Applying Solution-Adaptive Mesh Refinement in Engine Simulations

Long Liang, Yue Wang, Anthony Shelburn, Cheng Wang, Abhijit Modak, Ellen Meeks

ANSYS Inc.
5930 Cornerstone Court West, Suite 230, San Diego, CA 92121

1. Introduction

A new solution-adaptive mesh refinement capability has been implemented in ANSYS Forte CFD. In this paper, we discuss its implementation and application to engine simulations. This feature is built upon the automatic meshing framework in Forte. The automatic and on-the-fly mesh generation capability in Forte generates a Cartesian cell mesh and the solver uses an immersed boundary method. The mesh generation employs an octree data structure to represent the computational cells inside the computational domain, and all computational cells are perfectly orthogonal. In many scenarios of engine application, high mesh resolution is required to resolve fine geometrical structures or sharp gradients in a physical model. It is impractical and often unnecessary to apply tiny cells everywhere inside the whole domain; thus adaptive mesh refinement and coarsening can be important for obtaining high local mesh resolution while keeping the computational cost low.

There are three types of refinement controls implemented in Forte:

1) Fixed Mesh Refinement – Cells along certain boundary surfaces or in predefined volumes are refined to a user-specified level during specified time or crank-angle intervals.

2) Geometry-Adaptive Mesh Refinement – This control is mainly concerned with moving walls. For example, the cell size within valve gaps is dynamically controlled based on the valve lift profiles and user-specified minimum lift thresholds and minimum number of cells in the gap.

3) Solution-Adaptive Mesh Refinement – In this control, cells are adaptively refined or coarsened based on user-specified solution parameters (or their gradients) following certain refinement criteria. This control can help apply high resolution at locations where the high resolution is most needed. We refer to this control as SAM in this paper.

In typical engine cases, the first two types of refinement are always used. In this work we focus on the implementation and application of solution-adaptive mesh refinement (SAM).

Our first objective is to demonstrate the usage of SAM in different modeling components involved in engine simulations, including basic flow solution, spark ignition and flame propagation, and sprays. We explore different SAM control parameters, demonstrating that refinement is applied at expected locations, and then derive best practices for SAM application. A second objective is to take advantage of SAM’s flexibility to study the mesh sensitivity and mesh convergence of several key sub-models in Forte. A natural question about mesh refinement is: how much should the mesh be refined? The answer will certainly vary for different CFD implementations, mesh types, and sub-models. In this paper, we use SAM to explore the mesh sensitivity of the flame propagation model and spray model. From this we provide guidance for users to select cost effective refinement levels in their engine applications.

2. Methodology

2.1 Solution-Adaptive Mesh Refinement (SAM)

Solution adaptive meshing allows refining certain regions in the mesh based on a solution field (or gradient) at the current time step. Each SAM control includes a solution or gradient field, a refinement criterion, and a refinement level. Cells meeting the selected criteria will be refined one level per mesh update, up to the maximum depth specified in the control. Previously refined cells that no longer meet the criteria will be coarsened one level per mesh update, up to the default cell depth, unless influenced by another control.

Three types of criteria are available in Forte:

1) Absolute Value

This criterion allows user to specify an absolute range of a selected parameter. Cells whose field or gradient value is greater than the given lower bound and less than the given upper bound will be targeted for refinement. This option requires a-priori knowledge. It is useful and will behave well if this a-priori knowledge is available. For example, for the combustion simulation in a premixed charge SI engine, the mass fraction range of the combustion products can be pre-estimated. Thus, the mass fraction range of CO\(_2\) can be used in a SAM control to mesh the volume around the flame front using smaller cells. For other cases, such a-priori knowledge may be difficult to obtain.

2) Percentile

This control will target cells whose field value lies within the user-specified parameter percentile range, where the range is defined as follows:

\[ x_{low} = x_{min} + \frac{\text{percentile}_{low}}{100} \cdot (x_{max} - x_{min}) \]
\[ x_{high} = x_{min} + \frac{\text{percentile}_{high}}{100} \cdot (x_{max} - x_{min}) \]

The percentile control is more flexible than the absolute value control because it dynamically tracks the parameter range. However, the bandwidth of this control may not capture the degree of stratification of the selected control parameter, such that the percentile threshold values may require case-by-case adjustment.
3) Statistical
This control targets cells whose field value lies beyond a user-specified statistical significance of the bulk fluid value, given as a multiple of the standard deviation of the field or gradient average:

\[ x_{\text{threshold}} = x_{\text{mean}} + \sigma_{\text{threshold}} \cdot \sigma \]

where \( \sigma \) is standard deviation, and \( \sigma_{\text{threshold}} \) is the user-specified multiplier. This approach tracks the mean value of the control parameter well, and also dynamically adjusts the refinement targets based on the degree of stratification of the selected control parameter. This option is especially useful when applied to gradients. A \( \sigma_{\text{threshold}} \) value between 0.5 and 1.0 is typically adequate based on our experience. A lower value lowers the threshold used to target cells and thus leads to more refinement, while a higher value raises the threshold and results in less refinement.

Among these three SAM options, the statistical control is the most flexible one and requires the least amount of \textit{a-priori} knowledge. In this work, the statistical method is used in all the test cases and \( \sigma_{\text{threshold}} \) is fixed at 0.5.

2.3 Physical Models

Spark Ignition and Flame Propagation Models
Spark-ignition engines are characterized by flame initiation near the spark location followed by flame propagation into the engine cylinder. In Forte, the \( G \)-equation model is used to track the propagation of fully developed, premixed or partially premixed, turbulent flames inside the engine cylinder [2]. When the flame is initiated by the spark, the ignition-kernel flame has a structure that is typically smaller than the average grid size in the computational mesh. During this time, then, the kernel flame front is first tracked by a group of discrete flame “particles”. The calculation switches from this kernel flame model to the \( G \)-equation model after the flame structure grows bigger than a characteristic flow length scale.

The \( G \)-equation combustion model is based on the turbulent premixed or partially premixed flamelet theory by Peters [3], in which the mean flame front surface is tracked by an iso-surface of non-reacting scalar \( G \) (i.e., the \( G(x,t) = 0 \) iso-surface). In this model, a set of Favre-averaged level set equations was derived, including the equations for the Favre mean \( G \), and its variance, \( G^{\prime 2} \), as well as a model equation for the turbulent/laminar surface area ratio \( \sigma_T \), which, in turn, results in an explicit expression for the turbulent flame speed \( S_T \). These equations, together with the Reynolds averaged Navier-Stokes equations and the turbulence modeling equations, form a complete set to describe turbulent flame front propagation. In the \( G \)-equation model, the transport equations are closed by the turbulent flame speed correlation [3]. The transport equation of \( G \) is solved using the second order ENO (Essentially Non Oscillating) method.

Spray Models
Spray atomization of solid-cone injectors is modeled using the Kelvin-Helmholtz, Rayleigh-Taylor (KH-RT) hybrid break-up model in Forte. Droplet collisions were modeled using the Radius of Influence (ROI) collision model. A discrete multi-component (DMC) fuel-vaporization model is used to represent the vaporization of spray droplets.

In addition, an unsteady gas-jet model is applied to predict the relative velocity between the droplets and the gas to give improved entrainment rate predictions relative to mesh-based calculations [4,5]. This results in more accurate and less mesh-dependent drag predictions. Using the gas-jet model, the liquid penetration and droplet distribution inside the domain have been demonstrated to be mesh insensitive and predictive under many spray configurations, covering the whole range of injection pressures and injector nozzle sizes relevant to engine environments. However, using the original gas-jet implementation reported in Refs. [4,5], the velocity field in the spray plume and the vapor penetration length were found to be mesh sensitive in certain spray and mesh configurations. A recent development on the spray/gas coupling in Forte greatly reduces the mesh sensitivity of the gas phase parameters. With the help of SAM, we will study the mesh convergence of gas phase velocity and vapor penetration length using the latest spray models in Forte.

3. Results and Discussion

In this section, SAM was applied to several key components involved in engine simulation, including basic flow solution, flame propagation, and sprays.

3.1 Intake Charge Process
The impact of SAM on basic engine flow calculation is studied through the simulation of the intake charge process of an SI engine. This engine uses a simplified “generic” geometry created by ANSYS, which is mainly used for code testing and tutorial purposes. As shown in Fig 1, this geometry contains a symmetry boundary condition, which reduces the computational time by 50% compared to a whole engine calculation. The calculation starts at 330 °ATDC (near intake valve open) and ends at 700 °ATDC (near spark timing). The intake and exhaust boundary pressures are 0.8 bar and 1 bar, respectively.

![Figure 1. SI engine geometry used in this paper.](image)
The global mesh size is 3 mm. Cells along the system boundaries are refined to 1.5 mm using fixed refinement. The cell size within valve gaps are controlled using the geometry-adaptive mesh refinement. The smallest cells inside the gaps are 0.375 mm during valve motion. Calculations without and with SAM are compared. As mentioned earlier, the SAM calculation uses the statistical criterion with $\gamma_{\text{threshold}} = 0.5$. Velocity gradient was selected as the only refinement control parameter, and the smallest cell size in SAM is 0.75 mm.

The in-cylinder averaged pressure, mass, tumble ratio, turbulent kinetic energy, and number of cells are compared in Fig 2. With the fixed and geometry-adaptive refinements already in place, SAM has a negligible impact on these global parameters. The difference in trapped mass is less than 0.5%. The cell count in the SAM calculation is substantially larger, especially in the duration when the intake valves are open. As a result, the CPU time of the SAM calculation is roughly 30% longer.

In Fig. 3, the meshing details are compared at two sampled crank angle locations. The cut-planes go through the valve stems and they are shaded using velocity magnitude. Without SAM, the geometry-adaptive refinement keeps the valve gaps finely resolved, but there is an abrupt change in cell size outside the gap. By contrast, SAM creates a more natural transition of cell size by following the flow structure near the valve gap. Generally speaking, the bulk flow patterns look very similar with or without SAM, but SAM provides more details of the flow structures by putting smaller cells in the high velocity gradient regions. If it is desirable or required to capture these detailed flow structures in certain applications, SAM will be an effective tool for achieving this.

### 3.2 Spark Ignition and Flame Propagation

Using the same generic SI engine geometry shown in Fig.1, SAM was further applied to study the SI combustion models. At spark timing, the premixed mixture in the cylinder is a stoichiometric iso-octane/air mixture with around 15% residual. The spark timing is 705 °ATDC. In the discrete particle ignition kernel model, the initial kernel radius is set to 0.2 mm. Four cases are run and compared:

1) Fixed volume refinement in entire cylinder 1/4 level, 0.75 mm
2) SAM with CO$_2$ mass fraction gradient 1/4 level, 0.75 mm
3) SAM with temperature gradient 1/4 level, 0.75 mm
4) SAM with temperature gradient 1/8 level, 0.375 mm
The in-cylinder averaged pressure, CO mass, NO mass, and cell count are compared in Fig. 4. It can be seen that both temperature gradient and CO$_2$ mass fraction gradient are adequate candidates as SAM control parameters. Using SAM at the ¼ level achieves the same level of accuracy for global parameters as the full fixed refinement at that level, but requires fewer cells. The comparison of cases 3 and 4 indicates that the flame propagation model is insensitive to further refinement in the flame front zone. Using the ignition kernel model and the $G$-equation flame propagation model, it is unnecessary to use tiny cells inside the flame front zone. Cells on the order of 1 mm should be able to provide sufficient resolution for these models.

The meshing details and temperature contours of the fixed refinement and two temperature-based SAM calculations are compared in Fig. 5. When deeper refinement is used in SAM as shown in the SAM 0.375 mm case, the cell population is biased toward the small cell side, resulting in larger cell sizes outside the SAM refinement zone. There are many large cells completely unrefined in the burned zone. If such large cells caused by deep SAM refinement are undesired, users should consider using smaller global mesh size or using fixed volume refinement to make sure the regions beyond the flame front zone are properly resolved, for example, for emissions.

**Figure 4.** Impact of SAM and SAM criterion on pressure, emissions, and mesh size.

**Figure 5.** Impact of SAM on meshing details and flame propagation (shaded by temperature, range is 600-2600 K).
3.3 Non-Evaporating Spray

In the first spray test case, we focus on the momentum coupling between the liquid droplets and the gas phase, and thus the vaporization model is turned off. This case simulates a single-plume diesel spray under non-evaporating environment. Fuel (n-Hexadecane) at 320 K is injected into a vessel filled with nitrogen that is initially quiescent and at a back pressure of 50 bar. The fuel injector has a discharge coefficient of 0.7 and its orifice diameter is 250 µm. The fuel is injected at a constant mass flow rate of 5.33 mg/msec.

The global cell size is set to 1 mm. Velocity gradient is used as the only SAM control parameter. Three refinement levels are studied, in which the smallest cell sizes are 0.5, 0.25, and 0.125 mm, respectively. The simulation duration is 0.5 ms.

As shown in Fig. 6, the liquid penetration lengths agree well among all three cases. Figure 7 shows the spray particle distribution and velocity contours at 0.4 ms. Mesh lines on the cut-planes are shown in the images of the middle row. It can be seen that the high velocity gradient regions are refined as expected, which shows that the SAM refinement strategy is effective in capturing the moving front of sharp velocity gradients. With regard to the mesh sensitivity, the velocity contours do not show noticeable variation as the spray plume is further refined. In fact, the velocity fields of the 0.25 mm and 0.125 mm cases are almost identical. This indicates that the momentum coupling between the spray droplets and the gas phase is insensitive to mesh resolution, and mesh convergence can be achieved when the cell size are refined to 0.25 mm. However, the 0.5 mm result is really not much different from the finer mesh results. Thus even with 0.5 mm mesh, the spray results are expected to be reasonable in practical engine simulations.

![Figure 6. Liquid penetration lengths of non-evaporating spray.](image)

3.4 Evaporating Spray

Using SAM as a tool, we next study the mesh sensitivity of vapor penetration using an evaporating spray. The ECN spray A from the Engine Combustion Network [6] is used for this study. This case considers a single-component fuel (n-Dodecane) at 373 K injected into a cubic vessel. The vessel is filled with nearly-quiescent inert gas (89.71% N₂, 6.52% CO₂ and 3.77% H₂O) at a back pressure of 60.5 bar. The average temperature in the vessel at the start of injection is 891.9 K. The injector (SN 210677) has a discharge coefficient of 0.89 and an orifice diameter of 84 µm. An estimate of 3.46 mg fuel is injected within a duration of 1.54 ms, and a constant mass flow rate is adopted.

Similar to the non-evaporating spray, the evaporating case also uses 1 mm as the global cell size, and the minimum cell sizes in SAM also vary from 0.5 mm to 0.125 mm. In this case, in addition to velocity gradient, SAM uses the gradient of fuel vapor mass fraction as a second control variable. Both SAM controls use \( \sigma_{\text{threshold}} = 0.5 \) for the statistical criterion.

Figure 8 shows the calculated liquid and vapor penetration lengths compared to the experimental data. The liquid penetration lengths of all three SAM calculations are essentially identical. All calculated vapor penetration lengths agree with the measured data reasonably well. As the mesh is refined from 0.5 mm down to 0.125 mm, the vapor penetration prediction is improved slightly. Note that both liquid and vapor penetrations are slightly under-predicted in the current calculations, which used all default model constant values in Forte [7]. The simulation results are expected to improve further with some adjustment to the breakup model constants.
Fuel vapor mass fraction and velocity contours at 1.6 ms are compared in Fig. 9 and 10, respectively. These results demonstrate that the current spray models in Forte are able to achieve excellent mesh independency for these key gas phase parameters. Accurate spray solutions can be obtained even with 0.5 mm mesh, though deeper refinements could offer marginally better agreement with the experiment.

As an example, the cell arrangement on a cut-plane of the 0.125 mm SAM calculation is shown in Fig. 11. The refinement inside the spray plume appears to be due to the velocity gradient criterion, while the refinement along the outer envelope follows the shape of the fuel vapor contour.

4. Summary

The solution-adaptive mesh refinement (SAM) capability was applied to the simulation of intake charge flow, flame propagation, and sprays under conditions relevant to engines.

The statistical criterion of SAM produces expected refinement results when combined with a $\sigma_{\text{threshold}}$ value of 0.5 in all the applications tested. This criterion is a good starting point to explore SAM application.

For engine flow simulations, velocity gradient is an effective SAM control parameter. Adding SAM in the flow calculation has small impact on global solution parameters, but provides better resolution for detailed flow structures.

For spark ignition and flame propagation simulation, both CO$_2$ mass fraction gradient and temperature gradient are effective SAM control parameters. Using the $G$-equation model, it is adequate to use cells on the order of 1 mm in the flame front zone. The combustion results are insensitive to further refinement in the flame front zone, and thus it is unnecessary to use tiny cells to mesh the flame front.

The latest spray models in Forte have been demonstrated to be mesh insensitive in both non-evaporating sprays and evaporating sprays. Mesh convergence could be achieved without resorting to minuscule cells. Meaningful and accurate spray results can be obtained using 0.5 mm cells. Deeper refinement may be used if desired, but are expected to have marginal impact on the overall solution.

References
7. ANSYS Forte CFD, ANSYS, San Diego, CA.