

Forte Tutorials



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 18.2
August 2017

ANSYS, Inc. and
ANSYS Europe,
Ltd. are UL
registered ISO
9001:2008
companies.

Copyright and Trademark Information

© 2017 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT 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 located 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. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

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. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2008 companies.

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, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. Introduction	1
2. Simulating a Diesel Engine Using a Sector Mesh	3
2.1. Data Provided	3
2.1.1. Files Used in This Tutorial	3
2.1.2. Project Comparison Utility	3
2.1.3. Time Estimate	4
2.2. Generating the Sector Mesh	4
2.2.1. Define the Engine Parameters	4
2.2.2. Select the Mesh Topology	5
2.2.3. Specify the Mesh-Size Parameters	6
2.2.4. Generate the Mesh and Review	6
2.2.5. Import the Mesh into the Open Forte Project	7
2.2.6. Set Up the Case	7
2.2.6.1. Run the Case Using Defaults	12
2.2.6.2. Problem Results	13
2.3. Reference	13
3. Simulating Dual Fuel Combustion	15
3.1. Data Provided	15
3.1.1. Files Used in This Tutorial	15
3.1.2. Project Comparison Utility	16
3.1.3. Time Estimate	16
3.2. Pre-Processing the Chemistry Set	16
3.3. Importing the Mesh	17
3.3.1. Set Up the Case	17
3.3.1.1. Save the Project	23
3.3.1.2. Run the Case Using Defaults	23
3.3.1.3. Visualizing the Results	24
3.4. Reference	24
4. Spray Bomb Modeling	27
4.1. Data Provided	27
4.1.1. Files Used in This Tutorial	27
4.1.2. Project Comparison Utility	27
4.1.3. Time Estimate	28
4.2. Modeling Solid-Cone Spray Injection	28
4.2.1. Spray Bomb	29
4.2.1.1. Problem Description	29
4.2.1.2. Mesh Setup	31
4.2.1.3. Models Setup	32
4.2.1.3.1. Boundary Conditions	34
4.2.1.3.2. Initial Conditions	34
4.2.1.3.3. Parameter Study on Pressure	35
4.2.1.3.4. Simulation Controls	35
4.2.1.3.5. Chemistry Solver	35
4.2.1.3.6. Transport Terms	35
4.2.1.4. Output Controls	36
4.2.1.5. Save Project	36
4.2.1.6. Run Settings	36
4.2.1.7. Run Simulation	36
4.2.1.8. Results	37
4.3. Reference	39

5. Port-Injected Spark Ignition Engine	41
5.1. Data Provided	41
5.1.1. Files Used in This Tutorial	41
5.1.2. Project Comparison Utility	41
5.1.3. Time Estimate	41
5.2. Port-Injected Spark Ignition Engine	42
5.2.1. Problem Description	42
5.2.1.1. Import the Geometry	42
5.2.1.2. Subvolume Creation	43
5.2.1.3. Automatic Mesh Generation Setup	43
5.2.1.4. Models Setup	45
5.2.1.5. Boundary Conditions:	46
5.2.1.6. Initialization	48
5.2.1.7. Simulation Controls	49
5.2.1.8. Output Controls	50
5.2.1.9. Preview Simulation	52
5.2.1.10. Run Simulation	52
5.2.1.11. Run Settings	52
5.2.2. Results	53
6. Tracking Soot Particles Evolution in a Diesel Engine	57
6.1. Data Provided	57
6.1.1. Files Used in This Tutorial	57
6.1.2. Project Comparison Utility	57
6.1.3. Time Estimate	58
6.1.4. Prerequisites for This Tutorial	58
6.2. Project Setup	58
6.2.1. Sector Mesh Details	58
6.2.2. Chemistry Set Details	60
6.2.3. Transport Property Settings	60
6.2.4. Spray Model Settings	60
6.2.5. Soot Model Settings	62
6.2.6. Boundary Conditions	62
6.2.7. Initial Conditions	62
6.2.8. Simulation Control	63
6.2.9. Output Control	63
6.3. Project Results	63
7. Solving a Gasoline Direct Injection Engine Simulation	67
7.1. Data Provided	67
7.1.1. Files Used in This Tutorial	67
7.1.2. Project Comparison Utility	67
7.1.3. Time Estimate	68
7.1.4. Prerequisites for This Tutorial	68
7.2. Direct Injection Spark Ignition Engine	68
7.2.1. Problem Description	68
7.2.1.1. Import the Geometry	69
7.2.1.2. Subvolume Creation	70
7.2.1.3. Automatic Mesh Generation Setup	70
7.2.1.4. Models Setup	72
7.2.1.5. Boundary Conditions	75
7.2.1.6. Initialization	78
7.2.1.7. Simulation Controls	79
7.2.1.8. Output Controls	80

7.2.1.9. Preview Simulation	81
7.2.1.10. Run Simulation	81
7.2.1.11. Run Settings	82
7.2.2. Results	82
8. Two-Stroke Engine Simulation	89
8.1. Data Provided	89
8.1.1. Files Used in This Tutorial	89
8.1.2. Project Comparison Utility	90
8.1.3. Time Estimate	90
8.1.4. Prerequisites for This Tutorial	90
8.2. Two-Stroke Marine Engine Project Setup	90
8.2.1. Problem Description	90
8.2.1.1. Two-Stroke Engine Details	92
8.2.1.2. Automatic Mesh Generation Setup	92
8.2.2. Chemistry Set Details	94
8.2.3. Transport Property Settings	94
8.2.4. Spray Model Settings	94
8.2.5. Boundary Conditions	96
8.2.6. Initial Conditions	99
8.2.7. Simulation Controls	101
8.2.8. Output Controls	102
8.2.9. Preview Simulation	104
8.2.10. Run Settings	105
8.2.11. Run Simulation	105
8.3. Project Results	106
9. Multi-Cylinder Four-Stroke Engine Simulation	111
9.1. Data Provided	111
9.1.1. Accessing Tutorial Files	111
9.1.2. Files Used in This Tutorial	111
9.2. Engine Configuration and Problem Description	112
9.3. Multi-Cylinder Spark-Ignition Engine Project Setup	113
9.3.1. Geometry Import	113
9.3.2. Defining Reference Frames and Time Frames	113
9.3.2.1. Reference Frames	113
9.3.2.2. Time Frames	114
9.3.3. Automatic Mesh Generation Setup	115
9.3.4. Model Setup	116
9.3.5. Boundary Conditions	116
9.3.6. Initial Conditions	118
9.3.7. Simulation Controls	119
9.4. Result Visualization	119

Chapter 1: Introduction

The ANSYS Forte CFD™ software is designed for 3-D, multi-phase engine simulations with combustion. This manual consists of tutorials that illustrate how to use ANSYS Forte CFD to address a variety of engine designs including compression ignition and spark ignition. The tutorials generally represent realistic situations that might be encountered by practicing scientists or engineers. They have been chosen to demonstrate the wide range of software capabilities, and the different ways ANSYS Forte can be used.

In this manual, we address two major categories of engine simulations: Diesel modeling with sector mesh generation is covered in [Simulating a Diesel Engine Using a Sector Mesh \(p. 3\)](#), modeling of Dual Fuels is covered in [Simulating Dual Fuel Combustion \(p. 15\)](#), Evaporating Spray modeling is covered in [Spray Bomb Modeling \(p. 27\)](#), and Spark Ignition modeling with moving valves, and automatic mesh generation in [Port-Injected Spark Ignition Engine \(p. 41\)](#). Soot particle tracking is demonstrated in [Tracking Soot Particles Evolution in a Diesel Engine \(p. 57\)](#). In [Solving a Gasoline Direct Injection Engine Simulation \(p. 67\)](#), gasoline direct injection in an SI engine is modeled. In [Two-Stroke Engine Simulation \(p. 89\)](#), a two-stroke marine engine with intake and exhaust ports is described. This lesson also includes a restart file, pressure profiles, plus an .stl file, to try starting from only the geometry.

Note

Before working with these tutorials, we recommend that you first review the Forte Quick Start Guide to become familiar with the operation of the ANSYS Forte interface and how to set up, run, and visualize results.

Chapter 2: Simulating a Diesel Engine Using a Sector Mesh

For diesel engines, the engine combustion is often simulated from intake valve closure (IVC) to exhaust valve opening (EVO), rather than modeling the full air intake or exhaust flow processes involving the intake and exhaust ports, respectively. This is usually a reasonable approximation since the gas in the cylinder at IVC is a relatively homogeneous mixture of air and exhaust gas (due to internal residual or from exhaust-gas recycling), prior to fuel injection. Furthermore, the nozzle hole pattern of the fuel injection usually gives rise to a periodic symmetry, based on the number of nozzle holes. In this tutorial we describe the use of a sector mesh that takes advantage of such symmetry for simulating a diesel engine operating in low-temperature-combustion (LTC) mode. The LTC mode involves an early injection timing, relative to more conventional diesel operation.

A sector can represent the full geometry, since we can take advantage of the periodicity of the cylinder and injector nozzle-hole pattern. For example, an eight-hole injector allows simulation using a 45° sector (360/8). By using the symmetry of the problem in this way, the mesh created is much smaller and the simulation therefore runs faster than it would with a 360° mesh. Such a simplification usually cannot be made for spark-ignited engine cases due to asymmetries introduced by spark plugs, piston shape, and possibly intake ports.

2.1. Data Provided

2.1.1. Files Used in This Tutorial

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

and access the **Forte** tutorial files to download. The files for this tutorial include:

- **sandia_bowl_profile.csv**: Data describing the piston bowl shape. This profile can be imported or manually entered in the Profile Editor for use by the Sector Mesh Generator to define the bowl shape.
- **InjectionProfile.csv**: Data describing the spray injection. The data can be imported from this file through the User Interface or manually entered once you reach the Spray node on the Workflow tree through use of the Profile Editor.
- **UserCrankAngleOutputs.csv**: Table of data that contains crank angles going from -22 to +20.
- **Sandia_Engine_LTC_EarlyInj.ftsim**: A project file of the completed tutorial, for verification or comparison of your progress in the tutorial set-up.

The sample files are provided as a download. You have the opportunity to select the location for the files when you download and uncompress the sample files.

2.1.2. Project Comparison Utility

The Forte installation includes the `cgns_util export` command, which you can use to compare the parameter settings in the project file generated at any point during your tutorial set-up against the

provided **.ftsim**, which has the parameter settings for the final, completed tutorial. This command is described in the Forte User Guide.

Briefly, you can double-check project settings by saving your project and then running the `cgns_util` to export your tutorial project, and then to export the provided final version of the tutorial. Save both versions and compare them with your favorite diff tool, such as DIFFzilla. If all the parameters are in agreement, you have set up the project successfully. If there are differences, you can go back into the tutorial set-up, re-read the tutorial instructions, and change the setting of interest.

2.1.3. Time Estimate

As a guideline for your own simulations, this tutorial is estimated to take 19 minutes on a dual Intel® Xeon® processor E5-2690 at 2.90 GHz (8 total cores).

2.2. Generating the Sector Mesh



The Forte Sector Mesh Generator provides six different sample mesh topologies for different piston bowl shapes. The shape of the piston bowl determines which mesh topology is most appropriate. The overview of the process is:

1. Specify the bowl profile.
2. Enter engine parameters, such as bore, stroke, etc.
3. Select a topology for meshing.
4. Specify the control point locations for the mesh topology if necessary (for Topology 3, these are assumed and the profile must have exactly 3 points to define them).
5. Specify the number of cells between control points for the mesh.
6. Generate the mesh and import into Forte.

Note

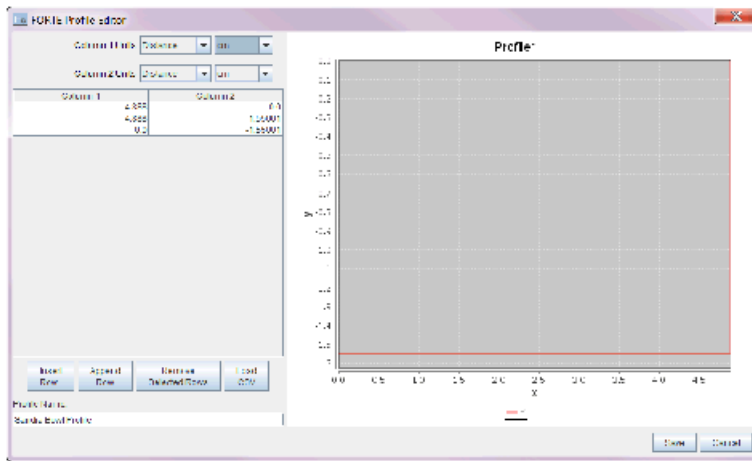
All Editor panel options that are not explicitly mentioned in this tutorial should be left at their default values. Changed values on any Editor panel do not take effect until you press the **Apply** button.

2.2.1. Define the Engine Parameters

1. To begin creating a sector mesh, go to the Workflow tree and click Geometry. This opens the Geometry icon bar. Click the **Launch Sector Mesh Generator**  icon to open the Sector Mesh Generator Utility (SMG).
2. Start by clicking the Engine Parameters node in the SMG Project tree to open an Editor panel.
3. In that panel, the first step is to import the simple bowl profile that was provided with the sample. To do this, go to the **Bowl Profile** pull-down menu and select **Create New...** and click the **Pencil**  icon. This opens the Profile Editor window.

4. In the Profile Editor, click the **Load CSV** button (you may need to expand the window vertically to see the button below the table) and then browse to, select, and open the **sandia_bowl_profile.csv** file. In the dialog, select **Comma** as the **Column Delimiter** and uncheck (turn OFF) the **Read Column Titles** box and click **OK**.
5. In the **.csv** file, the first column is the X-coordinate and the second column is the Z-coordinate. For this example the “bowl” is completely flat and the wall of the bowl is vertical. For this very simple geometry, only three coordinate points are needed to define the bowl. The profile should look like a flat line in the Profile Editor display, as shown in [Figure 1.12: Diesel sector sample case](#). At the bottom of the Profile Editor panel in the text entry field under **Profile Name**, enter a name for the profile, such as “Sandia Bowl Profile”. See [Figure 1.11: Sector Mesh Generator Utility with new Profile and Editor panel](#). Click **Save**.

Figure 2.1: Sector Mesh Generator Utility with New Profile and Editor panel

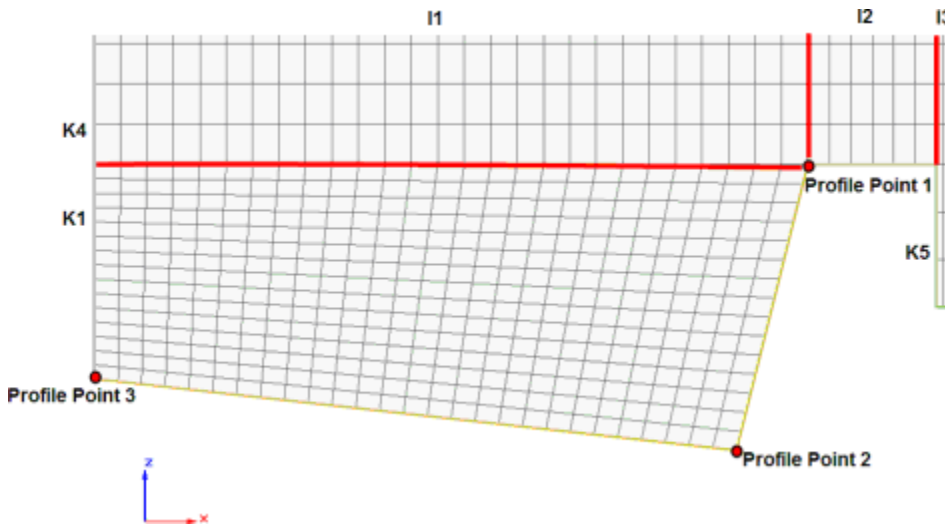


6. Next, in the Sector Mesh Generator, specify the engine parameters, which are:
 - **Sector Angle = 45 degrees**
 - **Bore = 13.97 cm**
 - **Stroke = 15.24 cm**
 - **Squish = 0.56 cm**
 - **Crevice Width = 0.167 cm**
 - Select and expand the **Include Crevice Block** option and set **Crevice Height = 3.72621 cm**. Click **Apply**. A diagram preview of the engine geometry appears.

2.2.2. Select the Mesh Topology

Once you have a profile, the first step in the mesh workflow is to specify the desired mesh topology that is appropriate for the imported piston-bowl shape. For this simple flat-piston-bowl geometry, Topology 3 will be used since it is a square-shaped bowl, as shown in [Figure 1.9: Template for Topology #3](#). For this topology there are only 3 coordinates needed to define the profile and exactly 3 are required.

Figure 2.2: Template for Topology #3




1. Now go to the Mesh Parameters node in the SMG tree. Select **Topology 3** in the **Topology** pull-down menu. If you would like to see the topology template selected, click the **Show Topology** button. It should look like that shown in [Figure 1.9: Template for Topology #3](#).


2.2.3. Specify the Mesh-Size Parameters

Next, enter the cell counts between the different control points, as indicated in the Topology #3 diagram, and in the schematic drawing in the 3-D display window. The parameters we will enter are as follows:

- **Circumferential Cell Count = 21**
- **Radial Cell Count i1 = 16**
- **Radial Cell Count i2 = 11**
- **Radial Cell Count i3 = 3**
- **Axial Cell Count k1 = 11**
- **Axial Cell Count k4 = 21**
- **Axial Cell Count k5 = 11**
- **Smoothing Passes = 20** (default value; works well for most cases)

2.2.4. Generate the Mesh and Review

1. Once all the settings are specified, click **Apply**. Review the outline of the profile information in the 3-D View panel to verify that the settings make sense.
2. Then use the **Generate a Mesh**  icon at the top of the Mesh Parameters Editor panel in the Sector Mesh Generator. ANSYS Forte will automatically generate the sector mesh. When it is finished, the resulting mesh

will appear for your review in the 3-D View window within the Sector Mesh Generator window, as shown in [Figure 1.12: Diesel sector sample case](#). (You may need to click the **Refit**  icon.)

Note

You can modify settings here to refine or coarsen the mesh, for example, and re-generate, or you can open (import) the newly created mesh into the sample project into Forte. Alternatively, you can use the **Save to File** button to export to a mesh file that can later be imported into ANSYS Forte.

2.2.5. Import the Mesh into the Open Forte Project

1. For this tutorial, click the **Import to Forte Project**  icon in the icon bar at the top of the SMG panel. Now you can go to the Simulate window and see the mesh that you just generated displayed in the main


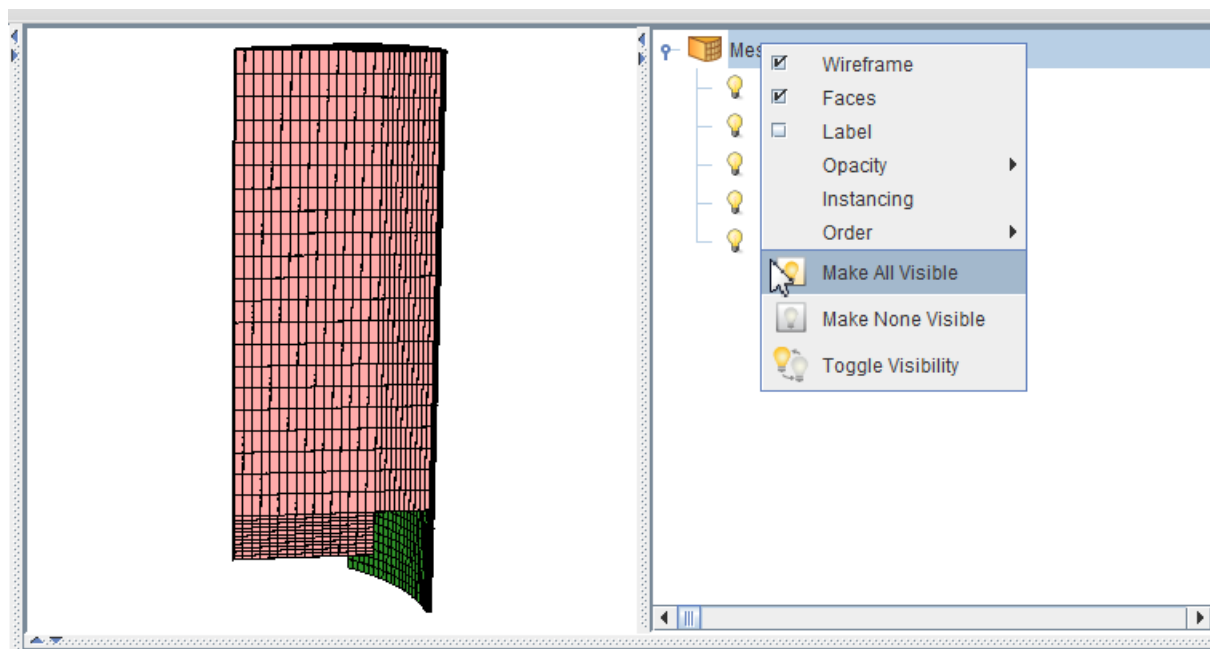
3-D View panel. To center and resize the mesh display, click the **Refit**  icon on the toolbar.

Figure 2.3: Diesel sector sample mesh as generated in the Sector Mesh Generator.



2. If the Sector Mesh Generator window is still open, you can close it.


2.2.6. Set Up the Case

Now that the mesh has been defined, you can set up the models and solver options using the guided tasks in the Forte Simulate Workflow tree. In many cases, default parameters are assumed and employed, such that no input is required. The required inputs and changes to the setup panels for this case are described here.

In the Workflow tree, expand Models. In the following steps you will turn ON (check-mark) several models and configure parameters for them in their Editor panels.

Note


You can change the display properties of items in the 3-D View by selecting the items in the Visibility tree and turning ON/OFF their visibility or right-clicking and selecting options from the context menu, such as display of the mesh or level of opacity.


1. Models > Chemistry: In the Workflow tree, under Models, select Chemistry. On the Chemistry icon bar, click the **Import Chemistry**  icon. This opens a file browser where you can navigate to and select a chemistry set file. For this project, however, we use a pre-installed chemistry set that comes with ANSYS Forte. This is a simplified, reduced n-heptane mechanism that can be used to represent the diesel fuel under conventional diesel-engine combustion conditions. To load this file, browse to the **System Data** directory and locate the pre-installed **Diesel_1comp_35sp.cks** file. The **System Data** directory can be accessed from this file browser by clicking the **data** radio button in the upper right-hand sub-panel of the browser window. The chemistry file is a standard CHEMKIN chemistry-set file. More information about the chemistry set and the files referenced within it can be found in the ANSYS Forte User Guide .

Note


You can respond **Yes** or **No** to the “View chemistry set information?” prompt. A **Yes** response displays the chemistry set file in a viewer.

2. Models > Transport: For this node, keep all the default settings.
3. Models > Spray Model: Since this is a direct-injection case, turn ON (check) Spray Model to display its icon bar (action bar) in the Editor panel.
 - a. For the basic Spray Properties, keep **Radius of Influence Model** for the **Droplet Collision Model** and set **0.2 cm** as the **Radius of Influence**. The **Use Vaporization Model** option should be **ON** (checked, its default value).
 - b. Create Injector: Click Spray Model in the Workflow tree. The icon bar provides two spray-injector options:

Hollow Cone or **Solid Cone**. For the diesel injector, click the **Solid Cone**  icon. In the dialog that opens, name the Spray Model as “**Injector 1**”. This opens another icon bar and Editor panel for the new solid-cone spray model. In the Editor panel, configure the model parameters for the solid-cone spray model.

- **Composition:** Select **Create New...** in the Composition drop-down menu under Settings and click the **Pencil**  icon to open the Fuel Mixture Editor. In the Fuel Mixture panel, click the **Add Species** button and select **nc7h16** (i.e., n-heptane) as the **Species**, **n-Tetradecane** as the **Physical Properties** and **1.0** as the **Mass Fraction**. (Note that you must press **ENTER** after entering the values in the table.)
- At the bottom of the panel, type a name such as “n-heptane”. Click **Save** and **Close** the window.
- Under the **Injection Type**, select Pulsed Injection and change the **Parcel Specification** to **Number Of Parcels** and set **Injected Parcel Count = 4,000**.

- Set **Inflow Droplet Temperature = 368.0 K**.
- Set **Spray Initialization** to **Constant Discharge Coefficient and Angle**, and **Discharge Coefficient = 0.7**, and **Mean Cone Angle = 15.0** degrees.
- Keep default values for **Droplet Size Distribution** and, under the **Solid Cone Breakup Model Settings**, the **KH Model Constants**, **RT Model Constants**, and **Use Gas-Jet Model**.
- Click **Apply**.

- c. Create a Nozzle: Click the **New Nozzle**  icon on the Spray icon bar and name the nozzle "**Nozzle 1**". Nozzle 1 then appears in the Workflow tree, and the Editor panel and icon bar transform to allow specification of the Nozzle geometry and orientation. In the Editor panel, set the Reference Frame parameters to specify the nozzle location and direction. Keep the default **Global Origin** and use the following settings:
- **Location > Coord. System = Cylindrical**, with **R = 0.15 cm**, **$\theta = 22.5$ degrees**, and **A = 19.368 cm**.
 - **Spray Direction > Coord. System = Spherical**, **$\theta = 104.0$ degrees**, and **$\varphi = 22.5$ degrees**.
 - Click **Apply**. You can see the nozzle appear at the top of the geometry. (You may wish to make the Geometry less opaque by right-clicking the Geometry item and selecting a lower value for **Opacity**, or change the color of the nozzle itself, both in the Visibility tree on the right side, to make the nozzle easier to see in the interior.)


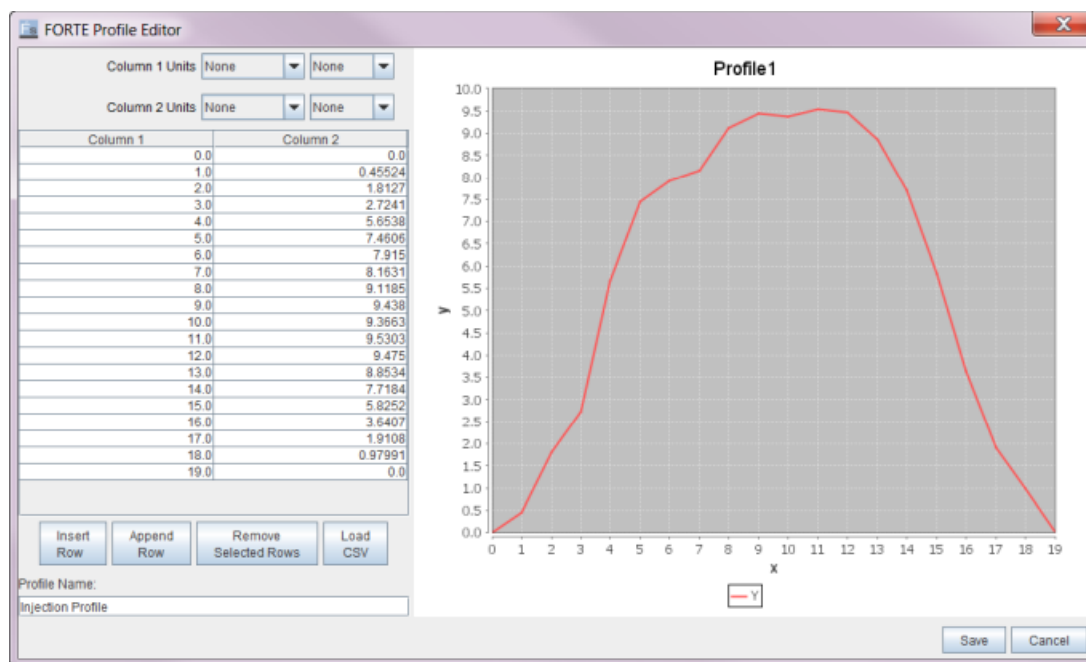

- d. Create an Injection: In the Workflow tree, click Injector 1 again and click the **New Injection**  icon on the **Injector 1** icon bar and name the injection **Injection 1**. The new Injection item appears in the Workflow tree, and the Editor panel and the icon bar transform to allow specification of the injection properties. In the Editor panel, keep the **Pulsed Injection Type** (default), select **Crank Angle** as the **Timing** option and then specify the **Start** of injection and **Duration** of injection as **-22.5** and **7.75** degrees ATDC, respectively. Set the **Total Injected Mass = 0.0535 g**. Click **Apply**.
- e. Injection Profile: Click the **Create new...** option in the profile selection menu next to **Velocity Profile**, then click the pencil icon to open a new window with the Profile Editor. The new Injection Profile item appears in the Workflow tree; also the Editor panel and the icon bar transform to the new Injection Profile. In the Profile Editor window, make sure all the **Units** are set to **None** for both columns (the dimensionless data is automatically scaled within ANSYS Forte to match the mass and duration of injection) and click the **Load CSV** button at the bottom of the panel (you may have to expand the panel size to see the button). Navigate to the **InjectionProfile.csv** file (see [Data Provided \(p. 3\)](#)). Select **Comma** as the **Column delimiter** and turn ON **Read Column Titles** and load the profile file. Alternatively, you can type in the Injection Profile data in the table on the Editor panel or copy and paste from a spreadsheet or 3rd-party editor. Go to **Profile Name** at the bottom-left of the panel and name the new profile **Injection Profile**. Once the data is entered, click **Save** in the Profile Editor and then click **Apply** in the Forte Editor panel.

Figure 2.4: Injection Profile Parameter Settings


4. Models > Soot Model: Turn ON the Soot Model. This creates the Settings item. In the Editor panel, accept all the defaults.
5. Boundary Conditions: Under Boundary Conditions in the Workflow tree, specify the Boundary conditions in the Editor panels associated with each of the four boundary conditions created by importing the mesh. By default the **Wall Model** for all of the wall boundaries will be set to **Law of the Wall**. Leave this default setting as well as the default check box that turns ON heat transfer to the wall.
 - Boundary Conditions > Piston: Set Piston Temperature = **500.0 K**. Turn ON **Wall Motion** with **Motion Type** set to **Slider Crank**. The other parameters should be:
 - **Stroke = 15.24**
 - **Connecting Rod Length = 30.48**
 - Bore = **13.97** (pre-determined by the Sector Mesh Generator)
 - Accept the defaults of **Piston is Offset = unchecked (OFF)** and **Vertices to Transform = All**.
 - For **Reference Frame**, accept the default **Global Origin** and **Direction** parameters.
 - Click **Apply**.
 - Boundary Conditions > Periodicity: Set the **Sector Angle = 45 degrees**, and multi-select both **Periodic A** and **Periodic B** boundaries on which to apply this boundary condition. Click **Apply**.
 - Boundary Conditions > Head: Set **Head Temperature = 470.0 K**. Click **Apply**.
 - Boundary Conditions > Liner: Set **Liner Temperature = 420.0 K**. Click **Apply**.
6. Initial Conditions > Region 1 Initialization: Specify the parameters for the initial conditions as follows:

- **Composition:** Select **Constant** and then **Create New** and click the **Pencil**  icon and to launch the Composition Editor. Set these parameters: set the **Composition** = **Mole Fraction** (not the default **Mass Fraction**). Then click the **Add Species** button and select both **o2** and **n2** to add. When both **o2** and **n2** are in the Species column in the **Composition** table, enter **0.126** for the **o2 Fraction** and **0.874** for **n2**. Name this "Composition 1" in the text field at the bottom of the Gas Mixture window. Click **Save** and close the Composition Editor.
- **Temperature = 362.0 K**
- **Pressure = 2.215 bar** (Note: This is not the default unit.)
- **Turbulence:** Select **Constant**, and then in the pull-down menu for the initial Turbulence parameters, select **Turbulent Kinetic Energy and Length Scale** as the way in which we will specify the initial turbulence. For this option we provide an explicit value for the initial turbulent kinetic energy, but use a length-scale approximation to determine the turbulence dissipation energy. Use these values.
- **Turbulent Kinetic Energy = 10,000cm2/sec2**
- **Turbulent Length Scale = 1.0 cm**
- **Velocity:** Select **Engine Swirl** in the **Velocity** pull-down and then specify the swirl profile parameters:
 - **Initial Swirl Ratio = 0.5**
 - **Initial Swirl Profile Factor = 3.11**
 - Initialize Velocity Components Normal to Piston = ON

Note

If you want to estimate the turbulent kinetic energy (TKE) as a fraction of the piston speed, let the fraction be F and stroke is ms^{-1} , the value you would enter then would be

$$TKE = F * 0.5 * \left(2 * \frac{\text{stroke} * \frac{\text{engine_speed_RPM}}{60}}{100} \right)^2 \text{ [m}^2/\text{s}^2\text{]}$$

- Click **Apply** in the Region 1 Initialization Editor panel.
7. Simulation Controls: Under Simulation Limits, for the **Simulation End Points**, specify:
- **Crank Angle Based**
 - **Initial Crank Angle = -165.0**
 - **Final Simulation Crank Angle = 125.0 degrees.** (These two angles correspond to Intake Valve Open and Exhaust Valve Closing, respectively.)
 - **RPM = 1,200.0**
 - **Cycle Type is 4-Stroke.**
 - Click **Apply**.

8. Simulation Controls > Time Step: For the **Advanced Time Step Control Options**, specify:
 - **Rate of Strain Factor = 1.2**
 - **Convection Factor = 0.5**
 - Click **Apply**.
9. Simulation Controls > Chemistry Solver:
 - Turn ON (check) **Use Dynamic Cell Clustering** and accept its defaults.
 - At the bottom of the panel, set **Activate Chemistry** to **Conditionally**, specifying **When Temperature is Reached**, setting **Threshold Temperature = 600.0 K**. Click **Apply**.
10. Output Controls > Spatially Resolved: For the **Spatially Resolved Output Control**, specify:
 - **Crank Angle Output Control**
 - **Output every = 5.0 degrees**.
 - Also check the box next to **User Defined Crank Angle Outputs** and then import the **UserCrankAngleOutputs.csv** table of data using the Profile Editor that contains crank angles going from -22 to +20, with **increment 1**, units of **Angle** and **Degree**, and file named **UserCrankAngleOutputs**. This assures that we get more resolved outputs around the spray, without requiring the same resolution throughout the simulation.
 - For **Spatially Resolved Species**, move these species to the **Selection** list: **nc7h16, o2, n2, co2, h2o, co, no**, and **no2**. Click **Apply**.
11. Output Controls > Spatially Averaged:
 - For the **Spatially Averaged Output Control**, select the **Crank Angle** option and specify **Output Every = 1.0 degree**.
 - For **Spatially Averaged Species**, select all and move all species to the **Selection** list. Click **Apply**.
12. Output Controls > Restart Data: In the Workflow tree, check the box for **Restart Data** and then in the Restart panel, check the box that says, **Write Restart File at Last Simulation Step**. Uncheck (turn OFF) any other boxes on the panel. Click **Apply**.

2.2.6.1. Run the Case Using Defaults


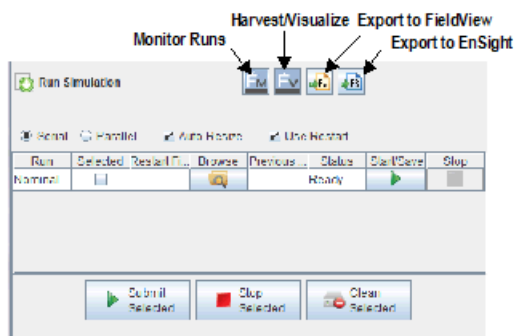
1. In the Workflow tree, click the Run Simulation node to open the run control interface.
2. Click the green arrow under **Start** to start the case running. The **Status** changes to "Running."
3. To interrupt the run, click the red button under **Stop**. Then the **Status** becomes "Stopped".
4. To start over or restart, click **Start** again.
5. The progress of the simulation can be monitored during a run. To monitor the averaged value of simulation variables, click the **Monitor Runs**  icon on the Run Simulation icon bar.

Figure 2.5: Run Simulation - Icon bar.

A new window will appear, with a list of variables that can be plotted on the left panel in the window (as described in the ANSYS Forte User Guide). To select one or more variables to plot, check the boxes in front of them in the Monitor Data panel.

6. When a run is finished, the **Status** becomes **Complete** in the Run panel of the main Simulate Interface window. In the Editor Panel, use the **Harvest/Visualize** button to save the harvested results as a **.ftres** file. Note that multiple runs can be selected for this purpose.
7. ANSYS Forte Visualizer will be loaded automatically following the “Visualize” step unless the **Harvest only** option was checked. You can follow the ANSYS Forte User Guide Visualization chapter to post-process the results.
8. You can save the project at this point using the **File > Save** command. Rename the project to avoid overwriting the original **.ftsim** file.

2.2.6.2. Problem Results

1. When a run is finished, the **Status** becomes **Complete** in the Run panel of the main Simulation interface window. In the Monitor Panel, use the **Harvest/Visualize** button to save the harvested results as a **.ftres** file. Note that multiple runs can be selected for this purpose.
2. Forte Visualizer will be loaded automatically following the “Harvest Results” step if the **Launch Visualizer after Harvest** option is checked. You can follow the ANSYS Forte User Guide Visualization chapter to post-process the results.

2.3. Reference

This case models the engine experiment from the following paper:

Singh, S., Reitz, R. D., and Musculus, M. P. B. Comparison of the characteristic time (CTC), representative interactive flamelet (RIF), and direct integration with detailed chemistry models against optical diagnostic data for multi-mode combustion in a heavy-duty DI diesel engine. SAE paper 2006-01-0055, 2006.

Chapter 3: Simulating Dual Fuel Combustion

Many engine companies are now investigating the technical challenges posed by dual-fuel combustion in engines. Dual-fuel combustion has the advantage of providing diesel-like efficiency with gasoline as the primary fuel, providing potential increases in efficiency of 50% while reducing emissions. Typical dual-fuel combustion designs involve the use of a small pilot injection of a liquid fuel that is relatively easy to ignite. The pilot is injected into a lean mixture of air and a more volatile fuel that is less inclined to autoignite, but which is port-injected to provide good mixing with the air prior to ignition. Diesel is typically used as the pilot and it can be injected in very small amounts, sometimes referred to as micro-pilots. Gasoline is the typical port-injected fuel for transportation applications and natural gas for power applications. Other liquid and gaseous fuels are of interest for power applications such as landfill gas, process gas and refinery gas. ANSYS Forte allows the use of accurate fuel models to capture fuel effects of both the ignition and flame propagation processes in a computationally efficient manner.

ANSYS Forte uses the well established G-equation model for tracking flame propagation. In spark-ignited engines, flame propagation is initiated by a spark event, using a kernel ignition model. For dual-fuel cases, however, the injection and auto-ignition of the liquid pilot fuel serves to initiate the flame propagation. In ANSYS Forte, the simulation can consider both auto-ignition and flame-propagation modes of combustion progress simultaneously. The ANSYS Forte auto-ignition mode of initiating flame propagation models takes advantage of a good fuel model's ability to accurately predict ignition under auto-ignition conditions of temperature and pressure. In addition, the flame-propagation model uses a locally calculated turbulent flame-speed that derives from fuel-specific flame-speed calculations using detailed kinetics models. In this way, both auto-ignition and flame-propagation benefit from accurate fuel models for low-temperature and high-temperature kinetics. The details of the autoignition-induced flame propagation model are included in the Forte Theory Manual.

The ANSYS Forte Simulation workflow allows the use of either a body-fitted mesh or an automatically generated mesh. In this tutorial, we will focus on the use of a mesh previously generated using the Sector Mesh Generator.

A sector can represent the full geometry because we can take advantage of the periodicity of injector nozzle-hole pattern and the symmetry of the cylinder. For example, a six-hole injector allows simulation using a 60° sector (360/6). By using the symmetry of the problem in this way, the mesh created is much smaller and the simulation therefore runs faster than it would with a complete 360° mesh.

This tutorial describes the use of 82% premixed gasoline with an 18% liquid diesel pilot for ignition. The tutorial is based upon work previously published that also looks at 78% gasoline and 85% gasoline cases (Puduppakkam et al. 2011, as described in [Reference \(p. 24\)](#)).

3.1. Data Provided

3.1.1. Files Used in This Tutorial

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

and select Forte tutorial files you wish to download. The files for this tutorial include:

- **itape.fmesh:** This is the sector mesh file. It can only be used with body-fitted mesh calculations.
- **Profile1.csv:** Data describing the spray injections. These data can be entered using cut and paste or using the **Import** option within the Profile Editor tool, which can be opened in the context of the spray-injection panel on the Simulation User Interface.
- **reduced-mech__RCCI-conditions_e.inp:** This is the chemistry input file specifically assembled for this tutorial, using Reaction Design's Model Fuels Consortium II (MFC II) database to assemble a multi-component master mechanism. This mechanism was then reduced specifically for use in the dual fuels engine case. Details of the mechanism can be found in Puduppakkam et al. (2011). After downloading this file, it must be pre-processed (see [Pre-Processing the Chemistry Set \(p. 16\)](#)).
- **Dual_Fuels_82percentGasoline_BFMesh_Tutorial.ftsimsim:** A project file of the completed tutorial, for verification or comparison of your progress in the tutorial set-up.

The sample files are provided as a download. You have the opportunity to select the location for the files when you download and uncompress the sample files.

3.1.2. Project Comparison Utility

The Forte installation includes the `cgns_util export` command, which you can use to compare the parameter settings in the project file generated at any point during your tutorial set-up against the provided `.ftsimsim`, which has the parameter settings for the final, completed tutorial. This command is described in the Forte User Guide.

Briefly, you can double-check project settings by saving your project and then running the `cgns_util` to export your tutorial project, and then to export the provided final version of the tutorial. Save both versions and compare them with your favorite diff tool, such as DIFFzilla. If all the parameters are in agreement, you have set up the project successfully. If there are differences, you can go back into the tutorial set-up, re-read the tutorial instructions, and change the setting of interest.

3.1.3. Time Estimate

As a guideline for your own simulations, this tutorial is estimated to take approximately 3 hours on a dual Intel® Xeon® processor E5-2690 at 2.90 GHz (8 total cores).

3.2. Pre-Processing the Chemistry Set

Note

All Editor panel options that are not explicitly mentioned in this tutorial should be left at their default values. Changed values on any Editor panel do not take effect until you press the Apply button.

The mechanism has been encrypted for use only with ANSYS Forte and ANSYS Chemkin. The encrypted mechanism `.inp` file must be pre-processed.

On the **Utility** menu, select **Pre-Processing**. Choose a **Working Directory** and select **New Chemistry Set**. Enter a **Short Name** for the chemistry set, such as **DualFuels**. For the **Gas-Phase Kinetics File** entry, browse to the directory where you saved the `.inp` file and select **reduced-mech__RCCI-conditions_e.inp**, and click the **Open/Create** button in the file browser. In the Pre-processing dialog, click the **Save As** button. The `.cks` file with the assigned **Short Name** should be in the **File Name** entry; click

Save. In the Pre-processing dialog, click **Run Pre-Processor** to create the **.cks** file that will be imported into ANSYS Forte. You will use this **.cks** file later in the tutorial. Close the confirmation dialog and the Pre-processing window.

3.3. Importing the Mesh

You can import many different mesh and geometry file types in ANSYS Forte. In this tutorial, we will import a sector-mesh file previously created using the Sector Mesh Generator, which is compatible with KIVA-3V formatted files (itape file).


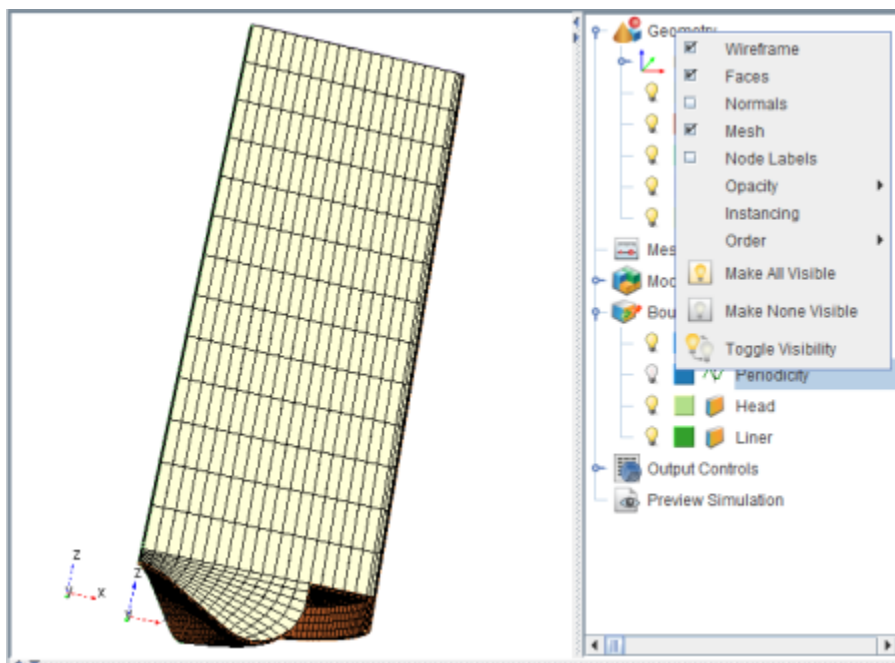
Select the Geometry item on the Workflow tree, then click the Import **Geometry**  icon and select **Body Fitted Mesh from KIVA-3V Format**. In the dialog that displays, select **Use the imported body-fitted mesh directly in the simulation**. Then select the downloaded **itape.fms** file described in [Files Used in This Tutorial \(p. 15\)](#). ANSYS Forte will automatically create the defined surfaces in the Boundary Conditions section of the Workflow tree based on boundary flags contained in this file. If you wish, you can rename the geometry elements by right-clicking on an element's name. As shown in [Figure 3.1: KIVA-3V geometry visualized in Simulate's 3-D View after import.](#) (p. 17), you can see the sector geometry in the 3-D View window in the center. The geometry elements and Boundary Conditions are listed in the Visualization tree on the right, where you can perform many operations such as turning the visibility of the mesh ON and OFF, increasing opacity, changing color, etc., by right-clicking on the element.




Figure 3.1: KIVA-3V geometry visualized in Simulate's 3-D View after import.



3.3.1. Set Up the Case

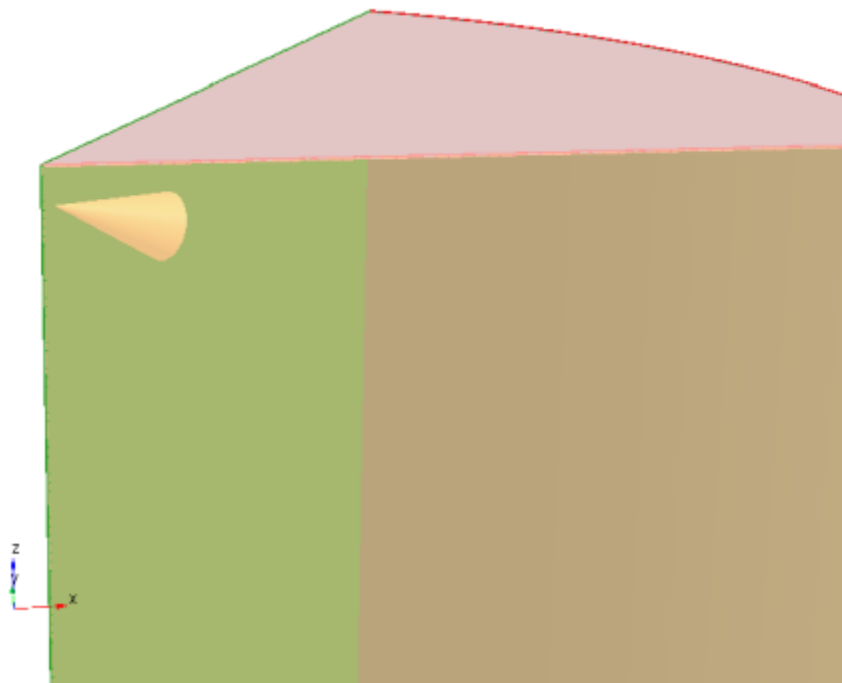
Now that the mesh has been imported, you can set up the models and solver options using the guided tasks in the Simulation Workflow tree. In many cases, default parameters are assumed and employed, such that no input is required. The required inputs and changes to the setup panels for this case are described here.


1. In the Workflow tree, expand Models. In the following steps you will turn ON (check-mark) some optional models and configure parameters for each model in its Editor panel.

2. Models > Chemistry: On the Chemistry icon bar, click the **Import Chemistry**  icon. This opens a file browser where you can navigate to and select a chemistry set file. For this project, however, we use a pre-installed chemistry set that comes with ANSYS Forte. This is a reduced mechanism specifically for these dual-fuel conditions. To load this file, browse to the directory where you saved pre-processed **.cks** file and locate the **.cks** file you created from the **reduced-mech_RCCI-conditions_e.inp** file (see [Pre-Processing the Chemistry Set \(p. 16\)](#)). The chemistry file (**.cks**) is a standard CHEMKIN chemistry-set file. More information about the chemistry set and the files referenced within it can be found in the Forte User Guide .
3. Models > Transport: For this node, keep all the default settings.
4. Models > Spray Model: Since this is a direct-injection case, turn ON (check) **Spray Model** to display its icon bar (action bar) in the Editor panel. In the panel, keep the defaults. Leave the **Use Vaporization Model** (default) check ON.
 - a. Add an Injector: The icon bar provides two spray-injector options: **Hollow Cone** or **Solid Cone**. For the diesel injector, click the **Solid Cone**  icon. In the dialog that opens, name the new solid-cone injector as "**Solid Injector**". This opens another icon bar and Editor panel for the new solid-cone injector. In the Editor panel, configure the model parameters for the solid-cone injector.
 - **Composition:** Select **Create New...** in the Composition drop-down menu under Settings and click the pencil icon to open the Fuel Mixture Editor. In the Fuel Mixture panel, click the **Add Species** button and select **nc7h16** (i.e., *n*-heptane) as the **Species**, **n-Tetradecane** as the **Physical Properties** and enter **1.0** as the **Mass Fraction**. At the bottom of the panel, you can keep the default name of "Fuel Mixture 1". Click **Save** and **Close** the window.
 - Change the **Parcel Specification** to **Number of Parcels**, and the **Injected Parcel Count** = **3,000**, its default value.
 - Set **Inflow Droplet Temperature** = **322.0 K**.
 - Set **Spray Initialization** to **Constant Discharge Coefficient and Angle**, and **Discharge Coefficient** = **0.7** and **Mean Cone Angle** = **10.0** degrees.
 - The **Droplet Size Distribution** is set to **Uniform Size** by default.
 - Under the **Solid Cone Breakup Model Settings**, leave the **KH Model Constants**, and **RT Model Constants**, at their default values, except for the **Size constant of RT Breakup**; set this value to **0.1**.
 - Use the **Gas Jet Model**, with the default parameters.
 - Click **Apply**.
 - b. Create a Nozzle: While Solid Injector is selected in the Workflow tree, click the **New Nozzle**  icon on the icon bar and name the nozzle **Nozzle**. "Nozzle" then appears in the Workflow tree, and the Editor panel and icon bar transform to allow specification of the Nozzle geometry and orientation. Set the parameters in the Editor panel to:
 - Reference Frame: Select **Global Origin** from the drop-down menu. For **Coord. System** = **Cylindrical**, with **R** = **0.097** cm, **Θ** = **30.0** degrees, and **A** = **18.44** cm.

- For **Spray Direction**, select **Spherical** from the drop-down menu and then $\Theta = 107.5$ degrees and $\phi = 30.0$ degrees.
- For **Nozzle Size**, select **Area** and set **Nozzle Area** = 0.000491 cm^2 .
- Click **Apply**. You can see the nozzle appear at the top of the geometry. (You may want to make the Geometry non-opaque or change the color of the nozzle itself in the Visibility tree to make the nozzle easier to see in the interior.)

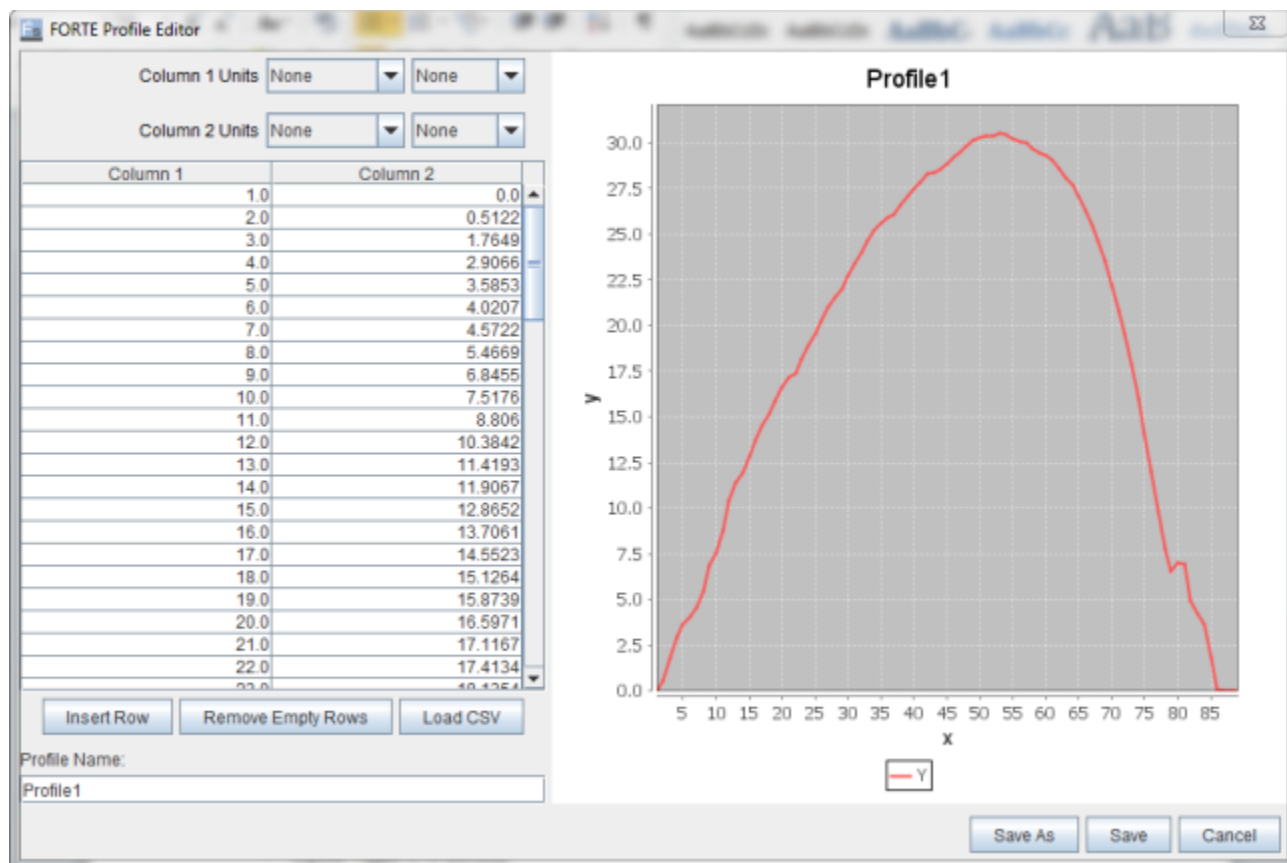
Figure 3.2: Spray cone shown after nozzle specification.



- c. Add an Injection: In the Workflow tree, click Solid Injector again and click the **New Injection**  icon on the **Solid Injector** icon bar and name the injection "**Injection 1**". The new Injection item appears in the Workflow tree, and the Editor panel and the icon bar transform to allow specification of the injection properties. In the Editor panel, select **Crank Angle** as the **Timing** option and then specify the **Start** of injection and **Duration** of injection as **-67.0** and **5.46** degrees, respectively. Set the **Total Injected Mass** = **0.015042 g**.
- d. Injection Profile: Click the **Create new...** option in the profile selection menu next to **Velocity Profile**, then click the pencil icon to open the Profile Editor. The new Injection Profile item appears in the Workflow tree, and the Editor panel and the icon bar transform to the new Injection Profile. In the Editor panel, make sure all the **Units** are set to **None** for both columns (the dimensionless data is automatically scaled within ANSYS Forte to match the mass and duration of injection) and click the **Load CSV** button at the bottom of the panel (you may have to expand the panel size to see the button). Then navigate to the **Profile1.csv** file (see [Files Used in This Tutorial \(p. 15\)](#)). Select **Comma** as the **Column delimiter** and turn ON **Read Column Titles** and load in the profile file. Alternatively, you can type in the Injection Profile data in the table on the Editor panel or copy and paste from a spreadsheet or 3rd-party editor.

Once the data are entered, go to **Profile Name** at the bottom-left of the panel and name the new profile "**Profile1**" and click **Save** in the Profile Editor and then click **Apply** in the Editor panel.

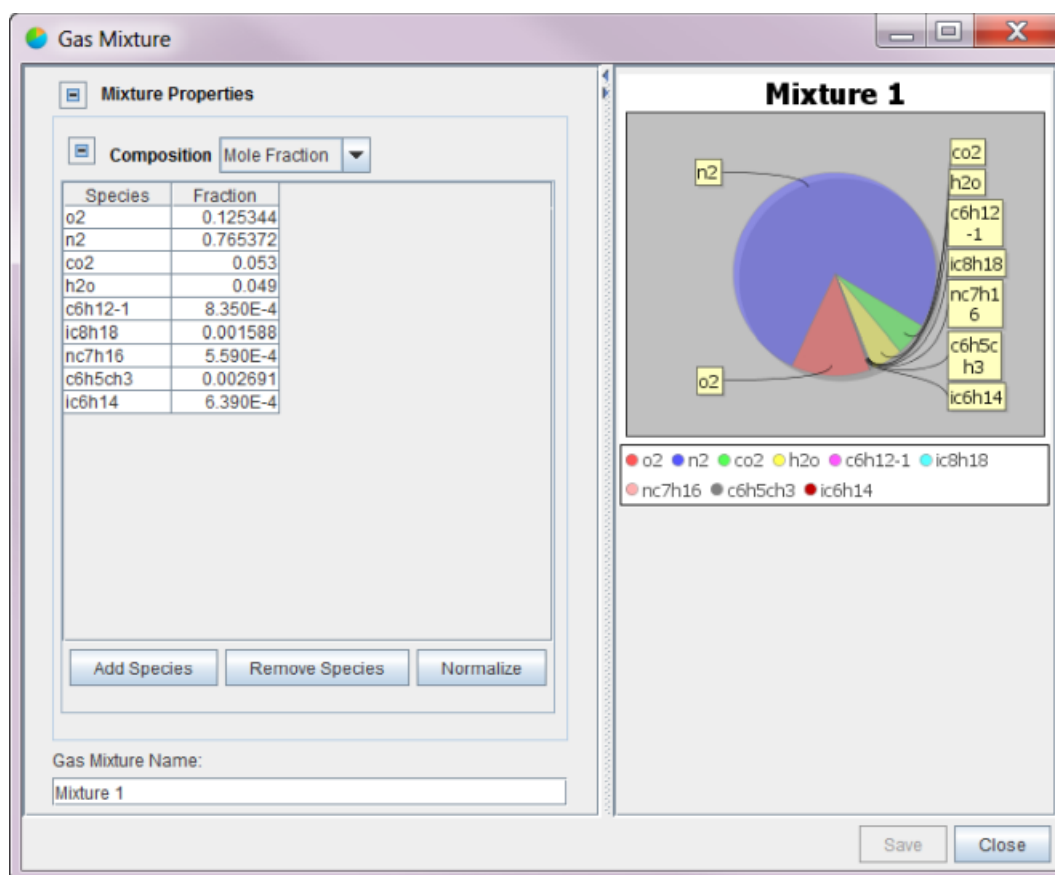
Figure 3.3: Injection profile parameter settings



5. Add a second injection. For the second injection, the start of injection is **-32.7 degrees ATDC** and the duration of injection is **2.73 CA** degrees. Use the same profile as above for this injection by selecting it from the pull-down menu next to Injection Profile. Enter the **Total Injected Mass** as **0.007521 g**.
6. Boundary Conditions: Under Boundary Conditions in the Workflow tree, specify the Boundary conditions in the Editor panels associated with each of the four boundary conditions created by importing the mesh. By default the **Wall Model** for all of the wall boundaries will be set to **Law of the Wall**. Leave this default setting as well as the default check box that turns ON heat transfer to the wall.
 - Boundary Conditions > Piston: Set Piston Temperature = **500.0 K**. Turn ON **Wall Motion** with **Motion Type** set to **Slider Crank**. The other parameters should be:
 - Stroke = 16.51
 - Connecting Rod Length = 26.16
 - Bore = 13.716 (this is pre-determined from the mesh)
 - Accept the defaults of **Piston is Offset** = **unchecked** (OFF).
 - For **Reference Frame**, accept the default **Global Origin** and **Direction** parameters.
 - For **Direction**, accept the defaults of **X** and **Y** = **0.0** and **Z** = **1.0** for the direction of piston motion.

- Click **Apply**.
 - Boundary Conditions > Periodicity: Keep the default **Sector Angle = 60 degrees**. The periodic boundaries, **Periodic A** and **Periodic B**, for this boundary condition are automatically selected, based on the boundary flags contained in the imported mesh.
 - Boundary Conditions > Head: Set **Head Temperature = 500.0 K**. Click **Apply**.
 - Boundary Conditions > Liner: Set **Liner Temperature = 430.0 K**. Click **Apply**.
7. Initial Conditions > Region 1 Initialization (Main): Specify the parameters for the initial conditions as follows:
- In the **Composition** drop-down menu, select **Create New...** and click the pencil icon to open the Gas Mixture Editor. In the Gas Mixture panel, change the **Composition** setting to **Mole Fraction** (not the default **Mass Fraction**). Then click the **Add Species** button and select the species in the list indicated in [Figure 3.4: Gas Mixture Editor \(p. 21\)](#). Once all the species are entered, provide the mole fraction values as indicated in [Figure 3.4: Gas Mixture Editor \(p. 21\)](#). Enter a name for the **Mixture** if you desire or leave the default as **Mixture 1** in the bottom left text box; click **Save**, then **Close** the window.

Figure 3.4: Gas Mixture Editor



- **Temperature = 391.0 K**
- **Pressure = 3.34 bar** (note this is not the default unit, so you need to use the pull-down next to the text box to select the correct units option)

- In the pull-down menu for the initial Turbulence parameters, select **Turbulence Intensity and Length Scale** as the way in which we will specify the initial turbulence. For this option we leave the settings with the default values:
 - **Turbulence Intensity Fraction = 0.1** (default value).
 - **Turbulent Length Scale = 1.0 cm** (default value).
 - **Velocity** is set to **Constant**.
 - Select **Engine Swirl** in the **Velocity** pull-down and then specify the swirl profile parameters:
 - **Initial Swirl Ratio = -0.7**
 - **Initial Swirl Profile Factor = 3.11**
 - **Initialize Velocity Components Normal to Piston = ON** (checked)
 - Also keep other options at their defaults.
 - Click **Apply** in the Region 1 Initialization Editor panel.
8. Simulation Controls: Under Simulation Limits, for the **Simulation End Points**, specify:
- **Crank Angle Based**
 - **Initial Crank Angle = -95.0**
 - **RPM = 1,300.0**
 - **Cycle Type** is **4-Stroke**.
 - **Final Simulation Crank Angle = 130.0 degrees**. (These two angles correspond to Intake Valve Open and Exhaust Valve Closing, respectively.) Click **Apply**.
9. Simulation Controls > Time Step:
- **Maximum Time Step Option = Constant** and enter the value **1.0E-5 sec**. (This value is selected so the tutorial example will run fast; the default value is 5.0E-6.)
 - Leave other settings at their defaults. Click **Apply**.
10. Simulation Controls > Chemistry Solver:
- Accept the default state of ON (check) for **Use Dynamic Cell Clustering** and accept its defaults.
 - Near the bottom of the panel, turn OFF (uncheck) **When temperature is Reached**. Set the **Activate Chemistry** drop-down menu to **Conditionally**, specifying **During Crank Angle Interval**, with **Starting** and **Ending** values of **-45.0** and **40.0**, respectively. Click **Apply**.

Note

More details about the chemistry solver options, such as Dynamic Cell Clustering and Dynamic Adaptive Chemistry, are available in the Forte User Guide.

11. Output Controls > Spatially Resolved: For the **Spatially Resolved Output Control**, specify:

- **Crank Angle Output Control.**
- **Output every = 5.0 degrees.**
- For **Spatially Resolved Species**, move all the fuel-related and emissions species, such as these, to the **Selection** list: **nc7h16, o2, co2, h2o, co, no, ic8h18, c6h12-1, c6h5ch3** and **ic6h14**. Click **Apply**.

12. Output Controls > Spatially Averaged:


- For the **Spatially Averaged Output Control**, select the **Crank Angle** option and specify **Output Every = 1.0 degree**.
- For **Spatially Averaged Species**, select **o2, co2, co, no, nc7h16, ic8h18, c6h12-1, c6h5ch3, ic6h14, n2, h2o**, and move all species to the **Selection** list. Click **Apply**.

13. Output Controls > Restart Data: In the Workflow tree, check the box for **Restart Data** and then in the Restart panel, check the box that says, **Write Restart File at Last Simulation Step**. Uncheck (turn OFF) any other boxes on the panel. Click **Apply**.

3.3.1.1. Save the Project

1. In the **File** menu, use the **Save As** command to rename and save this project. The project must be saved before running.

3.3.1.2. Run the Case Using Defaults

1. In the Workflow tree, click the Run Simulation node to open the run control interface.
2. Click the green arrow “play” button under **Start/Save** to start the case running. The **Status** changes to “Running.”
3. To interrupt the run, click the red button under **Stop**. Then the **Status** becomes “Stopped”.
4. To start over or restart, first select the file from which you want to start, by clicking on the Browse button in the Run row. Once an appropriate **first** file is selected, click the green **Start** again.
5. The progress of the simulation can be monitored during a run. To monitor the averaged value of simulation variables, click the **Monitor Runs**  icon in the Run Simulation icon bar. A new window will appear, with a list of variables that can be plotted on the left panel in the window. To select one or more variables to plot, check the boxes in front of them in the Monitor Data panel. These plots are shown in [Figure 3.4: Gas Mixture Editor \(p. 21\)](#) to [Figure 3.7: CO and UHC emissions vs. crank angle. \(p. 26\)](#).

Note

To change the units of the plotted variables, go back to the Simulation Interface, **Edit** menu and choose **Edit Preferences**. In the Preferences panel, select the units desired. The next time the Monitor window updates the plots, it will automatically reflect these updated unit preferences.

3.3.1.3. Visualizing the Results

1. When a run is finished, the **Status** becomes **Complete** in the Run panel of the main Simulation interface window. In the Monitor Panel, use the **Harvest/Visualize** button to save the harvested results as an **.ftres** file. Note that multiple runs can be selected for this purpose.
2. ANSYS Forte Visualizer will be loaded automatically following the “Harvest Results” step if the **Launch Visualizer after Harvest** option is checked. You can follow the Forte User Guide to post-process the results.
3. In the Visualizer, we are interested in creating some average line plots for Pressure, Net Heat Release Rate (Net HRR) and emissions such as CO vs. Crank Angle. The Visualizer view when you read in the **Nominal.ftres** file is shown in [Figure 3.5: Simulation result in Visualizer. \(p. 24\)](#). The geometry is in the 3-D Viewer window at the center and the Visualization tree is on the right. We will use a wizard on the left to create line plots for Pressure, Net HRR, UHC and CO. These figures are shown as [Figure 3.4: Gas Mixture Editor \(p. 21\)](#) and [Figure 3.7: CO and UHC emissions vs. crank angle. \(p. 26\)](#).

3.4. Reference

This case models the engine experiment from the following paper:

Karthik V. Puduppakkam, Long Liang, Chitrakumar V. Naik, Ellen Meeks, Sage L. Kokjohn and Rolf D. Reitz, “Use of Detailed Kinetics and Advanced Chemistry-Solution Techniques in CFD to Investigate Dual-Fuel Engine Concepts”, *SAE International Journal of Engines*, Vol. 4, No. 1, pp. 1127-1149, 2011.

Figure 3.5: Simulation result in Visualizer.

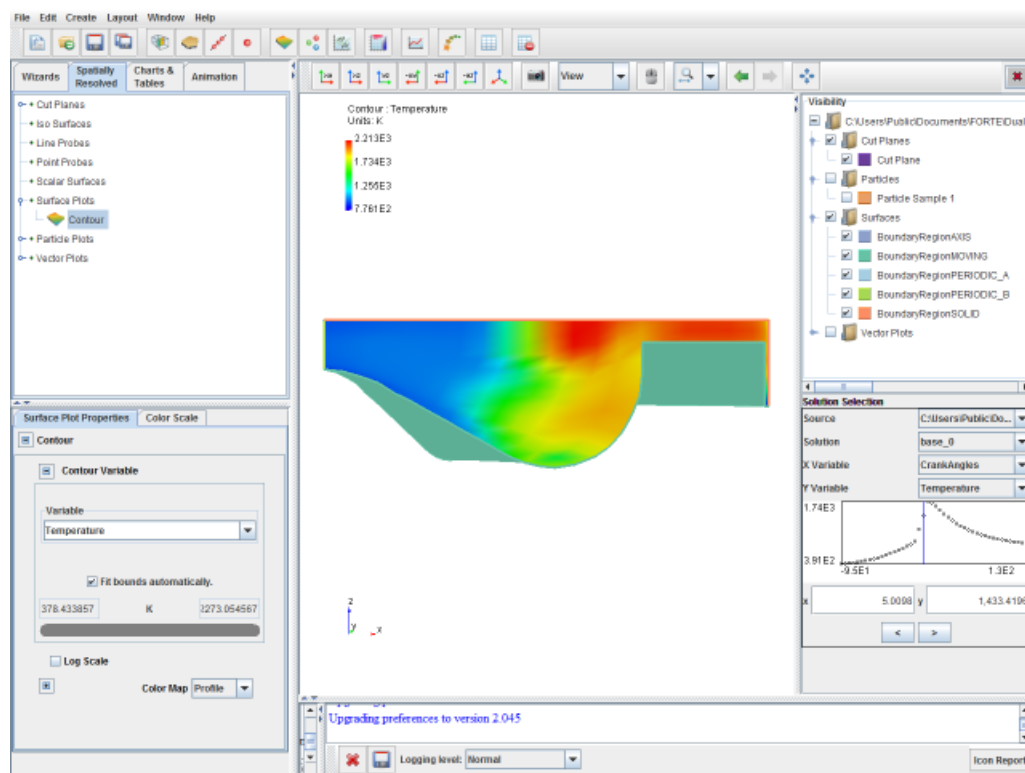


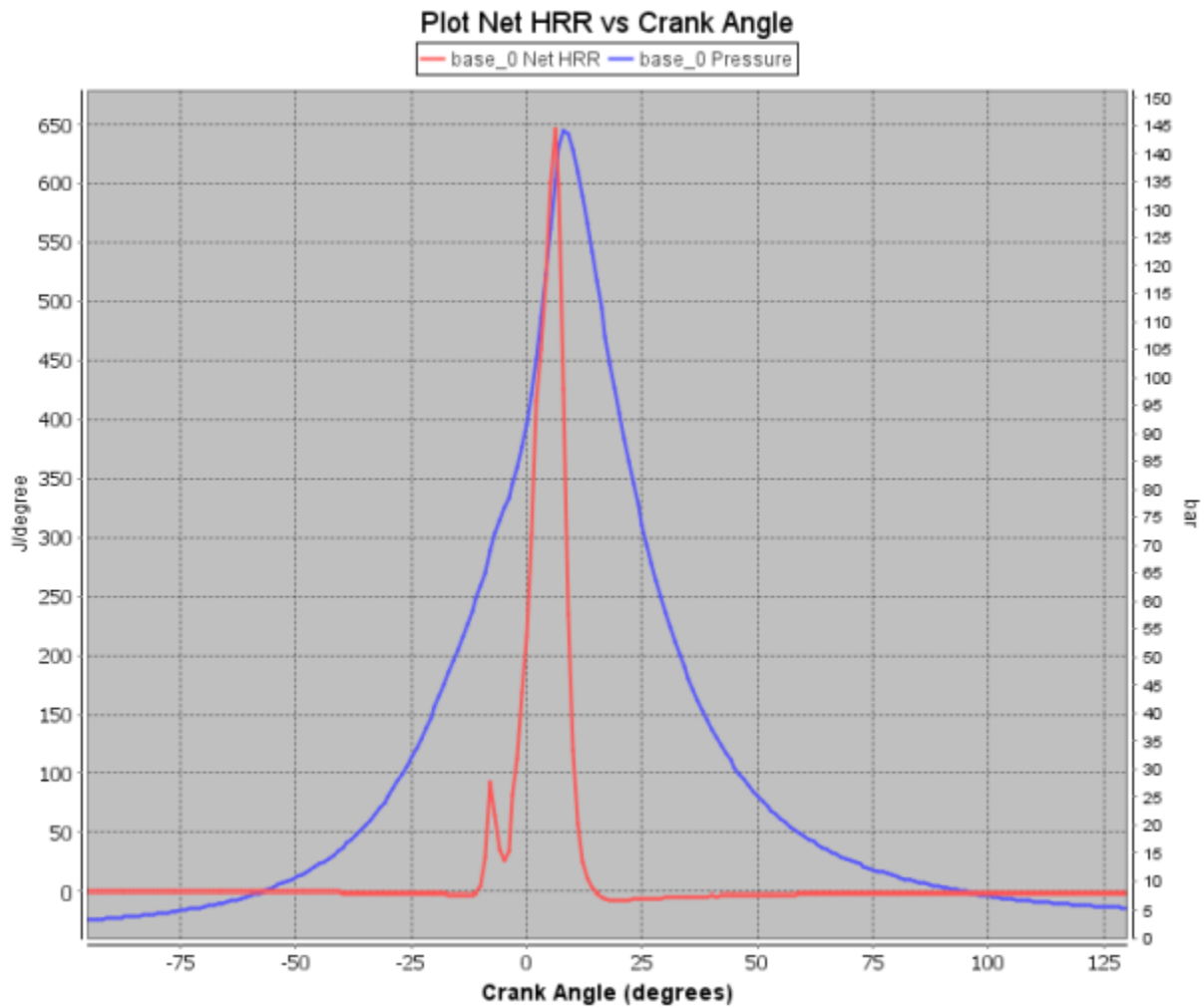
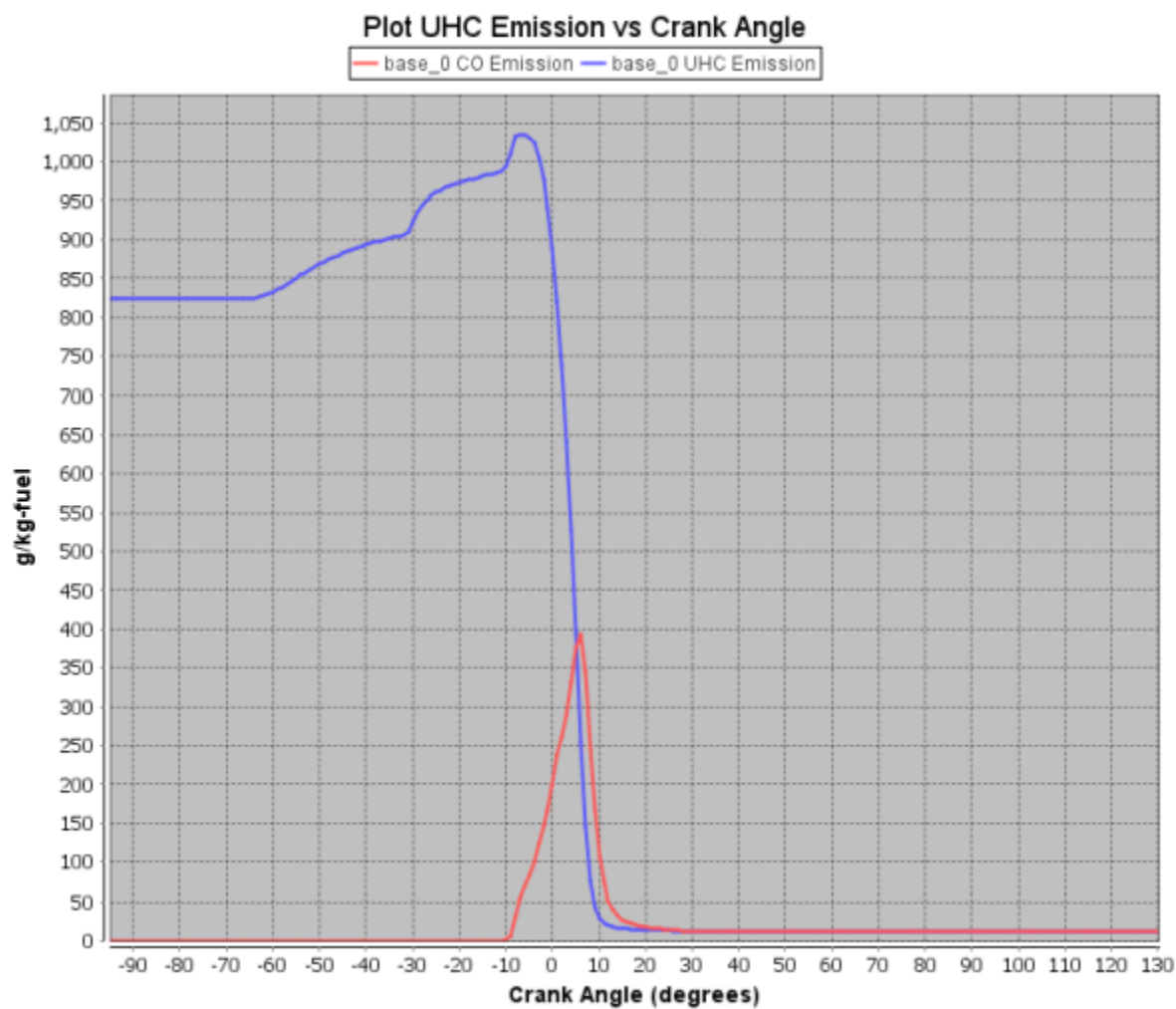
Figure 3.6: Pressure and net heat release rate vs. crank angle.

Figure 3.7: CO and UHC emissions vs. crank angle.



Chapter 4: Spray Bomb Modeling

Accurately capturing fuel injection, spray breakup and spray vaporization are important aspects of engine-design simulations for direct-injection engines. However, it is difficult to isolate the effects of spray behavior in a full engine simulation involving combustion, moving boundaries and complex turbulent flow. Spray bomb experiments are an excellent method for isolating the spray behavior prior to engine design and analysis. A spray bomb is a quiescent environment where liquid is injected without any moving boundaries, such that the spray characteristics can be examined in detail. Any engine simulation involving liquid injection must account for the key aspects of atomization such as liquid-jet breakup, droplet formation, secondary breakup, droplet collisions, droplet coalescence and vaporization. This tutorial describes how to use ANSYS Forte in spray bomb simulations that account for all of these phenomena.

4.1. Data Provided

4.1.1. Files Used in This Tutorial

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

and select the Forte tutorial files you wish to download. The files for this tutorial include:

- ***Evaporating_Spray_Tutorial.fmesh***: This is a Forte body-fitted mesh file. This file will provide the spray bomb geometry for this Automatic Mesh Generator (AMG) tutorial.
- ***Evaporating_Spray_Tutorial.ftsim***: A project file of the completed tutorial, for verification or comparison of your progress in the tutorial set-up.

The tutorial sample files are provided as a download. You have the opportunity to select the location for the files when you download and unzip or untar the sample files.

4.1.2. Project Comparison Utility

The Forte installation includes the `cgns_util export` command, which you can use to compare the parameter settings in the project file generated at any point during your tutorial set-up against the provided ***.ftsim***, which has the parameter settings for the final, completed tutorial. This command is described in the Forte User Guide.

Briefly, you can double-check project settings by saving your project and then running the `cgns_util` to export your tutorial project, and then to export the provided final version of the tutorial. Save both versions and compare them with your preferred diff tool, such as DIFFzilla. If all the parameters are in agreement, you have set up the project successfully. If there are differences, you can go back into the tutorial set-up, re-read the tutorial instructions, and change the setting of interest.

4.1.3. Time Estimate

As a guideline for your own simulations, this tutorial is estimated to take 4 hours on a cluster of 16 nodes with Intel® Xeon® processor E5-2690 at 2.90 GHz (16 cores).

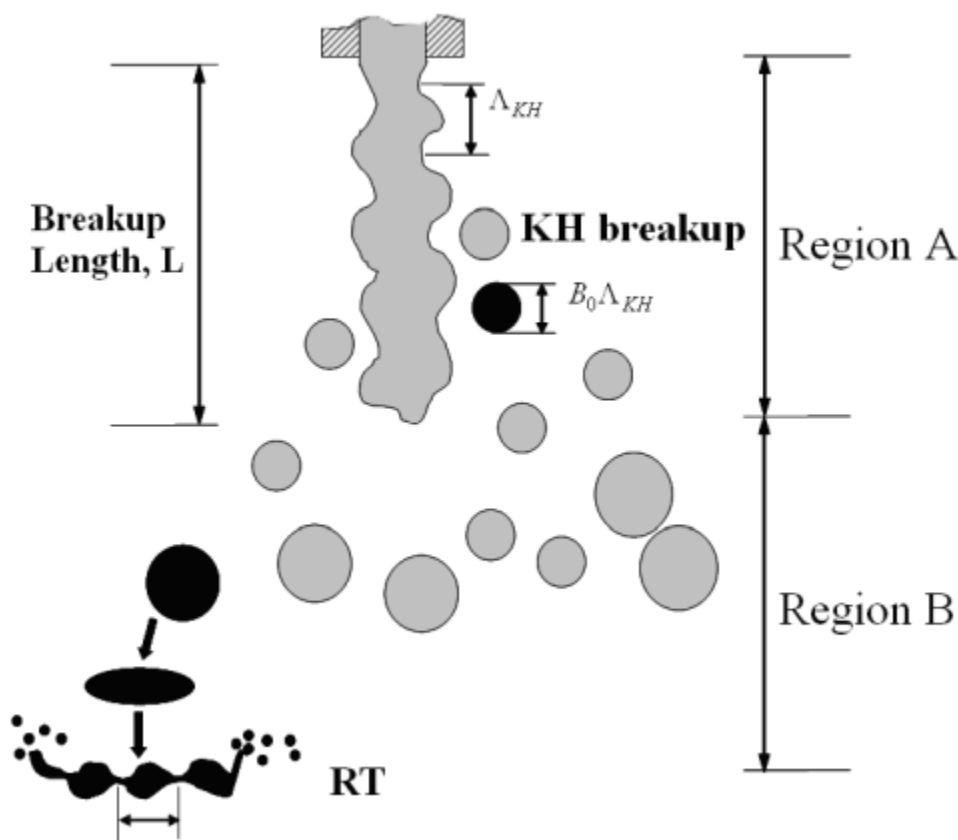
4.2. Modeling Solid-Cone Spray Injection

For solid-cone spray injections, Forte utilizes the Kelvin-Helmholtz/Rayleigh-Taylor (KH-RT) spray-breakup models that have been proven to be accurate over a wide range of direct-injection conditions. Together with the gas-jet model, which provides accurate gas-jet entrainment calculations without requiring mesh refinement around the spray, the ANSYS Forte spray model provides results that are insensitive to mesh resolution or simulation time step. These spray model components are described in detail in the Forte Theory Manual.

In many cases, the starting point for a spray injection specification is an estimate of the nozzle discharge coefficient and spray-cone angle. Forte's nozzle-flow model can optionally provide initial conditions for the spray model by including detailed calculation of the nozzle-flow including cavitation effects.

The droplet breakup model consists of two key models. The KH breakup model is applied within the breakup length (see [Figure 4.1: KH-RT spray model definitions. \(p. 28\)](#)). The RT model is then used further downstream with the KH model to predict secondary breakup. The unsteady gas-jet model effectively removes mesh dependency by calculating the gas velocity in the spray region using an analytical model.

Figure 4.1: KH-RT spray model definitions.



Forte allows you to set several constants that control the initial breakup in the KH region and the secondary breakup in the RT region. Defaults for these constants are provided in Forte that have been

determined after extensive validation with experimental data and should be used initially. However, there could be some cases where an adjustment in these constants would prove useful. The following table describes the specific spray constant name, its default value, a suggested range of adjustment and what the constant controls.

Table 4.1: Description of default ANSYS Forte spray constants.

Constant Name	Default Value	Adjustment Range	Description
Size Constant of KH Breakup	1.0	0.5 - 2.0	Affects radius of new droplets formed from initial breakup.
Time Constant of KH Breakup	40	10 - 80	Most important constant for controlling spray penetration length.
Critical Mass Fraction for New Droplet Generation	0.03	N/A	Determines fraction at which new droplets can leave their parent parcel.
Size Constant of RT Breakup	0.15	0.1 - 0.3	Affects the radius of the secondary droplets arising from the catastrophic breakup of the parent droplet
Time Constant of RT Breakup	1.0	0.5 - 2.0	Affects the characteristic time required to break the parent droplet
RT Distance Constant	1.9	1.5 - 3.0	Affects the initial breakup length when the RT model is initiated

4.2.1. Spray Bomb

4.2.1.1. Problem Description

This tutorial considers a cylindrical chamber with no moving walls and quiescent gas (see [Table 4.2: Spray bomb simulation case set-up and test conditions. \(p. 30\)](#)). In this case, pure *n*-heptane is injected into the chamber. This single component, fuel is injected from a single solid-cone injector, where the nozzle hole is located on the wall and the injection is directed towards the center of the chamber. The initial gas in the chamber is a blend of N₂, CO₂, and H₂O, such that the mixture is non-reacting. As the intention of this example is to represent diesel-fuel injection, we use the physical properties of *n*-tetradecane to represent the fuel. This combination of chemistry model being represented by one fuel surrogate (in this case, *n*-heptane) while the physical model is represented by another fuel surrogate (in this case, *n*-tetradecane) is an approach often taken in simulating diesel engine combustion. In this way, the spray-bomb simulation comparisons to experiment can be used to verify the behavior of the surrogate model approach as it will be used in a combustion simulation. For this case, a square injection profile

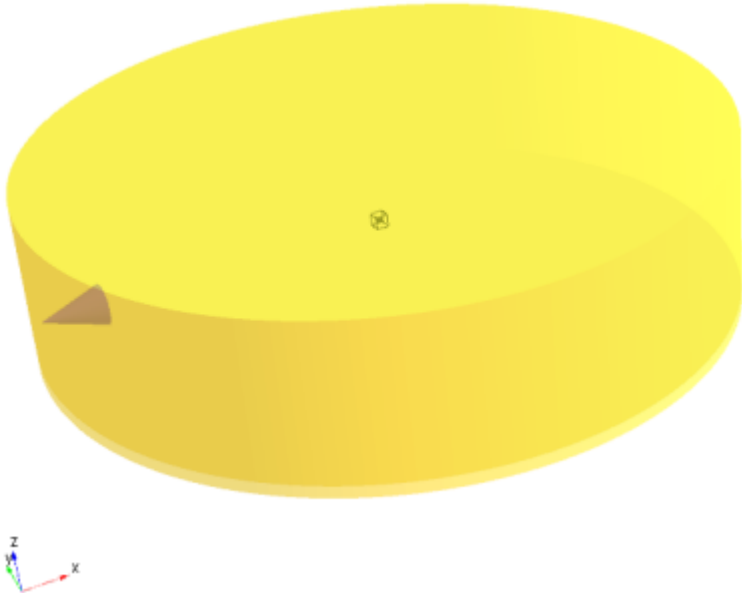
is used. Chemistry calculations are effectively stopped by setting the chemistry solver to begin at a temperature that will not be reached during the simulation.

This tutorial shows the use of the Automatic Mesh Generator. We import a previously generated mesh to provide the spray bomb geometry and select the option upon import to only import the geometry surfaces for use with Automatic Mesh Generation. Mesh sizes of 1 mm, 2 mm and 3 mm are then used to test mesh dependency. Results for penetration length of the three different mesh sizes are compared against experimental data from SAE Technical Paper 960034 by J.D. Naber and D.L. Siebers [1 (p. 39)].

Note

All Editor panel options that are not explicitly mentioned in this tutorial should be left at their default values. Changed values on any Editor panel do not take effect until you press the **Apply** button.



Table 4.2: Spray bomb simulation case set-up and test conditions.

Parameter	Evaporating Spray Case
Mesh	Disc-shaped constant-volume combustion chamber with 114.3 mm diameter and 28.6 mm thickness 
Fuel Type	Diesel DF2 (Experiment)Tetradecane (Surrogate)
Fuel Temperature	436 K
Injection Pressure	1,370 bar
Injection Duration	4 ms
Nozzle Diameter	257 μm
Chamber Conditions	P _a = 40; 83; and 170 bar

Parameter	Evaporating Spray Case
	$T_a = 1,000 \text{ K}$



4.2.1.2. Mesh Setup

In this tutorial, we will import an existing structured mesh with a size of 2 mm. Mesh sizes of 1 mm, 2 mm, and 3 mm will be investigated to demonstrate mesh independence of the solution. The spray bomb geometry and test conditions are shown in [Table 4.2: Spray bomb simulation case set-up and test conditions](#). (p. 30).

- Import the mesh using Geometry on the Workflow tree. Click the **Import Geometry**  icon. In the dialog, pull down and select **Body Fitted mesh from Kiva-3Vformat**, click **OK** and then select **Only import the geometry surfaces for use with Automatic Mesh Generation**. In the dialog that opens, navigate to the **Evaporating_Spray_Tutorial.fmesh** file in the location where you saved the downloaded sample files.
- The imported file displays in the 3-D View area, but may not be ideally zoomed or centered. Click the **Refit**  icon to center and resize the mesh.
- **Mesh Controls:** Under Mesh Controls, we need to set the **Material Point** to a location that will always be “inside” the boundaries and should be located at least one unit cell length away from any boundaries. To see the coordinate values, select Geometry > Reference Frames > Solid.1 in the left-hand Workflow tree. The Editor panel below will show the min/max coordinates. From here we can see that the minimum and maximum coordinate values are **-5.724** and **5.714 cm** for x- and y-axes, and **0** and **2.86 cm** for the z-axis. So, return to Mesh Controls > Material Point, and in the **Reference Frame = Global Origin** options, set **x=1 cm**, **y=2 cm** and **z=1 cm**, and click **Apply**. Once it is set, you can see the Material Point in the 3-D View panel (as a small cube) and it can be made visible/invisible in the Visibility tree on the right, under **Mesh Controls**. (You may need to adjust the opacity of the Solid.1 Geometry item in the Visibility tree.)
- In the Workflow tree, set the **Global Mesh Size** to **0.2 cm**. This will set the default size of cells to be 2 mm long on each side of the cell cube. Click **Apply**.

Note

This is within the recommended mesh range; a coarser setting of 0.3 cm could be used for tutorial purposes to produce a shorter runtime.

- Click the Mesh Controls node in the Workflow tree. In the icon bar, click the **New Surface Refinement Depth**  icon and name the new control “surfaces”. (This indicates that along the selected surface, cells of this smaller size will be used.) A new SurfaceDepth item appears under Mesh Controls in the Workflow tree. In the panel that appears, select the **Solid.1** surface from the Location list, set the mesh **Size as Fraction of Global Size** to **1/2** with an extension of **1** layer. Click **Apply**.
- Click the **Solution Adaptive Meshing**  icon on the Mesh Controls Editor panel and name the new control **SAM-Velocity**. Set the **Quantity Type = Gradient of Solution Field** and **Solution Variables = VelocityMagnitude**. **Bounds = Statistical** and **Sigma Threshold = 0.5**. Set the **Size as Fraction of Global Size = 1/4**. The refinement is **Active = Always**, and the **Location** option is **Entire Domain**. Click **Apply**.

4.2.1.3. Models Setup


Chemistry: After the mesh, the models are chosen. Chemistry is set with Models > Chemistry. Click the



Import Chemistry icon and import **Diesel_1comp_35sp.cks** from the installed ANSYS Forte **data** directory. An option is provided to view the chemistry details such as chemistry source, pre-processing log, gas phase input, gas phase output, thermodynamic input, transport input and transport output. Note that in this case, the chemistry is effectively turned off, so the main reason to import the chemistry set is to obtain the chemical species *nc7h16* that will represent the fuel and *n2* for the gas in the chamber.

Transport: The default RNG k-E model turbulence settings are used in this tutorial. Those are specified in the Editor panel for Models > Transport > Turbulence. The default fluid properties are also used, which are at Models > Transport.

Spray Model: To set up the spray model, turn on (check-mark) Models > Spray Model. First, choose the global **Spray Properties**, which include using the **Radius of Influence** model for droplet collisions with the default **Radius of Influence** set to 0.2 cm. Also check **Use Vaporization Model** to include the effects of vaporization, since this case is into a relatively hot gas, where vaporization will occur.

- **Create Injector:** Add a solid-cone spray injector through the Models > Spray Model panel, by clicking on the **Solid Cone Spray Injector**  icon. Name the injector (**Injector**) and configure the injector and its fuel. In the Injector Editor panel, select **Create New...** in the **Composition** drop-down menu and click the pencil icon to open the Fuel Mixture Editor. In the Fuel Mixture panel, click the **Add Species** button and select **nc7h16** (i.e., *n*-heptane) as the **Species**, **n-Tetradecane** as the **Physical Properties** and **1.0** as the **Mass Fraction**. (Note that you must press ENTER after entering the values in the table.) At the bottom of the panel, type a name such as "**n-heptane**". Click **Save** and **Close** the window.
- Using an **Injection Type** of **Pulsed Injection**, change the **Parcel Specification** to **Number of Parcels**, then set **Injected Parcel Count** = **5,000**.
- Set **Inflow Droplet Temperature** = **436.0 K**.
- Set **Spray Initialization** to **Constant Discharge Coefficient and Angle**, and **Discharge Coefficient** = **0.78**, and **Mean Cone Angle** = **19.04** degrees.
- Keep default values for **Droplet Size Distribution** and, under the **Solid Cone Breakup Model Settings**, for the **KH Model Constants**, **RT Model Constants**, and **Use Gas-Jet Model**. Click **Apply**.

Next we will add the nozzle and specify the injection parameters under this new injector (see [Table 4.4: Nozzle settings.](#) (p. 33) and [Table 4.5: Injection settings.](#) (p. 33)).

Table 4.3: Injector Settings

Parameter	Setting
Injected Spray Parcels	5,000
Inflow Droplet Temperature	436.0 K
Spray Initialization	Constant Discharge Coefficient and Angle
Discharge Coefficient	0.78
Mean Cone Angle	19.04


- **Create a Nozzle:** Click the **New Nozzle**  icon on the Injector icon bar and name the nozzle **"Nozzle"**. The Nozzle item then appears in the Workflow tree, and the Editor panel and icon bar transform to allow specification of the Nozzle geometry and orientation. Set the parameters in the Editor panel to use the **Reference Frame** parameters to specify the nozzle location and direction. Keep the **Global Origin** and use the following settings:

Table 4.4: Nozzle settings.

Parameter	Setting
Location	
Coordinate System	Cylindrical
R	5.7 cm
•	180.0 degrees
A	1.43 cm
Spray Direction	
Coord. System	Spherical
θ	90.0 degrees
•	0.0 degrees
Nozzle Size	
Nozzle Diameter	257 microns

- If you cannot see the nozzle in the 3-D View, you may need to adjust the opacity or color choice of the nozzle or the boundary conditions that may be blocking your view of it. Select the desired item in the Visualization tree (on the right side) and right-click to turn ON/OFF visualization, change opacity levels, or change the color assigned to that item.


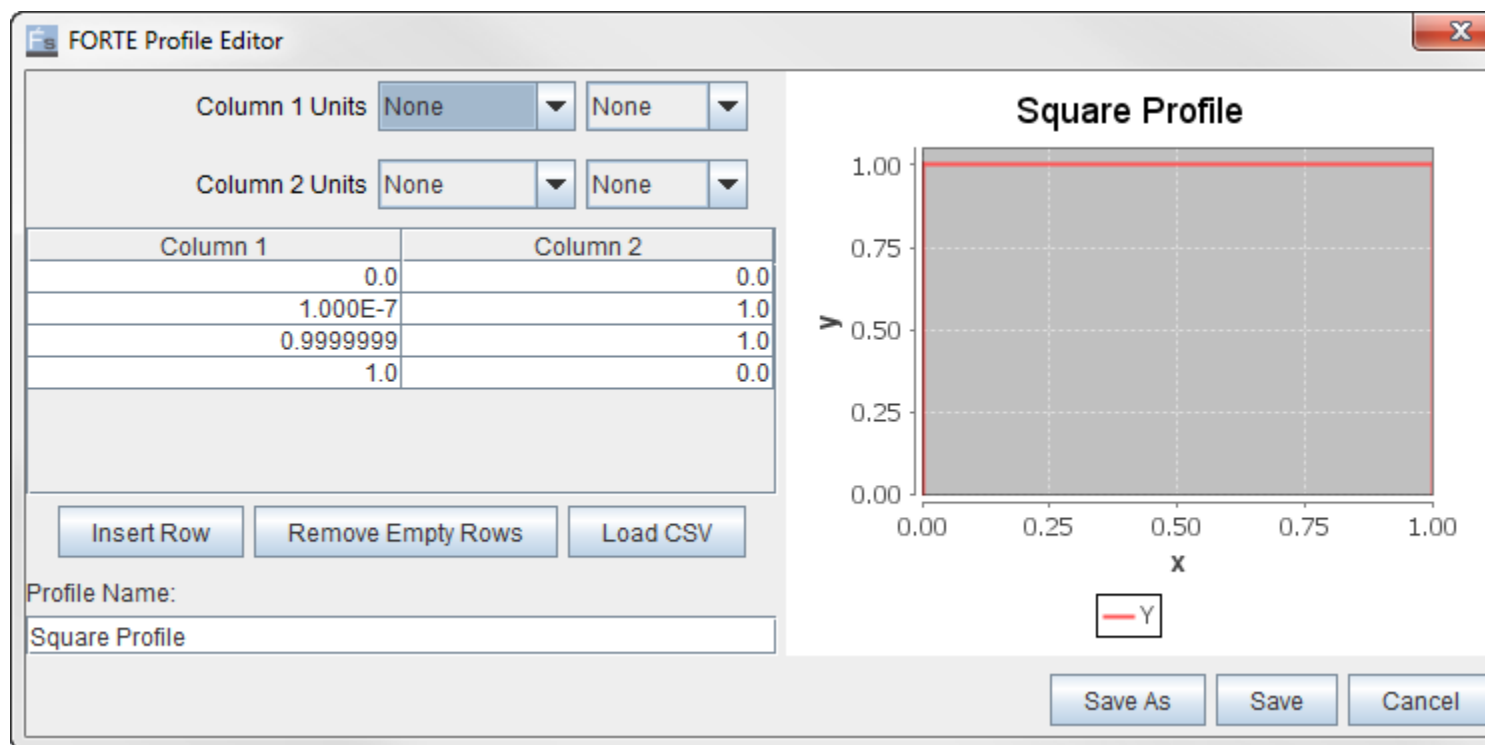
- **Create an Injection:** In the Workflow tree, click Injector again and click the **New Injection**  icon on the **Injector** icon bar and name the injection **Injection**. The new Injection item appears in the Workflow tree, and the Editor panel and the icon bar transform to allow specification of the injection properties. In the Editor panel, use the following settings:

Table 4.5: Injection settings.

Parameter	Setting
Timing	Time
Start	0.0 sec
Duration	0.004 sec
Velocity Profile	Square Profile
Total Injected Mass	0.058 g

The resulting injection profile can be viewed by clicking on the pencil icon next to the **Velocity Profile** entry box. In this window, you can also import a new velocity profile for the injection from a **.csv** file or make edits directly in the table (see [Figure 4.2: View injection profile. \(p. 34\)](#)).

Figure 4.2: View injection profile.

4.2.1.3.1. Boundary Conditions

Create a new boundary by right-clicking on the Boundary Conditions Workflow tree, and **Add a Wall**. Now, under the Boundary Conditions Workflow tree node, a new entry called Wall will be found. Under **Location** for the **Wall** boundary condition, select **Solid.1**. The only boundary condition that must be specified for the wall and liner is temperature. For both Boundary Condition > Head and Liner, select the **No Slip** model for handling the boundary layer near the wall and set the **Temperature = Constant** and **Temperature = 451 K**. Keep **Heat Transfer** checked (ON).

4.2.1.3.2. Initial Conditions


Initial conditions will be set only for the Default Initialization region. Specify the parameters for the initial conditions as follows:

- **Composition:** Use the Composition Editor to set these parameters: In the Gas Mixture Editor, keep the **Composition = Mass Fraction**. Then click the **Add Species** button and select **h2o**, **co2** and **n2** to add. When these 3 species are in the Species column in the **Composition** table, enter **0.9033** for the **n2 Fraction**, **0.0611** for **co2** and **1.0** for **h2o**. Name this **Composition 1** in the text field at the bottom of the Gas Mixture window. Click **Save** and close that window.
- **Temperature = 1,000.0 K.**
- **Pressure = 83.04 atm.**
- In the pull-down menu for the **Turbulence** parameters, select **Turbulence Intensity and Length Scale** as the way in which we will specify the initial turbulence. For this option we provide an explicit value for the initial turbulence intensity, but use a length-scale approximation to determine the turbulence dissipation energy. Use these values:
 - **Turbulence Intensity Fraction = 0.0.**

– **Turbulent Length Scale = 1.0 cm.**

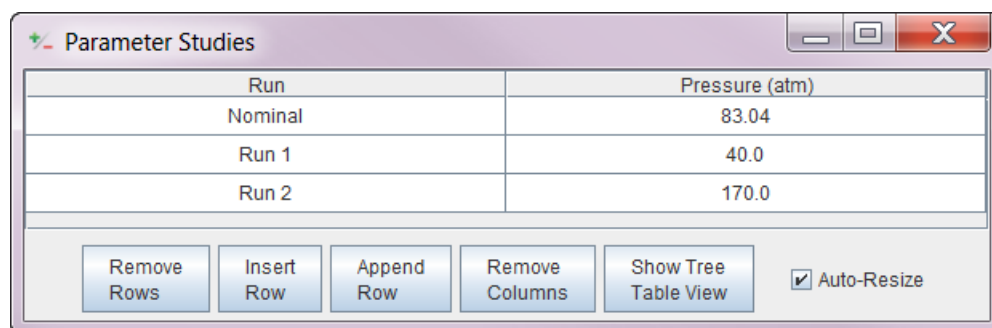
- Select **Velocity Components** in the **Velocity** pull-down and then specify the velocity components, using the default **Reference Frame** with **Global Origin** and **Cartesian** coordinates set at **0.0, 0.0, 0.0**.
- Click **Apply** in this panel.

4.2.1.3.3. Parameter Study on Pressure

To create a parameter study on pressure, click the Parameter Study  icon next to the **Pressure** setting (at 83.04 atm). In the resulting dialog, change the specification to **From End Points**, set these values: **First run value (A) 40.0, Last run value (B) 170.0, Number of runs (n) 2**. Click **OK**.

Close or minimize the Parameter Studies window. Note that the **Pressure** label in the Editor panel is now blue to indicate that it is associated with a parameter study.

Figure 4.3: Spray - Parameter study on pressure.



When this simulation runs, a run will occur at each of these pressure settings. See [Results \(p. 37\)](#) to see how the results vary with the changes in pressure.

4.2.1.3.4. Simulation Controls

Simulation controls allow you to define the simulation limits, time step, chemistry solver and transport terms.

Simulation Limits: Select the Simulation Controls node and then use **Simulation Limits** to select a **Time-based** simulation with a **Max. Simulation Time** of **0.004 sec**.

Time Step: Use Simulation Controls > Time Step to set the **Initial Simulation Time Step** to **5.0E-7 sec** and use a **Constant Max. Time Step Option** of **5.0E-6 sec**. **Advanced Time Step Control Options** can be left at the default settings.

4.2.1.3.5. Chemistry Solver

Use Simulation Controls > Chemistry Solver so **Activate Chemistry** is **Always Off** (at the bottom of the panel) to prevent the chemistry from being solved in this non-reacting case.

4.2.1.3.6. Transport Terms

The transport tolerances and maximum number of iterations are left at their default values.

4.2.1.4. Output Controls

Output controls determine what data are stored for viewing during the simulation and for creating plots, graphs and animations in the ANSYS Forte Visualizer.

Spatially Resolved: Set the **Temporal Output Control** reporting every **0.004 sec**. To create the **Spatially Resolved Species** list, be sure to move **nc7h16** to the **Selection** list. Also move **n2**, **co2**, and **h2o**. Under **Restart**, clear (uncheck) the option **Write Restart File at Last Simulation Step**. Click **Apply**.

Spatially Averaged: Set the **Temporal Output Control** reporting every **4.0E-5 sec**. To create the **Spatially Averaged Species** list, be sure to move **nc7h16** to the **Selection** list. Also move **co2**, **o2**, and **n2**. Click **Apply**.

4.2.1.5. Save Project

Save your project with the **Save** command in the **File** menu, but choose a new name for the project to avoid overwriting the original **.ftsim** file. The saving step includes validation of the project.

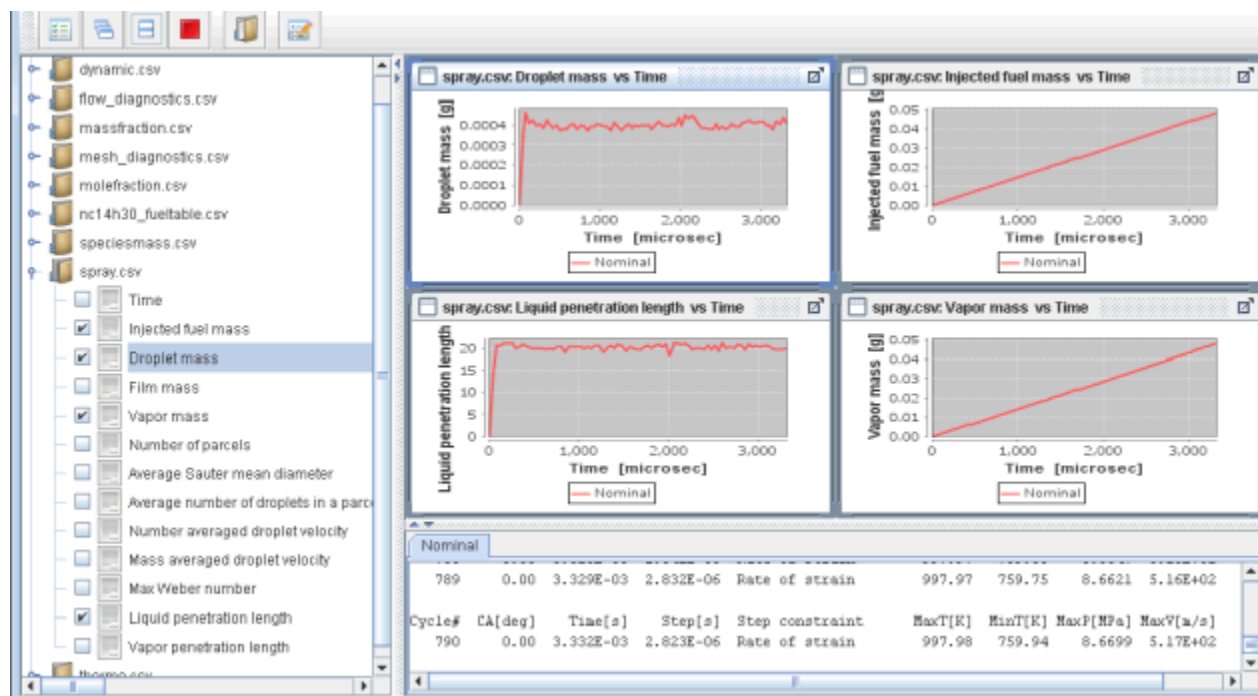
4.2.1.6. Run Settings

The settings here depend on the system and environment for your simulations. The default for the Run Settings panel is to have nothing selected.

Run Options: Under **Job Script Options**, change **Default Run Type** to **Parallel** and change the default **MPI Arguments** to **8**. (If you do not have MPI installed and configured, keep the default of a serial run.)

4.2.1.7. Run Simulation

All that is needed to be done now is to select Run Simulation and click on **Start** once ANSYS Forte has given you the green light and reports that it is ready for the simulation. You can monitor the results by clicking on the Monitor icon. In the Monitor window that opens, you can select which result you want to monitor for the spray such as **Mass Injected** and **Penetration Length**.

Figure 4.4: Monitor ANSYS Forte run

4.2.1.8. Results

This spray bomb case was run with three mesh sizes (1 mm, 2 mm and 3 mm) and then compared against experimental data from Naber and Siebers 1 (p. 39). The experimental data provided consisted of penetration depth for the test spray. The experimental data show a penetration depth of 22.6 mm as is shown in [Figure 4.5: Mesh sensitivity on spray penetration depth \(p. 38\)](#). The simulation results show good agreement with the spray penetration depth as is also seen in [Figure 4.5: Mesh sensitivity on spray penetration depth \(p. 38\)](#). It is important to note that ANSYS Forte defines the spray penetration depth that captures 95% of the liquid. These results also show that the spray is relatively insensitive to the mesh for the three mesh sizes considered. The 1 mm and 2 mm results lie on top of each other. The diversion of the results for 3 mm, suggest that size is a little too large to resolve the fluid mechanics overall and so for these types of cases we would recommend the 2 mm resolution. However, even the coarse mesh results are not far from the resolved-mesh solution, which is an important feature of the ANSYS Forte simulations. Another interesting feature is to look at the how the droplet mass and vapor mass evolve during the simulation time. These results are shown in [Figure 4.6: Mesh sensitivity on droplet mass and vapor mass \(p. 38\)](#), where the mesh is also shown to be insensitive to the mesh size, even with the coarsest mesh. Visualization of the spray results are shown in [Figure 4.7: Spray bomb penetration for 2 mm mesh. \(p. 39\)](#) at time = 0.0015 sec.

The results of the automated parameter study on ambient pressure in the spray bomb are shown as [Figure 4.8: Results of automated parameter study on ambient pressure in the spray bomb. \(p. 39\)](#). The spray penetration is decreased as the ambient pressure increases due to the increased chamber gas density and drag.

Figure 4.5: Mesh sensitivity on spray penetration depth

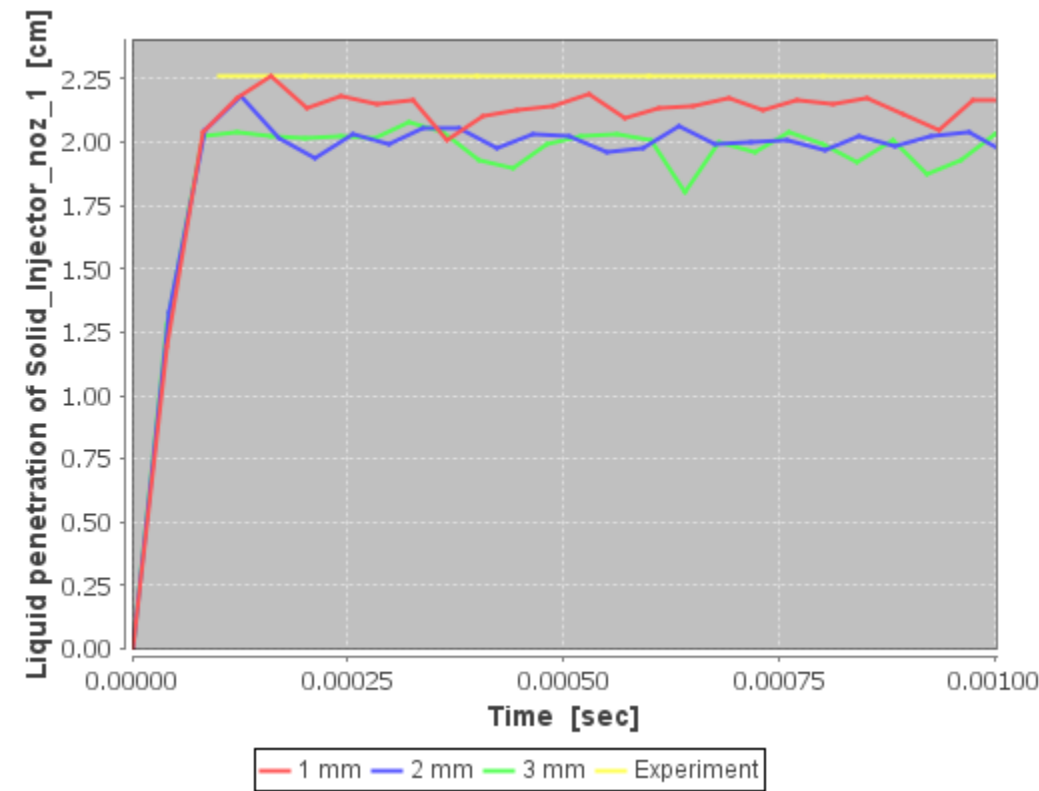


Figure 4.6: Mesh sensitivity on droplet mass and vapor mass

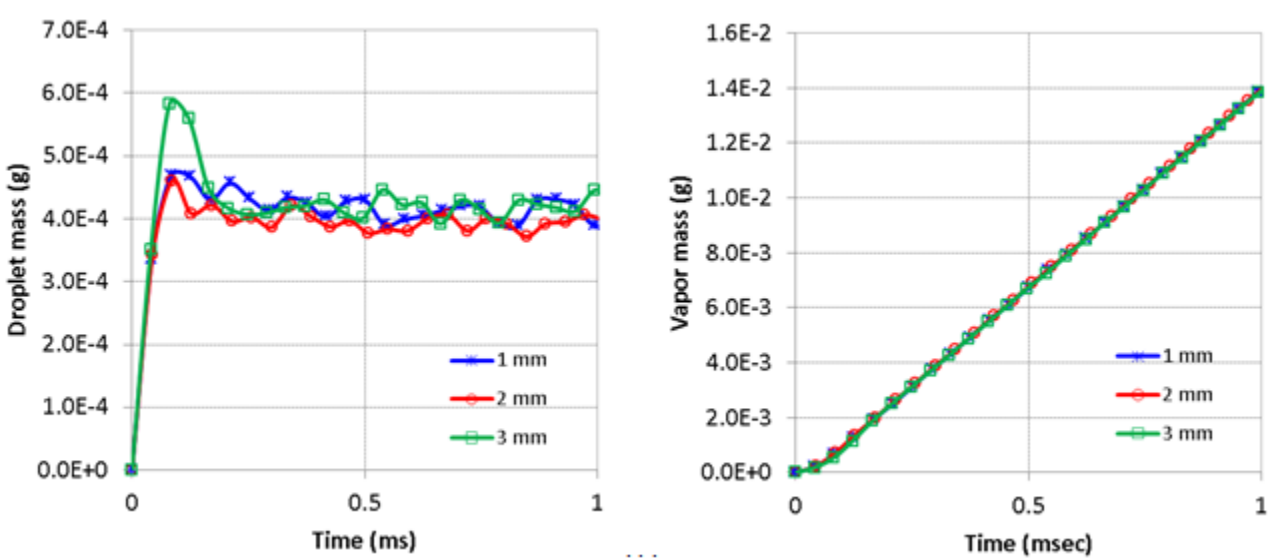


Figure 4.7: Spray bomb penetration for 2 mm mesh.

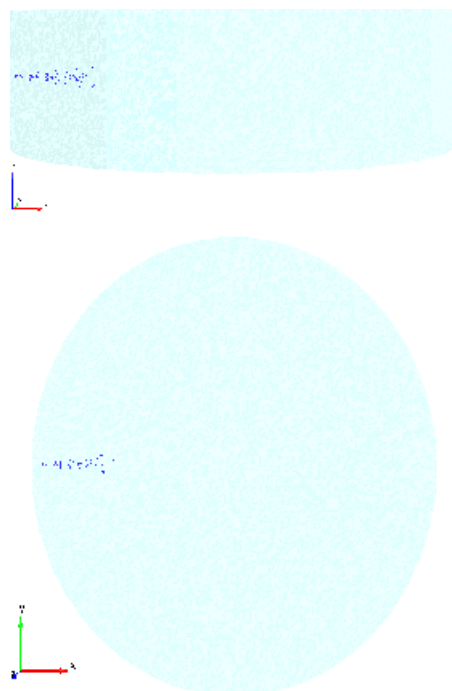
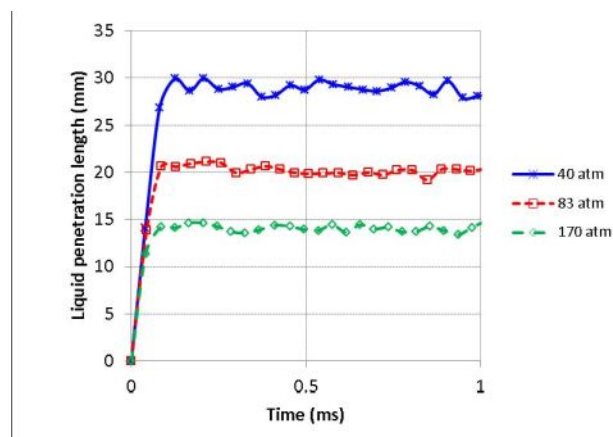


Figure 4.8: Results of automated parameter study on ambient pressure in the spray bomb.



4.3. Reference

This tutorial is based on the work in this published paper:

1. Naber, J. D. and Siebers, D. L., "Effects of Gas Density and Vaporization on Penetration and Dispersion of Diesel Sprays", SAE Technical Paper 9p60034, 1996.

Chapter 5: Port-Injected Spark Ignition Engine

This tutorial describes how to use ANSYS Forte CFD to simulate combustion in a port-injected spark ignition internal combustion engine with moving valves. Engine geometry is imported and ANSYS Forte's automatic mesh generation is used to create the computational mesh on-the-fly during the simulation.

5.1. Data Provided

5.1.1. Files Used in This Tutorial

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

and select the Forte tutorials desired for download. The files for this tutorial include:

- ***SIEngine_Sample.stl***: This is the geometry file.
- ***ExhaustLift.csv*** and ***IntakeLift.csv***: Two files of data describing the intake and exhaust valve lifts. Profiles such as these can be imported (from **.csv** files) or manually entered in the Profile Editor.
- ***Spatial_Output_CAs.csv***: Specifies when spatially resolved data such as velocity, temperature, species concentrations, etc., will be output.
- ***SIEngine_PortInjected_AMG_Tutorial.ftsim***: A project file of the completed tutorial, for verification or comparison of your progress in the tutorial set-up.

The sample files are provided as a download. You have the opportunity to select the location for the files when you download and uncompress the sample files.

5.1.2. Project Comparison Utility

The Forte installation includes the `cgns_util export` command, which you can use to compare the parameter settings in the project file generated at any point during your tutorial set-up against the provided **.ftsim**, which has the parameter settings for the final, completed tutorial. This command is described in the Forte User Guide.

Briefly, you can double-check project settings by saving your project and then running the `cgns_util` to export your tutorial project, and then to export the provided final version of the tutorial. Save both versions and compare them with your favorite diff tool, such as DIFFzilla. If all the parameters are in agreement, you have set up the project successfully. If there are differences, you can go back into the tutorial set-up, re-read the tutorial instructions, and change the setting of interest.

5.1.3. Time Estimate

As a guideline for your own simulations, this tutorial is estimated to take approximately 20.2 hours on a cluster with 16 nodes with dual Intel® Xeon® processor E5-2690 at 2.90 GHz (8 total cores).

5.2. Port-Injected Spark Ignition Engine


5.2.1. Problem Description

The port-fuel injection is approximated by a premixed, pre-vaporized blend of fuel and air in the intake port. There are two exhaust and intake valves and the simulation domain includes the intake and exhaust manifolds (Figure 5.1: Port-injected engine geometry with valves and ports defined (p. 43)). In other words, the initialization of the gas composition is used to specify the fuel-air mixture in the intake-port region and perfect mixing within the port is assumed. The case is reacting flow, with a single component, 59-species single-component fuel that is appropriate for engine simulations that are not concerned with knock. See the ANSYS Forte User Guide for details of the chemistry.

5.2.1.1. Import the Geometry

Note

All Editor panel options that are not explicitly mentioned in this tutorial should be left at their default values. Changed values on any Editor panel do not take effect until you press the **Apply** button. Always press the **Apply** button after modifying a value, before moving to a new panel or the Workflow tree.

In this tutorial, we will import an existing geometry into ANSYS Forte and set up the automatic mesh generation using a global mesh size and adapting the mesh near the valves. To import the geometry, go to the Workflow tree and click Geometry. This opens the Geometry icon bar. Click the **Import Geometry**  icon. In the resulting dialog, pull down and select **Surfaces from STL file**. In the dialog that opens with STL file import options, accept the defaults.

The mesh will be automatically generated during the simulations. When the file browser launches, navigate to the folder **Tutorial_SIENGINE_PFI_Automesh** then select **SIEngine_sample.stl**. Note that once you have imported the geometry, there are a number of actions that you can perform on the items in the Geometry node, such as scale, rename, transform, invert normals, or delete geometry elements.

The Geometry imports in an opaque mode and possibly preset zoom level. It is often helpful to Re-


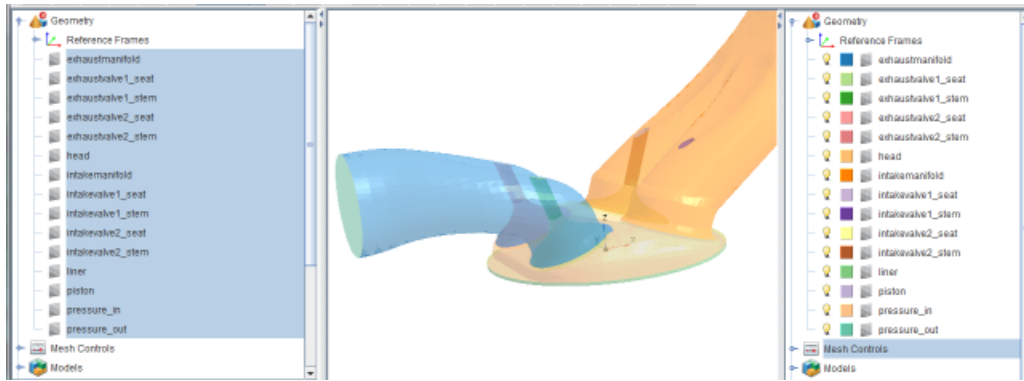
fit  the view or use the mouse wheel to re-zoom. To change opacity, right-click the Geometry node in the right-side Visibility tree and select **Medium** for the **Opacity** level of all geometry elements, as shown in Figure 5.1: Port-injected engine geometry with valves and ports defined (p. 43).

Figure 5.1: Port-injected engine geometry with valves and ports defined

5.2.1.2. Subvolume Creation

Subvolumes are useful to refine regions of interest, such as the chamber. The subvolume can then be used in the mesh controls to control the size of the mesh in the selected subvolume. On the Workflow tree, use Geometry > Sub-Volume to create a subvolume named **Sub-Volume-Cylinder**. Select the following surfaces to define the subvolume: **Head**, **Liner**, and **Piston**. For the Material point, accept the default **Reference Frame** using the **Global Origin**. Set the **Location** using **Cartesian Coord. System** to **X = 0, Y = 0 and Z = 1.0 cm**. Click **Apply**.

5.2.1.3. Automatic Mesh Generation Setup

The Material Point is the point in the domain that tells ANSYS Forte where the mesh will be generated and should be located at least one unit cell length away from any boundaries. This point must be inside of the domain throughout the entire simulation. Typically it is located near the head so it is still inside the domain at TDC.

- Mesh Controls > Material Point: Accept the default **Reference Frame** using the **Global Origin**. Set the Location using Cartesian Coord. **System** to **X = 0, Y = 0 and Z = 1.0 cm**. **Apply**.
- From the Workflow tree, use Mesh Controls > Global Mesh Size to set the global **Mesh Size** to **0.2 cm**.






Note

This is the recommended mesh size; a coarser setting of 0.3 cm could be used for tutorial purposes to produce a shorter runtime.


- Set the **Small Feature Deactivation Factor** to 0.5.

Next, you will refine the mesh around key geometric features such as valves and the piston, walls, and open boundaries ("continuative outflows"). The following steps show how the mesh is refined for key geometric features. From the Mesh Controls node, you can add **Point**, **Surface**, **Line**, or **Feature Refinements**, or **Small Feature Avoidance Controls**. We recommend the following general practices for setting mesh refinement in a full-cycle 4-stroke engine simulation that includes ports and valves:


1. **Refinement at Wall Boundaries.** Use $\frac{1}{2}$ the global mesh size where the mesh intersects with the wall boundaries. An extended layer of refinement is added by default to ensure that the refinement includes a full layer when the Cartesian mesh intersects curved or slanted boundaries.


- **Surface Refinement:** Click the Mesh Controls node in the Workflow tree. In the icon bar, click the **New Surface Refinement Depth**  icon and name the new control **AllWalls**. (This indicates that along the selected surfaces, cells of this smaller size and greater refinement will be used.) A new AllWalls item appears under Mesh Controls in the Workflow tree. In the panel that appears, select all of the items in the Location list, except be sure to exclude the Pressure_In and Pressure_Out open-boundary surfaces (we will set up refinement along these surfaces separately). Be sure the list's scroll bar is showing to ensure all the list items are displayed. Accept the default mesh **Size as Fraction of Global Size** as $\frac{1}{2}$. Set the **Number of Cell Layers** to **Extend Refinement from Surface** to **1**. Keep the default of **Active to Always** in the pull-down list below the refinement-layer entry. Click **Apply**. An initial refinement is applied over all wall boundaries with a $\frac{1}{2}$ **Size Fraction of Global Size**.
2. **Refinement at Open Boundaries (Continuative Outflows).** Again, we will use $\frac{1}{2}$ the global mesh size at continuative outflows, but use 2 extended layers of refinement, to make sure the continuative outflows have smooth refinement where the inlet intersects walls.
- In the Mesh Controls icon bar, click the **New Surface Refinement Depth**  icon and name the new control **OpenBoundaries**. In the panel that appears, select all of the open-boundary surfaces (the **Pressure_In** and **Pressure_Out** boundary surfaces) from the **Location** list. Accept the default mesh **Size as Fraction of Global Size** as $\frac{1}{2}$. Set the **Number of Cell Layers** to **Extend Refinement from Surface** to **2**. Click **Apply**. An initial refinement control is applied over all wall boundaries with a $\frac{1}{2}$ **Size Fraction of Global Size**.
3. **Refinement Around the Valve Stems.** Here we want to make sure we have smaller cells in the regions around the valve stems, where velocities will be greatest during intake and exhaust portions of the cycle.
- Click the **New Surface Refinement Depth**  icon (on the Mesh Controls icon bar) and name the new item **ValveStems**. Select **IntakeValve1_Stem**, **IntakeValve2_Stem**, and **ExhaustValve1_Stem**, and **ExhasutValve2_Stem** in the Location list. Set the **Size as Fraction of Global Size** to $\frac{1}{2}$ and set the **Number of Cell Layers** to **Extend Refinement from Surface** to **10**. **Make this Active = Always**.
4. **Refinement Around the Valve Ends that Seat to the Port.** As above, we want to assure sufficient refinement where velocities will be greatest during valve opening and closing.
- Click the **New Surface Refinement Depth**  icon and name the new item **ValveSeats**. Select **Exhaust-Valve1_Seat**, **ExhaustValve2_Seat**, **IntakeValve1_Seat** and **IntakeValve2_Seat** in the Location list. Set **Size as Fraction of Global Size** to $\frac{1}{4}$ and set the **Number of Cell Layers** to **Extend Refinement from Surface** to **1**. **Make this Active = Always**.
5. **Refinement Near Piston and Head at TDC positions (Squish regions):** The squish regions require refinement near TDC at the end of compression strokes to make sure that there are at least a couple of cells separating the head and piston surfaces in all locations. These controls are dynamic controls, since they are only necessary near the TDC position of the crank.
- Click the **New Surface Refinement Depth**  icon and name the new item **Squish1**. Select **Head**, **ExhaustValve1_Seat**, **ExhaustValve2_Seat**, **IntakeValve1_Seat**, **IntakeValve2_Seat**, **Liner** and **Piston** from the **Location** list. Change **Size as Fraction of Global Size** to $\frac{1}{4}$. Set **Number of Cell Layers** to **Extend Refinement from Surface** to **1**. Make this **Active = During Crank Angle Interval** and **Start angle = 340.0 degrees** and **End angle = 380.0 degrees**. Click **Apply**.
 - Use the **Copy** and **Paste** icons in the icon bar of the **Squish1** panel and name the new item **Squish2**. Make this **Active = During Crank Angle Interval** and set **Start angle = 690.0 degrees** and **End angle = 790.0 degrees**. Click **Apply**.

6. **Add a Point Refinement Depth for the region of the spark:** Capture the chemistry around the spark.

- Click the **Point Refinement Depth**  icon and name it **SparkRegion**. Set the Location as: **X = -0.36 cm, Y = 0 cm, Z = 0.33 cm**, with the **Radius of Application = 0.5 cm** and the **Refinement level = 1/4**. The refinement is **Active** between **Crank Angle 680 and 720 degrees**. Click **Apply**.


7. **Adaptive Refinements:** Adaptively refine based on the solution variables temperature and velocity.

- Click the **Solution Adaptive Meshing**  icon and name the new control **SAM-Temperature**. Set the **Quantity Type = Gradient of Solution Field** and **Solution Variables = Temperature**. **Bounds Option = Statistical** and **Sigma Threshold = 0.5**. Set the **Size as fraction of Global Size = 1/4**. The refinement is **Active** between **Crank Angle 680 and 800 degrees**, and the **Location** option is **Sub-Volumes = Sub-volume-Cylinder**. Click **Apply**.

- Click the **Solution Adaptive Meshing**  icon and name the new control **SAM-Velocity**. Set the **Quantity Type = Gradient of solution field** and **solution variable = VelocityMagnitude**. **Bounds Option = Statistical** and **Sigma Threshold = 0.5**. Set the **Refinement level = 1/2**. The refinement is **Active = Always**, and the **Location** option is **Entire Domain**. Click **Apply**.

5.2.1.4. Models Setup

Chemistry: Now that the mesh has been set up, assigning models is next. Assign the chemistry with


Models > Chemistry and use the **Import Chemistry**  icon and select the file **Gasoline_1comp_59sp.cks** from the ANSYS Forte data directory. If you are curious, you can view the chemistry details, such as chemistry source, pre-processing log, gas phase input, gas phase output, thermodynamic input, transport input and transport output.

- Flame Speed Model:** Use Models > Chemistry > Flame Speed Model to access the settings for the Flame Speed model. Keep all the defaults, except under **Turbulent Flame Speed** (at the bottom of the Editor panel), set **Turbulent Flame Speed Ratio (b1)** to **2.0**. Click **Apply**.

Transport: The default RNG k- ϵ model turbulence settings are used in this tutorial. Those are specified in the Editor panel for Models > Transport > Turbulence. The default fluid properties are also used, which are at Models > Transport.

Spark Ignition: Turn on the spark ignition model by checking the box at Models > Spark Ignition.

The spark ignition model defaults include a Kernel Flame to G-Equation Switch Constant of 2.0 and the Flame Development Coefficient of 0.5.

Use the **New Spark**  icon to create a new spark event and name it **Spark**.

Use Models > Spark Ignition > Spark to set up the details of the spark event. In the **Reference Frame**, use **Global Origin** and **Cartesian** coordinates for the **Location**, and set **X = -0.36 cm, Y = 1.0E-6 cm, and Z = 0.33 cm**. Select **Crank Angle** for **Timing** and **Starting Angle = 688.0 degrees, Duration = 10.0 degrees**. Under **Spark Energy**, set **Energy Release Rate = 20.0 J/sec**. Accept the default **0.5** for **Energy Transfer Efficiency** and **0.5 mm** for **Initial Kernel Radius**. Click **Apply**.

5.2.1.5. Boundary Conditions:

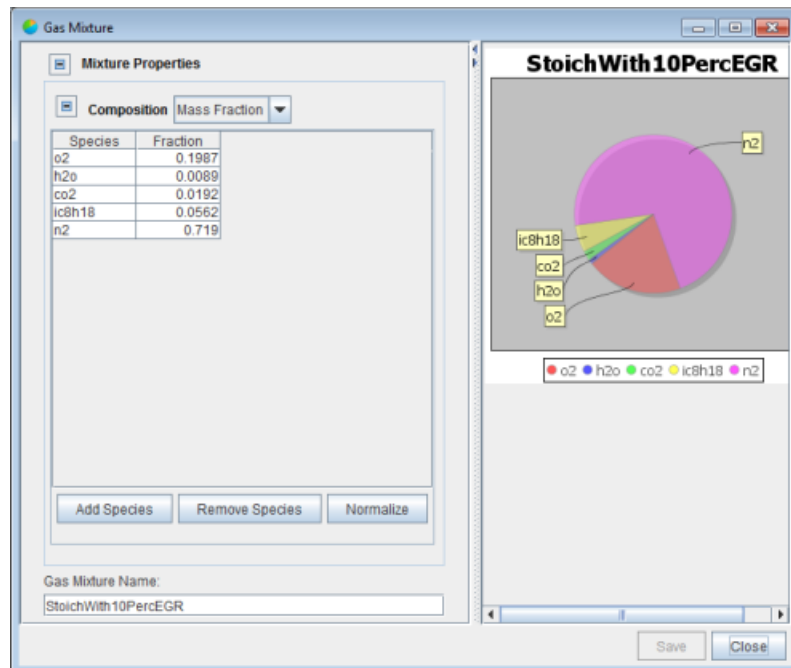
Boundary conditions are specified for each of the geometry elements in Boundary Conditions > *geometry-element-name*, where *geometry-element-name* is each of the items under Geometry in the Workflow tree.

Inlet: From the Boundary Conditions node, click the **New Inlet** icon and create Inlet. Select **Pressure_In** from the Location list. Select **Pressure Inlet** as the **Inlet Option** with a constant pressure of **0.8 bar**. You will set the inlet composition to the correct **Mass Fraction** values corresponding to the premixed fuel and air:

- **o2=0.1967**
- **h2o=0.0089**
- **co2=0.0192**
- **ic8h18=0.0562**
- **n2=0.7190**


To do this, select **Create New** in the Composition dropdown list above the **Location** list and click the **Pencil** icon. This opens the Gas Mixture panel, where you select **Add Species** and then choose the species you want from the mechanism (see [Figure 5.2: Gas Mixture Editor: Defining Inlet gas composition \(p. 46\)](#)). You can find a species more quickly by typing (the beginning of) the species name to filter the list. Save the inlet gas mixture as StoichWith10PercEGR. You will then see a list to define turbulence using **Turbulent Kinetic Energy and Length Scale** of **7,900 cm²/sec²** and **0.5 cm**, respectively.

Figure 5.2: Gas Mixture Editor: Defining Inlet gas composition




Outlet: From the Boundary Conditions node, click the **New Outlet** icon and create **Outlet**. Select the **Pressure_Out** geometry item and select the **Total Pressure** as the **Outlet Option**. Give the outlet a

Pressure of **0.9 bar**, which arises from the back-pressure of the exhaust system with an **Offset Distance to Apply Pressure** of **0.1 cm**. Set the Turbulence Boundary Conditions to **Turbulent Kinetic Energy** and **Length Scale** of **7,900 cm²/sec²** and **0.5 cm**, respectively.


Piston: From the Boundary Conditions node, click the **New Wall**  icon and name it **Piston**. Select the **Piston** item in the Location list. Set the **Temperature Option** to **Constant** and **420 K**. Turn **ON** the **Wall Motion** option and set the piston **Motion Type** to use a **Slider-Crank Model** with a **Stroke** of **7.95 cm** and a **Connecting Rod Length** of **13.81 cm** with **0.0 Piston Offset**. Change the **Movement Type** to **Moving Surface** and accept the default **Global Origin Reference Frame**.


Intake: Click the **New Wall**  icon and name it **Intake**. Select the IntakeManifold item in the Location list and set the **Temperature** to **300 K**.

Exhaust: Click the **New Wall**  icon and name it **Exhaust**. Select the **ExhaustManifold** item in the Location list and set the **Temperature** to **650 K**.

Liner: Click the **New Wall**  icon and name it **Liner**. Select the **Liner** item in the Location list and set the **Temperature** to **385 K**.


Head: Click the **New Wall**  icon and name it **Head**. Select the **Head** item in the Location list and set the **Temperature** to **385 K**.

Intake Valves: Click the **New Wall**  icon and name it **IntakeValve 1**. The valve specification is a little more complicated than the stationary walls, such that several steps are required within the wall panel. These steps specify the wall motion and the way we want the mesh to adapt to the gap opening near the valve seat when the valve opens or closes.

- Multiselect both the **IntakeValve1_Seat** and **IntakeValve1_Stem** items in the Location list.
- Check the Heat Transfer option and set the **Temperature** to the constant value of **385 K**.
- Turn ON (check) the **Wall Motion** and set the **Motion Type** to **Offset Table**.
- Accept the default **Global Origin** for the **Reference Frame**. Select **Spherical** for the Coord. **System** under **Direction** for the valve motion and set **Q=199 degrees** and **f=0.0 degrees**.
- To import the lift profile (named **IntakeLift.csv** in the same location as the **.stl** file you started this tutorial with), select **Create New** from the **Lift Profile** drop-down list and click the **Pencil**  icon. In the Profile Editor, click the **Load CSV** button and navigate to the **IntakeLift.csv** file. Accept the defaults in the import dialog and click **OK**. At the bottom of the Profile Editor window, name this **IntakeValves**. Ensure that the units in the first column are set to **Angle** and the second column is set to **cm**. **Save** the Profile.
- For Movement Type, change the pull-down menu to **Valve**. Then select just the surface boundary portion of the valve that comes into contact with the valve seat on the port, as well as the surface boundary that contains the seat region. In this case, select both the **IntakeManifold** and the **IntakeValve1_Seat**.
- Finally, set the **Valve Motion Activation Threshold** to **0.15 cm**. This indicates that the valve will not open until the lift value specified in the valve-lift profile exceeds this threshold. A smaller value will cause the valves to open sooner, but the mesh refinement required to resolve the gap will be higher. This is a trade-off that will need to be determined based on the goals and outcomes desired from the simulation. In addition

to the activation threshold, you may also change the minimum number of cells desired at the smallest gap opening. In this case, accept the default value for **Approx. Cells in Gap at Min. Lift** of **3.0**.

Follow the same procedure to set the boundary conditions for **IntakeValve 2**. This time you can select the existing **IntakeValves** profile instead of creating a new one. And for **IntakeValve 2**, the Location selected should be **IntakeValve2_Seat** and **IntakeValve2_Stem**, while the surfaces selected under **Multiselect the Valve Seat and Surface it Contacts** should be the IntakeManifold and IntakeValve2_Seat.

Exhaust Valves: Click the **New Wall**  icon and name it **ExhaustValve1**. Follow a similar procedure as for the Intake Valves. This time, select the **ExhaustValve1_Seat** and **ExhaustValve1_Stem** items in the Location list and set the **Temperature** to **497 K**. Turn ON the **Wall Motion** and set the **Motion Type** to **Offset Table** and the **Vertices to Transform** to **Interior**. Select **Global Origin** for the **Reference Frame**. Select **Spherical** for the **Coord. System** under **Direction** and set **Q = -199 degrees** and **f = 0.0 degrees** (note that this is the opposite direction from the intake valves, relative to the z-axis). Similarly to the intake valves, first import the **.csv** (named **ExhaustLift.csv**) file, accept the defaults, and name it **ExhaustValves**. Then follow the same procedures for specifying the **Movement Type** to **Valve**, selecting the **Valve Seat and Surface it Contacts** (**ExhaustManifold and ExhaustValve1_Seat**), and setting the **Valve Motion Activation Threshold** to **0.15 cm**.

Follow the same procedure to set the boundary conditions for ExhaustValve2. Use the **ExhaustValves** profile again for ExhaustValve2, but select the appropriate surfaces for the **ExhaustValve2** in the Location list and in the **Multiselect the Valve Seat and the Surface it Contacts** list.

5.2.1.6. Initialization


The domain is initialized with the operating conditions, species concentrations and temperatures. The Default Initialization species composition is at the expected exhaust composition, assuming complete combustion. The intake and exhaust must also be initialized to the boundary condition values. Set the following initialization parameters:

Default Initialization:

- Select Default Initialization in the Workflow tree. Set the **Initialization Order** to **2**. Since flow goes generally from the intake to the cylinder and then to the exhaust, this indicates that this default (cylinder) region will be the second in the initialization order precedence list.
- Select **Create New** in the **Composition** drop-down list and click the **Pencil** icon. This opens the Gas Mixture panel, where you select **Add Species**, set to a **Mass Fraction** of **n2=0.7192**, **co2=0.1923** and **h2o=0.0885** and **Save** this composition as **ExhaustEst_w10percEGR**.
- Set a **Temperature** of **1,000 K** and the **Pressure** to **1.0 bar**.
- The **Turbulence** initialization uses the **Turbulent Kinetic Energy and Length Scale** option with values **7,900 cm²/sec²** and **0.4 cm**, respectively.
- The **Velocity** is initialized using **Engine Swirl**, with an **Initial Swirl Ratio** of **-0.0739**, an **Initial Swirl Profile Factor** of **3.11**, and checking (ON) **Initialize Velocity Components Normal to Piston**.
- Click **Apply**.

Intake Initialization: The intake manifold Initial Condition is set to match the Boundary Condition at the Inlet. Since this is a separate port that can be closed off from the main cylinder region, we also need to set the equivalent of a material point to identify the region, as well as an initialization order

that helps determine what region takes precedence in initializing new cells that appear when gaps are opened.

- From the Initial Conditions Workflow tree item, select the **New Port Initialization**  icon and name it **Intake**.
- To identify the region, select a point for the **Location** under the **Reference Frame** selection, which is a point that will always be within the **Intake** port. Set the coordinates for this case to **X=6.0, Y=2.0, Z=5.0 cm**, which is a point just inside the inlet.
- Set the **Initialization Order** to **1**. Flow is expected to go from the intake to the main region; for this reason, we give it the first order in initialization precedence.
- Set the **Composition** by selecting in the previously saved profile, **StoichWith10PercEGR**. Set the **Temperature** to **300 K** and **Pressure** to **0.8 bar**.
- The **Turbulence** initialization uses **Turbulent Kinetic Energy and Length Scale** option set to **7,900 cm²/sec²** and **0.5 cm**, respectively. Click **Apply**.

Exhaust Initialization: The Initial Conditions of the Exhaust is set to match the Boundary Condition of the Outlet.

- From the **Initial Conditions** Workflow tree item, select the **New Port Initialization** icon and name it **Exhaust**.
- To identify the region, select a point for the **Location** under the **Reference Frame** selection, which is a point that will always be within the **ExhaustPort** region. Set the coordinates for this case to **X= -4.0, Y=2.0, Z=2.0 cm**, which is a point just inside the outlet.
- Set the Initialization Order to **3**. Flow is expected to go from the cylinder to the exhaust port; for this reason, we give it the last order in initialization precedence for the 3 regions defined.
- Set the **Composition** to the existing profile, **ExhaustEst_w10percEGR**. Set the **Temperature** to **650K** and **Pressure** to **0.9 bar**.
- The **Turbulence** initialization uses the **Turbulent Kinetic Energy and Length Scale** option; these are set to **7,900 cm²/sec²** and **0.5cm**, respectively. Click **Apply**.

5.2.1.7. Simulation Controls

Simulation controls allow you to define the simulation limits, time step, chemistry solver and transport terms.

Simulation Limits: Use the Simulation Controls panel to select a **Crank Angle**-based simulation from a CA of **340 to 900** degrees. Set **RPM = 1,500 rpm**. The engine **Cycle Type** is **4-Stroke**.

Time Step: Use Simulation Controls > Time Step and set the following parameters:

- **Initial Simulation Time Step** to **5.0E-7 seconds**
- Select **Restrict Time Step by Crank Angle**
- **Max. Crank Angle Delta Per Time Step** of **1.1 degrees**
- Set the **Max. Time Step Option** to **Constant** and set the value of **Max. Simulation Time Step** to **3.E-5 sec**. The time step will be adaptively determined throughout the simulation, based on local solution gradients, so this just sets the maximum value allowed.

The **Advanced Time Step Control Options** settings are kept at the defaults:

- **Time Step Growth Factor = 1.3**
- **Fluid Acceleration Factor = 0.5**
- **Rate of Strain Factor = 0.6**
- **Convection factor = 0.2**
- **Internal Energy Factor = 1.0**
- **Max. Convection Subcycles = 8**

Chemistry Solver: Simulation Controls > Chemistry Solver is also kept at the defaults, with the following parameters:

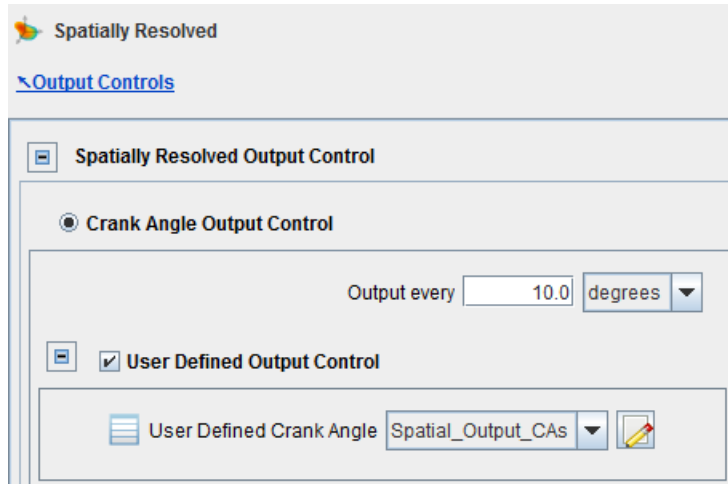
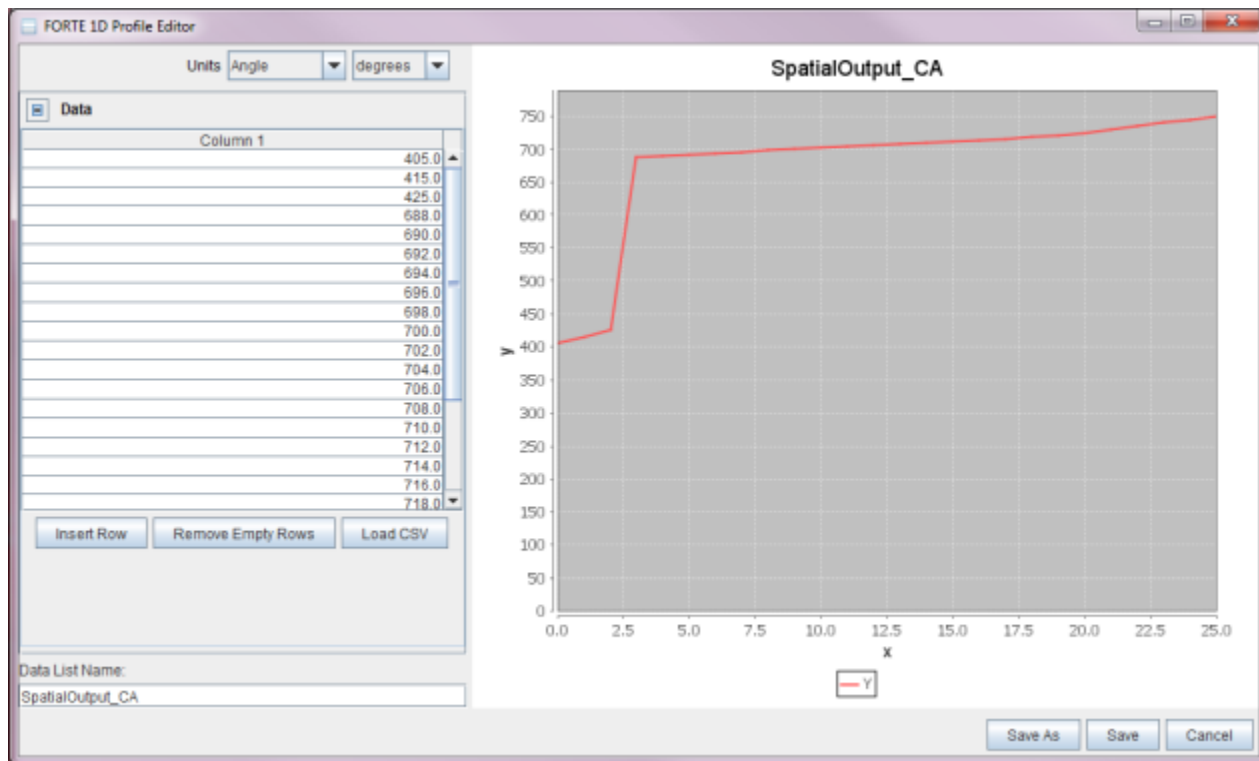
- **Absolute Tolerance = 1.0E-12**
- **Relative Tolerance = 1.0E-5**
- Use **Dynamic Cell Clustering** to take advantage of groups of cells with similar conditions. Select **2 features** to introduce **Dynamic Cell Clustering**: 1) **Max. Temperature Dispersion** of **10 K** and 2) a **Max. Equilibrium Ratio Dispersion** of **0.05**.
- To increase the time-to-solution speed, you have the ability to choose when chemistry is activated. In this tutorial, select **Activate Chemistry Conditionally**, and select **When Temperature is Reached** with **Threshold Temperature 600 K** and also select **During Crank Angle Interval** between **650** and **850** crank angle. This ensures that chemistry is active during the time that combustion is expected even if temperature does not rise above **600 K**. Click **Apply**.

Transport Terms: Use the default transport tolerances and maximum number of iterations.

5.2.1.8. Output Controls

Output controls determine what data are stored for viewing during the simulation and for creating plots, graphs and animations in ANSYS Forte Visualize.

Spatially Resolved: Allows you to control when spatially resolved data such as velocity, temperature, species concentrations, etc., will be output. In the **Spatially Resolved** panel, set the **Crank Angle Output Control** to report every **10 degrees** (to manage the size of the output file). You can optionally increase the frequency of output during the cycle by selecting **User Defined Output Control** and importing the **Spatial_Output_CAs.csv** file, which has a list of specific crank angles where spatially resolved output will occur (illustrated in [Figure 5.3: Output control panel - Spatially Resolved. \(p. 51\)](#)). Alternatively, you could select **User Defined Output Control**, and use the Profile Editor to create some other file specifying an output crank-angle profile (the provided profile is illustrated in [Figure 5.4: User-defined spatially resolved output controls. \(p. 51\)](#)). Select the following species for **Spatially Resolved Output**: **h2o, no, no2, co, co2, o2, ic8h18 and n2**.



Figure 5.3: Output control panel - Spatially Resolved.**Figure 5.4: User-defined spatially resolved output controls.**

Spatially Averaged: Allows you to control the output of values that are averaged across the domain. **Spatially Averaged** panels, set the **Crank Angle Output Control** to reporting every **1 degree**. Select the following species for Spatially Averaged Output: h2o, no, no2, co, co2, o2, ic8h18 and n2.

Restart Data: If you anticipate that the case will be stopped and you want the ability to restart it from the last time step solved, select Output Controls > Restart Data. You can specify certain Restart Points using a separate file. Turn on (check) **User Defined Restart Points** and use the Profile Editor to create a Restart profile. You can view, edit or import new Restart Points in this 1-D Profile Editor. Sometimes it is helpful in spark ignition cases to save a restart file after IVC but before the spark occurs (CA=687 in this case) so you can use the compression portion of the cycle as a start point in additional runs. Create a new profile for this purpose called **RestartOutput** and add one line with CA value set to 650.

5.2.1.9. Preview Simulation

You can view the profiles for the Piston, Intake Valves, Exhaust Valves and Mesh Refinement during the simulation by selecting Preview Simulation > Boundary Motion > Simulation Preview. This is an excellent way to check whether you have valve overlap and that the settings are correct.

As a method for checking the automatically generated mesh, you can generate a Preview Mesh. Select Preview Simulation > Mesh Generation and then click the **New Automatic Mesh Plot**  icon, name this new automatic mesh generation plot "FTDC", select **Crank Angle** as the **Time Option** and set it to **720 CA**. Then click **Apply** to save the settings and then on the **Generate Mesh**  icon. ANSYS Forte will generate the preview mesh and display it in the 3-D View window. It is a good practice to look at meshes at key points in the cycle such as Firing Top Dead Center (FTDC), Exhaust Valve Opening (EVO), Intake Valve Closed (IVC), Intake Valve Open (IVO) and Exhaust Valve Closed (EVC). If you want to see the cut plane where the mesh will be generated, click the Plane Filter box and specify an origin point and normal direction for the cut plane.

Use the information in the following list to set up these **New Automatic Mesh Plots**.

1. FTDC @ CA=720: **Point (X=0.0cm, Y=0.0cm, Z=0.0cm); Normal (X=0.0cm, Y=1.0cm, Z=0.0cm)**
2. EVO @ CA=192: **Point (X=0.0cm, Y=2.0cm, Z=0.0cm); Normal (X=0.0cm, Y=1.0cm, Z=0.0cm)**
3. IVC @ CA=601: **Point (X=0.0cm, Y=-2.0cm, Z=0.0cm); Normal (X=0.0cm, Y=1.0cm, Z=0.0cm)**
4. IVO @ CA=363: **Point (X=0.0cm, Y=-1.0cm, Z=0.0cm); Normal (X=0.0cm, Y=1.0cm, Z=0.0cm)**

The visibility of the **Automatic Mesh Plots** is controlled in the Visibility tree under Preview Simulation and Mesh Generation.

5.2.1.10. Run Simulation

To complete the lesson, select Run Simulation on the Workflow tree and, once ANSYS Forte displays the green **START** button on the Run Simulation panel and reports a "Ready" status, click **Start**. You can monitor the results by clicking on the **Monitor Runs**  icon. In the Monitor window that opens, you can select the run you wish to monitor.

5.2.1.11. Run Settings

The settings here depend on the system and environment for your simulations. The default for the Run Settings panel is to have nothing selected.

Run Options: This tutorial does not require changes to this panel's defaults; adjust them as necessary for your environment.

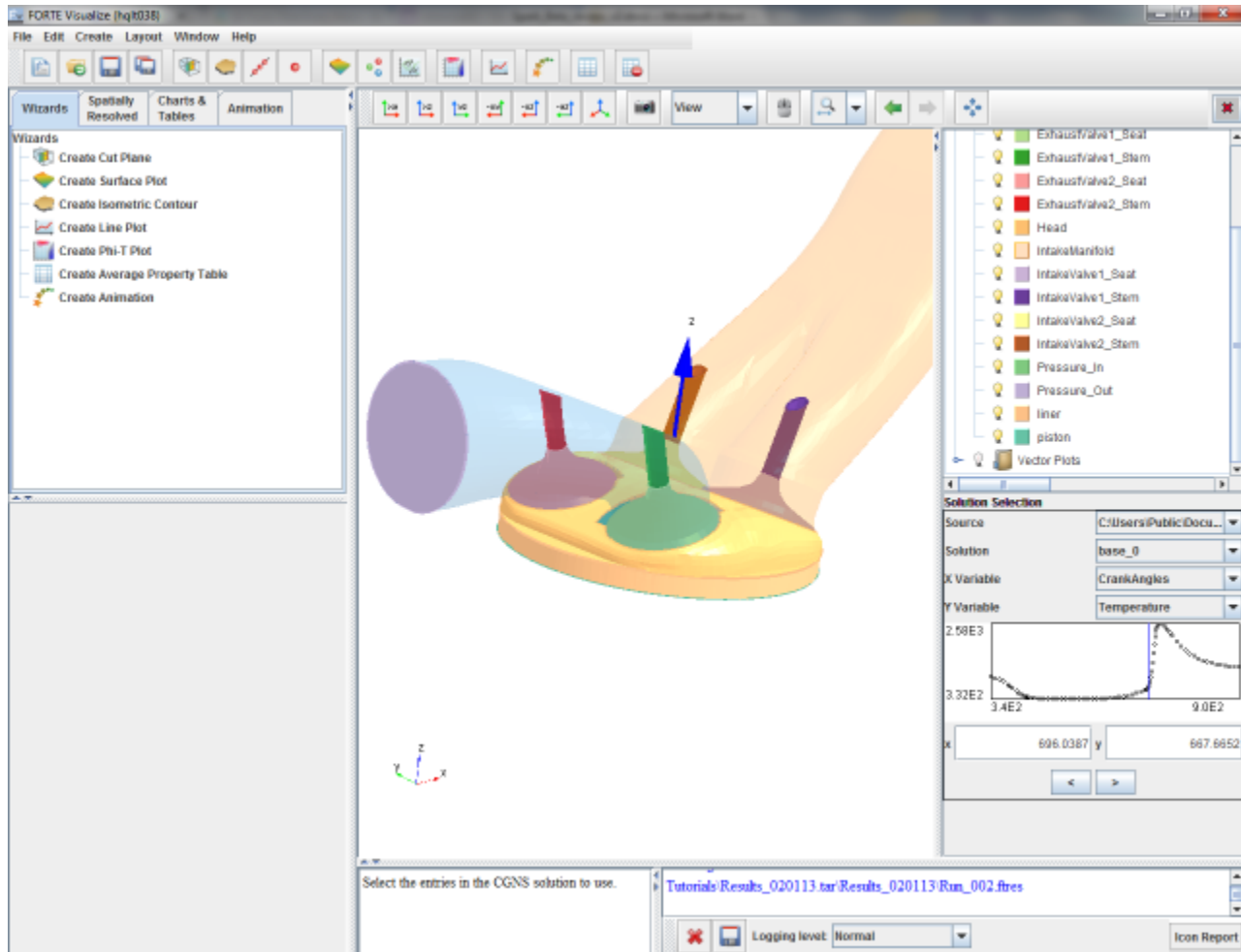
Windows Settings: This tutorial does not require changes to this panel's defaults; adjust them as necessary for your environment.

Linux Settings: This tutorial does not require changes to this panel's defaults; adjust them as necessary for your environment.

5.2.2. Results

To view the results of the simulation, open the Visualizer from the ANSYS Forte Launcher. In Visualize, open the solution file for the case (**Nominal.ftres**). Figure 5.5: Screen view of the solution file in Visualize. (p. 53) shows the screen view once the solution file has been loaded. Use **Edit > Edit Preferences** to set the units for pressure to **bar**.

Figure 5.5: Screen view of the solution file in Visualize.



Line plots can easily be created using the Line Plot Wizard. We will use the Line Plot Wizard and select the **X** and **Y Variables** to **Crank Angle** and **Net HRR**, respectively. We will add the **Pressure** after the initial line plot is created. Make sure that **Average** is selected at the top of the dialog box. On the lower left you will see the drop-down menu for the **Y Variable**. Multi-select (control-click) the **Pressure Variable** and it will display in the graph on a second Y-axis. The pressure and net heat release rate are shown in Figure 5.6: Pressure (blue line) and Net Heat Release Rate (red line). (p. 54). The NO_x and CO emissions index (g/kg-fuel) during the cycle are shown in Figure 5.7: NO_x and CO emissions index (g/kg-fuel). (p. 54). Another interesting feature is to use the Iso-Contour Wizard to track the flame front across the cylinder volume, as shown in Figure 5.8: Iso-contours at 1000 K showing the progression of the flame front from CA 694-702 degrees. (p. 55). Use a temperature value of 1000 K for this purpose.

Figure 5.6: Pressure (blue line) and Net Heat Release Rate (red line).

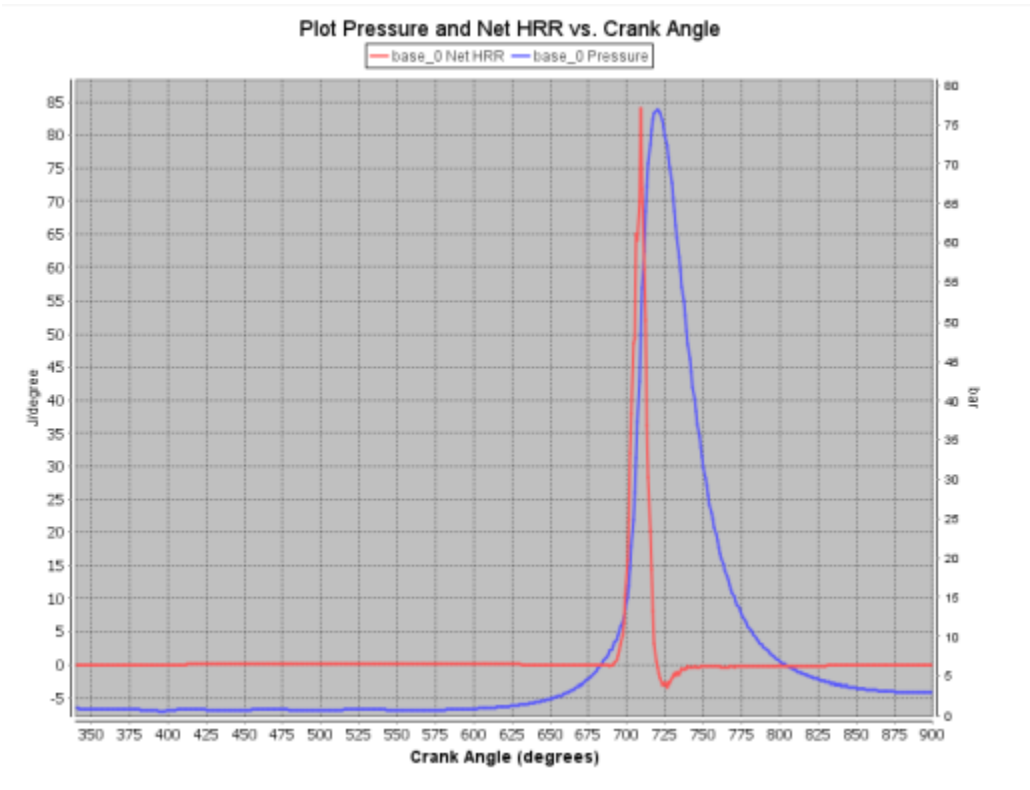


Figure 5.7: NOx and CO emissions index (g/kg-fuel).

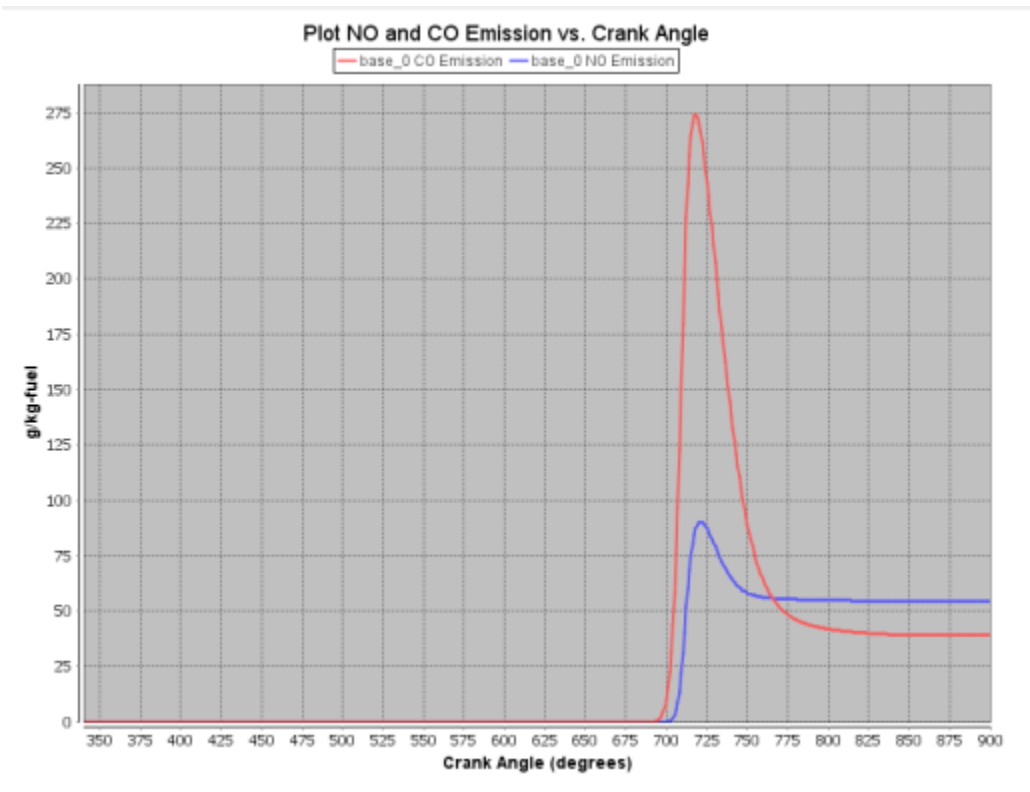
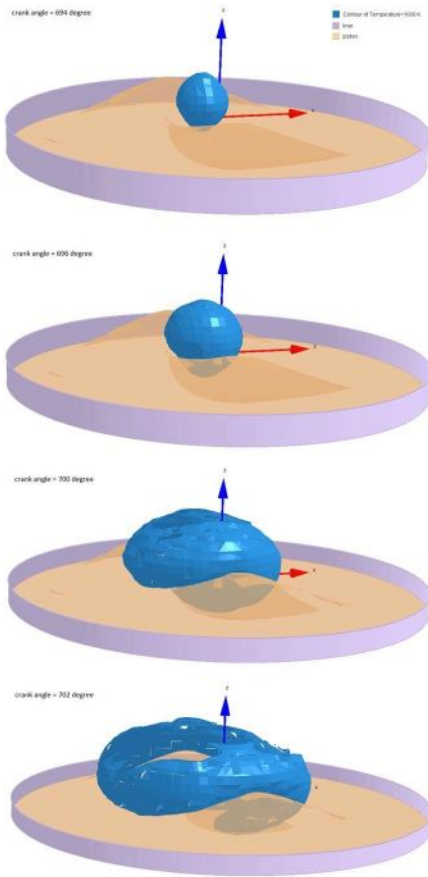


Figure 5.8: Iso-contours at 1000 K showing the progression of the flame front from CA 694-702 degrees.



Chapter 6: Tracking Soot Particles Evolution in a Diesel Engine

This tutorial presents the soot particle tracking capability in ANSYS Forte, applying the method of moments to a diesel engine case. With the soot particle tracking feature, several spatially resolved values of soot can be simulated, including particle number density, volume fraction and average diameter. The tutorial explains the setup for using the particle tracking capability, and presents visualizations of soot predictions.

6.1. Data Provided

6.1.1. Files Used in This Tutorial

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

and select the Forte tutorials you wish to download. The file for this tutorial is:

- **Soot-particle-tracking__tutorial.ftsims**: A project file of the completed tutorial, for verification or comparison of your progress in the tutorial set-up. The project file includes all the relevant geometry, chemistry set and spray profile details.

The tutorial sample file is provided as a download. You have the opportunity to select the location for the file when you download and uncompress the sample files.

Note

This tutorial is based on a fully configured sample project that contains the tutorial project settings. The description provided here covers the key points of the project set-up but is not intended to explain every parameter setting in the project. The .ftsims file has all custom and default parameters already configured; the text highlights only the significant points of the tutorial.

6.1.2. Project Comparison Utility

The Forte installation includes the `cgns_util export` command, which you can use to compare the parameter settings in the project file generated at any point during your tutorial set-up against the provided **.ftsims**, which has the parameter settings for the final, completed tutorial. This command is described in the Forte User Guide.

Briefly, you can double-check project settings by saving your project and then running the `cgns_util` to export your tutorial project, and then to export the provided final version of the tutorial. Save both versions and compare them with your preferred diff tool, such as DIFFzilla. If all the parameters are in agreement, you have set up the project successfully. If there are differences, you can go back into the tutorial set-up, re-read the tutorial instructions, and change the setting of interest.

6.1.3. Time Estimate

As a guideline for your own simulations, this tutorial is estimated to take approximately 6.5 hours on an Intel® processor E5-2690 at 2.90 GHz (8 total cores).

6.1.4. Prerequisites for This Tutorial

We recommend starting with the Forte Quick Start Guide , which explains the workflow of the ANSYS Forte user interface, before doing this tutorial.

6.2. Project Setup

The ***Soot-particle-tracking__tutorial.ftsim*** project file has been preconfigured with all the information that will be discussed in this section. You do not need to input any values but can just follow along, reading the instructions and viewing the settings in the loaded ***.ftsim***. This chapter nevertheless explains step-by-step the process of setting up the project, as an illustration of the features in the user interface.

Open the project file, ***Soot-particle-tracking__tutorial.ftsim***, from the location where you stored the downloaded tutorial files.

6.2.1. Sector Mesh Details

A simple engine configuration is used in this tutorial for demonstration purposes. The details of the configuration and simulation settings are presented in [Table 6.1: Details of the diesel engine geometry used in this tutorial \(p. 58\)](#).

Table 6.1: Details of the diesel engine geometry used in this tutorial

Compression ratio	15
Displacement volume (cm ³)	785
Sector angle	45
Bore (cm)	10
Stroke (cm)	10
Squish (cm)	0.1
Flat piston bowl depth, diameter (cm)	1.5, 6
Crevice width and height (cm)	0.1, 1.8

The bowl profile used in the Sector Mesh Generator is described in [Table 6.2: Details of the bowl profile in the ANSYS Forte profile editor \(p. 58\)](#).

Table 6.2: Details of the bowl profile in the ANSYS Forte profile editor

Column 1 Distance (cm)	Column 2 Distance (cm)
3	0
3	-1.5
0	-1.5

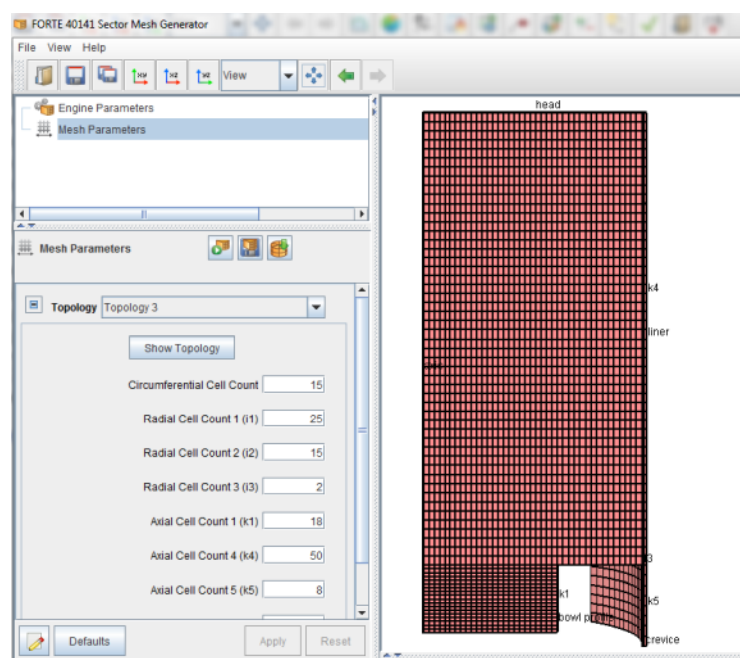
The 45° sector mesh was created using the Sector Mesh Generator in ANSYS Forte. The values described in [Table 6.1: Details of the diesel engine geometry used in this tutorial \(p. 58\)](#) can be accessed in the

Soot-particle-tracking_tutorial.ftsim project file, by launching the Sector Mesh Generator, as shown in Figure 6.1: Sector Mesh Generator settings for this tutorial. (p. 59). In the Sector Mesh Generator, Topology 3 was deemed the most appropriate to represent this piston bowl geometry. The mesh parameters for this topology were selected as follows:

1. ~1.2 mm resolution along the radial direction.
2. 3° along the azimuthal direction.
3. Along the z-direction:
 - 0.8 mm in the piston bowl, to capture spray and events close to TDC accurately.
 - 2 mm resolution in the squish region. Four minimum cells will be present, as specified in the Mesh Controls Editor panel, to ensure resolution close to TDC. Away from TDC, the mesh is set to be coarse in the squish region, for the sake of faster simulation in this simplistic demonstration case.

These settings resulted in a sector mesh with 38,490 cells when the piston is at bottom dead center (BDC).

Figure 6.1: Sector Mesh Generator settings for this tutorial.



In the Mesh Controls > Remeshing Editor panel, the **Smooth** check box is unchecked. All other controls are left at default values.

6.2.2. Chemistry Set Details

On the Models > Chemistry Editor panel, the ***Diesel_3comp_243sp__soot-particle-tracking.cks*** chemistry set has been selected.

Note

This chemistry set requires a special license feature; please contact your Account Representative if you are not able to load the chemistry set and you are interested in using the Particle Tracking option in ANSYS Forte.

This chemistry set includes a gas-phase mechanism (***Diesel_3comp_243sp__soot-particle-tracking_chem.inp***), a surface mechanism file (***surf_soot_chem.inp***) and a thermodynamic data file (***Diesel_3comp_243sp__soot-particle-tracking_therm.dat***).

The gas-phase mechanism contains 243 species participating in 1756 reactions. This mechanism has been generated for use with the 66.8/33.2 wt% n-decane/AMN diesel surrogate. It has reaction pathways to accurately predict soot precursors needed for the surface soot mechanism, all the way from small hydrocarbons such as acetylene to polycyclic aromatic species such as pyrene.

If you were to create your own chemistry set for use with particle tracking, the surface mechanisms would have to conform to certain standards, as defined in the Chemkin Input Manual and Chemkin Theory Manual. The soot surface mechanism that is part of this tutorial's chemistry set includes:

1. Soot nucleation through multiple pathways.
2. HACA- and PAH-based soot growth pathways.
3. Soot oxidation through O₂ and OH.

6.2.3. Transport Property Settings

The default values are used in the Transport property panel. In the Transport > Turbulence panel, the **RNG k-epsilon** model is used, with default values.

6.2.4. Spray Model Settings

Under Models > Spray model, the spray models, nozzle and fuel injection values have been set.

The nozzle-1 settings are provided in [Table 6.3: Nozzle settings \(p. 60\)](#).

Table 6.3: Nozzle settings

Location	
Reference frame	Global origin
Coordinate system	Cylindrical
R	0.1 cm
q	22.5 degrees
A	11.8 cm
Spray direction	

Location	
Reference frame	Global origin
Coordinate system	Spherical
q	110 degrees
f	22.5 degrees
Nozzle size	
Nozzle diameter	150 micron

The following spray models are being used in this project, with default ANSYS Forte settings:

1. KH droplet breakup model
2. RT droplet breakup model
3. Gas-jet
4. Vaporization model
5. Radius of influence droplet collision model

The diesel fuel was simulated using a 2-component surrogate consisting (¹) of 66.8 weight% *n*-decane/33.2 weight% 1-methylnaphthalene. This particular surrogate was designed to model a European diesel fuel with cetane number of 55. This surrogate has a threshold sooting index of 36, placing it on the higher end of the typical diesel fuel sooting index. [Table 6.4: Details of both nozzle \(Nozzle-1\) and fuel injection \(Injection-1\) \(p. 61\)](#) provides details of the nozzle and the fuel injection. These values are entered under the Models > Solid Injector-1 node. Specifically, the nozzle location and spray direction inputs are set on the Nozzle-1 Editor panel, and the fuel injection-related inputs are specified on the Injection-1 Editor panel.

On the Solid Injector-1 panel, parcel specification has been set using the Number of parcels option. The Injected parcel count has been set to **3000**.

The **Inflow Droplet Temperature** has been set to **320 K**.

Spray was initialized using the **Constant Discharge Coefficient** and **Angle** option. A **Constant Discharge Coefficient** of **0.7** has been used, with a **Constant Mean Cone Angle** of **15°**.

The nozzle and fuel injection specifications are provided in [Table 6.4: Details of both nozzle \(Nozzle-1\) and fuel injection \(Injection-1\) \(p. 61\)](#).

Table 6.4: Details of both nozzle (Nozzle-1) and fuel injection (Injection-1)

Nozzle details	
Direct-injector nozzle hole	150 µm
Direct-injector included angle (°)	140
Fuel injection details	
Start of injection (° ATDC)	-15
Duration of injection (°)	10

¹species names: *n*-decane is nc10h22; 1-methylnaphthalene is a2ch3.

Nozzle details	
Total Injected mass (mg)	30
Injection velocity profile	A slightly modified square profile: (0,0), (1E-7,1), (0.999999,1), (1,0)

6.2.5. Soot Model Settings

Under Models > Soot Model, choose **Soot Model**. There are three options for modeling soot in ANSYS Forte. One option is to define a “pseudo-gas” soot model that is included in the gas-phase chemistry mechanism input. In this case, you would not select Models > Soot Model. Otherwise, you may either use a built-in 2-step model, which is one option on the Soot Model panel. Another option is the Method of Moments. This option enables the particle tracking capability. In this tutorial, **Method of Moments** is selected.

In this tutorial, with the **Method of Moments** option selected in the Soot Model panel, four inputs are required. Details regarding these parameters can be found in the ANSYS Forte Theory Manual .

1. **Number of Moments:** 3-6 moments can be used in the simulations. Typically, it is recommended that 3 moments are sufficient; that value of **3** is used in this tutorial.
2. **Scaling Factor for Moments:** This value changes the units of the (internal) solution variable for particle moments. A recommended value for typical problems is the default **1.0E+12**, as the scaling helps preserve the positivity of the solution during computation.
3. **Scaling Factor for Surface Species:** This value changes the units of the (internal) solution variable for particle surface species. For example, setting it to 1.0E+12 results in pico-moles, while a value of 1 would mean that the unit should be moles. A recommended value for typical problems is the default **1.0E+12**, as the scaling helps preserve the positivity of the solution during numerical computation.
4. Coagulation is included in this tutorial, with a **Coagulation Collision Efficiency** of **1.0**. This coagulation collision efficiency term is a combined correction factor to the coalescent collision between particles. The van der Waals forces can enhance the collision frequency while non-coalescent collision can reduce the frequency.

6.2.6. Boundary Conditions

Piston: The piston temperature is set to **500 K**. The **Law of the Wall** model is used. Under **Wall motion**, the **Bore** and **Stroke** values are set to **10 cm**, and the **Connecting Rod Length** is set to **15 cm**.

Periodicity: The **Sector Angle** is set to **45°**.

Head: The **Temperature** is set to **470 K**, and the **Law of the Wall** model is used.

Liner: The **Temperature** is set to **420 K**, and the **Law of the Wall** model is used.

6.2.7. Initial Conditions

An ~20% EGR case is considered here. This results in an initial composition with a mass fraction of:

- $O_2=0.18$
- $N_2=0.76$

- $\text{CO}_2=0.04$
- $\text{H}_2\text{O}=0.02$

The initial **Temperature** and **Pressure** are set to values of **450 K** and **1.0 bar**. The initial **Turbulent Kinetic Energy** is set to **10,000 cm²/sec²**, and the **Turbulent Length Scale = 1 cm**. Under Velocity, Engine Swirl is chosen and an **Initial Swirl Ratio** of **1.0** is used. The Initial Swirl Profile factor is left at the default value of 3.11.

6.2.8. Simulation Control

The simulation Start Crank Angle (i.e., **Initial Crank Angle**, under Simulation Controls in the Workflow tree) is set to **-130 degrees ATDC**, the **Engine Speed** is specified as **2000 RPM**, and the simulation **Final Simulation Crank Angle** is set to **+130 degrees ATDC**.

6.2.9. Output Control

To allow visualizing fuel and soot precursor species, the following species have been named in the **Spatially Resolved** and **Spatially Averaged** Output Control Editor panel. Also included in the list are species for air, combustion products, and NOx.

1. **Pyrene a4**
2. **Acetylene c2h2**
3. **Benzene c6h6**
4. **n-Decane, a fuel surrogate nc10h22**
5. **AMN, a fuel surrogate a2ch3**

The spatially resolved values are output at intervals of **5 crank angles**, and the spatially averaged values every **1.0 crank angle**.

6.3. Project Results

Spatially averaged plots appear below first, followed by some spatially resolved plots.

The simulation results for the pressure and heat release rate are shown in [Figure 6.2: Calculated pressure and heat release rate curves. \(p. 64\)](#). The fuel ignites slightly after TDC.

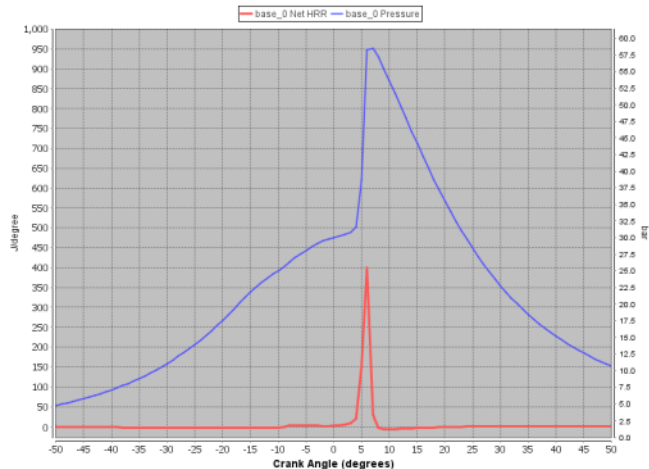
Figure 6.2: Calculated pressure and heat release rate curves.

Figure 6.3: Predicted average particle diameter as a function of crank angle. (p. 64) shows a line plot of the spatially averaged average particle diameter and total soot mass, as a function of crank angle. These values are impacted by particle coagulation as well as by soot surface chemistry, including nucleation, growth and oxidation. While the total soot mass decreases after ~10 CAD ATDC due probably to consumption of soot by oxidation, the particle diameter increases until ~30 CAD ATDC due to a combination of soot growth kinetics and particle coagulation. The average diameter peaks at around 20 nm for this case.

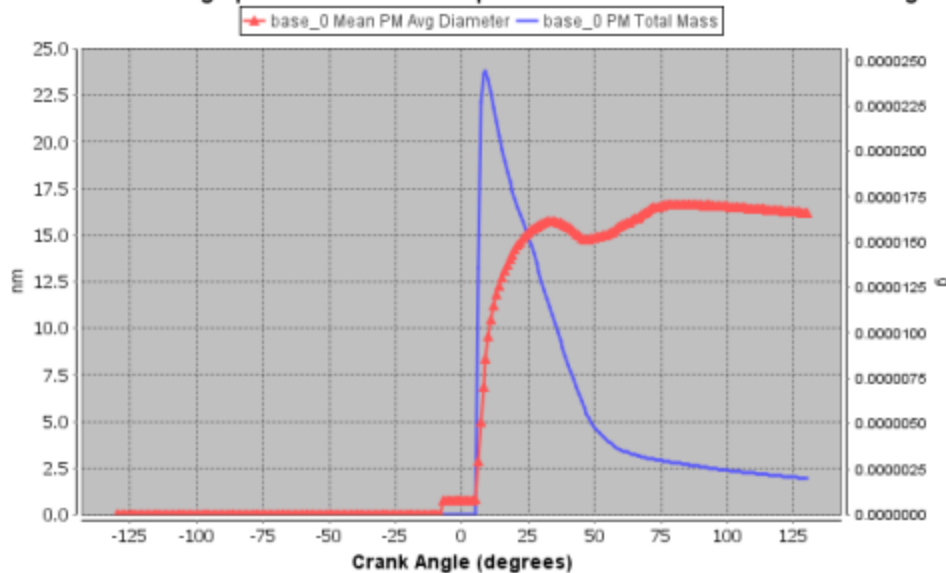
Figure 6.3: Predicted average particle diameter as a function of crank angle.**Predicted average particle diameter and particle mass as a function of crank angle**

Figure 6.4: Average particle diameter iso-surface of 20 nm, as a function of crank angles. (p. 65) uses iso-surfaces (50 nm) to show the spatially resolved values of average particle diameter, at 30-70 crank angle degrees. Of the three crank angles explored here, the 20 nm particles peak at 50 CAD, and the location of these relatively large particles is in the squish zone.

Figure 6.4: Average particle diameter iso-surface of 20 nm, as a function of crank angles.

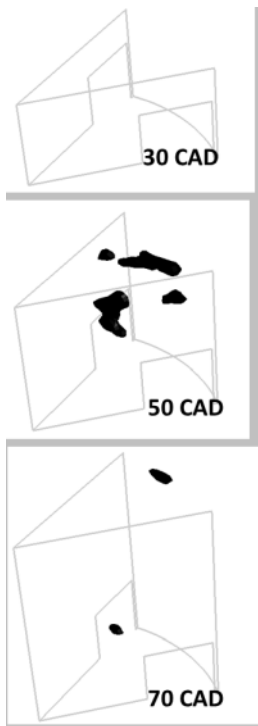
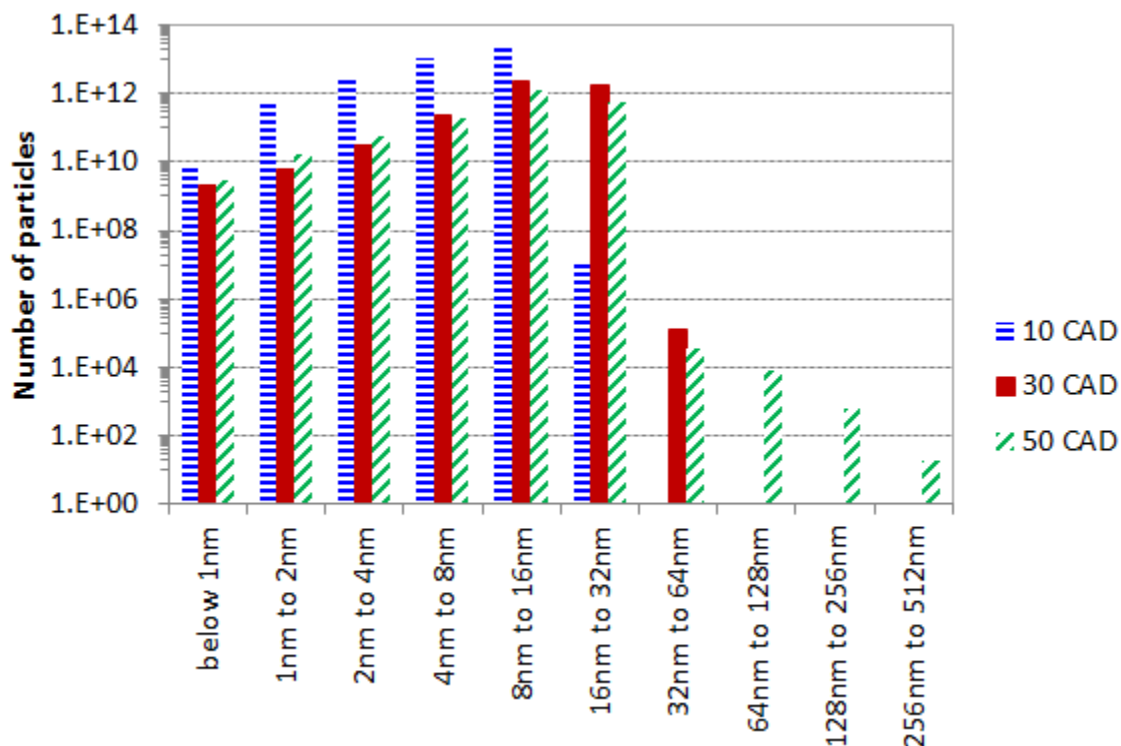


Figure 6.5: Average particle size distribution at different crank angles. (p. 65) shows the average particle size distribution. It can be seen that initially at 5-10 CAD, the particle count is high but the particle size is low with the majority of particles being smaller than 4 nm. With increasing crank angle, the lower, zoomed-in plot shows that larger diameter particles form but have lower particle count.

Figure 6.5: Average particle size distribution at different crank angles.



Chapter 7: Solving a Gasoline Direct Injection Engine Simulation

This tutorial describes how to use ANSYS Forte to simulate combustion in a direct injection spark ignition internal combustion engine with moving valves. Engine geometry is imported and ANSYS Forte's automatic mesh generation is used to create the computational mesh on-the-fly during the simulation.

7.1. Data Provided

This section describes the provided files, time required, prerequisites, and a utility for comparing your generated project file (**.ftsim**) with the one provided in the tutorial download.

7.1.1. Files Used in This Tutorial

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

and select the Forte tutorials you wish to download. The files for this tutorial are:

- **Forte_GDI_Tutorial.stl**: This is the geometry file.
- **exhaust_valve_lift.csv** and **intake_valve_lift.csv**: Two files of data describing the intake and exhaust valve lifts. Profiles such as these can be imported (from **.csv** files) or manually entered in the Profile Editor
- **spatial_output.csv**: Specifies when spatially resolved data such as velocity, temperature, species concentrations, etc., will be output.
- **Forte_GDI_Tutorial.ftsim**: A project file of the completed tutorial, for verification or comparison of your progress in the tutorial set-up.

The tutorial sample file is provided as a download. You have the opportunity to select the location for the file when you download and uncompress the sample files.

Note

This tutorial is based on a fully configured sample project that contains the tutorial project settings. The description provided here covers the key points of the project set-up but is not intended to explain every parameter setting in the project. The **.ftsim** file has all custom and default parameters already configured; the text highlights only the significant points of the tutorial.

7.1.2. Project Comparison Utility

The Forte installation includes the `cgns_util export` command, which you can use to compare the parameter settings in the project file generated at any point during your tutorial set-up against the provided **.ftsim**, which has the parameter settings for the final, completed tutorial. This command is described in the Forte User Guide.

Briefly, you can double-check project settings by saving your project and then running the `cgns_util` to export your tutorial project, and then to export the provided final version of the tutorial. Save both versions and compare them with your favorite diff tool, such as DIFFzilla. If all the parameters are in agreement, you have set up the project successfully. If there are differences, you can go back into the tutorial set-up, re-read the tutorial instructions, and change the setting of interest.

7.1.3. Time Estimate

As a guideline for your own simulations, this tutorial is estimated to take approximately 33.2 hours on a cluster with 16 nodes of dual Intel® Xenon® processors E5-2690 at 2.90 GHz (8 cores).

7.1.4. Prerequisites for This Tutorial

We recommend starting with the Forte Quick Start Guide , which explains the workflow of the ANSYS Forte user interface, before doing this tutorial.

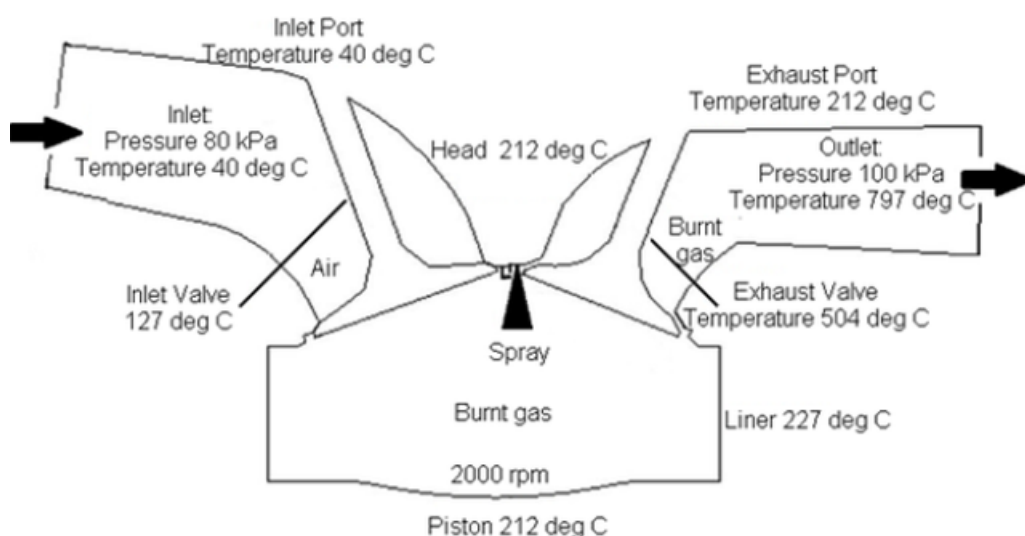
7.2. Direct Injection Spark Ignition Engine

The next sections describe the problem, including how to set up the simulation that is represented in the provided `.ftsim` file, and some results relating to surfaces, spray, and flame, that you can generate.

7.2.1. Problem Description

A three-dimensional single cylinder CFD simulation, of a four-stroke spray-guided Gasoline Direct Injection (GDI) Spark Ignition (SI) engine, is performed in this tutorial. Detailed boundary conditions are shown in [Figure 7.1: Schematic of GDI engine \(p. 68\)](#). Engine simulation is started from intake valve opening (IVO) and fuel is injected during the intake stroke. Homogeneous fuel air mixture is compressed and the spark ignited 15° before compression Top Dead Center (TDC).

Figure 7.1: Schematic of GDI engine



This tutorial illustrates the following steps in setting up and solving a Gasoline Direct Injection combustion simulation.

- Read an existing geometry into the Forte system.
- Define mesh setup and mesh the geometry.

- Set up the injection event.
- Set up the spark timing event.
- Set up the boundary and initial conditions.
- Specify the simulation controls.
- Specify the output variables and frequency.
- Run the simulation.
- Examine the results in the report.

7.2.1.1. Import the Geometry

Note

All Editor panel options that are not explicitly mentioned in this tutorial should be left at their default values. Changed values on any Editor panel do not take effect until you press the **Apply** button. Always press the **Apply** button after modifying a value, before moving to a new panel or to the Workflow tree.

In this tutorial, we will import an existing geometry into ANSYS Forte and set up the automatic mesh generation using a global mesh size and adapting the mesh near the valves. The geometry used will be half symmetric. To import the geometry, go to the Workflow tree and click Geometry. This opens the Geometry icon bar. Click the **Import Geometry**



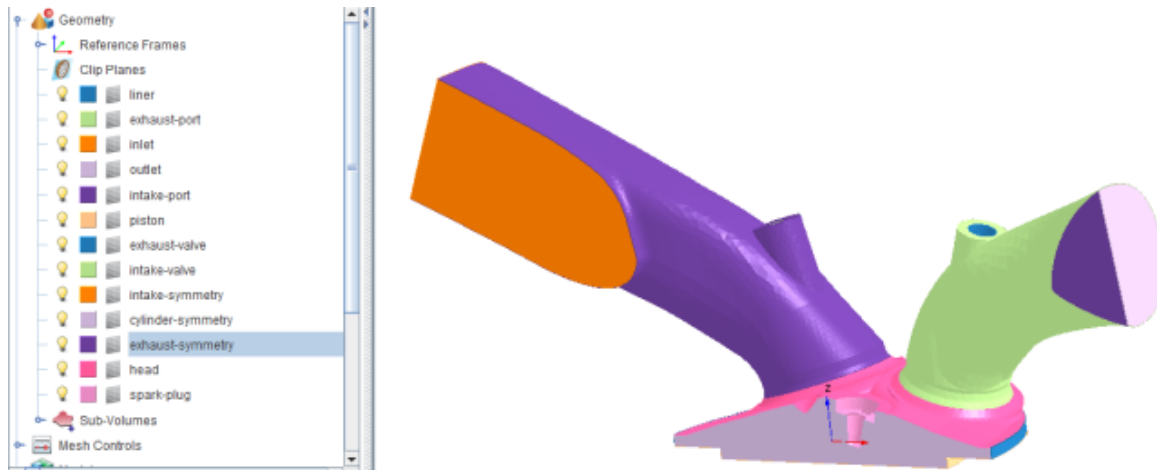
icon. In the resulting dialog, pull down and select **Surfaces from STL file**. Browse to and select the file called **Forte_GDI_Tutorial**. In the dialog that opens with STL file import options, accept the defaults for units and tolerances. Note that the geometry should always be in centimeters units when read into Forte. If it is not, the geometry can be scaled using the appropriate scaling factor to get it into centimeters. Note that once you have imported the geometry, there are a number of actions that you can perform on the items in the Geometry node, such as scale, rename, transform, invert normals, or delete geometry elements.

Once the geometry is read in, you should see the following surfaces: **liner, exhaust-port, inlet, outlet, intake-port, piston, head, exhaust-valve, intake-valve, intake-symmetry, cylinder-symmetry, exhaust-symmetry, spark-plug**.

The Geometry imports in an opaque mode and possibly preset zoom level. It is often helpful to Refit

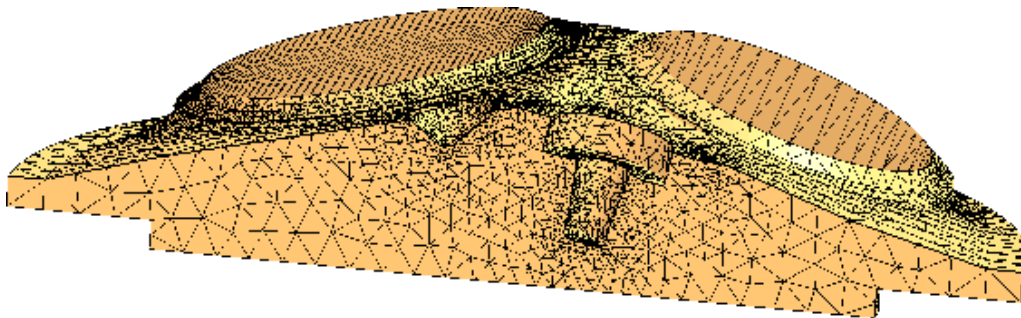


the view or use the mouse wheel to re-zoom. To change opacity, right-click the Geometry node in the right-side Visibility tree and select Medium for the Opacity level of all geometry elements, as shown in [Figure 7.2: Engine geometry with valves and ports defined \(p. 70\)](#).

Figure 7.2: Engine geometry with valves and ports defined

7.2.1.2. Subvolume Creation

Subvolumes are useful to refine regions of interest, such as the chamber. The subvolume can then be used in the mesh controls to control the size of the mesh in the selected subvolume. Select the following surfaces to define the subvolume: **cylinder-symmetry**, **head**, **liner**, **piston**, and **spark_plug**. You should see a triangulated region as in [Figure 7.3: Subvolume for the chamber \(p. 70\)](#).

Figure 7.3: Subvolume for the chamber



7.2.1.3. Automatic Mesh Generation Setup

The Material Point is the point in the domain that tells ANSYS Forte where the mesh will be generated and should be located at least one unit cell length away from any boundaries. This point must remain inside of the domain throughout the entire simulation. Typically it is located near the head so it is still inside the domain at TDC.

- Mesh Controls → Material Point: Accept the default **Reference Frame** using the **Global Origin**. Set the **Location** using **Cartesian Coord. System** to **X = 0.25, Y = 3.25, and Z = 0.25 cm**. Click **Apply**.
- From the Workflow tree, use Mesh Controls → Global Mesh Size to set the **Global Mesh Size** to **0.2 cm**.

Note

This is the recommended mesh size; a coarser setting of 0.3 cm could be used for tutorial purposes to produce a shorter runtime.

- Add a Solution Adaptive Mesh refinement: Click the **Solution Adaptive Meshing**  icon on the Mesh Controls Editor panel, and name the new control **SAM-Temperature**. Set the **Quantity Type = Gradient of Solution Field** and **Solution Variables = Temperature**. **Bounds Option = Statistical** and **Sigma Threshold = 0.5**. Set the **Size as fraction of Global Size = 1/4**. The refinement is **Active** between **Crank Angle 700** and **800 degrees**, and the **Location** option is **Sub-Volumes = chamber**. Click **Apply**.
- Click the **Solution Adaptive Meshing**  icon on the Mesh Controls Editor panel and name the new control **SAM-Velocity**. Set the **Quantity Type = Gradient of solution field** and **solution variable = VelocityMagnitude**. **Bounds Option = Statistical** and **Sigma Threshold = 0.5**. Set the **Size as fraction of Global Size = 1/2**. The refinement is **Active = Always**, and the **Location** option is **Entire Domain**. Click **Apply**.
- Accept the default setting for the **Small Feature Deactivation Factor**.

Next, you will refine the mesh around key geometric features such as valves and the piston, walls, and open boundaries (“continuative outflows”). The following steps show how the mesh is refined for key geometric features. From the Mesh Controls node, you can add **Point, Surface, Line, or Feature Refinements**, or **Small Feature Avoidance Controls**. Create the refinements specified in [Table 7.1: Details of the refinement types and settings \(p. 71\)](#). These are also good guidelines for the level of refinement for your own engine simulations.

Table 7.1: Details of the refinement types and settings

Name	Location	Refinement Type	Size Fraction	Cell Layers	Active
wall	Cylinder symmetry, exhaust-symmetry, intake, symmetry, exhaust-port, intake-port, head, piston, liner, spark-plug	Surface	1/2	1	Always
wall	Inlet, outlet	Surface	1/2	2	Always
spark	0.551, 0.0945, -0.1678 cm	Point (radius = 0.6 cm)	1/4	N/A	Always
valves	exhaust-valve, intake-valve	Surface	1/4	2	Always
tdc1	head, piston, liner	Surface	1/4	2	340-380 CA
tdc2	head, piston, liner	Surface	1/4	2	700-740 CA
chamber	Chamber	Secondary volume	1/2	N/A	Always

7.2.1.4. Models Setup

Chemistry: Now that the mesh has been set up, assigning models is next. Assign the chemistry with Models → Chemistry and use the **Import Chemistry**



icon and select the file **Gasoline_1comp_59sp.cks** from the ANSYS Forte data directory. If you are curious, you can view the chemistry details, such as chemistry source, pre-processing log, gas phase input, gas phase output, thermodynamic input, transport input, and transport output.

- **Flame Speed Model:** Use Models → Chemistry → Flame Speed Model and select the Table Library Option. Select Create New, then add ic8h18 as the fuel species and specify **ic8h18_flame_library** as the name. For the **Turbulent Flame Speed** settings, keep the defaults except for **Turbulent Flame Speed Ratio (b1)**, which should be set to **1.2**. Click **Apply**.

Transport: The default RNG k-epsilon model turbulence settings are used in this tutorial. Those are specified in the Editor panel for Models → Transport → Turbulence. The default fluid properties are also used, which are at Models → Transport.

Spray Model: Turn on the spray model by checking the box at Models → Spray Model. For Spray modeling in Forte, you will create an Injector first, then and Injection events and Nozzles to the injector. You can have multiple injectors in a Forte model and they can have different fuels.

Create a new **Solid Cone Injector**



and name it **Solid Injector**. Specify the following for the injector:

- For **Composition**, select **Create New...** then select **ic8h18** as the fuel species and specify the mass fraction as **1.0**. Specify **gasoline** as the name and click **Save**.
- Set **Injection Type** to **Pulsed Injection** and **Parcel Specification** to **Droplet Density** with a **Droplet Number Density** of **1**.
- Set the **Inflow Droplet Temperature** to **400K**.
- For **Spray Initialization**, select **Constant Discharge Coefficient and Angle** and specify the **Discharge Coefficient** as **0.7** and the **Mean Cone Angle** as **14.0 degrees**.
- Select **Rosin-Rammler Distribution** for the **Drop Size Distribution**, use **3.5** for the **Shape Parameter** and specify the **Initial Sauter Mean Diameter** as **120 micron**.
- Specify the following for the KH and RT model constants:
 - **Size Constant of KH Breakup = 0.5**
 - **Time Constant of KH Breakup = 10**
 - **Critical Mass Fraction for New Droplet Generation = 0.03**
 - Activate the **SMR Conservation in KH Breakup** option (Note: This option is typically activated for gasoline injections, but not for diesel injections.)
 - **Size Constant of RT Breakup = 0.1**

- **Time Constant of RT Breakup = 1.0**
- **RT Distance Constant = 1.9**
- Activate the **Use Gas Jet Model** and specify **0.5** for the **Gas Entrainment Constant**.

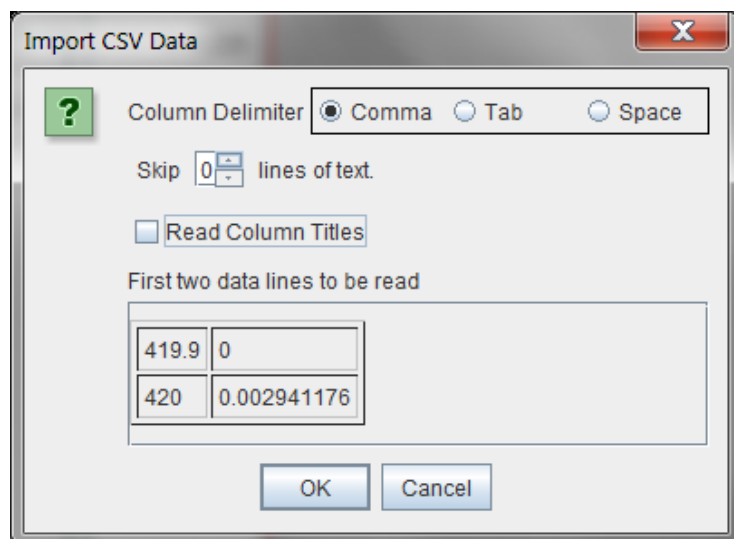
Now add an Injection by clicking the **Injection**



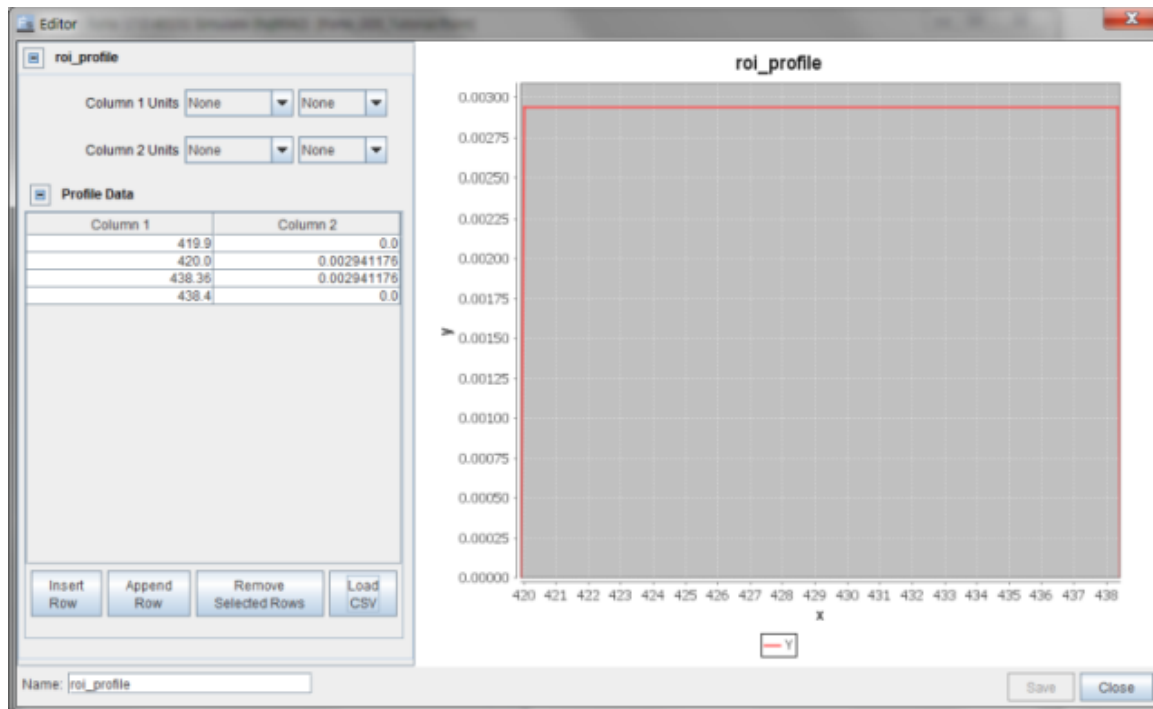
icon at the top of the Injector Editor panel or by right-clicking on the Solid Injector and selecting **Add > Injection**. Specify the **Injection Type** as **Pulsed** and the **Timing** as **Crank Angle**. Specify the **Start of injection** as **420 degrees** and the **Duration** as **18.4 degrees**.

For the **Velocity Profile**, select **Create New...**, then click the **Load CSV** button. Browse to the directory containing the tutorial files and select the file called **injection_profile.csv**. Clear the **Read Column Titles** option as shown in [Figure 7.4: Import injection profile from CSV dialog \(p. 73\)](#).

Figure 7.4: Import injection profile from CSV dialog



The profile should look like the image in [Figure 7.5: Injection profile after import \(p. 74\)](#) after import. Specify **roi_profile** as the profile name.

Figure 7.5: Injection profile after import

Now that the injection event has been added, we need to specify the nozzles. The nozzles require input for the location, direction, and nozzle hole diameter (or area). Since this is a half-symmetry model, three nozzles will be specified.

On the Solid Injector panel, click the New Nozzle



icon or right-click Solid Injection and click **Add > Nozzle**. Name the new nozzle **nozzle1**.

For the **Location**, specify **X = -5.296mm**, **Y = 0.634mm**, and **Z = 6.468mm** and for **Spray Direction** specify **X = 3.0415mm**, **Y = 2.73735mm**, and **Z = -9.12449mm**. For the **Nozzle Size**, select the **Diameter** option and specify a nozzle diameter of **300 micron**.

Now that the first nozzle has been created, copy and paste **nozzle1** and name it **nozzle2** using the **Copy**



and **Paste**



icons. For the **Location**, specify **X = -5.65mm**, **Y = 1.1885mm**, and **Z = 6.5991mm** and for **Spray Direction** specify **X = 1.49781mm**, **Y = 5.08692mm**, and **Z = -8.4782mm**.

Repeat the copy and paste process by copying **nozzle2** and naming it **nozzle3** to create the last nozzle. For the **Location**, specify **X = -6.7647mm**, **Y = 0.72441mm**, and **Z = 6.1653mm** and for **Spray Direction** specify **X = -2.90254mm**, **Y = 2.74977mm**, and **Z = -9.16592mm**.

Now that all the nozzles have been created, they should appear similar to the model in [Figure 7.6: Nozzles after specifying location and orientation \(p. 75\)](#).

Figure 7.6: Nozzles after specifying location and orientation

Spark Ignition: Turn on the spark ignition model by checking the box at Models → Spark Ignition.

The spark ignition model defaults include a **Kernel Flame to G-Equation Switch Constant** of **2.0**, **Min. Kernel Radius for Kernel to G-equation** switch of **0.1cm**, **Flame Development Coefficient** of **0.5**, and **Number of Flame Particles for Each Spark Plug** of **3000**.

Use the **New Spark**




icon to create a new spark event and name it **Spark**.

Use Models → Spark Ignition → Spark to set up the details of the spark event. In the **Reference Frame**, use **Global Origin** and **Cartesian** coordinates for the **Location**, and set **X= 0.551 cm**, **Y=0.0945 cm**, and **Z=-0.1678 cm**. Select **Crank Angle** for **Timing** and **Starting Angle = 705.0 degrees**, **Duration = 10.0 degrees**. Under **Spark Energy**, set **Energy Release Rate = 20.0 J/sec**. Accept the default **0.5** for **Energy Transfer Efficiency** and **0.25 mm** (note that the unit is mm) for **Initial Kernel Radius**. Click **Apply**.

7.2.1.5. Boundary Conditions

Boundary conditions are specified for each of the geometry elements in Boundary Conditions → *geometry-element-name*, where *geometry-element-name* is each of the items under Geometry in the Workflow tree.

Inlet: From the Boundary Conditions node, click the **New Inlet**  icon and create an Inlet. Select **Pressure_In** from the **Location** list. Select **Pressure Inlet** as the **Inlet Type** with a constant pressure of **8E+04 Pa**. You will set the inlet composition to the correct **Mass Fraction** values corresponding to the premixed fuel and air:

- **o2=0.233**
- **n2=0.767**


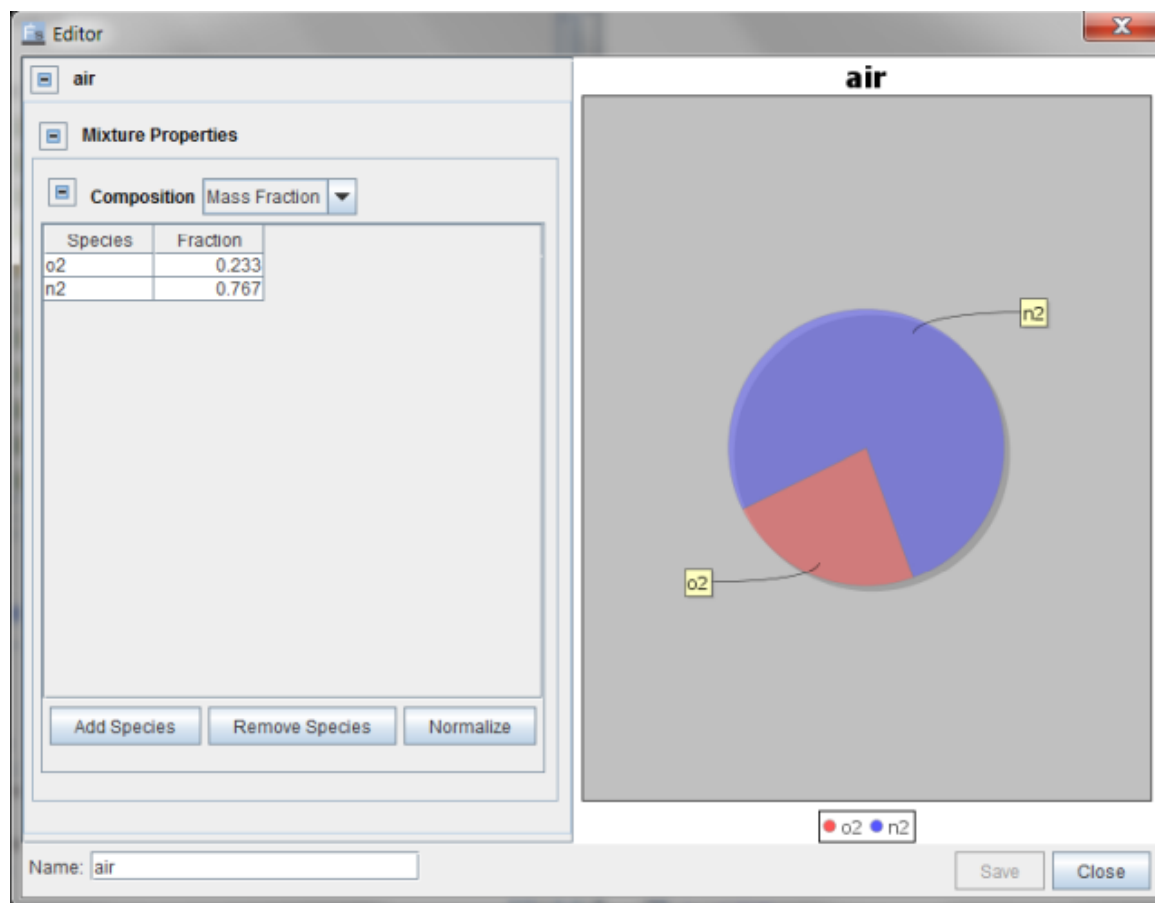
To do this, select **Create New** in the **Composition** drop-down list above the **Location** list and click the **Pencil**  icon. This opens the Gas Mixture panel, where you select **Add Species** and then choose the species you want from the mechanism (see [Figure 7.7: Gas Mixture Editor: Defining inlet gas composition \(p. 76\)](#)). You can find a species more quickly by typing (the beginning of) the species name to filter the list. **Save** the inlet gas mixture as **air**. You will then see a list to define turbulence using **Turbulent Kinetic Energy and Length Scale** of **10000 cm2/sec2** and **1 cm**, respectively.

Figure 7.7: Gas Mixture Editor: Defining inlet gas composition


Outlet: From the Boundary Conditions node, click the **New Outlet** icon and create **Outlet**. Select the **Pressure_Out** geometry item and select the **Pressure Outlet** as the **Outlet Type**. Give the outlet a **Pressure** of **1E+05 Pa**, which arises from the back-pressure of the exhaust system with an **Offset Distance to Apply Pressure** of **0.0 cm**. Set the **Turbulence Boundary Conditions** to **Turbulent Kinetic Energy** and **Length Scale** of **10000 cm2/sec2** and **1 cm**, respectively.


Piston: From the Boundary Conditions node, click the **New Wall** icon and name it **Piston**. Select the **Piston** item in the **Location** list. Set the **Temperature Option** to **Constant** and **485 K**. Turn ON the **Wall Motion** option and set the piston **Motion Type** to use a **Slider-Crank Model** with a **Stroke** of **9.0 cm** and a **Connecting Rod Length** of **14.43 cm** with **0.0 Piston Offset**. Change the **Movement Type** to **Moving Surface** and accept the default **Global Origin Reference Frame**. For **Direction**, the piston will move along the Z-direction, specify **1.0** for **Z**.

Intake: Click the **New Wall** icon and name it **intake-port**. Select the **intake-port** item in the **Location** list and set the **Temperature** to **313 K**.

Exhaust: Click the **New Wall** icon and name it **exhaust-port**. Select the **exhaust-port** item in the **Location** list and set the **Temperature** to **485 K**.

Liner: Click the **New Wall** icon and name it **liner**. Select the **liner** item in the **Location** list and set the **Temperature** to **500 K**.

Head: Click the **New Wall**  icon and name it **head**. Select the **head** item in the **Location** list and set the **Temperature** to **485 K**.



Intake Valves: Click the **New Wall**  icon and name it **intake-valve**. The valve specification is a little more complicated than the stationary walls, such that several steps are required within the wall panel. These steps specify the wall motion and the way we want the mesh to adapt to the gap opening near the valve seat when the valve opens or closes.


- Select the **intake-valve** in the **Location** list.
- Check the **Heat Transfer** option and set the **Temperature** to the constant value of **400 K**.
- Turn ON (check) the **Wall Motion** and set the **Motion Type** to **Offset Table**.
- Accept the default **Global Origin** for the **Reference Frame**. Select **Cartesian** for the **Coord. System** under **Direction** for the valve motion and set **X = 0.241922 cm**, **Y = 0.0 cm**, **Z = -0.970296 cm**.
- To import the lift profile (named **intake_valve_lift.csv** in the same location as the **.stl** file you started this tutorial with), select **Create New** from the **Lift Profile** drop-down list and click the **Pencil**





icon. In the Profile Editor, click the **Load SV** button and navigate to the **intake_valve_lift.csv** file. Clear the **Read Column Titles** option and press **OK**. At the bottom of the Profile Editor window, name this **intake_valve_lift**. Ensure that the units in the first column are set to **Angle** and the second column for **Distance** is set to **m**. Save the Profile.

- For **Movement Type**, change the pull-down menu to **Valve**. Then select just the surface boundary portion of the valve that comes into contact with the valve seat on the head, as well as the surface boundary that contains the seat region. In this case, select both the **head** and the **intake-valve**.
- Finally, set the **Valve Motion Activation Threshold** to **0.02 cm**. This indicates that the valve will not open until the lift value specified in the valve-lift profile exceeds this threshold. A smaller value will cause the valves to open sooner, but the mesh refinement required to resolve the gap will be higher. This is a trade-off that will need to be determined based on the goals and outcomes desired from the simulation. In addition to the activation threshold, you may also change the minimum number of cells desired at the smallest gap opening. In this case, specify the **Approx. Cells in Gap at Min. Lift** as **1.5**.

Exhaust Valves: Click the **New Wall**  icon and name it **exhaust-valve**. **New Wall**  icon and name it **exhaust-valve**. Follow a similar procedure as for the Intake Valves. This time, select the **exhaust-valve** item in the **Location** list and set the **Temperature** to **777 K**. Turn ON the **Wall Motion** and set the **Motion Type** to **Offset Table** and the **Vertices to Transform** to **Interior**. Select **Global Origin** for the **Reference Frame**. Select **Cartesian** for the **Coord. System** under **Direction** and set **X = -0.327218 cm**, **Y = 0.0 cm**, and **Z = -0.944949 cm**. Similarly to the intake valves, first import the **.csv** (named **exhaust_valve_lift.csv**) file, clear the **Read Column Titles** option, and name it **exhaust_lift_profile**. Then follow the same procedures for specifying the **Movement Type** to **Valve**, selecting the **Valve Seat and Surface it Contacts** (**exhaust-valve** and **head**), and setting the **Valve Motion Activation Threshold** to **0.02 cm**. Specify the **Approx. Cells in Gap at Min. Lift** as **1.5**.

Symmetry: Click the **New Symmetry**  icon and name it **symmetry**. Select the **cylinder-symmetry**, **exhaust-symmetry**, and **intake-symmetry** item in the **Location** list.

Spark Plug: Click the **New Wall**  icon and name it exhaust-valve. **New Wall**  icon and name it **spark-plug**. Select the **head** item in the **Location** list and set the **Temperature** to **485 K**.

7.2.1.6. Initialization

The domain is initialized with the operating conditions, species concentrations and temperatures. The Default Initialization species composition is at the expected exhaust composition, assuming complete combustion. The intake and exhaust must also be initialized to the boundary condition values. Set the following initialization parameters:

Default Initialization:


- Select Default Initialization in the Workflow tree. Set the **Initialization Order** to **2**. Since flow goes generally from the intake to the cylinder and then to the exhaust, this indicates that this default (cylinder) region will be the second in the initialization order precedence list.
- We will use the Forte Composition Calculator to compute the exhaust gas composition. Click the **Composition Calculation** button




in the Toolbar or go to **Utility > Composition Calculation**. In the Composition calculation, specify the **Fuel Mass** (27 mg), select the fuel mixture in the Liquid input, for **Air Flow** specify **Phi** and set **Phi = 1.0**, **EGR Fraction = 0.0**, and for **Internal EGR** select the **Estimate from CR** and specify a **CR** of **10**. Next, set the **Calculate** option to **Exhaust** and click **Create Mixture**. Specify a name of **exhaust_gas**. In the Initial Conditions, select **exhaust_gas** for the **Composition**.

- Set a **Temperature** of **1,070 K** and the **Pressure** to **1.05359E+05 Pa**.
- The **Turbulence** initialization uses the **Turbulent Kinetic Energy and Length Scale** option with values **10,000 cm2/sec2** and **1.0 cm**, respectively.
- The **Velocity** is initialized using **Velocity Components** and all the values are set to **zero**.
- Click **Apply**.

Intake Initialization: The intake manifold Initial Condition is set to match the Boundary Condition at the Inlet. Since this is a separate port that can be closed off from the main cylinder region, we also need to set the equivalent of a material point to identify the region, as well as an initialization order that helps determine what region takes precedence in initializing new cells that appear when gaps are opened.

- From the Initial Conditions Workflow tree item, select the **New Port Initialization**  icon and name it **intake**.
- To identify the region, select a point for the **Location** under the **Reference Frame** selection, which is a point that will always be within the **Intake** port. Set the coordinates for this case to **X=-5.6838, Y=1.4295, Z=4.2858 cm**, which is a point just inside the inlet.
- Set the **Initialization Order** to **1**. Flow is expected to go from the intake to the main region; for this reason, we give it the first order in initialization precedence.
- Set the **Composition** by selecting in the previously saved profile, **air**. Set the **Temperature** to **313 K** and **Pressure** to **8.0E+04 Pa**. The Turbulence initialization uses **Turbulent Kinetic Energy and Length Scale** option set to **10,000 cm2/sec2** and **1.0 cm**, respectively. Click **Apply**.

Exhaust Initialization: The Initial Condition of the Exhaust is set to match the Boundary Condition of the Outlet.

- From the Initial Conditions Workflow tree item, select the **New Port Initialization**  icon and name it **exhaust**.
- To identify the region, select a point for the **Location** under the **Reference Frame** selection, which is a point that will always be within the **ExhaustPort** region. Set the coordinates for this case to **X=4.7892, Y=1.5061, Z=3.1552 cm**, which is a point just inside the outlet.
- Set the **Initialization Order** to **3**. Flow is expected to go from the cylinder to the exhaust port; for this reason, we give it the last order in initialization precedence for the 3 regions defined.
- Set the **Composition** to the existing profile, **exhaust_gas**. Set the **Temperature** to **1070K** and **Pressure** to **1.0E+05 Pa**.
- The **Turbulence** initialization uses the **Turbulent Kinetic Energy and Length Scale** option; these are set to **10,000 cm²/sec²** and **1.0 cm**, respectively. Click **Apply**.

7.2.1.7. Simulation Controls

Simulation controls allow you to define the simulation limits, RPM, time step, chemistry solver, and transport terms.

Simulation Limits: Use the Simulation Controls panel to select a **Crank Angle**- based simulation from a **CA** of **328 to 880.2 degrees**. Set **RPM = 2,000 rpm**. The engine **Cycle Type** is **4-Stroke**.

Time Step: Use Simulation Controls → Time Step and set the following parameters:

- **Initial Simulation Time Step** to **5.0E-7 seconds**
- **Max. Crank Angle Delta Per Time Step** of **1.1 degrees**
- Set the **Max. Time Step Option** to **Constant** and set the value of **Max. Simulation Time Step** to **1.E-5 sec**. The time step will be adaptively determined throughout the simulation, based on local solution gradients, so this just sets the maximum value allowed. You may also consider reducing the time-step using the **Time Varying** option. This can be helpful for injection and spark events.

The Advanced Time Step Control Options settings are kept at the defaults:

- **Time Step Growth Factor = 1.3**
- **Fluid Acceleration Factor = 0.5**
- **Rate of Strain Factor = 0.6**
- **Convection factor = 0.2**
- **Internal Energy Factor = 1.0**
- **Max. Convection Subcycles = 8**

Chemistry Solver: Simulation Controls → Chemistry Solver is also kept at the defaults, with the following parameters:

- **Absolute Tolerance = 1.0E-12**
- **Relative Tolerance = 1.0E-5**
- Use **Dynamic Cell Clustering** to take advantage of groups of cells with similar conditions. Select 2 features to introduce Dynamic Cell Clustering: 1) **Max. Temperature Dispersion** of **10 K** and 2) a **Max. Equilibrium Ratio Dispersion** of **0.05**.
- To increase the time-to-solution speed, you have the ability to choose when chemistry is activated. In this tutorial, select **Activate Chemistry Conditionally**, turn on (check) **After First Spark Event**, and select **When Temperature is Reached** with **Threshold Temperature 600 K** and also select **During Crank Angle Interval** between **670 and 850 Crank Angle**. This ensures that chemistry is active during the time that combustion is expected even if temperature does not rise above 600 K. Click **Apply**.

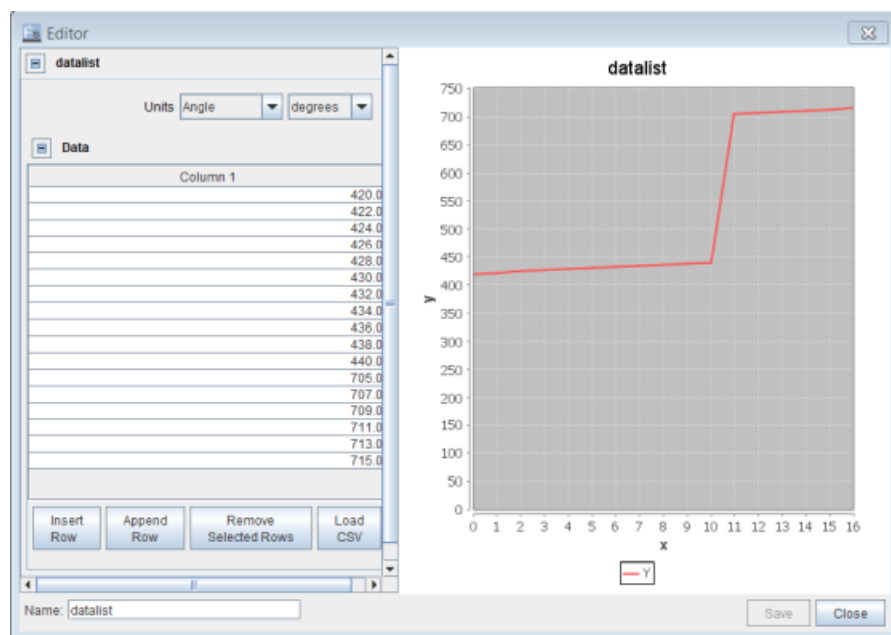
Transport Terms: Use the default transport tolerances and maximum number of iterations.

7.2.1.8. Output Controls

Output controls determine what data are stored for viewing during the simulation and for creating plots, graphs, and animations in ANSYS Forte Visualize, CFD-Post, EnSight, or FieldView.

Spatially Resolved: Allows you to control when spatially resolved data such as velocity, temperature, species concentrations, etc., will be output. In the Spatially Resolved panel, set the **Crank Angle Output Control** to report every **10 degrees** (to manage the size of the output file). You can optionally increase the frequency of output during the cycle by selecting **User Defined Output Control** and importing the *spatial_output.csv* file, which has a list of specific crank angles where spatially resolved output will occur. Alternatively, you could select **User Defined Output Control**, and use the Profile Editor to create some other file specifying an output crank-angle profile (the provided profile is illustrated in [Figure 7.8: Profile of spatially resolved output control data \(p. 80\)](#)). Keep the default species and solution variables selected for output. Reducing the variables selected will help reduce file sizes.

Figure 7.8: Profile of spatially resolved output control data





Spatially Averaged: Allows you to control the output of values that are averaged across the domain. On the Spatially Averaged panel, set the **Crank Angle Output Control** to reporting every **1 degree**. Keep the default species and solution variables selected for output.

Restart Data: If you anticipate that the case will be stopped and you want the ability to restart it from the last time step solved, select Output Controls → Restart Data. You can specify certain Restart Points using a separate file. Turn on (check) **User Defined Restart Points** and use the Profile Editor to create a Restart profile. You can view, edit, or import new Restart Points in this 1-D Profile Editor. Sometimes it is helpful in spark ignition cases to save a restart file after IVC but before the spark occurs (CA=687 in this case) so you can use the compression portion of the cycle as a start point in additional runs. Create a new profile for this purpose called **restart_output** and add two lines with **CA** value set to **415** and **700** which correspond to points just before injection and spark timing.

7.2.1.9. Preview Simulation

You can view the profiles for the Piston, Intake Valves, Exhaust Valves, and Mesh Refinement during the simulation by selecting Preview Simulation > Boundary Motion > Simulation Preview. This is an excellent way to check whether you have valve overlap and that the settings are correct.

As a method for checking the automatically generated mesh, you can generate a Preview Mesh. Select Preview Simulation → Mesh Generation and then click the **New Automatic Mesh Plot**  icon, name this new automatic mesh generation plot **720CA**, select **Crank Angle** as the **Time Option** and set it to **720 CA**, specify the **Normal** by setting **Y = 1.0**. Then click **Apply** to save the settings and then on the **Generate Mesh**  icon. ANSYS Forte will generate the preview mesh and display it in the 3-D View window. It is a good practice to look at meshes at key points in the cycle such as Firing Top Dead Center (FTDC), Exhaust Valve Opening (EVO), Intake Valve Closed (IVC), Intake Valve Open (IVO) and Exhaust Valve Closed (EVC). If you want to see the cut plane where the mesh will be generated, click the **Plane Filter** box and specify an origin point and normal direction for the cut plane.

7.2.1.10. Run Simulation


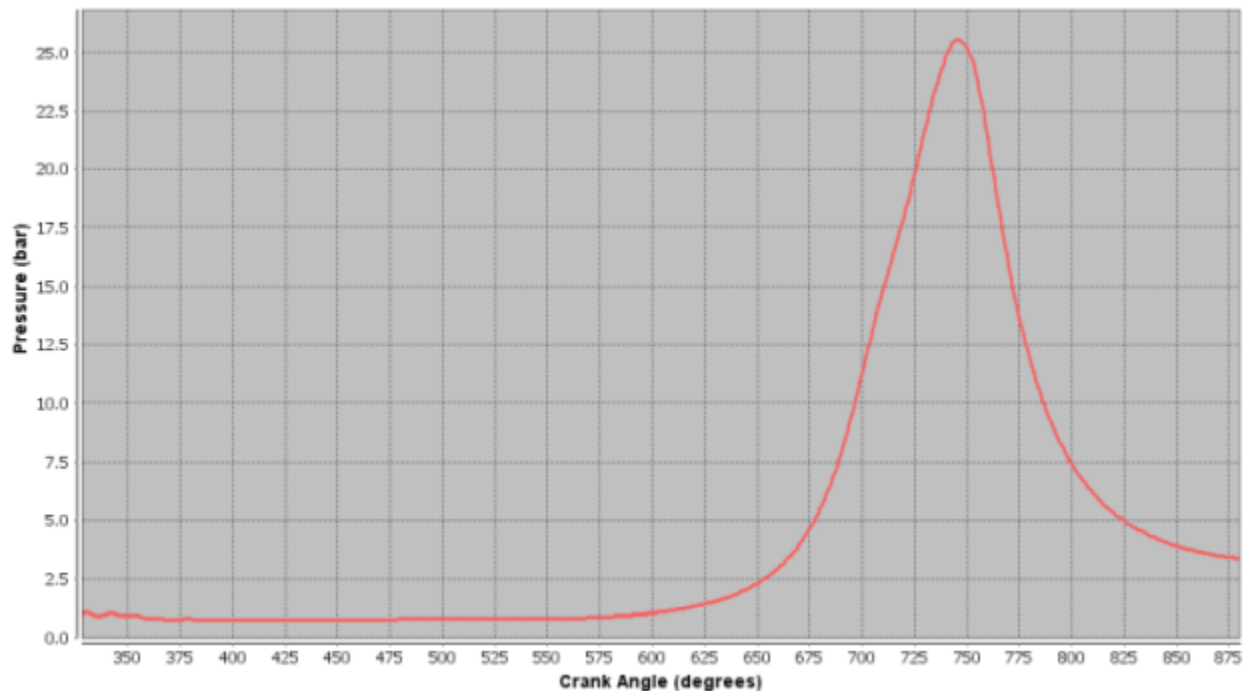
To complete the lesson, select Run Simulation on the Workflow tree and, once ANSYS Forte displays the green **START** button on the Run Simulation panel and reports a “Ready” status, click **Start**. You can monitor the results by clicking on the **Monitor Runs**  icon. In the Monitor window that opens, you can select the run you want to monitor. Open the Forte Monitor dialog, then select **Pressure** under the **thermo.csv** grouping. The pressure trace should look like the one in [Figure 7.9: Pressure trace result from Forte \(p. 82\)](#).

Figure 7.9: Pressure trace result from Forte

7.2.1.11. Run Settings

The settings here depend on the system and environment for your simulations. The default for the Run Settings panel is to have nothing selected.

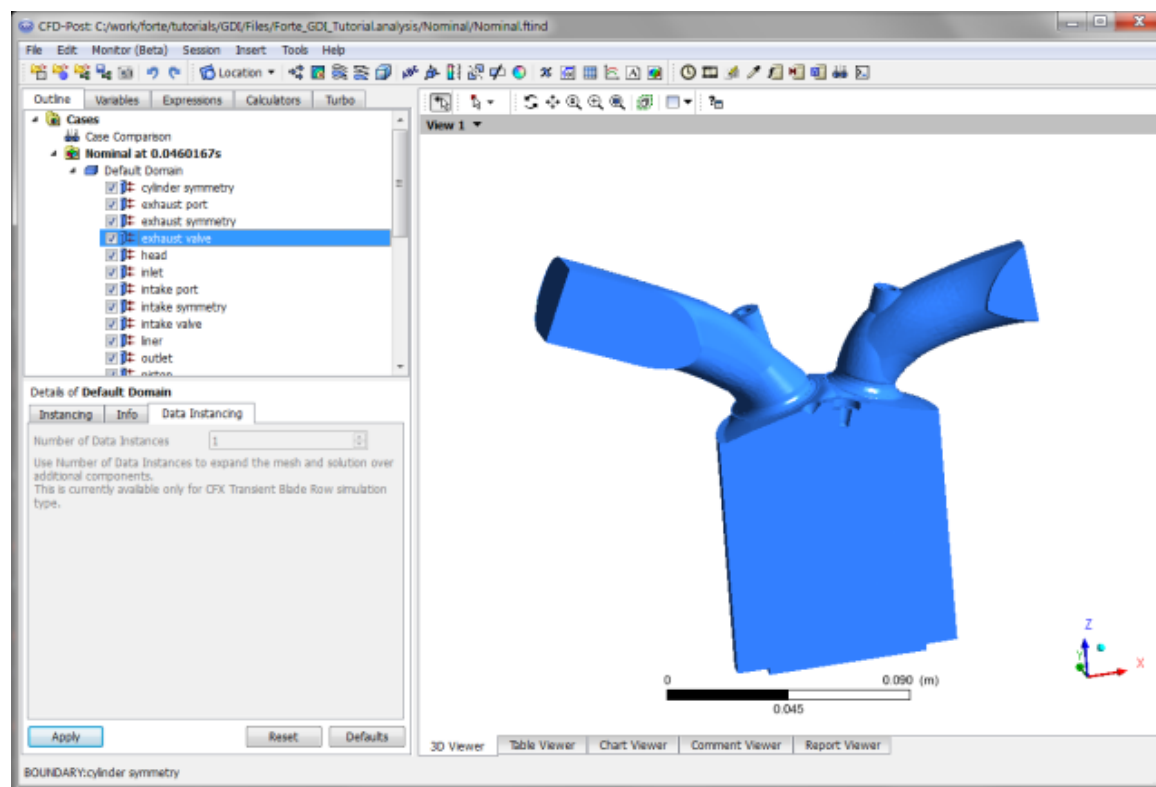
Run Options: Under **Job Script Options**, change **Default Run Type** to **Parallel** and change the default **MPI Arguments** to **8**. (If you do not have MPI installed and configured, keep the default of a serial run.)

Windows Settings: Under **Job Script Options**, change **Default Run Type** to **Parallel** and change the default **MPI Arguments** to **8**. (If you do not have MPI installed and configured, keep the default of a serial run.)

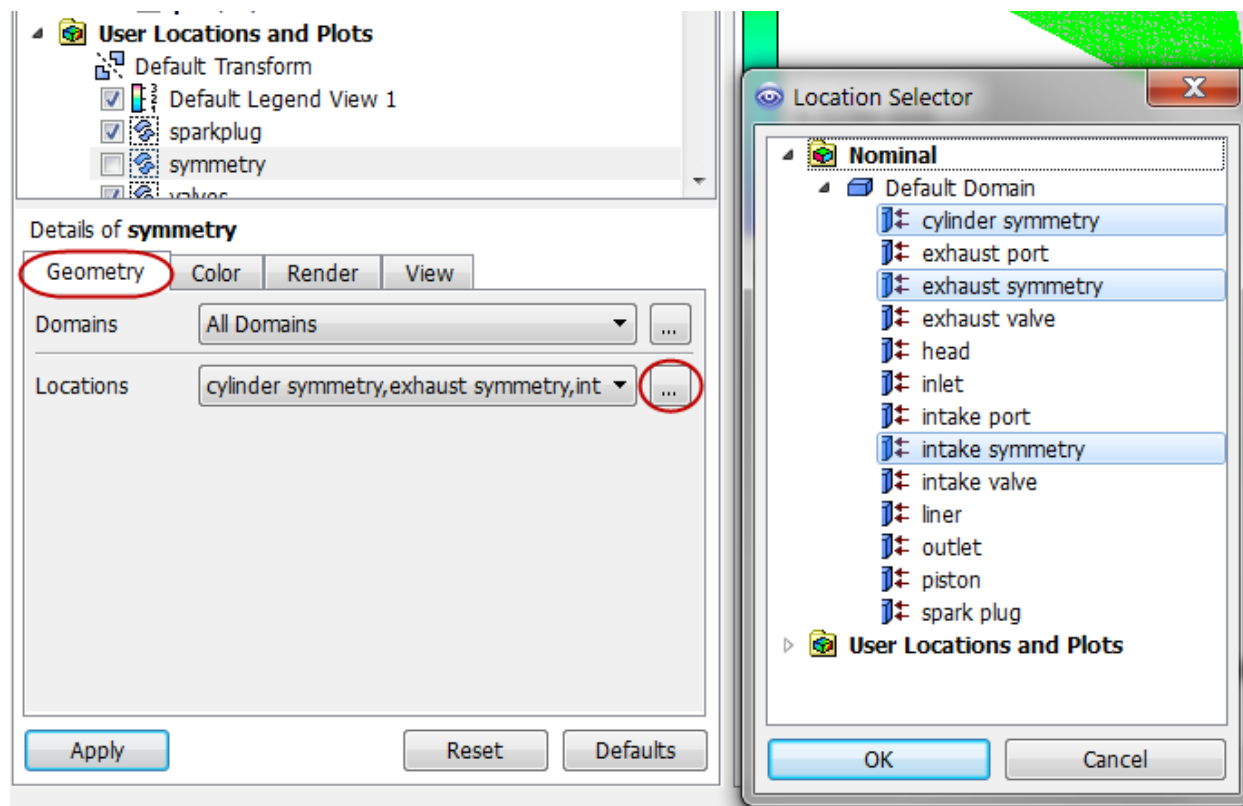
Linux Settings: This tutorial does not require changes to this panel's defaults; adjust them as necessary for your environment.

7.2.2. Results

To view the results of the simulation, open the results in CFD-Post. In CFD-Post, open the solution file for the case (**Nominal.ftind**). [Figure 7.10: Screen view of the solution file in CFD-Post \(p. 83\)](#) shows the screen view once the solution file has been loaded.

Figure 7.10: Screen view of the solution file in CFD-Post

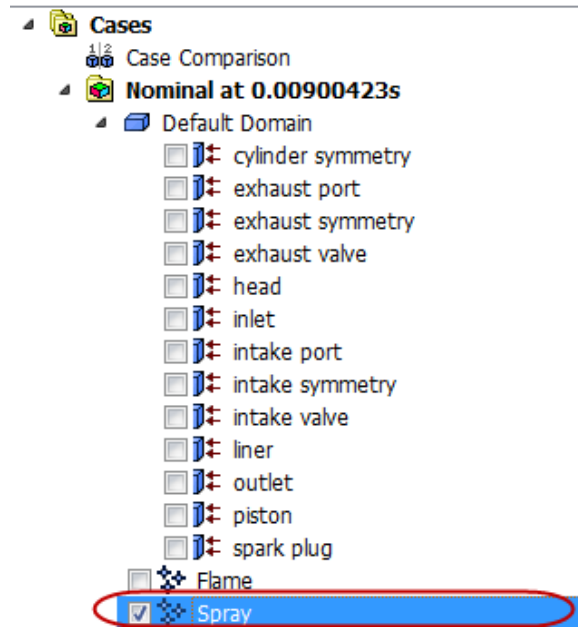
Once you have read in the model, you can create surface groups to group the surfaces. In this case, we will create a surface group for the spark plug, walls, symmetry boundaries, and the valves. To create a surface group, right-click **User Locations and Plots Insert > Locations > Surface Group** and specify the name **symmetry**. On the Geometry tab, select the surfaces names **cylinder symmetry**, **exhaust symmetry**, and **intake symmetry** and click **Apply** as in [Figure 7.11: Creating surface groups in CFD-Post](#) (p. 84).

Figure 7.11: Creating surface groups in CFD-Post

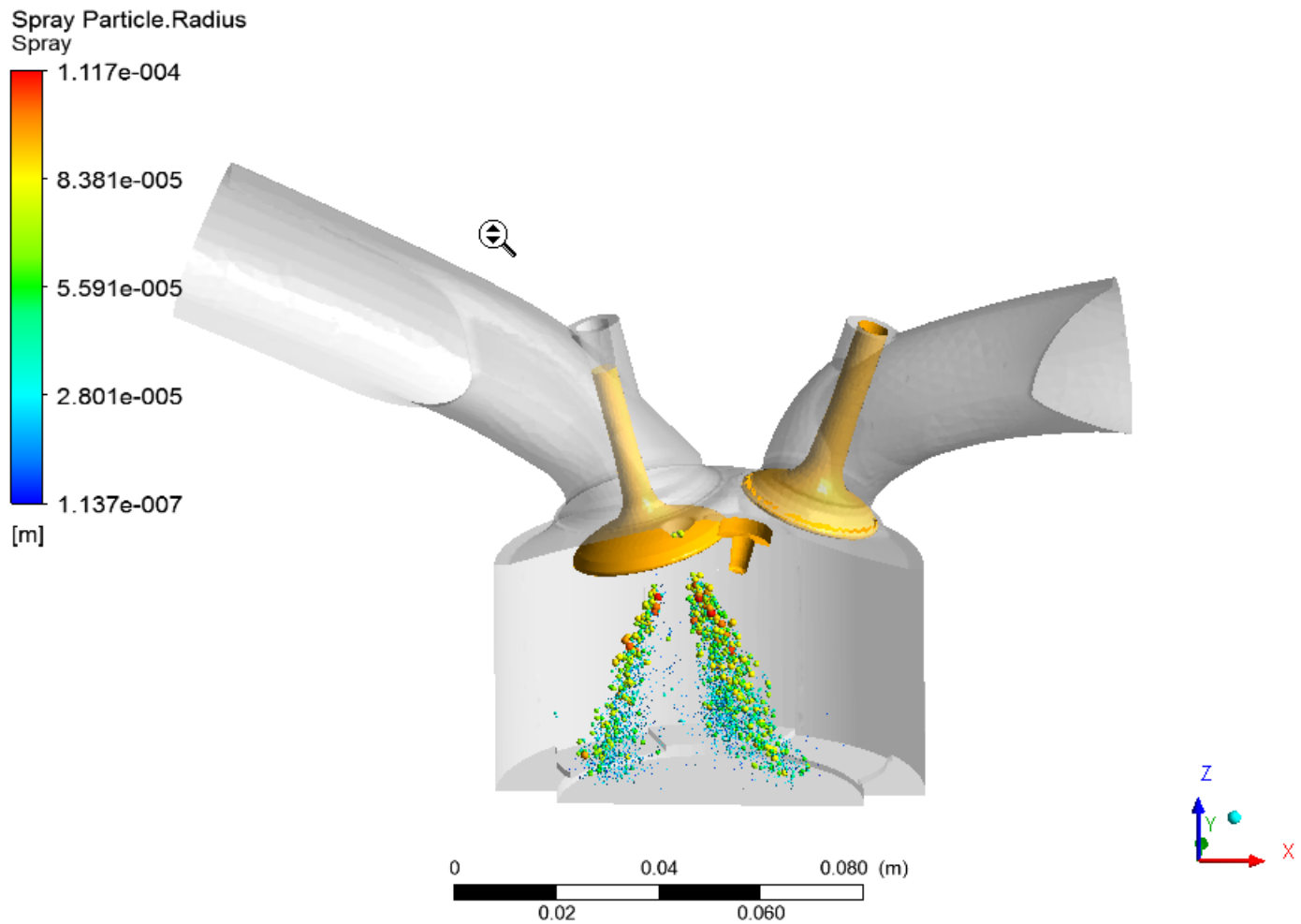
On the Color tab, choose **gray** as the color. On the Render tab, specify a **Transparency** of **0.5**. Repeat these steps create the following surface groups:

Surface Group Name	Surface Selections	Color	Transparency
Sparkplug	Spark plug	Orange	0.5
Valves	Intake valve, exhaust valve	Orange	0.5
Walls	Exhaust port, head, inlet, intake port, liner, outlet, piston	Gray	0.5

To visualize the spray particles, turn on the **Spray** object in the list as in [Figure 7.12: Turning on the spray particles \(p. 85\)](#).

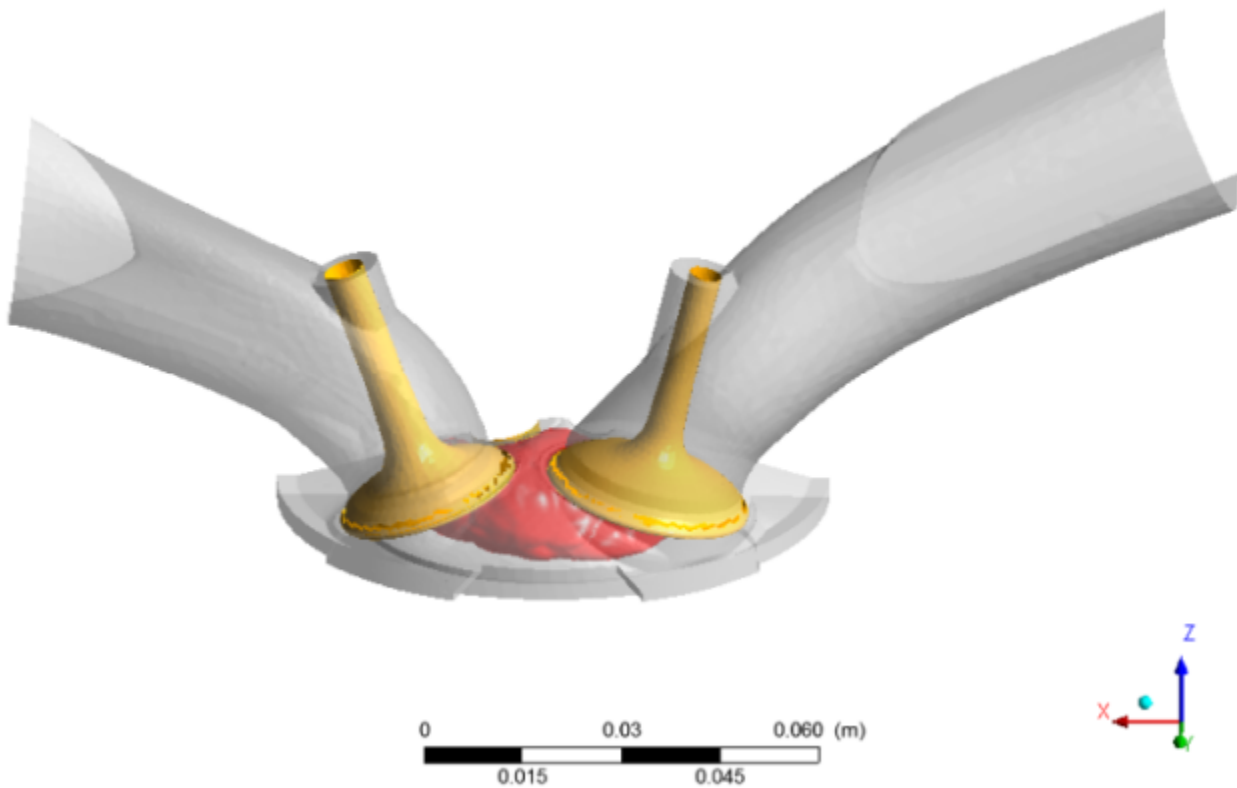
Figure 7.12: Turning on the spray particles

When the spray is activated, all the particles will be a single color. You can use a variable to change the color by going to the Color tab. Set the **Mode** to **Variable** and select **Spray Particle Radius**. You can also adjust the droplet size rendering on the Symbol tab; a value of **0.3** works well for this case. Go to **Tools > Timestep Selector** and select crank angle **436CA**. With these changes, the spray should resemble [Figure 7.13: Spray particles in the combustion chamber \(p. 86\)](#).

Figure 7.13: Spray particles in the combustion chamber

You may also want to visualize the location of the flame front for spark ignited engines. This can be accomplished by creating an **Iso-Surface** for variable **G** and setting the value to **0.0**. Right-click **User Locations and Plots > Location** and select **Iso-Surface**. Name the surface **flame**. For **Variable**, select **G** and specify a value of **0.0**. On the Color tab, set **Mode** to **Constant** and choose a **red** color. Go to **Tools > Timestep Selector** and choose a time step after spark timing, for example, **728CA**, and you should see the flame location, as in [Figure 7.14: Iso-surface of the flame showing the flame location at 728CA](#) (p. 87).

Figure 7.14: Iso-surface of the flame showing the flame location at 728CA



Note

A CFD-Post state file (***Forte_GDI_Tutorial_PostProcessing.cst***) is included with this tutorial that will create all the settings discussed in the post-processing section.

Chapter 8: Two-Stroke Engine Simulation

This tutorial describes how to use ANSYS Forte CFD to simulate combustion for a two-stroke marine engine with intake and exhaust ports. Details on how to use ANSYS Forte's automatic mesh generation with sliding interfaces, a requirement for 2-stroke engines, is discussed. The tutorial also demonstrates profile creation and the import and loading of an `.stl` file to start the simulation from the geometry.

8.1. Data Provided

The following sections describe the provided files, time required, prerequisites, and a utility for comparing your generated project file (**.ftsim**) with the one provided in the tutorial download.

8.1.1. Files Used in This Tutorial

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

and select the Forte tutorials, and the desired files to download. The files for this tutorial include:

- **FORTE-2stroke.ftsim**: A project file of the completed tutorial, for verification or comparison of your progress in the tutorial set-up. The project file includes all the relevant geometry, chemistry set, and spray profile details.
- **intake_pressure_profile.csv** and **exhaust_pressure_profile.csv**: Two files of data describing the profile of the time-varying pressure for the intake and exhaust.
- **time_step_size.csv**: Adaptive time-step profile.
- **output_crank_angles.csv**: Profile of crank-angle points for spatially resolved output.
- **FORTE_2stroke.stl**: Geometry file for the 2-stroke engine in this project without the chemistry set, spray profile, and other project details. (Optional)
- **restart_crank_angles.csv**: Profile defining restart points. (Optional)
- **Nom-VelMag_Vector-Centerline.avi**: An animation showing the velocity magnitude and piston motion of the meshed system. (Optional)

The tutorial sample file is provided as a download. You have the opportunity to select the location for the file when you download and uncompress the sample files.

Note

This tutorial is based on a fully configured sample project that contains the tutorial project settings. The description provided here covers the key points of the project set-up but is not intended to explain every parameter setting in the project. The **.ftsim** file has all custom and

default parameters already configured; the text highlights only the significant points of the tutorial.

8.1.2. Project Comparison Utility

The Forte installation includes the `cgns_util export` command, which you can use to compare the parameter settings in the project file generated at any point during your tutorial set-up against the provided `.ftsim`, which has the parameter settings for the final, completed tutorial. This command is described in the Forte User Guide.

Briefly, you can double-check project settings by saving your project and then running the `cgns_util` to export your tutorial project, and then to export the provided final version of the tutorial. Save both versions and compare them with your favorite diff tool, such as DIFFzilla. If all the parameters are in agreement, you have set up the project successfully. If there are differences, you can go back into the tutorial set-up, re-read the tutorial instructions, and change the setting of interest.

8.1.3. Time Estimate

As a guideline for your own simulations, this tutorial is estimated to take approximately 9.5 hours on an Intel® Xenon® processor E5-2690 at 3.00 GHz (20 total cores).

8.1.4. Prerequisites for This Tutorial

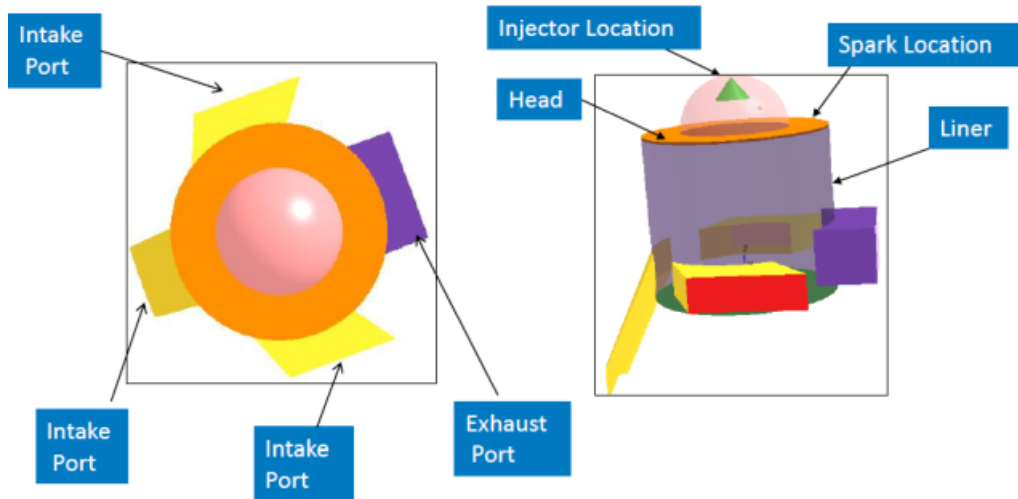
We recommend starting with the ANSYS Forte Quick Start Guide, which explains the workflow of the ANSYS Forte user interface, before doing this tutorial.

8.2. Two-Stroke Marine Engine Project Setup

The next sections describe the problem, including how to set up the simulation that is represented in the provided `.ftsim` file, and some results relating to surfaces, spray, and flame, that you can generate.

8.2.1. Problem Description

In this case, a full 3-D geometry is used to simulate a 2-stroke marine engine, including combustion. The fuel in this engine is gasoline and is direct-injected into the combustion chamber.

Figure 8.1: Components of direct-injection two-stroke engine simulation project

The **FORTE_2stroke.ftsim** project file has been preconfigured with all the information that will be discussed in this section. You do not need to input any values but can just follow along, reading the instructions and viewing the settings in the loaded **.ftsim**. This chapter nevertheless explains step-by-step the process of setting up the project, as an illustration of the features in the user interface.

Open the project file, **FORTE_2stroke.ftsim**, from the location where you stored the downloaded tutorial files.

In this tutorial, we will import an existing geometry into ANSYS Forte and set up the automatic mesh generation using a global mesh size and adapting the mesh near the valves. To import the geometry, go to the Workflow tree and click Geometry. This opens the **Geometry** icon bar. Click the **Import Geometry**



icon. In the resulting dialog, pull down and select Surfaces from STL file. In the dialog that opens with STL file import options, accept the defaults.

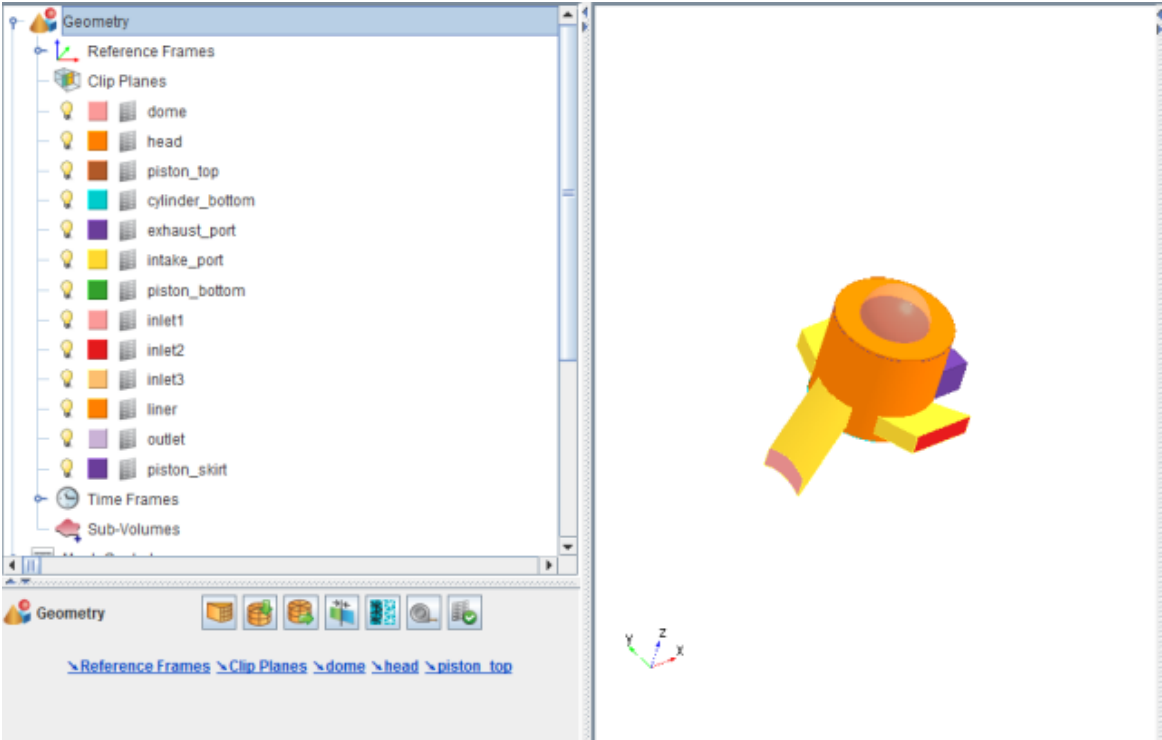
The mesh will be automatically generated during the simulations. When the file browser launches, navigate to the folder Tutorial_2stroke, then select FORTE-2stroke.stl. Note that once you have imported the geometry, there are a number of actions that you can perform on the items in the Geometry node, such as scale, rename, transform, invert normals, or delete geometry elements.

The Geometry imports in an opaque mode and possibly preset zoom level. It is often helpful to Refit



the view or use the mouse wheel to re-zoom. To change opacity, right-click the Geometry node in the right-side Visibility tree and select Medium for the Opacity level of all geometry elements, as shown in [Figure 8.2: Two-stroke geometry after import \(p. 92\)](#).

Figure 8.2: Two-stroke geometry after import



8.2.1.1. Two-Stroke Engine Details

A simple engine configuration is used in this tutorial for demonstration purposes. The details of the configuration and simulation settings are presented here.

Table 8.1: Details of the diesel engine geometry used in this tutorial

Item	Value	Units
Cycle type	2-stroke	
Fuel injection system	Direct injection, hollow core	
Compression ratio	12.65	
Bore	8.58	cm
Stroke	6.73	cm
Squish	13.97	cm
Fuel	Gasoline	

The mesh is created using the Automated Mesh Generator in ANSYS Forte. The values described in ??? can be accessed from the *FORTE_2stroke.fts* project file. If you did not already have a pre-configured geometry, you could import a Fluent Mesh file, STL, KIVA-3V, or CGNS geometry file. Fluent Mesh files are the preferred format for importing geometry because they are guaranteed to be water-tight. The Fluent Mesh file can be created as a surface mesh for the geometry in ANSYS Meshing.

8.2.1.2. Automatic Mesh Generation Setup

Material Point: On this panel, we must set the point that defines where ANSYS Forte will mesh. This point, the Material Point, must always lie inside the geometry during the entire cycle and should be

located at least one unit cell length away from any boundaries. On the Material Point Editor panel, define the point's location as **X = 0.0, Y = 0.0, Z = 6.9 cm**.

Global Mesh Size: Set the **Global Mesh Size** to **2.0 mm**. Other Mesh Controls should be set to these values:

Table 8.2: Settings for Mesh Control refinements

Item	Refinement type	Refinement location	Refinement level	Refinement layers	Crank angle active range
AllSurfaces	Surface Refinement	All walls except inlets and outlet	1/2		
OpenBoundaries	Surface Refinement	Inlets and Outlet	1/2	2 layers	
TDC	Surface Refinement	Head and Piston Top	1/4	2 layers	350° - 370°
PortWalls2	Surface Refinement	Intake and Exhaust Port	1/2	2	
Combustion	Point	(0.0, 0.0, 6.0) cm Radius of 4.4 cm	1/2		274° - 400°
Coarse Feature	Feature	Feature <i>angle</i> =60° Feature <i>Radius of application</i> =7.0 mm Intake/Exhaust Port, Liner, Piston Skirt, Piston Top	1/2		
Fine Feature	Feature	Feature <i>angle</i> =60° Feature <i>radius of application</i> =2.0 mm Liner, Piston Skirt, Piston Top	1/4		

8.2.2. Chemistry Set Details

Chemistry: Now that the mesh has been set up, assigning models is next. Assign the chemistry using Models → Chemistry and use the **Import Chemistry**



icon and select the file **Gasoline_1comp_59sp.cks** from the ANSYS Forte data directory. If you are curious, you can view the chemistry details, such as chemistry source, pre-processing log, gas phase input, gas phase output, thermodynamic input, transport input, and transport output.

Flame Speed Model: On the Workflow tree, use Models → Chemistry → Flame Speed Model and select the **Table Library Option**. Select **Create New**, then add **ic8h18** as the fuel species and specify **iso_octane** as the name. Click **Save**. Close the Library Editor panel. For the **Turbulent Flame Speed** settings, keep the defaults. Click **Apply**.


8.2.3. Transport Property Settings

The default values are used in the Transport property panel. In the Transport → Turbulence panel, the **RNG k-epsilon** model is used, with default values.

8.2.4. Spray Model Settings

Models → Spray Model: Since this is a direct-injection case, turn ON (check) Spray Model in the Workflow tree to display its icon bar (action bar) in the Editor panel. In the panel, keep the defaults. Leave the **Use Vaporization Model** (default) check ON.

Add an Injector: The icon bar provides two spray-injector options: **Hollow Cone** or **Solid Cone**. For this

injector, click the **Hollow Cone**  icon. In the dialog that opens, name the new hollow-cone injector as **Hollow Cone Injector**. This opens another icon bar and Editor panel for the new hollow-cone injector. In the Workflow tree under the Hollow Cone Injector node, add a Nozzle and an Injection. You can either right-click the Hollow Cone Injector node and select **Add** or select that Workflow tree item or use the icons in the panel's action bar.

On the Hollow Cone Injector panel, create a new **Composition**, select **ic8h18** (iso-octane) in the **Species** list, specify the physical properties as **iso-octane**, and specify the **Mass Fraction** as **1.0**. Use **iso-octane** for the **Vaporization properties** and name the Composition **gasoline**. Click **Save**.

For the injection, specify the following parameters:

Table 8.3: Injection settings used in this tutorial

Input	Value	Units
Injected Parcel Count	3000	-
Inflow Droplet Temperature	363	K
Injection Pressure	51.7	bar
Inwardly Opening Nozzle	Checked (ON)	-
Mean Cone Angle	54.0	degrees
Liquid Jet Thickness	15.0	degrees
Droplet Size Distribution	Rosin-Rammler Distribution	-
Shape Parameter	3.5	-

Input	Value	Units
Breakup Length Model Constant	12.0	-

With the Hollow Cone Injector selected, add a Nozzle to the injector by clicking the **New Nozzle**



icon. On the Nozzle Editor panel, set the parameters as described in [Table 8.4: Nozzle settings used in this tutorial \(p. 95\)](#). Click Apply. You can see the nozzle appear at the top of the geometry. (You may want to make the Geometry non-opaque or change the color of the nozzle itself in the Visibility tree to make the nozzle easier to see in the interior.)

Table 8.4: Nozzle settings used in this tutorial

Location	
Reference Frame	Global origin
Coordinate system	Cartesian
X	0.0 cm
Y	0.0 cm
Z	9.221 cm
Spray Direction	
Reference Frame	Global origin
Coordinate System	Spherical
Θ	180 degrees
Φ	0.0 degrees
Nozzle Size	
Nozzle Area	0.001647 cm ²

Injection: On the Injection panel, specify the **Start of Injection** as **275 degrees**, the **Duration of Injection** as **12.42 degrees**, specify the **Velocity Profile** as a **Square Profile**, and the **Injected Mass** as **0.007292 g**.

Spark Ignition: Turn on the spark ignition model by checking the box at Models → Spark Ignition. Keep the defaults for the **Spark Ignition** settings.

Use the **New Spark**



icon to create a new spark event and name it **Spark**.

Use Models → Spark Ignition → Spark to set up the details of the spark event. In the Location, for **Reference Frame**, use **Global Origin** and **Cartesian** coordinates for the **Location**, and set **X= 1.233 cm**, **Y=0.0 cm**, and **Z=7.76 cm**. Select **Crank Angle** for **Timing** and **Starting Angle = 319.0 degrees**, **Duration = 9.0 degrees**. Under **Spark Energy**, set **Energy Release Rate = 25.0 J/sec**. Accept the default **0.5** for **Energy Transfer Efficiency** and **0.5 mm** (note that the unit is mm) for **Initial Kernel Radius**. Click **Apply**.


8.2.5. Boundary Conditions

Boundary conditions specify inlets, outlets, and walls. Once you create one boundary condition, you can copy and paste it (using the **Copy**



and **Paste**



icons on the icon bar in the Editor panel), then modify it to create a second boundary condition. Select a boundary surface, either from the list in the workflow tree or using the **Select from Screen**  tool to pick a surface in the 3-D View to associate with a boundary condition.


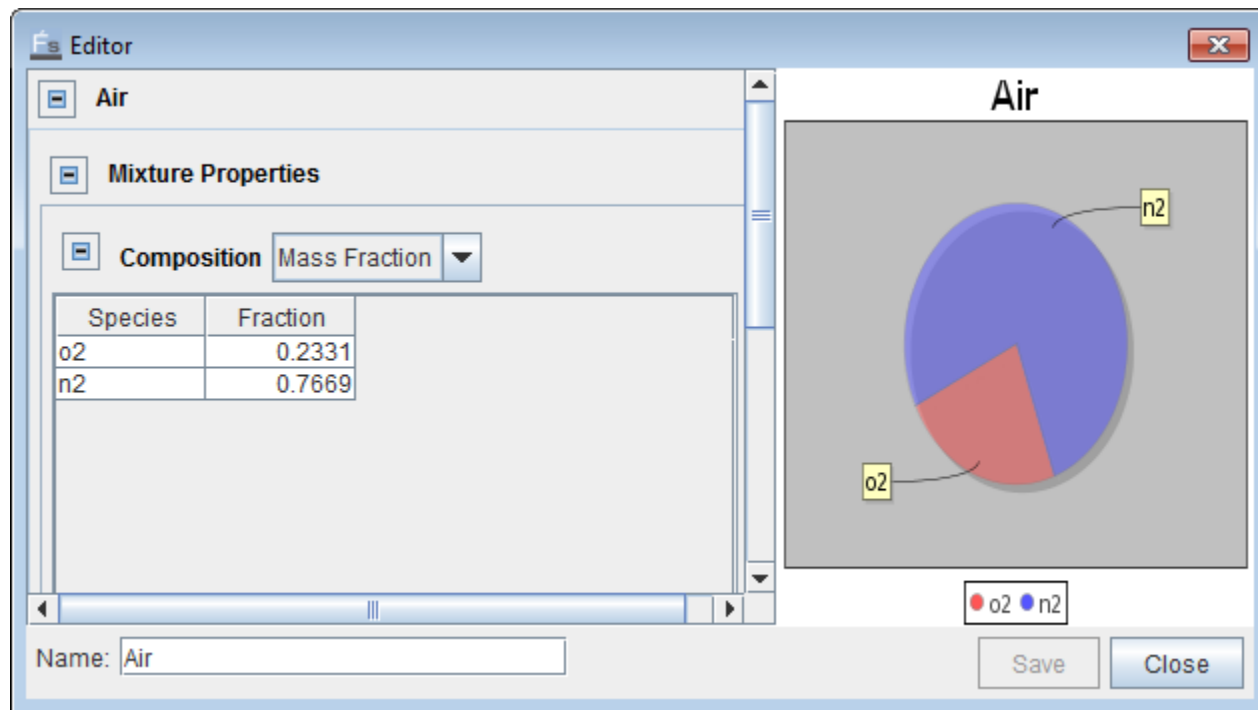
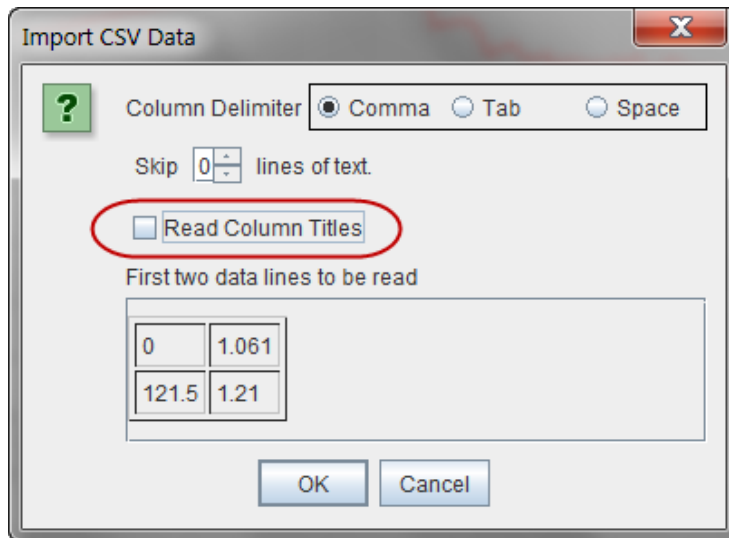
Inlet: From the Boundary Conditions node, in the Editor panel, click the **New Inlet**  icon and create an Inlet named **Inlet1**. At the top of the Editor panel, select Create New... for the Composition. Click the **Add Species** button and then select **o2** (oxygen) and **n2** (nitrogen) as the species. Specify **Mass Fraction** of **0.23** for **o2** and **0.77** for **n2**, as shown in [Figure 8.3: Composition editor specifying inlet Air mixture \(p. 96\)](#). Name the composition **Air** and click **Save** then **Close**.

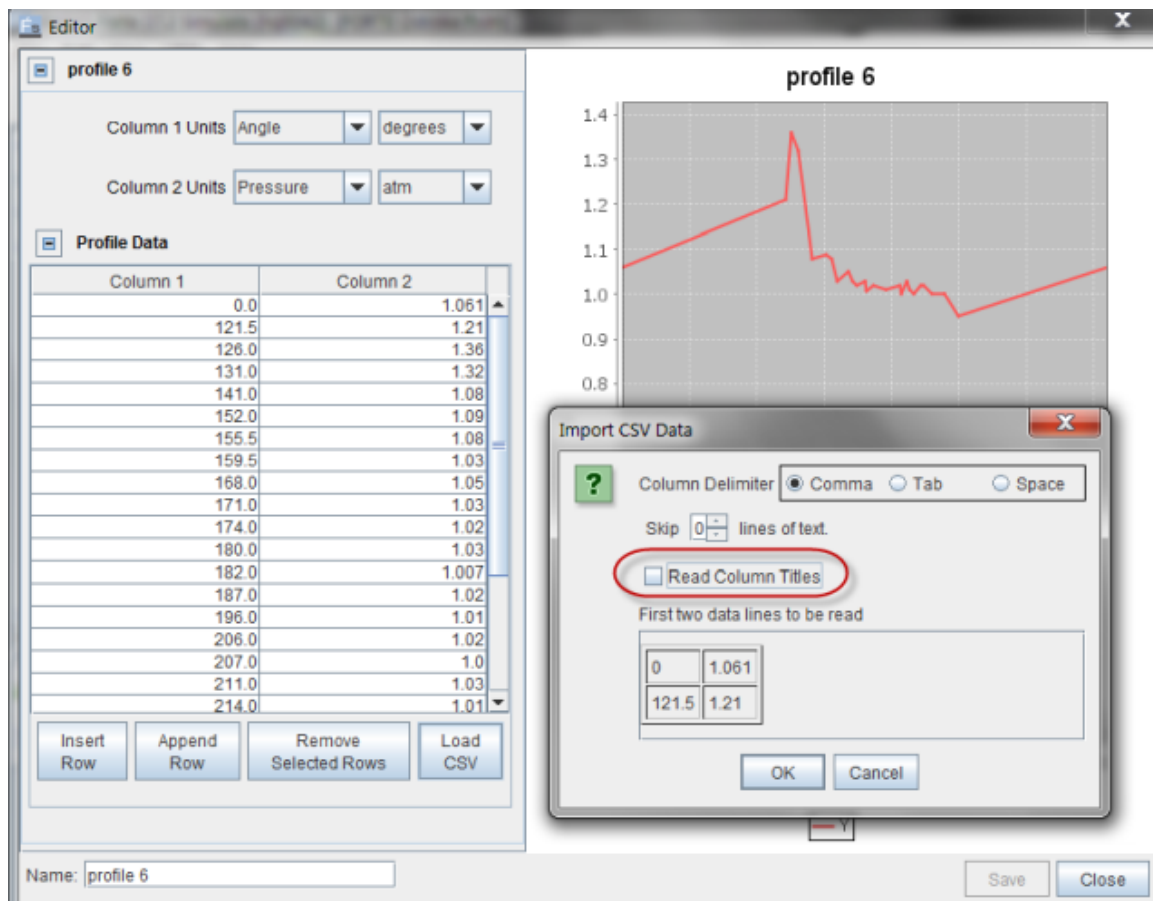
Figure 8.3: Composition editor specifying inlet Air mixture



In the Editor panel, select **inlet1** from the **Location** list. Select **Total Pressure, Time Varying** as the **Inlet Option**. For the pressure profile, select **Create New** and import the pressure profile using the **Load CSV** option. The name of the pressure profile is **intake_pressure_profile.csv**. On the Load CSV dialog, clear (deselect) the **Read Column Titles** option (because the file does not contain any column titles), as shown in [Figure 8.4: Import CSV data for pressure profile \(p. 97\)](#). The unit for Column 1 should be **Angle** and **degrees** and for Column 2 the units should be **Pressure** and **bar**.

Figure 8.4: Import CSV data for pressure profile

The pressure profile should look like the profile shown in [Figure 8.5: Importing the intake pressure profile](#) (p. 97).

Figure 8.5: Importing the intake pressure profile

Define **Turbulence** using values for **Turbulent Kinetic Energy** and **Length Scale** of **3,600 cm²/sec²** and **1.0 cm**, respectively. On the Editor panel, accept **Assume Isentropic** for the **Temperature Value Type**. Click **Apply**.


Now that the first inlet has been created, copy and paste **inlet1** and name it **inlet2** using the **Copy**




and **Paste**.



icons. Repeat this step to create **inlet3**. Update the Location to **inlet2** and **inlet3**, respectively, for each new boundary condition, **inlet2** and **inlet3**.

Outlet: From the Boundary Conditions node, go to the Editor panel and click the **New Outlet**  icon and create **Outlet**. Select **Total Pressure, Time Varying** as the **Outlet Option**. For the pressure profile, select **Create New** and import the pressure profile using the **Load CSV** option. The name of the pressure profile is **exhaust_pressure_profile.csv**. On the Load CSV dialog, deselect the **Read Column Titles** option, because the file does not contain any column titles. The unit for Column 1 should be **Angle** and **degrees**, and for Column 2 the units should be **Pressure** and **bar**. Set the **Turbulence Boundary Conditions** to **Turbulent Kinetic Energy and Length Scale** of **3600 cm2/sec2** and **1 cm**, respectively.

Piston: From the Boundary Conditions node, go to the Editor panel and click the **New Wall**  icon and name it **Piston_Moving**. Select the **piston_top** item in the **Location** list. Set the **Temperature Option** to **Constant** and **450 K**. Turn **ON** the **Wall Motion** option and set the piston **Motion Type** to use a **Slider-Crank** with a **Stroke** of **6.73 cm** and a **Connecting Rod Length** of **13.97 cm** with **0.0 Piston Offset**. To set the axis for translation, under **Direction**, the **Reference Frame** is **Global Origin** and the **Coordinate System** is **Cartesian**. Set **X=0.0 cm**, **Y=0.0 cm**, **Z=1.0 cm**. Change the **Movement Type** to **Sliding Interface** and select the **liner** and **piston_skirt** from the list of surfaces. For this setting, the moving piston should be selected along with the surface it is sliding/moving past.

Note

The RPM is specified on the Simulation Controls Editor panel.

Intake: Click the **New Wall**  icon and name it **IntakePort**. Select the **intake_port** item in the **Location** list and turn **ON** the **Heat Transfer** option and set the **Temperature** to **300 K**.

Now reproduce the Intake Port with **Copy**



and **Paste**



four times to create four new boundaries as detailed in the following table.

Table 8.5: Details of boundaries created from Copy and Paste operations

Item	Boundary Name	Location	Boundary Condition
1	ExhaustPort	exhaust_port	Wall Model=Law of the Wall $T_{\text{wall}}=490 \text{ K}$ Heat Transfer=ON

Item	Boundary Name	Location	Boundary Condition
2	Liner	cylinder_bottom liner	Wall Model=Law of the Wall $T_{\text{wall}}=400 \text{ K}$ Heat Transfer=ON
3	Head	dome head	Wall Model=Law of the Wall $T_{\text{wall}}=400 \text{ K}$ Heat Transfer=ON
4	Piston_Anchored	piston_bottom piston_skirt	Wall Model=Law of the Wall $T_{\text{wall}}=450 \text{ K}$ Heat Transfer=ON


8.2.6. Initial Conditions

The domain is initialized with the operating conditions, species concentrations and temperatures. The Default Initialization species composition is at the expected exhaust composition, assuming complete combustion. The intake and exhaust must also be initialized to the boundary condition values. Set the initialization parameters as described in the following sections.


Initialization Order: When looking at the piston in the TDC position, each isolated region should have an initialization region with a material point specified, which is typically the combustion chamber, the exhaust port, and the intake port. In many cases, there will be multiple intake regions that will require their own initialization, as in this case.

For 2-stroke engine cases, the combustion chamber should have an initialization order of 1. The intake ports should then have initialization orders of 2, 3, and 4, respectively. Lastly, the exhaust port would have an initialization order of 5.

Default Initialization:

- Select Default Initialization in the Workflow tree. In the Editor panel, set the **Initialization Order** to **1**. (For 2-stroke engines, the combustion chamber should have an **Initialization Order** of **1**.) Select **Create New** in the **Composition** drop-down list and click the **Pencil**  icon. This opens the Gas Mixture panel, where you select **Add Species**. Set a **Mass Fraction** of **o2=0.1123911**, **n2=0.7421674**, **co2=0.0996009**, and **h2o=0.0458404**, and **Save** this composition as **Exhaust_Est**. On the Editor panel, set a **Constant Temperature** of **1,220 K** and a **Constant Pressure** of **2.7 bar**.
- The **Turbulence** initialization uses the **Constant** option and the **Turbulent Kinetic Energy and Length Scale** option with values **3,600 cm2/sec2** and **1.0 cm**, respectively.
- The **Velocity** is initialized by using **Constant Velocity** and selecting **Velocity Components** and accepting defaults.
- Click **Apply**.

Intake Initialization: The intake Initial Condition is set to match the Boundary Condition at the Inlet. Since this is a separate port that can be closed off from the main cylinder region, we also need to set the equivalent of a material point to identify the region, as well as an initialization order that helps determine what region takes precedence in initializing new cells that appear when gaps are opened.

- From the Initial Conditions Editor panel, select the **New Secondary Region From Material Point**  icon and name it **Intake1**.
- To identify the region, select a point for the **Location** under the **Reference Frame = Global Origin** selection, which is a point that will always be within the **Intake1** port. Set the **Cartesian** and the coordinates to **X=0.0, Y=6.0, Z=1.2 cm**, which is a point just inside the inlet.
- Set the **Initialization Order** to **2**. Initialization order follows flow order.
- Set the **Composition** by selecting in the previously saved profile, **Air**. Set the **Temperature** to **Constant** and **300 K** and **Pressure** to **Constant** and **1.013 bar**. The **Turbulence** initialization is **Constant** and uses the **Turbulent Kinetic Energy and Length Scale** option set to **3,600 cm²/sec²** and **1.0 cm**, respectively. The **Velocity** is initialized by using **Constant Velocity** and **Velocity Components** and accepting defaults. Click **Apply**.

- Copy**




and **Paste**



Intake1 twice, naming the new initializations **Intake2** and **Intake3**. All the settings are the same, except for the **Initialization Order** and **Location**:

Intake #	Coordinates	Initialization Order
Intake2	X= -4.8, Y=0.0, Z= -1.0 cm	3
Intake3	X=0.0, Y= -5.0, Z=1.2 cm	4

Exhaust Initialization: The Initial Conditions of the Exhaust are set to match the Boundary Condition of the Outlet.

- From the Initial Conditions panel, select the **New Secondary Region from Material Point**  icon and name it **ExhaustPort**.
- To identify the region, select a point for the **Location** under the **Reference Frame** selection, which is a point that will always be within the **ExhaustPort** region. Set the coordinates for this case to **X=6.0, Y=0.0, Z=1.4 cm**.
- Set the **Initialization Order** to **5**. Flow is expected to go from the cylinder to the exhaust port; for this reason, we give it the last order in initialization precedence for the 5 regions defined.
- Set the **Composition** to **Constant** and the existing profile, **Exhaust_Est**. Set the **Temperature** to **490 K** and **Pressure** to a **Constant 0.981 bar**.


- The **Turbulence** initialization uses a **Constant Turbulent Kinetic Energy and Length Scale** option; these are set to **3,600 cm²/sec²** and **1.0 cm**, respectively. Click **Apply**.
- The **Velocity** is initialized using **Constant Velocity** and selecting **Velocity Components** and accepting defaults. Click **Apply**.

8.2.7. Simulation Controls

Simulation controls allow you to define the simulation limits, RPM, time step, chemistry solver, and transport terms.

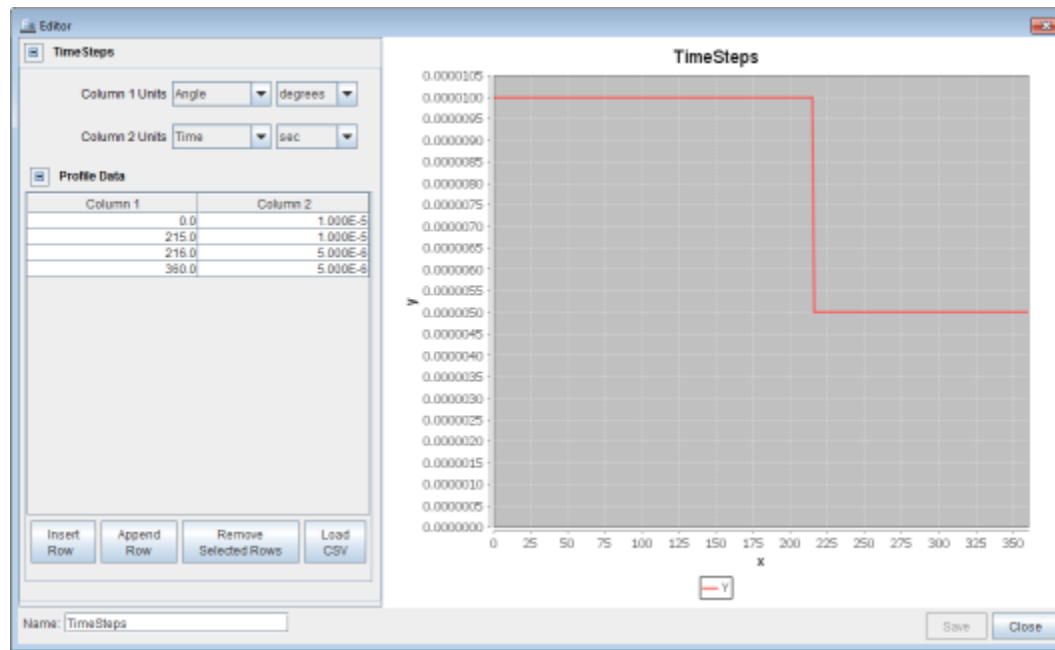
Simulation Limits: ANSYS Forte uses adaptive time stepping, with user-specified initial time step and maximum time step. Use **Crank Angle-based** limits, the simulation Start Crank Angle (i.e., **Initial Crank Angle**, under Simulation Controls in the Workflow tree) is set to **97.0 degrees ATDC**, and the simulation **Final Simulation Crank Angle** is set to **+455 degrees ATDC**. Set **RPM** to **2,000.0 rpm** and **Cycle Type** to **2-Stroke**.

Time Step: ANSYS Forte uses adaptive time stepping, with user-specified initial time step and maximum time step. Use Simulation Controls → Time Step and accept the defaults except:

- Set the **Max. Time Step Option** to **Time Varying**, and set up a profile to control the variation in time steps. This allows the maximum time step to be reduced at times of high interest, such as the spark and injection, and decreased during other intervals. In the drop-down list for **Max Time Step Profile**, select **Create New** and click the **Pencil**  icon. Because this is a 2-stroke engine, the simulation should run from **0 to 360 CA degrees**. After setting the following parameters, click **Save**.

Column 1	Column 2
Units = Angle and degrees	Units = Time and sec
0.0	1.00E-5
215.0	1.00E-5
216.0	5.00E-6
360.0	5.00E-6
Name of profile: TimeSteps	

The Editor panel for creating the profile will resemble [Figure 8.6: TimeSteps profile for Max Time Step Profile \(p. 102\)](#).

Figure 8.6: TimeSteps profile for Max Time Step Profile

Chemistry Solver: Simulation Controls → Chemistry Solver is also kept at the defaults for the following solver tolerances:

Do not activate **Dynamic Adaptive Chemistry**. DAC should not be used for this case since there are fewer than 500 species. For cases with 500+ species, DAC may provide additional speedup in the simulation.

- Use **Dynamic Cell Clustering** to take advantage of groups of cells with similar conditions. Select 2 features to introduce Dynamic Cell Clustering: 1) **Max. Temperature Dispersion of 10 K** and 2) a **Max. Equilibrium Ratio Dispersion of 0.05**.
- To increase the time-to-solution speed, you have the ability to choose when chemistry is activated. In this tutorial, select **Activate Chemistry Conditionally**, turn ON (check) **After Fuel Injection Starts**, and select **When Temperature is Reached** with **Threshold Temperature = 600 K**. This ensures that chemistry is active during the time that combustion is expected. Click **Apply**.

8.2.8. Output Controls

Output controls determine what data are stored for viewing during the simulation and for creating plots, graphs, and animations in ANSYS Forte Visualize, CFD-Post, EnSight, or FieldView.

The following species are named in both the **Spatially Resolved** and **Spatially Averaged and Spray** Output Control Editor panels, to be written to the results file. Fuel, oxidizer, and common emissions species are automatically added. Move these species into the **Selection** list.

- n2
- o2
- co2
- h2o

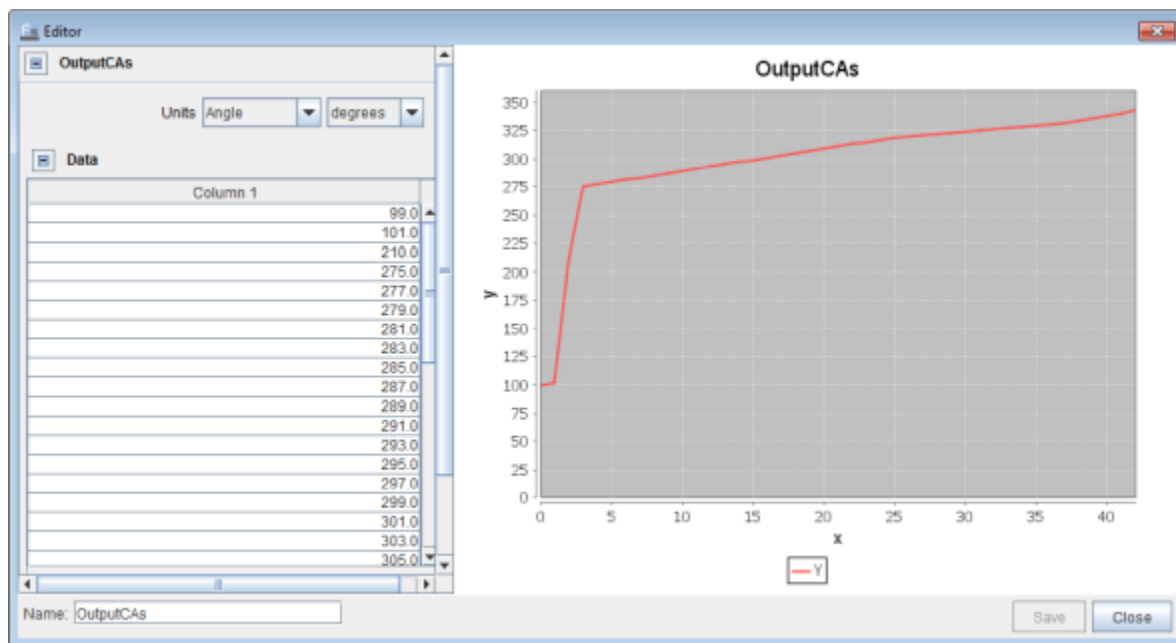
- co
- no
- no2
- ic8h18

The spatially resolved values are output at intervals of **5 crank angles**, and the spatially averaged values every **1.0 crank angle**. On each of the panels, click **Apply**.

Output Controls → Spatially Resolved: On this Editor panel, you can also optionally increase the frequency of output during the cycle by selecting **User Defined Output Control** and importing the **output_crank_angles.csv** file, which has a list of specific crank angles where spatially resolved output will occur. Specify the name of the profile as **output_crank_angles**. Alternatively, you could select **User Defined Output Control**, and use the Profile Editor to create the same profile, or some other file specifying an output crank-angle profile (the provided profile is illustrated in [Figure 8.7: Output CAs profile for Spatially Resolved output](#) (p. 103)). Click **Save** once the profile is imported and the name is specified.

Output Controls → Spatially Averaged: This Editor panel allows you to control the output of values that are averaged across the domain. Keep the default species and solution variables selected for output. Reducing the variables selected will help reduce file sizes.

Figure 8.7: Output CAs profile for Spatially Resolved output




Restart Data: If you anticipate that the case will be stopped and you want the ability to restart it from the last time step solved, select Output Controls → Restart Data. You can specify certain Restart Points using a separate file. Turn ON (checkmark) **User Defined Restart Points** and use the Profile Editor to create a Restart profile. You can view, edit, or import new Restart Points in this 1-D Profile Editor. Sometimes it is helpful in spark ignition cases to save a restart file after IVC but before the spark occurs (CA=687 in this case) so you can use the compression portion of the cycle as a start point in additional runs. Create a new profile for this purpose called **Restart_crank_angles** and add the lines with **CA** value set to **274.0, 318.0, 400.0 CA degrees**. Click **Save**.

8.2.9. Preview Simulation

You can view the profiles for accurate boundary motion and appropriate mesh quality with the Preview Simulation options.

Boundary Motion: Preview boundary motion with the player buttons on the Boundary Motion Editor panel or by right-clicking the Boundary Motion item in the Workflow tree and selecting a command. During the preview, a dotted line tracker provides a representation of the piston translation and the location of the injection and spark events (in the Editor panel), and the piston's motion (in the 3-D View if visibility settings are appropriate).


Preview Mesh: As a method for checking the automatically generated mesh, you can generate a Preview Mesh. Select Preview Simulation → Mesh Generation and then click the **New Automatic Mesh Plot**  icon, name this new automatic mesh generation plot **Start**, select **Crank Angle** as the **Time Option** and set it to **98.0 CA**. Then click **Apply**.

Copy



and **Paste**



the Start mesh twice. Name the first copy **Open** with **Start = 127.0 degrees** and **X, Y, Z = 0.0 cm**. Name the third mesh **TDC** with **Start = 360.0 degrees** and **X, Y, Z = 0.0 cm**. Now for each of the three meshes in turn, click the **Generate Mesh**  icon. ANSYS Forte will generate the preview mesh and display it in the 3-D View window. It is a good practice to look at meshes at key points in the cycle such as Firing Top Dead Center (FTDC), Exhaust Valve Opening (EVO), Intake Valve Closed (IVC), Intake Valve Open (IVO) and Exhaust Valve Closed (EVC).

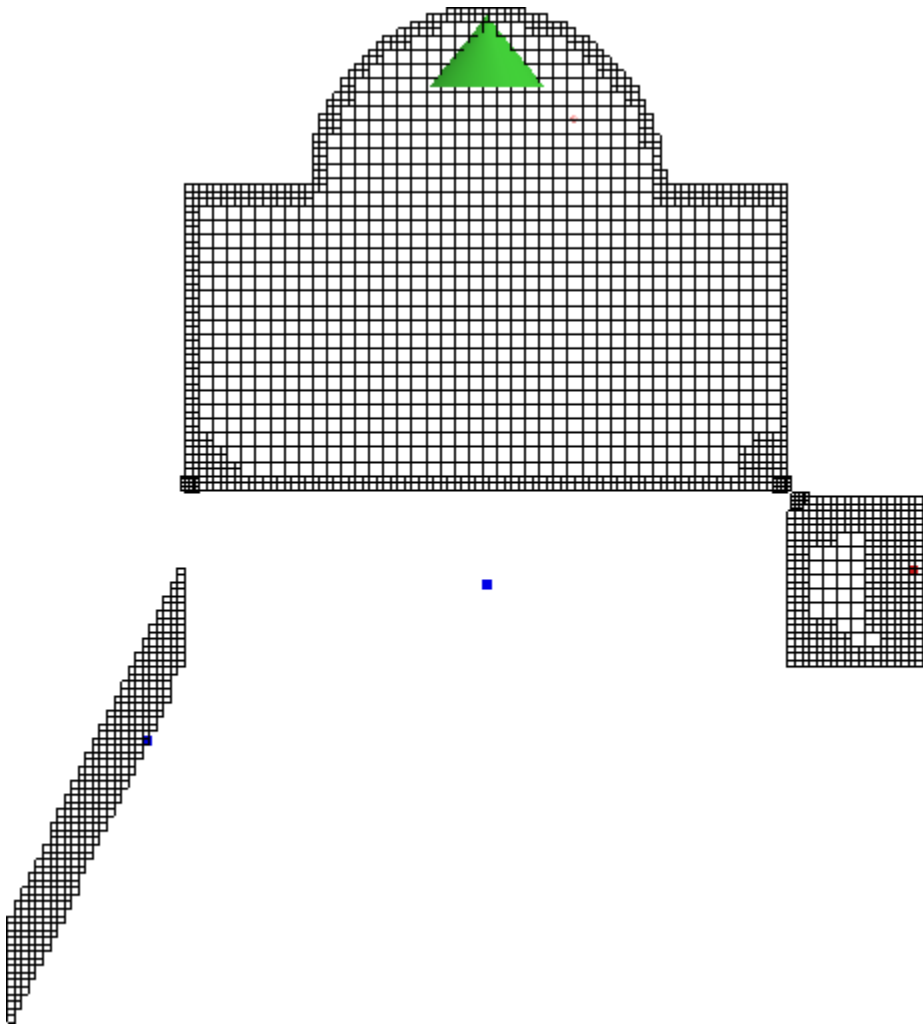
If you want to see the cut plane where the mesh will be generated, click the **Plane Filter** box and specify an origin point and normal direction for the cut plane. This adjustment has been added to the three meshes in this example. To create a cut plane, on each Mesh Generation Editor panel, turn ON (check mark) the **Plane Filter** box, and set the values as specified in [Table 8.6: Specifications for Cut Planes in Mesh Preview Editor panels](#) (p. 104)

Table 8.6: Specifications for Cut Planes in Mesh Preview Editor panels

	Start mesh	Open mesh	TDC mesh
Setting for Mesh Preview			
Start (CA degrees)	98.0	127.0	360.0
Settings for Cut Plane			
Plane Filter > Point > Ref. Frame > Global Origin > Cartesian	X=-0.0, Y=0.0-2.3, Z=0.0	X=-0.0, Y=0.0, Z=0.0	X=-0.0, Y=0.0, Z=0.0
Plane Filter > Normal > Ref. Frame > Global Origin > Cartesian	X=-0.0, Y=1.0, Z=0.0	X=-0.0, Y=1.0, Z=0.0	X=-0.0, Y=1.0, Z=0.0

The faces for the mesh have been hidden such that only the mesh lines are shown. Also, the boundary condition and geometry surfaces have been hidden. [Figure 8.8: Preview mesh at 98 CA degrees, the start of the simulation \(p. 105\)](#)

Figure 8.8: Preview mesh at 98 CA degrees, the start of the simulation



8.2.10. Run Settings

The settings here depend on the system and environment for your simulations. The default for the Run Settings panel is to have nothing selected.

Run Options: This tutorial does not require changes to this panel's defaults; adjust them as necessary for your environment.

Windows Settings: This tutorial does not require changes to this panel's defaults; adjust them as necessary for your environment.

Linux Settings: This tutorial does not require changes to this panel's defaults; adjust them as necessary for your environment.

8.2.11. Run Simulation

The last step is to submit the simulation to the solver or prepare for submission on a cluster.


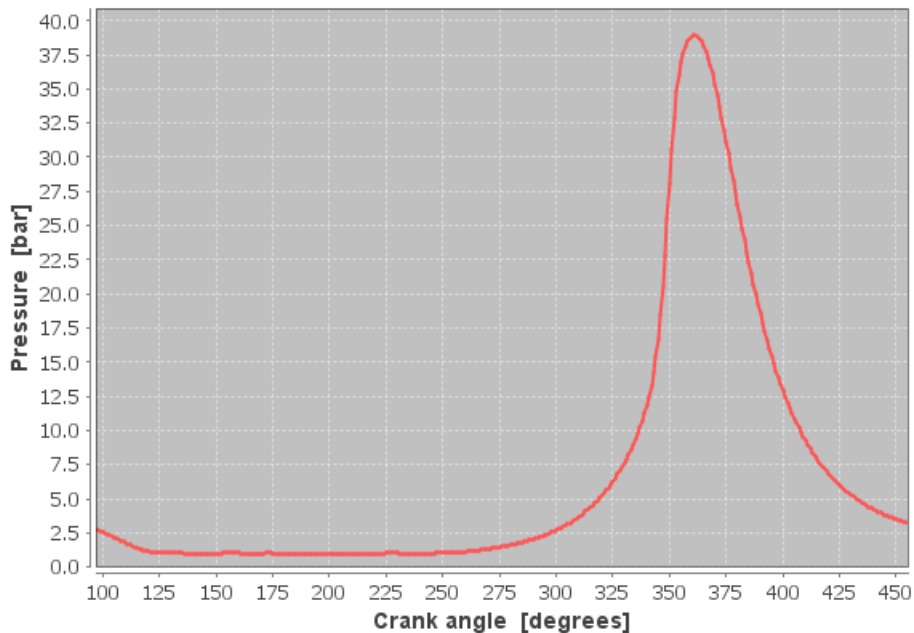
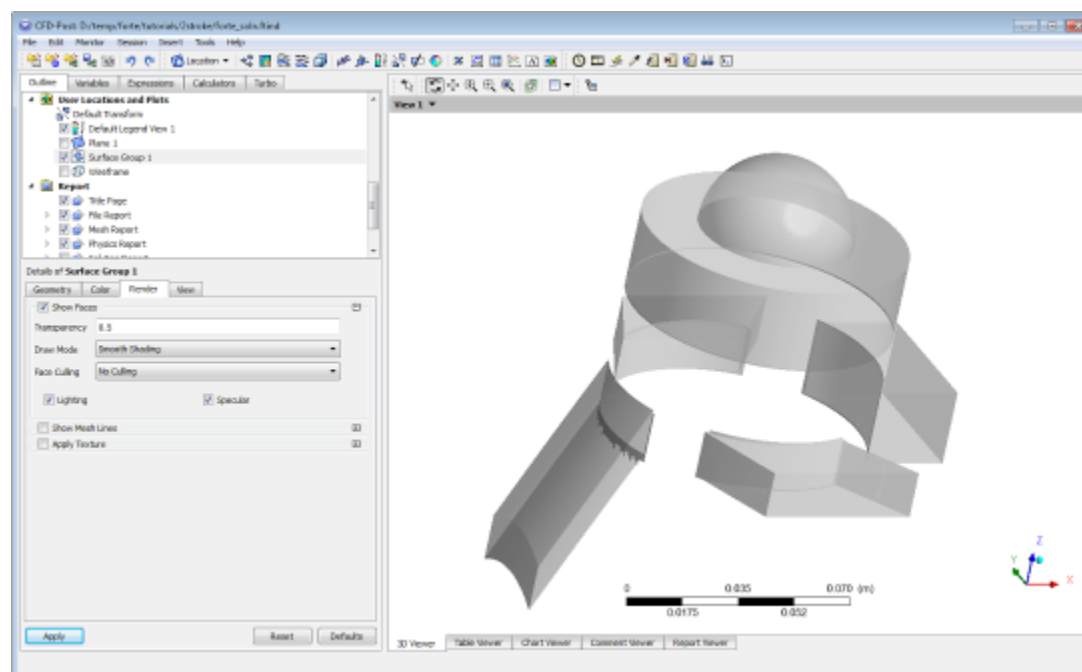
To complete the lesson, select Run Simulation on the Workflow tree and, once ANSYS Forte displays the green **Start** button on the Run Simulation panel and reports a “Ready” status, click **Start**. You can monitor the results by clicking on the **Monitor Runs**  icon. In the Monitor window that opens, you can select the run you want to monitor. The pressure trace should look like the one in [Figure 8.9: Monitoring the pressure from Run Simulation \(p. 106\)](#). This simulation takes about 11 hours to complete on 20 cores (Intel® Xeon® E5-2690 @2.9GHz).

Figure 8.9: Monitoring the pressure from Run Simulation



8.3. Project Results

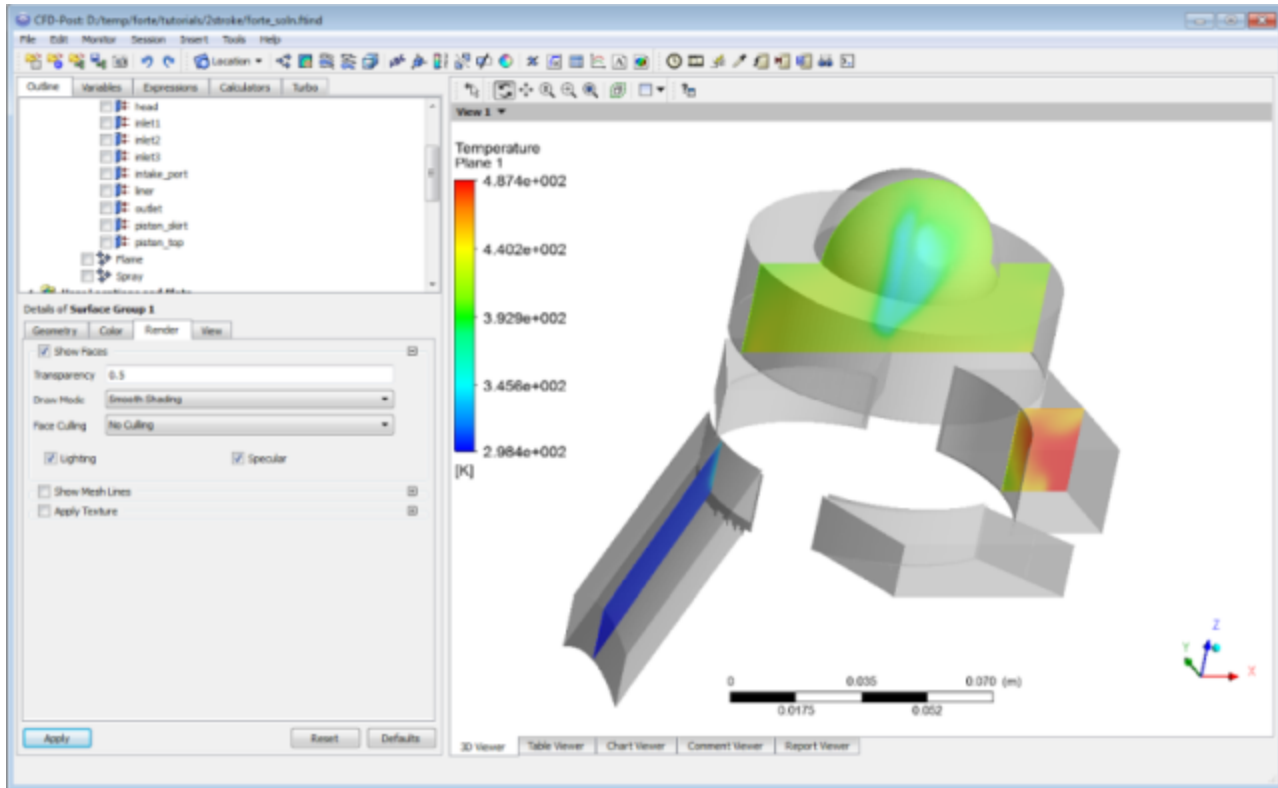
To view the results of the simulation, open the results in CFD-Post. In CFD-Post, open the solution file for the case (**Nominal.ftind**). [Figure 8.10: Screen view of the solution file in CFD-Post \(p. 107\)](#) shows the screen view once the solution file has been loaded.

Figure 8.10: Screen view of the solution file in CFD-Post

Once you have read in the model, you can create surface groups to group the surfaces. In this case, we will create a surface group for the spark plug, walls, symmetry boundaries, and the valves. To create a surface group, right-click **User Locations and Plots** **Insert > Locations > Surface Group** and specify the name **surfaces**. On the **Color and Render** tab, you can change the color of the surfaces and the transparency. In [Figure 8.10: Screen view of the solution file in CFD-Post \(p. 107\)](#), the surfaces are colored gray and the transparency is set to a value of 0.5.

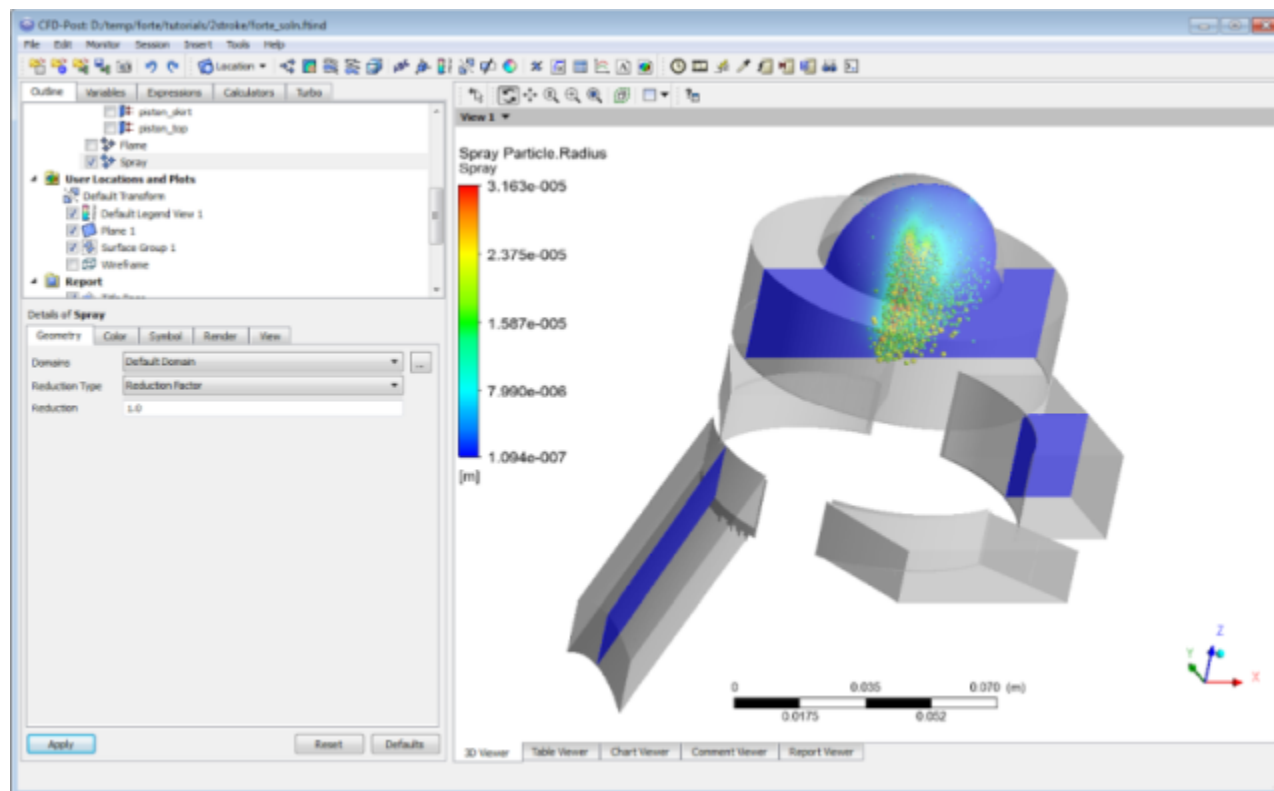
To visualize the solution in the combustion chamber, create a cut plane by right-clicking **User Locations and Plots > Insert > Location > Plane**. Specify the **Method** as a **ZX plane**. On the **Color** tab, set the **Mode** to **Variable** and specify the **Variable** as **Temperature**, click **Apply**. The temperature is now displayed on the cut plane, as shown in [Figure 8.11: Temperature contour plot on a plane through the centerline of the combustion chamber \(p. 108\)](#).

Figure 8.11: Temperature contour plot on a plane through the centerline of the combustion chamber

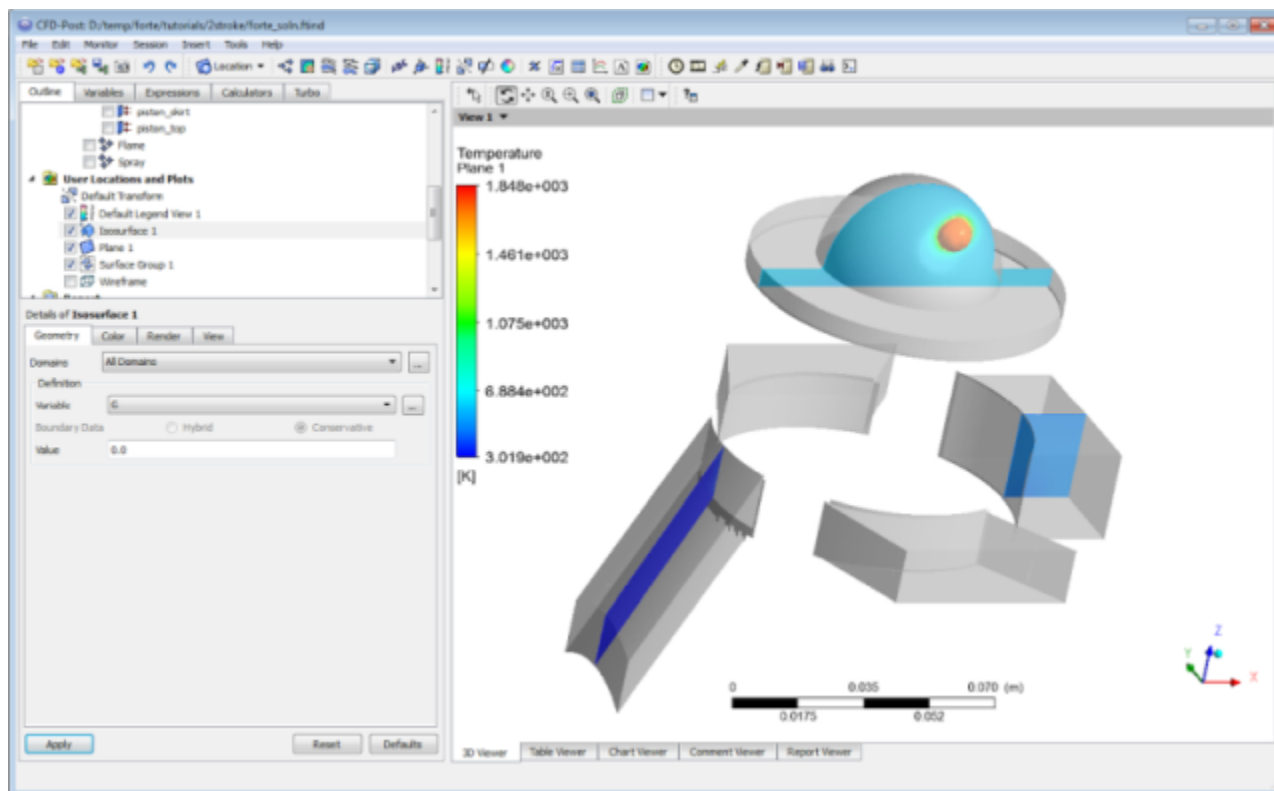


To visualize the spray injection into the chamber, activate the **Spray** option. Double-click **Spray** to edit the settings. On the Color tab, set **Mode** to **Variable** and select **Spray Particle Radius**. Set the **Range** to **Local**. To visualize the spray at a specific crank angle, go to **Tools > Timestep Selector**. In this case, crank angle **291.02** is specified.

Figure 8.12: Spray particles in the chamber at 291 CA degrees



Finally, to visualize the position of the flame, we need to create an iso-surface. To create the iso-surface, right-click **User Locations and Plots** and select **Insert > Location > Isosurface**. Specify **flame** as the name of the surface. Specify **G** as the variable and specify **red** as the color for the surface on the Color tab. To see the flame location, we must specify a crank angle after spark timing. Spark timing in this case is 319CA degrees, so you can use the menu command **Tools > Timestep Selector** to select a crank angle after spark timing.

Figure 8.13: Isosurface at 334 CA degrees showing the location of the flame front

Chapter 9: Multi-Cylinder Four-Stroke Engine Simulation

This tutorial describes how to use ANSYS Forte CFD to simulate engine processes in an inline four-cylinder four-stroke spark-ignition engine. The purpose of this tutorial is to demonstrate several key points of project setup that are specific to multi-cylinder use cases:

- Use (spatial) Reference Frames to facilitate the setup of location and orientation parameters in each cylinder.
- Use Time Frames to facilitate the setup of timing parameters in each cylinder.
- Approximate initial conditions in each cylinder region for a given starting crank angle.
- Monitor output parameters that are averaged for each cylinder region.

Although the example used is an inline four-cylinder four-stroke spark-ignition engine, the concept and methodology described in this tutorial are also applicable to other multi-cylinder configurations, such as V-shaped arrangement, opposed piston engines, as well as other types of engine operations, such as spray combustion in diesel engines, HCCI, etc.

This tutorial is based on a fully configured Forte project (.ftsim) file. The steps described only highlight the key points specific to multi-cylinder engines and are not intended to explain all the entire detailed setup. The tutorial assumes that learner is already familiar with the fundamental steps for setting up a single-cylinder engine in Forte. Such prerequisite knowledge can be obtained by studying tutorials in previous chapters, such as the ANSYS Forte GDI tutorial, [Solving a Gasoline Direct Injection Engine Simulation](#) (p. 67).

9.1. Data Provided

9.1.1. Accessing Tutorial Files

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

After downloading, you have the opportunity to select the location for the files when you download and unzip the sample files.

9.1.2. Files Used in This Tutorial

The files for this tutorial include:

- **multicylinder_4stroke_engine_tutorial.ftsim**: A completely configured ANSYS Forte project file.
- **intake_valve_lift_profile.csv**: Lift profiles of the intake valves, tabulated for the crank angle range of [0, 720] °CA ATDC.
- **exhaust_valve_lift_profile.csv**: Lift profiles of the exhaust valves, tabulated for the crank angle range of [0, 720] °CA ATDC.

- **multicylinder_4stroke_engine_tutorial.stl**: An STL file that contains the multi-cylinder geometry. This is an optional file, which could be used as part of an exercise setting up this Forte project without using pre-configured files.

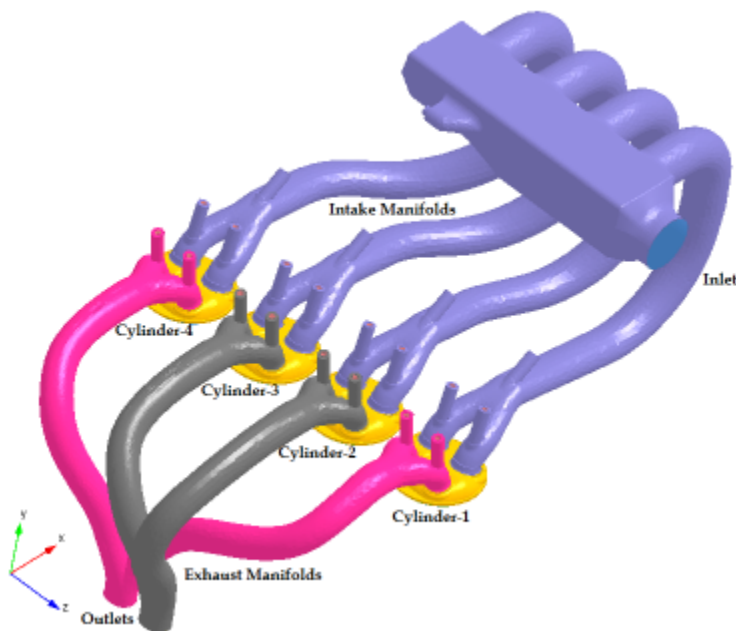
9.2. Engine Configuration and Problem Description

The configuration of the inline four-cylinder engine is shown in [Figure 9.1: Configuration of the inline four-stroke multi-cylinder engine \(p. 112\)](#). The figure shows the numbering of the four cylinders, the intake manifolds merging at a plenum, the inlet located on the side of the plenum, as well as two groups of exhaust manifolds. One group of exhaust manifolds is connected to Cylinder-1 and Cylinder-4, the other is connected to Cylinder-2 and Cylinder-3. In this geometry, the crank shaft axis of the engine (not shown) is parallel to the Z-axis and the cylinder axes are parallel to the Y-axis. If you prefer, you can rotate the geometry so the cylinder axes are parallel to the z-axis. To do the rotation, select all the geometry surfaces under **Geometry**, right-click, select **Transform Mesh**, and then select **Rotate X by 90 degrees**. The setup steps described below are based on the original orientation, not a transformed geometry. The engine geometry will serve as the surface mesh for the automatic mesh generation. Automatic Mesh Generation requires that all the pistons are positioned at their top-dead-center (TDC) location and all valves are at their closed location in the initial surface mesh.

There is one spark per cylinder central-mounted near the top of each cylinder head. The firing order of this engine is **1-3-4-2**, which means that there is a 180 °CA phase delay between any two consecutive spark events.

In this tutorial, the fuel/air mixture preparation process inside the intake manifolds is simplified and no port fuel injection is simulated. Instead, the fuel and air are treated as premixed stoichiometric mixture, which enters the computational domain through the inlet connected to the plenum.

Figure 9.1: Configuration of the inline four-stroke multi-cylinder engine




9.3. Multi-Cylinder Spark-Ignition Engine Project Setup

This section will go over the simulation setup steps, with emphasis on several aspects that are specific to multi-cylinder simulation.

Note

Time frames can be added under **Geometry > Time Frames**. However, the default state for the **Time Frames** node in the Workflow tree is hidden. To see it, use the **Preferences** command in the **Edit** menu and check **Enable Reference Time Frame Controls**. Then **Exit** and restart ANSYS Forte.

9.3.1. Geometry Import

(Optional) If you wish to set up the case without using the preconfigured files, you can start with importing the supplied STL geometry file. On the **Geometry** Editor panel, click the **Import Geometry**  icon, and choose the **Surface from one or more STL files** option, and then browse for the STL file. This step is needed only if you want the exercise of setting up the case from scratch. These steps are not detailed here.

9.3.2. Defining Reference Frames and Time Frames

Reference frames and time frames in ANSYS Forte allow you to define transforms to apply in space and time.

9.3.2.1. Reference Frames

Reference frames used in ANSYS Forte are spatially transformed coordinate systems. They can be defined in the Workflow tree under **Geometry > Reference Frames**. A new reference frame can be defined by referring to its parent coordinate system and through transformation of either or both the origin and orientation. In a multi-cylinder geometry, by defining equivalent local reference frames for all cylinders, you can use the same set of location and orientation parameters for all cylinders.

To make it convenient to set up equivalent local coordinate systems for all the cylinders, the geometry must meet some basic requirements relating to alignment with the coordinate axes in the global coordinate system. Generally, the minimum requirement is to let the crank shaft axis be parallel to one of the coordinate axes in the global coordinate. This tutorial case satisfies this requirement because its crank shaft axis is parallel to the Z-axis.

Now let us define a local reference frame for each cylinder. There are no fixed rules for setting the origin of each cylinder's own reference frame, but a convenient way is to use the center of the piston as the local origin. Using **Cylinder-1** as an example, under **Geometry**, click surface **piston_1**. The **piston_1** Editor panel shows the **Min** and **Max** coordinate values along X-, Y-, Z-directions in the global reference frame. The center of **piston_1** on the X-Z plane can be calculated as

$$X_{\text{center}} = 0.5 \cdot (X_{\text{min}} + X_{\text{max}}) = 0.5 \cdot (-3.949 + 3.948) = 0 \text{ cm}$$

$$Z_{\text{center}} = 0.5 \cdot (Z_{\text{min}} + Z_{\text{max}}) = 0.5 \cdot (31.15 + 39.05) = 35.1 \text{ cm}$$

Since Y_{min} and Y_{max} are the same, you can pick Y_{min} as Y_{center} . In this specific example, it is beneficial to let all four cylinders have the same Y_{center} value.

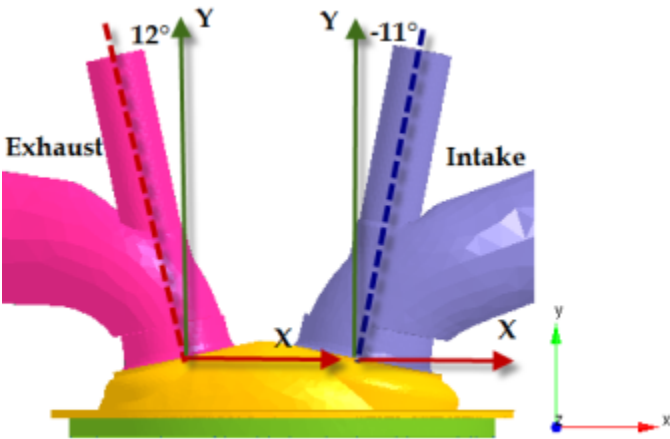
In the fully configured `.ftsim` file, you can see four **Reference Frames** for the four cylinders: **Frame-C1**, **Frame-C2**, **Frame-C3**, and **Frame-C4**, listed in the Workflow tree. Their origins are calculated based on the method described above and the values are listed in [Table 9.1: Origins of local reference frames for four cylinders \(p. 114\)](#). We can see that the distance along the Z-direction between the axes of two consecutive cylinders is 9.5 cm. These four reference frames do not involve orientation transformation. Orientation transformation is needed in a V-type multi-cylinder engine.

Table 9.1: Origins of local reference frames for four cylinders

(cm)	Frame-C1	Frame-C2	Frame-C3	Frame-C4
Origin X	0	0	0	0
Origin Y	19.7	19.7	19.7	19.7
Origin Z	35.1	25.6	16.1	6.6

In this tutorial engine, all the intake valves are tilted on the X-Y plane by -11° relative to the Y-axis, and all the exhaust valves are tilted in a similar manner, but by 12° . The tilt angles are illustrated in [Figure 9.2: Tilt angles of intake and exhaust valve stems \(p. 114\)](#). Two reference frames are created for intake valves and exhaust valves, respectively, such that the Y-axes of these two reference frames are aligned with the moving directions of the valves. The names are `Frame_Intake_Valves_Dir` and `Frame_Exhaust_Valves_Dir`. The transformation only involves rotation about the Z-axis. These two reference frames can be used to define the moving directions of the valves. They can also be used to adjust the valve gaps at seated position if needed.

Figure 9.2: Tilt angles of intake and exhaust valve stems



9.3.2.2. Time Frames

Note

If you cannot see **Time Frames** listed under **Geometry** in the Workflow tree, edit your Preferences as described in the note in [Note \(p. 113\)](#) early in this tutorial.

Time frames are convenient tools for managing all the timing parameters in a multi-cylinder context. A time frame requires input of a crank angle offset. An event associated with the time frame will be **delayed** by the offset value relative to the global crank angle. By defining an appropriate time frame for each cylinder, you can use the same set of timing parameters for all the cylinders in the configuration.

Such timing parameters include, for example, piston motion specification and valve lift profiles, spark timing, and injection timing.

The **Global Time** is a prepopulated item, which corresponds to the global crank angle and serves as a reference. It cannot be edited. You can add four new time frames, one for each cylinder. Since the firing order of this engine is 1-3-4-2, the crank angle offset values can be set as **0 °CA, 180 °CA, 360 °CA, 540 °CA** for **Cylinder 1, 3, 4, 2**, respectively. Note that **Cylinder-1** is essentially associated with the global crank angle, and therefore it is used as a reference cylinder.

9.3.3. Automatic Mesh Generation Setup

A multi-cylinder case requires that one of the cylinders be defined as the reference cylinder and associated with the **Default Initialization** region under **Initial Conditions**. The **Material Point** under **Mesh Controls** is expected to be a location inside the **Default Initialization** region. Since **Cylinder-1** is assigned as the reference cylinder in this tutorial, the material point should be inside **Cylinder-1**. As mentioned earlier, since the origin of the reference frame for **Cylinder-1** is defined as the center of the piston surface at the piston's TDC position, you can set the material point a distance above the local origin. The location selected should be able to remain inside the combustion chamber at all times while the piston travels from its BDC to TDC and the valves move from a fully closed position to a fully lifted position. In this sample, the **Material Point** is set to **(X, Y, Z) = (0.0, 1.0, 0.0) cm** in the **Frame-C1** reference frame. The **Global Mesh Size** is set to **4.0 mm**.

Several refinement controls for surface or volume are used to obtain higher mesh resolution at system boundaries and at various locations that are expected to see high gradients for velocity or temperature.

You can go through the refinement control items under **Mesh Controls** in the fully configured project file to check their detailed specifications.

Under **Mesh Controls** on the Workflow tree, **AllWalls**, **OpenBoundaries**, **ValveStems**, **Chamber_BL**, and **ValveSeats** are surface refinement controls that are applied at all times. **ValveSeats** specifies a **1/8** refinement for all valve seat surfaces in all the cylinders. In a typical engine geometry, cells near the valve gaps must be sufficiently refined (approximately 0.5 mm) to ensure that the computational domain of the intake/exhaust manifolds and the cylinders are cleanly separated when the corresponding valves are closed.

IVO_n and **EVO_n**, in which *n* is a numeral, are interval-based surface refinements applied to the combustion chamber walls during intake-valve opening or exhaust-valve opening processes. The cylinder-specific time frames are convenient when setting up these crank-angle-interval-based controls. By associating such a control with a cylinder's local time frame, you can specify the start angle and end angle values with respect to the cylinder's local TDC, and the same crank angle values can be used for all the cylinders. For example, the crank angle interval for refinement controls **IVO_1** to **IVO_4** has the same value, **[323, 563] °CA** ATDC. The only difference for this timing control is the time-frame selection. Note that the **Location** should point to the corresponding surfaces in each cylinder.

SparkPlug_1 to **SparkPlug_4** are spherical volume refinement controls that specify **1/8** refinement in a small region surrounding each spark plug. The center of the refinement volume will be set at the spark location and the **Radius of Application** is **0.4 cm**. By specifying the coordinates of the refinement

center relative to the origin of each cylinder's local reference frame, the same center coordinates can be used for these four controls.

Note


As an option, you can limit this refinement to 10 °CA before spark and 60 °CA degree after spark in each cylinder.



9.3.4. Model Setup

Chemistry: The single-component 59-species gasoline chemical kinetics mechanism that is included in Forte's chemistry mechanism database is used to simulate the combustion of gasoline/air mixture, in which gasoline is approximated by iso-octane. The chemistry set can be found in the `data` folder within the Forte installation. The name of the chemistry set is `Gasoline_1comp_59sp.cks`.

Flame Speed: The Gülder formulation of the power law will be used to compute laminar flame speeds in the flame propagation model. For all the other model inputs on the **Flame Speed Model** panel, the default values are used.

Spark Ignition: Turn on the spark ignition model by checking the checkbox in front of **Spark Ignition** in the Workflow tree. On the **Spark Ignition > Settings** panel, the checkbox in front of **Flame Propagation Model** is checked by default. This means the flame propagation will be modeled using the discrete particle ignition kernel (DPIK) model and the G-equation model. Keep the default values for all the model constants.

Sparks: In this engine, each cylinder contains one spark plug. The spark plug structure is included in the cylinder head surfaces. On the **SparkIgnition** Editor panel, you can create four spark plug items to specify their location, spark duration, energy discharge rate, etc. Start with adding the spark plug for the first cylinder by clicking the **Spark**  icon on the **SparkIgnition** Editor panel. By specifying the spatial reference frame as **Frame-C1**, the coordinates of the spark plug's location can be specified with respect to this cylinder's local reference frame. The coordinates of the spark gap are **(-0.4, 1.35, 0.0) cm**. Similarly, by linking this spark's timing parameter to **Time Frame Cylinder 1**, the spark timing value can be specified with respect to this cylinder's local TDC. The spark timing is 20°CA before the firing TDC, so the starting angle of spark is set to **700°CA** ATDC. The default values are kept for spark energy and initial kernel radius. After the first spark, **Spark 1**, has been configured, you can then duplicate it

(Copy  and Paste ) to create the spark plugs in the other three cylinders. After each copy and paste, you can simply adjust the **Time Frame** and (spatial) **Reference Frame** to their respective local frames.

Spray: (Optional) This tutorial does not involve spray simulation, but fuel sprays within each cylinder or within the intake ports of each cylinder could be set up in a similar manner as the spark setup. You can start with configuring the injector for the reference cylinder, and then duplicate that injector for the other cylinders. For each duplicated injector, you must adjust the time frame as well as the reference frames for each nozzle to use the cylinder's local frames. Note that the time frame is applied on the **Injector** level, that is, the injection timings of all the **Injection** pulses added under an injector are assumed to use the same time frame specified on the **Injector** Editor panel.

9.3.5. Boundary Conditions

In the context of multi-cylinder engines, the boundary conditions can be divided into two groups, one is shared by all cylinders and the other is cylinder-specific. The cylinder-specific boundary conditions

include head, liner, and moving surfaces (pistons and valves). For the cylinder-specific boundary conditions, you can first do the configuration for the reference cylinder, and then duplicate that for the rest of the cylinders. For each duplicated boundary condition item, the only parameters that must be adjusted are the **Ref. Frames** (for locations or directions) and **Time Frames** (for timings). Of course, the **Location** selection should be adjusted accordingly as well.

In this tutorial, boundary conditions shared by all the cylinders include **Inlet**, **Outlet_1**, **Outlet_2**, **IntakeManifold**, **ExhaustManifold1**, and **ExhaustManifold2**. The first three are open boundaries, which involve time profiles for boundary pressure and temperature. The time frame associated with these time profiles should be set to **Global Time**. In this tutorial, port fuel injection will not be simulated. Instead, the mixture coming into the domain through the inlet is assumed to be a premixed stoichiometric air/fuel mixture.

The boundary conditions **head** and **liner** are static wall boundaries. They are cylinder-specific, but do not require a time frame input, therefore the head surfaces of all cylinders are grouped as one boundary condition and the same is done for the liner surfaces. If preferred, you can split each group into four individual boundary condition items. Such a split may be needed if you want to define different wall boundary temperatures for different cylinders.

Each moving surface requires the specification of a **Wall Motion**. The wall motion for pistons and valves in different cylinders can easily be synchronized with the help of the cylinder-specific time frames. In the fully configured ANSYS Forte project file, **Piston_1**, **IntakeValve1-1**, **IntakeValve2-1**, **ExhaustValve1-1**, **ExhaustValve2-1** are the moving boundary conditions for **Cylinder_1** (the last numeral in the BC name indicates the cylinder index). If setting up the project on your own, you could first create these five boundary conditions, specify the **Time Frame** as **Cylinder 1**, and specify the **Ref. Frame** used by the wall motion direction as **Frame-C1**. Then, duplicate these BCs for the other cylinders.

For a duplicated piston, adjust three items to reflect the local cylinder:

- **Location** selection;
- **TimeFrame**; and
- **Ref. Frame** (for motion direction).

For a duplicated valve, adjust four items:

- **Location** selection;
- **Time Frame** (for lift profiles);
- **Ref. Frame** (for motion direction); and
- the selection of **Valve Seat and the Surface it Contacts**.

All the other parameters can be kept the same for different cylinders, including the valve lift profiles.

Note that **Frame-C1** to **Frame-C4** are used to specify the valve motion direction in the current project file. An alternative (probably easier) way is to use the **Ref. Frames** mentioned earlier, **Frame_Intake_Valves_Dir** and **Frame_Exhaust_Valves_Dir**. For example, when using **Frame_Intake_Valves_Dir** to specify the moving direction of the intake valves, you can simply specify the Cartesian direction vector as $(X, Y, Z) = (0, -1, 0)$. The same motion direction setting can be applied to all intake valves because they all move along the same direction. The same approach can be applied to the exhaust valves.

9.3.6. Initial Conditions

A multi-cylinder simulation should intrinsically also be a multi-cycle simulation. This is because the cylinders will be in different stages (strokes) of an engine cycle no matter where the simulation starts, and it will take multiple engine cycles to let the simulation reach a “converged” state starting from approximated initial conditions. Nonetheless, it is beneficial to provide a good estimate for initial conditions as this can help the simulation approach the “convergence” faster.

According to the lift profiles of the intake and exhaust valves and the minimum valve lift distance, the intake valves are open in crank angle interval [380, 580] °CA ATDC and the exhaust valves are open in [172, 380] °CA ATDC in each cylinder’s local time frame. Thus, the firing TDC of this engine is 0°CA ATDC (or 720°CA ATDC). In this tutorial, we will set the starting crank angle to 0°CA ATDC. The starting angle is the global crank angle, which is also in sync with the local crank angle in Cylinder-1. At this starting angle, the piston positions and strokes each cylinder is about to start are listed in [Table 9.2: Piston positions and strokes to begin in each cylinder \(p. 118\)](#):

Table 9.2: Piston positions and strokes to begin in each cylinder

	Piston Position	Stroke about to start
Cylinder 1	Firing TDC	Expansion/Work
Cylinder 2	BDC (180° after Firing TDC)	Exhaust
Cylinder 3	BDC (180° before Firing TDC)	Compression
Cylinder 4	Non-Firing TDC	Intake

ANSYS Forte requires that the reference cylinder (Cylinder-1 in this case) be defined as the **Default Initialization** region. This default region will use the global material point for mesh creation and thus does not require a separate material point input. In addition, the **Region Type** of the Default Initialization region is assumed to be Cylinder/Primary, and thus it does not require a **Region Type** input.

Chamber-2, Chamber-3, and Chamber-4 are three new regions for the other three cylinders, and they are based on material points. By linking these regions to their respective spatial **Ref. Frames**, the material point coordinates can have the same value as the coordinates of the global material point. Note that the global **Material Point** under **Mesh Controls** in the Workflow tree is linked to **Ref. Frame Frame-C1**. For these three non-default cylinder regions, the **Region Type** should be specified as **Cylinder/Primary**. Note that the cylinder numbering will follow the order as they appear on the Workflow tree under **Initial Conditions**.

At the starting crank angle, Cylinder-1 is beginning the expansion stroke, the mixture is approximated as complete combustion products, the temperature and pressure are also approximated as peak values during combustion, 2000 K and 20 bar, respectively. Cylinder-2 is starting the exhaust stroke, so the composition is also set to complete combustion products, but the temperature and pressures are lower than those in Cylinder-1. They are set to 1200 K and 1.5 bar, respectively. Cylinder-3 is starting the compression stroke. Physically, the mixture inside this cylinder should be a fuel/air mixture mixed with a small fraction of residual gas. For simplicity, the initial composition is approximated as a stoichiometric fuel/air mixture. The temperature and pressure in Cylinder-3 should be similar to the values in the intake manifold. They are set to 340 K and 1 bar, respectively. Cylinder-4 is at the non-firing TDC and starting the intake stroke, the composition is mainly combustion products. The temperature and pressure are approximated as 600 K and 1 bar. The default settings are used in the fields for initial turbulence parameters and initial velocity.

Intake Port, ExhaustManifold_1, and ExhaustManifold_2 are three regions with **Region Type** defined as **Other**. For **Intake Port**, the initial composition is approximated as stoichiometric air/fuel mixture.

Temperature and **Pressure** are set to **313 K** and **1.1 bar**, respectively. For the two exhaust ports, composition is set to combustion products, and **Temperature** and **Pressure** are set to **500 K** and **1 bar**, respectively. The material points can be placed anywhere inside the respective manifold geometry.

The **Initialization Order** values of all the initialization regions should be set following the “go-with-the-flow” rule. The order of the intake port should be 1, followed by the four cylinders, and finally the two exhaust manifolds.

The **Composition Calculation Utility** contained in the ANSYS Forte Setup Interface can be used to prepare different mixtures required by the initial conditions.

9.3.7. Simulation Controls

A multi-cylinder engine simulation must use the **Crank Angle Based** option on the **Simulation Controls** Editor panel. The initial and final crank angles exposed on the **Simulation Controls** panel are both global values. In the fully configured project, the case is set up to run two complete engine cycles.

On the **Time Step** Editor panel, the **Max Simulation Time Step**, dtmax, uses a constant value (**1.E-5 s**) in the current project. If a time varying dtmax profile is used instead, the profile is required to cover the crank angle range of [0, 720] °CA ATDC. This profile will be applied to all cylinders within each cylinder’s local time frame. Specifically, a max time step value is first determined in each cylinder separately based on the profile and the cylinder’s local crank angle value. Then, the minimum value among all the cylinders will be used as the global **Max Simulation Time Step**.

For the chemistry activation control on the **Chemistry Solver** panel, it is recommended not to check the **After First Spark Event** and **After Fuel Injection Starts** boxes because these controls are not well-defined in the multi-cycle, multi-cylinder context. Instead, you can use the **During Crank Angle Interval** option to activate chemistry for a certain duration within an engine cycle. Similar to the **Max Simulation Time Step** control, the interval-based chemistry activation control is also assumed to be applied in all the cylinders based on each cylinder’s local time frame. For example, in this tutorial, the interval is set to [-40, 100] °CA ATDC. This means chemistry will be activated 40°CA before the Firing TDC and deactivated 100°CA after the Firing TDC in all four cylinders.

9.4. Result Visualization

In a multi-cylinder calculation, cylinder-specific spatially-averaged solutions are reported on a per-cylinder basis. The cylinder-specific CSV output files include:

- thermos.csv
- dynamic.csv
- flame.csv
- massfraction.csv
- molefraction.csv
- speciesmass.csv

The cylinder index will be appended to the file names to indicate which cylinder they are for. For example, you will see thermo1.csv, thermo2.csv, etc. These files include both the global crank angle and the cylinder’s local crank angle. The local crank angles are always converted to values within the range

of $[0, 720)$ °CA ATDC. For two-stroke engines, the local crank angles will be in the range of $[0, 360)$ °CA ATDC.

The rest of the CSV files are reported either based on the average over the whole computational domain or reported on a per-boundary-condition basis, and they have the same format as single-cylinder calculations.