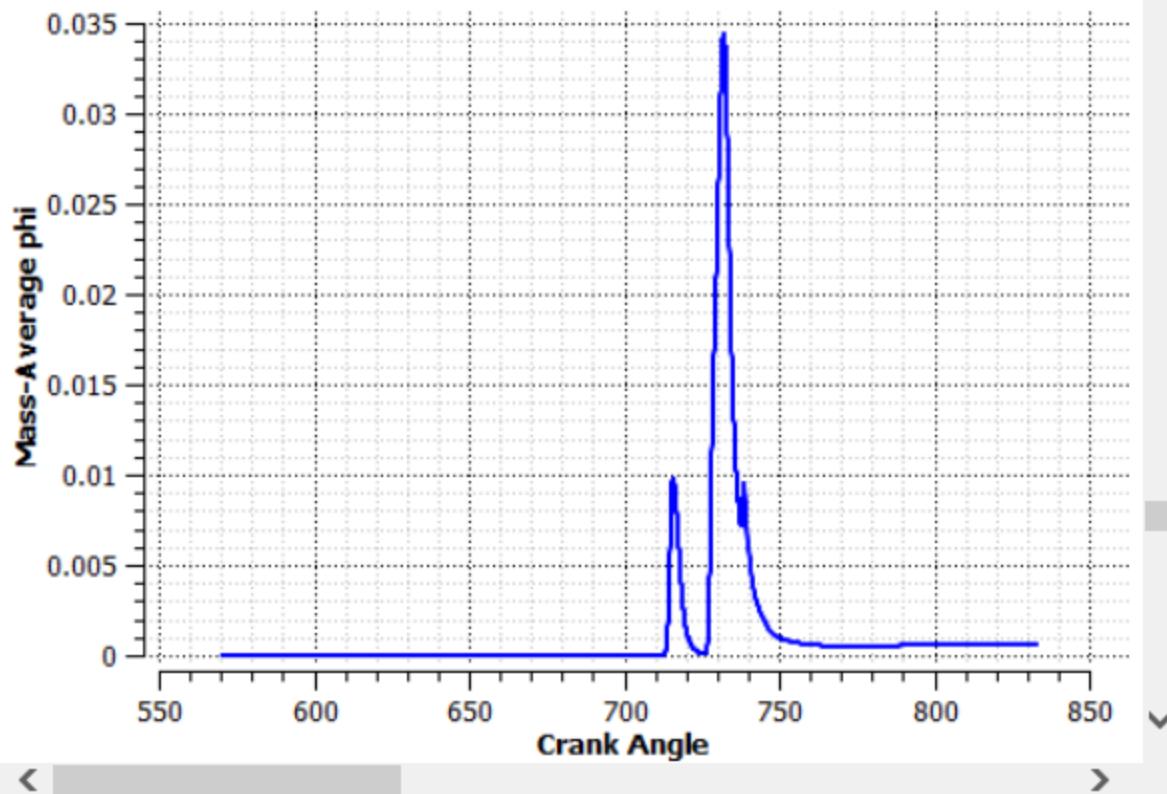


Chart 9. Monitor: Mass-Average Turbulent Kinetic Energy (k) (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

**Monitor: Mass-Average Turbulent Kinetic Energy (k)
(ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)**

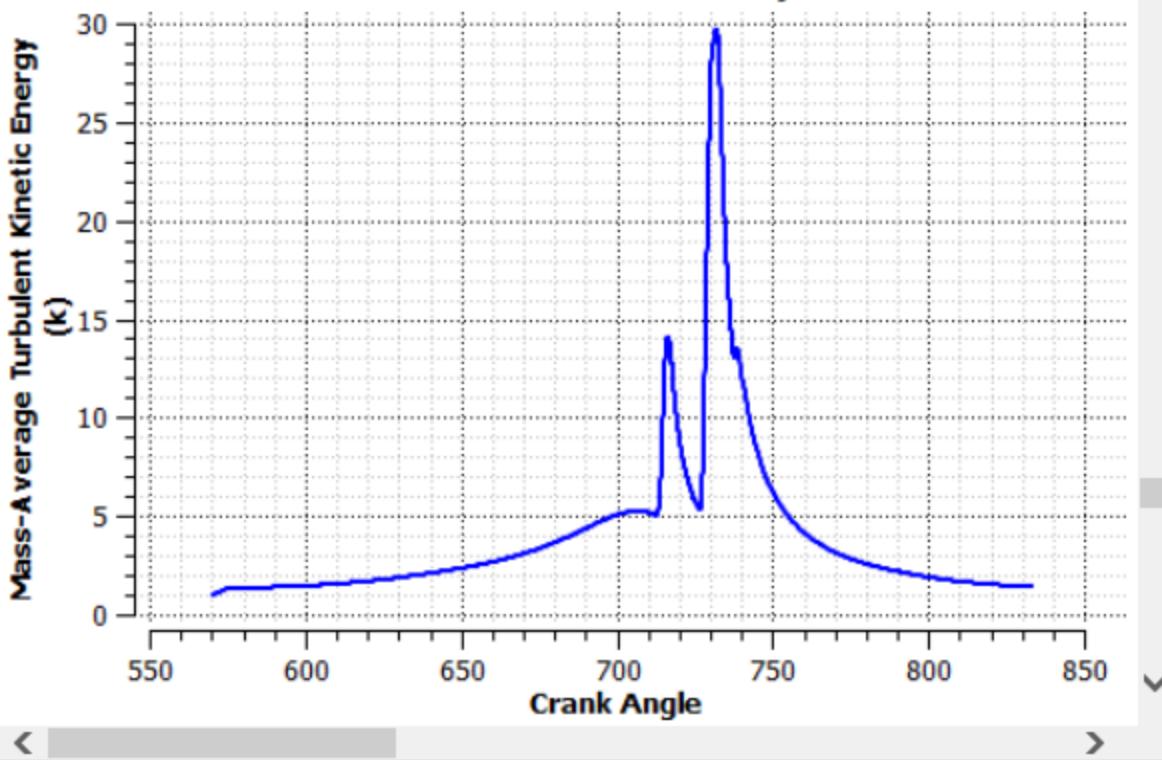
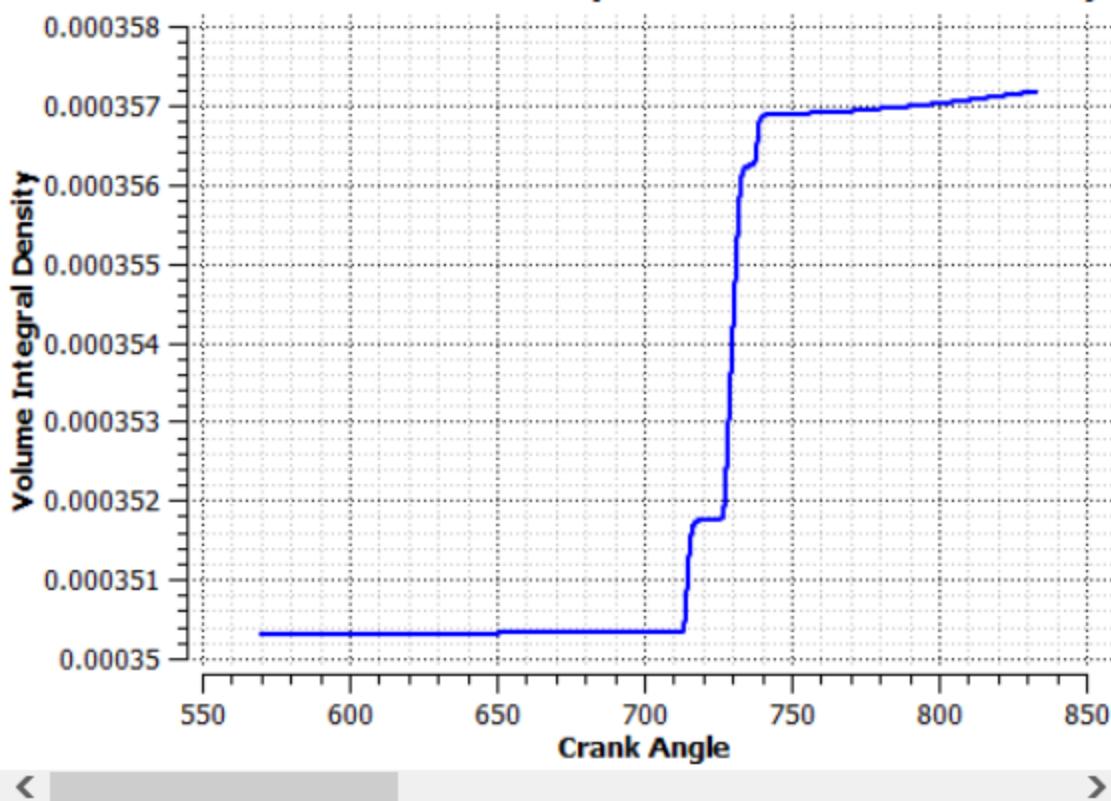


Chart 10. Monitor: Volume Integral Density (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

**Monitor: Volume Integral Density (ice-fluid-piston
ice-fluid-chamber-top ice-fluid-chamber-bottom)**



- The report also includes the plots of **Max Static Pressure**, **Max Static Temperature**, and **Max Velocity Magnitude** on the surfaces **ice-fluid-piston**, **ice fluid chamber-top** and **ice fluid chamber-bottom** at different crank angles.

Chart 11. Monitor: Max Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

Monitor: Max Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

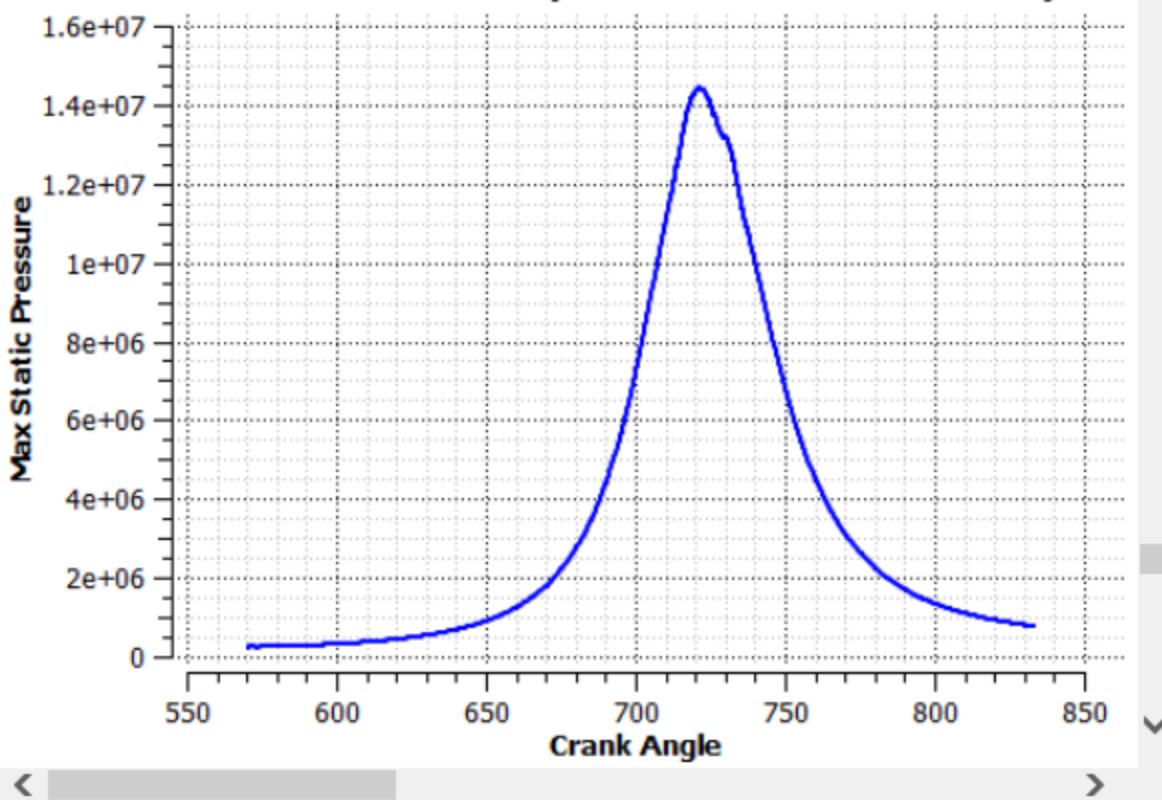


Chart 12. Monitor: Max Static Temperature (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

Monitor: Max Static Temperature (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

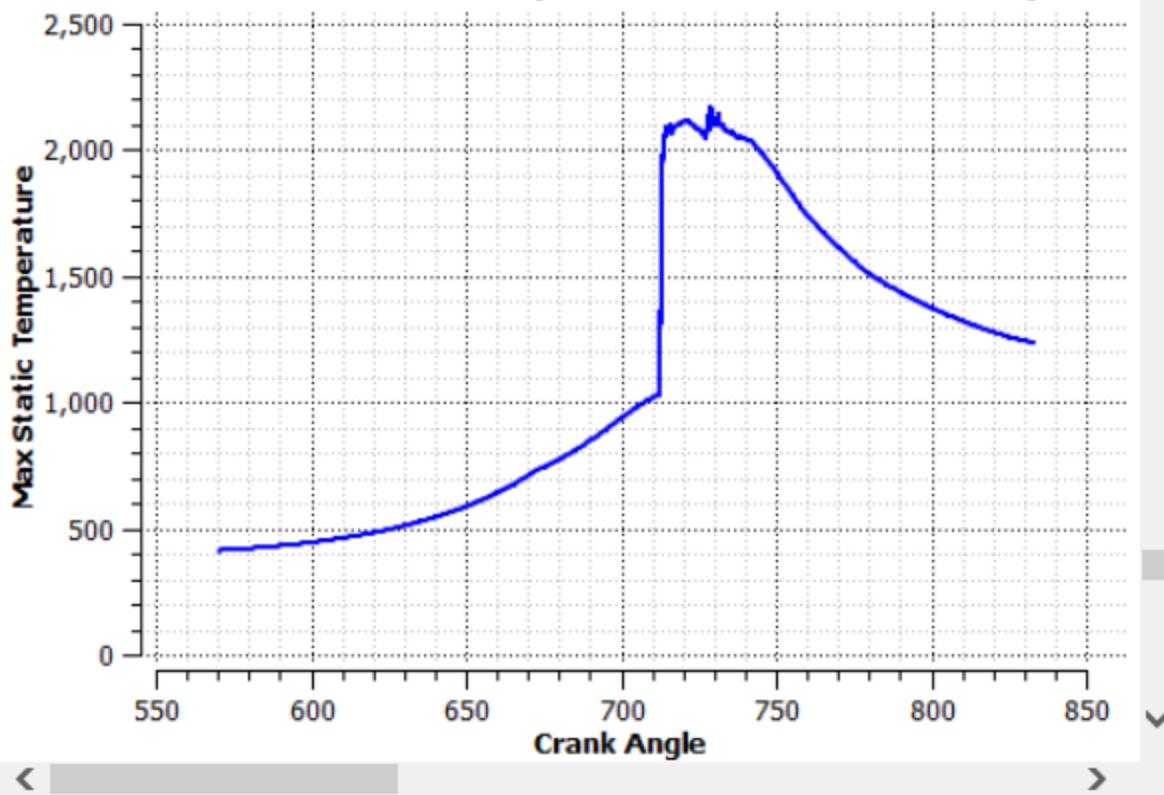
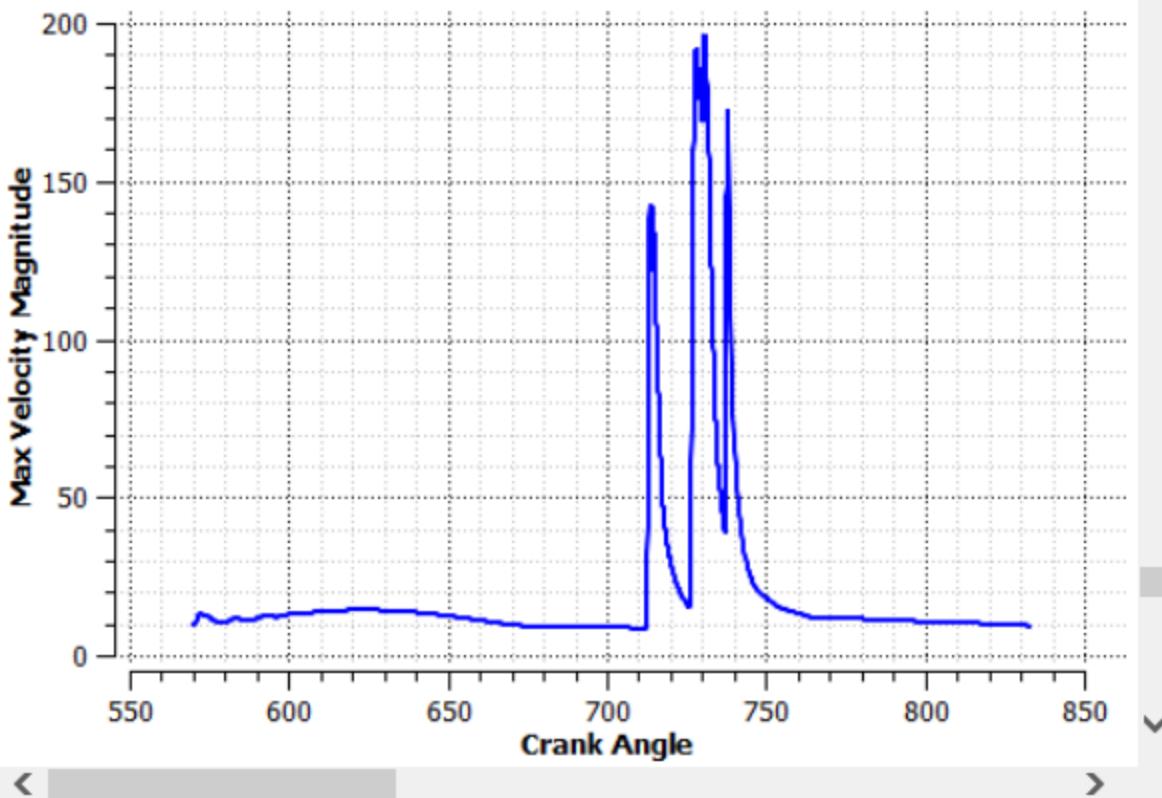
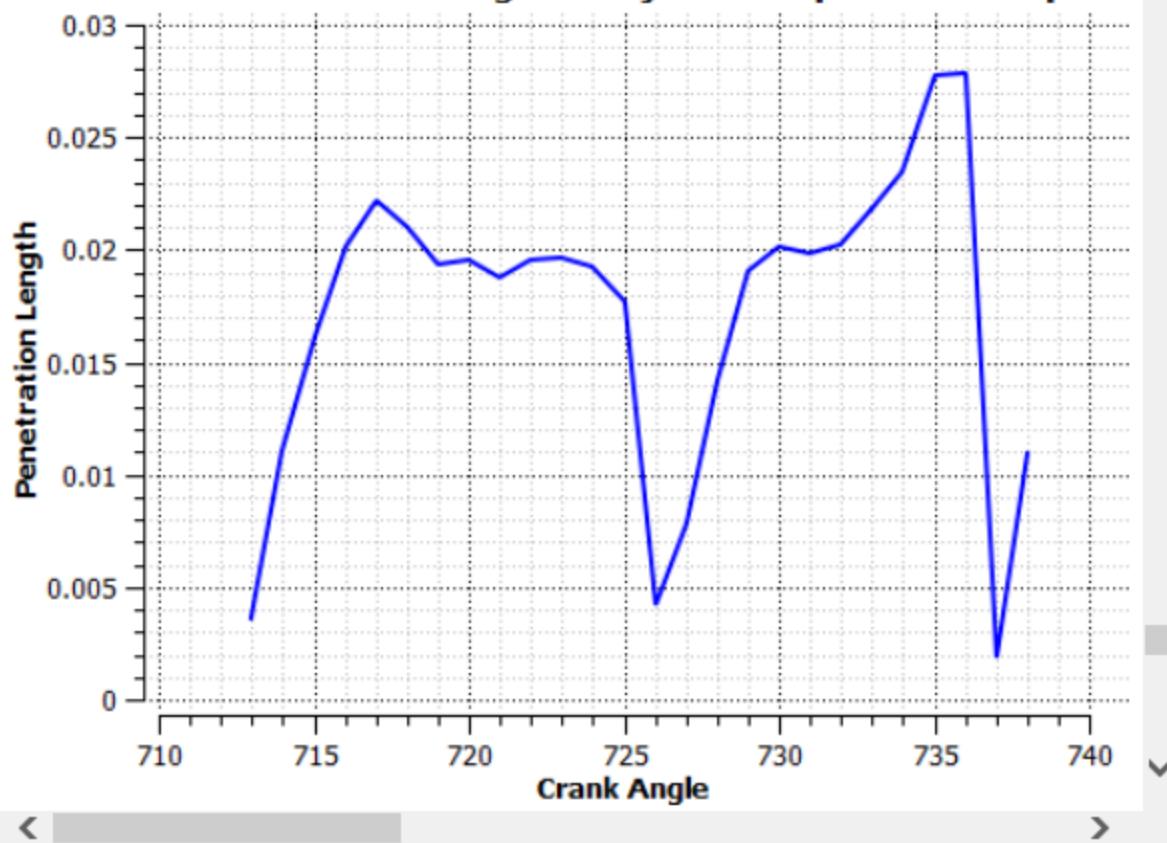


Chart 13. Monitor: Max Velocity Magnitude (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

Monitor: Max Velocity Magnitude (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)



- Chart plotting the **Penetration length of injection-0 per Time Step** is also included in the report..

Chart 14. Penetration length of injection-0 per Time Step**Penetration length of injection-0 per Time Step**

- Monitor plots of **Volume Average Static Pressure**, **Volume Average Static Temperature**, and **Volume Static Pressure** against **Crank Angle** can be checked in the report.

Chart 15. Monitor: Volume-Average Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

Monitor: Volume-Average Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

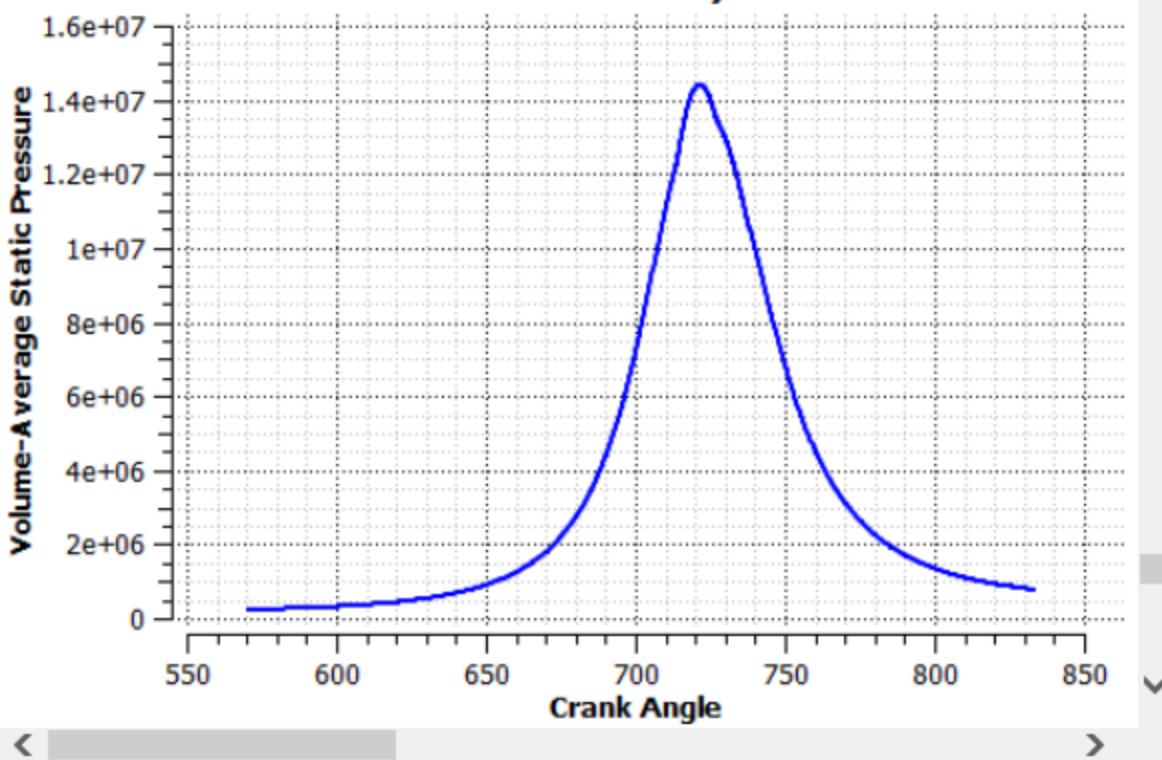


Chart 16. Monitor: Volume-Average Static Temperature (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

Monitor: Volume-Average Static Temperature (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

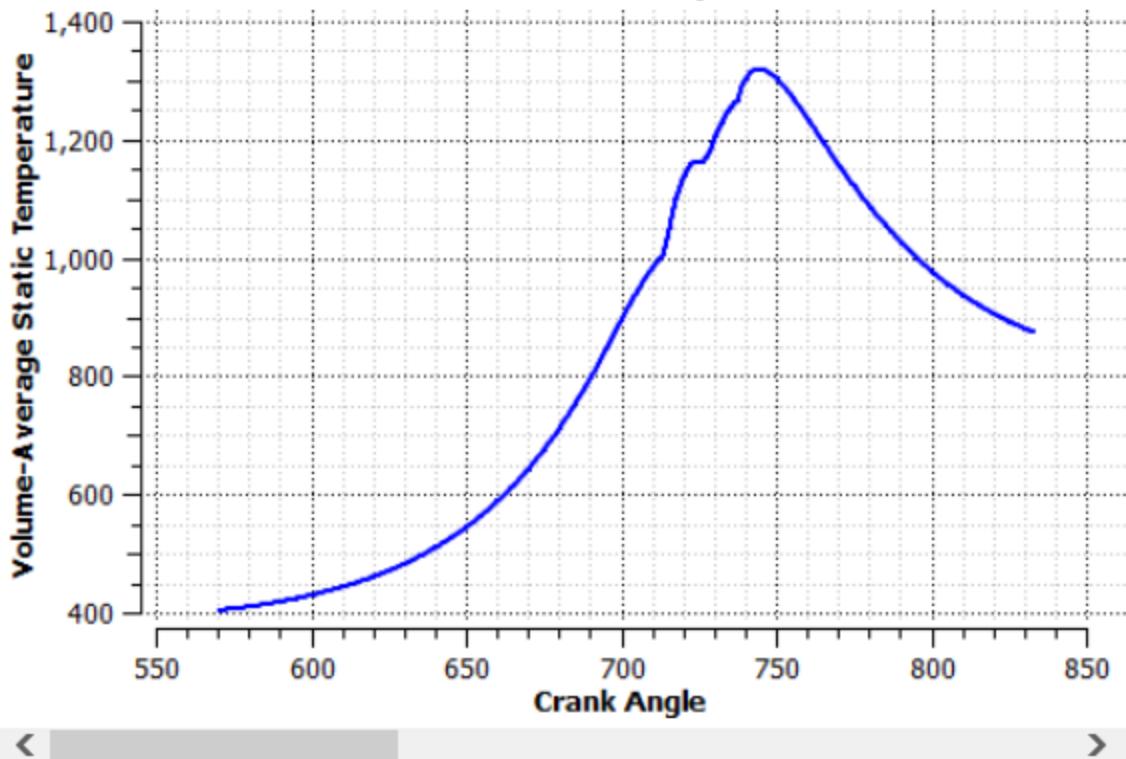
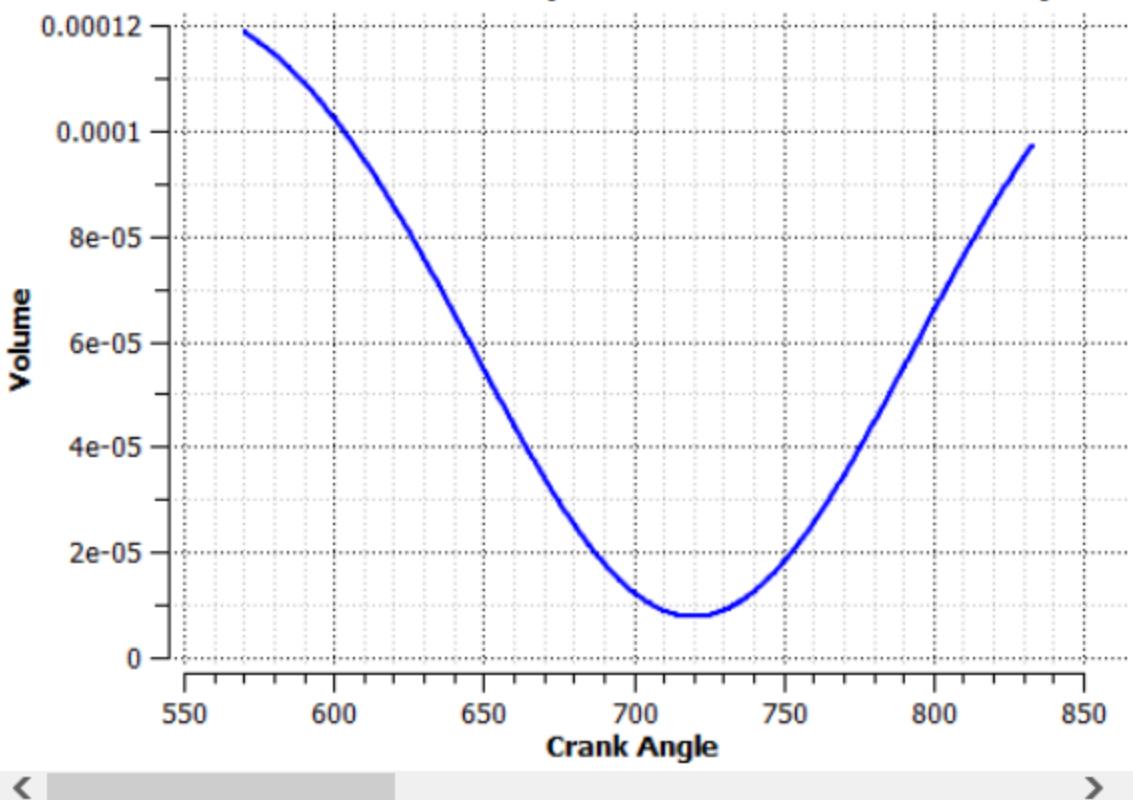


Chart 17. Monitor: Volume Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)

Monitor: Volume Static Pressure (ice-fluid-piston ice-fluid-chamber-top ice-fluid-chamber-bottom)



- Monitor plots of **Total mass influid for all injections per Time Step**, **Total mass injected for all injections per Time Step**, and **Total mass evaporated for all injections per Time Step** against **Crank Angle** can be checked in the report.

Chart 18. Total mass influid for all injections per Time Step

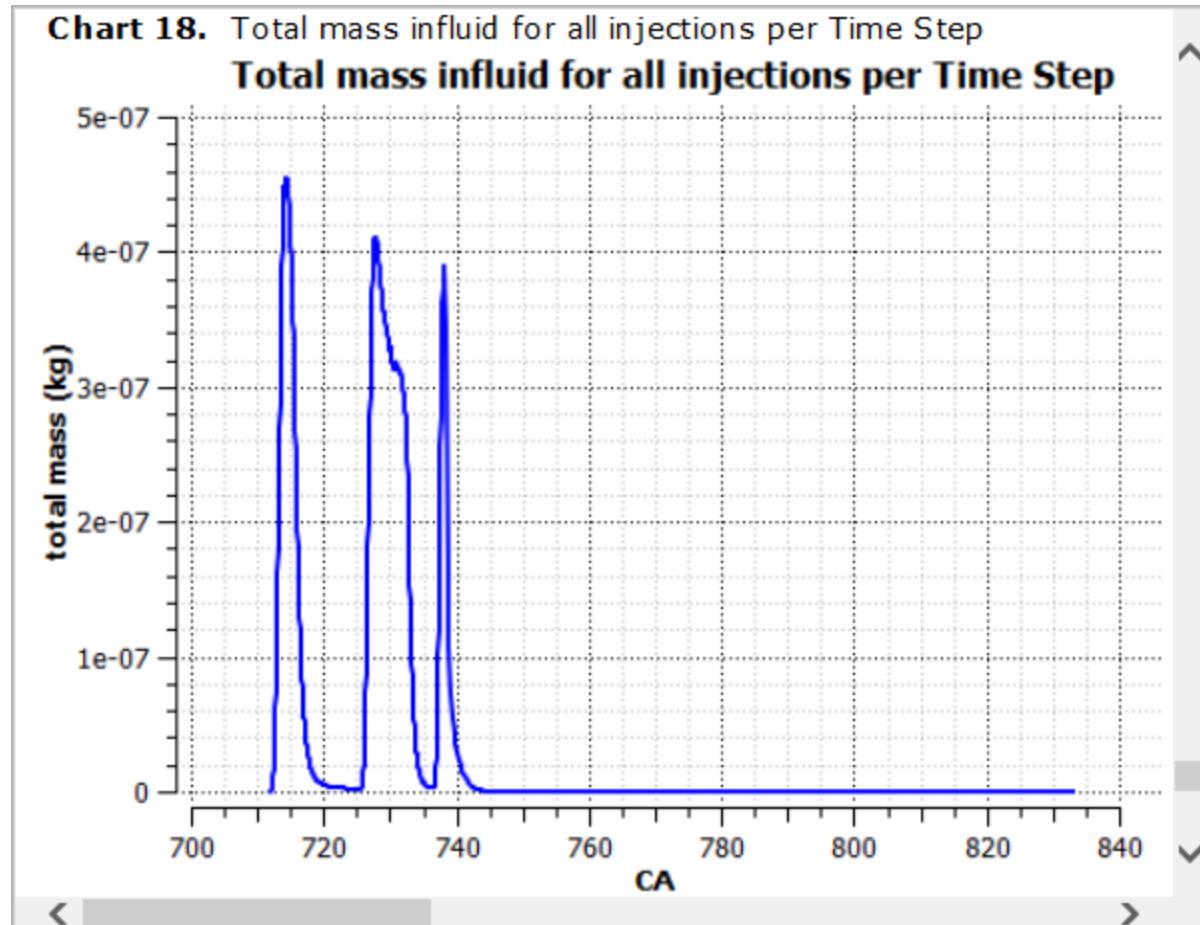


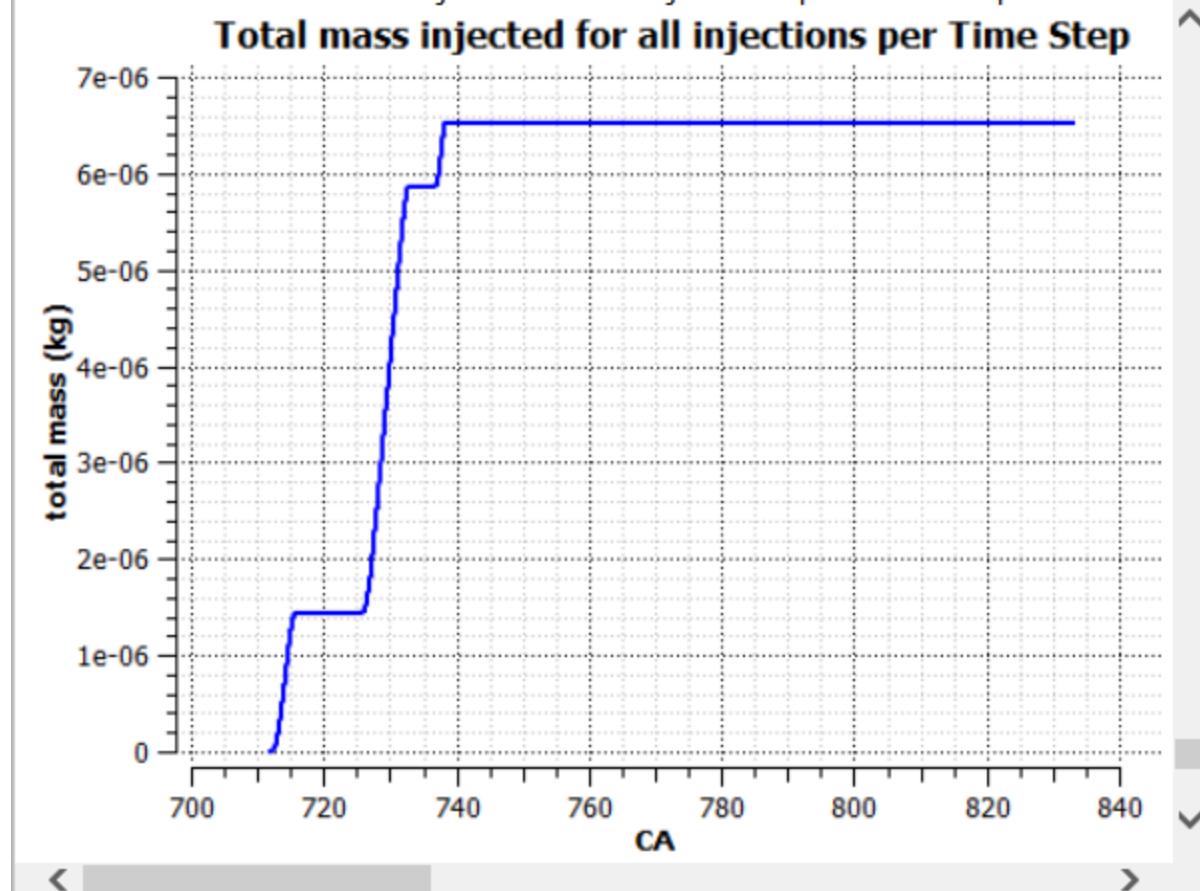
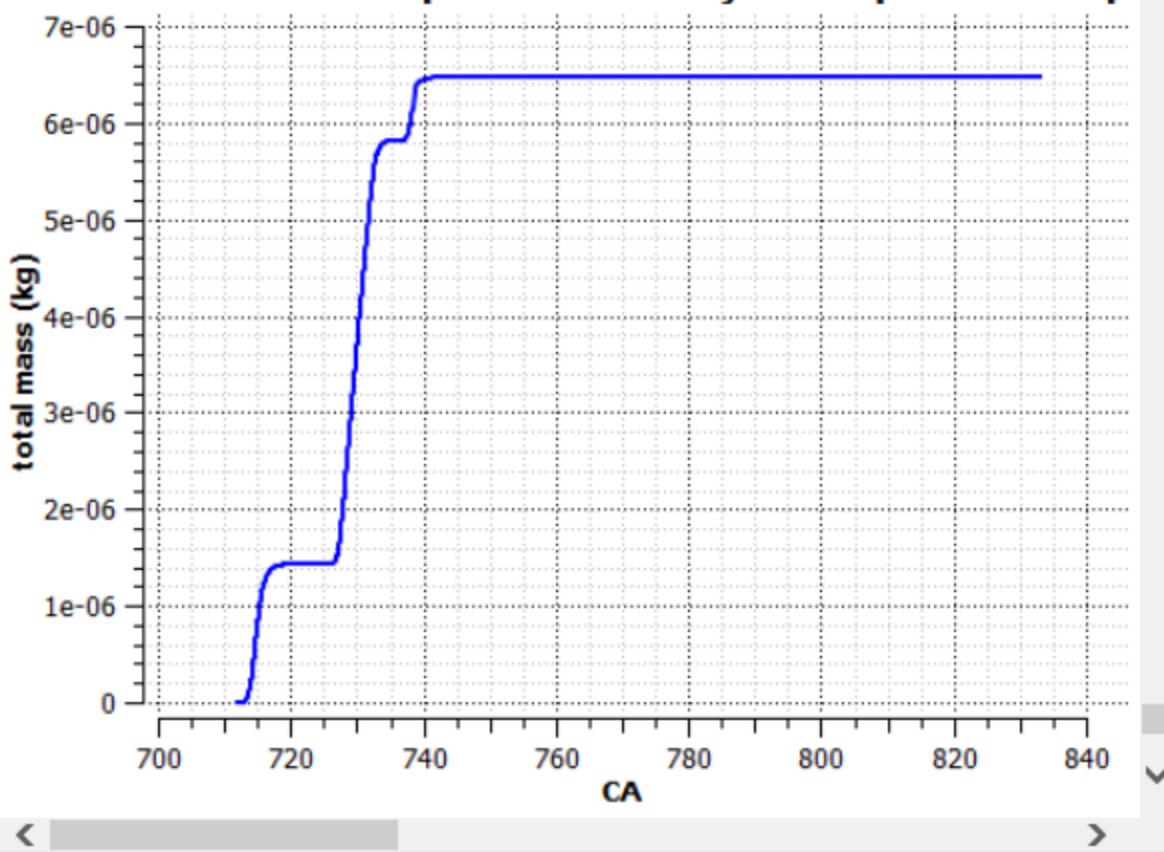
Chart 19. Total mass injected for all injections per Time Step

Chart 20. Total mass evaporated for all injections per Time Step

Total mass evaporated for all injections per Time Step

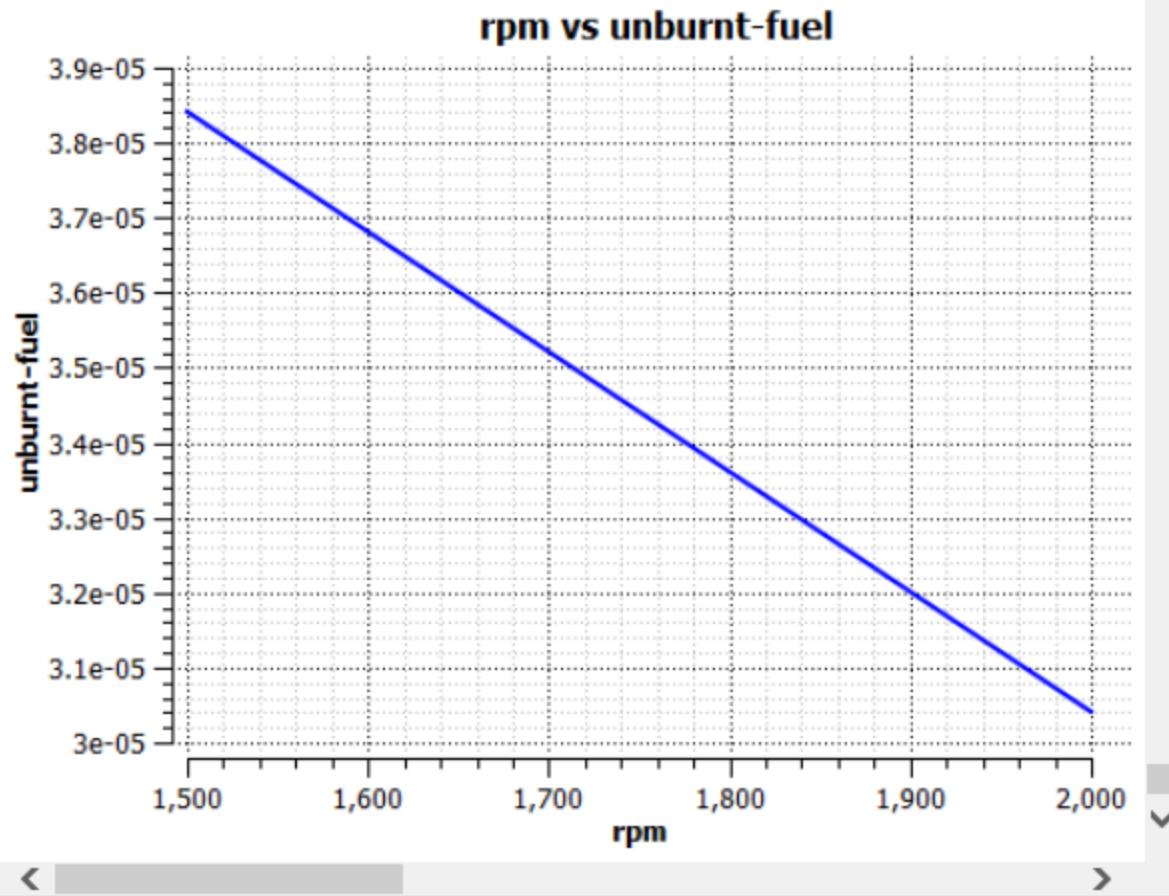


- Lastly in the report you can see the chart of **rpm vs unburnt-fuel**.

5. Design Points Report

5.1. Design Points Parameter values Charts

Chart 21. rpm vs unburnt-fuel



This concludes the tutorial which demonstrated the setup and solution for the combustion simulation of a sector of an IC engine.

3.8. Summary

In this tutorial Diesel Unsteady Flamelet (DUFL) non-premixed combustion model was used to simulate turbulent combustion process. Pressure trace and heat release rate were examined. You also learned how to use parametric system in Workbench for varying engine rpm and examining its effect on fuel burning.

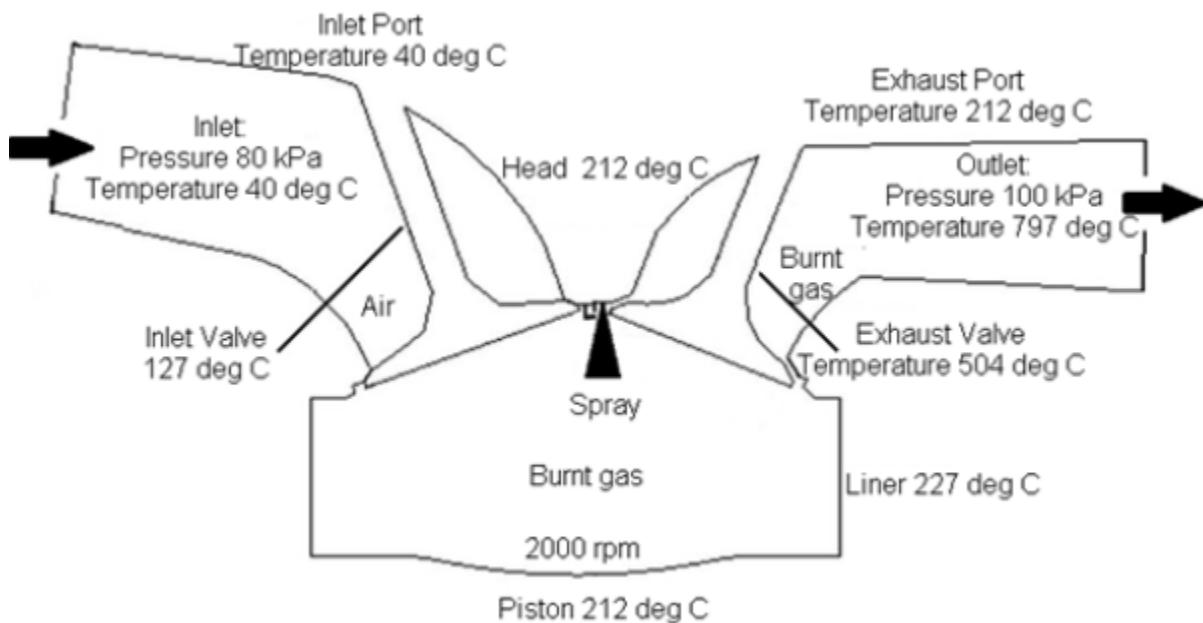
3.9. Further Improvements

This tutorial presents streamlined workflow between pre-processing, solver and post processing for ease of diesel sector simulations and performing parametric analysis of engine operating variables. You may switch on pollutant models and compare Nox and Soot.

Chapter 4: Tutorial: Solving a Gasoline Direct Injection Engine Simulation

A three dimensional single cylinder CFD simulation, of a 4-stroke spray guided Gasoline Direct Injection (GDI) Spark Ignition (SI) engine, is performed in this tutorial. Detailed boundary conditions are shown in [Figure 4.1: Problem Schematic \(p. 173\)](#). Engine simulation is started from Intake valve opening (IVO) and fuel is injected during the intake stroke. Homogeneous fuel air mixture is compressed and spark ignited 15° before compression Top Dead Center (TDC).

Figure 4.1: Problem Schematic



This is followed by power stroke and subsequent exhaust stroke. This tutorial illustrates the following steps in setting up and solving a Gasoline Direct Injection engine combustion simulation.

- Launch IC Engine system.
- Read an existing geometry into IC Engine system.
- Decompose the geometry.
- Define mesh setup and mesh the geometry.
- Run the simulation.
- Examine the results in the report.

This tutorial is written with the assumption that you are familiar with the IC Engine system and that you have a good working knowledge of ANSYS Workbench. For more information on IC Engines refer

to the user guide of Internal Combustion Engines in Workbench, on the ANSYS Customer Portal or in the ANSYS Help Viewer.

4.1. Preparation

4.2. Step 1: Setting the Properties

4.3. Step 2: Performing the Decomposition

4.4. Step 3: Meshing

4.5. Step 4: Setting up the Simulation

4.6. Step 5: Running the Solution

4.7. Step 6: Obtaining the Results

4.8. Summary

4.9. Further Improvements

4.1. Preparation

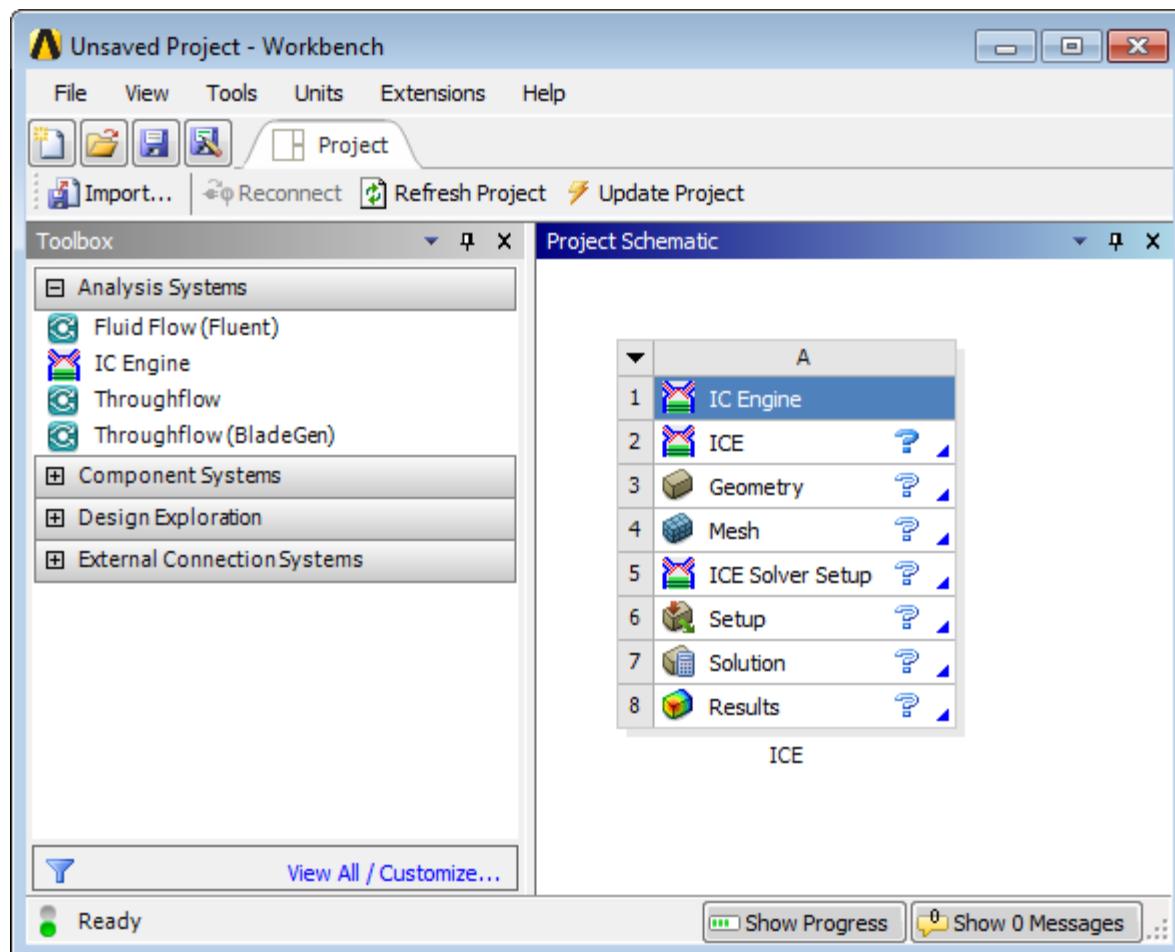
1. Copy the files (`tut_gdi_comb.x_t`, `comb_lift.prof`, `footprint.txt`, `massflowrate.csv`, and `velocity-0.7cd.csv`) to your working folder.

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

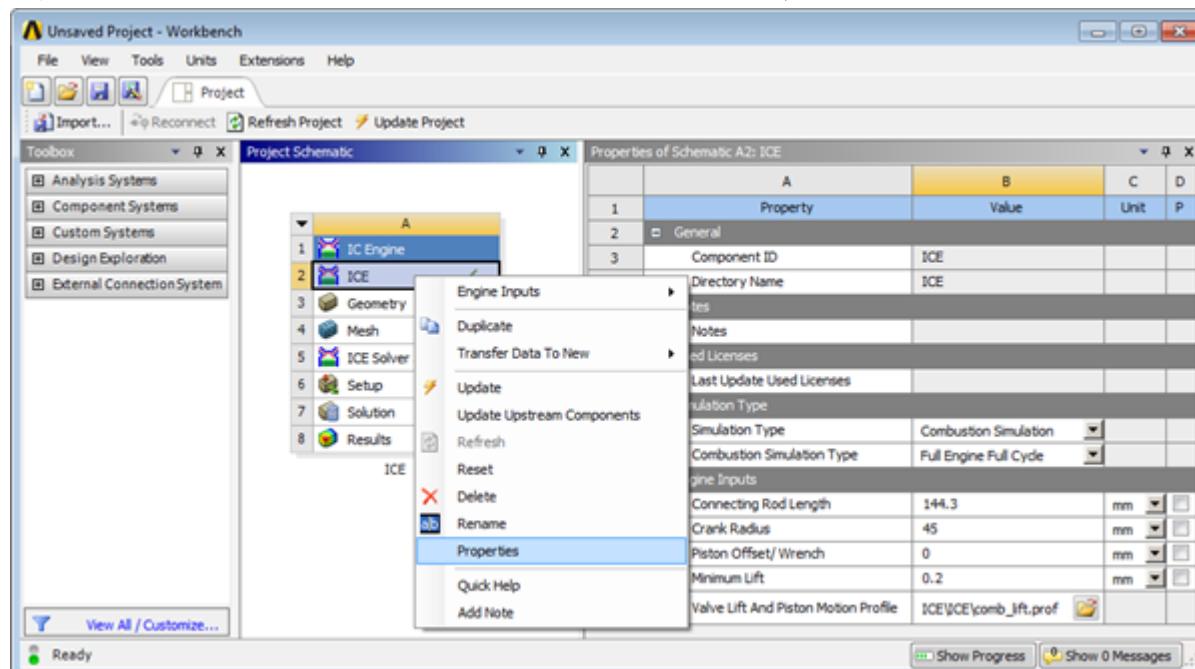
2. Start Workbench.

4.2. Step 1: Setting the Properties

1. Create an IC Engine analysis system in the Workbench interface by dragging or double-clicking on **IC Engine** under **Analysis Systems** in the **Toolbox**.



2. Right-click on **ICE**, cell 2, and click **Properties** (if it is not already visible) from the context menu.

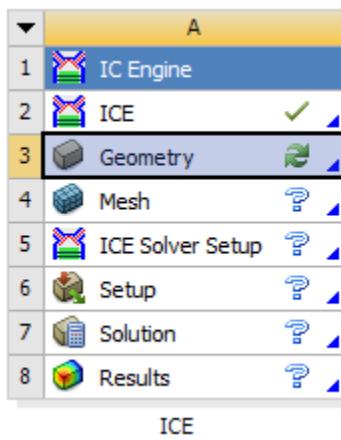


3. Select **Combustion Simulation** from the **Simulation Type** drop-down list.
 4. Select **Full Engine Full Cycle** from the **Combustion Simulation Type** drop-down list.

5. Enter 144.3 for **Connecting Rod Length**.
6. Enter 45 for **Crank Radius**.
7. Retain **0.2** for **Minimum Lift**.
8. Click **Browse File** next to **Lift Curve**. The **File Open** dialog box opens. Select the valve profile file `comb_lift.prof` and click **Open**.

4.3. Step 2: Performing the Decomposition

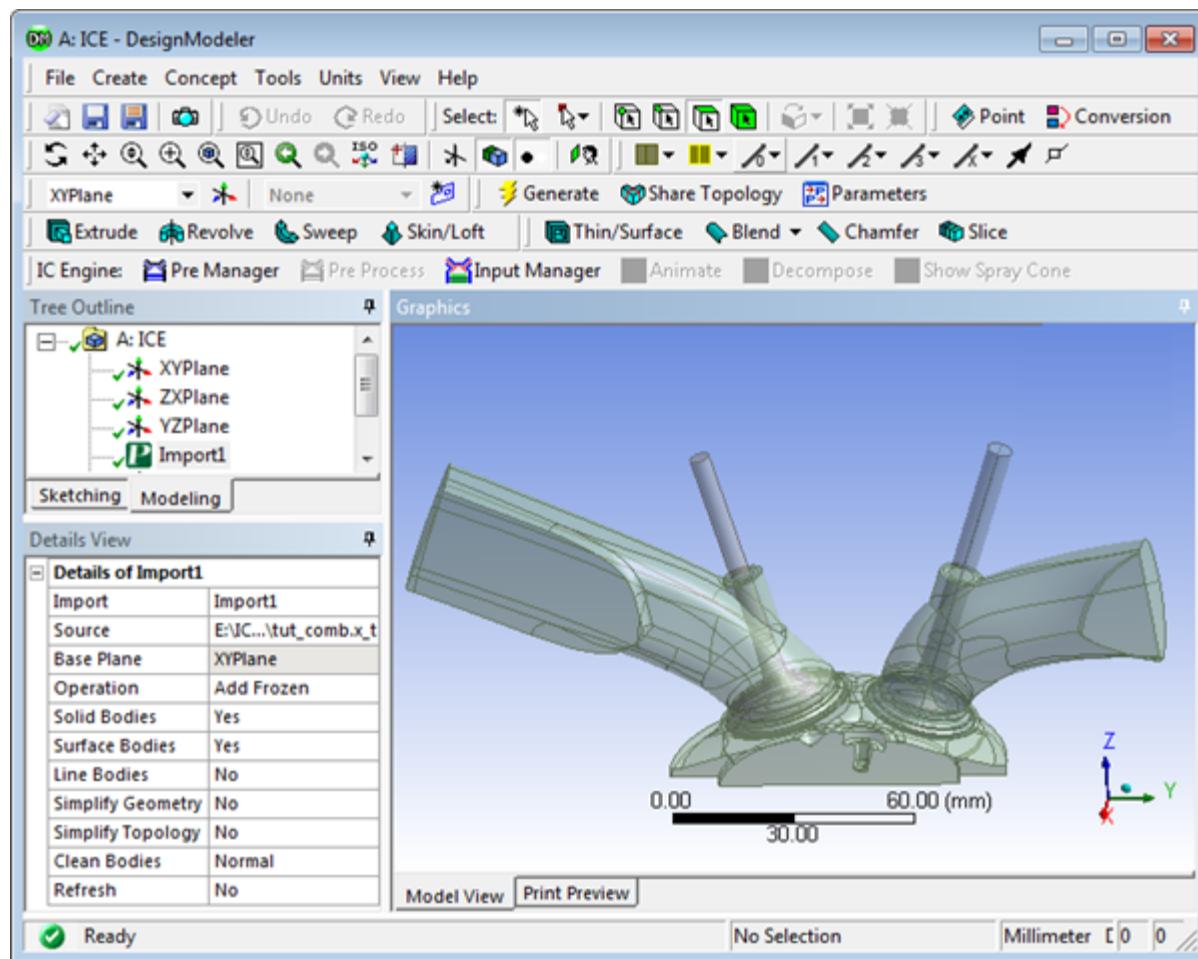
Here you will read the geometry and prepare it for decomposition. Double-click on the **Geometry** cell to open the DesignModeler.



1. Select **Millimeter** from the **Units** menu.
2. Import the geometry file, `tut_gdi_comb.x_t`.

File > Import External Geometry File...

3. Click **Generate** to complete the import feature.

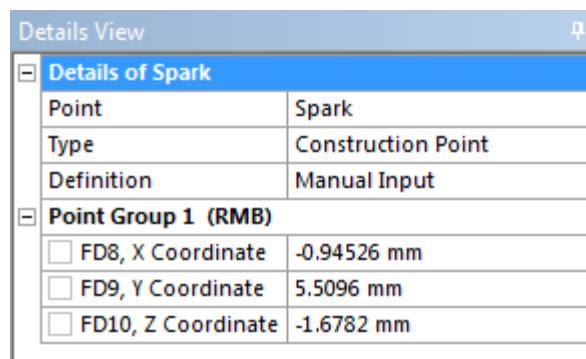


4. Create a spark point.

Note

Before creating a point ensure that the **Display Points** button  is pressed to enable you to see the point created in the graphics window.

Create > Point



- In the **Details View** enter Spark for **Point**.
- Select **Construction Point** from the **Type** drop-down list.

- c. Select **Manual Input** from the **Definition** drop-down list.
 - d. Under the **Point Group 1** enter -0.94526, 5.5096, and -1.6782 for **X Coordinate**, **Y Coordinate**, and **Z Coordinate**, respectively.
 - e. Right click and click **Generate** from the context menu. See [Figure 4.3: Spark, Beam and Footprint Points \(p. 180\)](#).
5. Similarly create a beam point before invoking **Input Manager**.

Create >Point

Details View	
Details of Beam	
Point	Beam
Type	Construction Point
Definition	Manual Input
Point Group 1 (RMB)	
<input type="checkbox"/> FD8, X Coordinate	0 mm
<input type="checkbox"/> FD9, Y Coordinate	-6 mm
<input type="checkbox"/> FD10, Z Coordinate	8.58 mm

- a. In the **Details View** enter Beam for **Point**.
 - b. Select **Construction Point** from the **Type** drop-down list.
 - c. Select **Manual Input** from the **Definition** drop-down list.
 - d. Under the **Point Group 1** enter 0, -6, and 8.58 for **X Coordinate**, **Y Coordinate**, and **Z Coordinate**, respectively.
 - e. Right click and click **Generate** from the context menu. See [Figure 4.3: Spark, Beam and Footprint Points \(p. 180\)](#).
6. To create a footprint point you need to create a plane first.

- a. Create a plane.

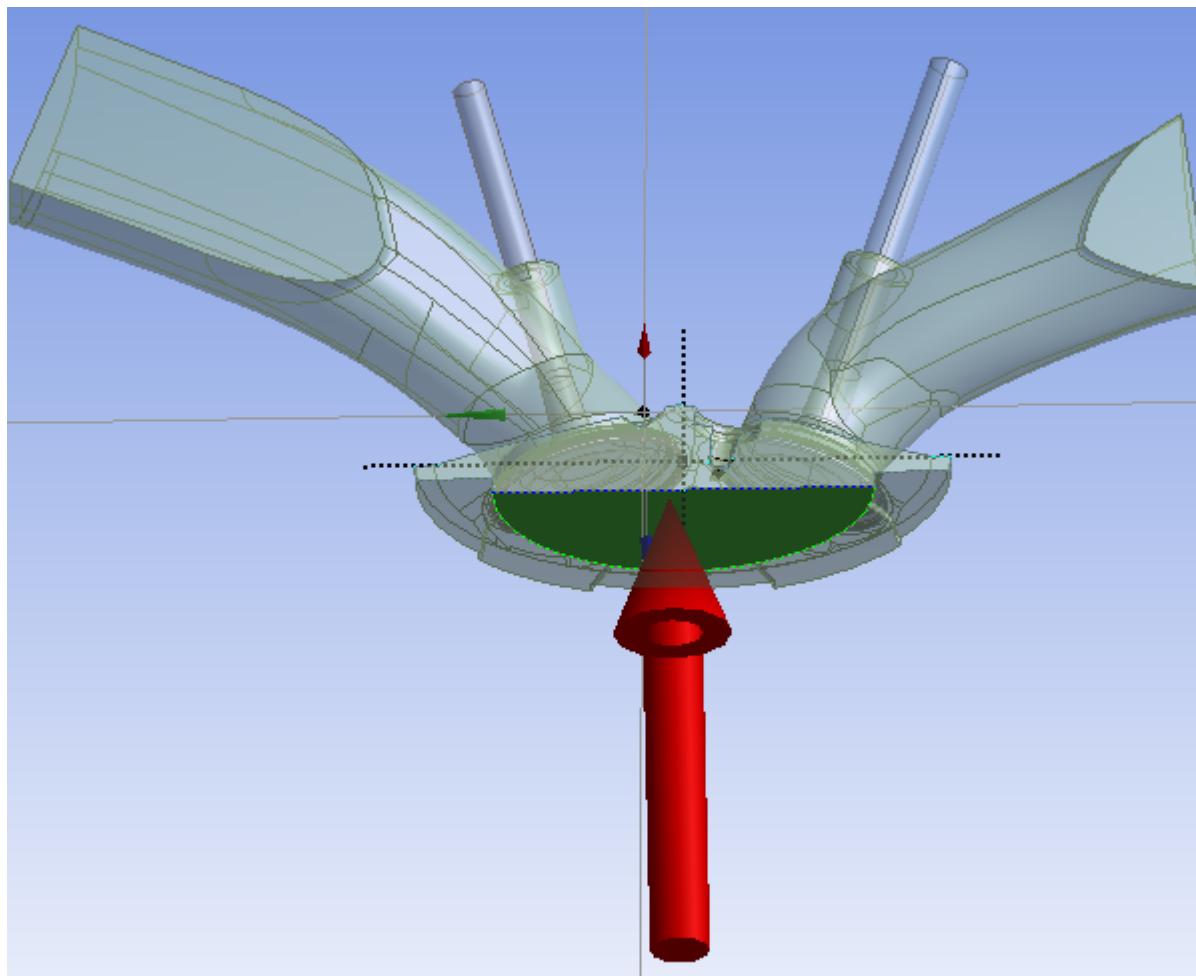
Create >New Plane

Details View	
Details of inj_plane	
Plane	inj_plane
Type	From Point and Normal
Base Point	PF Point
Normal Defined by	Face Normal
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

- b. In the **Details View** enter inj_plane for **Plane**.

- c. Select **From Point and Normal** from the **Type** drop-down list.
- d. Select the beam point created (**Beam**) for **Base Point**.
- e. Select the face and then the direction as shown in [Figure 4.2: Direction for Normal Defined by \(p. 179\)](#) for **Normal Defined by**.

Figure 4.2: Direction for Normal Defined by



- f. Right click and click **Generate** from the context menu.
- g. Create the footprint point.

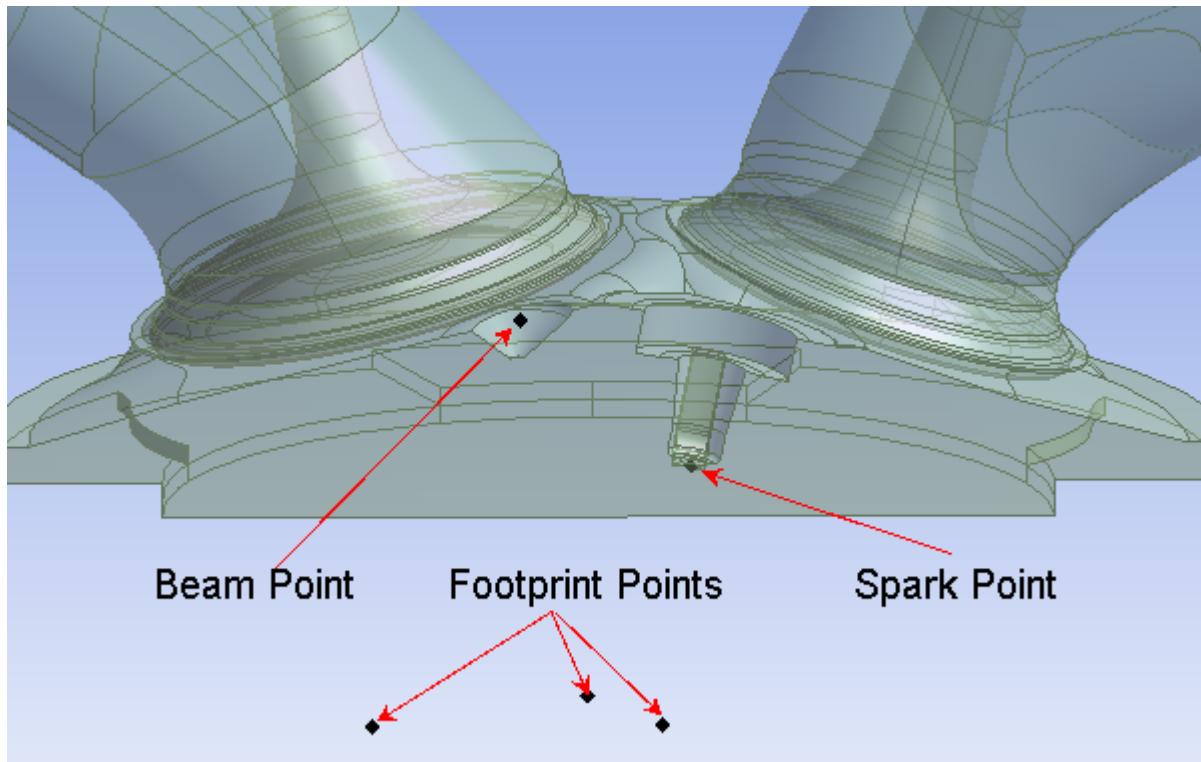
Create >Point

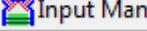
- h. In the **Details View** enter **Footprint** for **Point**.

Details View	
Details of Footprint	
Point	Footprint
Type	Construction Point
Definition	From Coordinates File
Coordinates File	E:\ICETutorials...\footprint.txt ...
Coordinates Unit	Millimeter
Base Plane	inj_plane
Tolerance	Normal
Refresh	No
# Points generated	3

- i. Select **Construction Point** from the **Type** drop-down list.
- j. Select **From Coordinates File** from the **Definition** drop-down list.
- k. Click the button next to **Coordinates File** text box (...) and select the file `footprint.txt`.
- l. Select the plane created, `inj_plane` for **Base Plane**.
- m. Retain the default settings for the rest, right click and click **Generate** from the context menu. See [Figure 4.3: Spark, Beam and Footprint Points \(p. 180\)](#).

Figure 4.3: Spark, Beam and Footprint Points



7. Click **Input Manager**  located in the **IC Engine** toolbar.

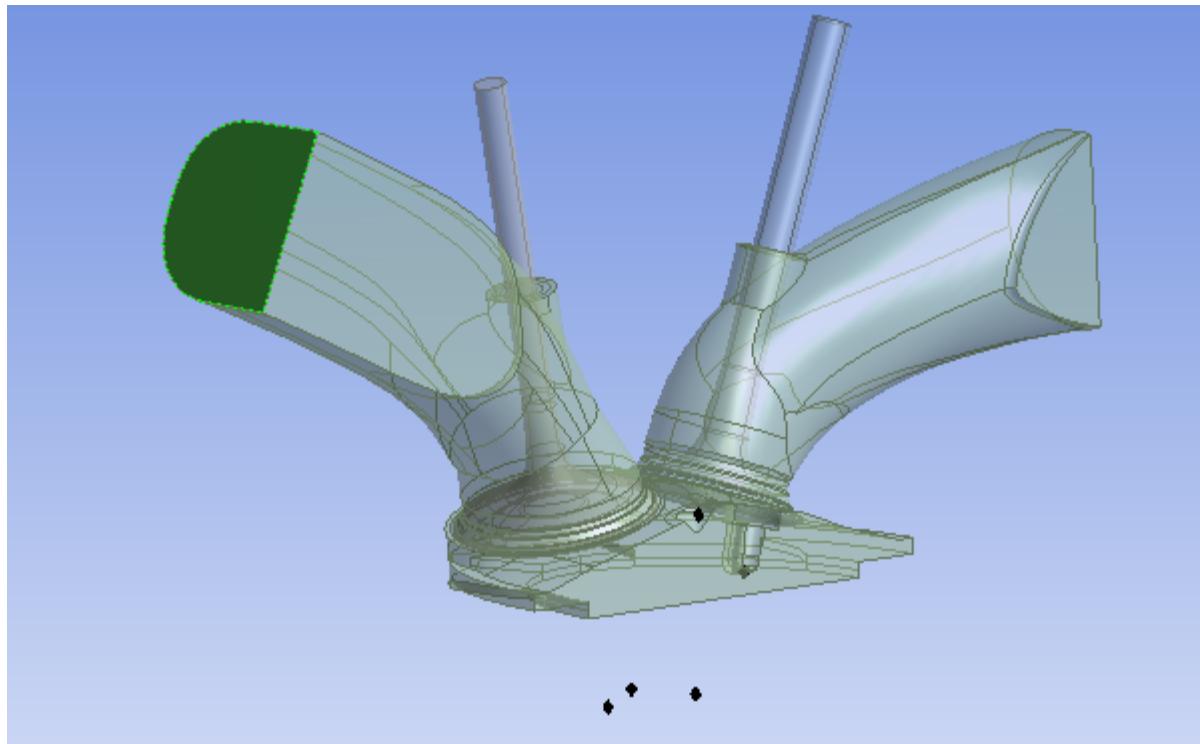
Details of InputManager1	
Name	InputManager1
Decomposition Position	Specified Angle
<input type="checkbox"/> FD1, Decomposition Angle	0 °
Inlet Faces	Not selected
Outlet Faces	Not selected
Cylinder Faces	Not selected
Symmetry Face Option	No
Topology Option	Full Topology
Crevice Option	No
Validate Compression Ratio	No
Spark Points	Not selected
+ IC Valves Data 1 (RMB)	
+ IC Animation Inputs (RMB)	
+ IC Injection 1 (RMB)	
- IC Advanced Options (RMB)	
V Layer Slice	Yes
V Layer Slice Angle	15 °
V Layer Approach	4 Layers
Piston Profile Option	No
Decompose Chamber	Yes
Decompose Chamber Inputs	Automatic

- a. Retain selection of **Specified Angle** for **Decomposition Position**.
- b. Enter 323 for **Decomposition Angle**.

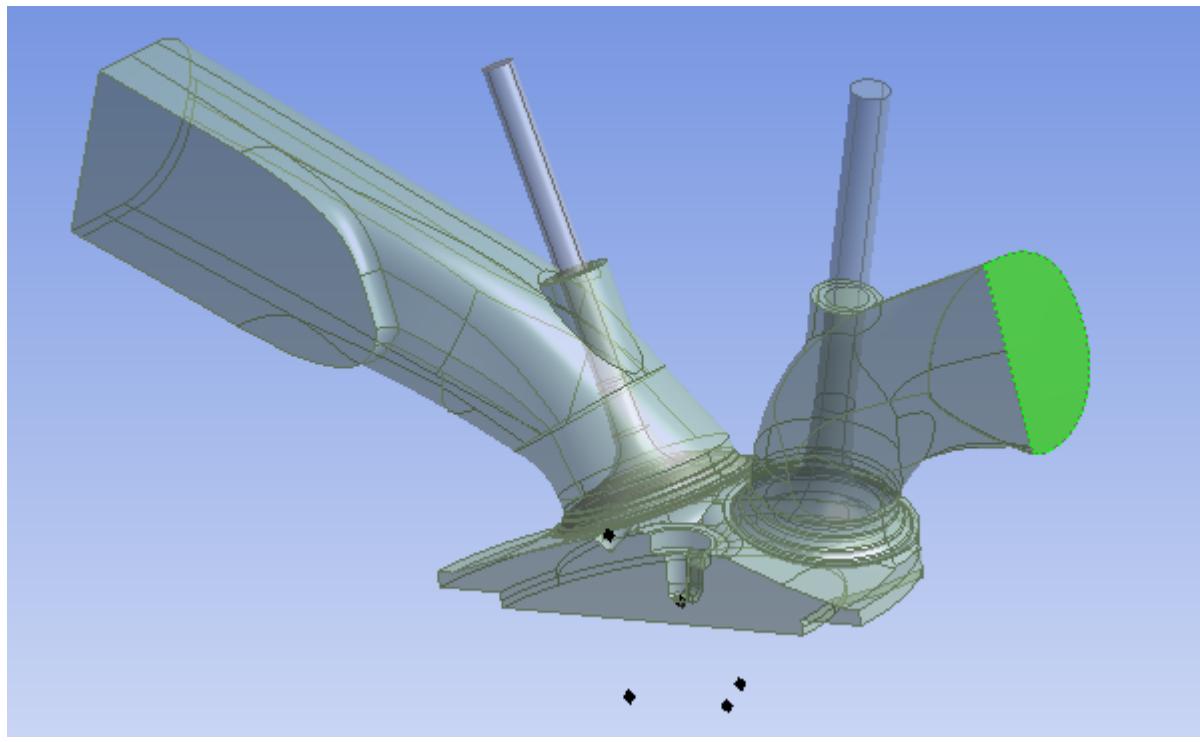
Note

This is just before IVO.

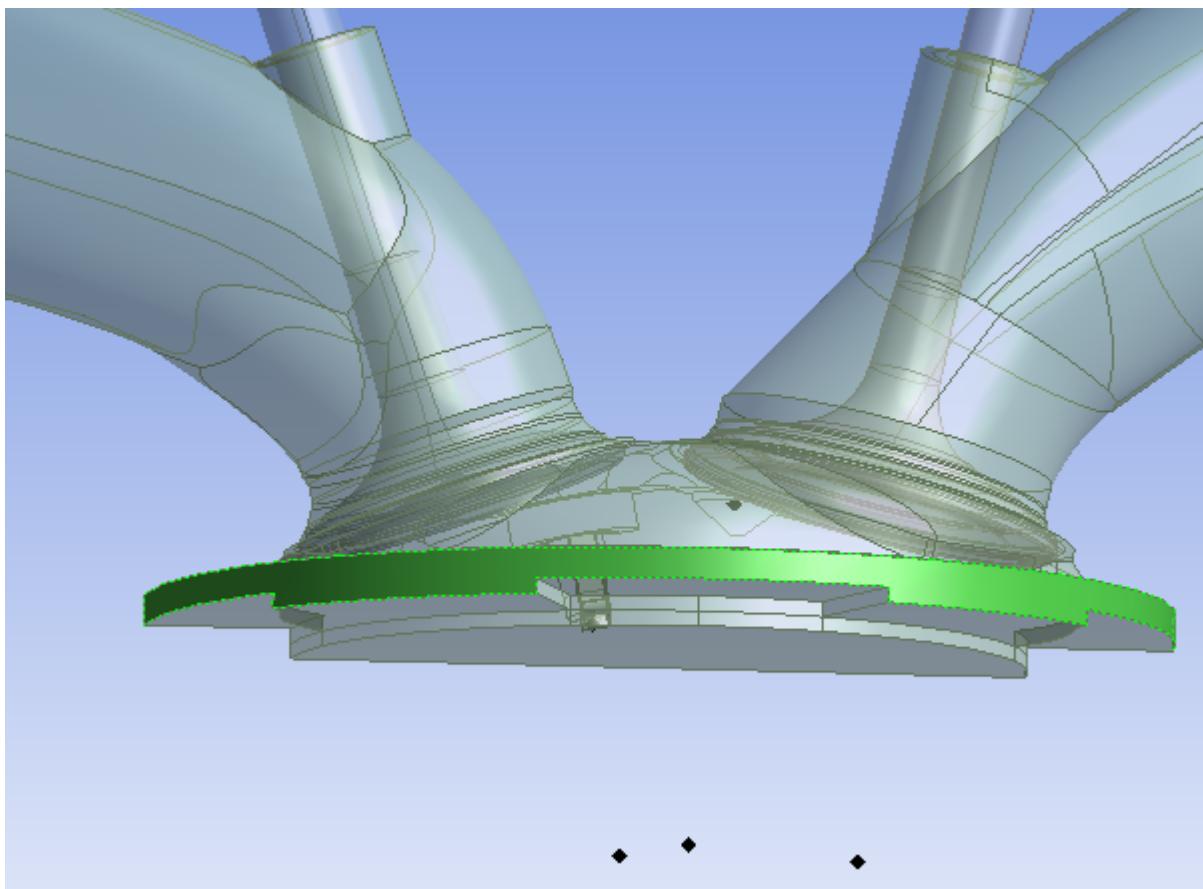
- c. Click next to **Inlet Faces**, select the face of the inlet valve and click **Apply**.



- d. Click next to **Outlet Faces**, select the face of the exhaust valve and click **Apply**.

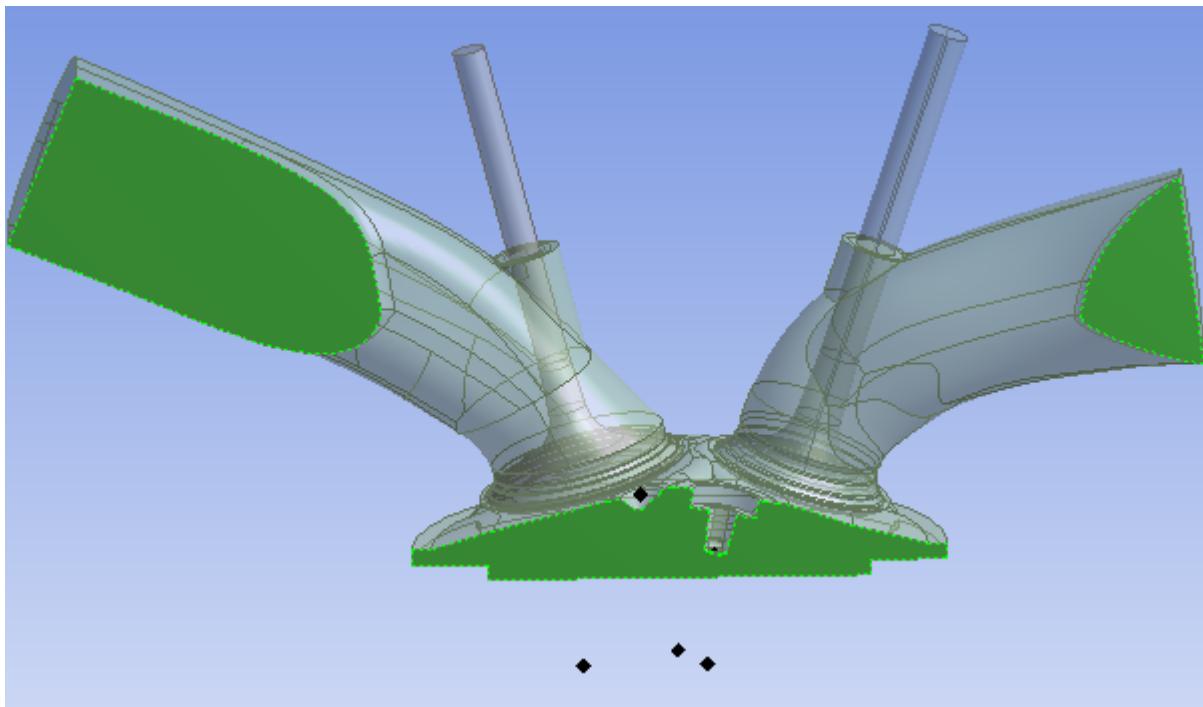


- e. Select the face as shown in [Figure 4.4: Cylinder Face \(p. 183\)](#) for **Cylinder Faces** and click **Apply**.

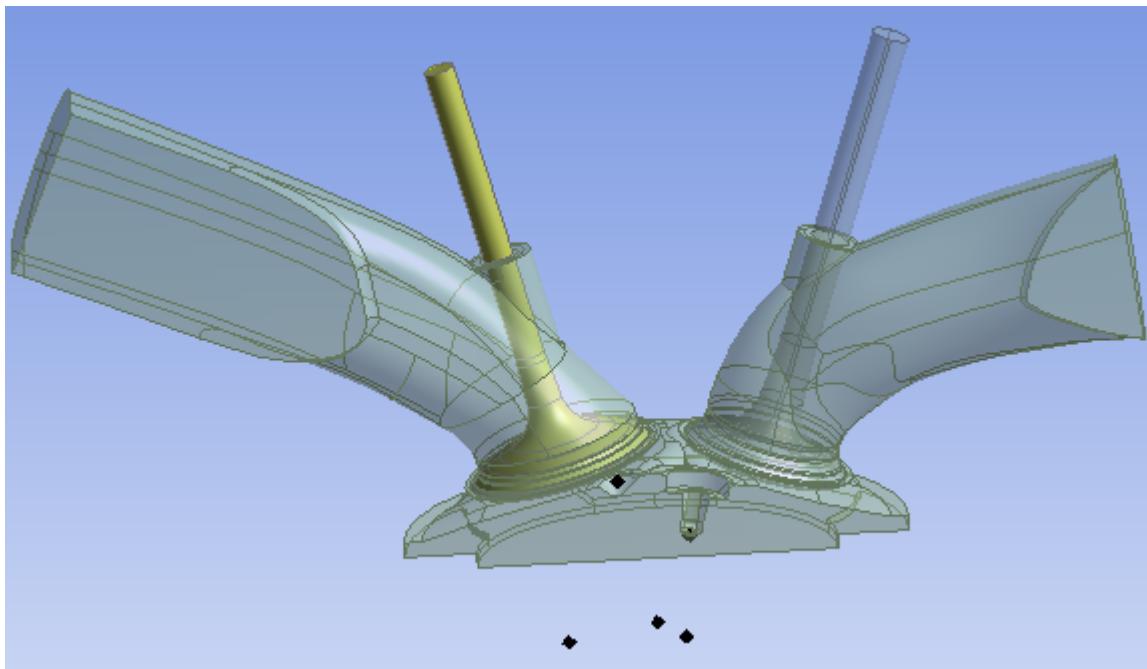
Figure 4.4: Cylinder Face

- f. **Symmetry Face Option** will be set to **Yes**. Retain the selection.
- g. Select the three faces shown in [Figure 4.5: Symmetry Faces \(p. 184\)](#) for **Symmetry Faces** and click **Apply**.

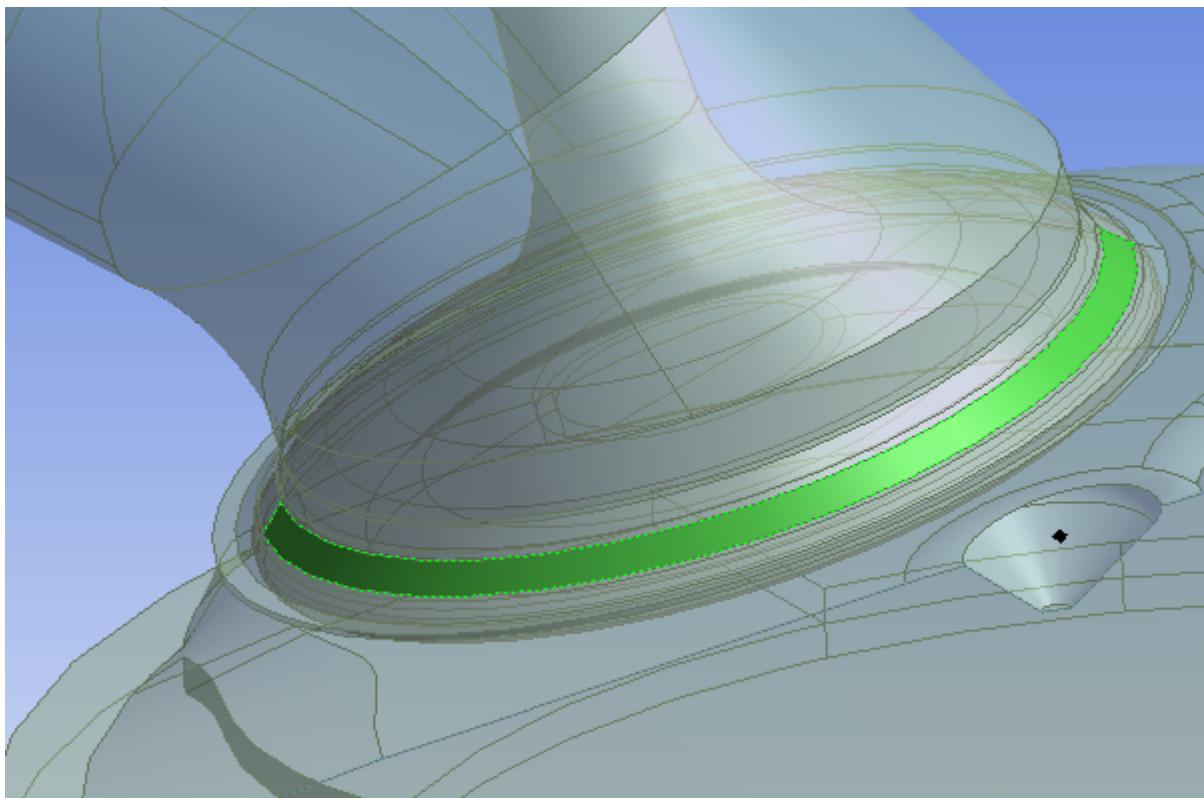
Figure 4.5: Symmetry Faces



- h. Retain the selection of **Full Topology** from the **Topology Option** drop-down list.
- i. Retain selection of **No** for **Crevice Option**.
- j. Select **Yes** for **Validate Compression Ratio**.
- k. Enter **10 . 25** for **Compression Ratio**.
- l. Select the **Spark** point created for **Spark Points**. See [Figure 4.3: Spark, Beam and Footprint Points \(p. 180\)](#).
- m. Select the valve body as shown in [Figure 4.6: Intake Valve \(p. 185\)](#) for **Valve Bodies** and click **Apply**.

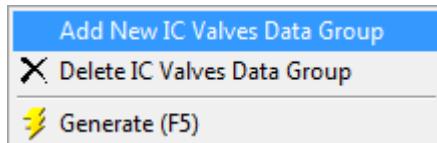
Figure 4.6: Intake Valve

- n. Select the valve seat face as shown in [Figure 4.7: Intake Valve Seat \(p. 185\)](#) for **Valve Seat Faces** and click **Apply**.

Figure 4.7: Intake Valve Seat

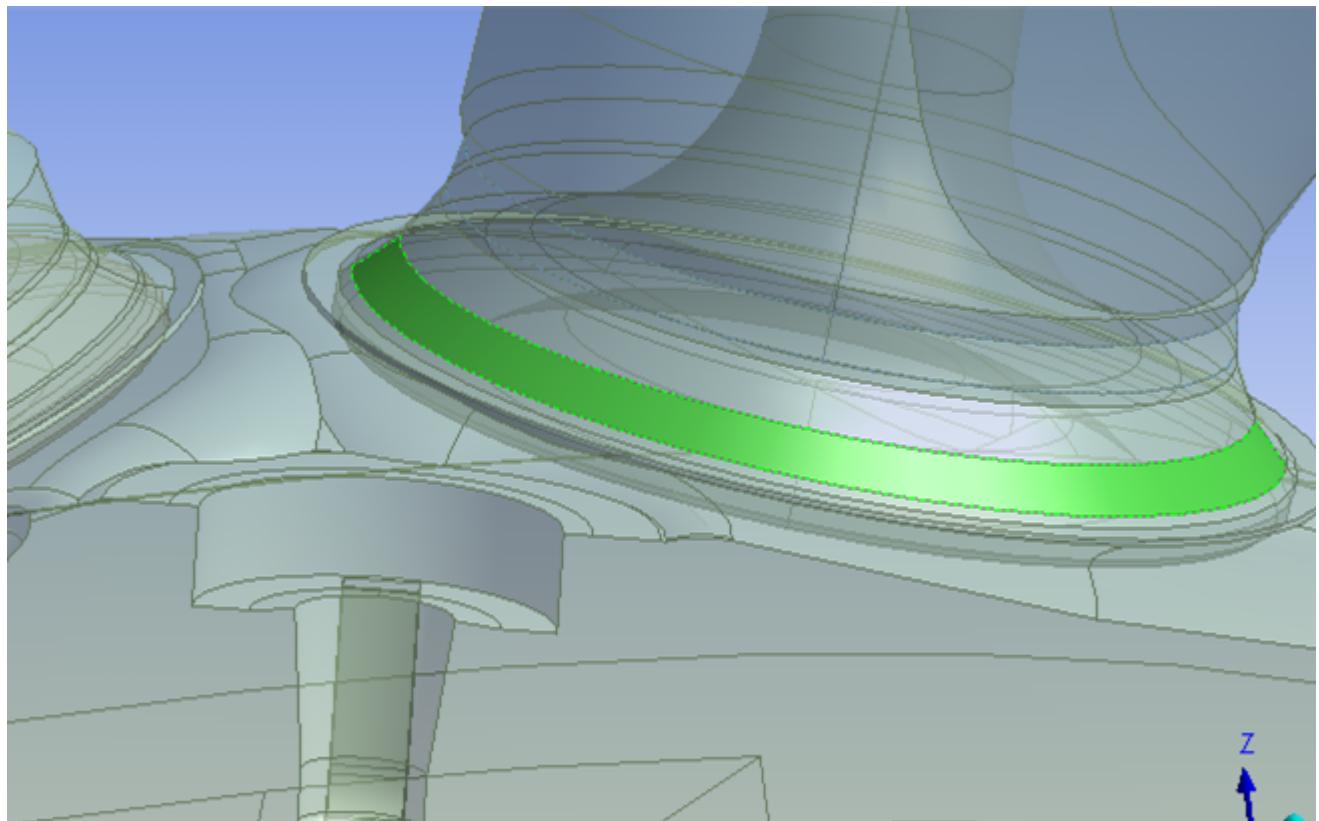
- o. Select **invalve1** from the **Valve Profile** drop-down list.

- p. Right-click on **IC Valves Data** in the **Details of InputManager** and select **Add New IC Valves Data Group** from the context menu.



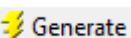
- q. In this **IC Valves Data** group following the steps for the intake valve, set the other valve body to **ExValve** and set its profile to **exvalve1**. Select the valve seat face of that valve as shown in [Figure 4.8: Exhaust Valve Seat \(p. 186\)](#).

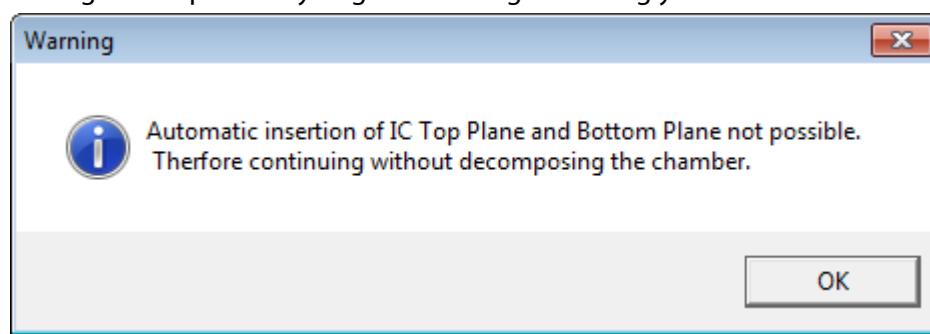
Figure 4.8: Exhaust Valve Seat



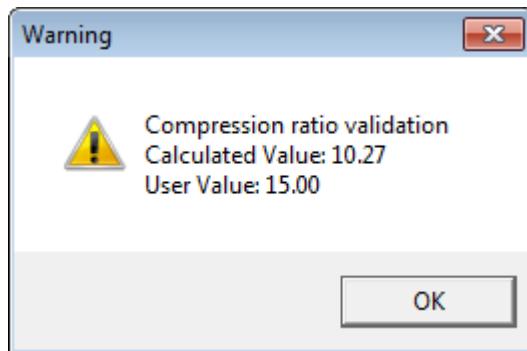
- r. Under **IC Injection 1** select the option **Beam Origin, Footprint** from the **Spray Location Option** drop-down list.
- s. Select the **Beam** point created before for **Injection Beam Origin**. See [Figure 4.3: Spark, Beam and Footprint Points \(p. 180\)](#). Click **Apply**.
- t. Select the 3 points after creating **Footprint** as shown in [Figure 4.3: Spark, Beam and Footprint Points \(p. 180\)](#) for **Footprint Point**.

Details View	
Details of InputManager1	
Name	InputManager1
Decomposition Position	Specified Angle
<input type="checkbox"/> FD1, Decomposition Angle	323 °
Inlet Faces	1 Face
Outlet Faces	1 Face
Cylinder Faces	1 Face
Symmetry Face Option	Yes
Symmetry Faces	3 Faces
Topology Option	Full Topology
Crevise Option	No
Validate Compression Ratio	Yes
Compression Ratio	10.25
Spark Points	1 Point
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	invalve1
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	exvalve1
IC Animation Inputs (RMB)	
IC Injection 1 (RMB)	
Spray Location Option	Beam Origin, Footprint
Injection Beam Origin	1 Point
Footprint Point	3 Points
IC Advanced Options (RMB)	

- u. After all the settings are done click **Generate** .
8. Click **Decompose** () located in the **IC Engine** toolbar).
9. During decomposition you get a warning informing you that the chamber will not be decomposed.



- The chamber will be decomposed between the **insert_angle** and the **delete_angle**. Click **OK** to continue.
10. Then you get a warning about the compression ratio validation.



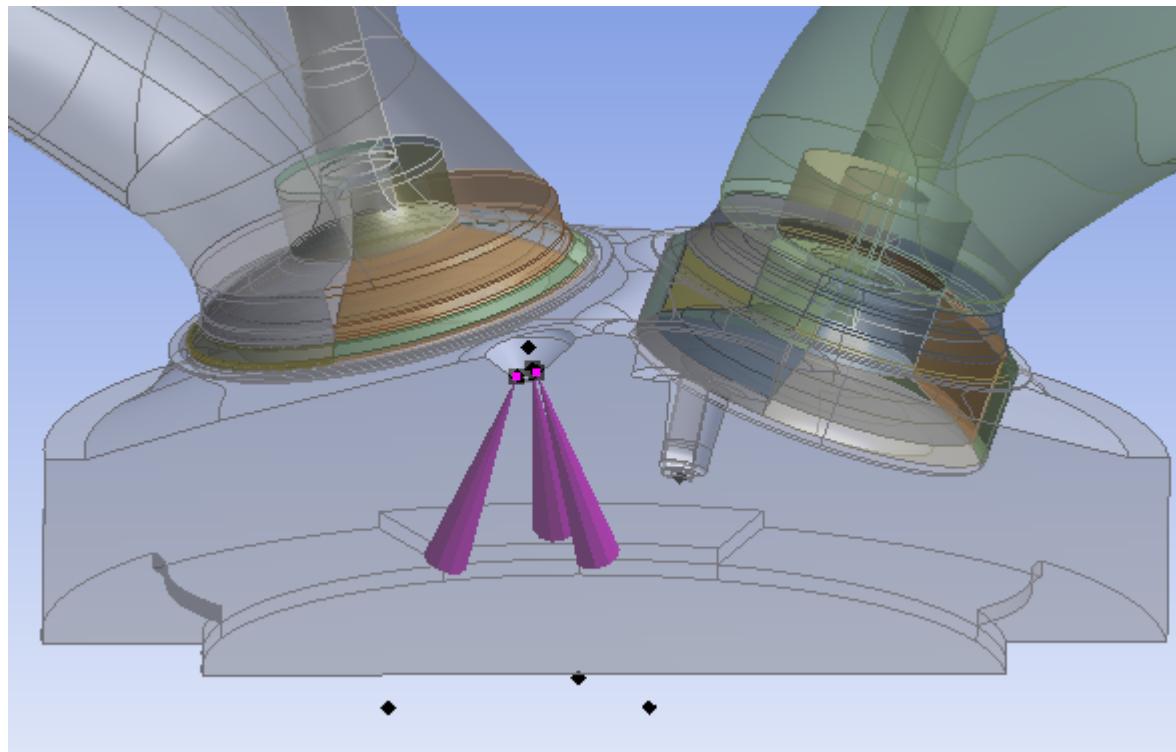
Click **OK** to continue.

Note

The decomposition process will take a few minutes.

11. Click **Show Spray Cone** to check the view of the injections.

Figure 4.9: Decomposed Geometry



12. Close the DesignModeler.

-
13. Save the project by giving it a proper name (`demo_comb.wbpj`).

Note

Once decomposition is completed you can open the `icSolverSettings.txt` file saved in the `~demo_comb_files\dp0\ICE\ICE` folder and check the **insert_angle** and the **delete_angle**.

4.4. Step 3: Meshing

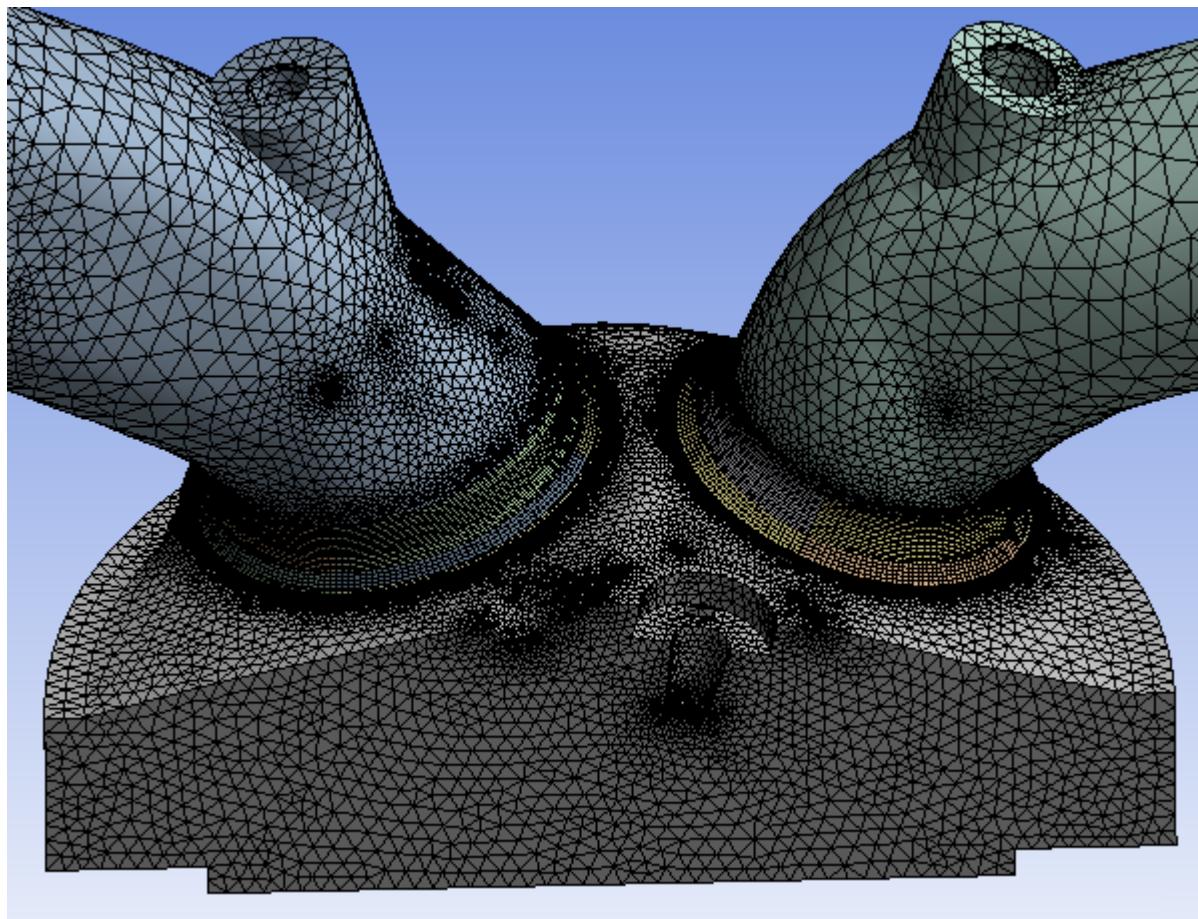
Here you will mesh the decomposed geometry.

1. Right-click on **Mesh**, cell 4, and click **Update** from the context menu. In a single step it will first create the mesh controls, then generate the mesh and finally update the mesh cell.
-

Note

If you want to check or change the mesh settings click **Edit Mesh Settings in Properties of Schematic A4: Mesh under IC Engine**. For this tutorial you are going to retain the default mesh settings. This meshing process will take a few minutes.

Figure 4.10: Meshed Geometry



2. Save the project.

File > Save

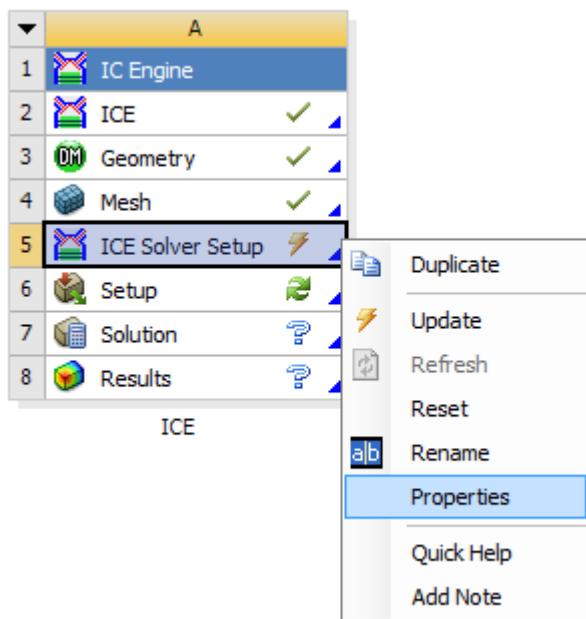
Note

It is a good practice to save the project after each cell update.

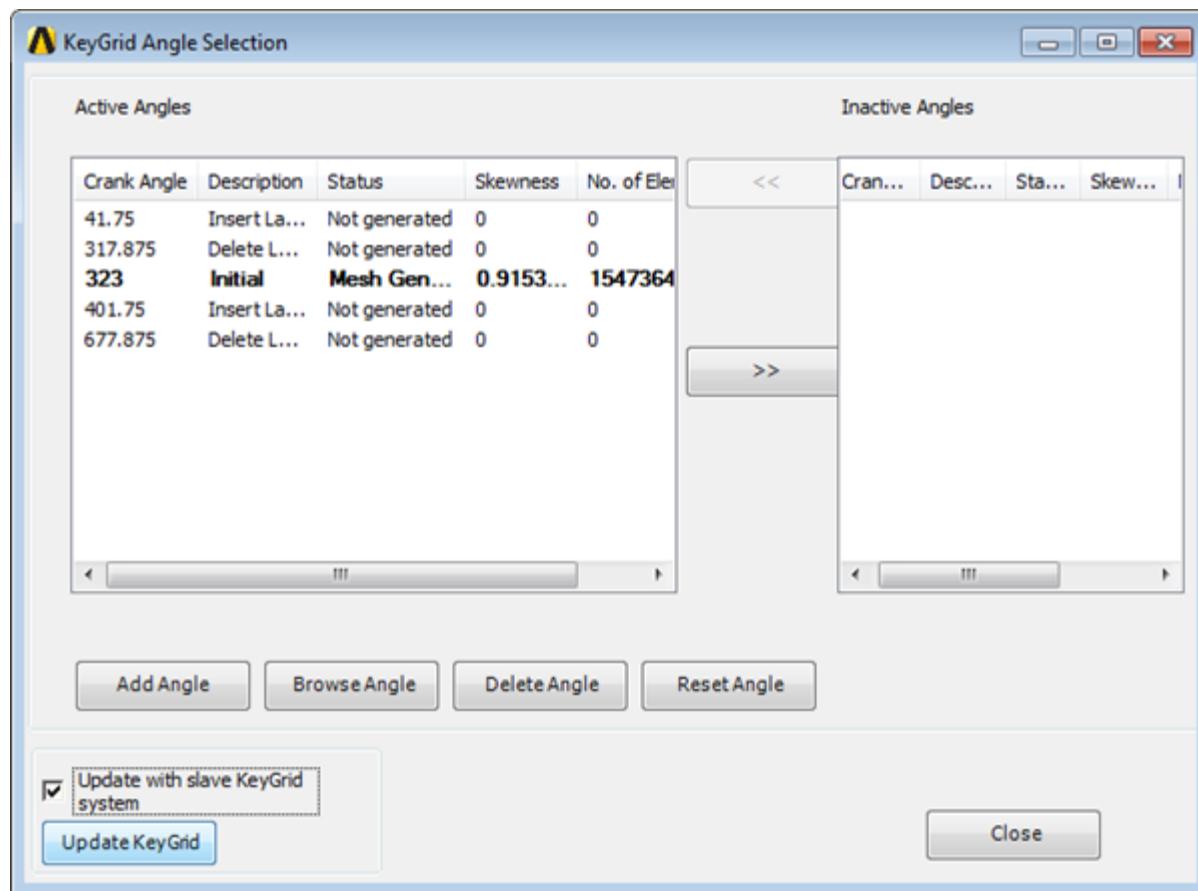
4.5. Step 4: Setting up the Simulation

After the decomposed geometry is meshed properly you can set keygrids, boundary conditions, monitors, and postprocessing images. You can also decide which data and images should be included in the report.

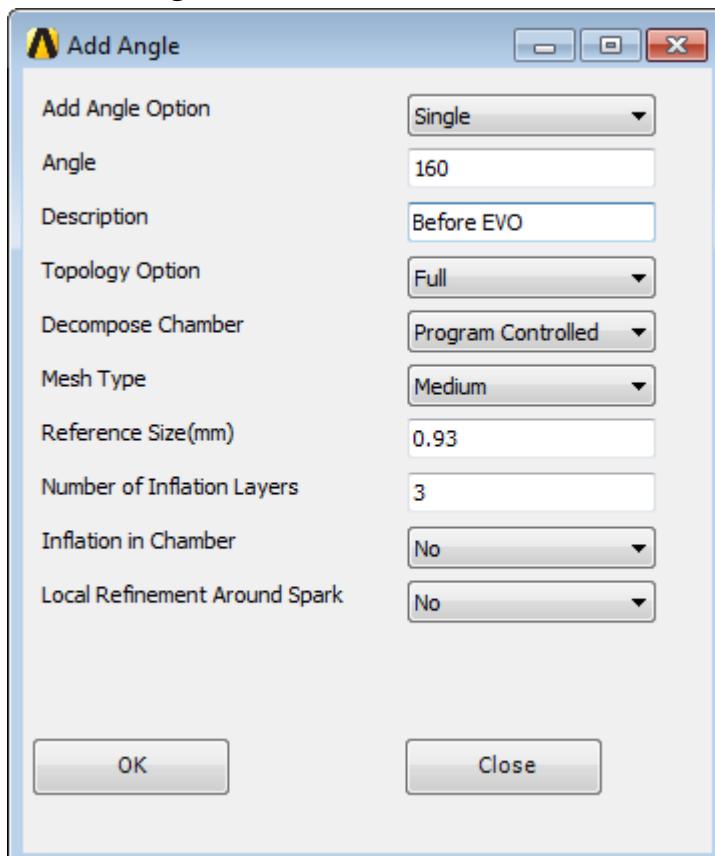
1. If the **Properties** view is not already visible, right-click **ICE Solver Setup**, cell 5, and select **Properties** from the context menu.



2. To set the KeyGrids select **Yes** from the **KeyGrid** drop-down list.
3. Click **Select KeyGrid Angles** to open the **KeyGrid Angle Selection** dialog box.



a. Click Add Angle.



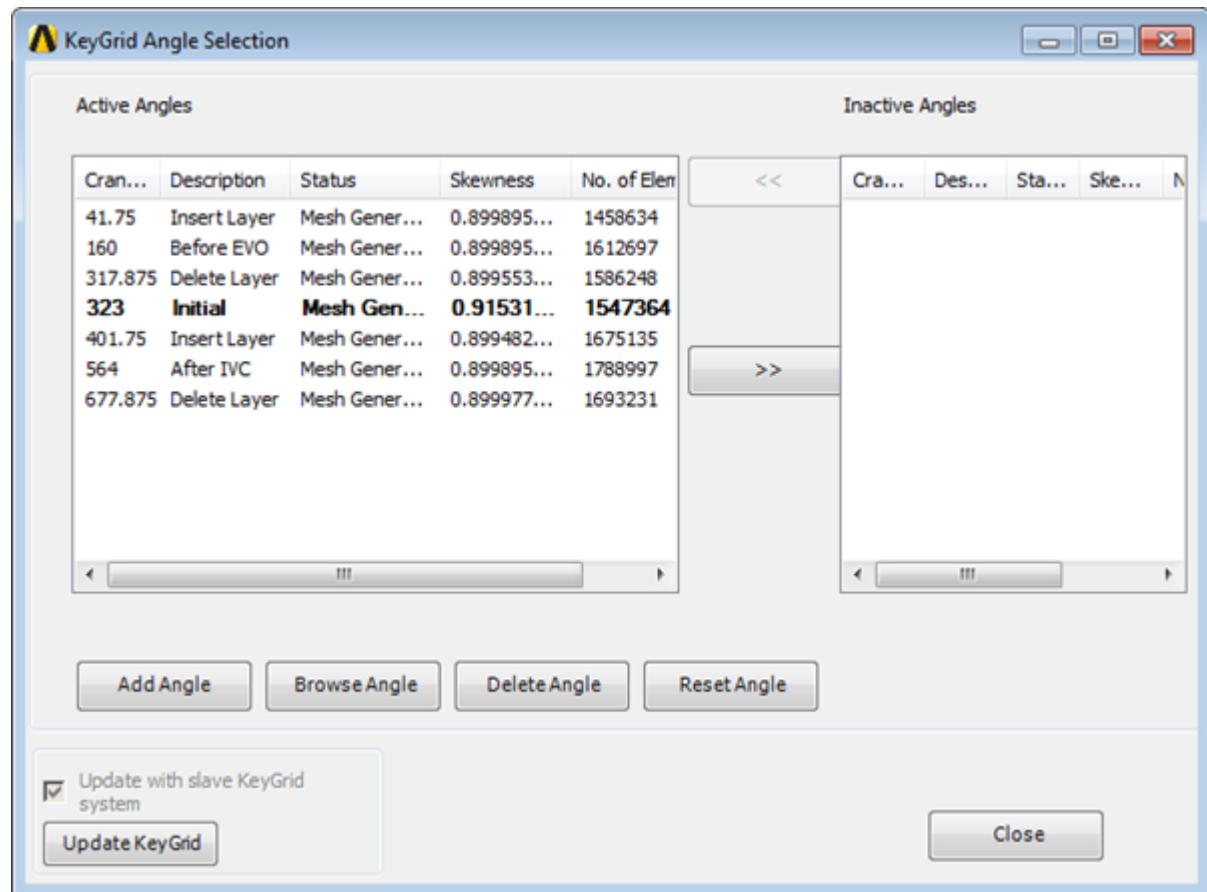
- i. Enter 160 for **Angle**.
- ii. Rename the **Description** to Before EVO.
- iii. Retain the default settings and click **OK** to close the **Add Angle** dialog box.

- b. Similarly add another angle at 564 and rename it to After IVC.
- c. For angle **564** select **Yes** for **Spark Refinement Around Spark**.
- d. Enable **Update with slave KeyGrid system**.
- e. Click **Update KeyGrid**.

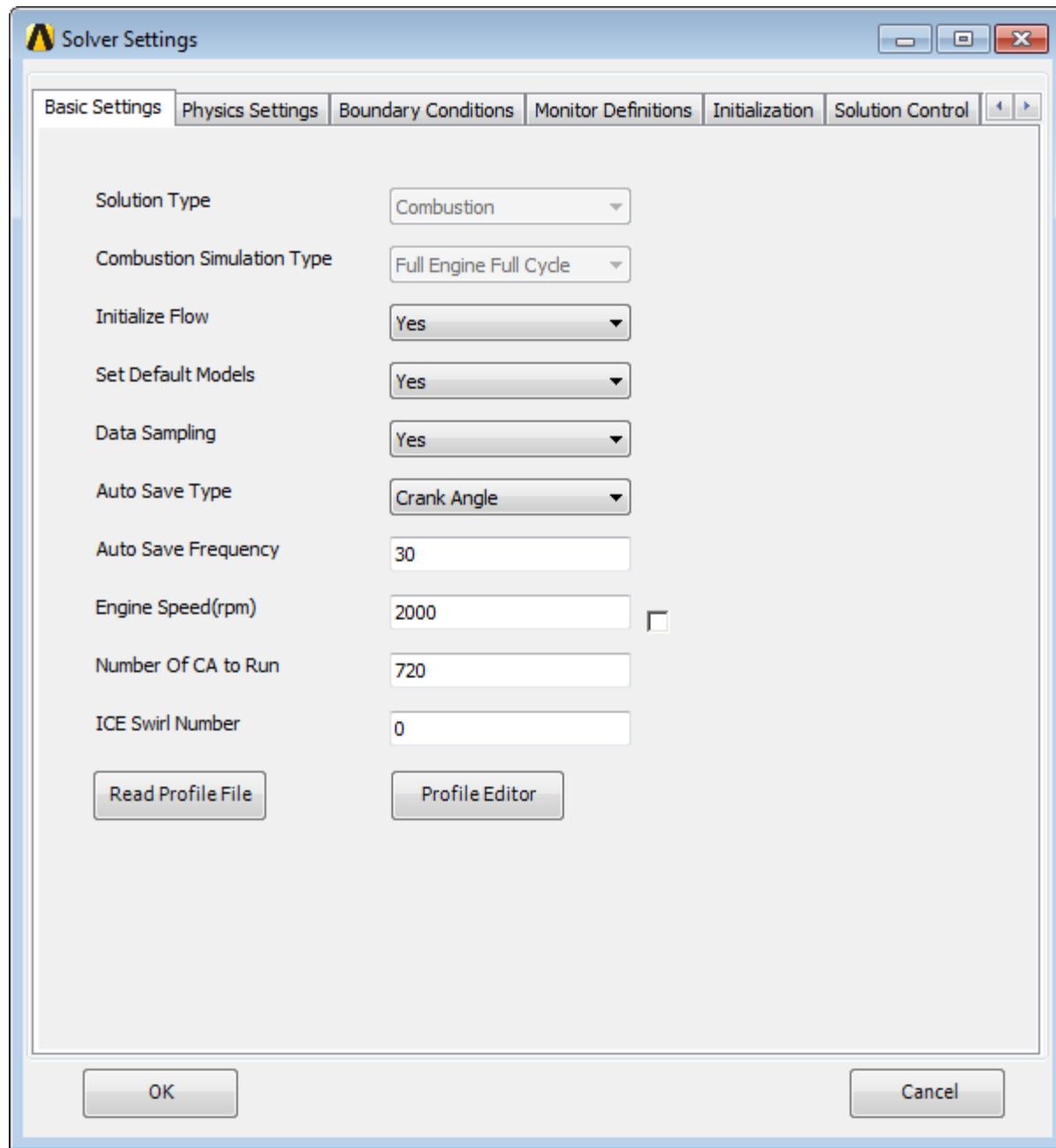
Note

While the KeyGrids are being updated in the slave system you can setup the solver.

After all the Keygrids have been updated click **Close** to close the **KeyGrid Angle Selection** dialog box.



4. Click **Edit Solver Settings** to open the **Solver Settings** dialog box.
 - a. In the **Basic Settings** tab enter 2000 for **Engine Speed(rpm)** and 0 for **ICE Swirl Number**.

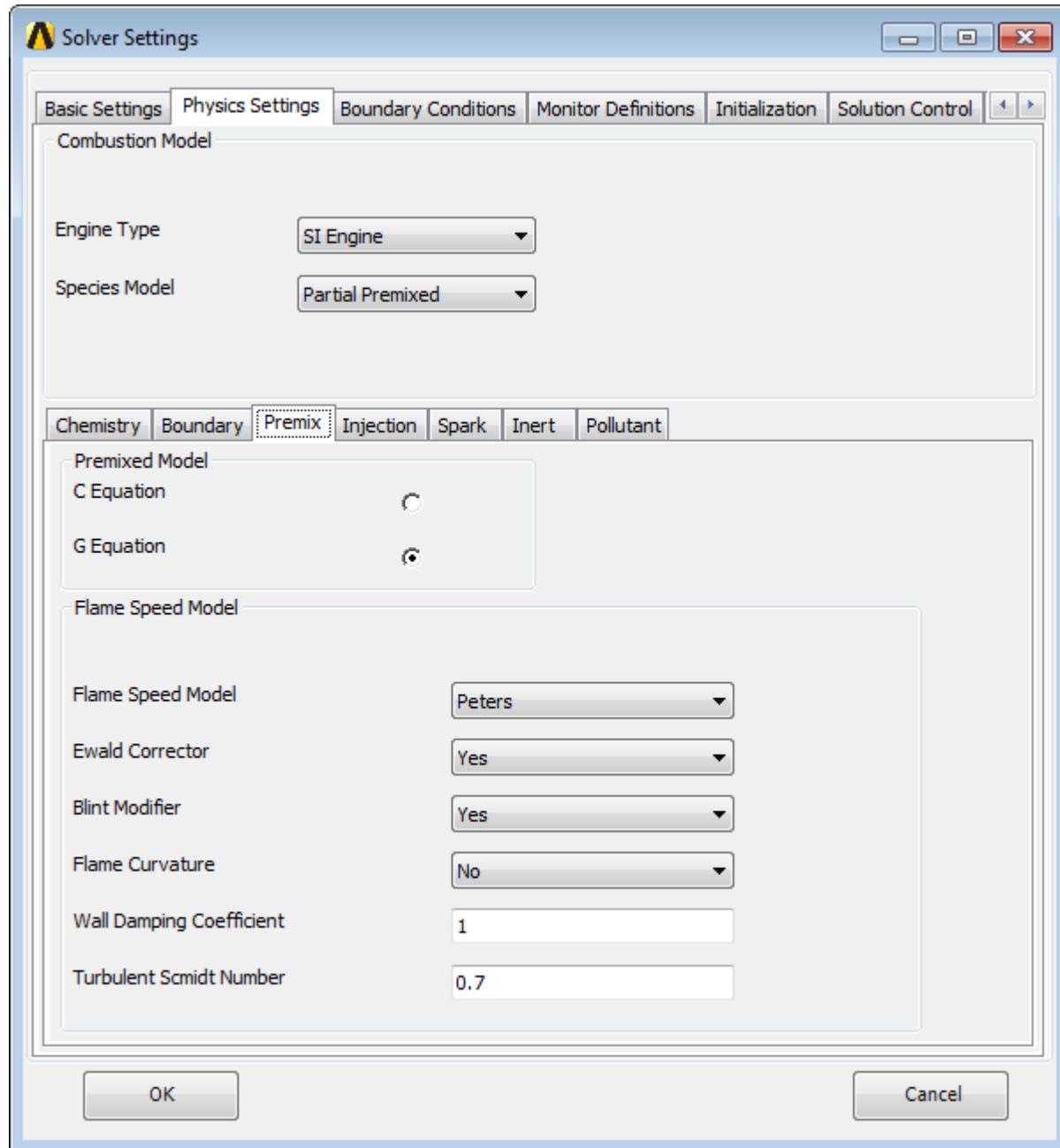


b. Click **Read Profile File**.

- i. In the **Browse profile file** click **Profile Files (.prof) (*.prof)** and select **CSV Files (.csv) (*.csv)**.
- ii. Select the file `massflowrate.csv` from your working folder and click **Open**.
- iii. In the **Read CSV File** dialog box that open rename the **Profile Name** as `massflowrate` and click **OK**.
- iv. Similarly read the `velocity-0.7cd.csv` file and rename it as `velocity`.

You can click **Profile Editor** and check the profile charts.

- c. In the **Physics Settings** tab select **SI Engine** from the **Engine Type** drop-down list and **Partial Premixed** from the **Species Model** drop-down list.

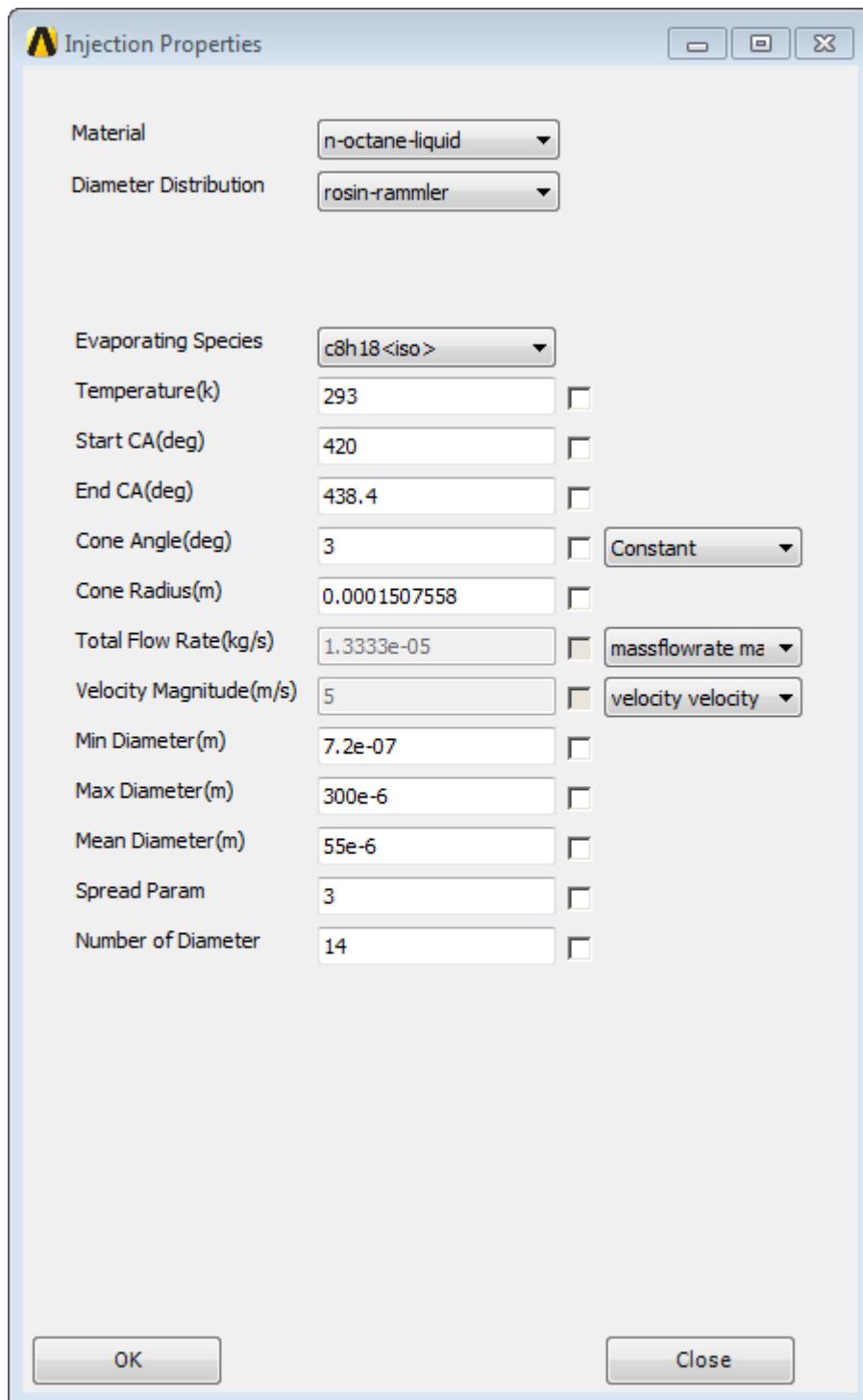


- i. Click the **Premix** tab.
 - A. Select **G Equation** from the **Premixed Model** list.
 - B. Select **Peters** from the **Flame Speed Model** drop-down list.
 - C. Select **Yes** for **Blind Modifier**.
- ii. Click the **Injection** tab you can see that three injections have been created.

- A. Select all three injections —**injection-0**, **injection-1**, and **injection-2** and click **Edit**.

Note

You can select all three injections by pressing **Ctrl** key and then selecting.

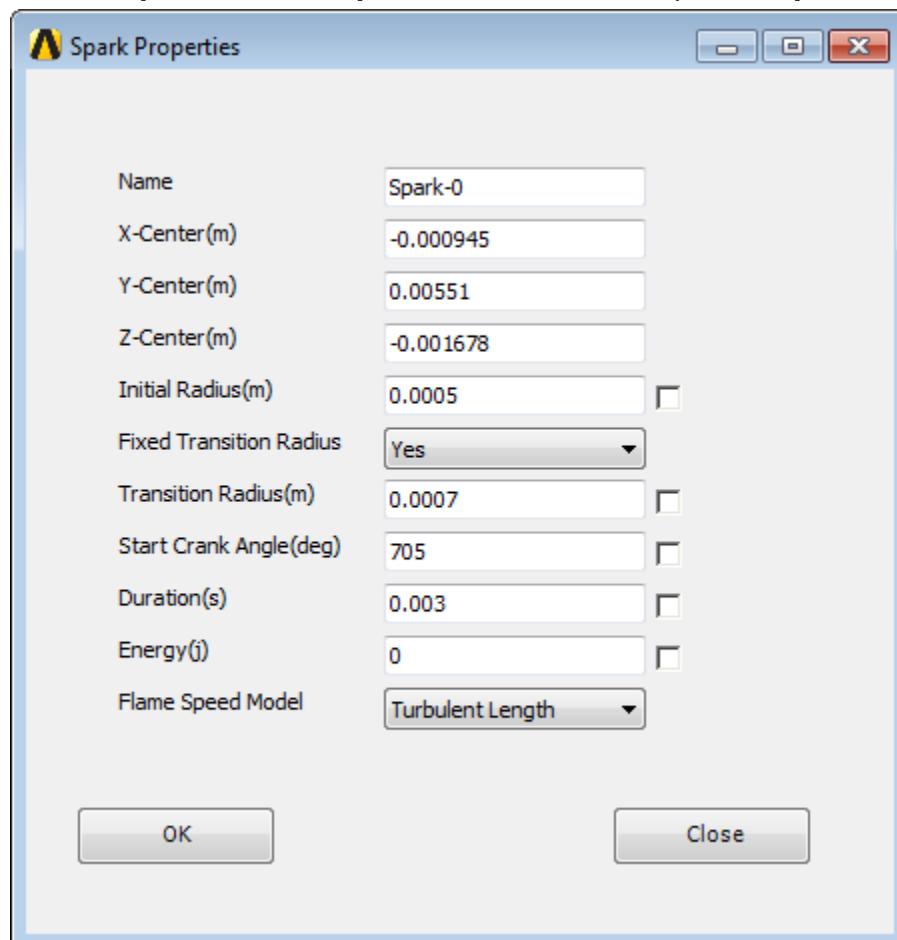


- B. Set the values in the **Injection Properties** dialog box as shown in the Table 4.1: Injection Properties (p. 196).

Table 4.1: Injection Properties

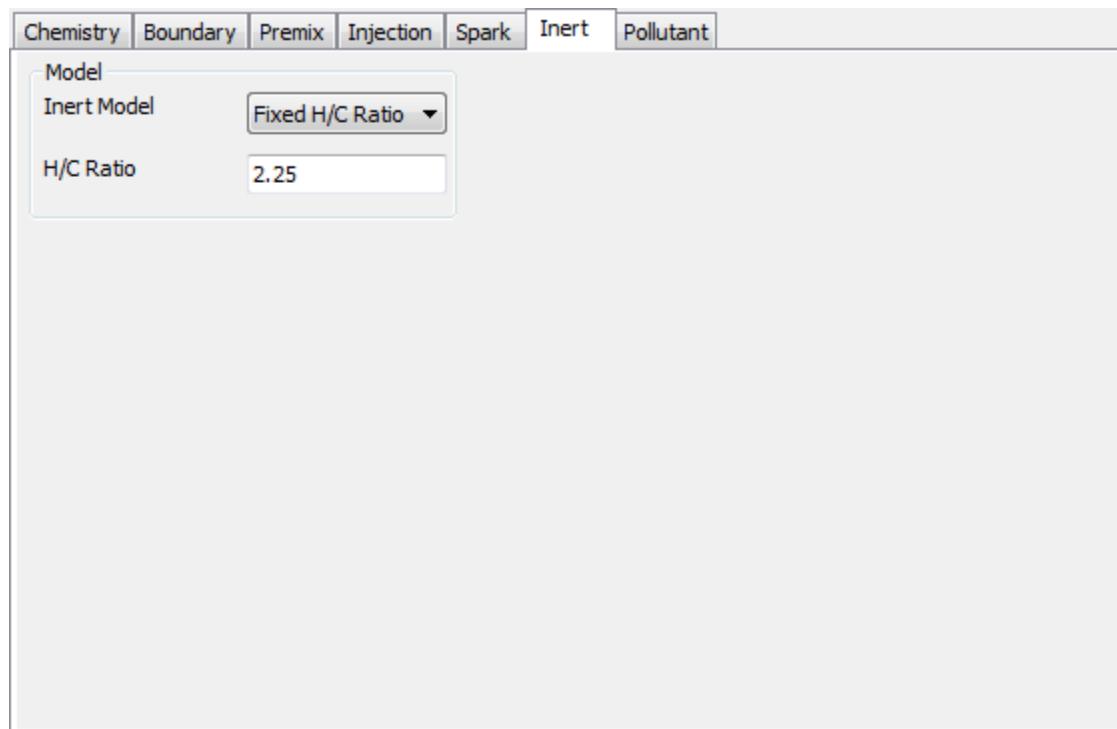
Parameter	Value
Material	n-octane-liquid
Evaporating Species	c8h18<iso>
Temperature	293
Start CA	420
End CA	438 . 4
Cone Angle	3
Total Flow Rate	massflowrate massflowrate
Velocity Magnitude	velocity velocity
Max Diameter	300e-6
Mean Diameter	55e-6
Spread Param	3

- iii. Click the **Spark** tab. Select **Spark-0** and click **Edit** to open the **Spark Properties** dialog box.

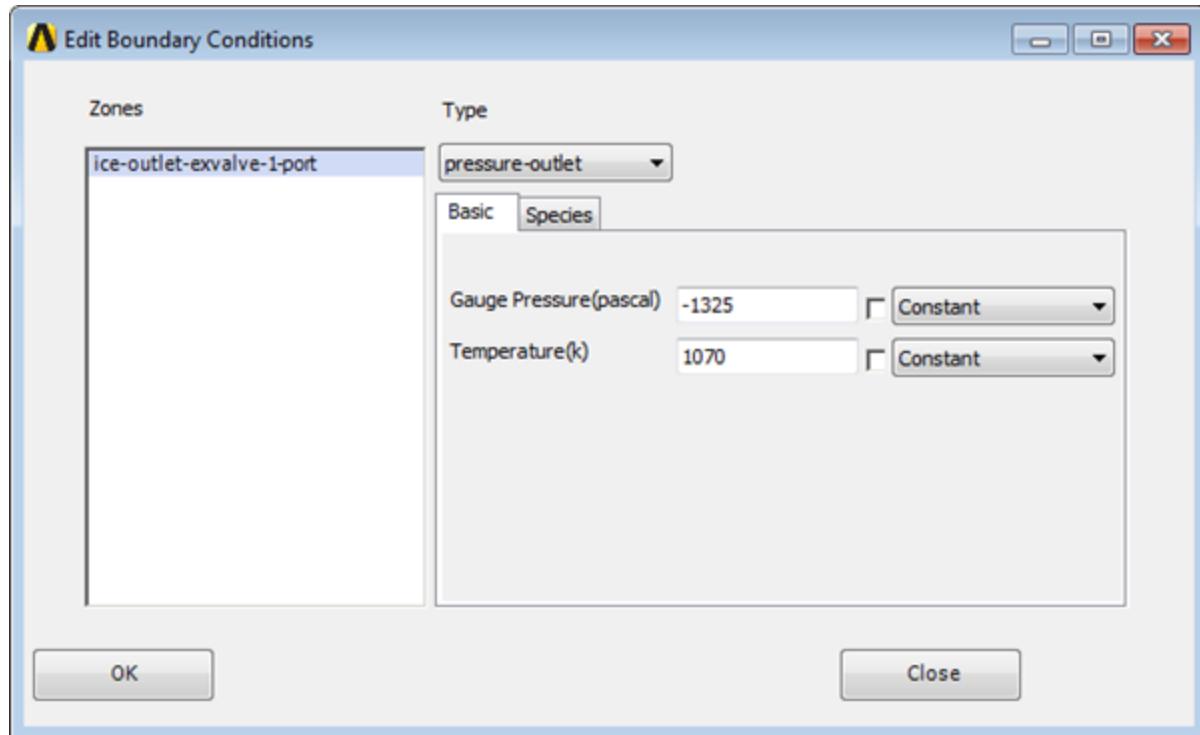


- A. Enter 705 for **Start Crank Angle**.

- B. Select **Turbulent Length** from the **Flame Speed Model** drop-down list.
 - C. Click **OK** to close the **Spark Properties** dialog box.
- iv. Click the **Inert** tab.



- A. Select **Fixed H/C Ratio** from the **Inert Model** drop-down list.
 - B. Retain the default setting of **2.25** for **H/C Ratio**.
- d. Click the **Boundary Conditions** tab.
- i. Double-click **ice-outlet-exvalve-1-port** to open the **Edit Boundary Conditions** dialog box.



- A. Enter -1325 pascal for **Gauge Pressure**.
 - B. Enter 1070 k for **Temperature**.
 - C. In the **Species** tab enter 0 . 98 for **Inert Stream**.
 - D. Click **OK** to close the dialog box.
- ii. Select **ice-inlet-invalve-1-port** and click **Edit**.
- A. Enter -21325 pascal for **Gauge Pressure**.
 - B. Enter 313 k for **Temperature**.
 - C. Click **OK** to close the **Edit Boundary Conditions** dialog box.
- iii. Click **Create** to open the **Create Boundary Conditions** dialog box.

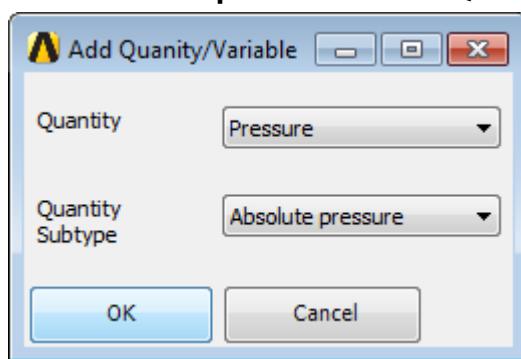
Set the **Temperature** for the zones as shown in the table.

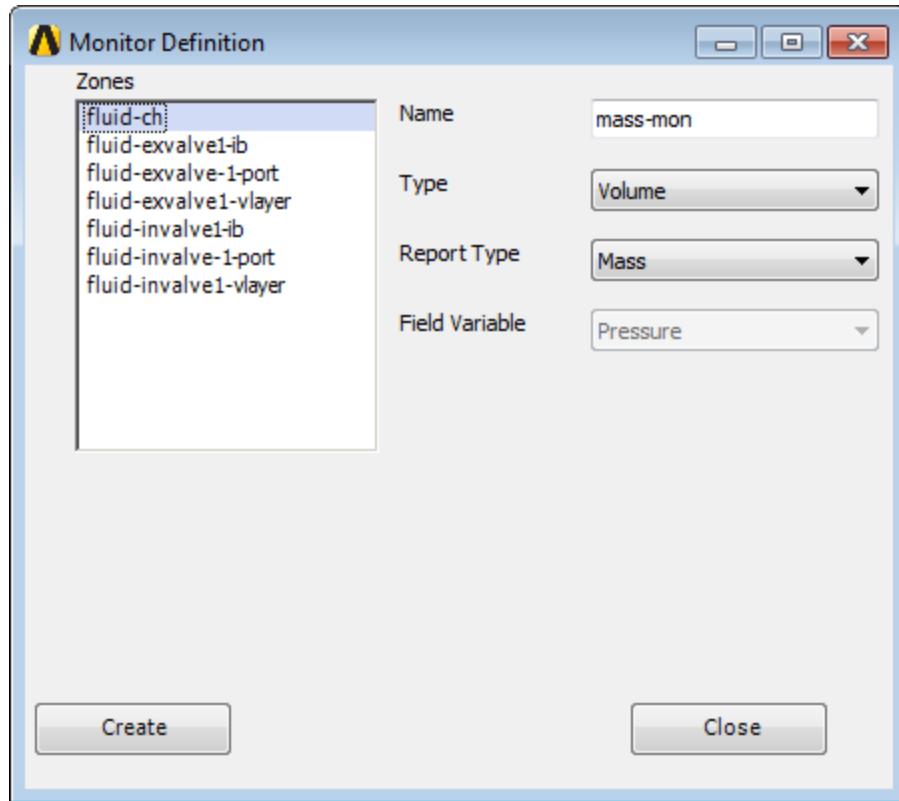
Table 4.2: Boundary Conditions

Part	Zone	Boundary Condition
Head	cyl-head, invalve1-ch, and exvalve1-ch	485k
Piston	piston	485k
Liner	cyl-tri	500k
Exhaust valve	exvalve1-ib, exvalve1-ob, and exvalve1-stem	777k
Exhaust port	exvalve1-port and exvalve1-seat,	485k

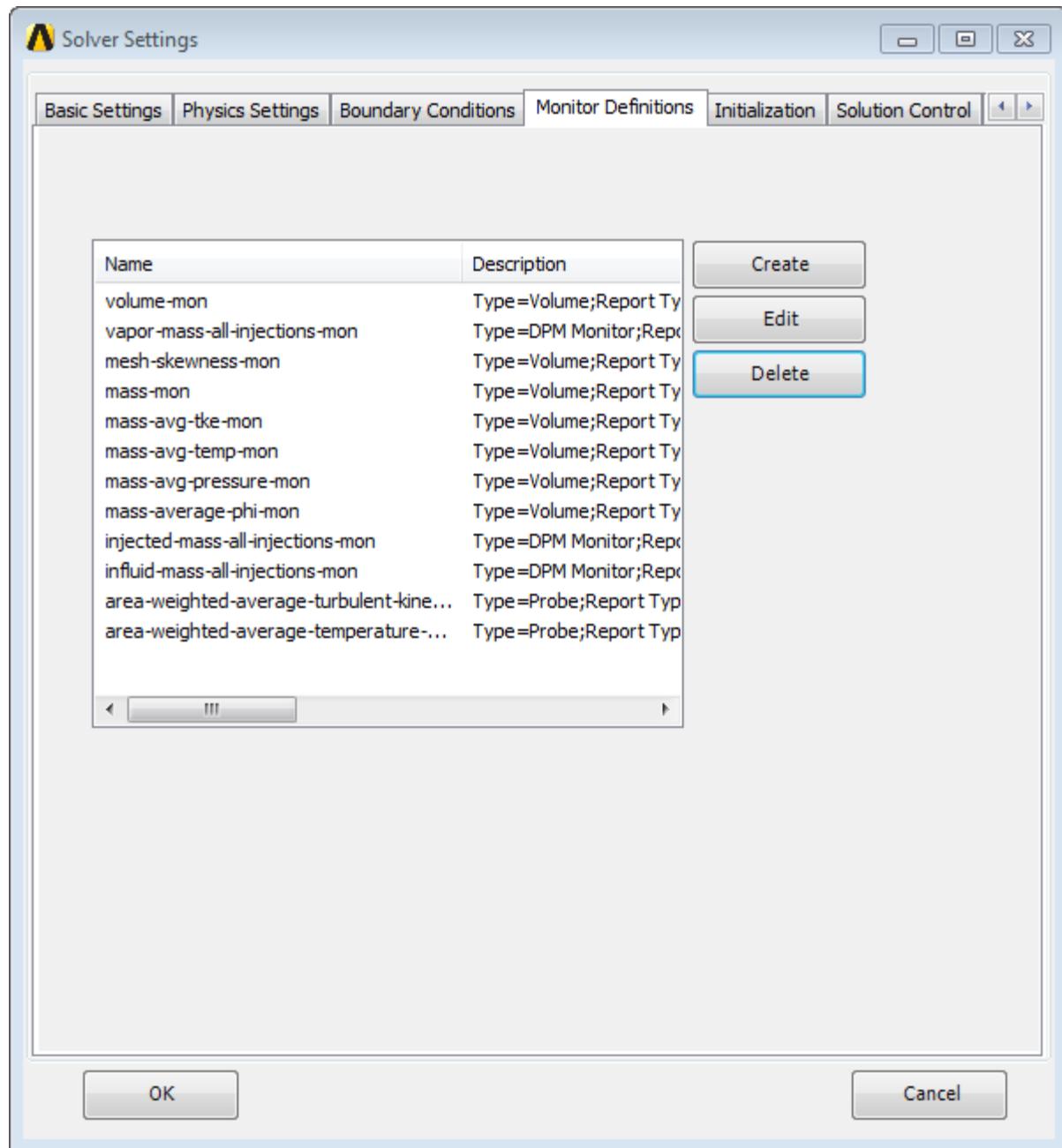
Intake valve	invalve1-ib, invalve1-ob, and invalve1-stem	400k
Intake port	invalve1-port and invalve1-seat,	313k

- iv. After creating all the above boundary conditions close the **Create Boundary Conditions** dialog box.
- e. In the **Monitor Definitions** tab you can see that four volume monitors have been set on the zone **fluid-ch**. You will create some additional monitors.
 - i. Select the monitor **mass-avg-pressure-mon** from the list and click **Edit**.
 - A. Select **New Variable** from the **Field Variable** drop-down list.
 - B. In the **Add Quantity/Variable** dialog box select **Pressure** from the **Quantity** drop-down list.
 - C. Select **Absolute pressure** from the **Quantity Subtype** drop-down list and click **OK**.
 - D. Close the **Monitor Definition** dialog box.
- ii. Click **Create** to open the **Monitor Definition** dialog box.

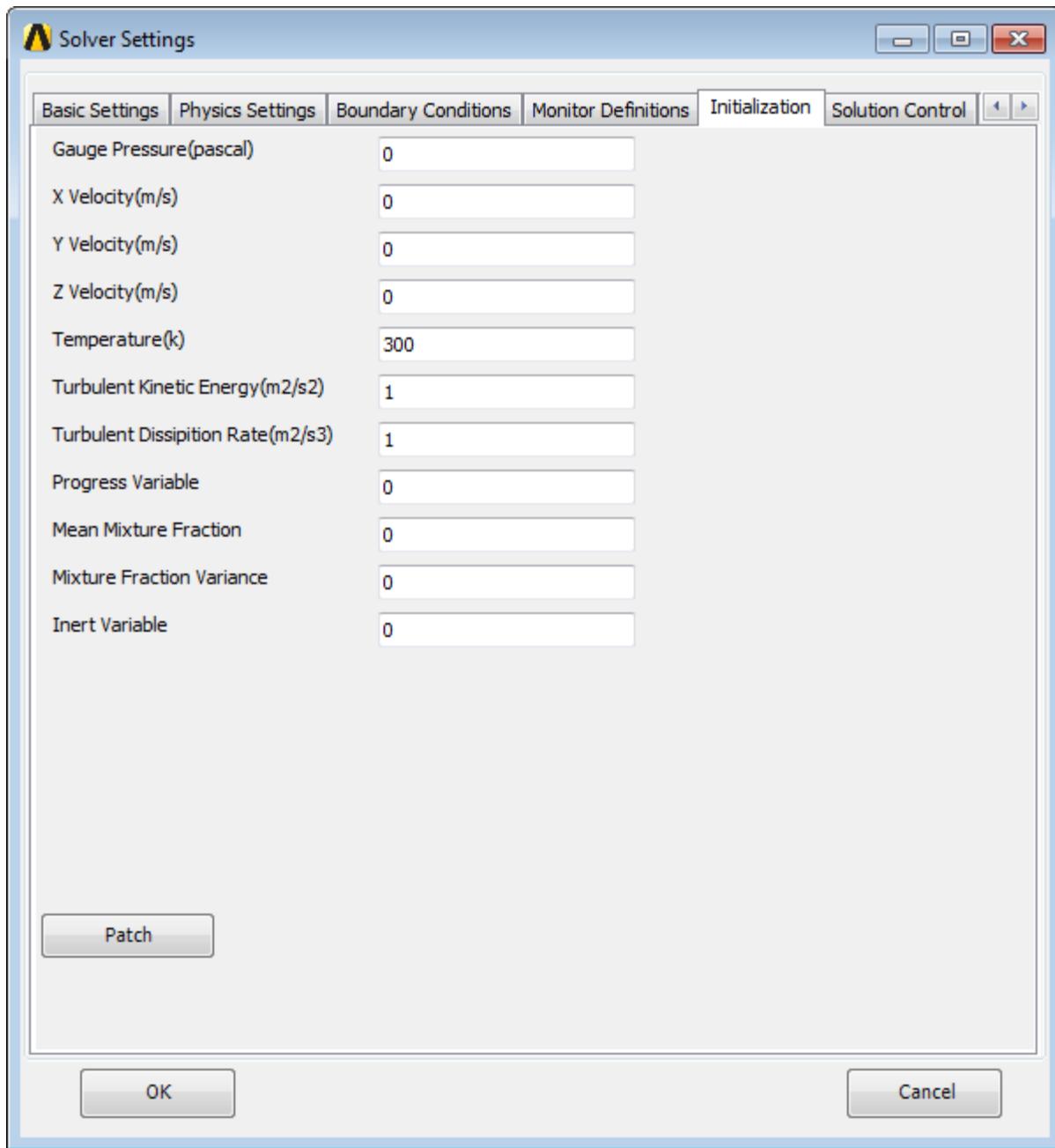




- iii. Select **fluid-ch** from the list of **Zones**.
 - A. Select **Volume** from the **Type** drop-down list.
 - B. Select **Mass** from the **Report Type** drop-down list.
 - C. Click **Create**.
- iv. For the same zone create a volume monitor by selecting **Volume** from the **Report Type** drop-down list.
- v. Similarly for the same zone create a volume monitor with **Mass-Average** selected from the **Report Type** drop-down list. From the **Field Variable** drop-down list select **Phi**.
- vi. Select **DPM Monitor** from the **Type** drop-down list.
 - A. Select **influid-mass** from the **Injection Fate** drop-down list.
 - B. Select **all-injections** from the list of **Injection** and click **Create**.
 - C. Similarly retaining the selection of **all-injections**, select **injected-mass** from the **Injection Fate** drop-down list and click **Create**.
 - D. Create another monitor by selecting **vapor-mass** from the **Injection Fate** drop-down list



- f. In the **Initialization** tab you can see the default set values for the various parameters.



- i. Click **Patch** to open the **Patching Zones** dialog box.
- ii. For the inlet port, select **fluid-invalve-1-port**, **fluid-invalve-1-vlayer**, and **fluid-invalve-1-ib** from the list of **Zone**.
- iii. Select **Pressure** from the list of **Variable**.
- iv. Enter **-21325** for **Value(pascal)** and click **Create**.
- v. Similarly patch the same zones for **Temperature** value **313 k**.
- vi. For outlet port, patch zones **fluid-exvalve-1-port**, **fluid-exvalve-1-vlayer**, and **fluid-exvalve-1-ib** to **Pressure** equal to **-1325 pascal**, **Temperature** equal to **1070 k**, and **Inert Variable** to **1**.

- vii. For chamber select **fluid-ch** from the list under **Zone** and patch the **Temperature** to 1070 k, **Pressure** to 4025 pascal, and **Inert Variable** to 1.

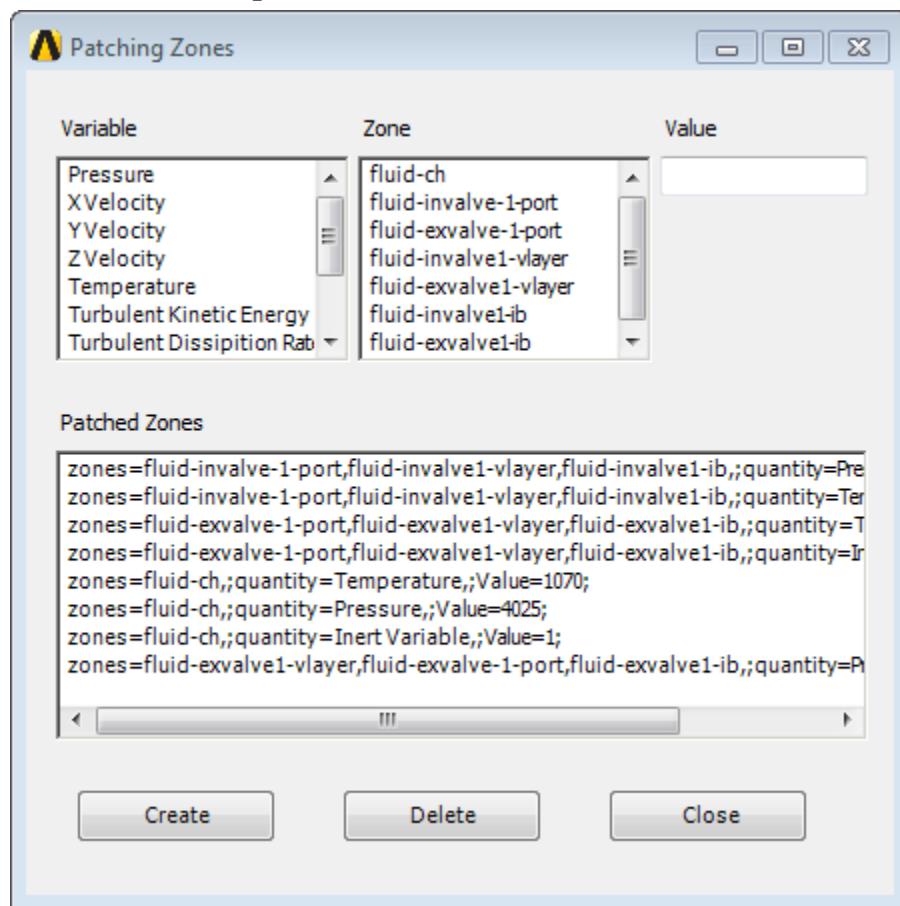
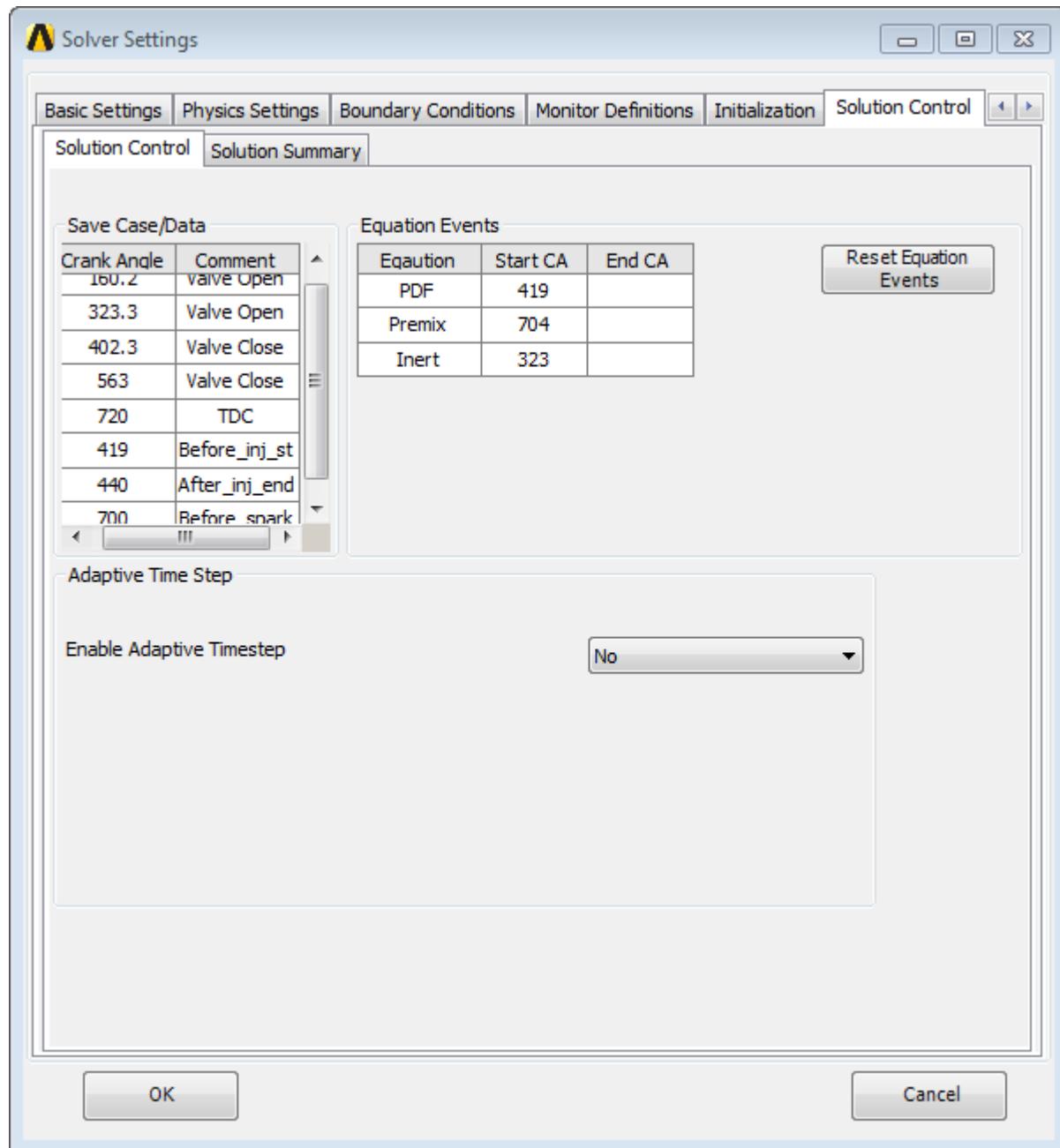


Table 4.3: Patching

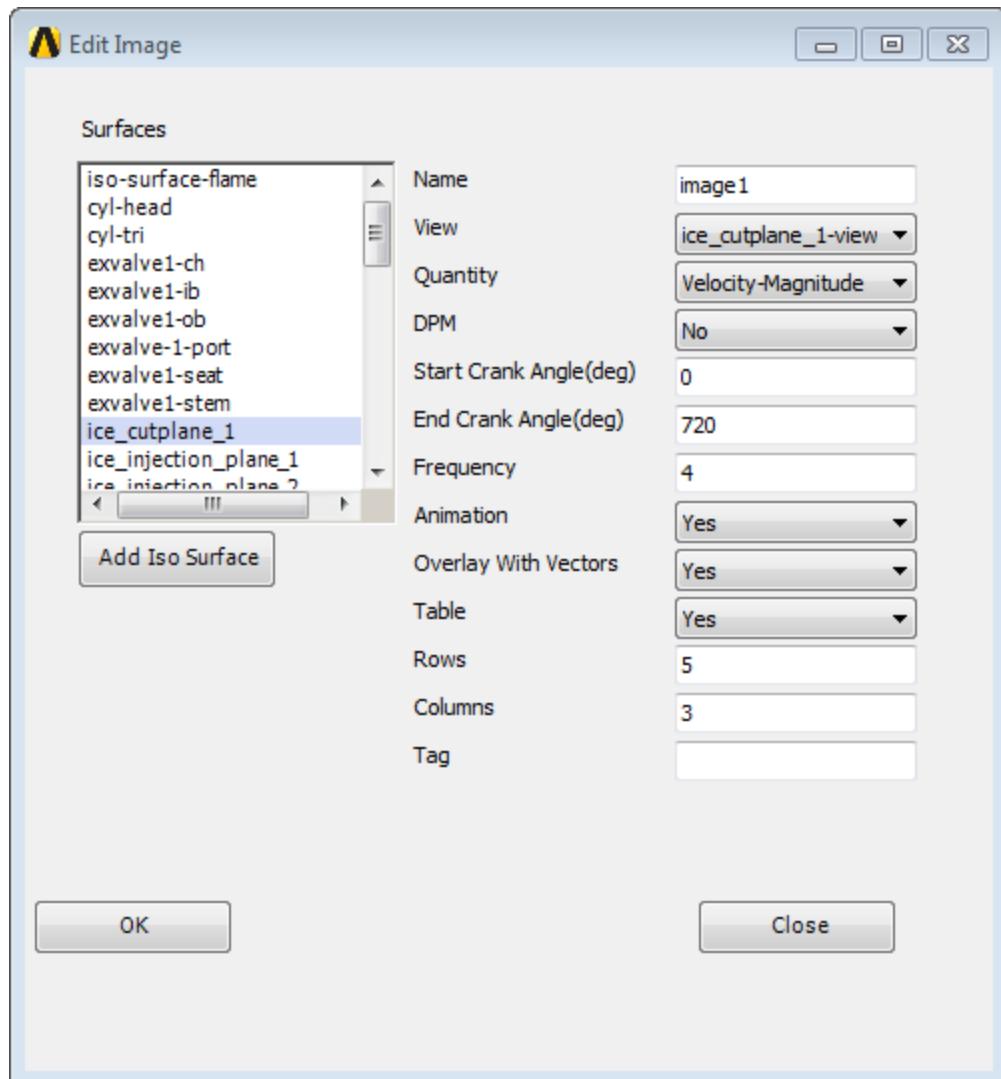
Variable	Zones	Value
Pressure	fluid-invalve-1-port, fluid-invalve-1-vlayer, fluid-invalve-1(ib)	-21325 pascal
Temperature	fluid-invalve-1-port, fluid-invalve-1-vlayer, fluid-invalve-1(ib)	313 k
Pressure	fluid-exvalve-1-port, fluid-exvalve-1-vlayer, fluid-exvalve-1(ib)	-1325 pascal
Temperature	fluid-exvalve-1-port, fluid-exvalve-1-vlayer, fluid-exvalve-1(ib)	1070 k
Pressure	fluid-ch	4025 pascal
Temperature	fluid-ch	1070 k
Inert Variable	fluid-ch	1
Inert Variable	fluid-exvalve-1-port, fluid-exvalve-1-vlayer, fluid-exvalve-1(ib)	1

viii. Close the **Patching Zones** dialog box.

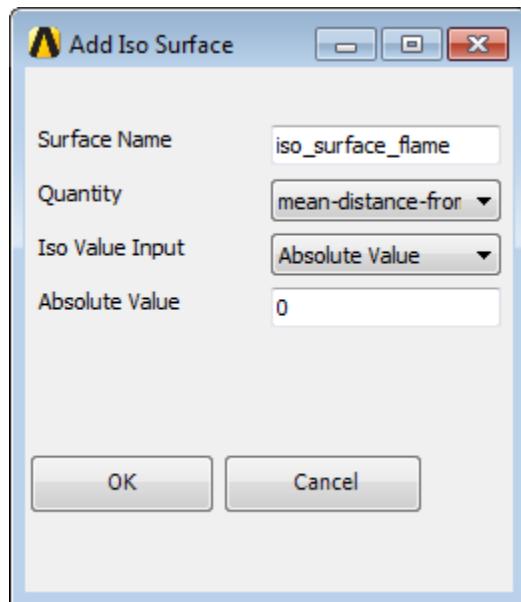
- g. In the **Solution Control** tab enter 419 under **Start CA for PDF**.



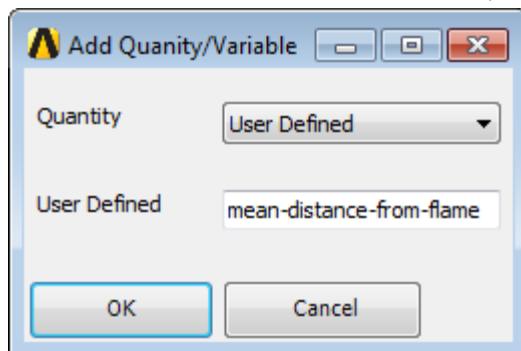
- i. Right-click in **Save Case/Date** group box and select **Insert Row Below** from the context menu.
- ii. Enter 419 for **Crank Angle** and add **Before_inj_st** as **Comment**.
- iii. Similarly add angle 440 as **After_inj_end** and 700 as **Before_spark**.
- h. In the **Post Processing** tab you can see that velocity-magnitude contours on the surface of cut-plane will be saved during simulation and displayed in a table format in the report.
- i. Double-click **image-1** to open the **Edit Image** dialog box.



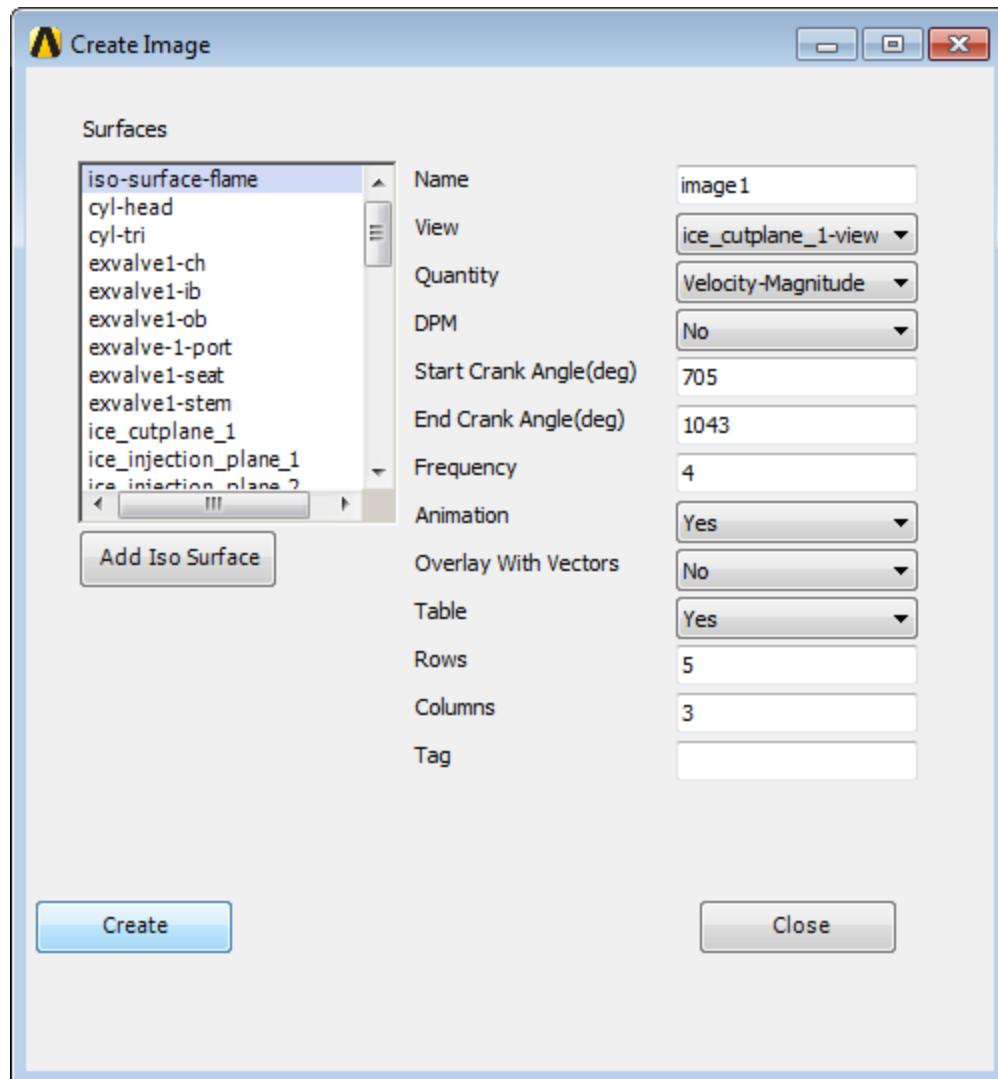
- A. Select **Yes** from the **Overlay with Vectors** drop-down list.
- B. Click **OK** to close the **Edit Image** dialog box.
- ii. Click **Create** in the **Post Processing** tab.
 - A. Select **ice_cutplane_1** from the list of **Surfaces**.
 - B. Select **Temperature** from the **Quantity** drop-down list.
 - C. Select **No** for **DPM**.
 - D. Select **Yes** from the **Overlay with Vectors** drop-down list.
 - E. Click **Create**.
- iii. Click **Add Iso Surface**.



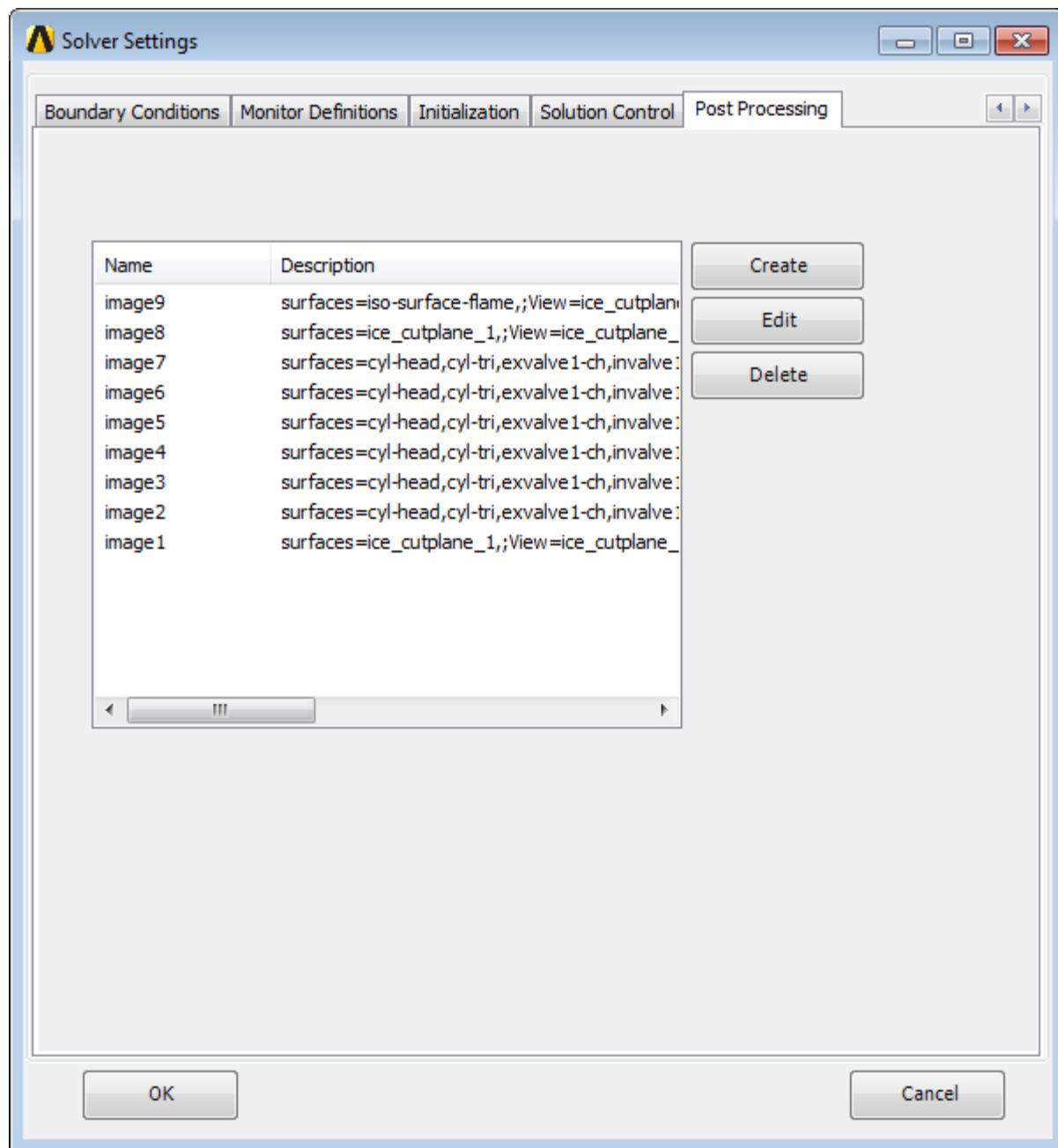
- A. In the **Add Iso Surface** dialog box enter **iso-surface-flame** for **Surface Name**.
- B. Select **New Variable** from the **Quantity** drop-down list.



- I. In the **Add Quantity/Variable** dialog box select **User Defined** from the **Quantity** drop-down list.
 - II. Enter **mean-distance-from-flame** for **User Defined**.
 - C. Retain **Absolute Value** for **Iso Value Input**.
 - D. Retain **0** for **Absolute Value**.
 - E. Click **OK** to close the **Add Iso Surface** dialog box.
- iv. To add another post processing image, select **iso-surface-flame** from the list of **Surfaces**.



- Retain the selection of **ice_cutplane_1-view** for **View**.
- Select **mean-distance-from-plane** from the **Quantity** drop-down list.
- Select **No** from the **DPM** drop-down list.
- Enter **705** for **Start Crank Angle(deg)**.
- Enter **1043** for **End Crank Angle(deg)**.
- Click **Create** and close the **Create Image** dialog box.



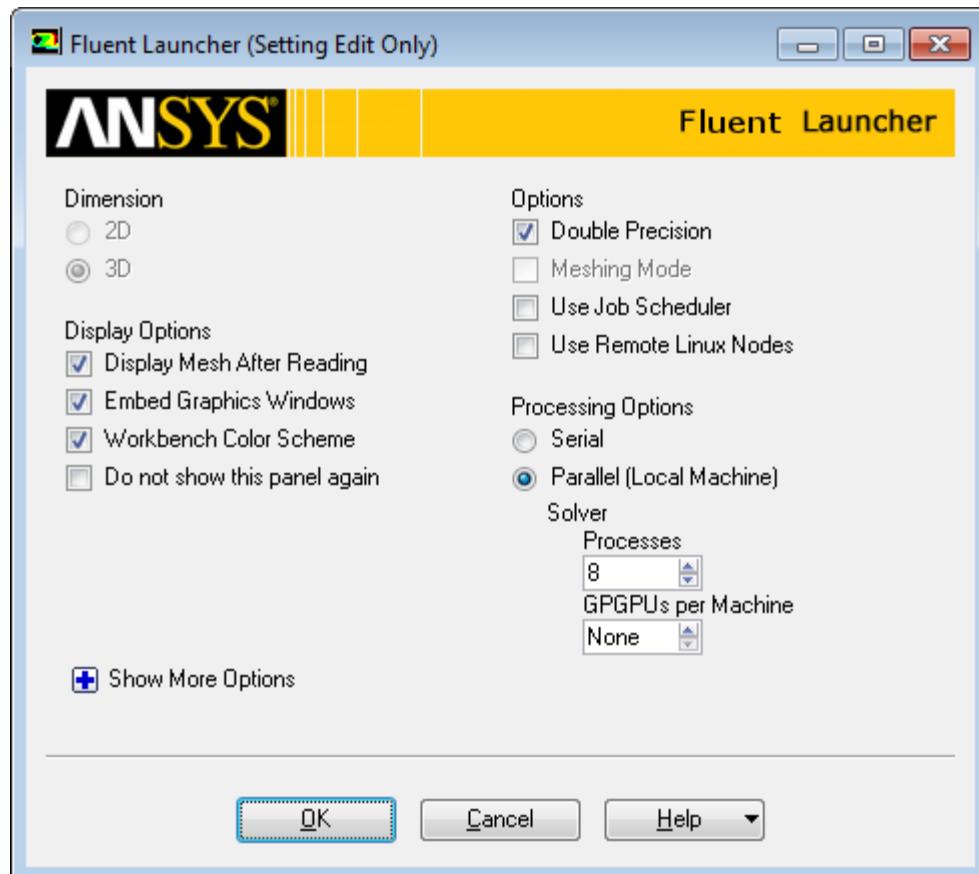
5. Click **OK** to close the **Solver Settings** dialog box.
6. Save the project.

File > Save

4.6. Step 5: Running the Solution

In this step you will start the simulation.

1. Double-click on the **Setup** cell.

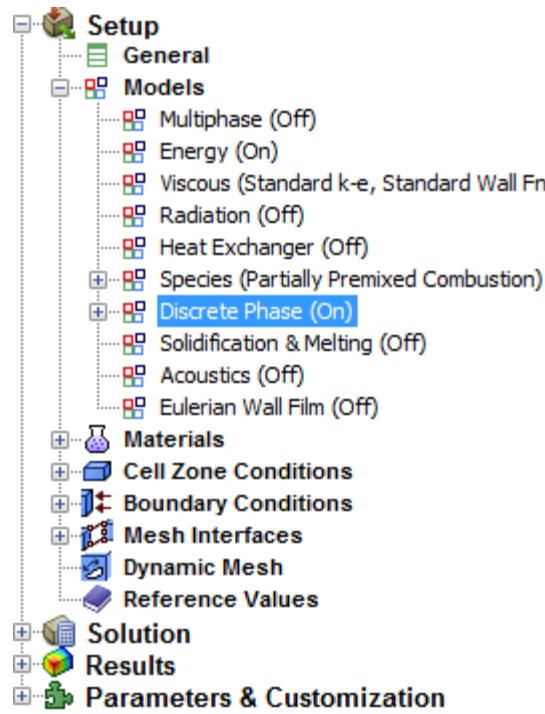


2. You can run the simulation in parallel with increased number of processors to complete the solution in less time.
3. Click **OK** in the **FLUENT Launcher** dialog box.

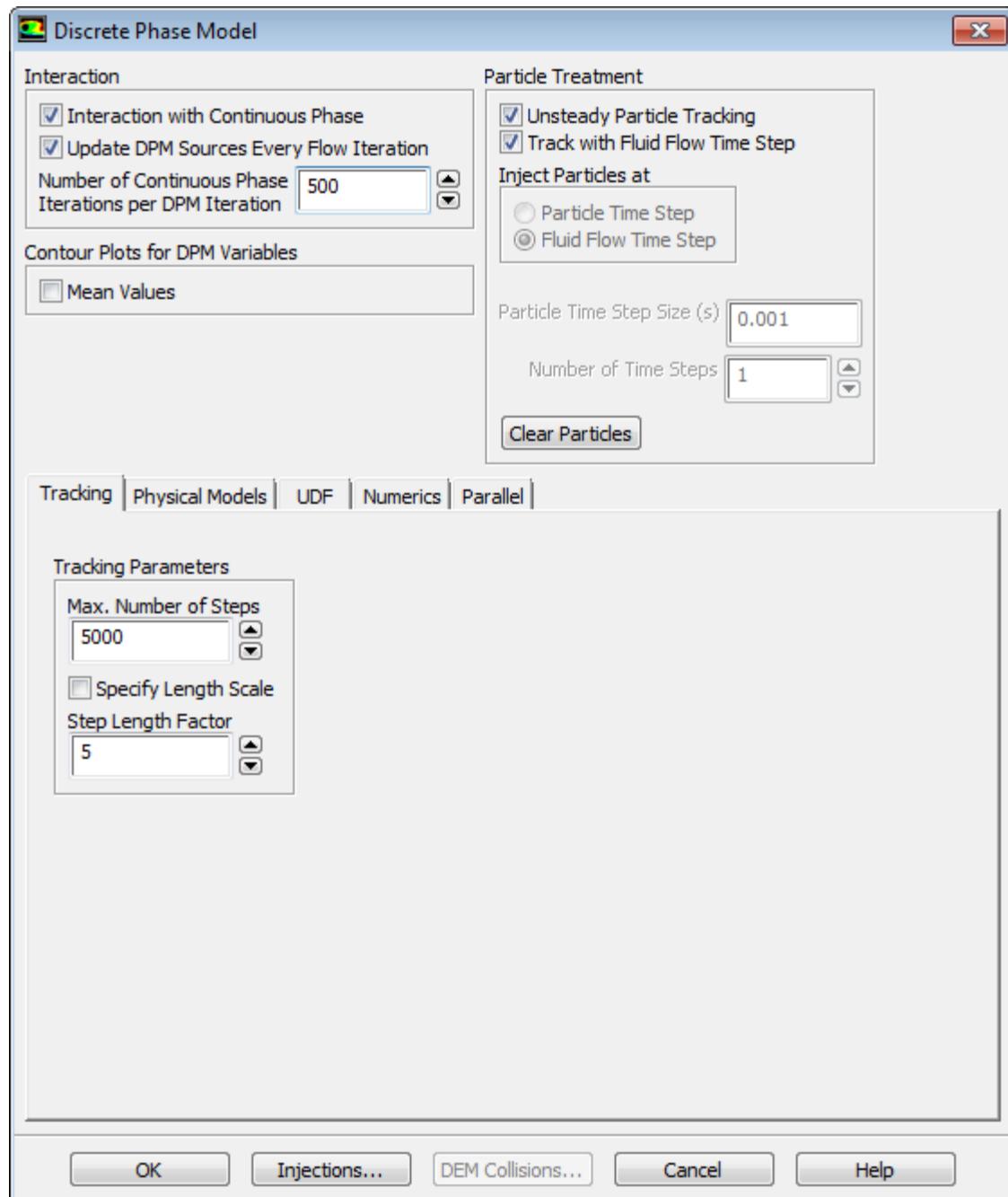
Note

ANSYS Fluent opens. It will read the mesh file and setup the case.

4. In the tree expand **Models** and double click on **Discrete Phase**.

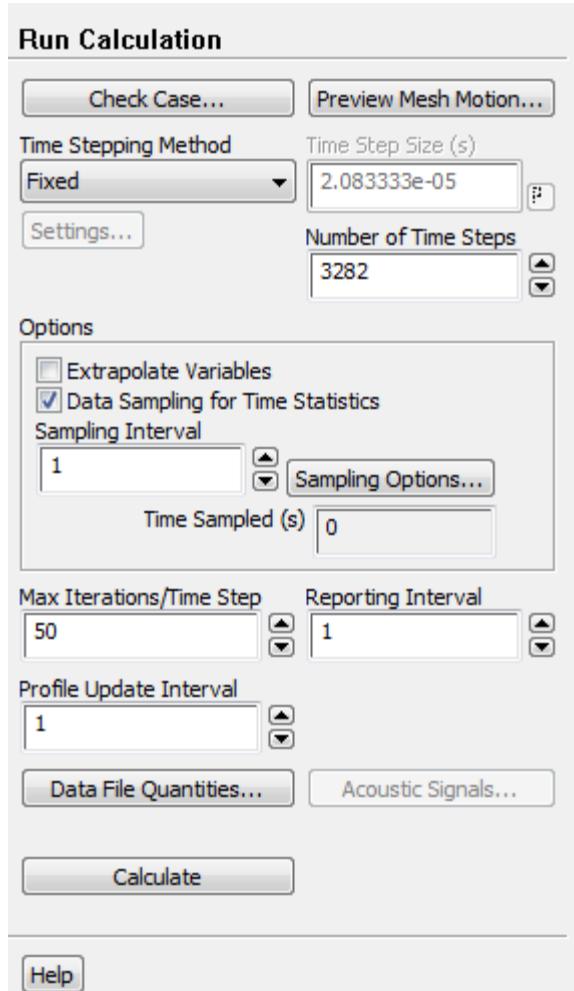


- In the **Discrete Phase Model** dialog box enter 500 for **Number of Continuous Phase Iterations per DPM Iteration** and click **OK**.



This will reduce solution time.

5. Click **Run Calculation** in the navigation pane.



- a. Retain the default set **Number of Time Steps**.

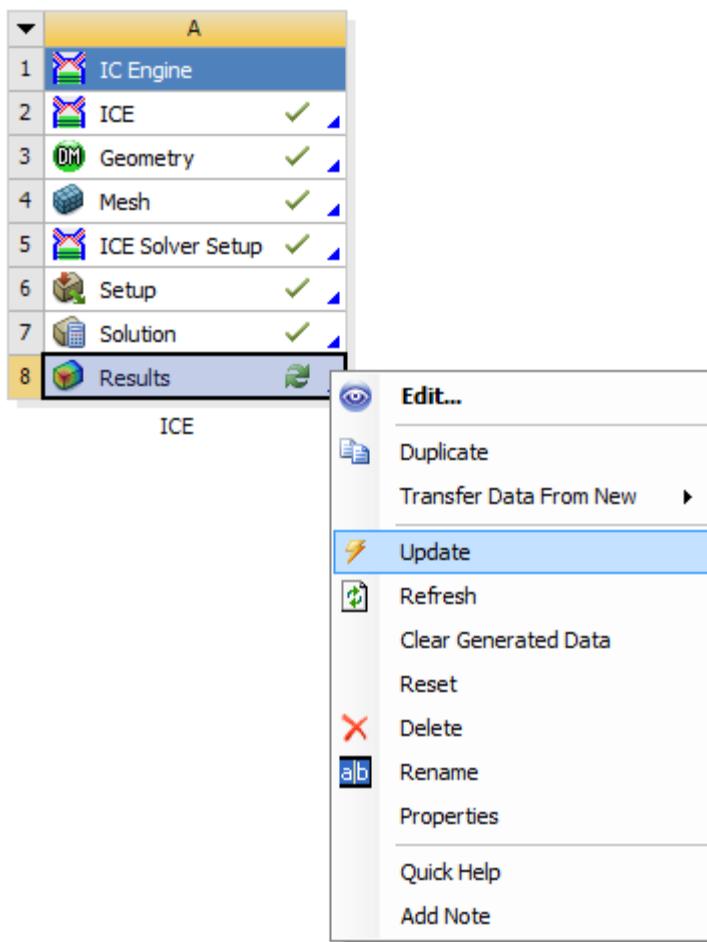
Note

The **Number of Time Steps** are automatically calculated from the entered start and end crank angles.

- b. Click **Calculate**.

4.7. Step 6: Obtaining the Results

1. Right-click on the **Results** cell and click **Update** from the context menu.



- Once the **Results** cell is updated, view the files by clicking **Files** from the **View** menu.

View >Files

- Right-click **Report.html** from the list of files, and click **Open Containing Folder** from the context menu.
 - In the **Report** folder double-click **Report.html** to open the report.
 - You can check the node count and mesh count of the cell zones in the table, **Mesh Information for ICE**.

1. File Report

Table 1. File Information for Gdi Tut

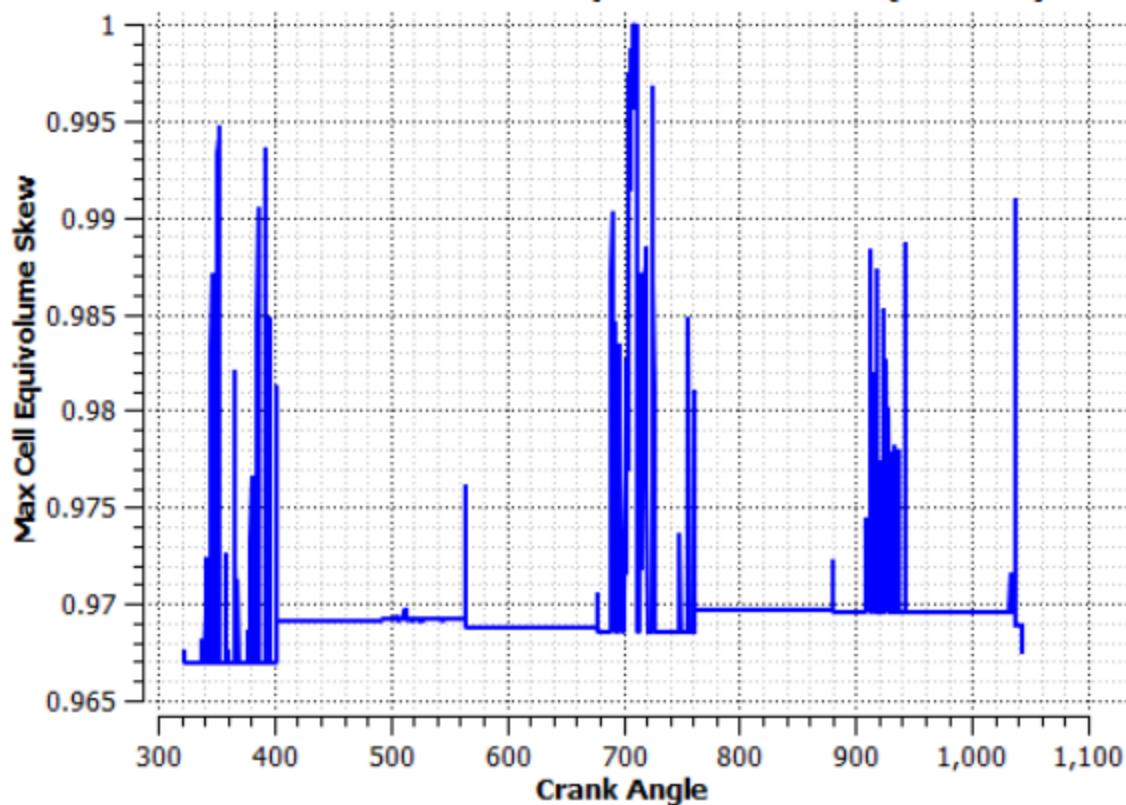
Case	Gdi Tut
File Path	E:\ICETutorials16\GDI Tutorial\demo_comb_files\dp0\ICE\Fluent\ICE-1-ca1043.150.dat.gz
File Date	21 December 2014
File Time	03:44:08 AM
File Type	FLUENT
File Version	16.0.0

2. Mesh Report

Table 2. Mesh Information for Gdi Tut

Domain	Nodes	Elements
fluid ch	154558	816822
fluid exvalve 1 port	65861	220436
fluid exvalve1 ib	18696	15580
fluid exvalve1 vlayer	112000	102600
fluid invalve 1 port	114287	360149
fluid invalve1 ib	7776	6192
fluid invalve1 vlayer	28800	21000
fluid layer cylinder	0	0
fluid piston	0	0
All Domains	501978	1542779

You can see the **Max Cell Equivolume Skew** monitor.

Chart 1. Monitor: Max Cell Equivolume Skew (fluid-ch)**Monitor: Max Cell Equivolume Skew (fluid-ch)****Table 3.** Cell count at crank angles

Crank Angle	Cell Count
0.000e+00	1.597e+06
1.800e+02	1.667e+06

- You can see the boundary conditions, under-relaxations factors, and other setup conditions under **Setup**.

3.1. Physics

Table 4. Boundary Conditions

Type	Zones	Values
wall (invalve1)	invalve1-ch	Temperature (k) 485
	invalve1-stem, invalve1-ob, invalve1-ib	Temperature (k) 400
wall (exvalve1)	exvalve1-ch	Temperature (k) 485
	exvalve1-stem, exvalve1-ob, exvalve1-ib	Temperature (k) 777
wall (invalve-port)	invalve1-port	Temperature (k) 313
wall (exvalve-port)	exvalve1-port	Temperature (k) 485
pressure-inlet	ice-inlet-invalve1-port	Gauge Total Pressure (pascal) -21325
		Supersonic/Initial Gauge Pressure (pascal) 0
		Total Temperature (k) 313
wall	cyl-head	Temperature (k) 485
wall	cyl-tri	Temperature (k) 500
wall	piston	Temperature (k) 485
pressure-outlet	ice-outlet-exvalve1-port	Gauge Pressure (pascal) -1325
		Backflow Total Temperature (k) 1070
wall	invalve1-seat	Temperature (k) 313
wall	exvalve1-seat	Temperature (k) 485

3.2. Relaxations

Table 5. Relaxation changes through events

Crank Angle	Pressure	Density	Body Forces	Momentum	Turbulent Kinetic Energy	Turbulent Dissipation Rate	Turbulent Viscosity	Energy	Temperature	Mean Mixture Fraction	Mixture Fraction Variance	G Equation	G Variance	Inert	Discrete Phase Sources
0.000	0.200	1.000	1.000	0.500	0.400	0.400	1.000	1.000	1.000	0.900	0.900	0.500	0.600	1.000	1.000
180.200	0.500	**	**	0.500	0.600	0.600	**	**	1.000	**	**	**	**	**	**
182.200	0.300	**	**	0.500	0.400	0.400	**	**	1.000	**	**	**	**	**	**
222.200	0.500	**	**	0.500	0.600	0.600	**	**	1.000	**	**	**	**	**	**
225.200	0.200	**	**	0.500	0.400	0.400	**	**	1.000	**	**	**	**	**	**

- The table of the dynamic mesh events can also be seen in the report.

3.3. Dynamic Mesh Setup

Table 6. Dynamic Mesh Events

At Crank Angle (deg)	Name	Description
0.000	dt-bound-start-at-0.00(0.25)	
40.750	dt-insert-layer1-start-at-40.75(0.125)	
41.750	key-grid-41.750, insert-interior-layer-1	
42.750	dt-insert-layer1-end-at-42.75(0.25)	
160.000	key-grid-160.000	
160.200	reduce-urf-due-to-open-exvalve1, change-positivity-at-valve-open, open-exvalve1, start-smoothing-at-exvalve1-open, dt-exvalve1-open-start-at-160.20(0.125), write-solution-point-at-ca-160.200	epsilon=0.8, k=0.8, mom=0.5, pressure=0.5, temperature=1. Reducing URFs 1deg before valve opening for solution stability. Saves solution files at this point.
162.200	increase-urf-due-to-open-exvalve1, change-positivity-after-valve-open	epsilon=0.4, k=0.4, mom=0.5, pressure=0.3, temperature=1. Increasing URFs for accelerating the solution.
163.200	change-positivity-after-valve-open	
165.200	dt-exvalve1-open-end-at-165.20(0.25)	
180.000	save-residual-plot-180	Saves the residual plot image from last saved iteration to the current iteration.
185.300	stop-smoothing-after-exvalve1-open	Stops smoothing in vlayer and starts layering.
316.875	dt-delete-layer1-start-at-316.88(0.125)	
317.875	key-grid-317.875, delete-interior-layer-1	
318.875	dt-delete-layer1-end-at-318.88(0.25)	
322.300	ice-inert-egr-reset	
323.000	key-grid-323.000, inert-equation-on-event	
323.300	reduce-urf-due-to-open-invalve1, change-positivity-at-valve-open, open-invalve1, start-smoothing-at-invalve1-open, dt-invalve1-open-start-at-323.30(0.125), write-solution-point-at-ca-323.300	epsilon=0.8, k=0.8, mom=0.5, pressure=0.5, temperature=1. Reducing URFs 1deg before valve opening for solution stability. Saves solution files at this point.
325.300	increase-urf-due-to-open-invalve1, change-positivity-after-valve-open	epsilon=0.4, k=0.4, mom=0.5, pressure=0.3, temperature=1. Increasing URFs for accelerating the solution.
326.300	change-positivity-after-valve-open	
328.300	dt-invalve1-open-end-at-328.30(0.25)	
348.400	stop-smoothing-after-invalve1-open	Stops smoothing in vlayer and starts layering.
360.000	save-residual-plot-360	Saves the residual plot image from last saved iteration to the current iteration.



365.400	start-smoothing-before-exvalve1-close	Stops layering in vlayer and starts smoothing.
397.300	change-positivity-at-valve-close, dt-exvalve1-close-start-at-397.30(0.125)	
400.750	dt-insert-layer2-start-at-400.75(0.125)	
401.750	key-grid-401.750, insert-interior-layer-2	
402.300	close-exvalve1, dt-exvalve1-close-end-at-402.30(0.125), write-solution-point-at-ca-402.300	Deleting interface between vlayer and chamber for stopping flow. Saves solution files at this point.
402.750	dt-insert-layer2-end-at-402.75(0.25)	
403.300	change-positivity-at-valve-close	
404.300	change-positivity-at-valve-close	
419.000	pdf-equation-on-event, write-solution-point-at-ca-419.000	Saves solution files at this point.
420.000	enable-pt-cal-act-for-injection-2, enable-pt-cal-act-for-injection-1, enable-pt-cal-act-for-injection-0, enable-writing-dpm-monitors-to-file, dt-injection-0-unsteady-ca-start-at-420.00(0.05)	
438.400	disable-pt-cal-act-for-injection-2, disable-pt-cal-act-for-injection-1, disable-pt-cal-act-for-injection-0, dt-injection-0-unsteady-ca-end-at-438.40(0.25)	
440.000	write-solution-point-at-ca-440.000	Saves solution files at this point.
528.700	start-smoothing-before-invalve1-close	Stops layering in vlayer and starts smoothing.
540.000	save-residual-plot-540	Saves the residual plot image from last saved iteration to the current iteration.
558.000	change-positivity-at-valve-close, dt-invalve1-close-start-at-558.00(0.125)	
563.000	close-invalve1, dt-invalve1-close-end-at-563.00(0.25), write-solution-point-at-ca-563.000	Deleting interface between vlayer and chamber for stopping flow. Saves solution files at this point.
564.000	change-positivity-at-valve-close, key-grid-564.000	
565.000	change-positivity-at-valve-close	
676.875	dt-delete-layer2-start-at-676.88(0.125)	
677.875	key-grid-677.875, delete-interior-layer-2	
678.875	dt-delete-layer2-end-at-678.88(0.25)	
700.000	write-solution-point-at-ca-700.000	Saves solution files at this point.
704.000	premix-g-eqn-equation-on-event, premix-g-var-equation-on-event	
720.000	save-residual-plot-720, dt-bound-end-at-720.00(0.25), write-solution-point-at-ca-720.000	Saves the residual plot image from last saved iteration to the current iteration. Saves solution files at this point.

You can check the engine inputs.

3.4. IC Engine System Inputs

Engine Inputs

Engine Speed (rev/min) : 2000
Crank Radius (mm) : 45
Piston Pin Offset/Wrench (mm) : 0
Connecting Rod Length (mm) : 144.3
Minimum Lift (mm) : 0.2

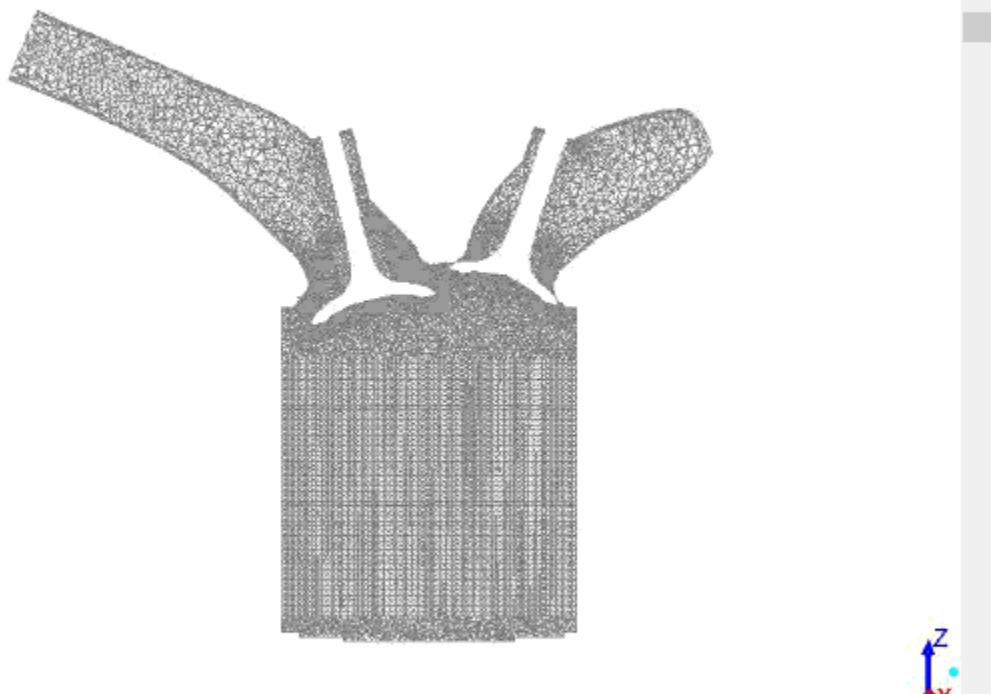
Journal Customization

Pre Iteration Journal File : N/A
Post Iteration Journal File : N/A

- Check the animation of mesh on the cut-plane in the section **Solution Data**.

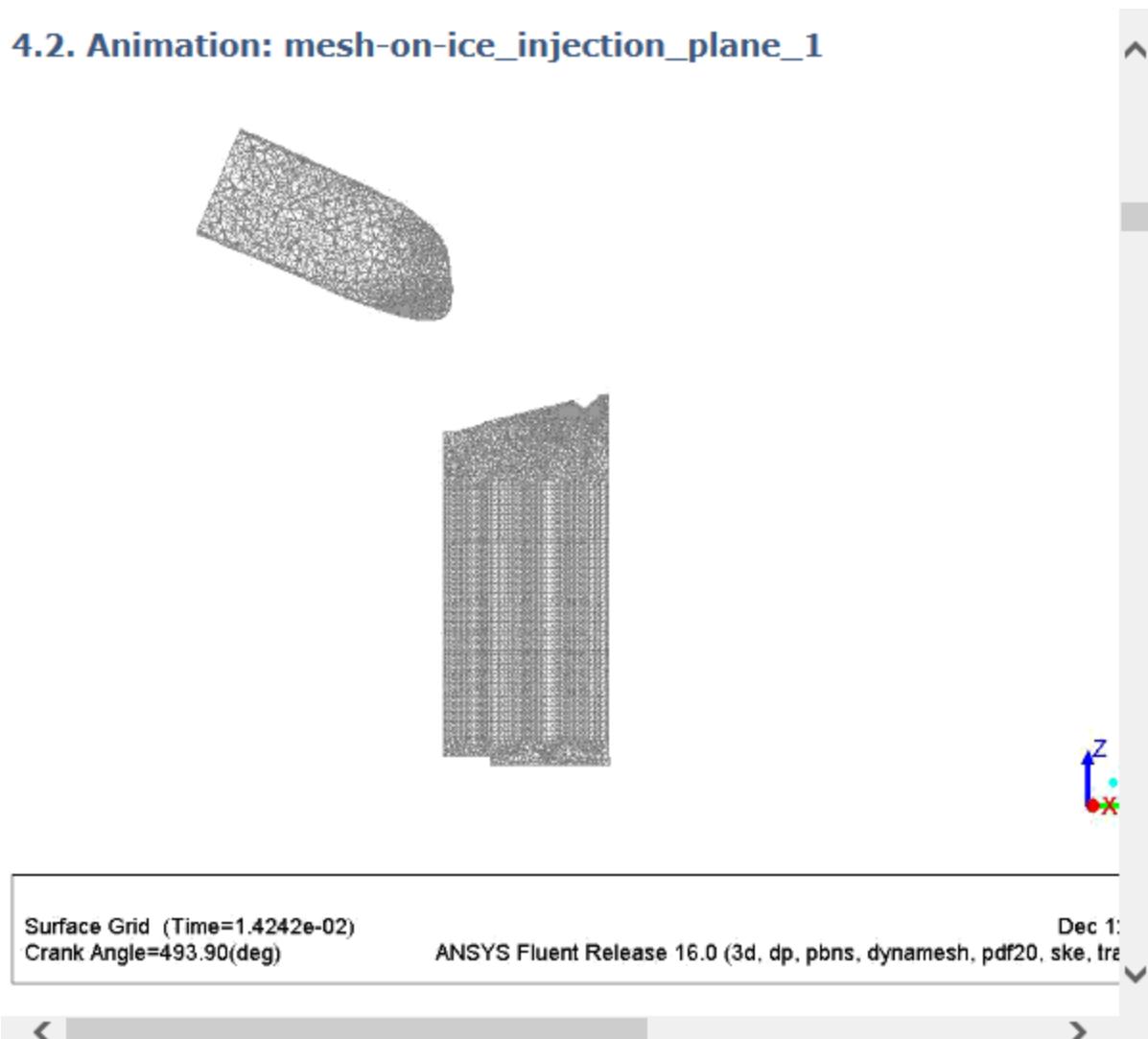
4. Solution Data

4.1. Animation: mesh-on-ice_cutplane_1



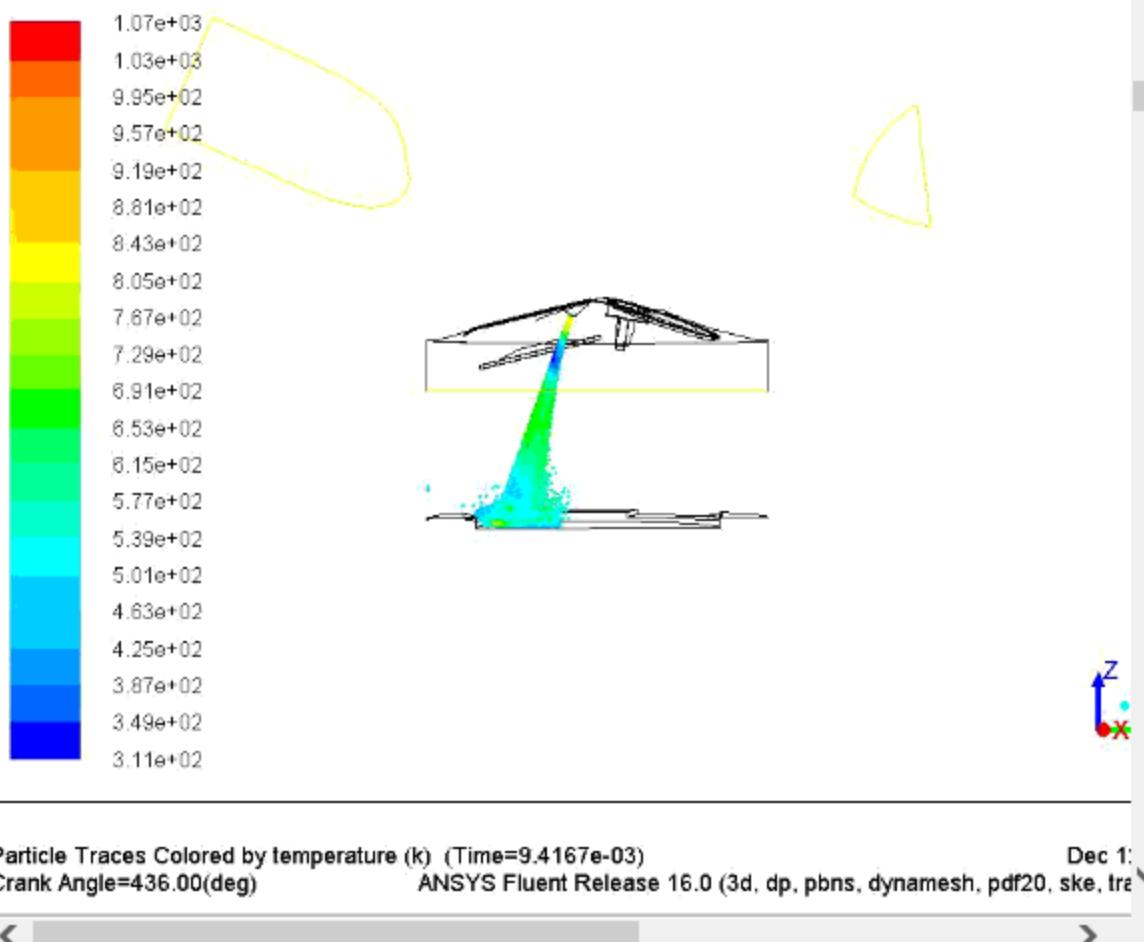
There are also additional mesh animations on the three injection planes.

4.2. Animation: mesh-on-ice_injection_plane_1

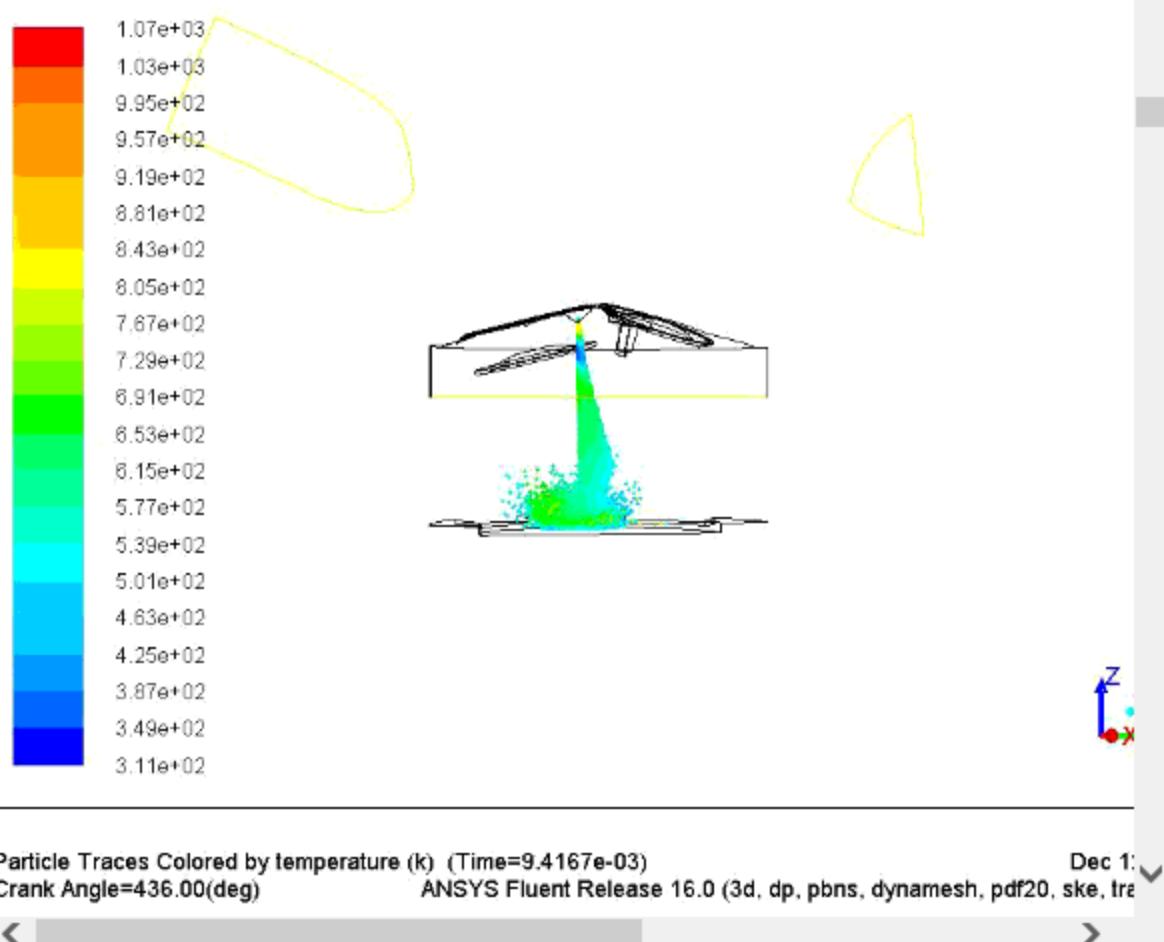


- Check the animations of particle traces of three injections. The animation are shown from different views.

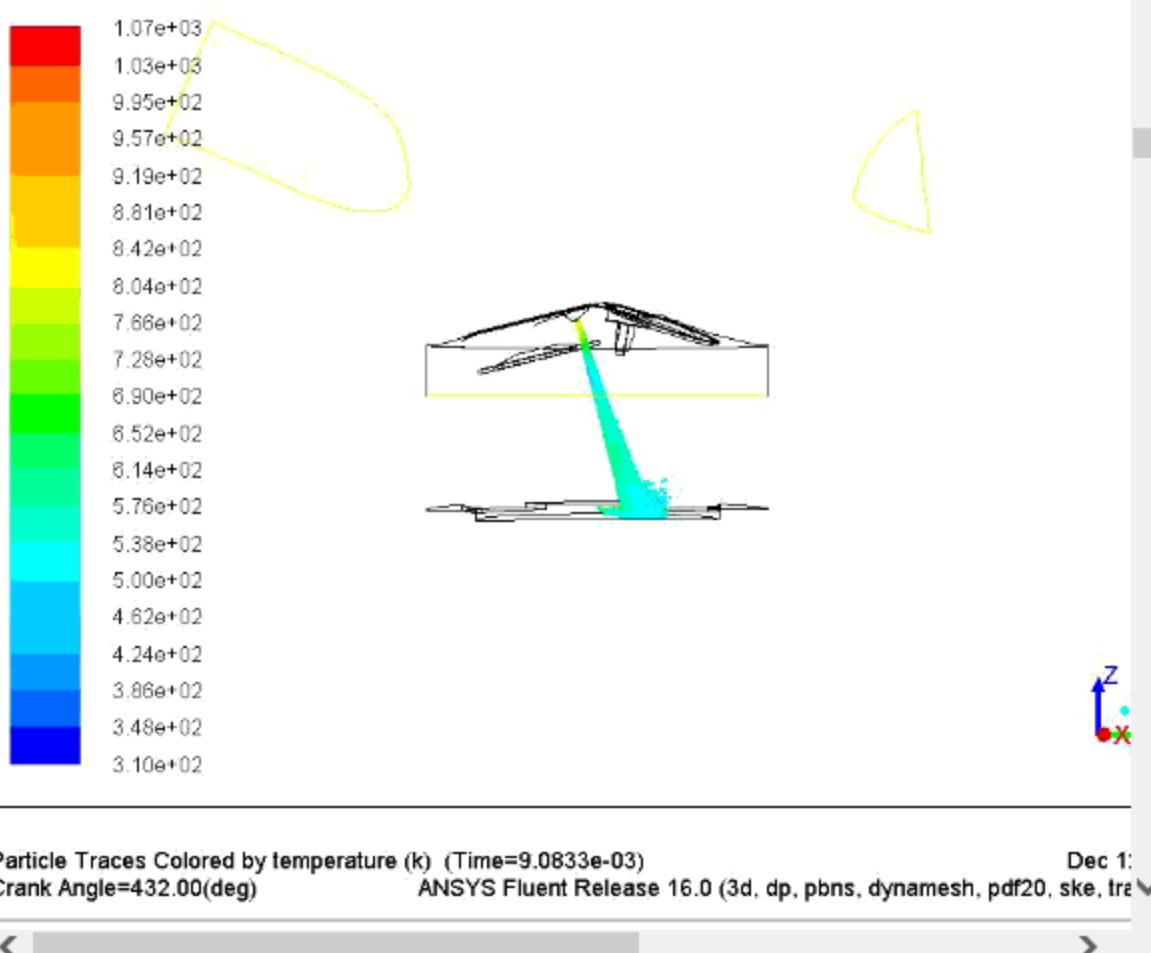
4.5. Animation: pt-temperature on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



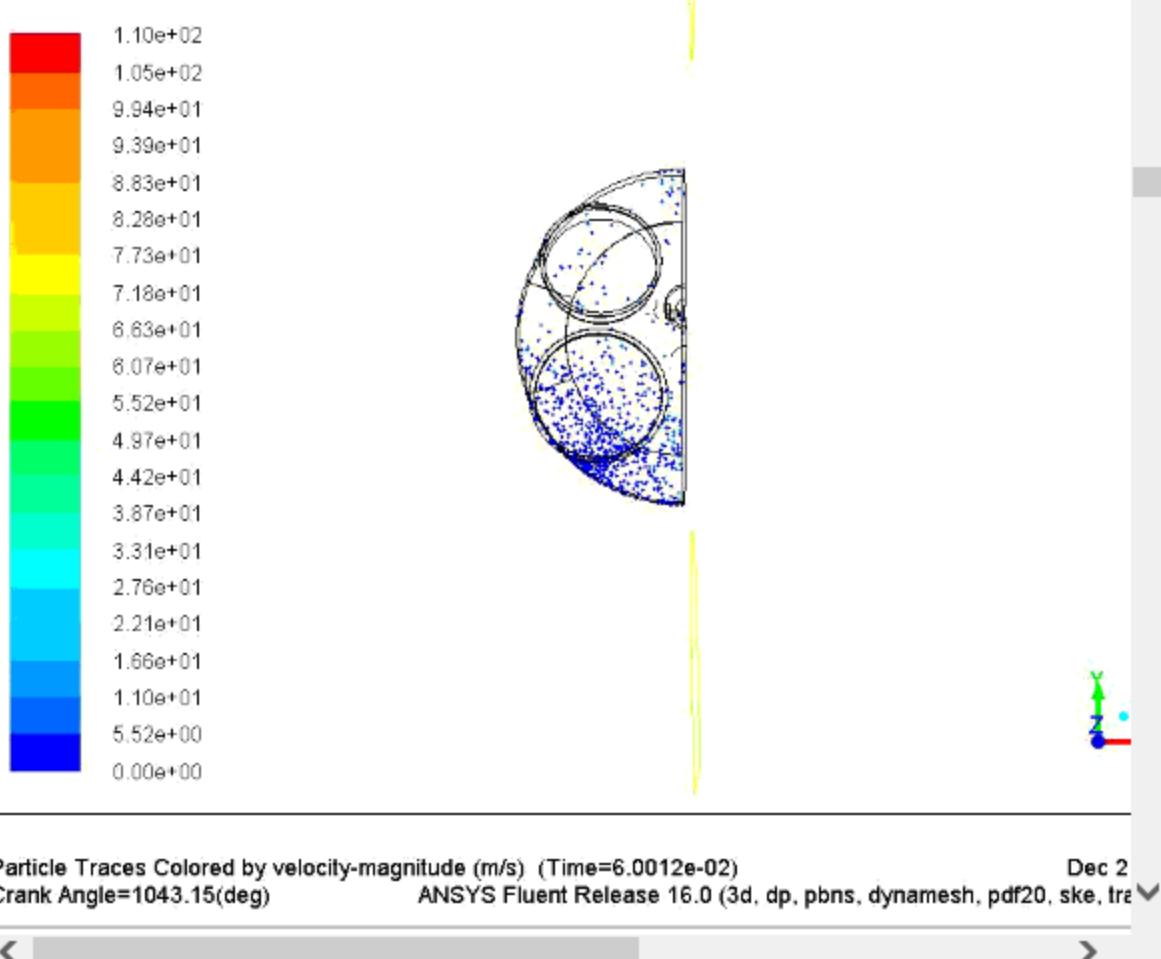
4.6. Animation: pt-temperature on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



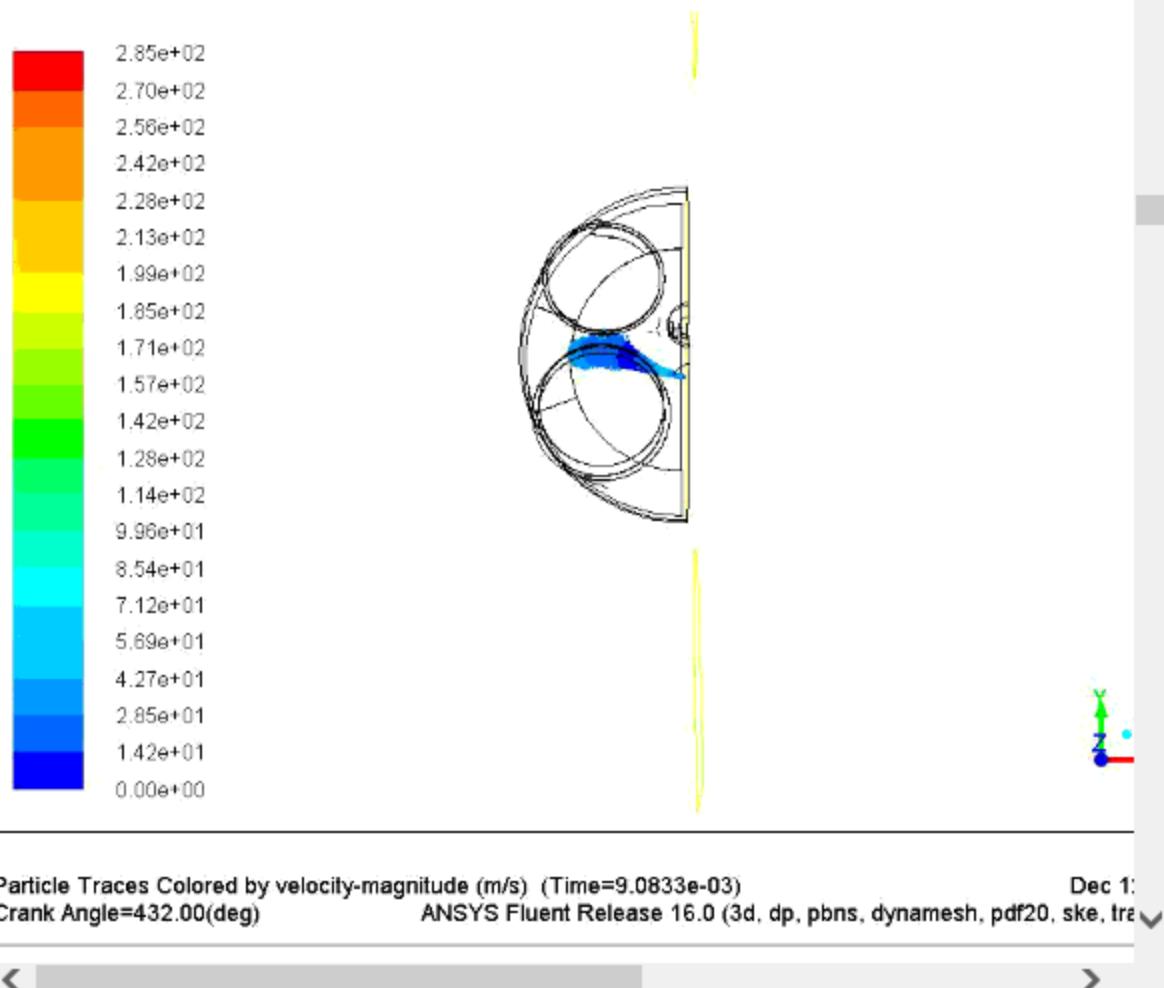
4.7. Animation: pt-temperature on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



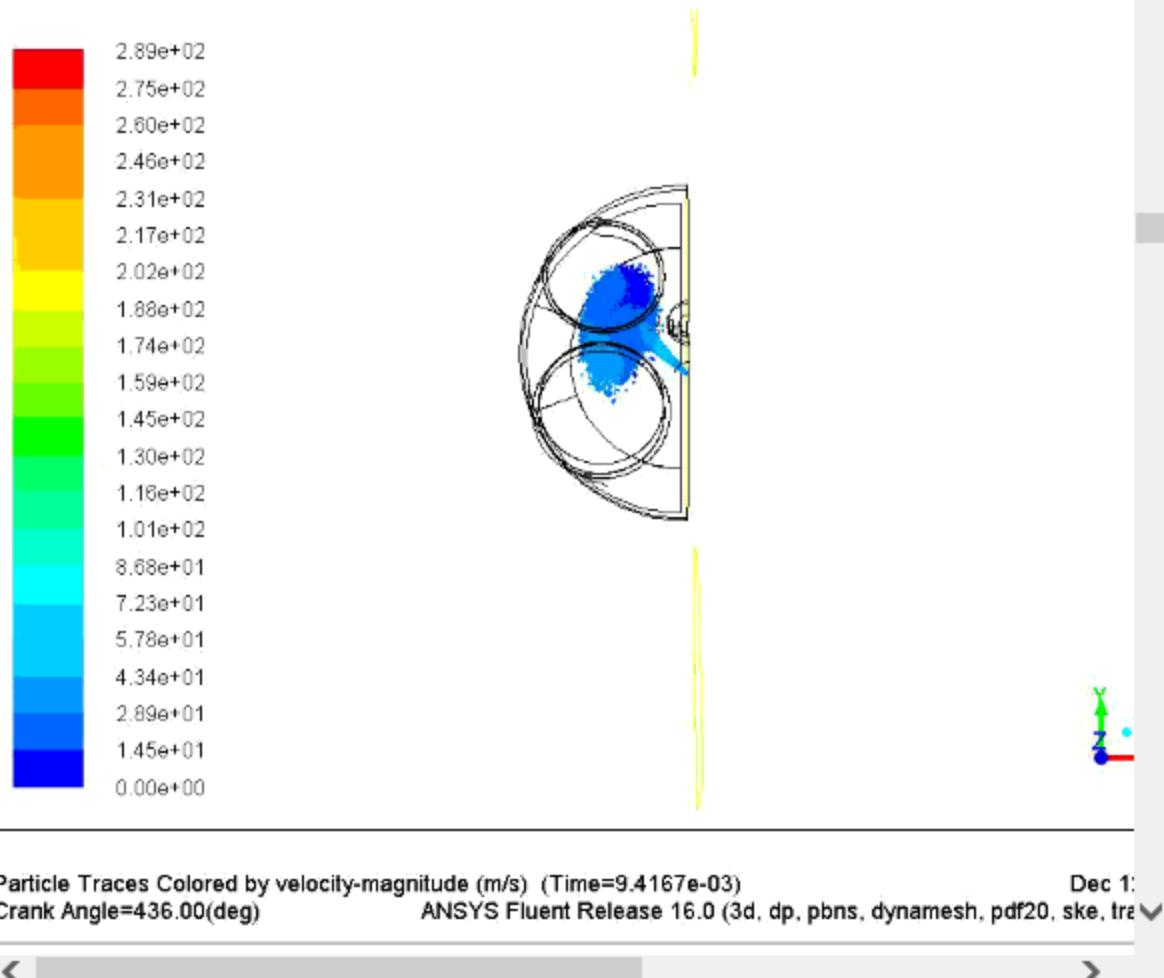
4.8. Animation: pt-velocity-magnitude on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



4.9. Animation: pt-velocity-magnitude on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port

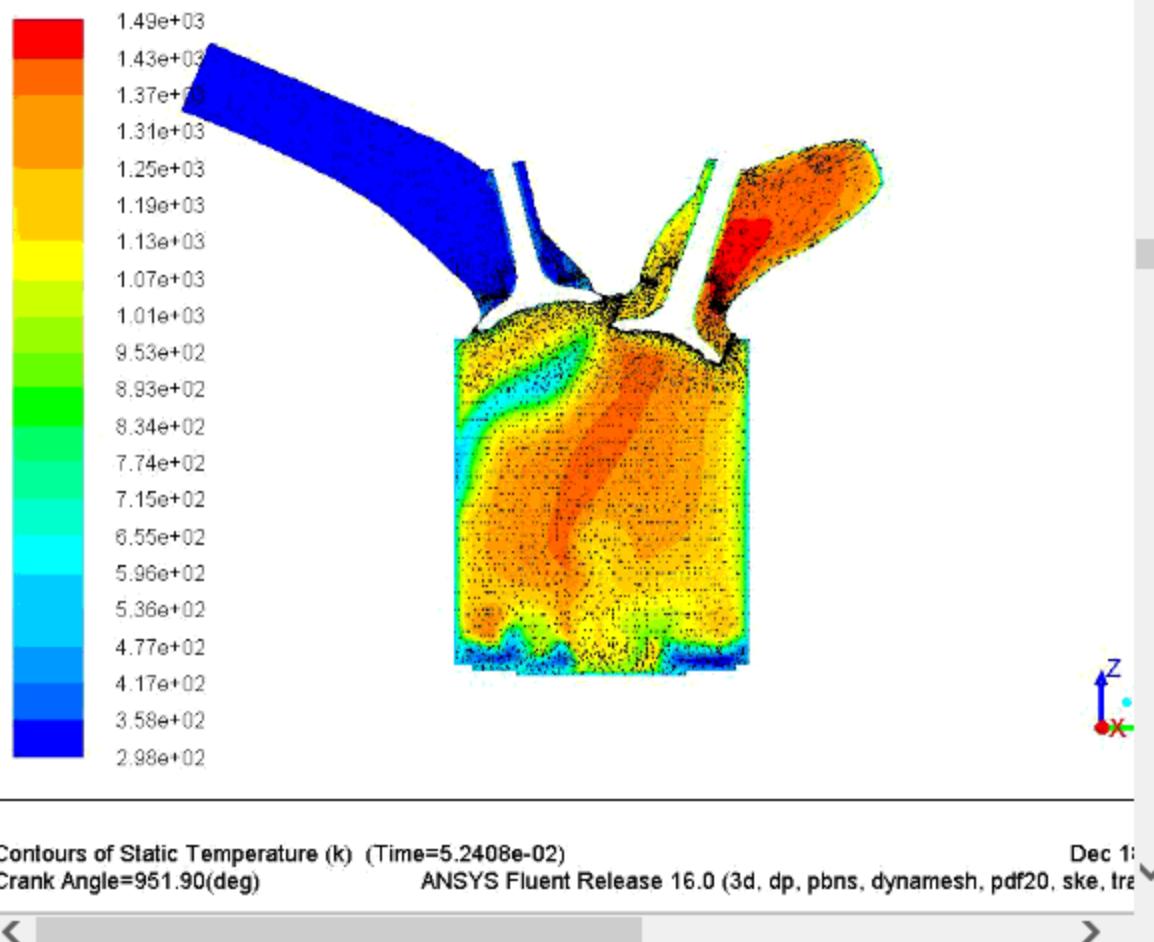


4.10. Animation: pt-velocity-magnitude on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port

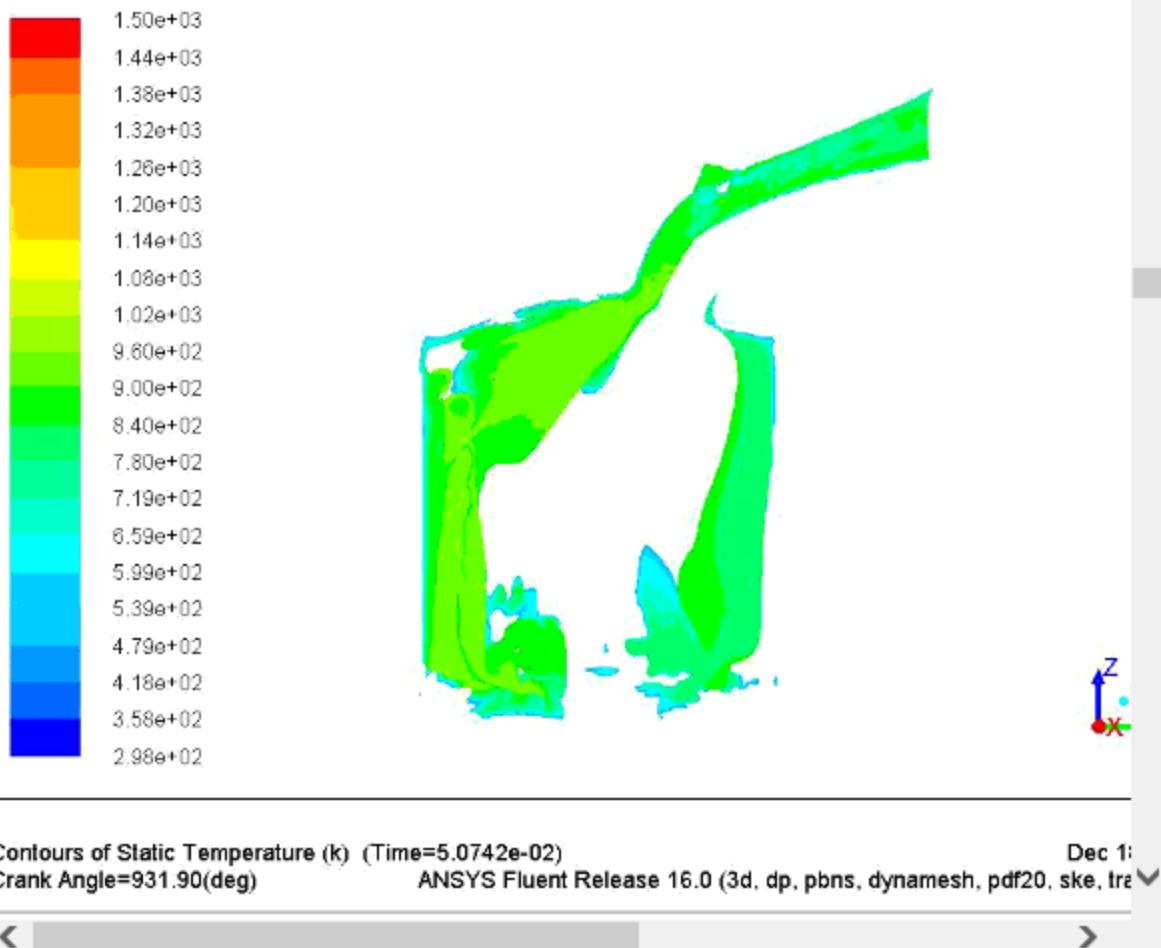


- Check the animations of temperature on cut-plane as well as the iso surface.

4.11. Animation: temperature on ice_cutplane_1

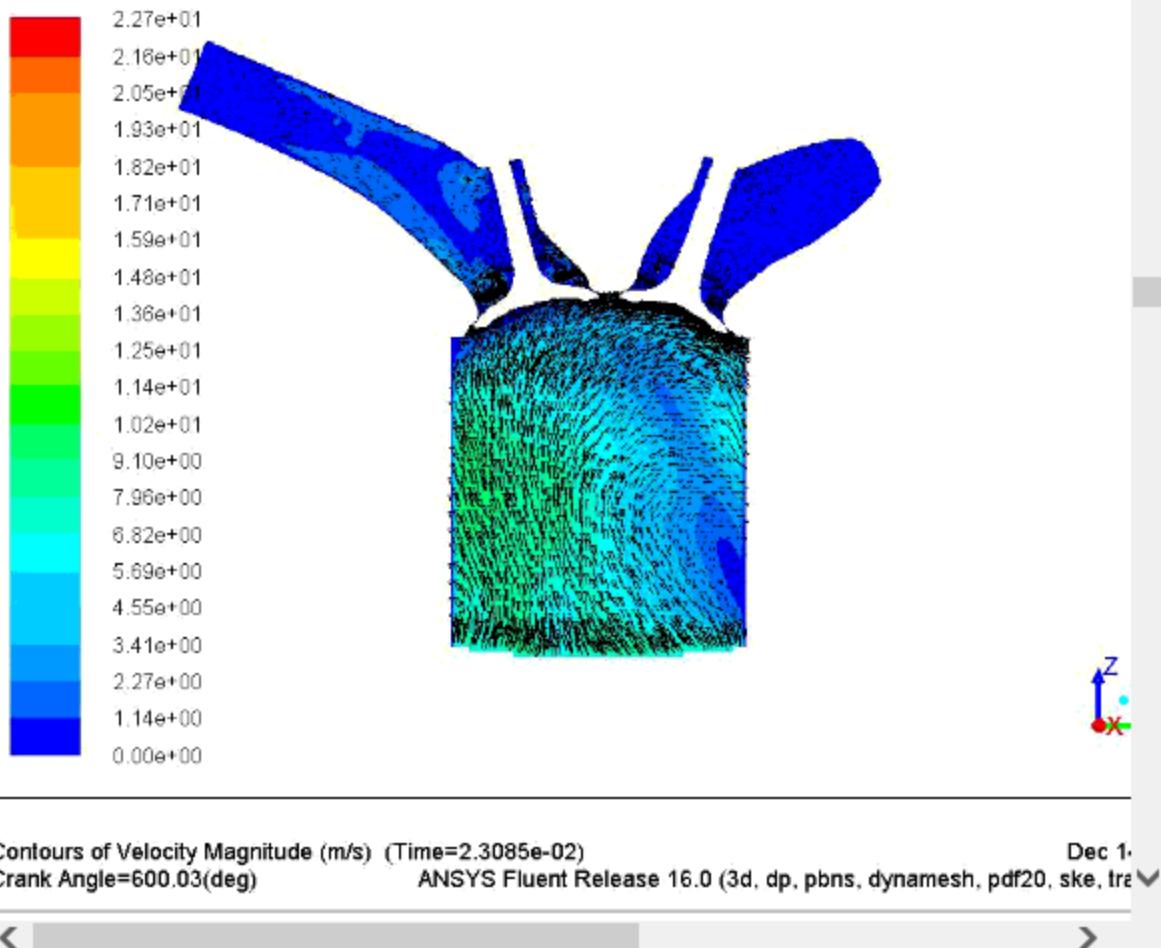


4.12. Animation: temperature on iso-surface-flame



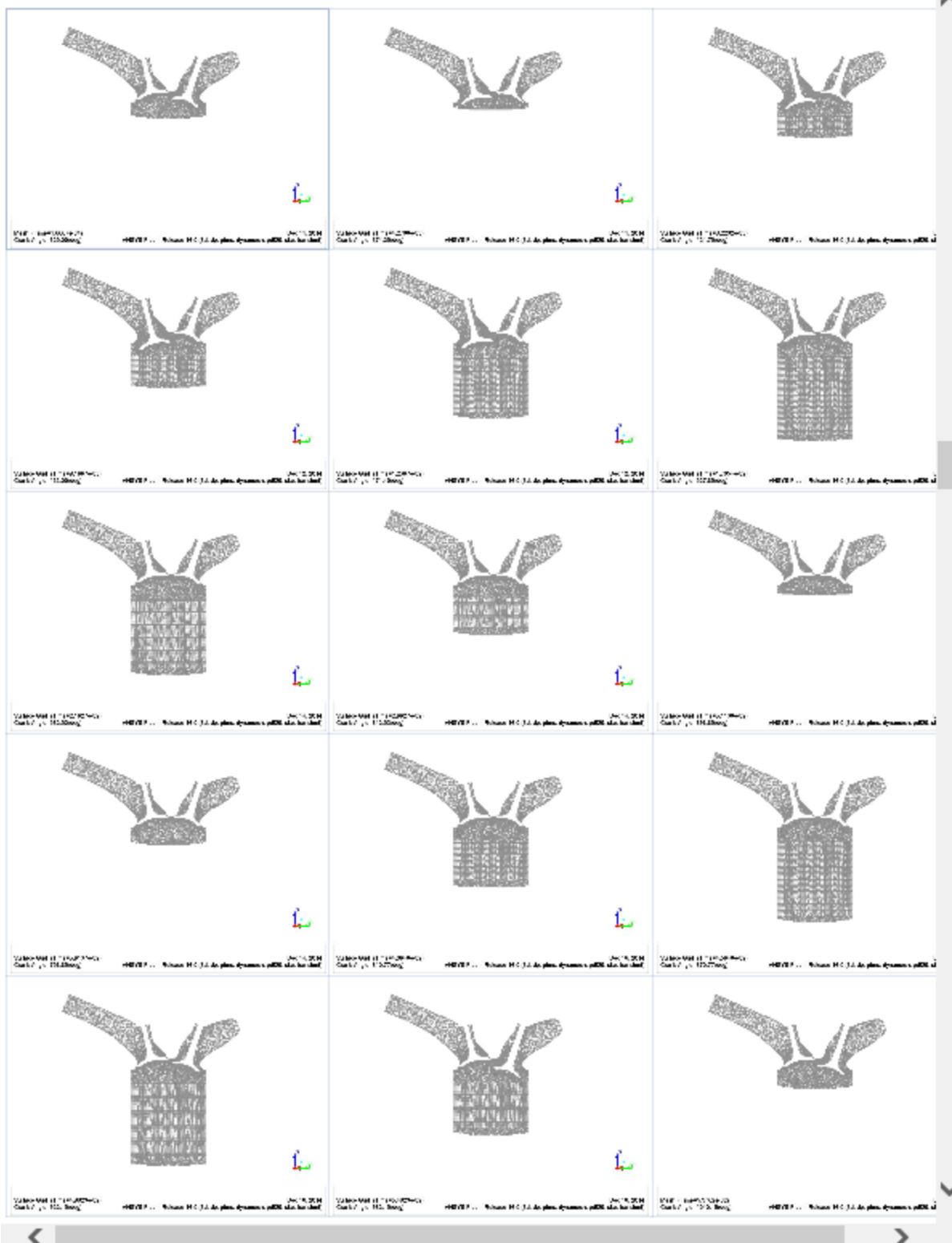
- Check the animation of velocity magnitude on the cut-plane.

4.13. Animation: velocity-magnitude on ice_cutplane_1



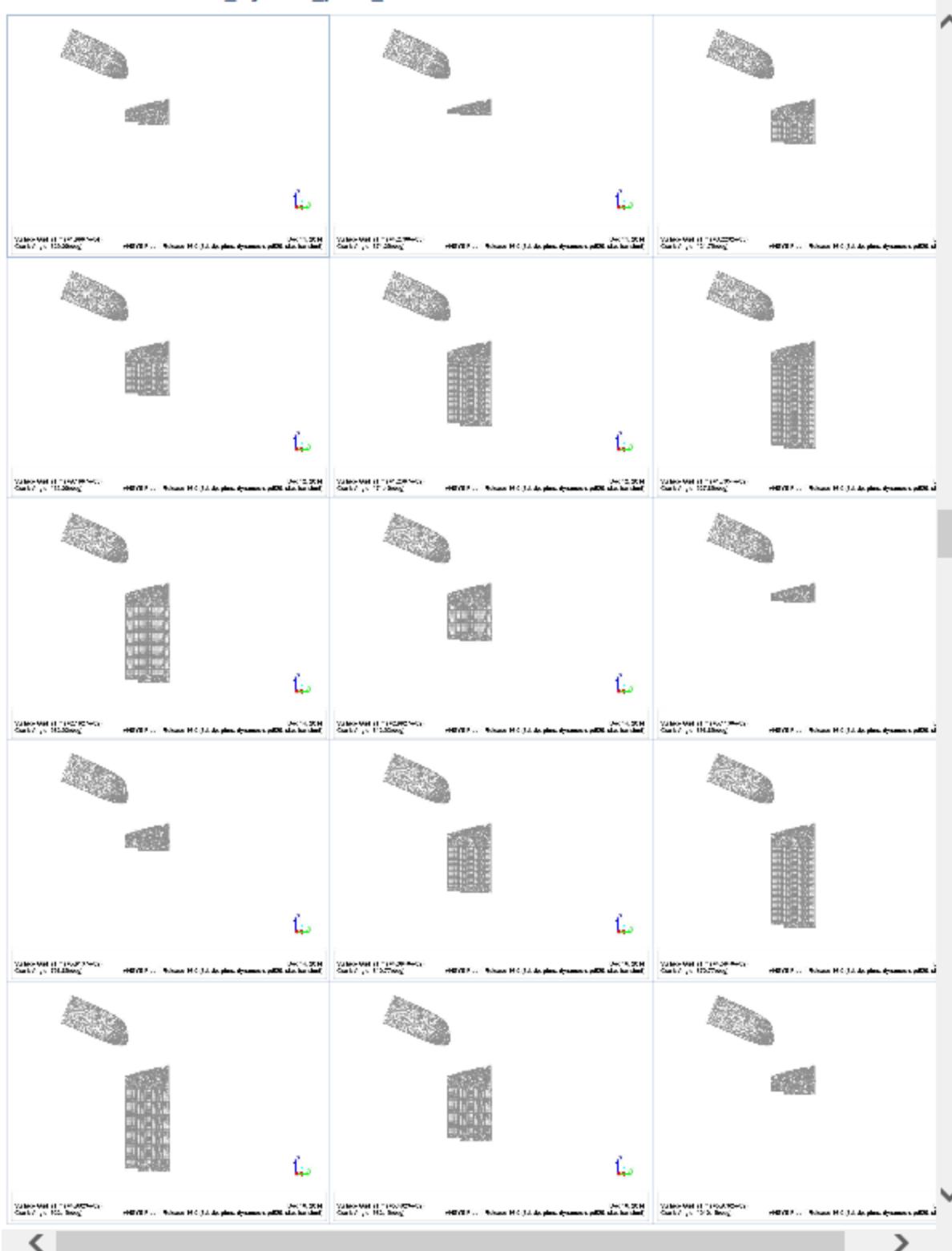
- In a **Table** you can observe the mesh images at various stages of the simulation.

4.14. Table: mesh-on-ice_cutplane_1



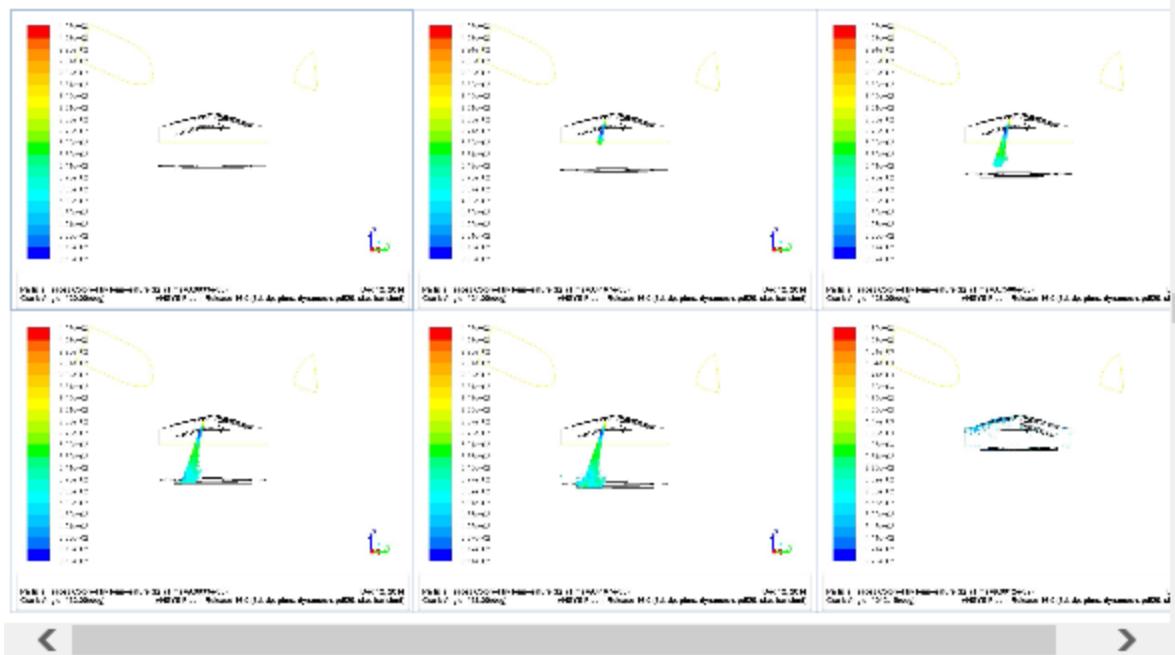
There are also additional mesh animations on the three injection planes.

4.15. Table: mesh-on-ice_injection_plane_1

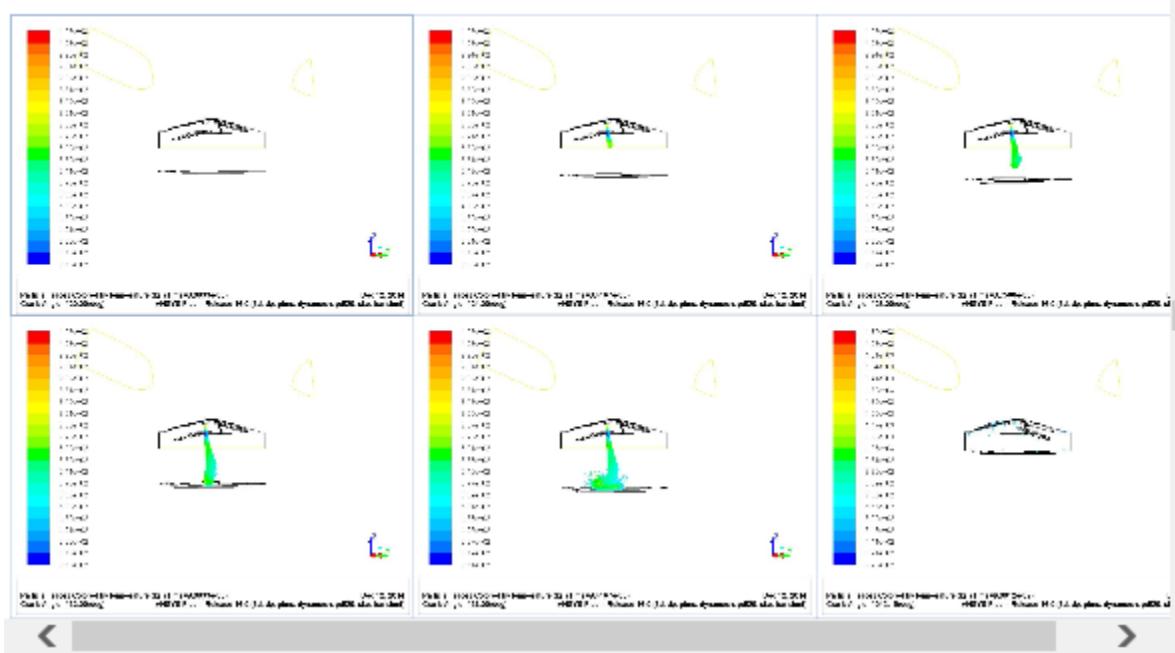


- You can observe the particle traces images at various stages of simulation from two different views in the different tables.

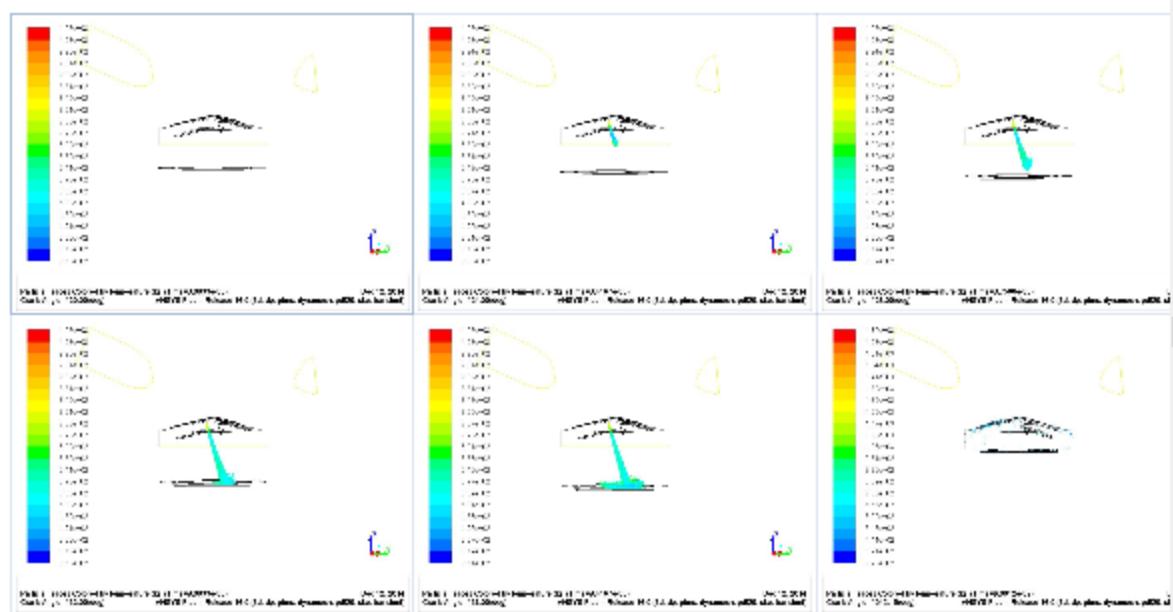
4.18. Table: pt-temperature on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



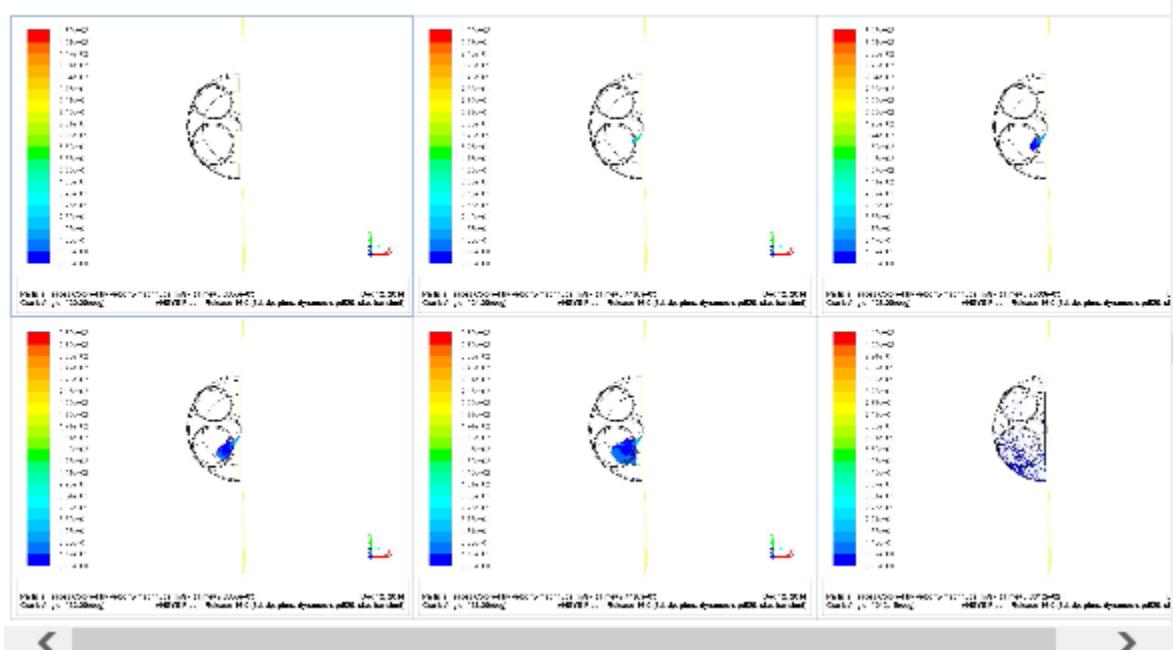
4.19. Table: pt-temperature on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



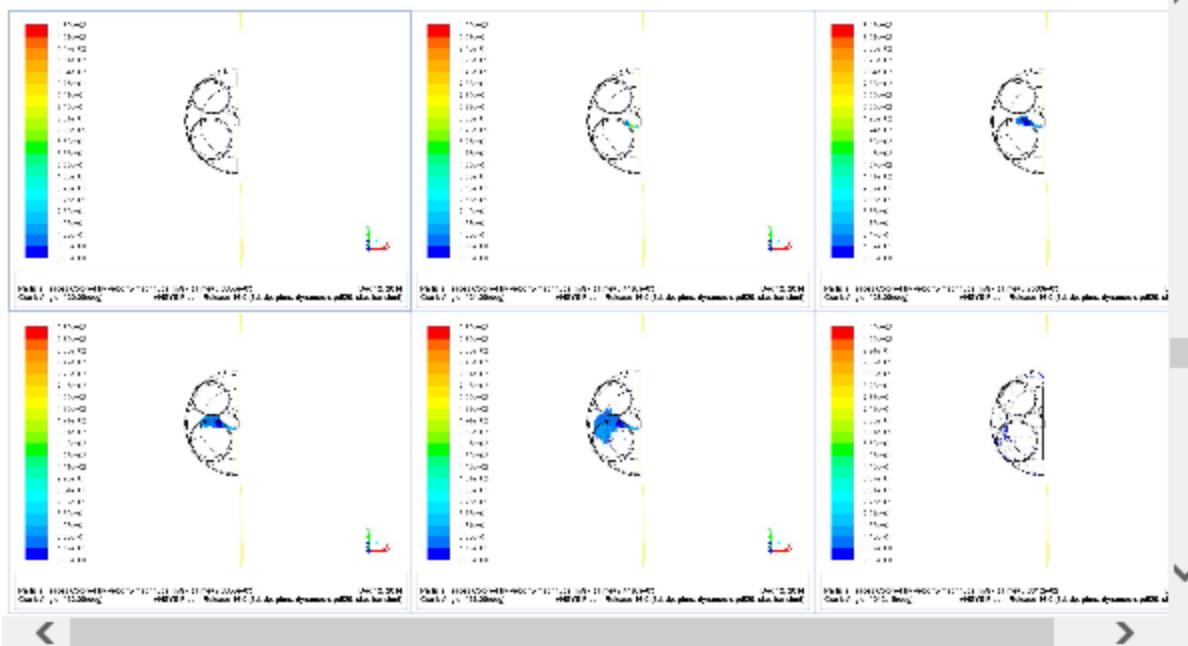
4.20. Table: pt-temperature on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



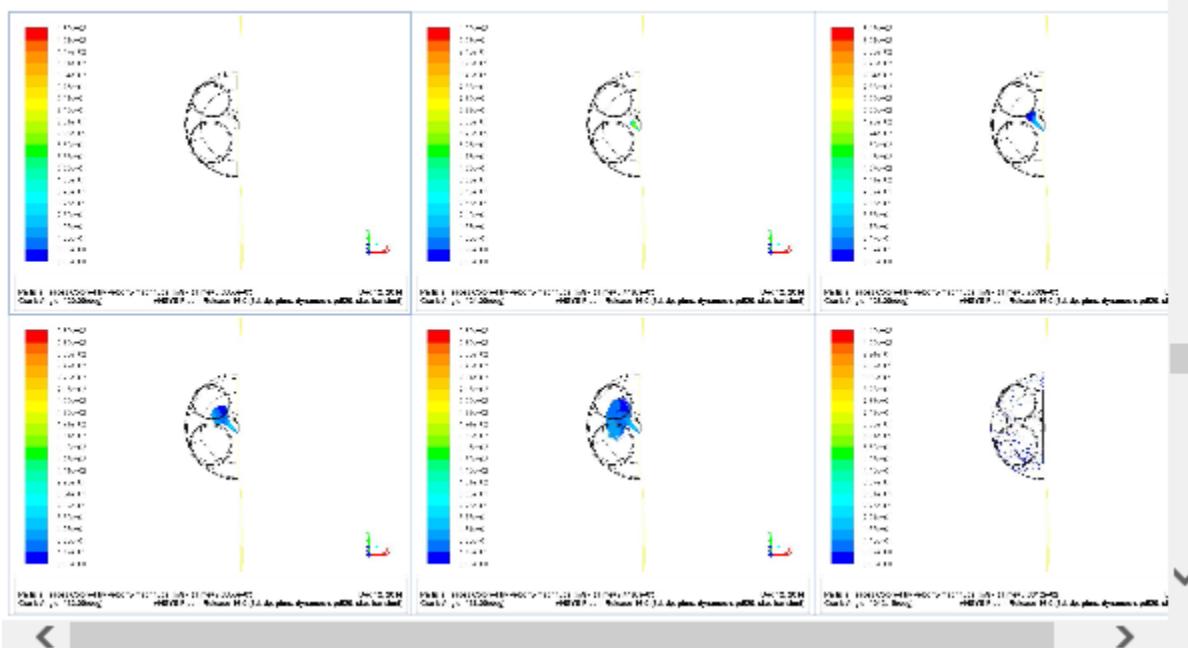
4.21. Table: pt-velocity-magnitude on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



4.22. Table: pt-velocity-magnitude on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port

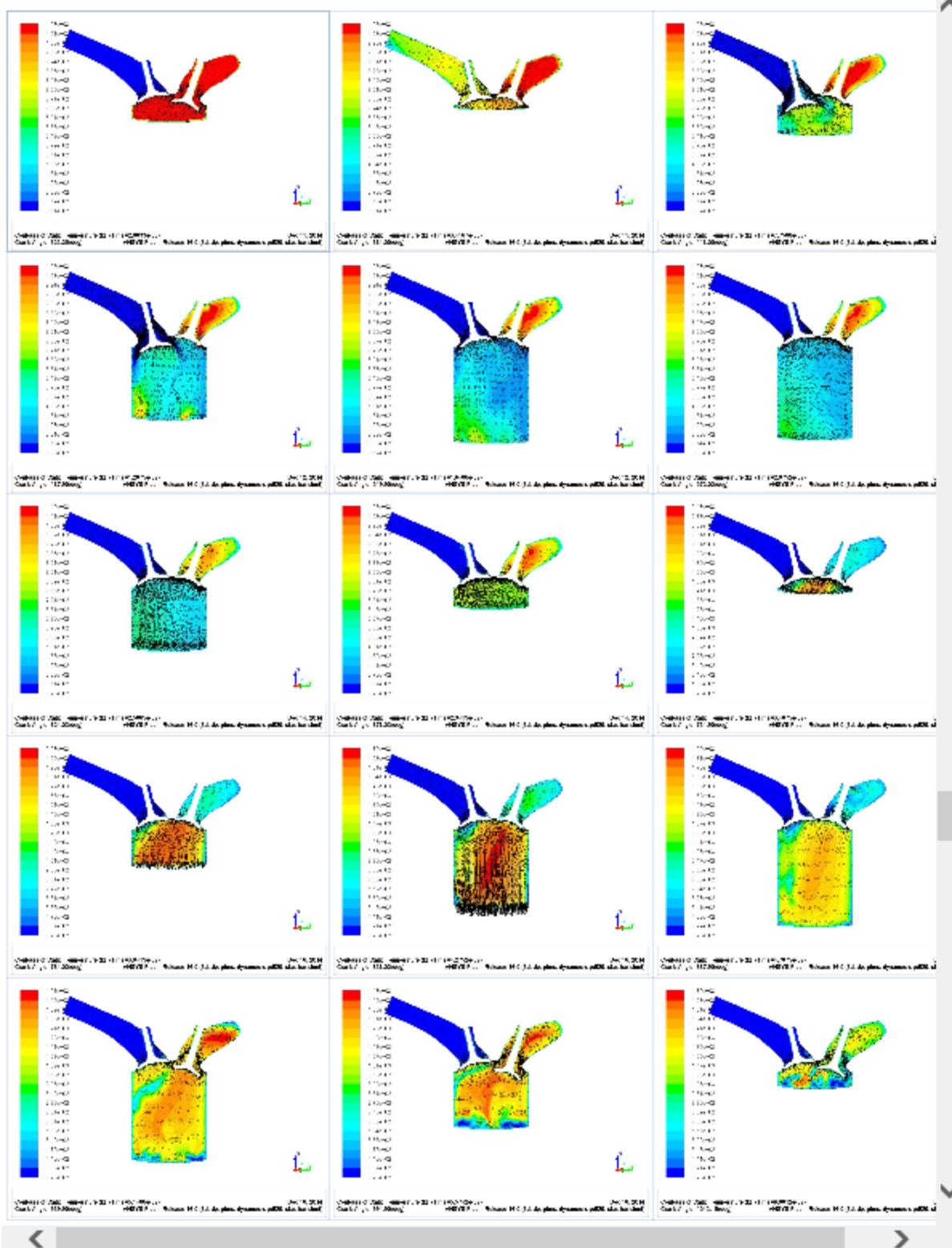


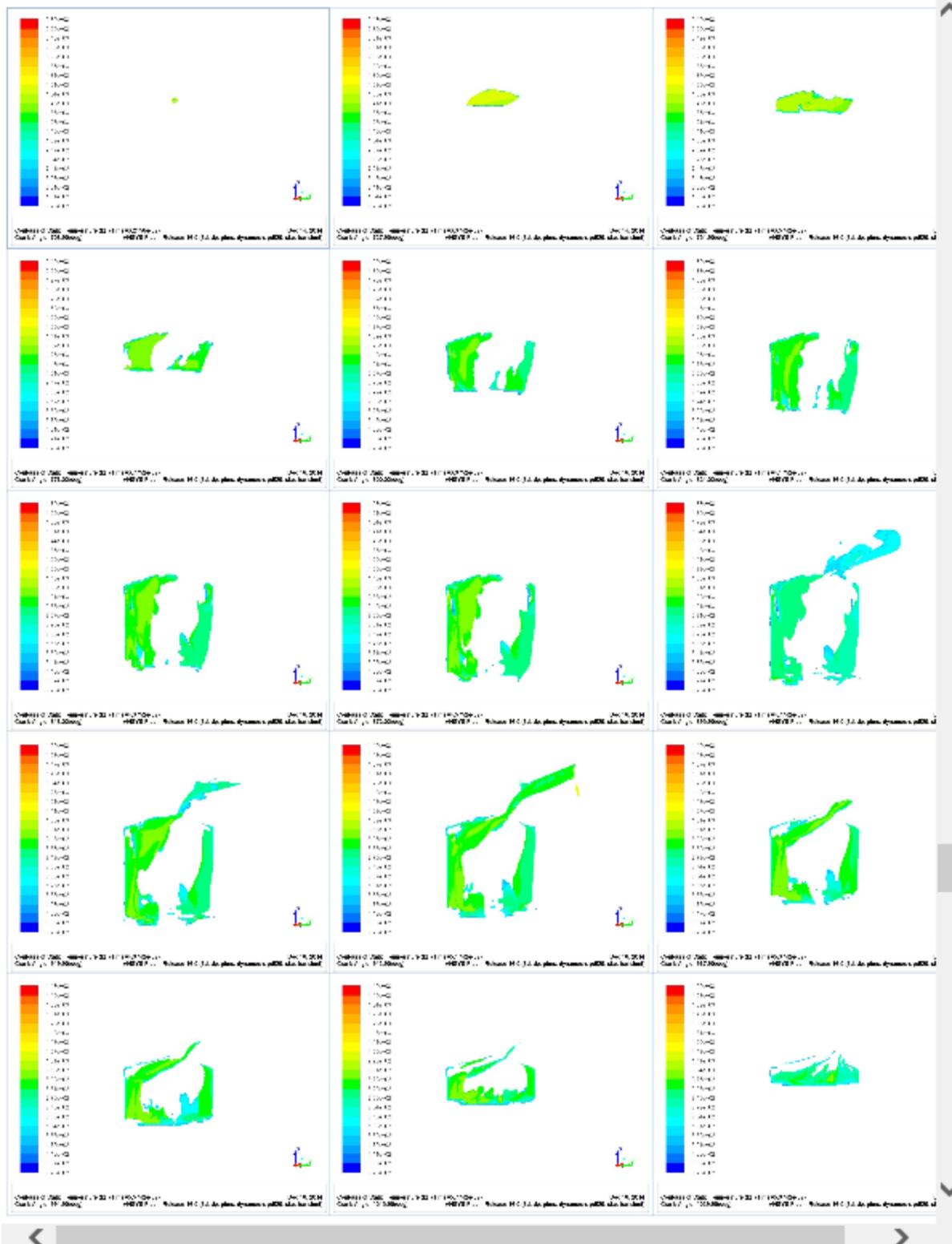
4.23. Table: pt-velocity-magnitude on cyl-head, cyl-tri, exvalve1-ch, invalve1-ch, piston, symm-cyl-tri, symm-invalve-1-port, symm-exvalve-1-port



- In another **Table** you can observe the temperature contours on the cut plane as well as on the iso-surface.

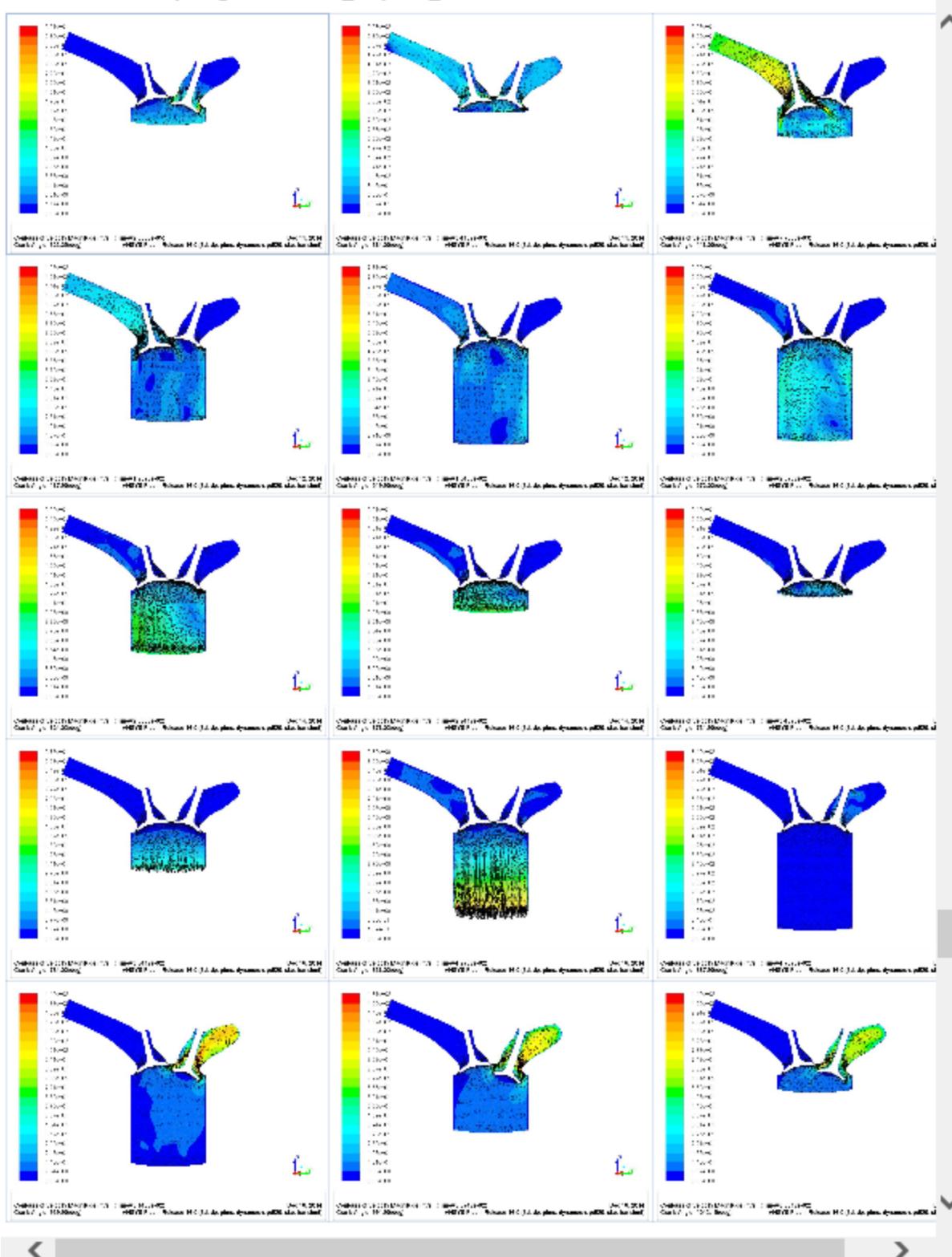
4.24. Table: temperature on ice_cutplane_1



4.25. Table: temperature on iso-surface-flame

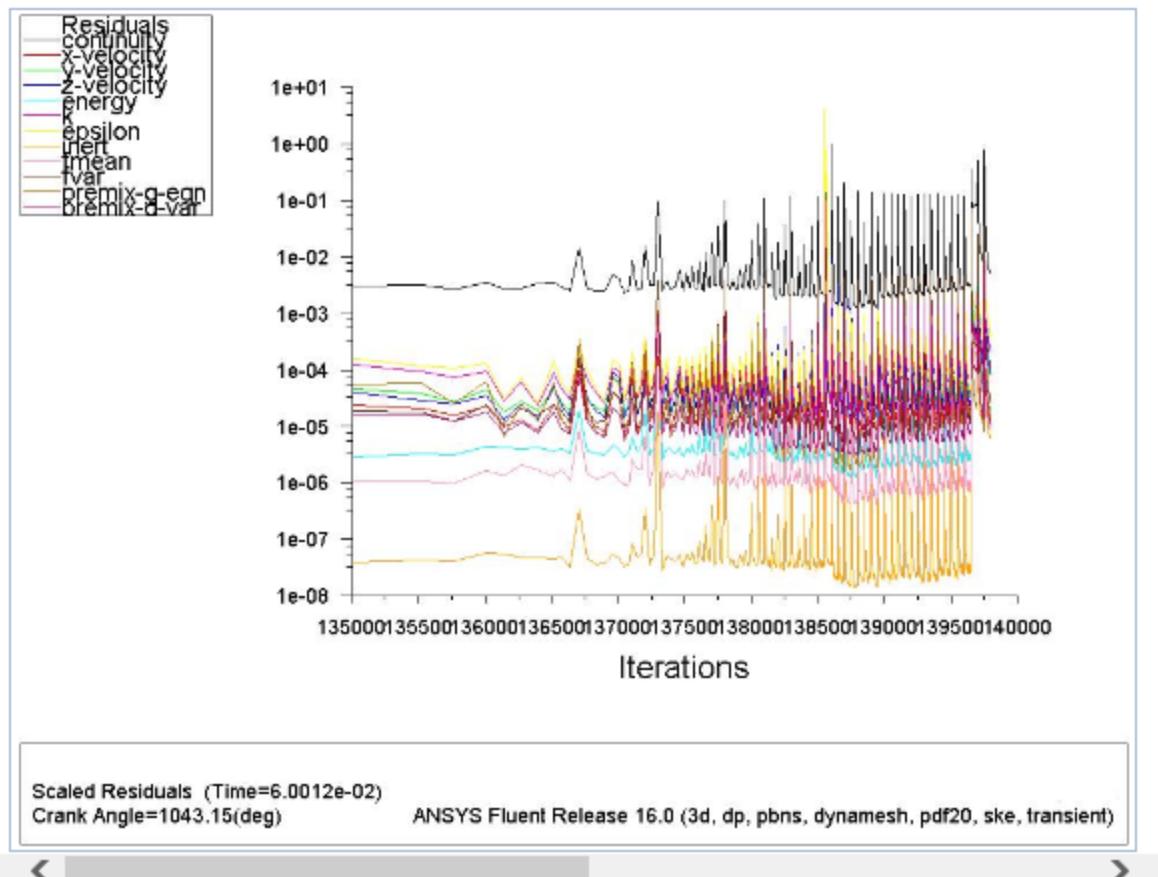
- You can also observe the velocity-magnitude contours on the cut-plane in another **Table**.

4.2b. Table: velocity-magnitude on ice_cutplane_1



- You can check the residual plot in the **Table: Residuals**.

4.27. Table: Residuals



- Under **Charts** you will find plots for the last iteration residual values, **Swirl Ratio**, **Tumble Ratio**, and **Cross Tumble Ratio**.

4.28. Charts

Chart 2. Last iteration residual values corresponding to each time step

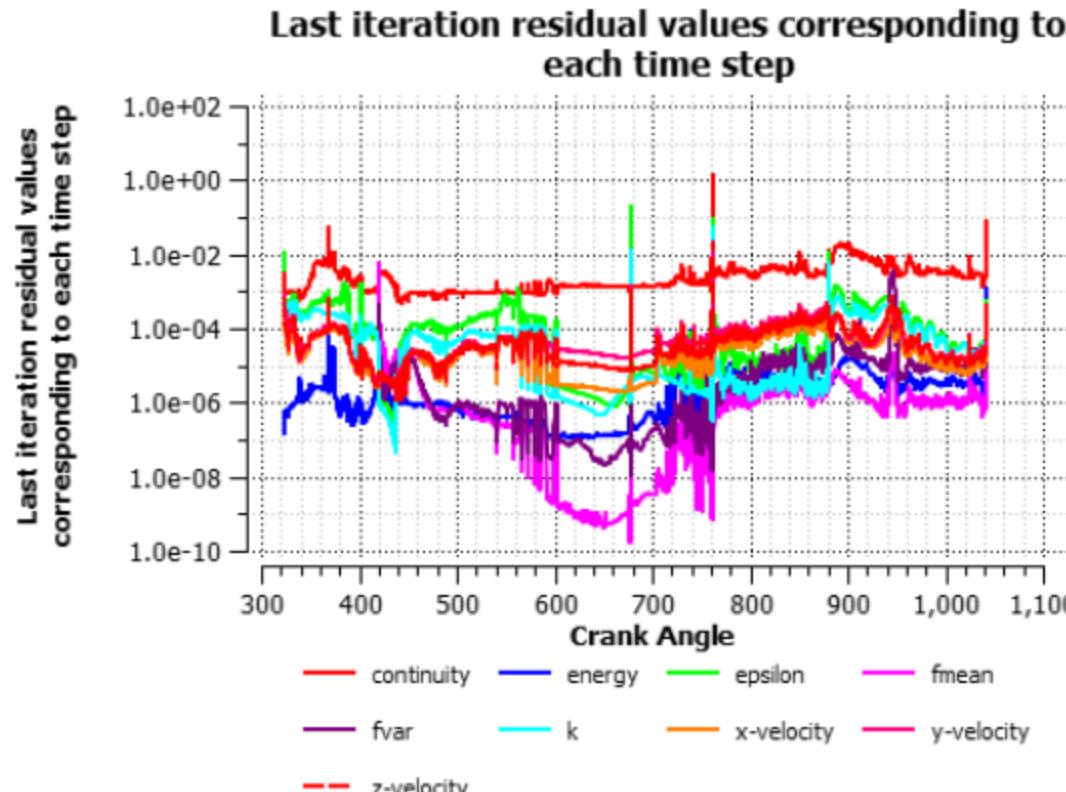


Chart 3. Swirl Ratio

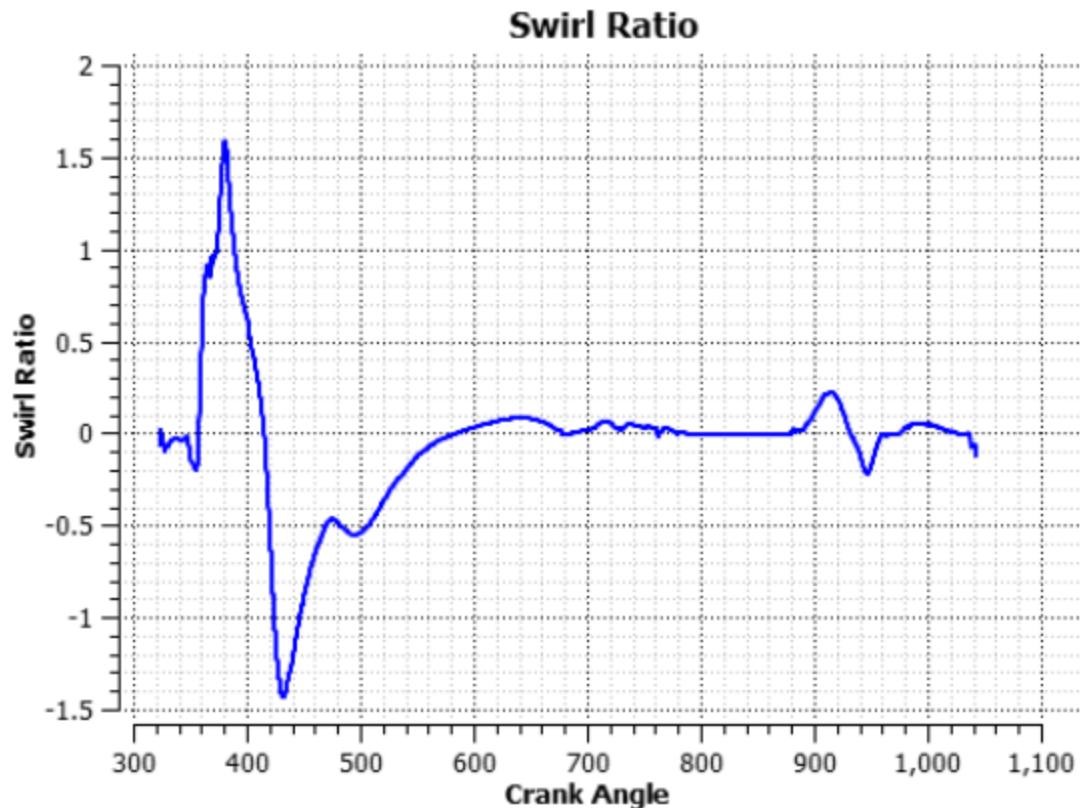


Chart 4. Tumble Ratio

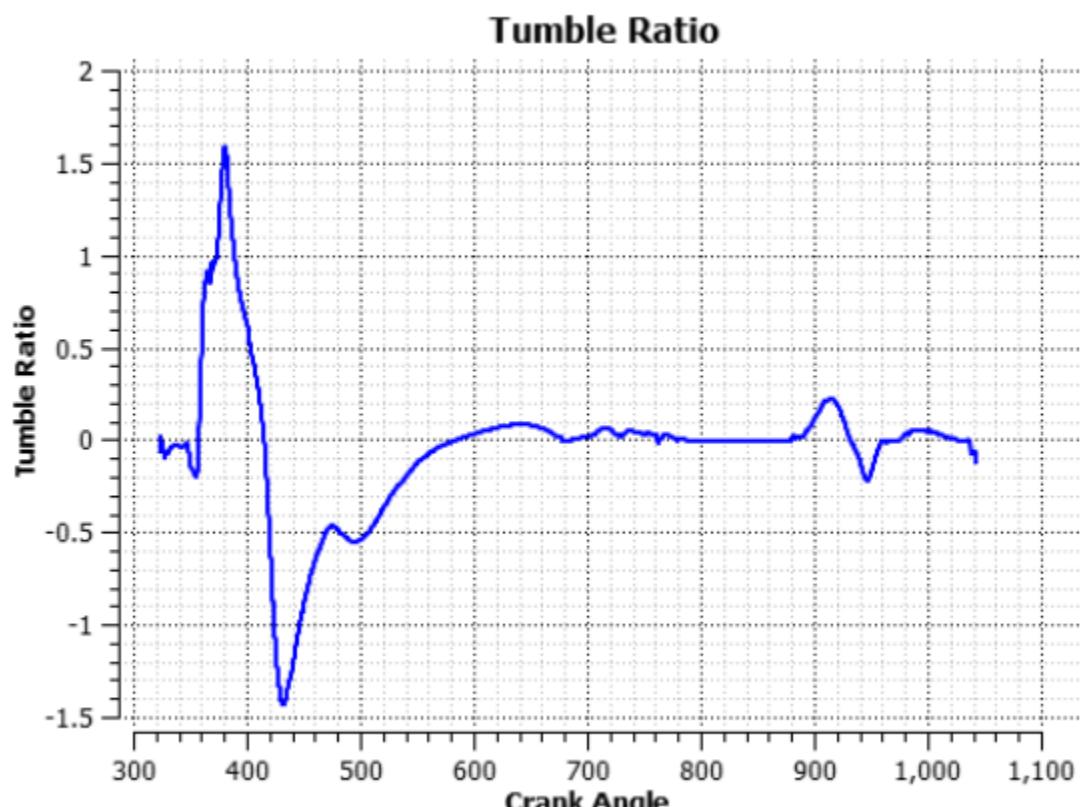
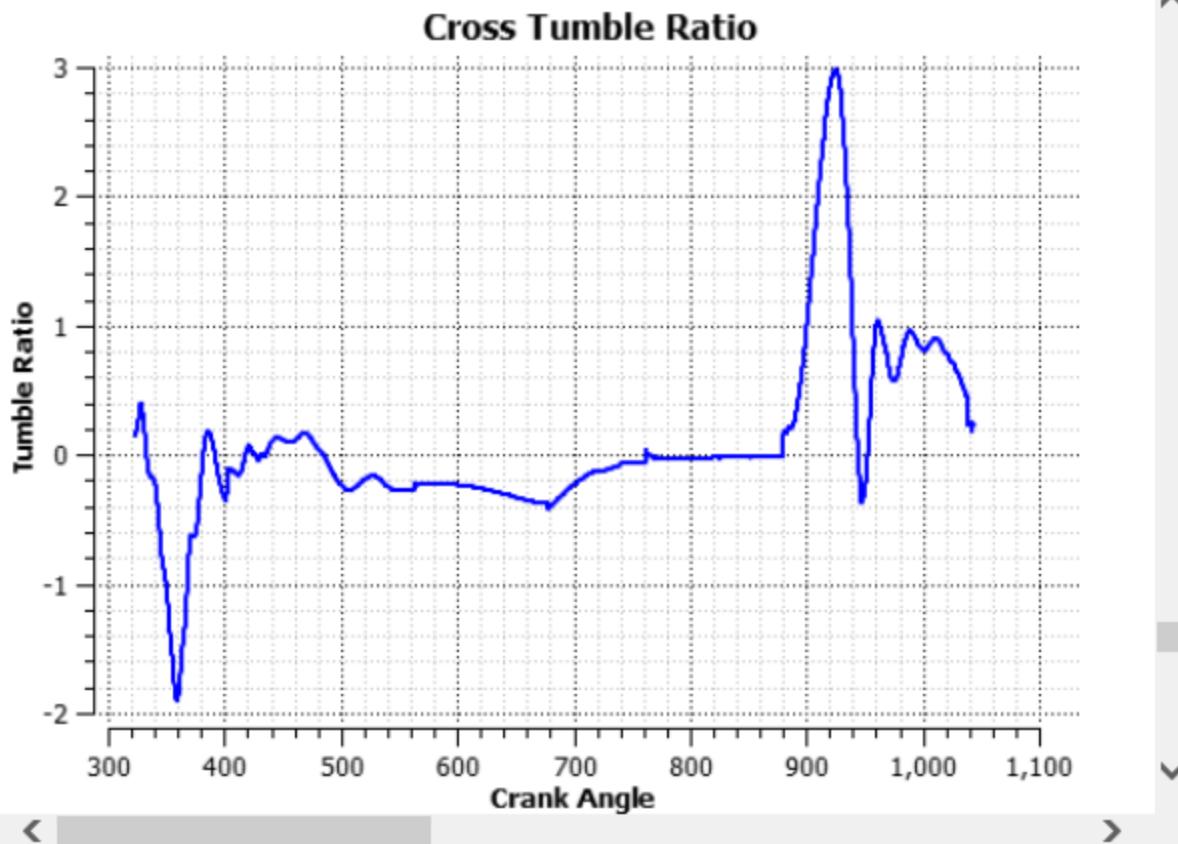


Chart 5. Cross Tumble Ratio

- You will find plot of **AHRR** (Apparent Heat Release Rate) and **Area-Weighted Average Static Temperature**. Apparent heat release rate is defined as:

$$AHRR = \frac{\gamma}{\gamma-1} \times P \times \frac{V_2 - V_1}{A_2 - A_1} + \frac{1}{\gamma-1} \times V \times \frac{P_2 - P_1}{A_2 - A_1}$$

where

γ = 1.35 (can also be computed from Fluent)

V = Volume of sector, m³ X number of sectors

P = Absolute pressure, Pa

A = Crank Angle

Chart 6. Apparent Heat Release Rate on (fluid-ch)

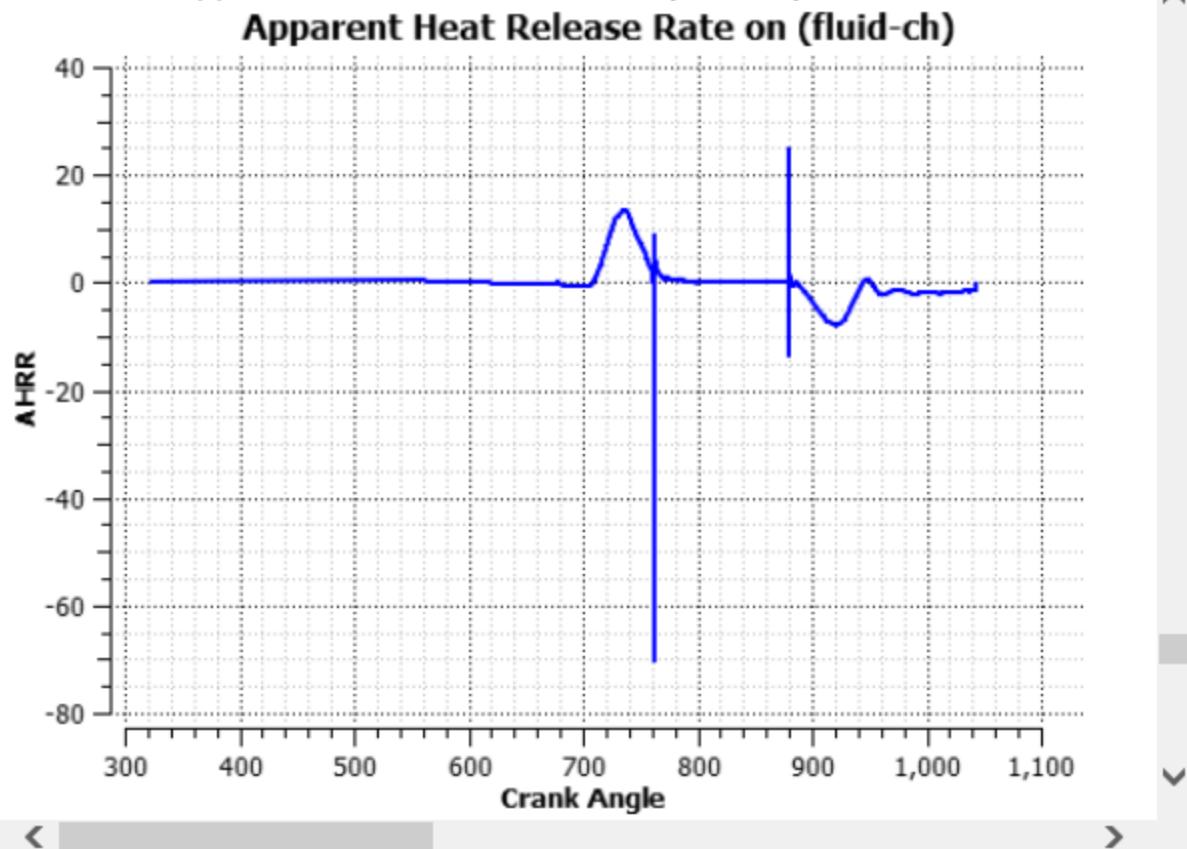
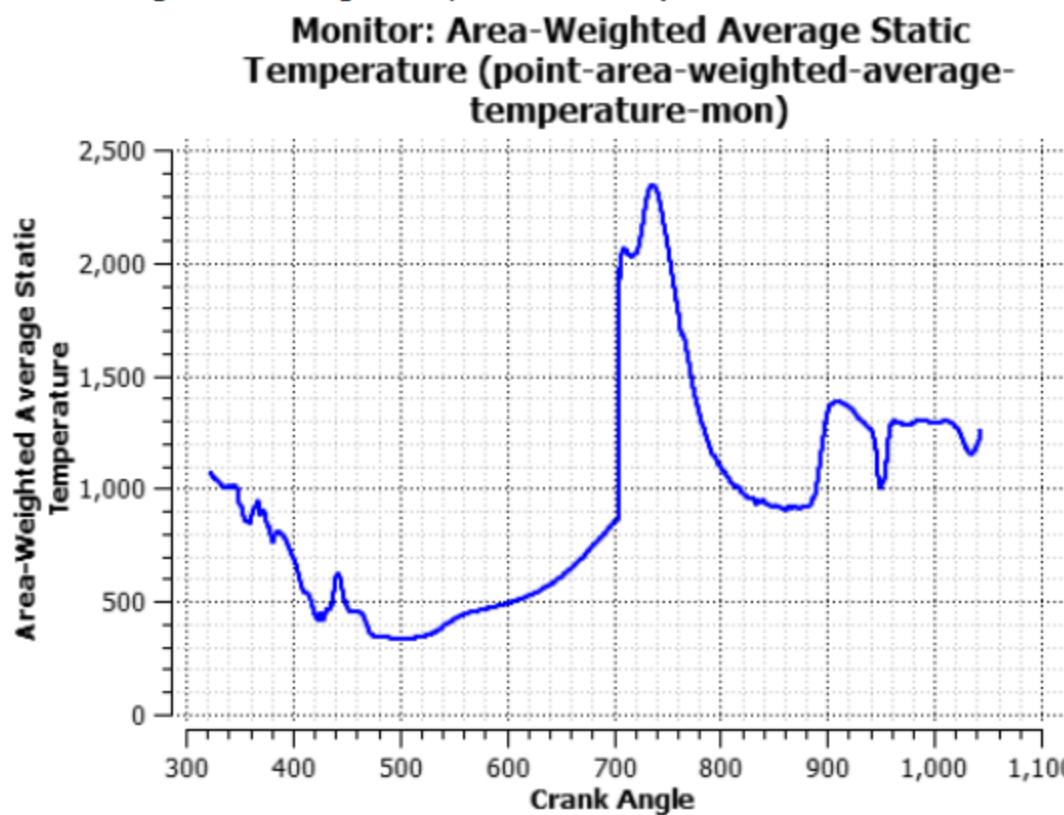


Chart 7. Monitor: Area-Weighted Average Static Temperature (point-area-weighted-average-temperature-mon)



- Monitors of **Mass-Average Turbulent Kinetic Energy** and **Number of Iterations per Time Step** are plotted against the crank angle.

Chart 8. Monitor: Area-Weighted Average Turbulent Kinetic Energy (k) (point-area-weighted-average-turbulent-kinetic-energy-mon)

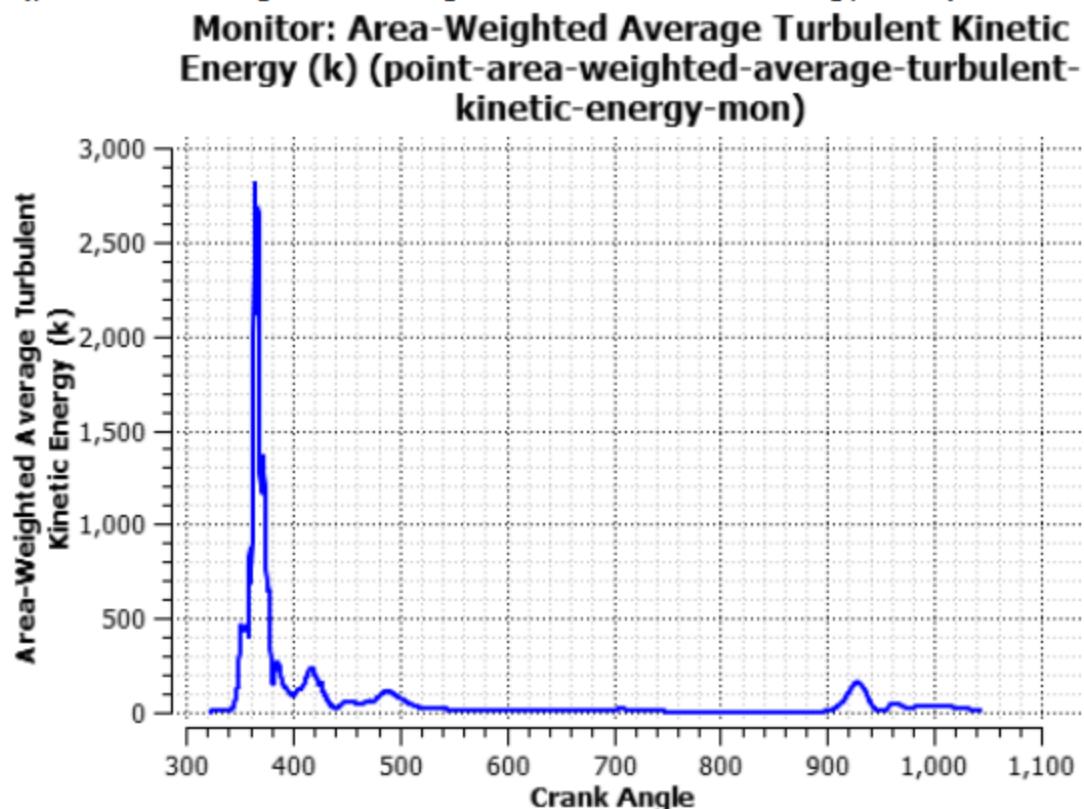
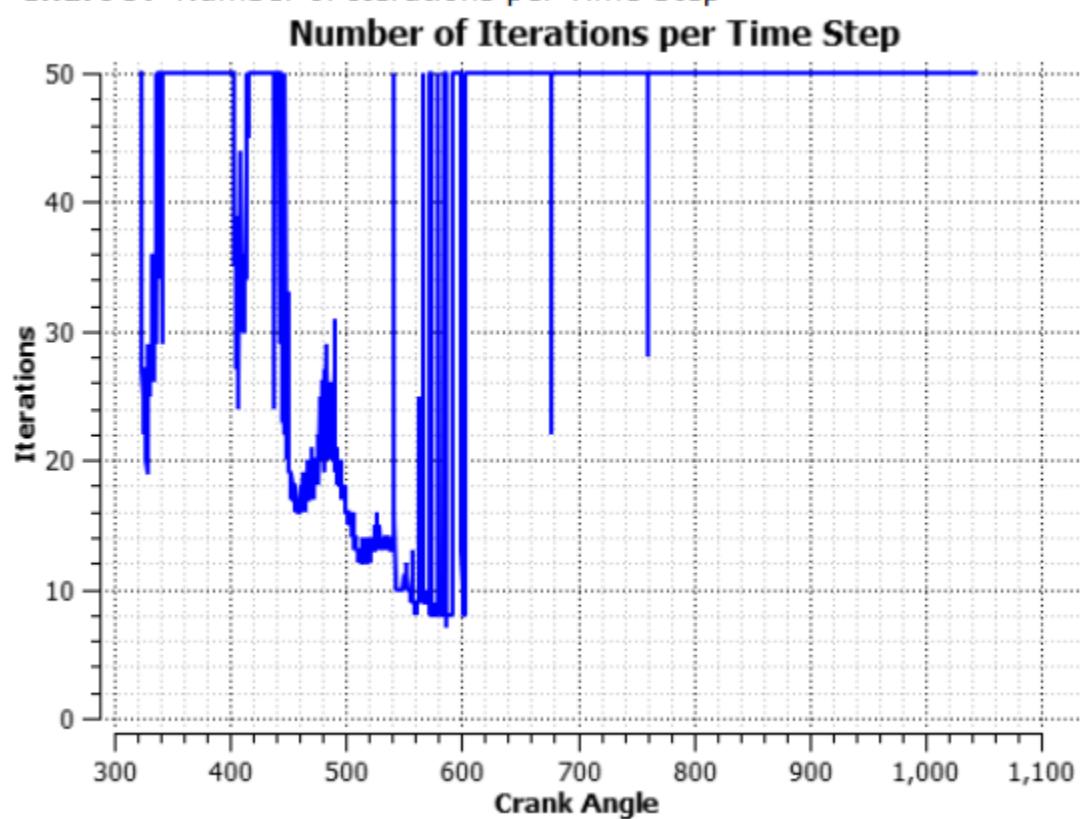


Chart 9. Number of Iterations per Time Step



- The report also includes the plots of **Mass-Average Absolute Pressure**, **Mass-Average Static Temperature**, **Mass-Average Turbulent Kinetic Energy**, and **Mass Static Pressure** on **fluid-ch** at different crank angles.

Chart 10. Monitor: Mass-Average Absolute Pressure (fluid-ch)

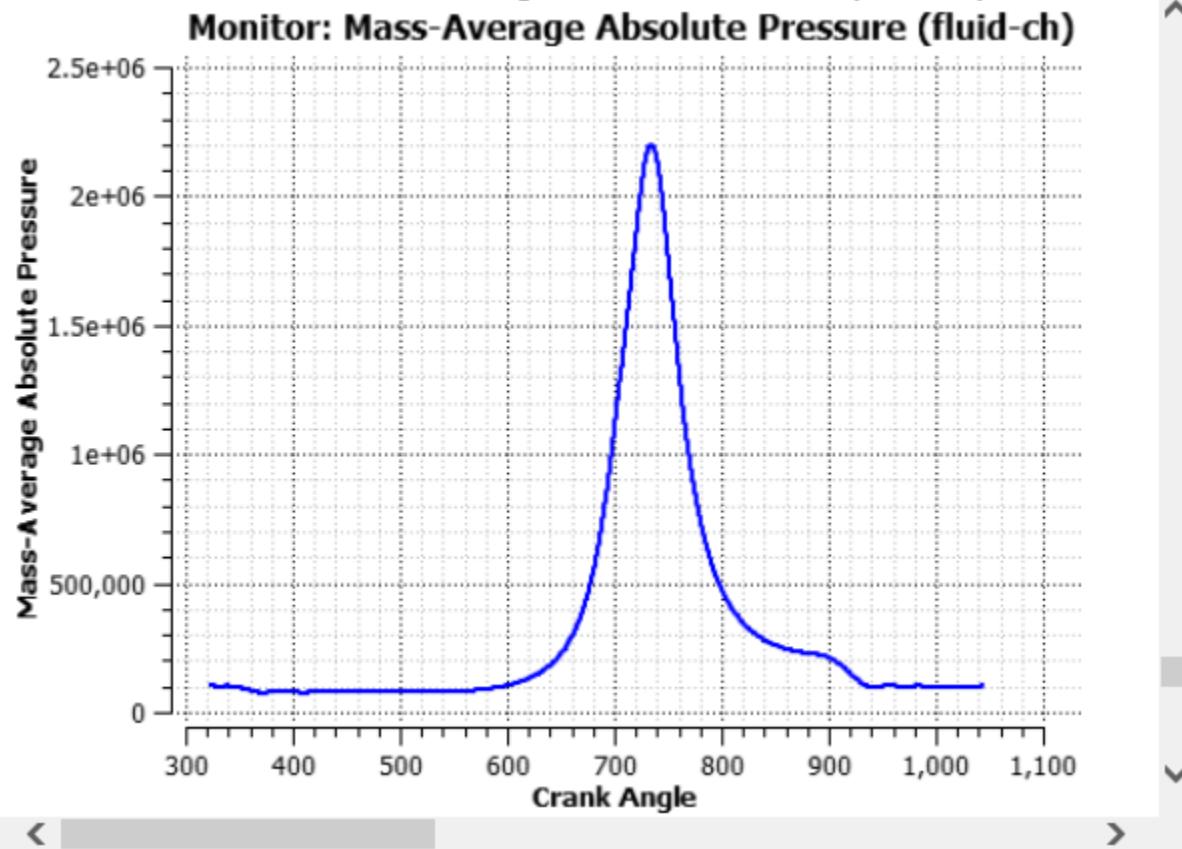


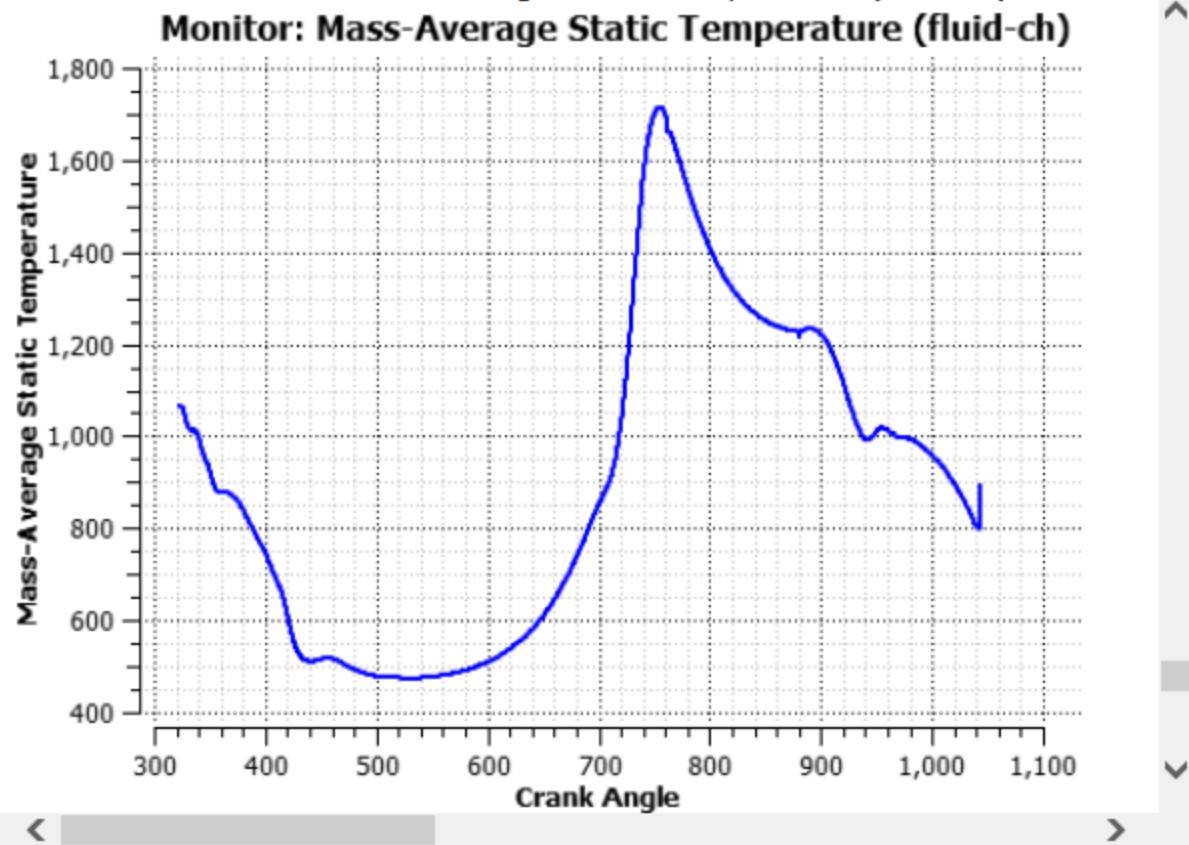
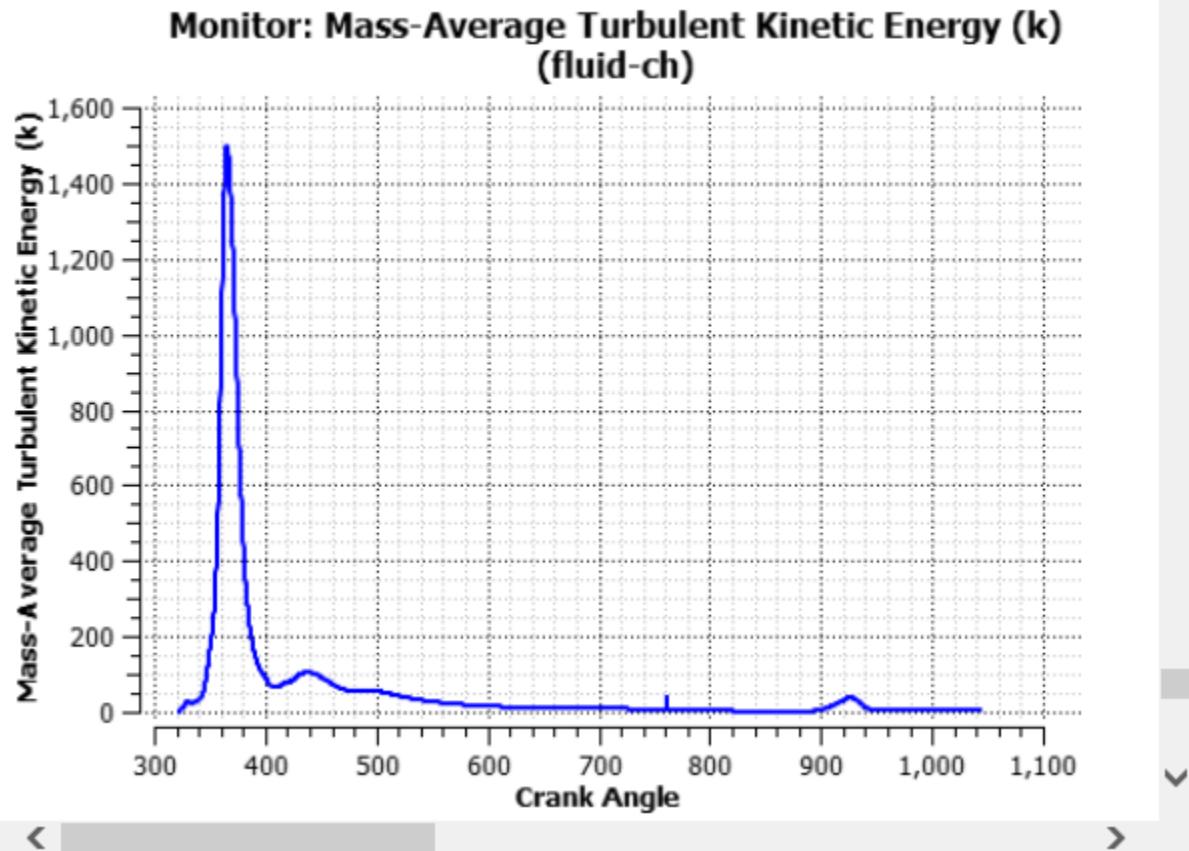
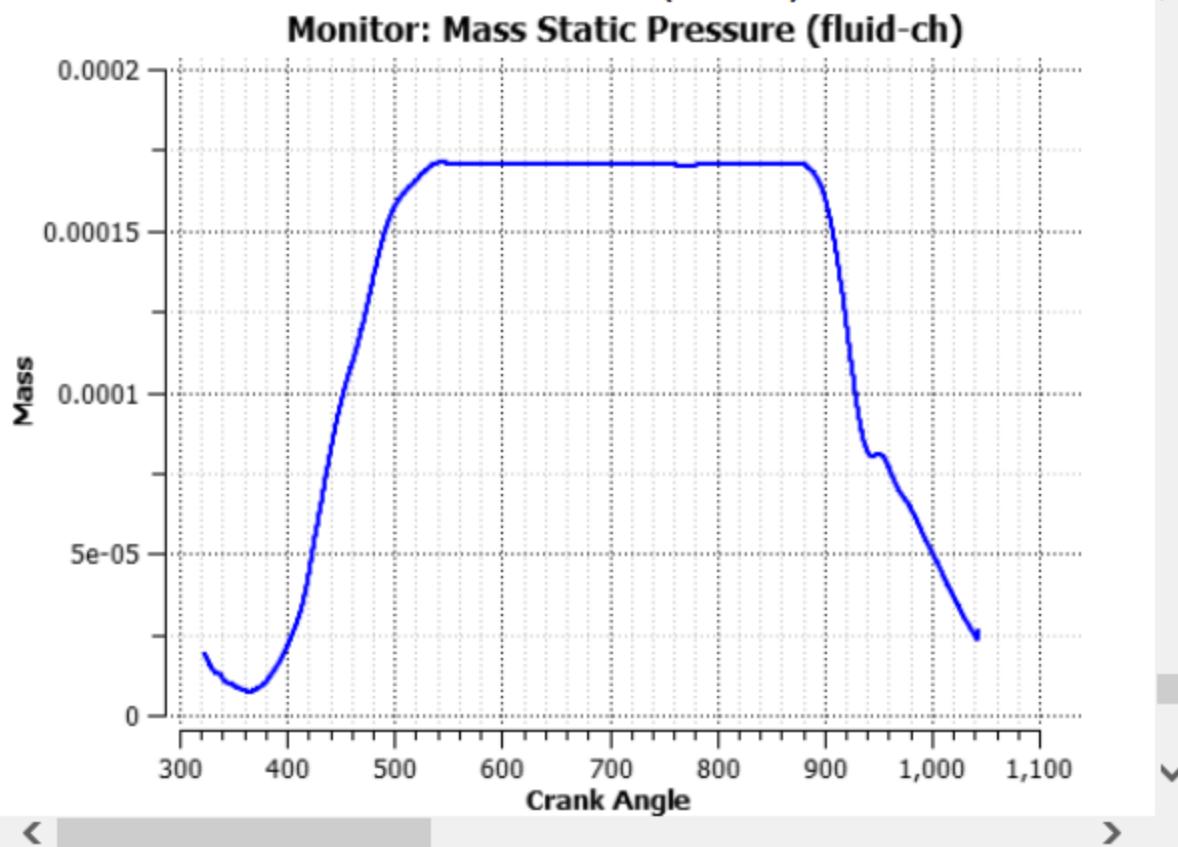
Chart 11. Monitor: Mass-Average Static Temperature (fluid-ch)**Chart 12.** Monitor: Mass-Average Turbulent Kinetic Energy (k) (fluid-ch)

Chart 13. Monitor: Mass Static Pressure (fluid-ch)

- Chart plotting the penetration length of the three injections is also included in the report.

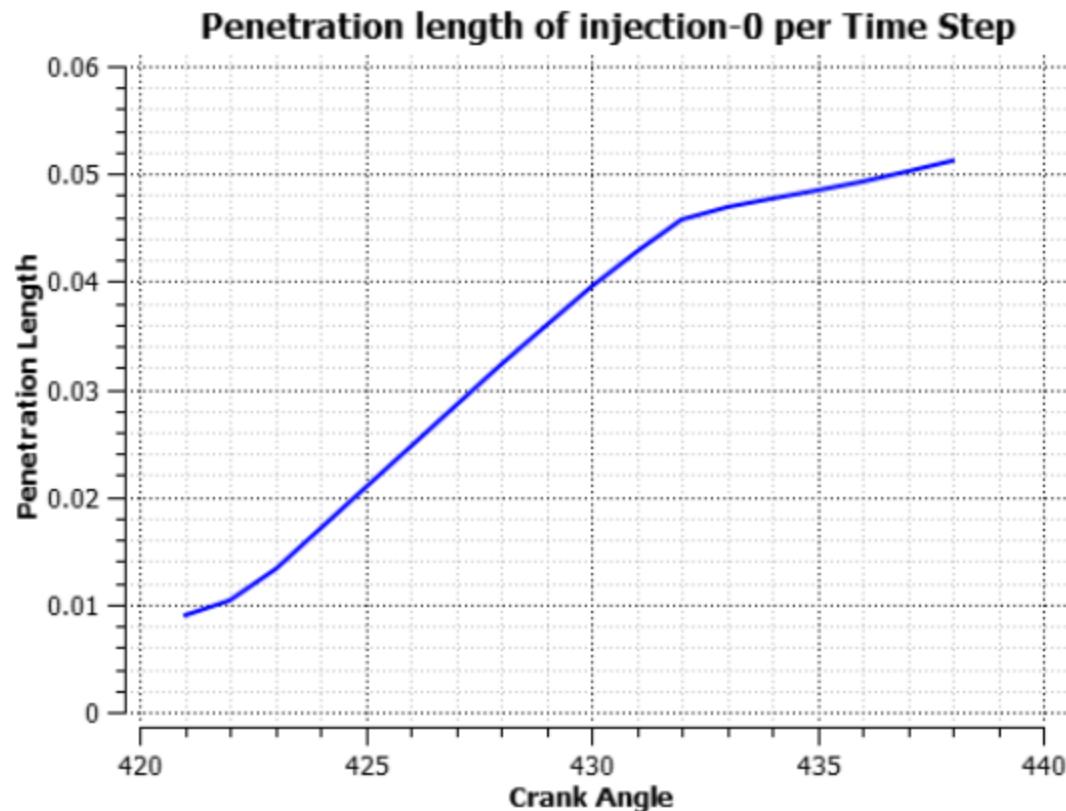
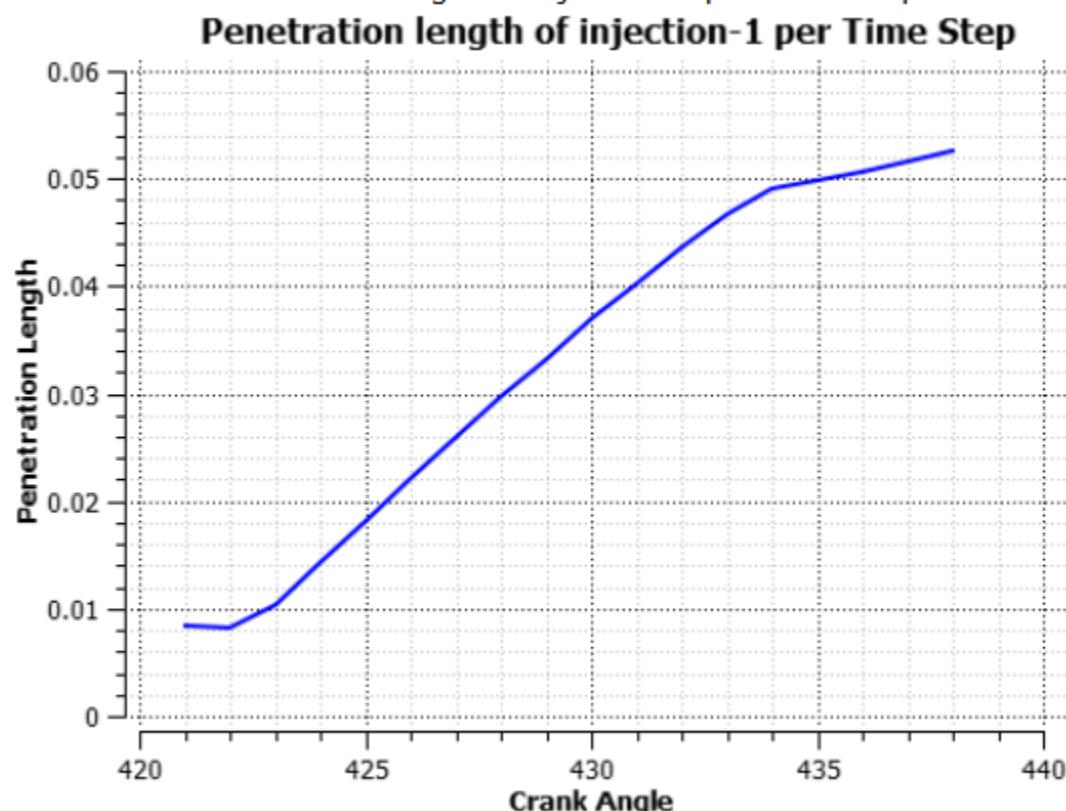
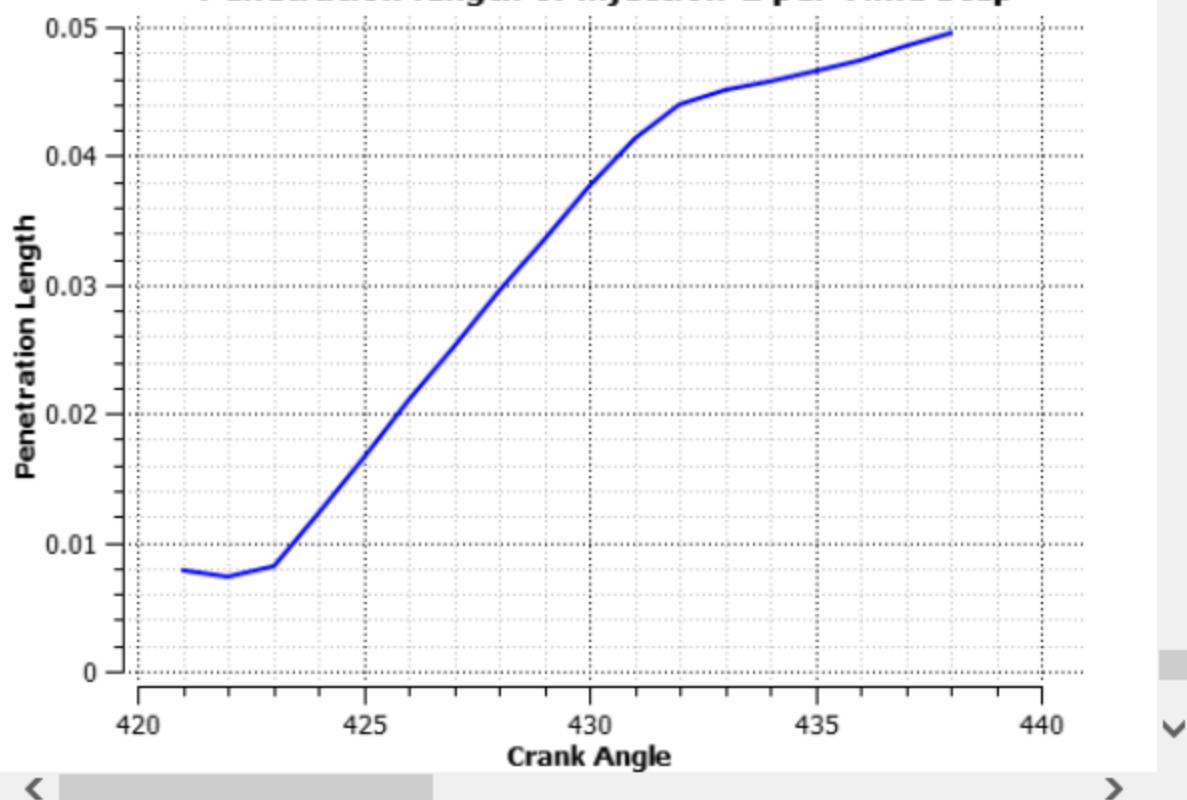
Chart 14. Penetration length of injection-0 per Time Step**Chart 15.** Penetration length of injection-1 per Time Step

Chart 16. Penetration length of injection-2 per Time Step
Penetration length of injection-2 per Time Step



- Monitor plots of **Volume Static Pressure**, **Total mass influid for all injections per Time Step**, **Total mass injected for all injections per Time Step** and **Total mass evaporarted for all injections per Time Step** against **Crank Angle** can be checked in the report.

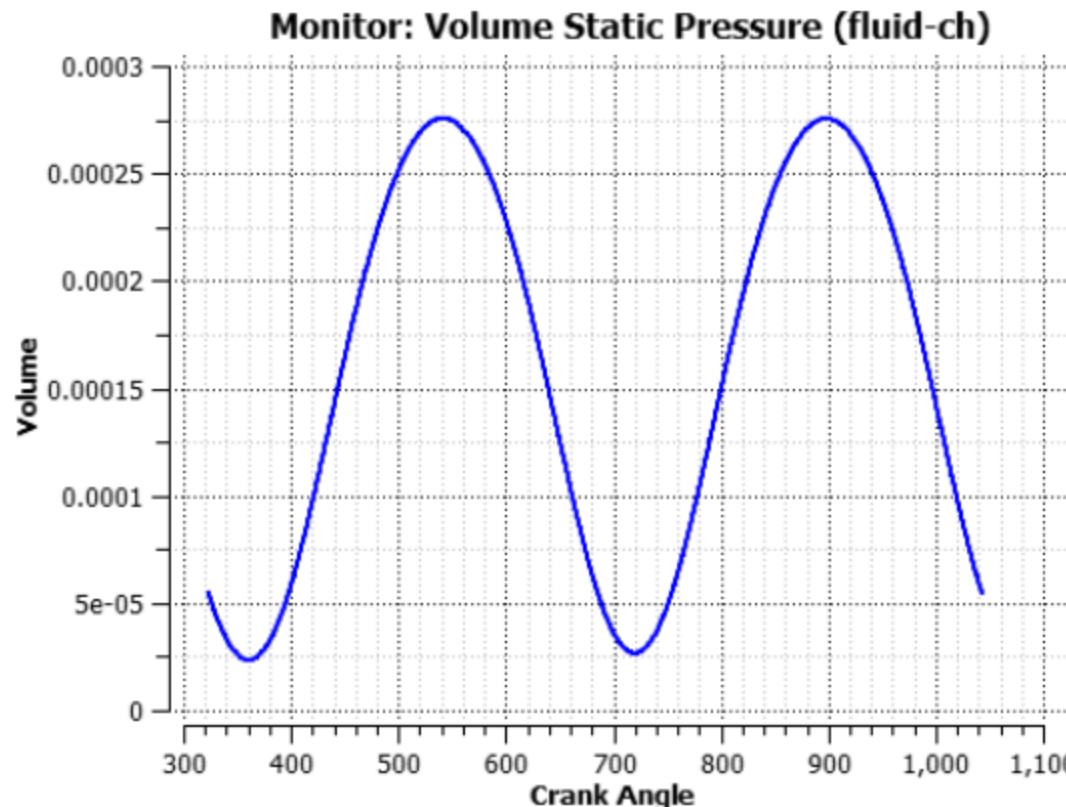
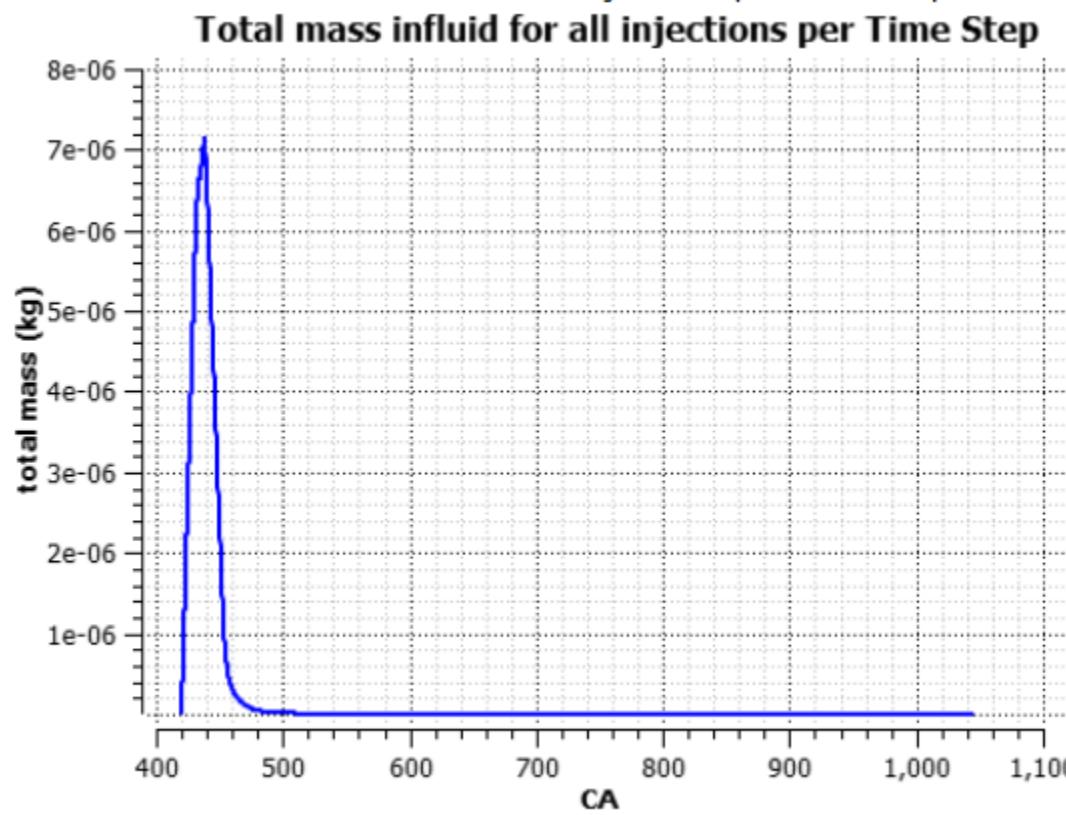
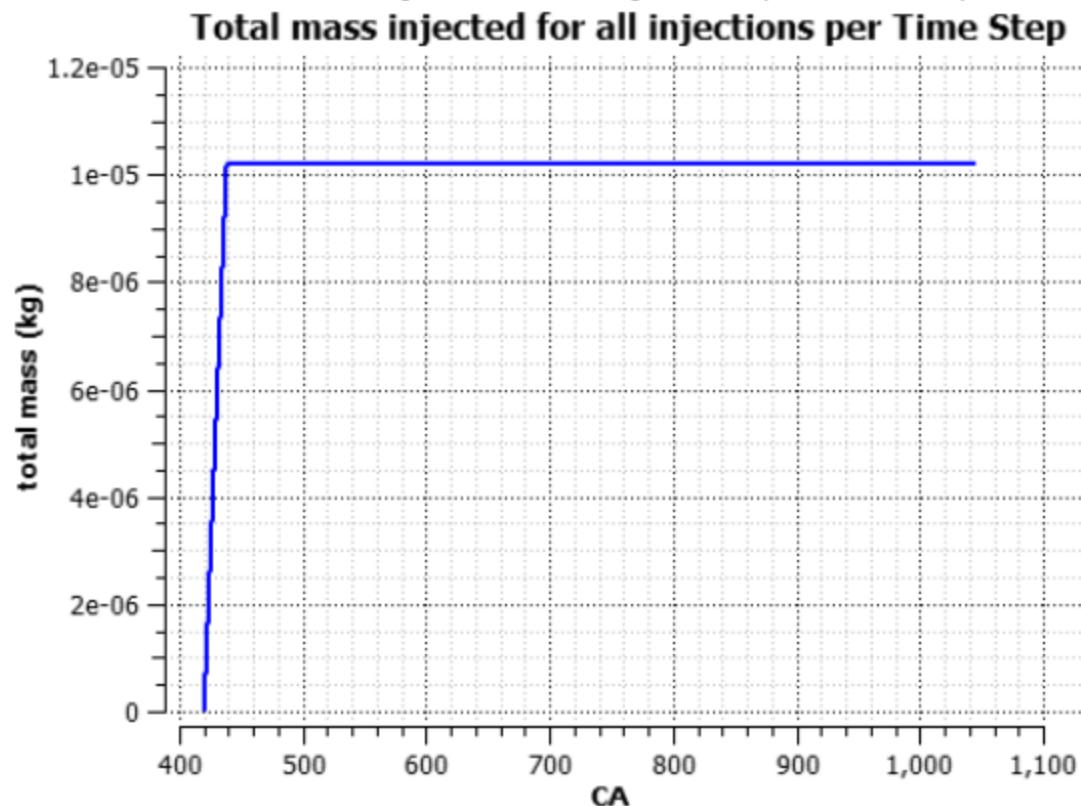
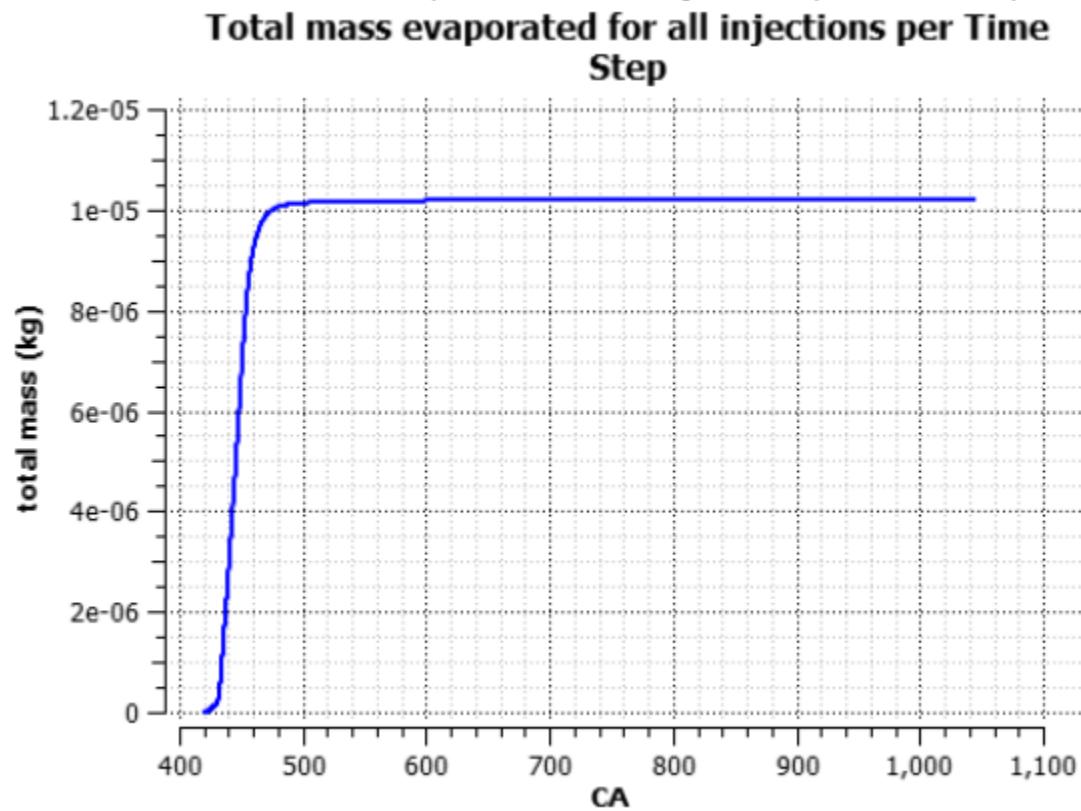
Chart 17. Monitor: Volume Static Pressure (fluid-ch)**Chart 18.** Total mass influid for all injections per Time Step

Chart 19. Total mass injected for all injections per Time Step**Chart 20.** Total mass evaporated for all injections per Time Step

This concludes the tutorial which demonstrated the setup and solution for the combustion simulation of a sector of an IC engine.

4.8. Summary

In this tutorial spray was injected in engine using Discrete Phase Model of Fluent. Fuel air mixture was then spark ignited by using spark model in Fluent. Partially premixed Combustion model was used for simulating turbulent combustion process. Pressure trace and heat release rate were examined.

You have also learnt on how to use key-grids and use different meshing strategies.

4.9. Further Improvements

This tutorial presents streamlined workflow between pre-processing, solver and post processing for ease of GDI simulations. You may obtain more accurate solution by using temperature and/or composition dependent material properties.

Index

