Grid Control

CONVERGE Studio Workflow
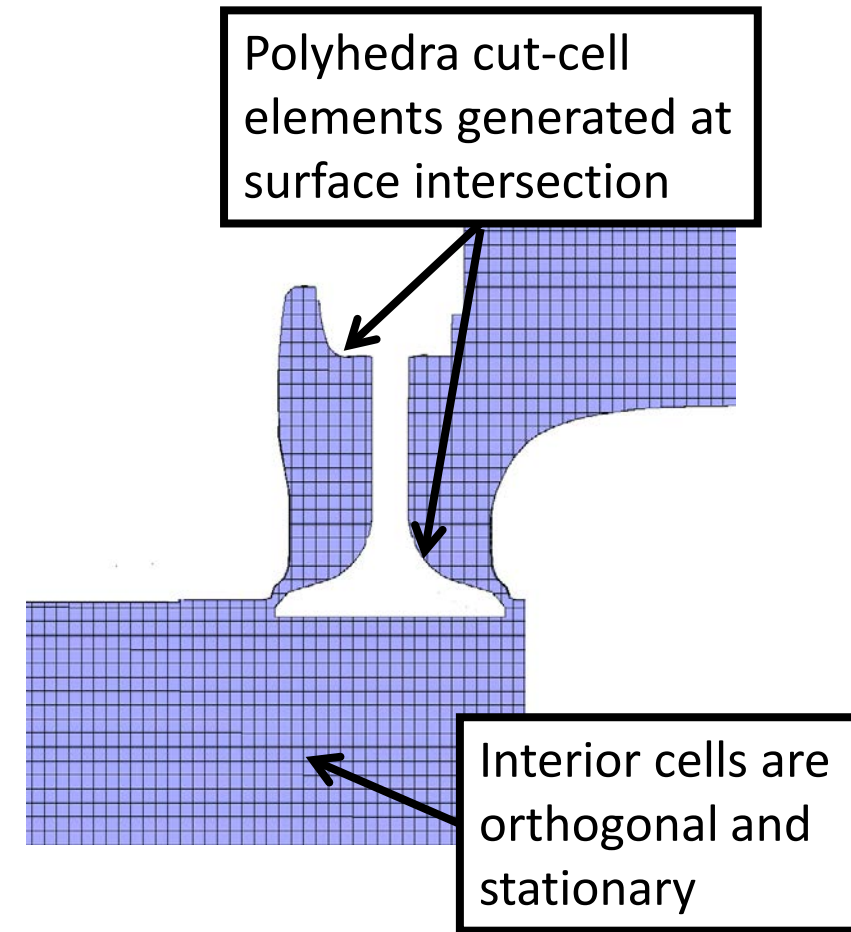
- ***Case Setup* module**
  - o Begin a project
  - o Import the surface geometry
  - o Prepare the surface
  - o **Configure case setup**
    - ▪ Boundary conditions and region definitions
    - ▪ Initialization
    - ▪ **Grid control**
    - ▪ Physical models (turbulence, spray, combustion, sources, CHT, VOF, etc.)
    - ▪ Advanced options
  - o Export input and data files to the Case Directory

---Run CONVERGE simulation--

- *Line Plotting* module

- *Post-Processing 3D* module

CONVERGE
CFD SOFTWARE

Key Technology: Cut-Cell Cartesian Meshing

- CONVERGE automatically generates a body-fitted volume mesh using a patented cut-cell technique

  o Interior cells are orthogonal hexahedrals, which is ideal for the solver algorithm

  o Interior cells remain stationary, which removes numerical diffusion associated with a moving mesh

  o Cells intersected by the geometry surface are "cut" by the surface to form arbitrary-sided polyhedra

  o The resulting volume mesh is body-fitted and is an exact representation of the surface geometry

Polyhedra cut-cell elements generated at surface intersection

Interior cells are orthogonal and stationary

CONVERGE
CFD SOFTWARE

Key Technology: Cell Pairing

- CONVERGE trims the cells at the intersecting surface, which results in some irregularly-shaped cut-cells

- If the volume of a cut-cell is less than 30% of the volume of the original cell, then a cell pairing process combines the cut-cell and the adjacent cell that shares most faces with the cut-cell

- The center of the paired cell becomes the volumetric center of the combined cell

- Values of transport entities (velocity, temperature, pressure) are shared by the regular cell and the cut-cell

Advantages in CONVERGE Grid Generation

- Unlike other CFD codes, CONVERGE maintains the original surface definition with high fidelity such that the volumes and areas will be correct regardless of the mesh resolution
  - CONVERGE maintains the surface definition for all grid control techniques
- The solver and meshing algorithm are tightly coupled (there are no mesh files created)

Grid Control Strategies

- There are four ways to manipulate the grid size in CONVERGE
    - Base grid size: Assign dx, dy, dz for the base grid
        - The base grid size is the largest grid size available in the simulation
        - All other grid control strategies are specified with respect to the base grid size
    - Grid scaling: Refine or coarsen the entire mesh at a specified time
    - Fixed embedding: Refine a specified portion of the grid at a specified time
    - Adaptive Mesh Refinement (AMR): Automatically refine the grid based on local flow conditions such as temperature or velocity
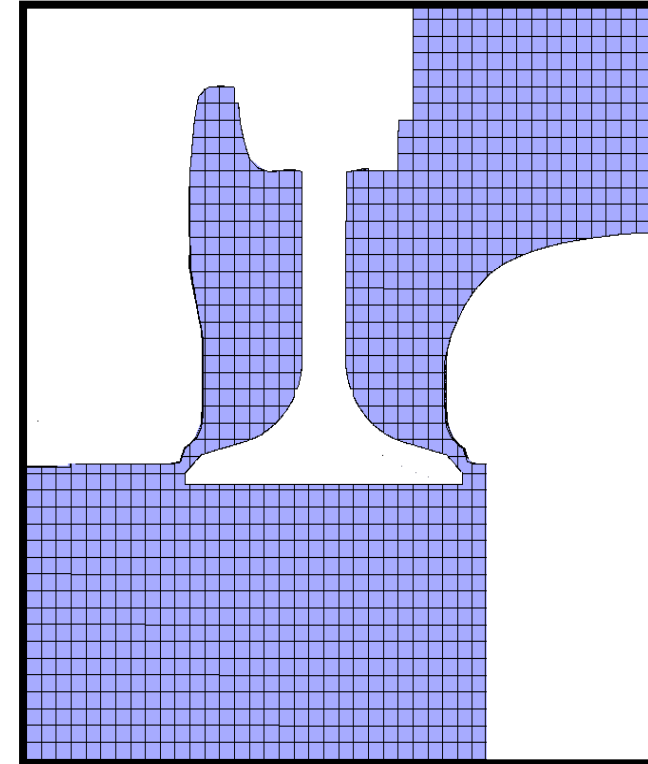

Base mesh

Combining Strategies

- You can specify as many grid control strategies as desired

- The grid control strategies can be activated and deactivated at various times

- If more than one grid refinement technique is activated in a single cell at a particular time, the method that calls for the largest refinement level will be used (*i.e.,* the effects are not additive)

Grid Control in a New Simulation

- When preparing a new simulation, it may be helpful to start with a coarse base mesh and no refinement techniques
  - o This allows for a rapid solution while testing the case setup and boundary motion
  - o Once you have verified the setup, activate the desired grid control techniques
- Because the base mesh size is a simulation input, in CONVERGE it is straightforward to perform a grid resolution study to determine the resolution required for reasonable grid independence

How to Set Up Grid Control

- To set up grid control, go to *Case Setup > Grid Control*

    o <u>Base grid</u> is always enabled

    o Click <u>Adaptive mesh refinement</u>, <u>Fixed embedding</u>, and <u>Grid scaling</u> to activate these options
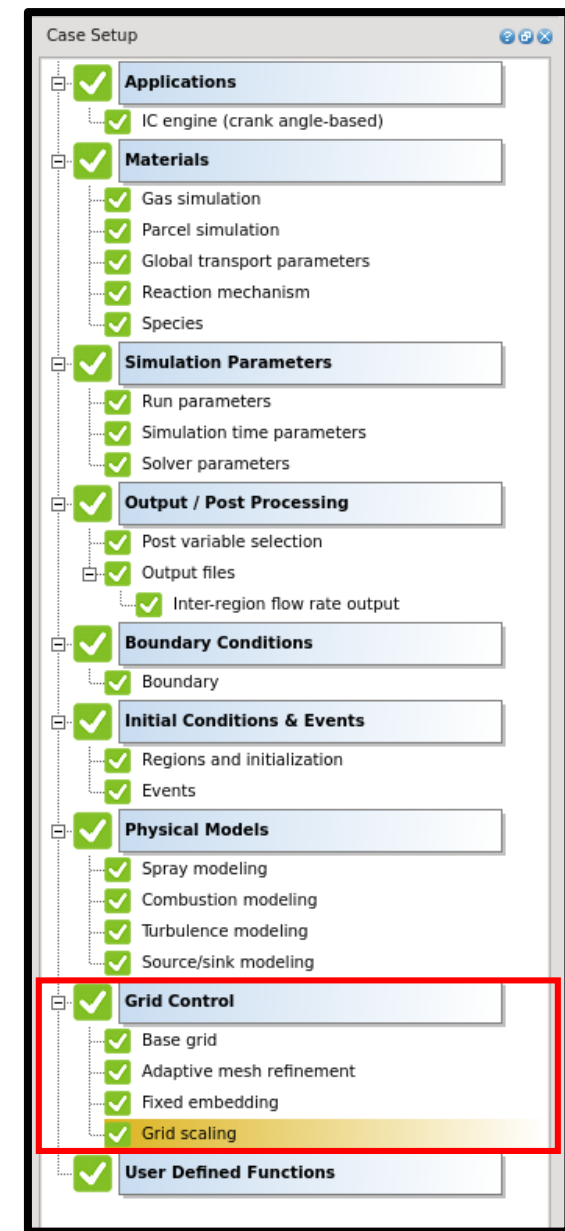
Base Grid

- Go to *Case Setup > Grid Control > Base grid* to set values for *dx, dy,* and *dz* (in *meters*)
- Click <u>Suggestion from target cell count</u> and enter a target cell count to get a suggested base grid size
  - The suggestion does not include other grid control techniques
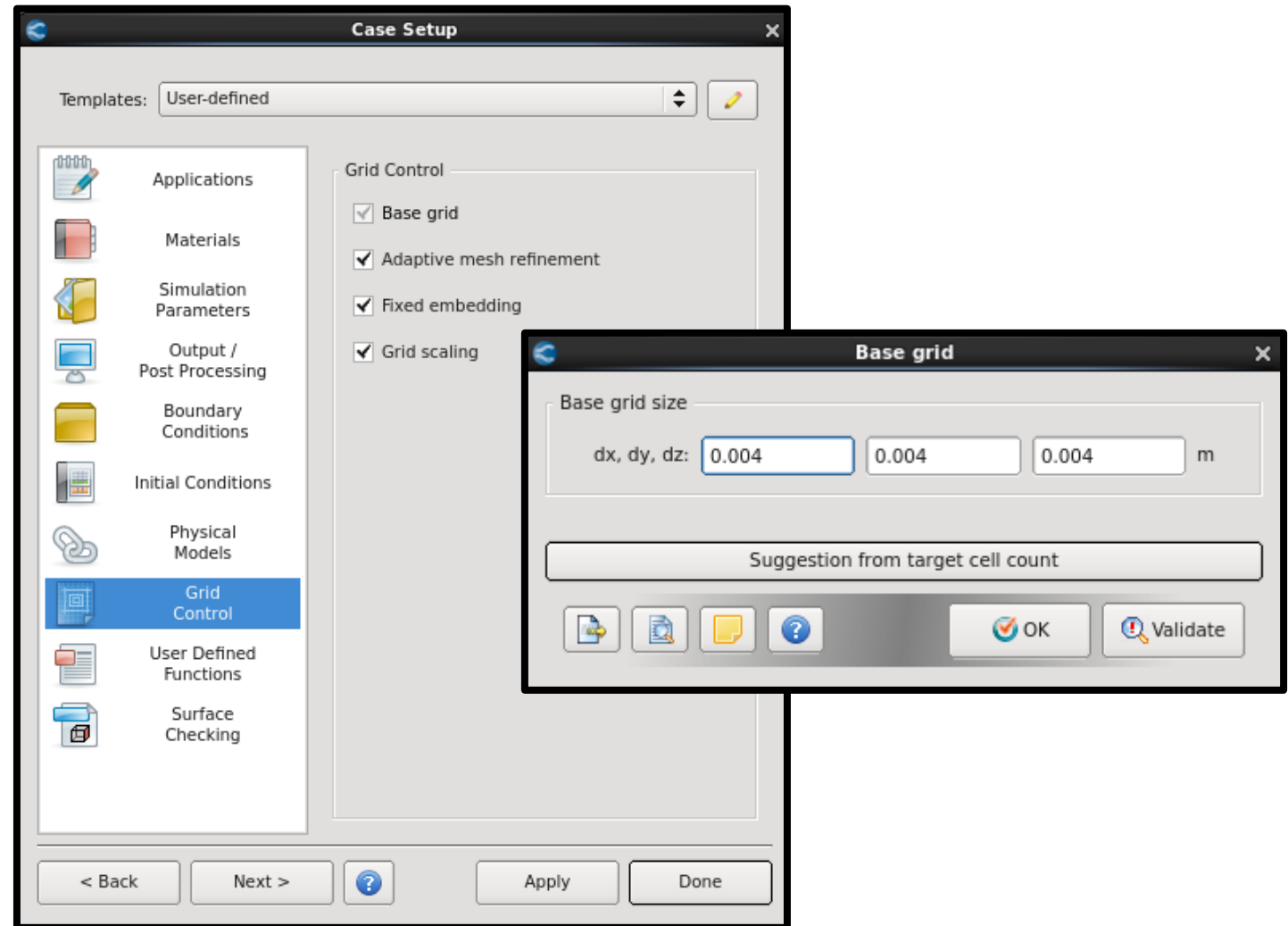
Grid Scaling (1/2)

- Grid scaling changes the base mesh size for the entire domain at a specified time during the simulation

- Grid scaling is useful for steady state flows to speed up convergence

- The base grid (*dx_base*) can be coarsened or refined by any power of two:

$$scaled\ grid = dx\_base/2^{grid\_scale}$$

- *grid_scale* must be an integer
  - Negative integers coarsen the grid
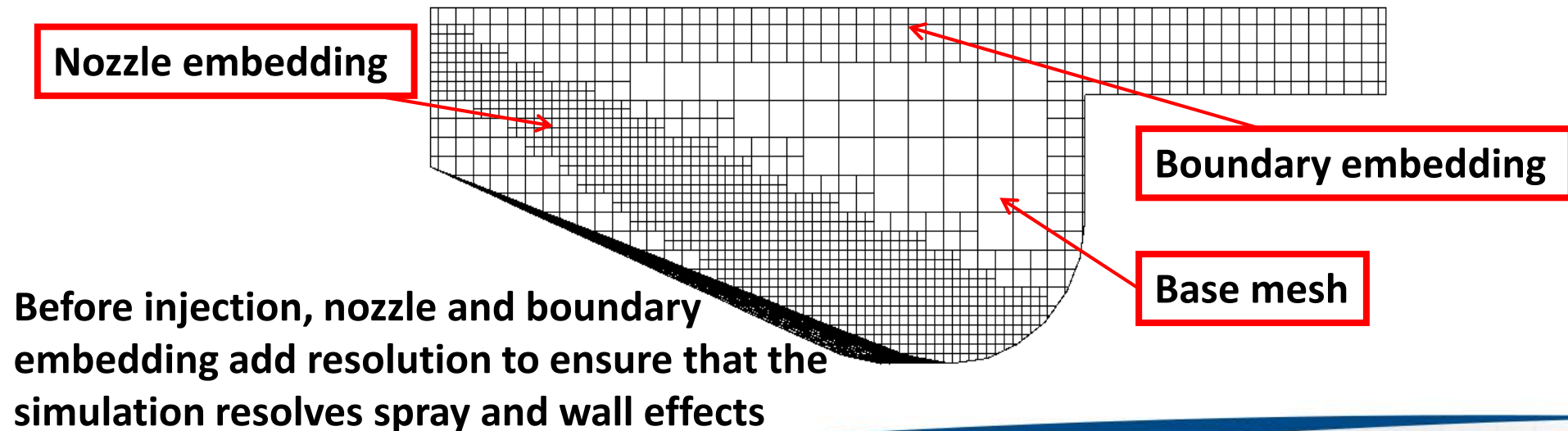 - Positive integers refine the grid

Grid Scaling (2/2)

- Go to *Case Setup > Grid Control > Grid scaling*
  - Enter an integer or check <u>Use file</u> to set up different grid scaling values at different times
    - The grid scaling values must be in ascending order
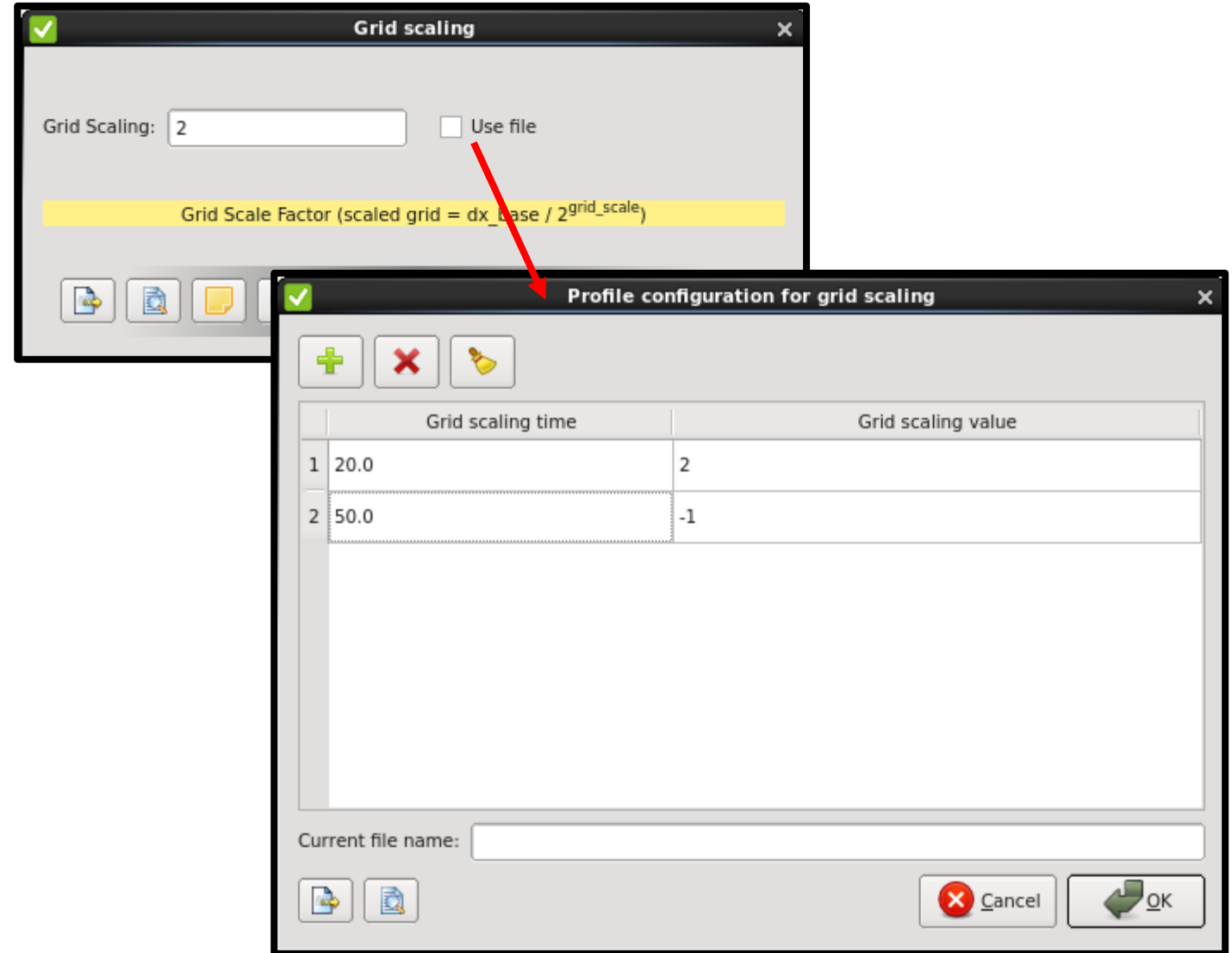
Fixed Embedding (1/4)

- Fixed embedding refines the grid at specified locations and times to capture important phenomena such as spark injector and spark energy deposition

- Embedding techniques can be permanent or set to coincide with critical events such as spray initiation or a spark event

Nozzle embedding

Boundary embedding

Base mesh

Before injection, nozzle and boundary embedding add resolution to ensure that the simulation resolves spray and wall effects

CONVERGE
CFD SOFTWARE

Fixed Embedding (2/4)

- For each embedding technique, an embed scale is specified to indicate how many levels of embedding should be included

- The embed scale is governed by the following equation:

$$embed\ cell\ size = dx\_base / 2^{embedding\ scale}$$

- Negative values for *embedding scale* (*i.e.*, coarsening) are not allowed for grid embedding
  - This is in contrast to grid scaling, in which both refining and coarsening are allowed

Fixed Embedding (3/4)

- Go to *Case Setup > Grid Control > Fixed embedding*
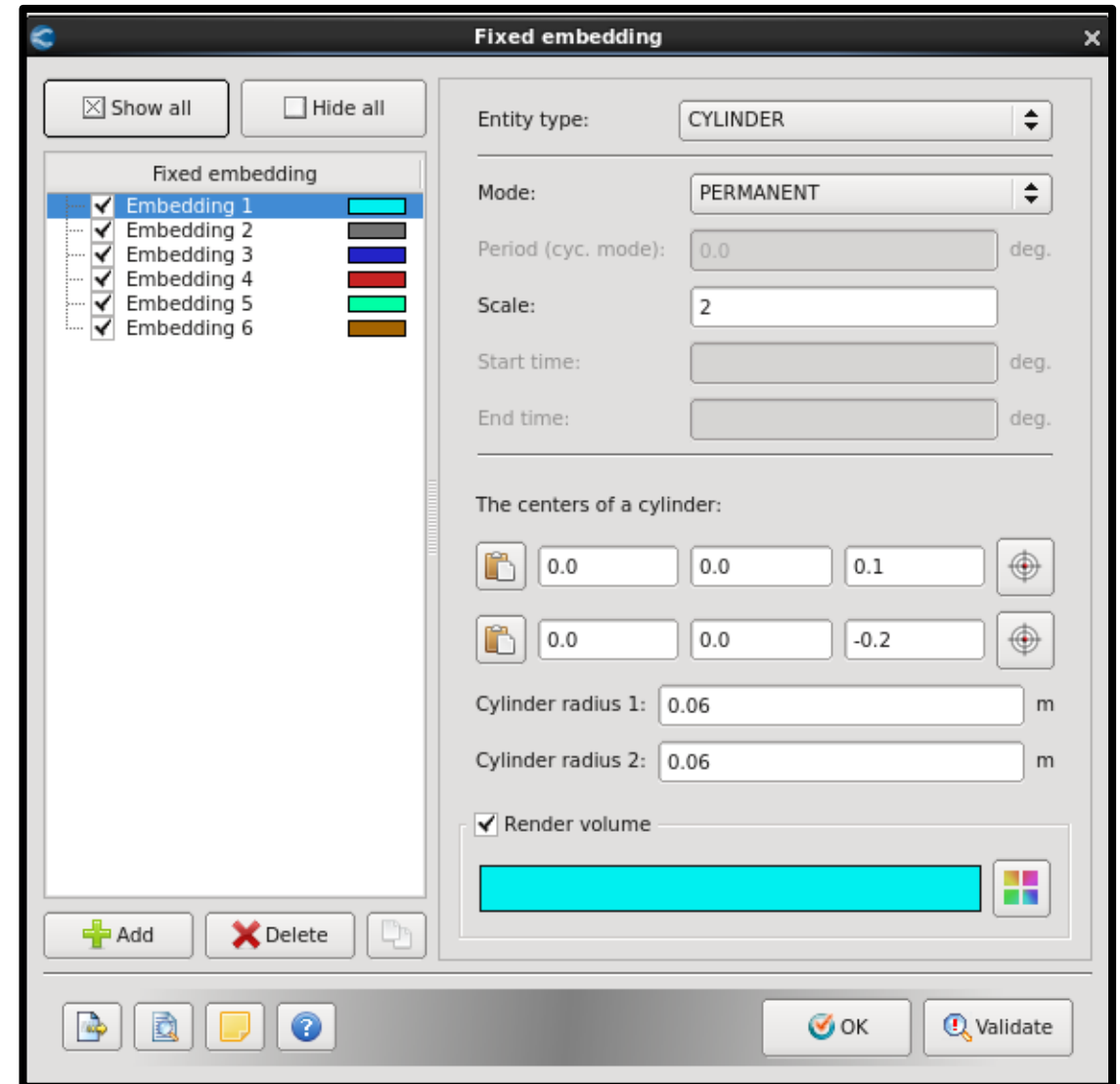
Fixed Embedding (4/4)

- Fixed embedding can add mesh resolution in various locations
    - Boundary embedding: at a boundary of interest
    - Sphere embedding: in a sphere of interest
    - Cylinder embedding: in a user-defined cylinder
    - Nozzle embedding: near a nozzle (spray)
    - Injector embedding: near an injector (spray)
    - Box embedding: in a user-defined box of interest
    - Region embedding: in a specific region
- These techniques are described on the following slides

Fixed Embedding: Boundary

- Boundary embedding refines a boundary (typically a wall) and a specified number of layers of additional cells around the boundary

- For a moving surface (such as a valve), the embedding will move with the surface

- Only one boundary embedding per boundary
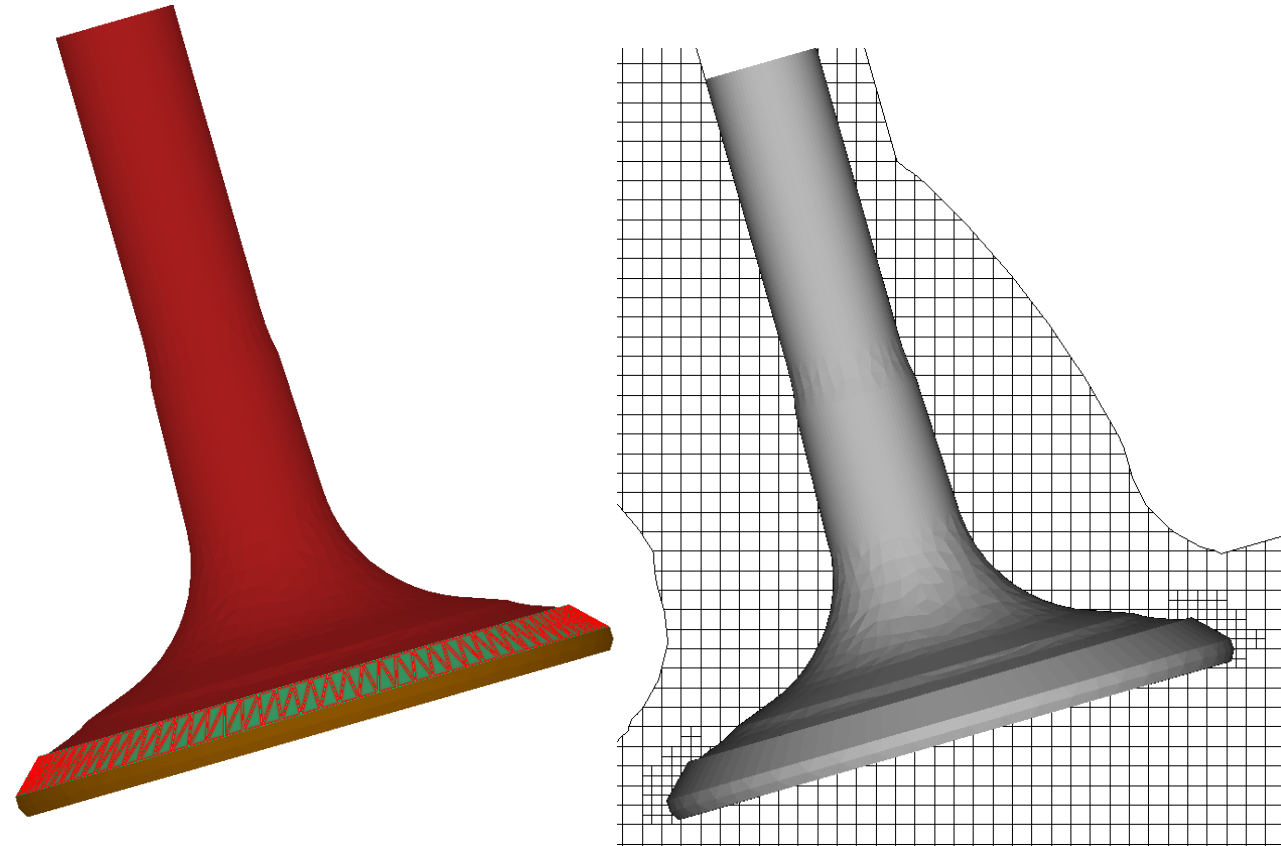
Fixed Embedding: Sphere

- Sphere embedding allows you to specify a spherical region in which fixed embedding will be applied

- Often sphere embedding is used to resolve the spark plug immediately before discharge

- Another common use is to resolve injector nozzle immediately the start of the Lagrangian
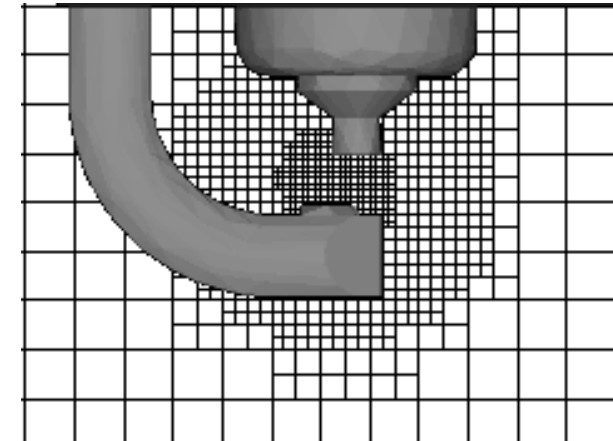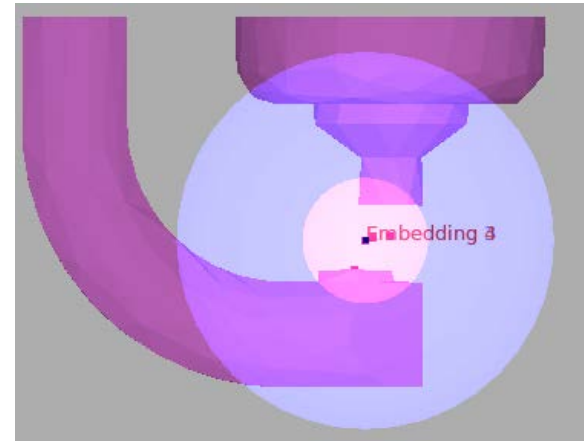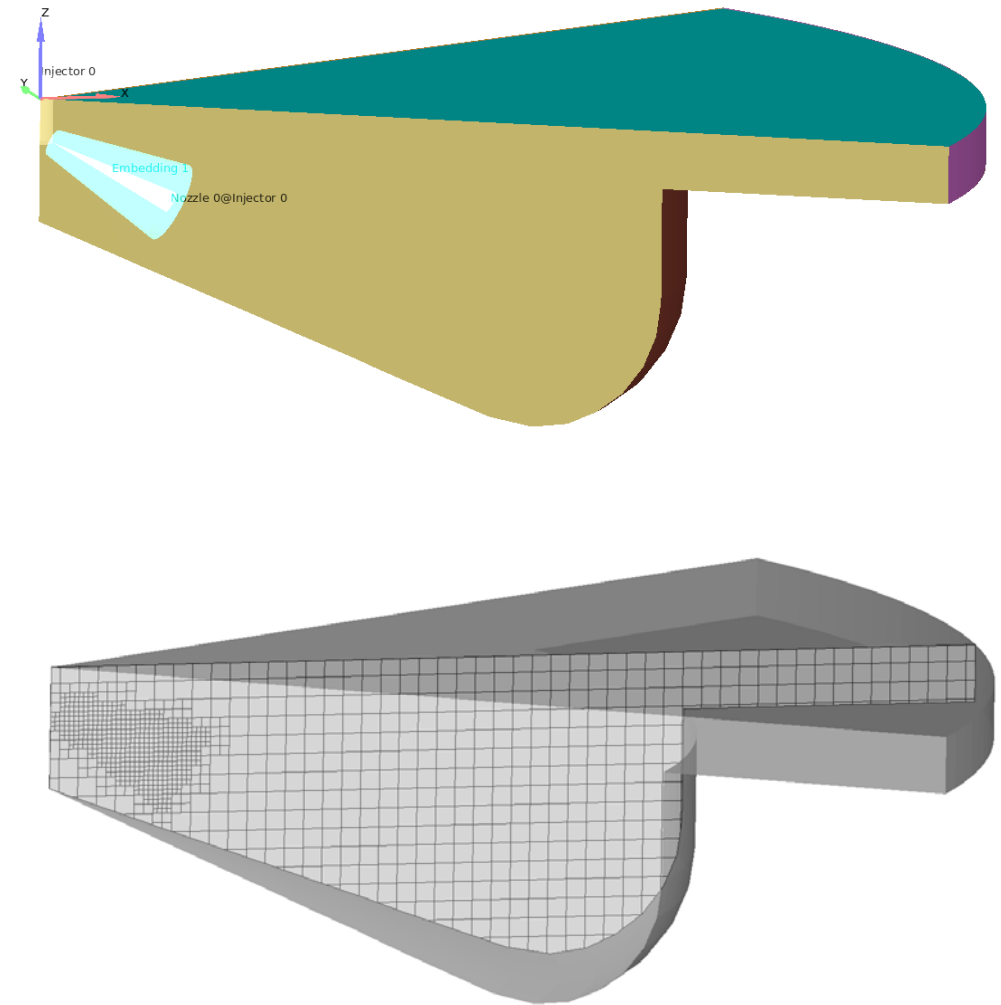
Fixed Embedding: Cylinder

- CYLINDER embedding allows you to specify a cylindrical region where fixed embedding will be applied
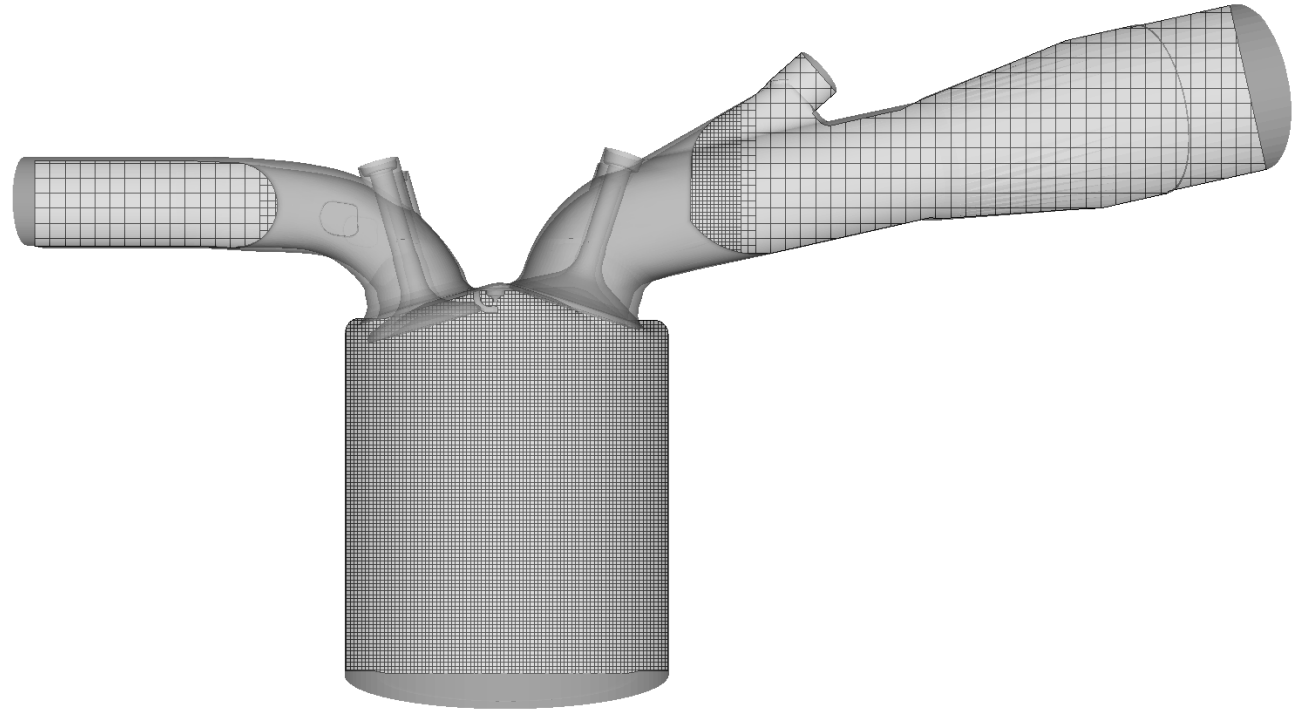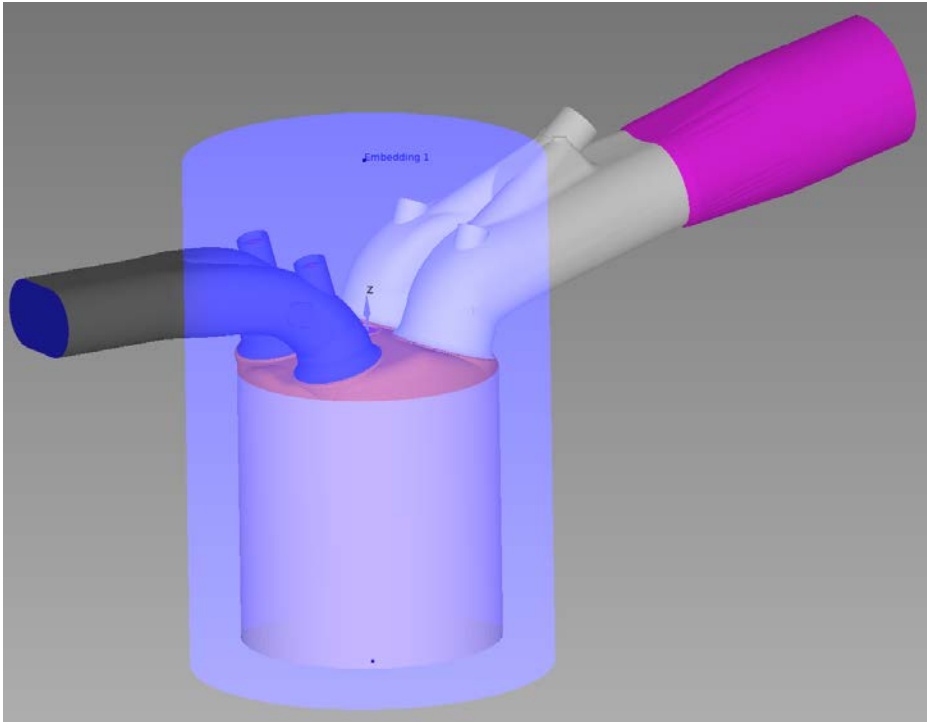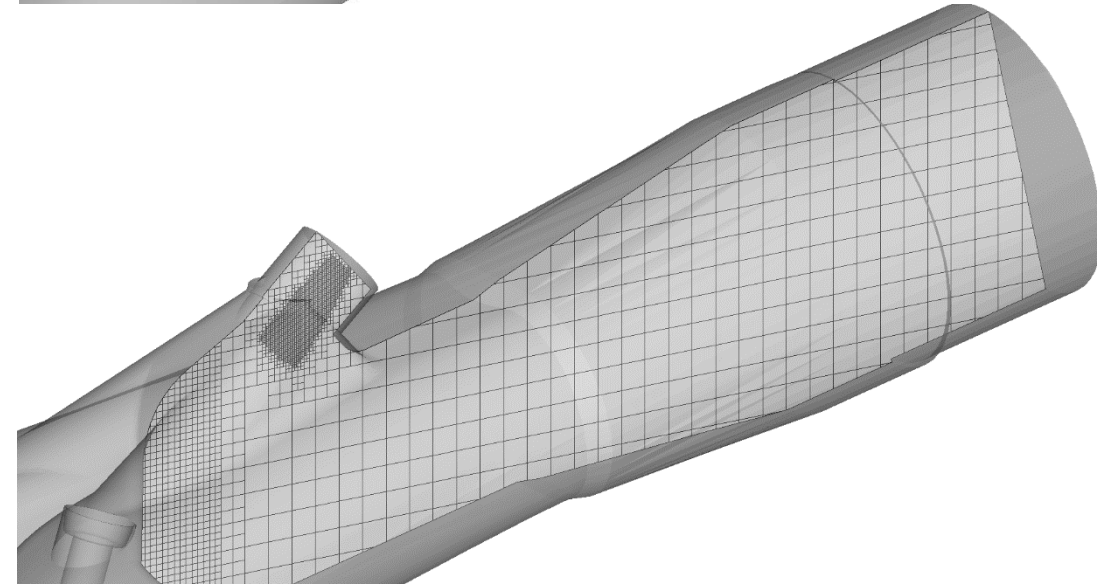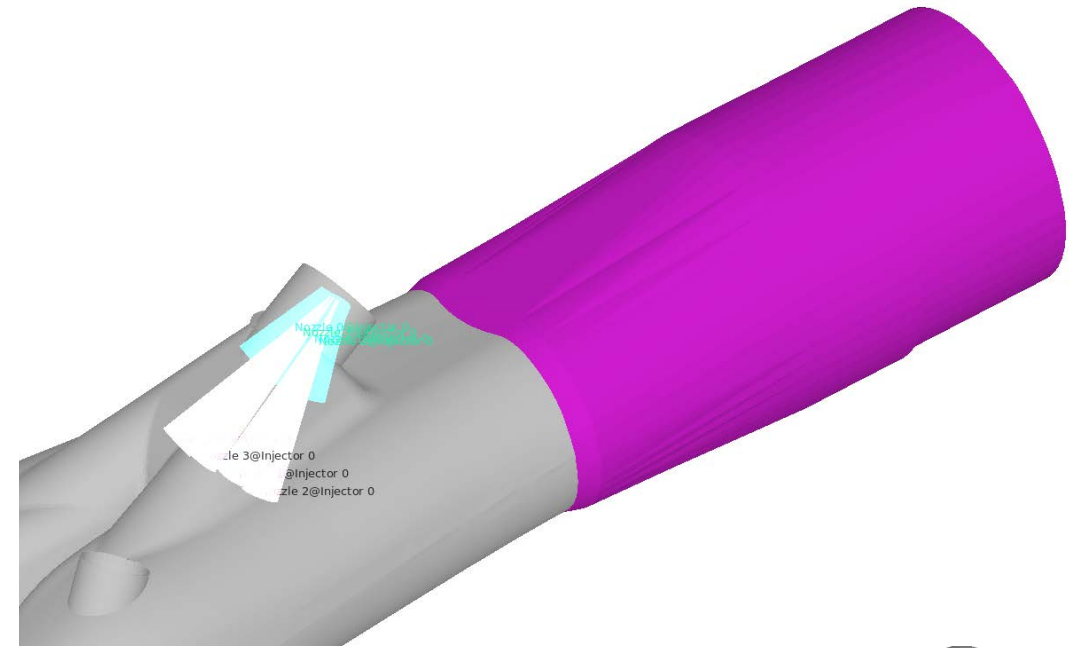
Fixed Embedding: Nozzle

- Nozzle embedding and injector embedding are closely related as both resolve the spray region

- The nozzle embedding option is used for convenience to specify conical embedding around a nozzle (equivalently, cylinder embedding can be used)

- To use nozzle embedding, define at least one nozzle (*Case Setup > Physical Models > Spray modeling*)

Fixed Embedding: Injector

- CONVERGE groups liquid injections into injectors and nozzles

- An injector is a group of nozzles with some of the same characteristics

- Injector embedding is similar to nozzle embedding except that it embeds for all nozzles in the injector
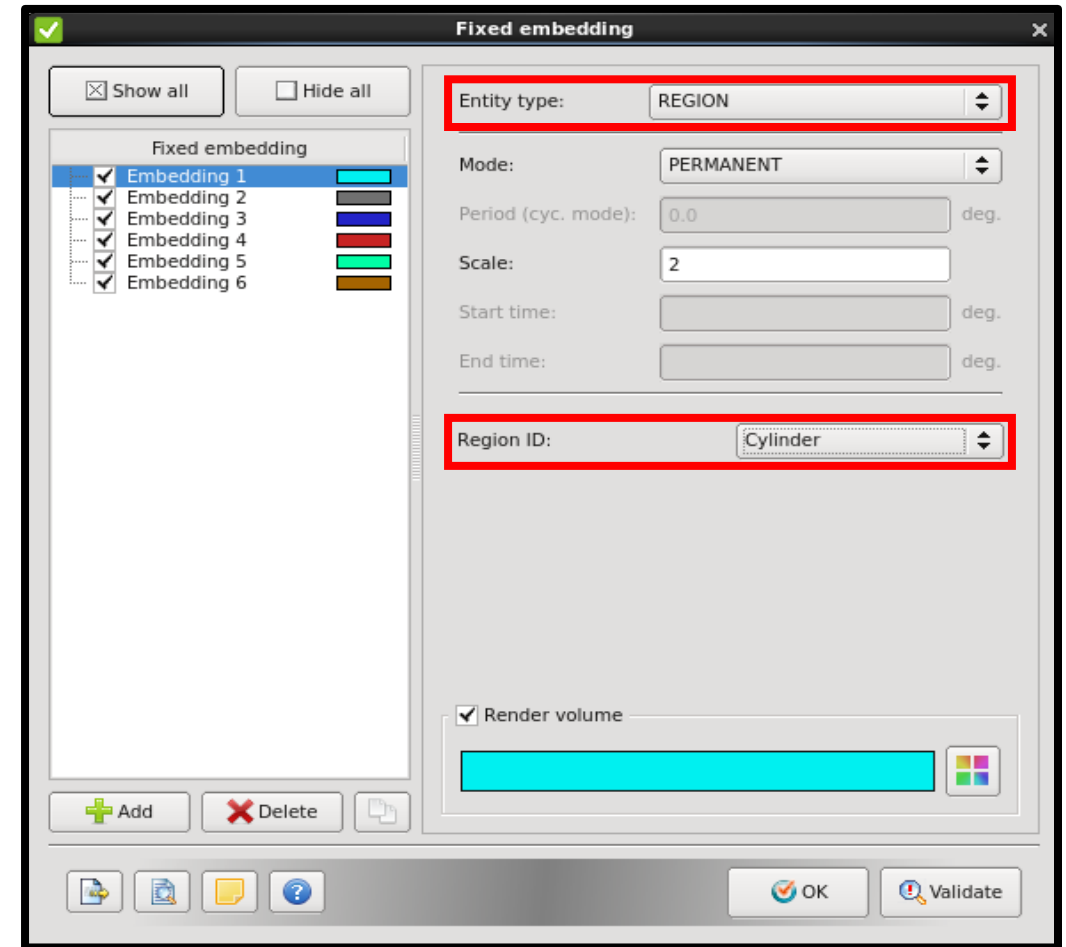
Fixed Embedding: Box

- Box embedding allows you to specify a box region where fixed embedding will be applied

- Go to *Case Setup > Grid Control > Fixed embedding* and select the <u>BOX</u> entity type
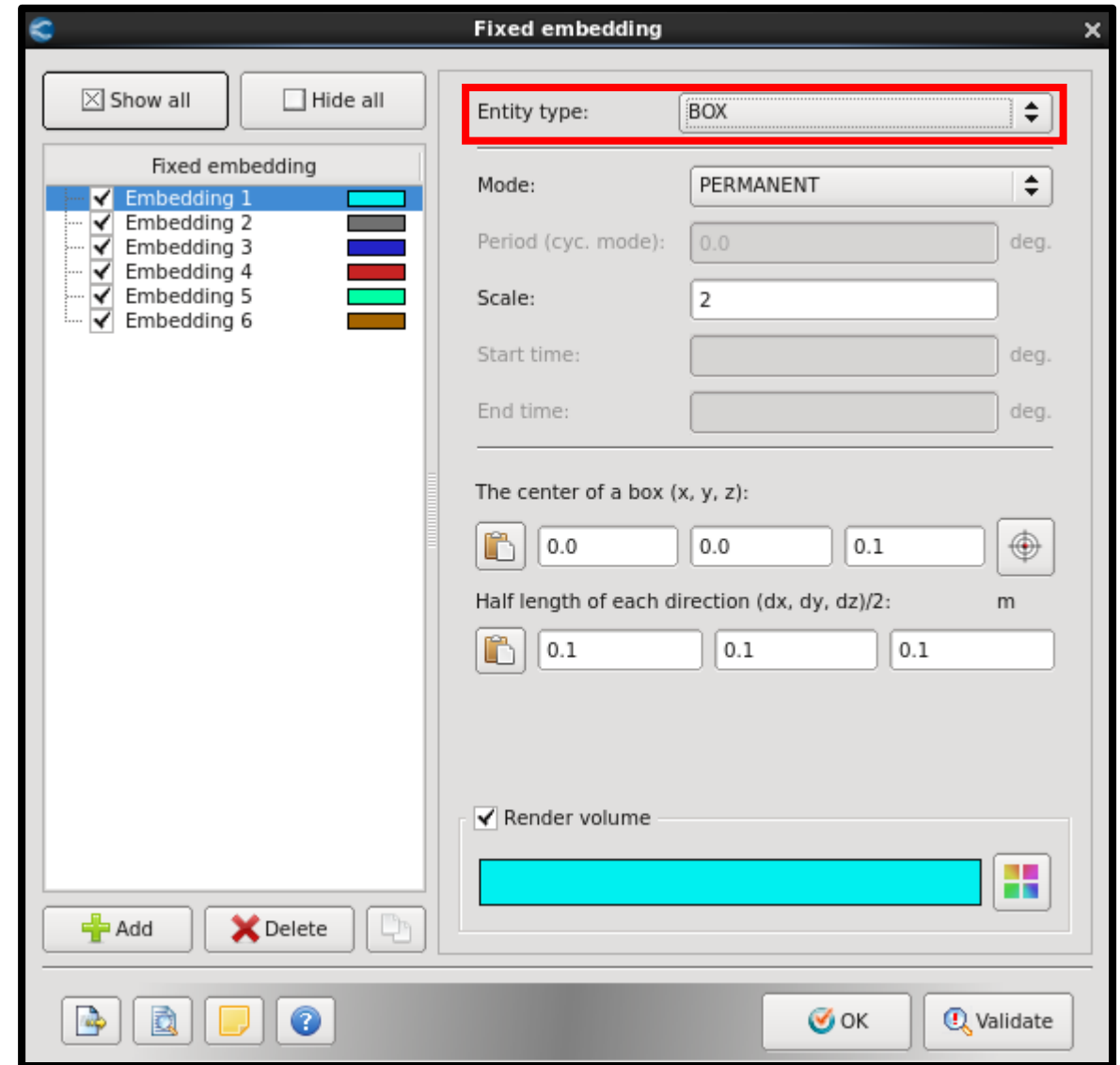
Fixed Embedding: Region

- Region embedding allows you to specify a region to which fixed embedding will be applied

- Go to *Case Setup > Grid Control > Fixed embedding* and select the <u>REGION</u> entity type

Fixed Embedding Timing Control

- There are three options for fixed embedding timing control to be specified for each fixed embedding technique
    - o PERMANENT: Embedding technique always used
    - o SEQUENTIAL: Embedding technique starts and ends at specified times
    - o CYCLIC: Embedding technique cycles on and off (you specify the start time, end time, and cyclic period)
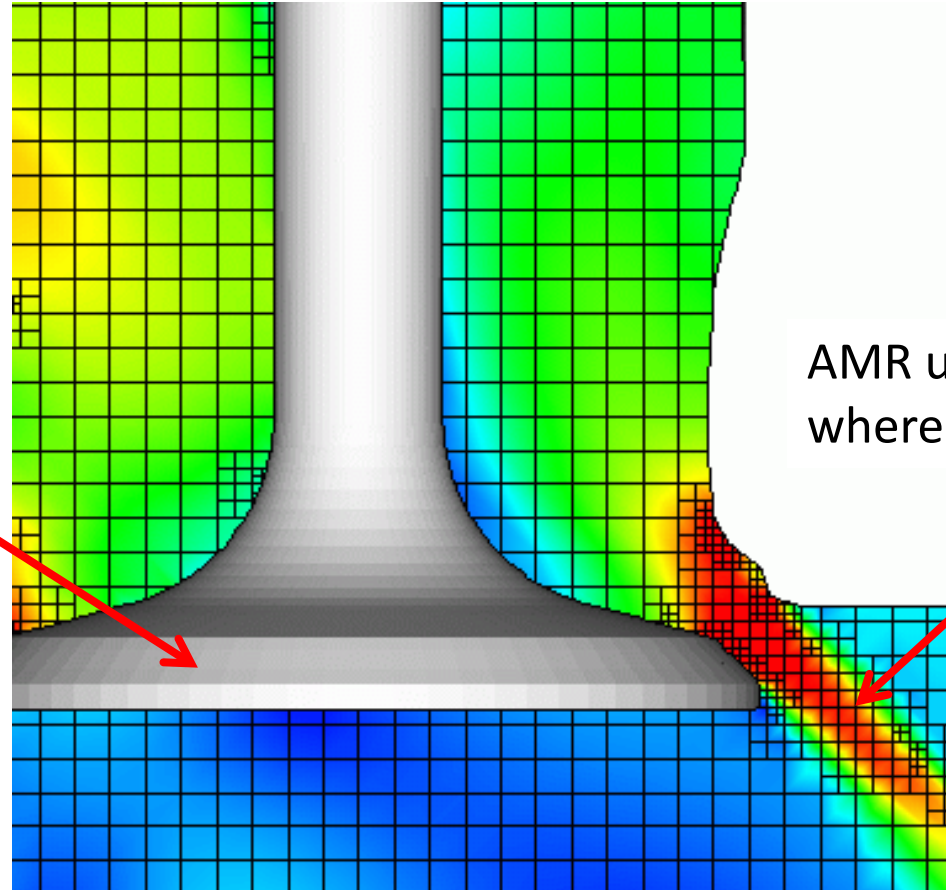
Adaptive Mesh Refinement (1/5)

- AMR automatically enhances the mesh resolution based on curvatures (second derivatives) in field variables

- AMR can be permanent or activated at specified times

- AMR is activated by on a region-by-region basis

- AMR can be activated for velocity, temperature, species, passives, y+ (a dimensionless wall distance), or void fraction

Adaptive Mesh Refinement (2/5)

- You specify a maximum number of AMR cells that, in conjunction with the base grid size, determines the overall mesh resolution and run times
  - This forces CONVERGE to prioritize AMR so that cells are used efficiently and resolution is added only where it is needed most
- To prevent large cell size ratios, you can block CONVERGE from using AMR in response to sub-grid scale quantities near a wall
  - Specify a boundary and a yplus value to be maintained for that boundary

Adaptive Mesh Refinement (3/5)

Fixed embedding used on the valve seat (specified in input file *a priori*)
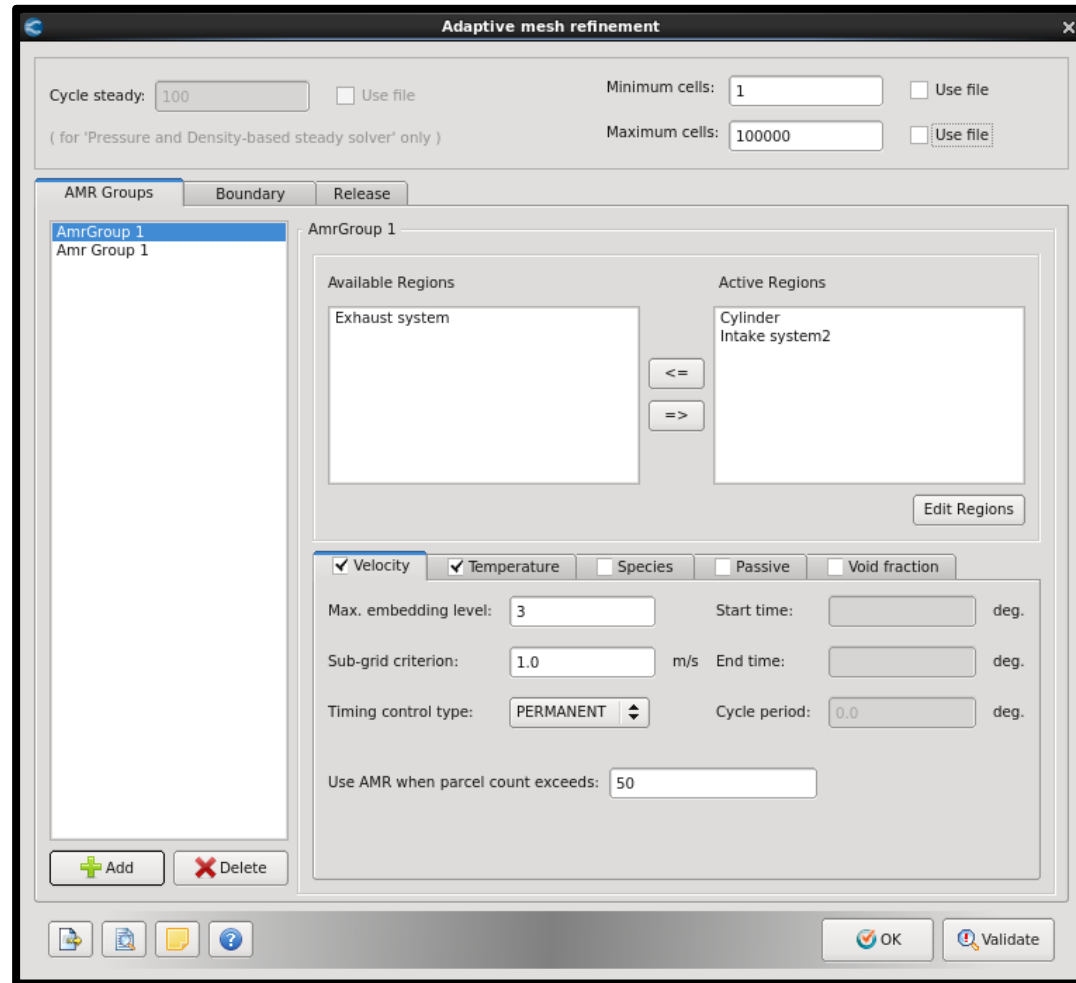
AMR used to resolve flow where curvatures are large

Representative mesh in valve region that contains both AMR and fixed embedding

Adaptive Mesh Refinement (4/5)

- Go to *Case Setup > Grid Control > Adaptive mesh refinement*

Adaptive Mesh Refinement (5/5)

- You may need to limit the wall cell refinement on selected boundaries based on Y+

- If the wall adjacent cell Y+ is smaller than the target Y+, then the refinement is released (i.e., the cell is coarsened)
  - However, CONVERGE will not coarsen the cell if the cell connectivity criteria (2:1) would be violated

- If the ratio of _wall adjacent cell Y+_ to _Y+ target_ is smaller than the _Y+ Ratio_, then the neighboring cells will be coarsened to maintain 2:1 cell connectivity criteria

- Go to _Case Setup > Grid Control > Adaptive mesh refinement > Release_ to set up Y+ AMR

- We recommend a Y+ Ratio of 1/3
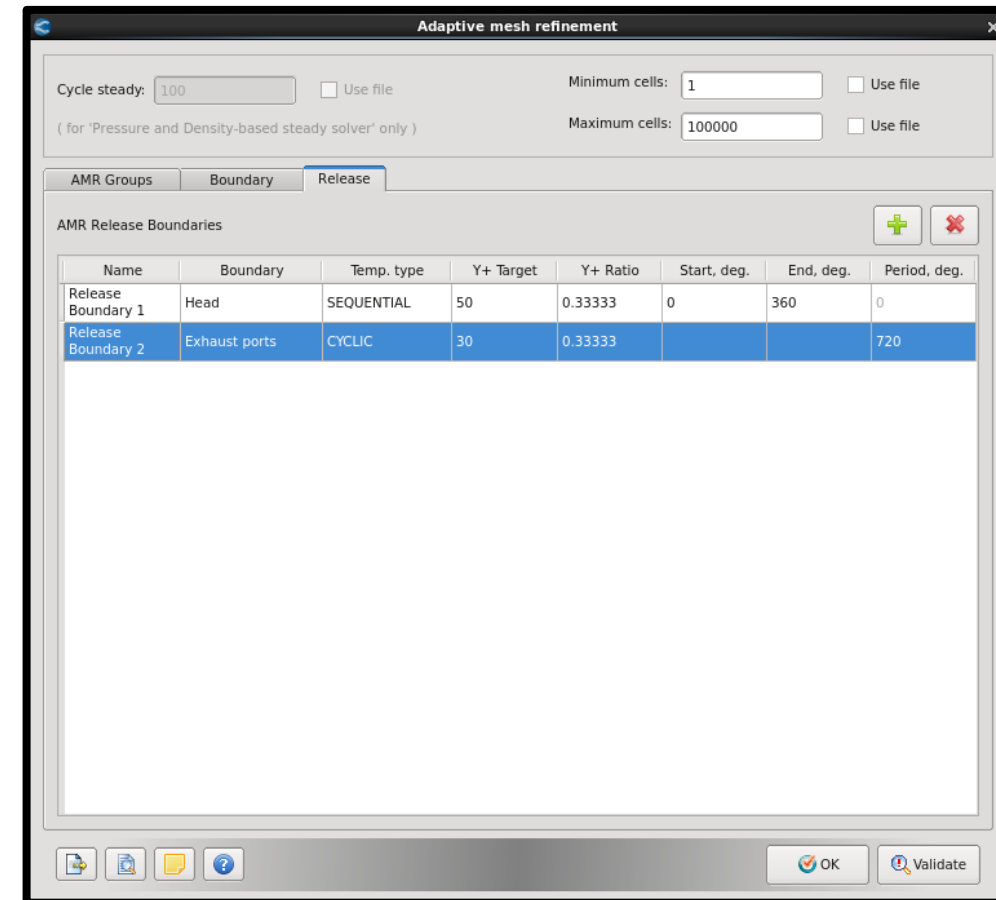
AMR-SGS

- To calculate the sub-grid scalar (SGS) field:

$$\phi' = \phi - \bar{\phi}$$

ϕ' is the sub-grid scalar field, ϕ is the actual field, and $\bar{\phi}$ is the resolved field

- A truncated infinite series approximates the sub-grid field for any scalar:

$$\phi' \cong -\frac{dx_k^2}{24}\frac{\partial^2 \bar{\phi}}{\partial x_k^2}$$

dx_k is the grid spacing for a given rectangular cell

- o If the absolute value of the sub-grid field exceeds the user-specified value *(e.g., amr_vel_sgs_embed* for velocity) in a given cell, CONVERGE applies embedding
- o If the absolute value of the sub-grid field is below one-fifth of the user-specified value, CONVERGE releases the cell (*i.e.,* removes the embedding)

- CONVERGE will stop refining the mesh at the user-specified embed level even if the target Y+ is not met
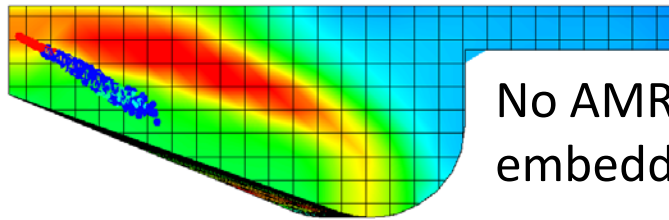
CONVERGE
CFD SOFTWARE

Examples of AMR
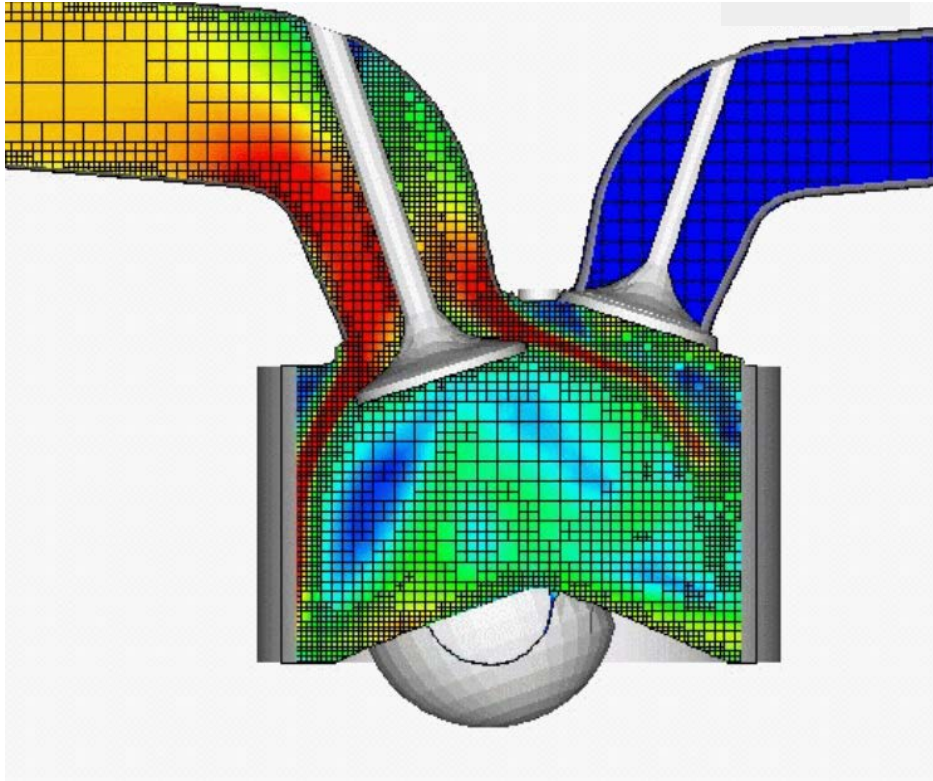
AMR capturing flapping jet found in intake flow simulation
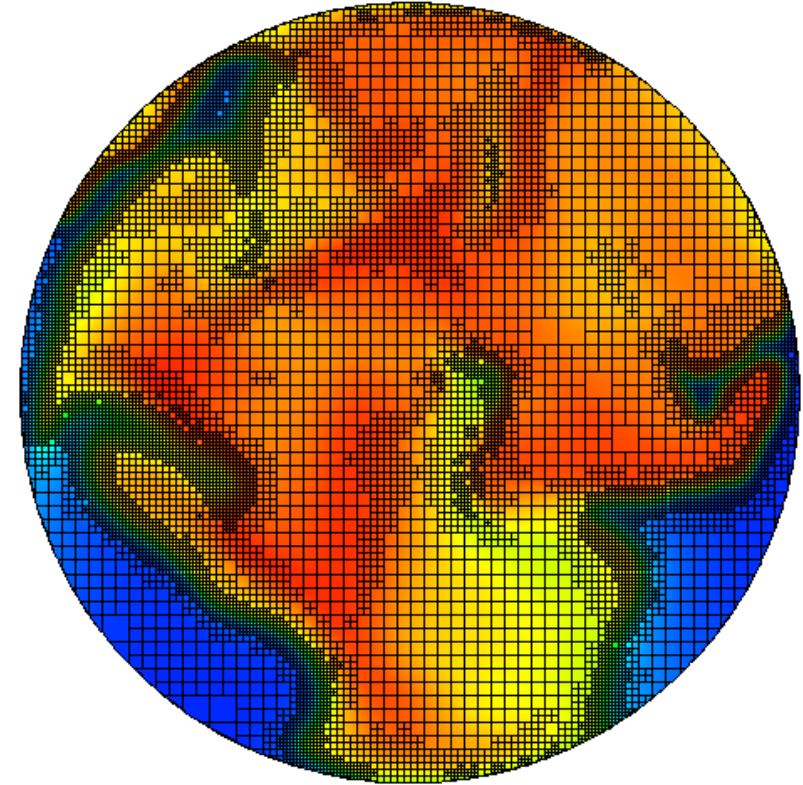
AMR capturing flame front for spark-ignited engine simulation using detailed chemistry

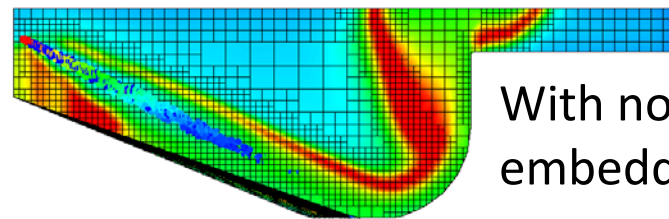CONVERGE
CFD SOFTWARE

Effects of Mesh Resolution
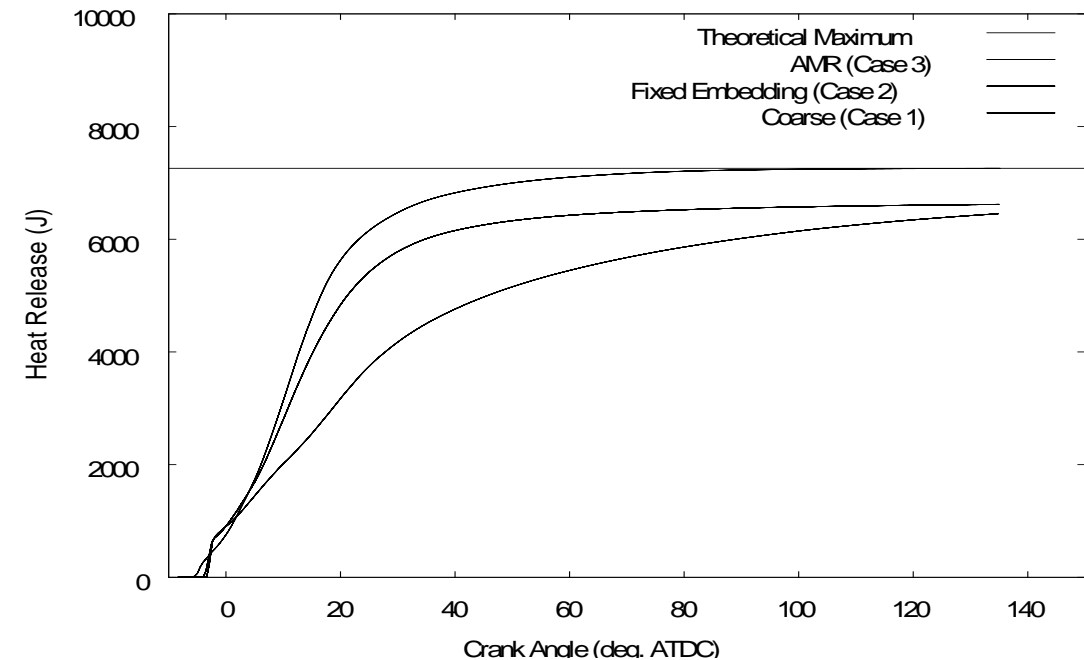
Temperature contours for diesel simulation

No AMR or embedding
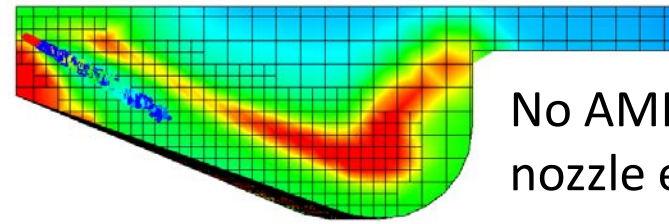
No AMR but with nozzle embedding

With nozzle embedding and AMR

Grid Resolution Recommendations

- Convergent Science provides example cases covering various applications

- These example cases are set up with our recommended physical, numerical, and grid settings

- These recommendations are based on our experience in obtaining better numerical predictions at an acceptable computational cost

- We recommend using these settings as a starting point for your simulation and then using your judgement in customizing these grid settings to better suit your particular problem

- We recommend performing grid sensitivity analyses to understand the accuracy/cost trade-off

CONVERGE
CFD SOFTWARE

THANK YOU!

CONVERGECFD.COM

CONVERGE
CFD SOFTWARE