Preparing the Piston and Valves for Motion

CONVERGE CFD SOFTWARE

© 2015 Convergent Science. All Rights Reserved.

CONVERGE Studio Workflow

- ***Case Setup*** **module**
  - Begin a project
  - Import the surface geometry
  - Prepare the surface
    - Prepare engine sector using *Make surface* tool
    - **Prepare piston and valves for motion**
  - Configure case setup
  - Export input and data files to the Case Directory

----------Run CONVERGE simulation----------

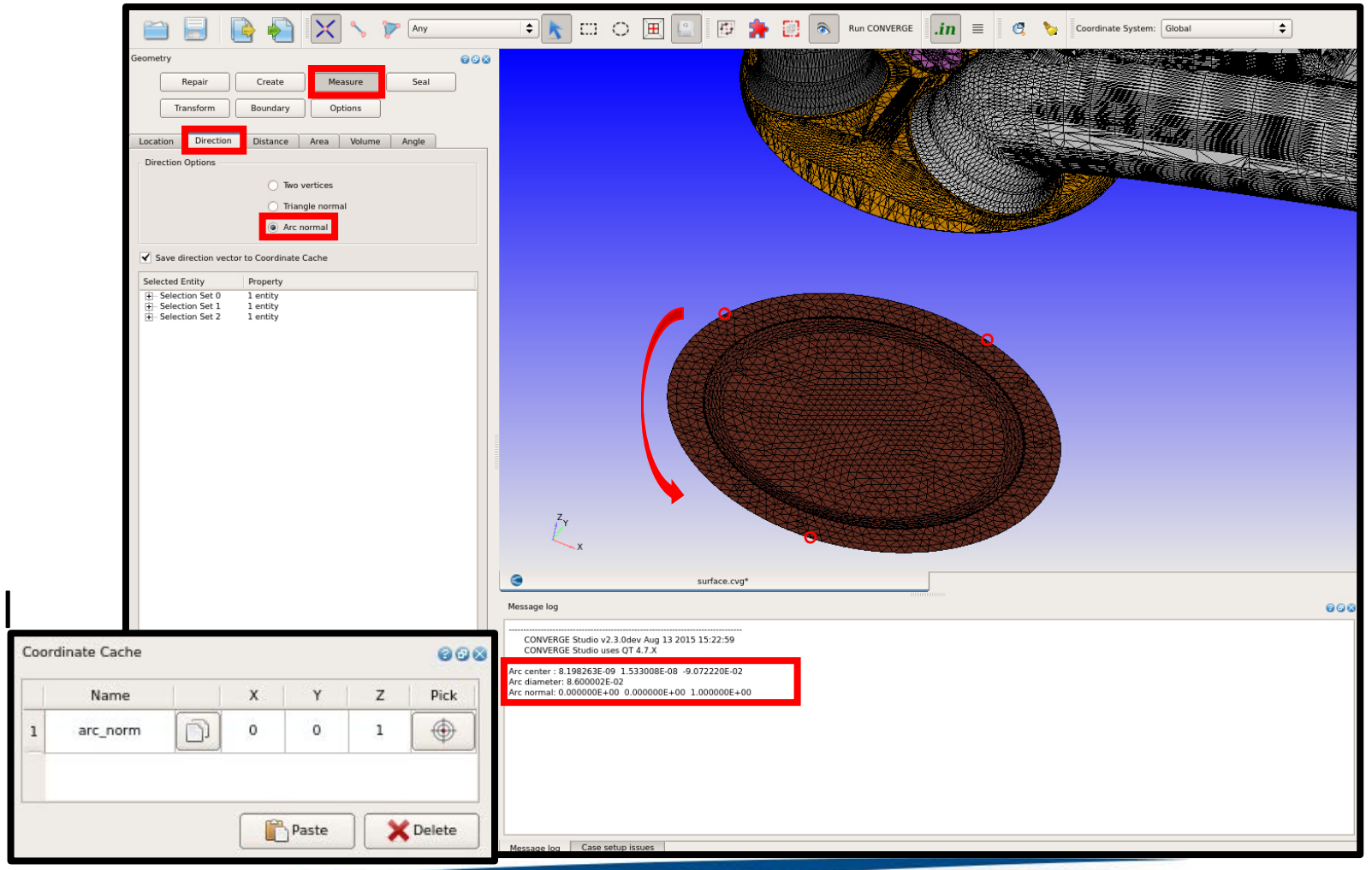
- *Line Plotting* module
- *Post-Processing 3D* module

Why Prepare the Piston for Motion

- When you import an **.stl* file, the piston and liner can become misaligned
  - Realign them before running a simulation
- You must ensure the piston motion for an inline engine is in the z direction for accurate velocity initialization and engine motion output
  - For V-engines, the piston motion direction has to be user-specified to obtain accurate velocity initialization and engine motion output
- An IC engine simulation must use the correct compression ratio, and the piston geometry must be located at bottom dead center (BDC)
  - Use the compression ratio tool in CONVERGE Studio to calculate the compression ratio and move the piston

Align the Piston with the Liner (1/3)

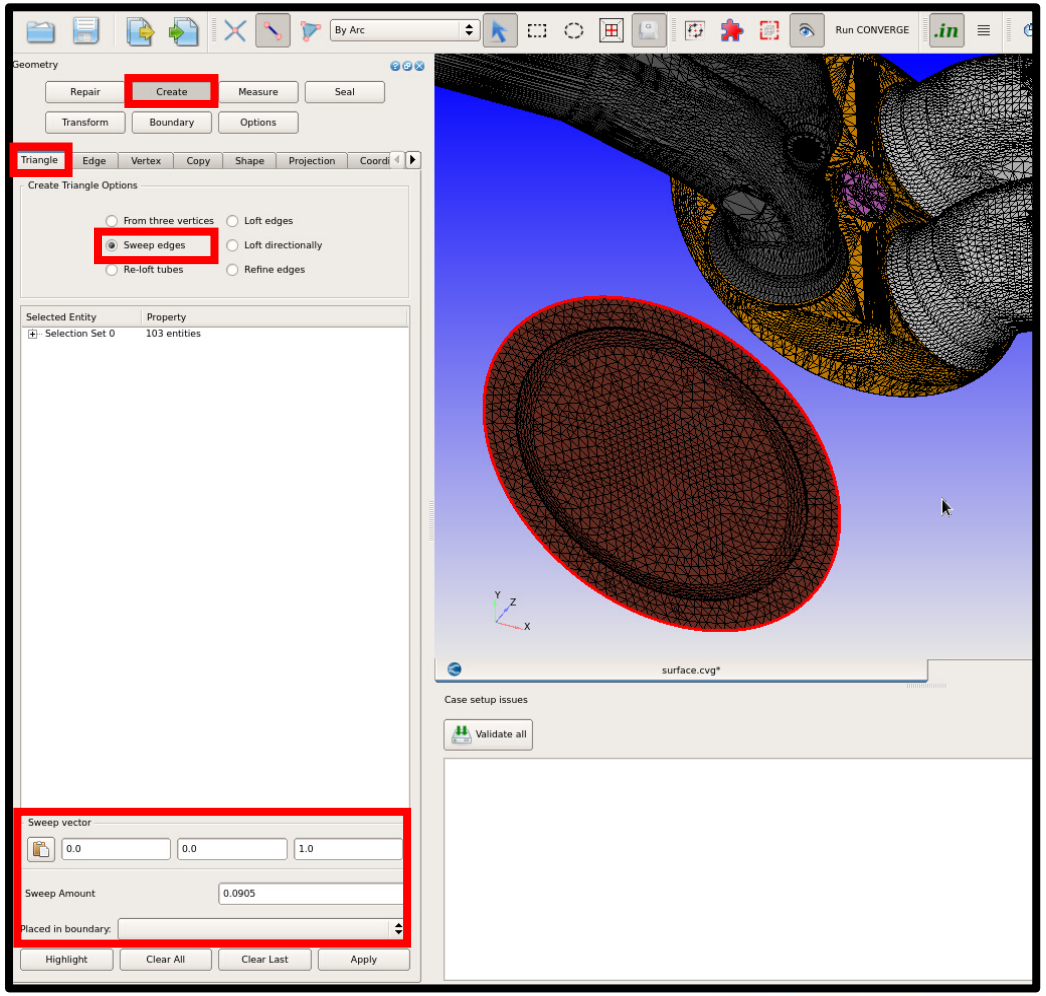
1) Go to *View Options* and hide the liner
2) Go to *Geometry* > *Measure* > *Direction* and click Arc normal
3) Select 3 vertices per the right-hand rule
4) Verify that the arc normal vector is (0.0, 0.0, 1.0)



CONVERGE CFD SOFTWARE

Align the Piston with the Liner (2/3)

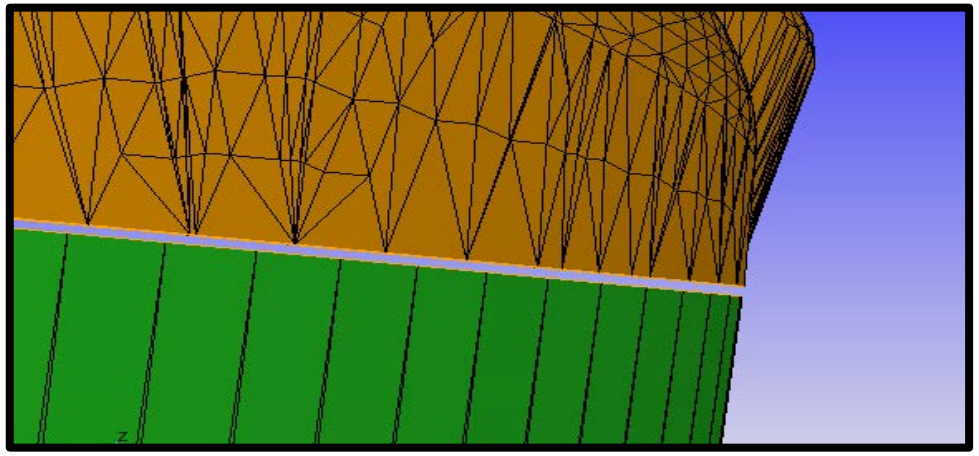
5) Go to *Geometry* > *Create* > *Triangle* and click Sweep edges
6) Select the piston edge
7) Set the Sweep vector to the arc normal
8) Set the Sweep Amount slightly larger than the stroke (an additional 0.0005 *meters*) and flag triangles to the liner boundary

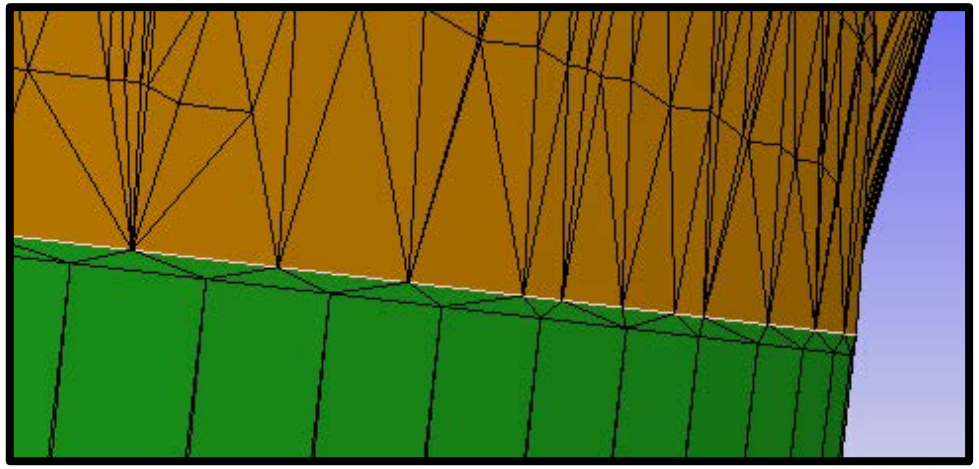


Align the Piston with the Liner (3/3)

There should be a small gap between the head and the liner

9) Go to *Geometry* > *Create* > *Triangle* and click Loft directionally
10) Select the liner edge, then the head edge
11) Set the Align vector to the arc normal





Compression Ratio Calculation

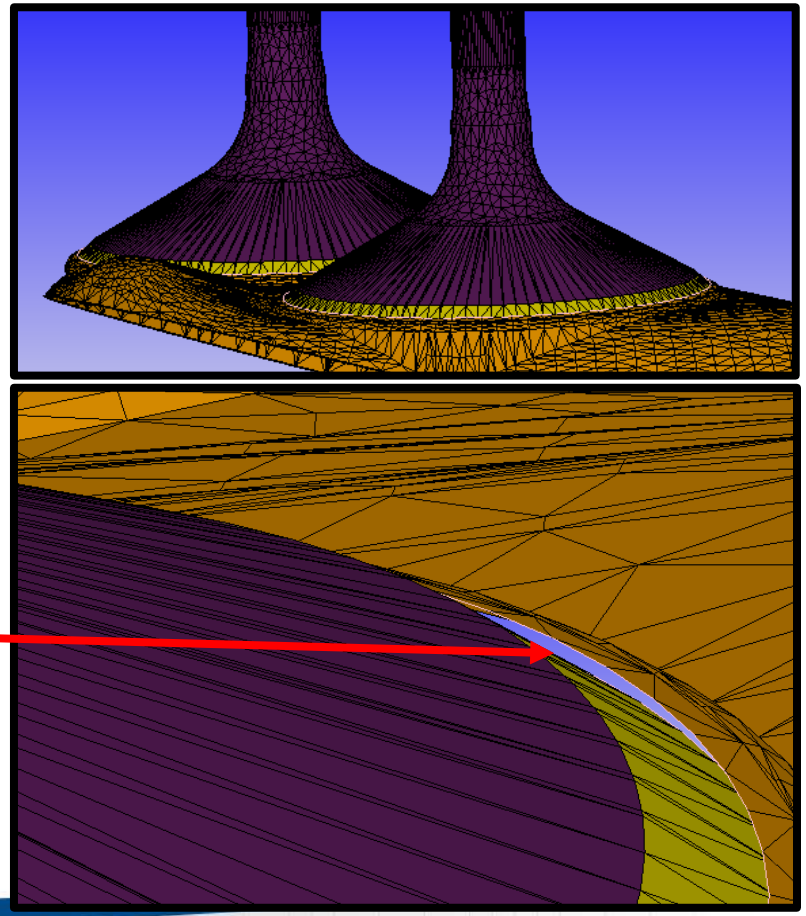
- The compression ratio (CR) is calculated as

$$CR = \frac{\text{maximum cylinder volume}}{\text{minimum cylinder volume}}$$

- CONVERGE Studio needs close any gaps in the domain in order to calculate the CR, which can be done two ways
  - Manually
  - By using the *Compression Ratio calculation* dialog box

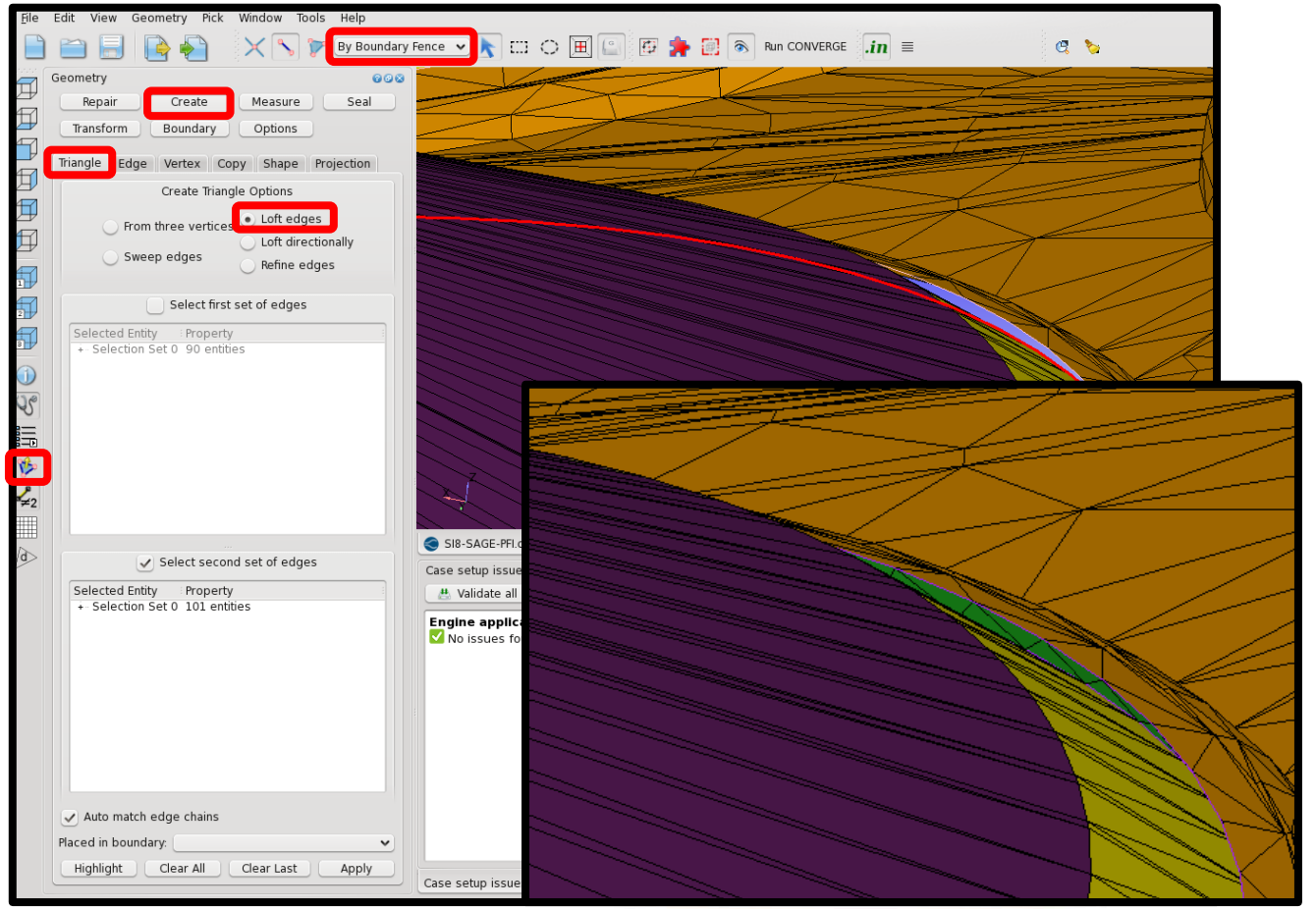
Manually Close the Combustion Chamber (1/2)

1) Go to *Geometry > Boundary > Fence* and click Reconstruct Fences from Existing Regions
2) Use *View Options* to hide the intake port and the liner
3) Rotate the geometry to see the open space between the head and one of the intake valves



Manually Close the Combustion Chamber (2/2)

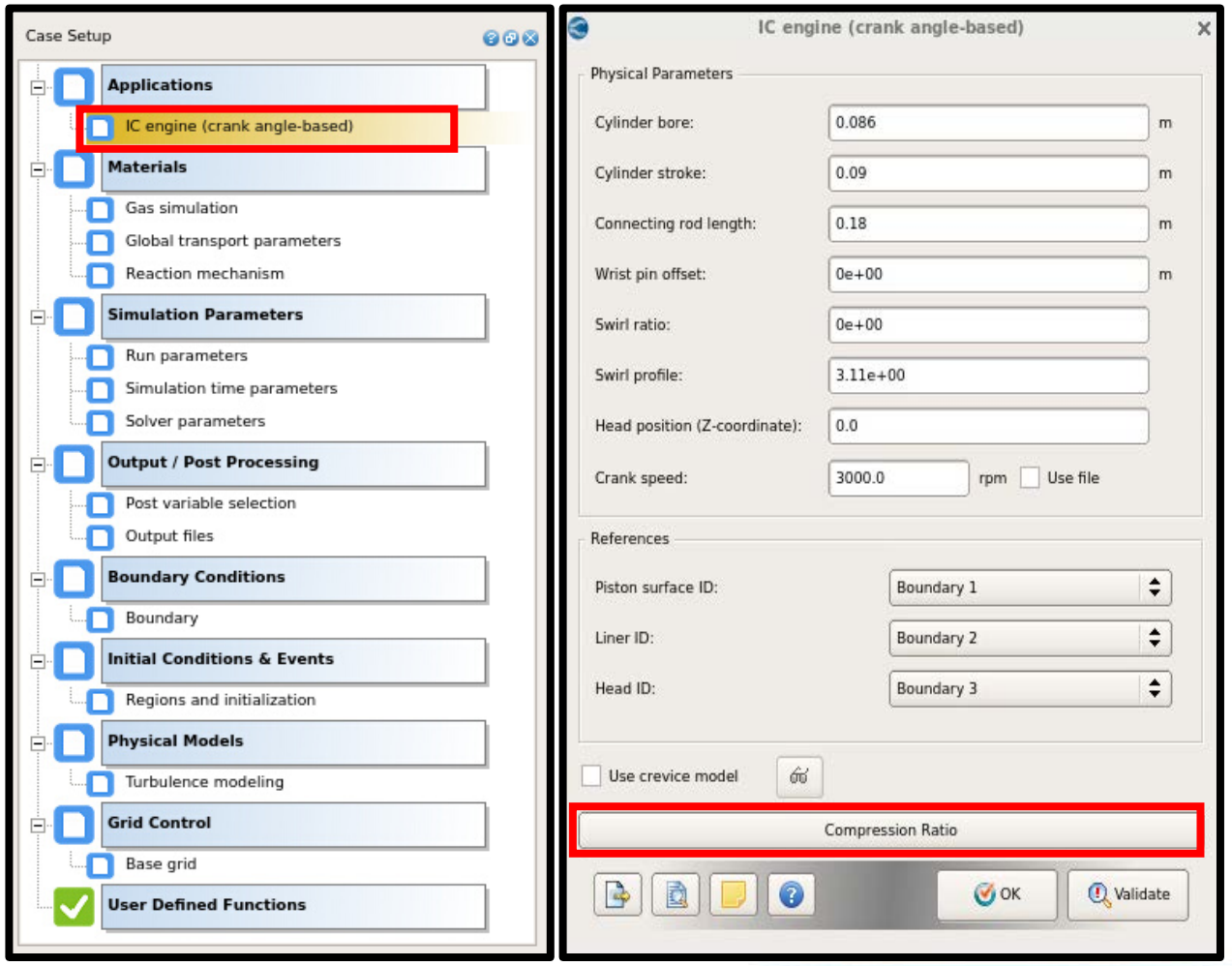
4) Go to *Geometry* > *Create* > *Triangle* and select Loft edges
5) Select one open edge, click Select second set of edges, and select the second open edge
6) Click Apply
7) Verify that the normal vectors of the new triangles point in towards the cylinder
8) Repeat steps 1-7 to close the gaps between the head and the remaining valves



CONVERGE CFD SOFTWARE

Calculate Compression Ratio and Move Piston (1/3)

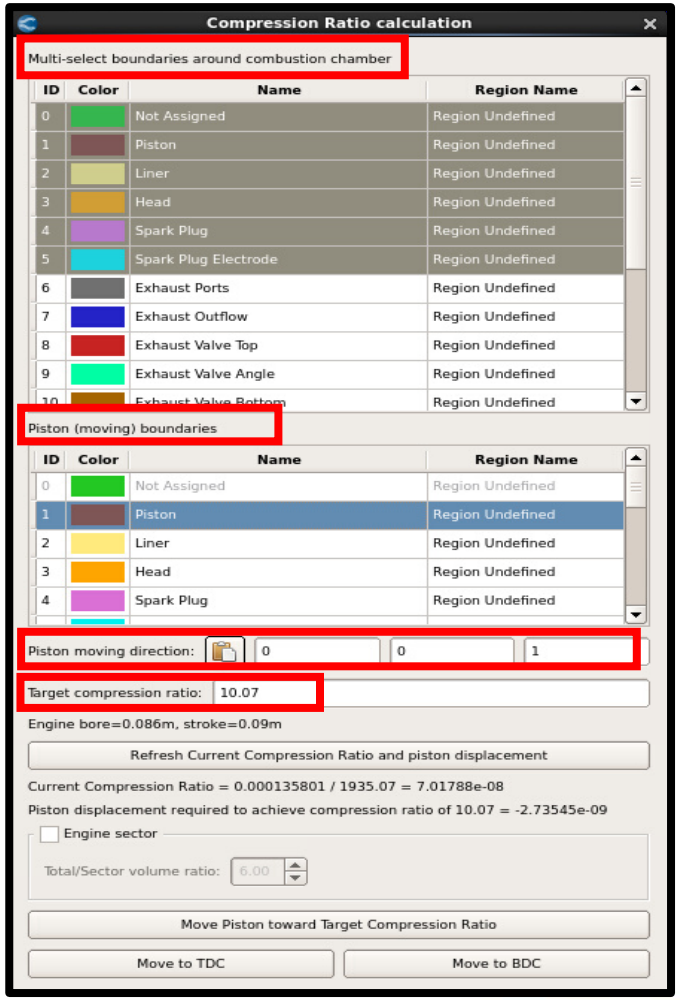
1) Go to the *Case Setup > IC engine (crank angle-based)*
2) Specify your engine parameters
3) Click Compression Ratio



CONVERGE CFD SOFTWARE

Calculate Compression Ratio and Move Piston (2/3)

4) Select the boundaries around the combustion chamber
5) Select the *Piston (moving) boundaries*
6) Change the Piston moving direction if the piston motion is not in the z direction
7) Set the Target compression ratio

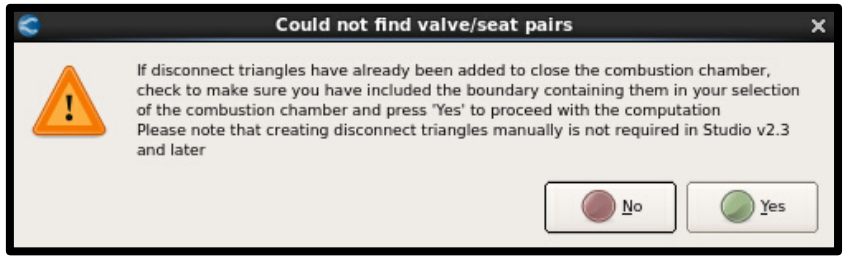


CONVERGE CFD SOFTWARE

Calculate Compression Ratio and Move Piston (3/3)

8) Click on the Refresh… button

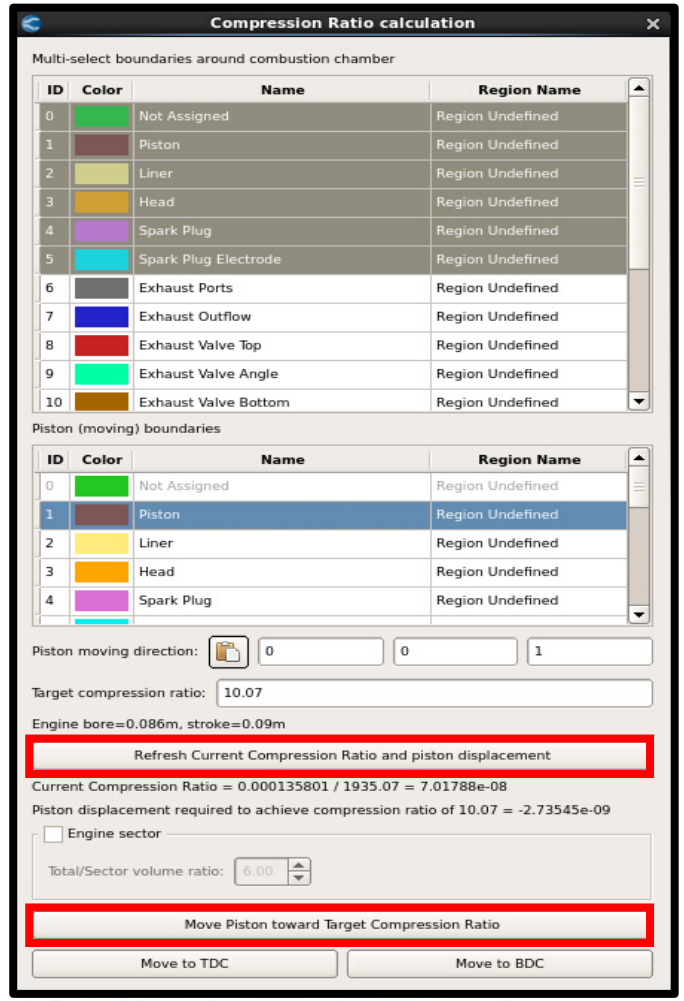
- If you manually closed the combustion chamber, a popup box will ask you to verify this; click Yes
  - If you click No, the calculation stops



- If you have not manually closed the combustion chamber, CONVERGE Studio does it and calculates the CR

9) Click Move Piston… until the current CR is acceptable

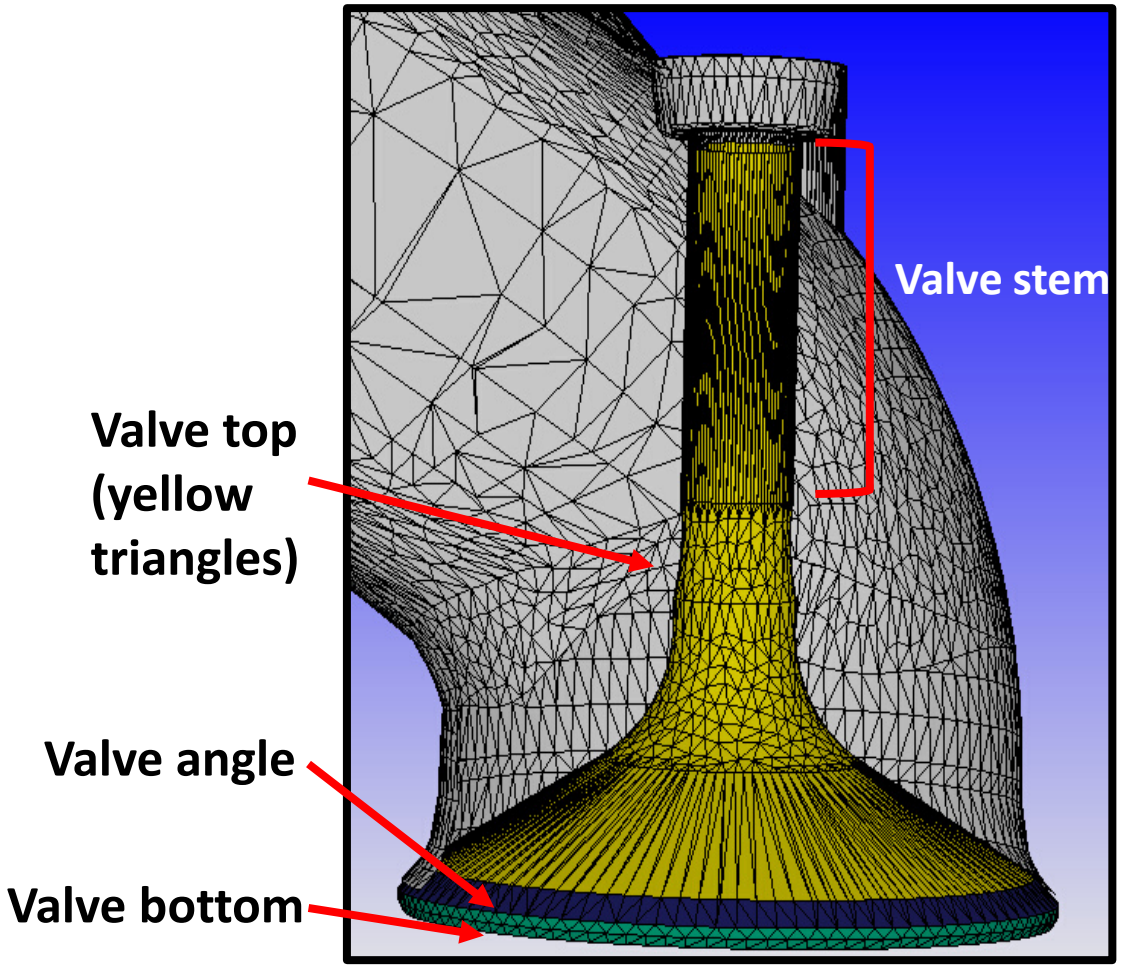
- When you click Move Piston…, CONVERGE Studio will move the piston and recalculate the CR



CONVERGE CFD SOFTWARE

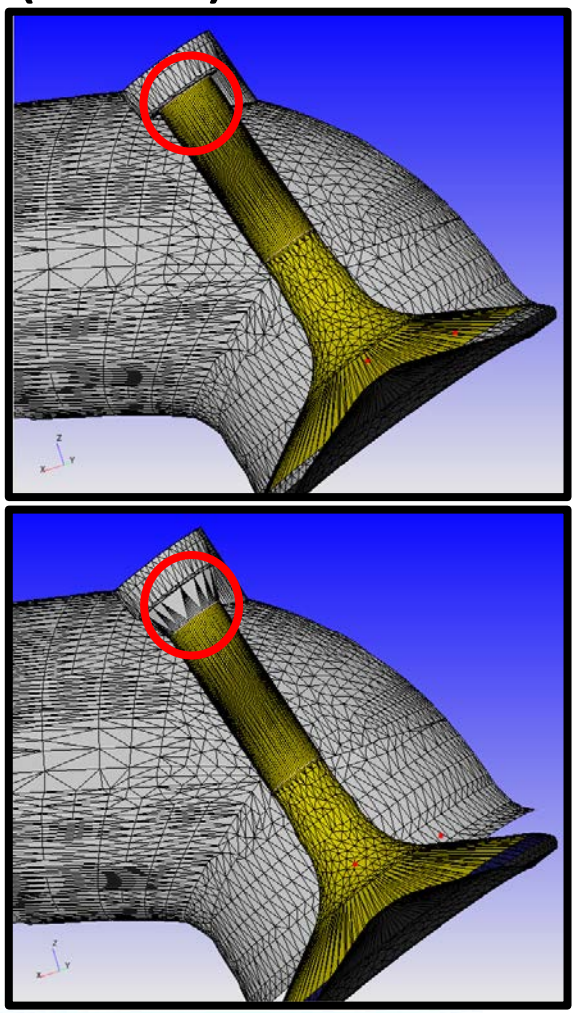
Why Prepare the Valves for Motion (1/2)

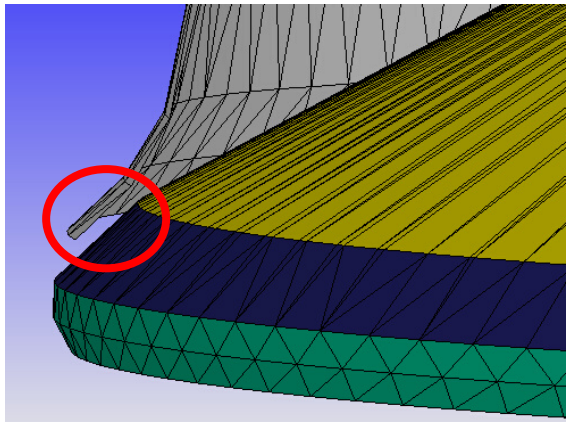
- When you import a surface geometry file, the valve stem may be misaligned with respect to the valve motion
  - The valve stem is the straight portion of the valve top
  - Realign the valve stem before running a simulation



Why Prepare the Valves for Motion (2/2)

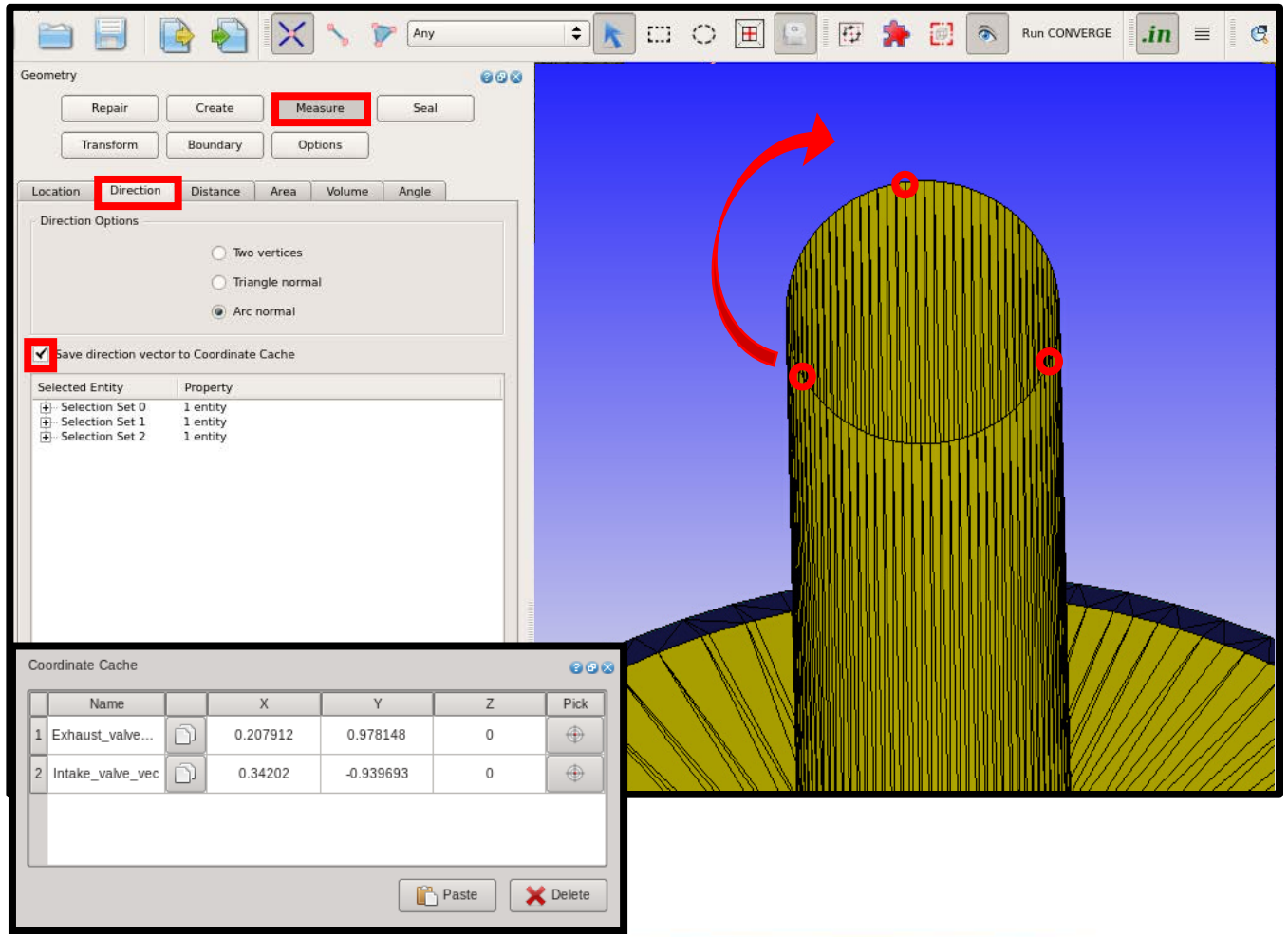
- Valve motion causes deformation in the surface geometry
  - You must create new connecting triangles between the port and the valve stem
- To avoid intersection defects, the valves must be at an open position
  - We recommend a minimum lift of 0.2 *mm*





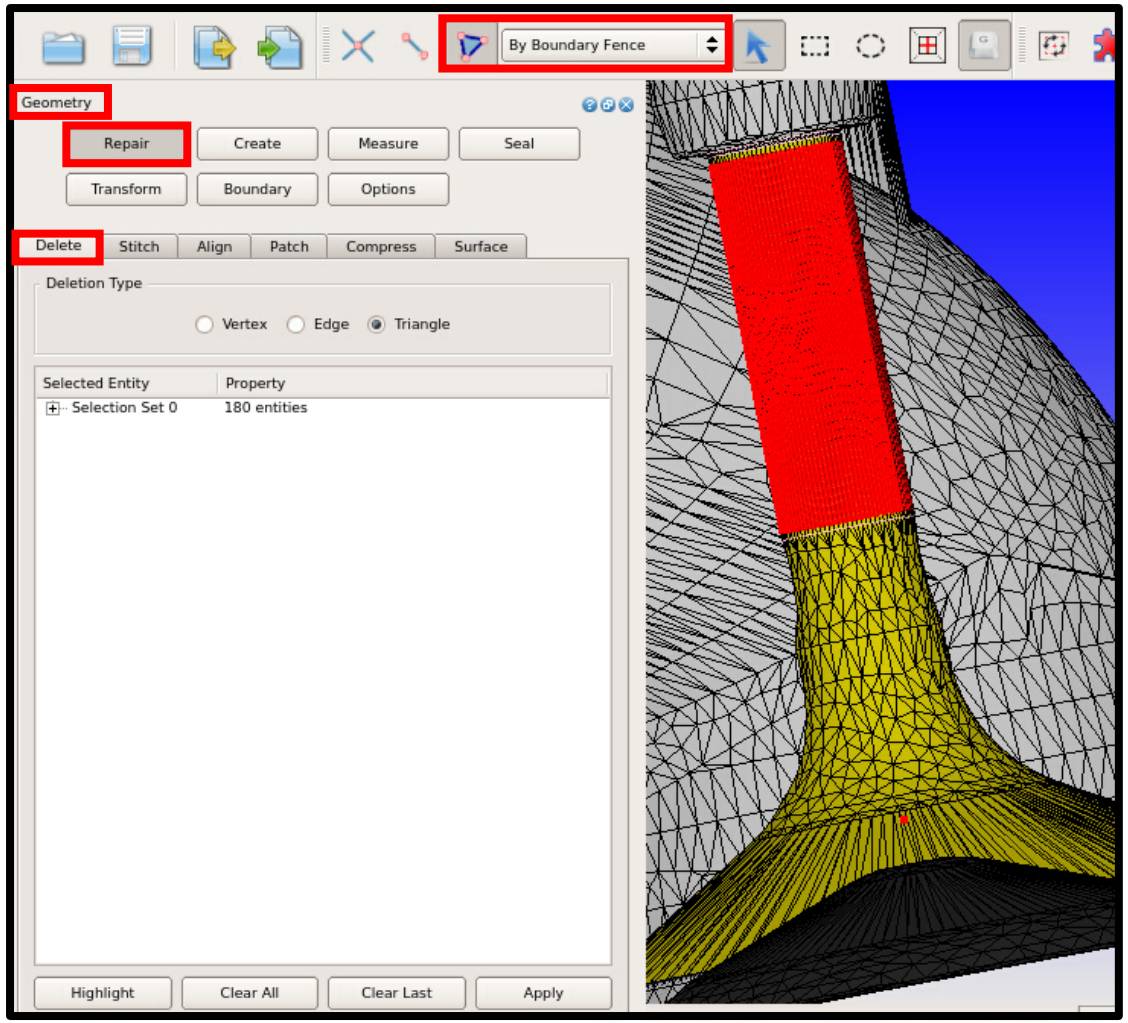
Measure Direction of Valve Motion

1) Go to *Geometry* > *Measure* > *Direction* and click Arc normal
2) Select three vertices along the same arc
3) Verify that Save direction vector… is checked
4) In the *Coordinate Cache*, rename the vector for easy identification
   - Open the cache via *View* > *Coordinate Cache*



Delete the Valve Stem

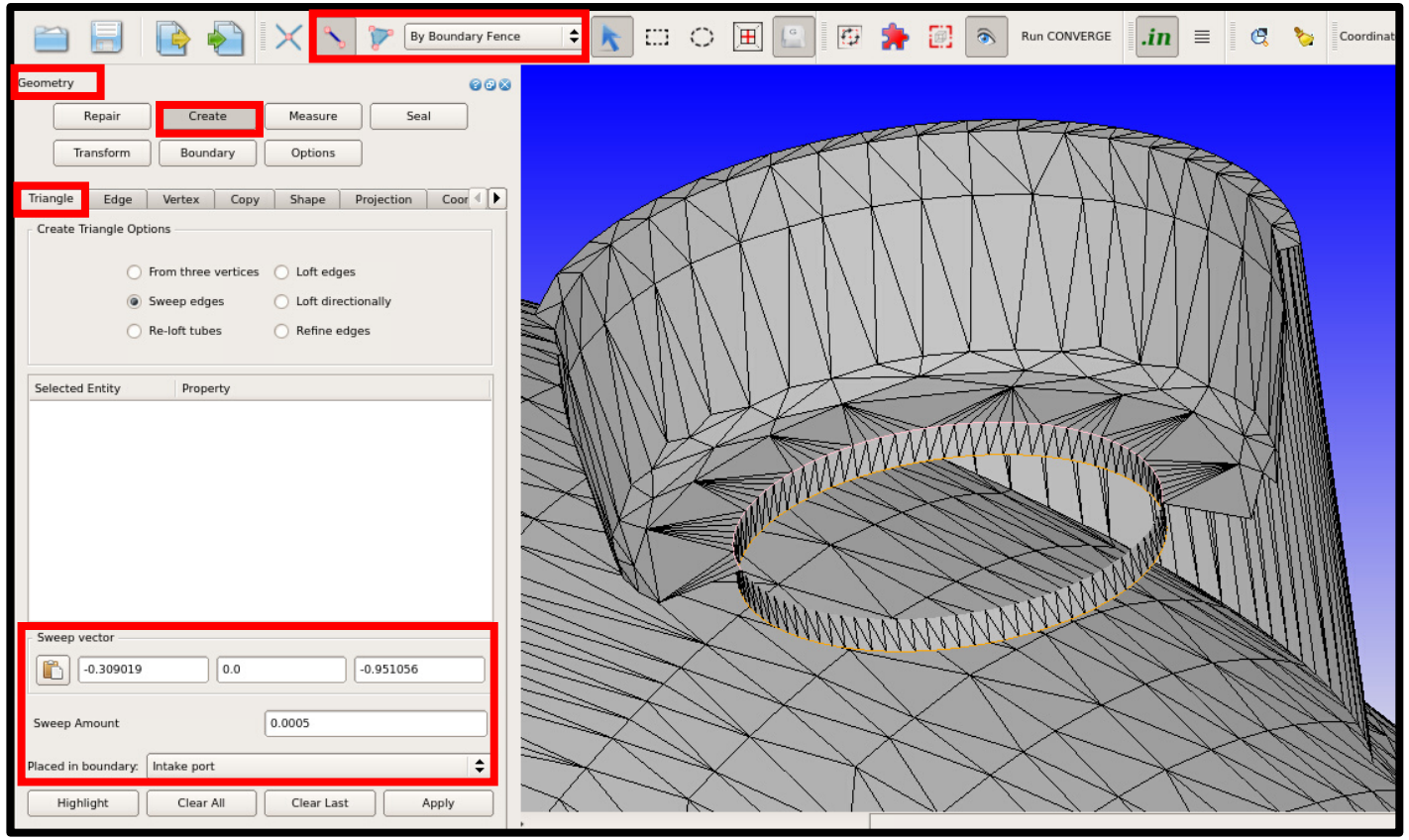
1) Place boundary fences around the top and bottom of the valve stem
2) Go to *Repair > Delete > Triangle*
3) Select the existing valve stem triangles <u>By Boundary Fence</u>



CONVERGE CFD SOFTWARE

Sweep Edges at Port

1) Go to *Geometry* > *Create* > *Triangle* and click Sweep edges
2) Use the  button to paste the valve motion vector into the Sweep vector field
3) Enter a small Sweep Amount (*e.g.*, 0.5 *mm*)
4) Select the port hole By Boundary Fence
5) Flag the newly created triangles to the correct boundary

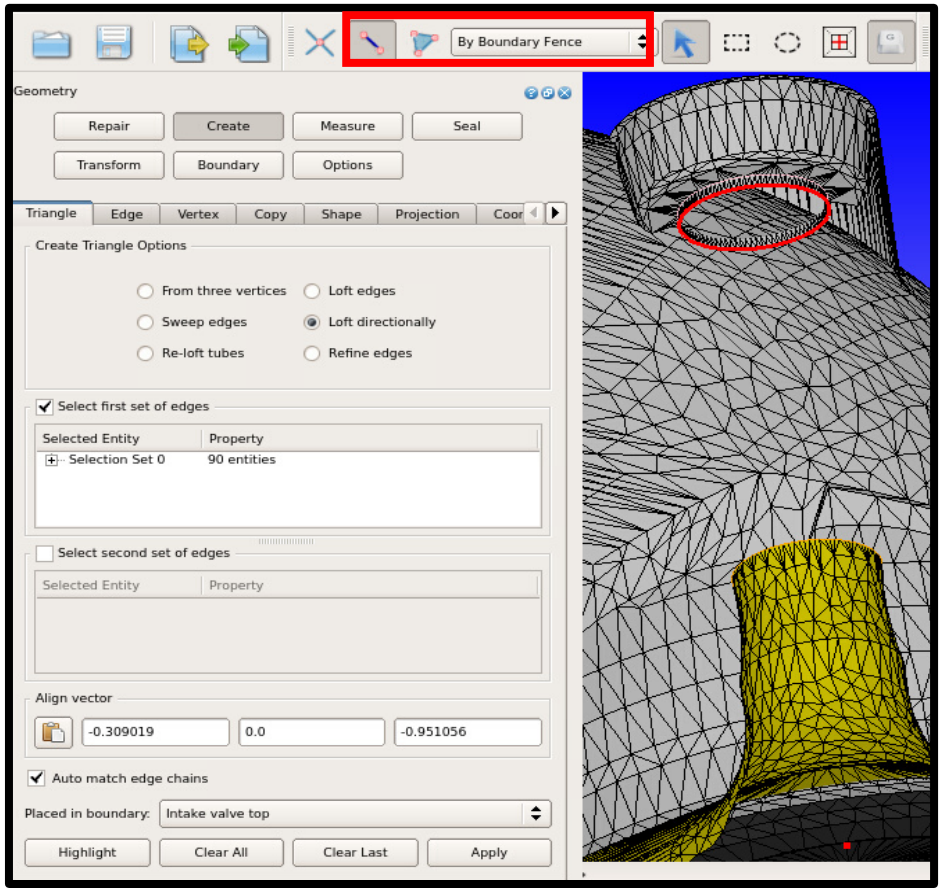


CONVERGE CFD SOFTWARE

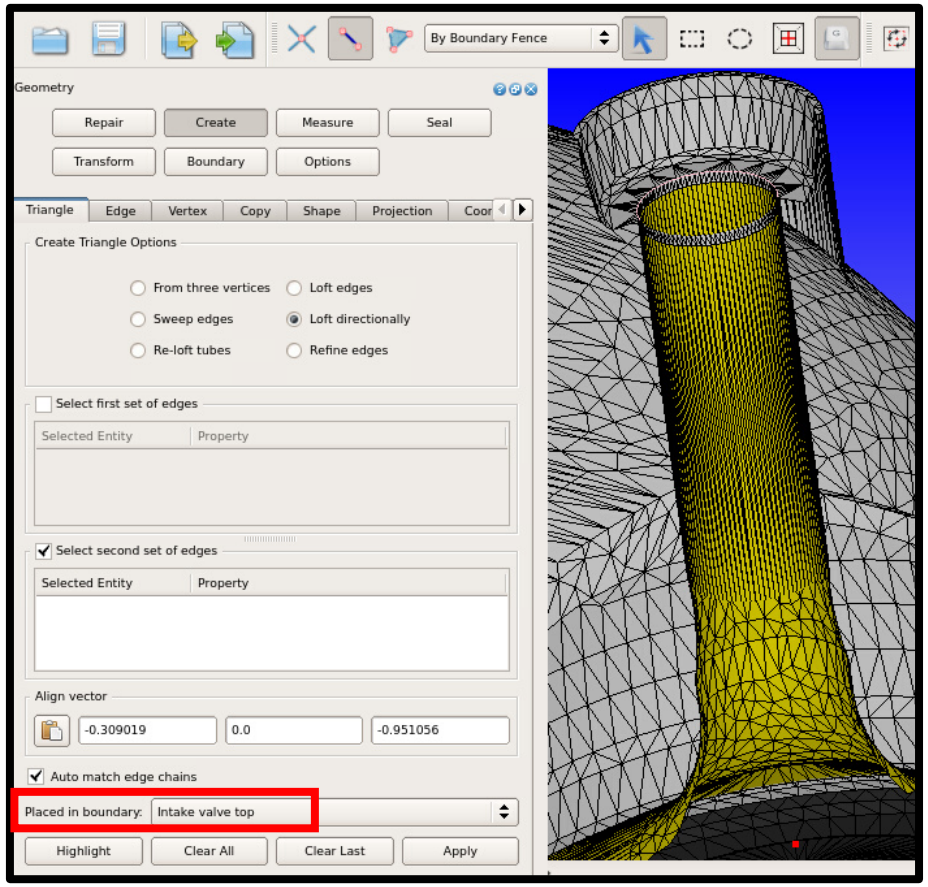
Connect Valve Top to Swept Edges (1/2)

1) Go to *Geometry* > *Create* > *Triangle* and click Loft directionally
2) Verify that boundary fences are in place and that Select first set of edges is selected and then select the bottom edge of the newly created triangles at the port
3) Check Select second set of edges and select the ring of edges at the top of the valve
4) Use the *Coordinate Cache* to paste the valve motion vector into the Align vector field

Connect Valve Top to Swept Edges (2/2)



Select the swept edges and the top of the valve using the By Open Edge or By Boundary Fence selection

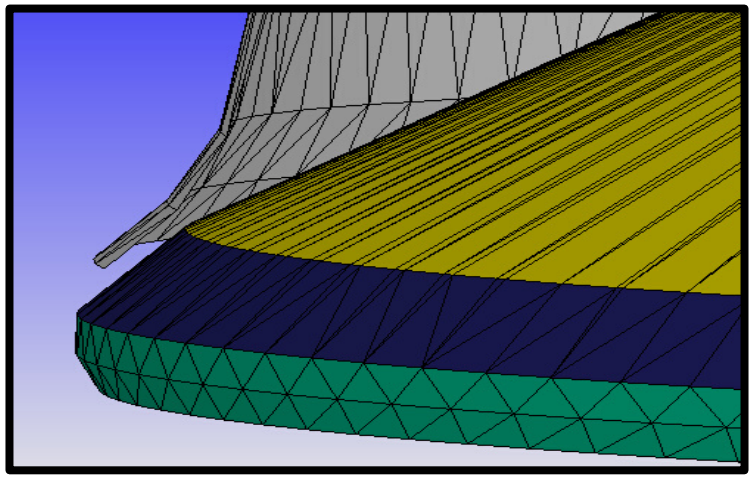


Automatically flag newly aligned triangles to the valve top boundary

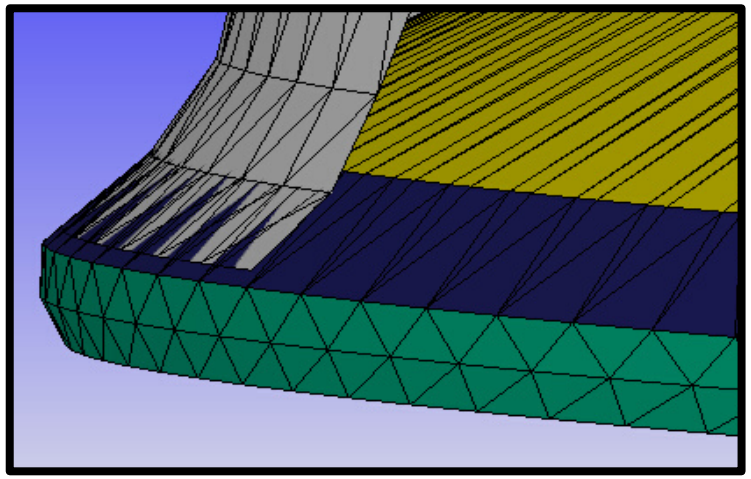
Move Valves to Minimum Lift (1/2)

1) Go to *Geometry* > *Transform* > *Translate* and click Selected Boundary
2) Use the button to select the saved valve motion vector
3) Select the valve top, angle, and bottom boundaries
4) Use a small Translation Amount (*e.g.*, 0.00001 *m*) to move these boundaries until the valve angle triangles no longer intersect the valve seat triangles
5) Move these three boundaries approximately 0.2 *mm* **away** from the valve seat (*e.g.,* toward the cylinder)

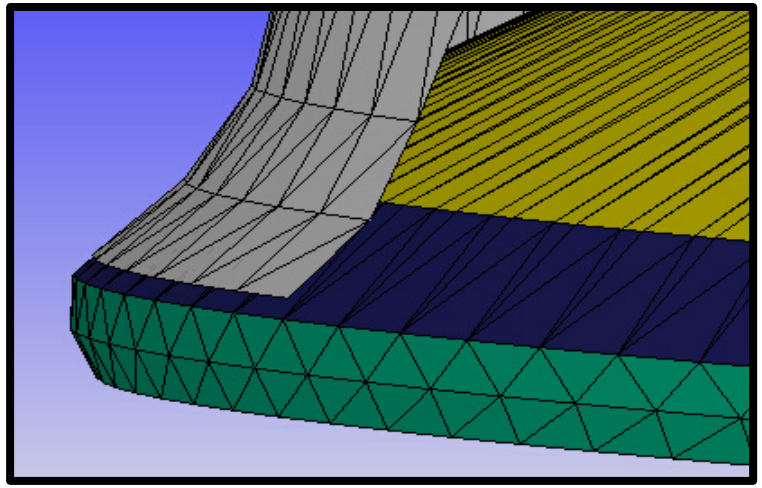
Move Valves to Minimum Lift (2/2)



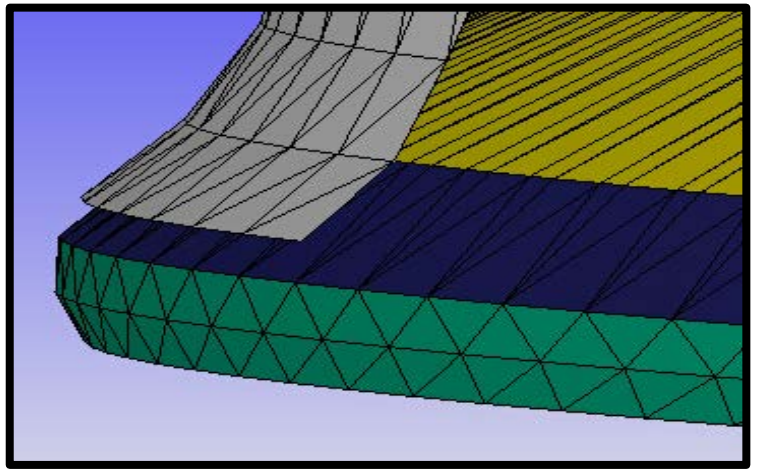
After importing a geometry, the valve may sit far below the valve seat



After importing a geometry, the valve angle may intersect the valve seat



After small translations, the valve angle is barely separated from the valve seat

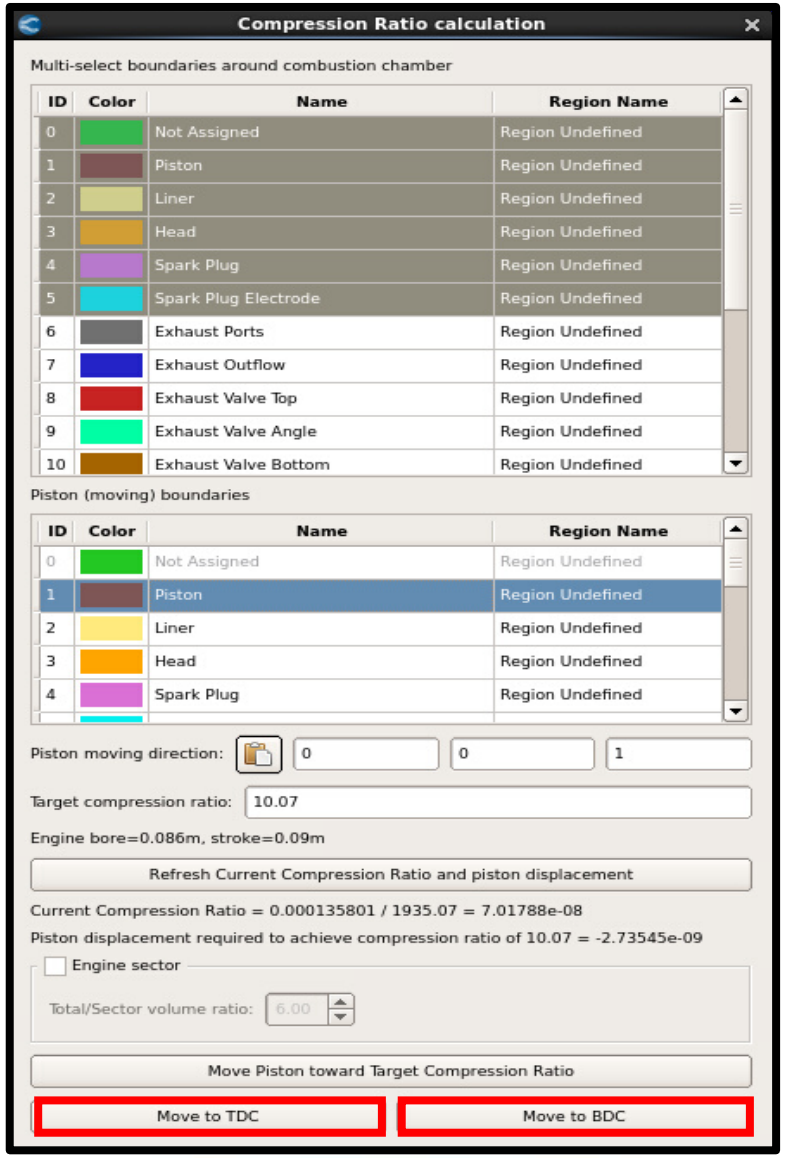


The valve sits 0.2 *mm* from the valve seat

Check for Errors (1/2)

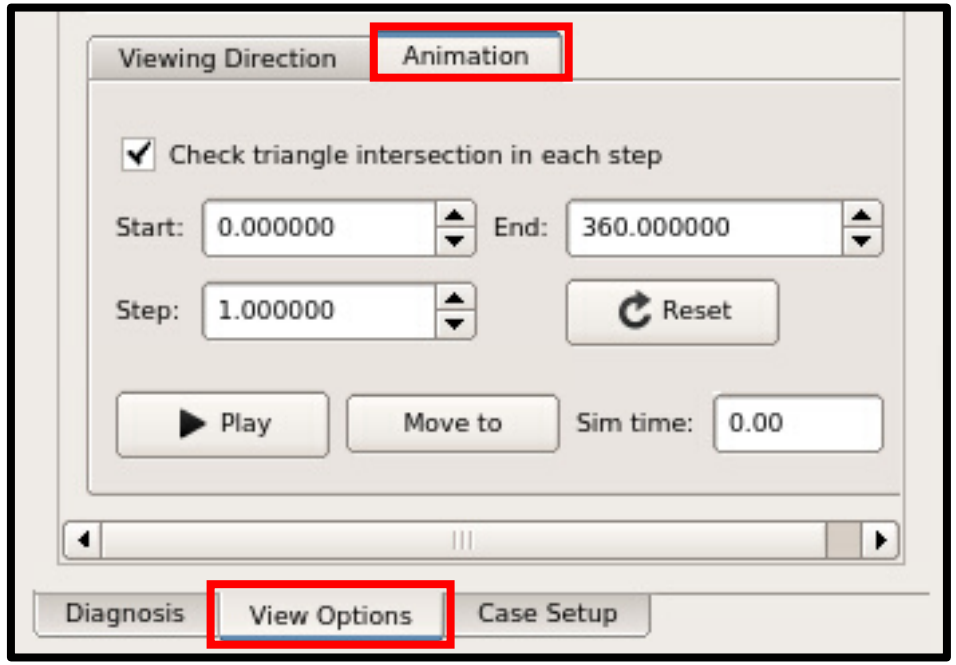
- You can check for errors in the piston motion using the *Compression Ratio calculation* dialog box

1) Click Move to TDC
2) Use the *Diagnosis* dock to identify errors (*e.g.*, intersections or overlapping triangles) at top dead center (TDC)
3) Use the tools in the *Geometry* dock to resolve any errors
4) Repeat these steps using Move to BDC



Check for Errors (2/2)

- In *View Options*, you can animate the geometry to check for intersection defects due to piston and valve motion
    1) Check the Check triangle… box
    2) Set start and end times along with the time step in between
    3) Click Play to set geometry into motion
- The *Message log* displays an error message if an intersection defect occurs



THANK YOU!

CONVERGECFD.COM

CONVERGE CFD SOFTWARE

© 2015 Convergent Science. All Rights Reserved.