CFD Simulation

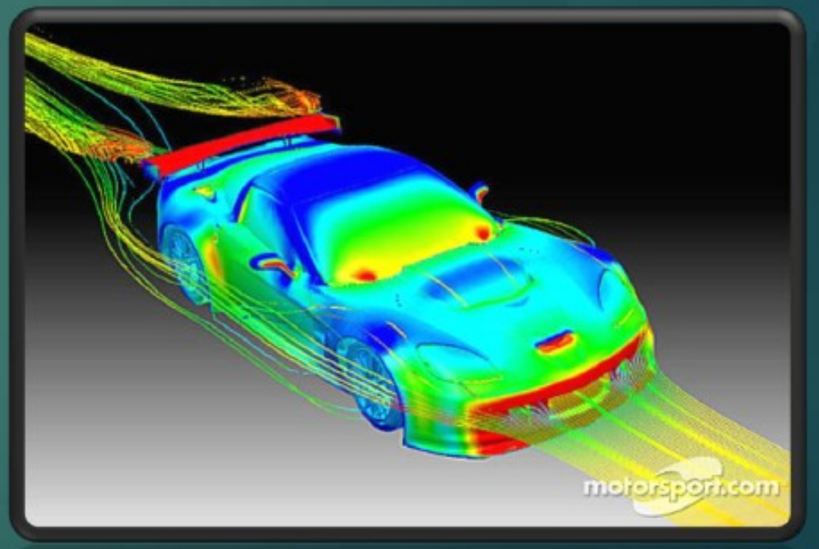
——take the gas extraction model as a sample

What is CFD?

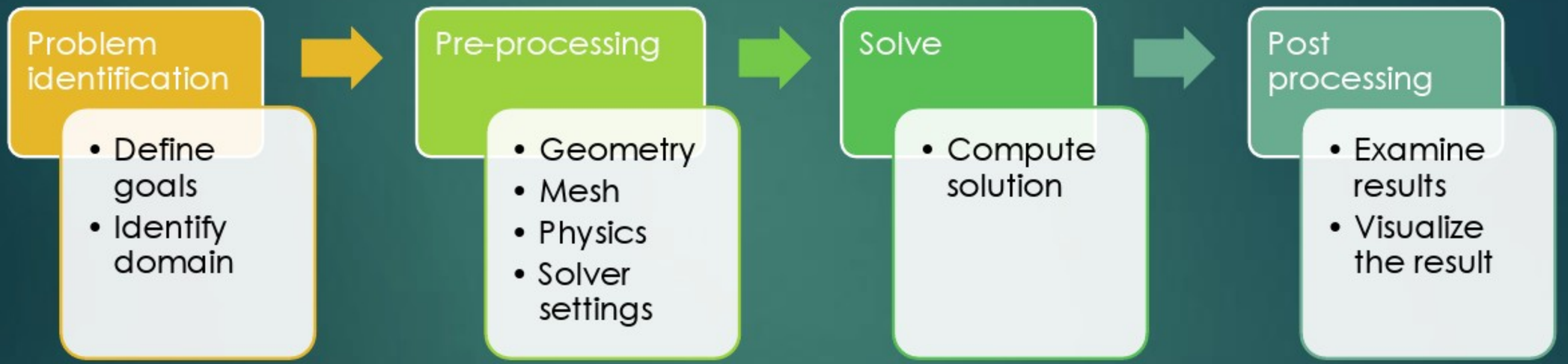
- CFD is the abbreviation of **Computational fluid dynamics** , which is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows, heat and mass transfer, chemical reactions and other relevant physical phenomena.

- CFD simulation is generally applied as follows
  - ✓ conceptual design
  - ✓ the detailed design of product
  - ✓ find problem and improve design

- CFD is the supplement of physics experiment



The process of CFD simulation

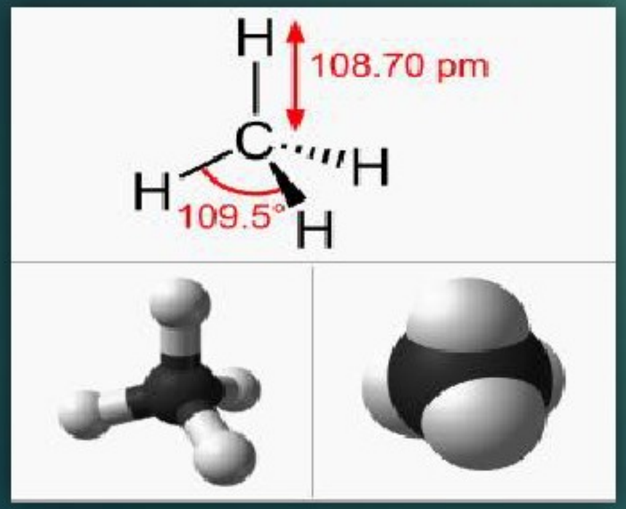


The application in goaf gas extraction

- Problem identification

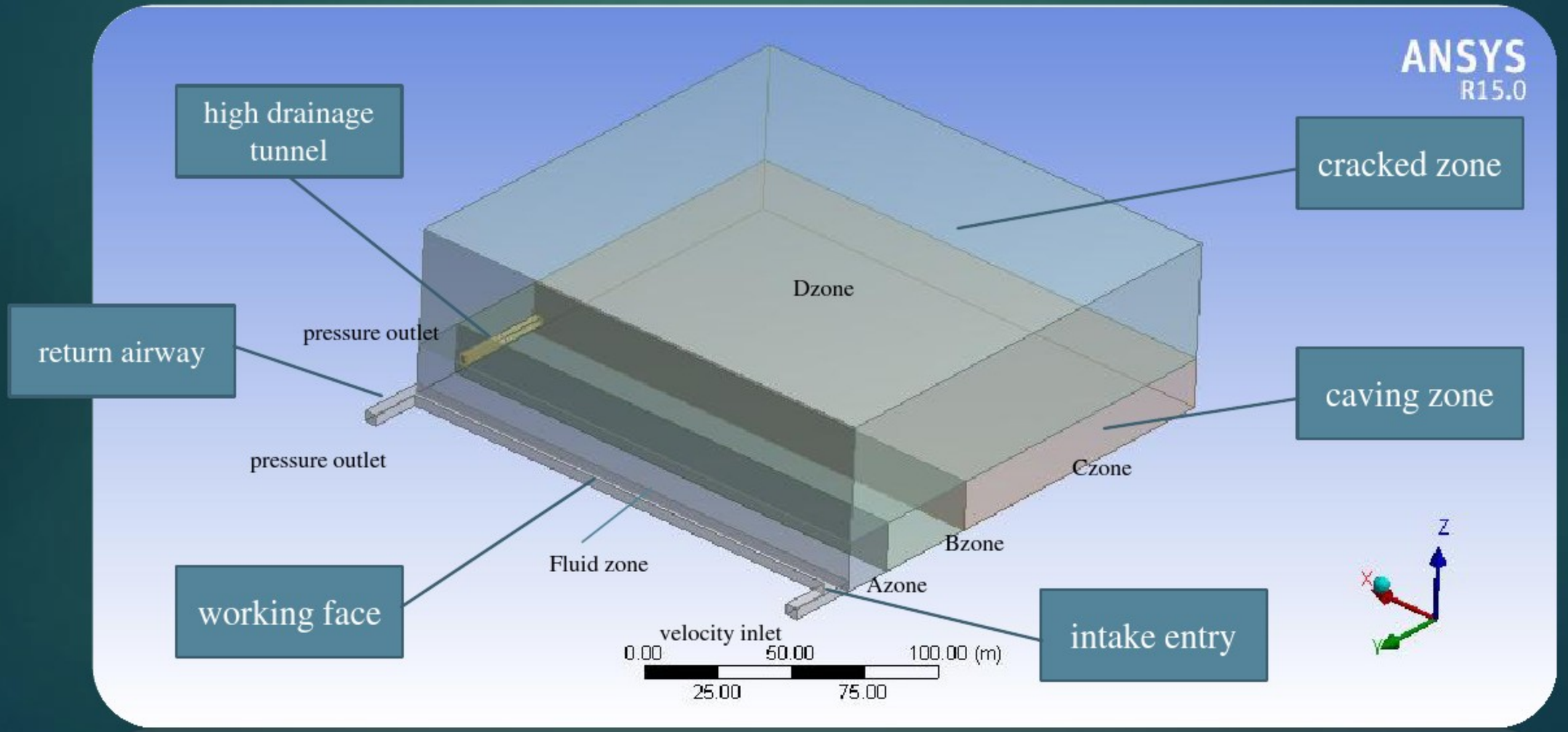
Gas of fully mechanized goaf is one of the main parts of the workplace gas emission. So research workface goaf gas flow pattern and the implementation of drainage measures in mined-out area is to reduce the face amount of gas emission and the upper corner of the primary means of transfinite. The current ventilation extraction methods cannot solve the gas problem, so we intend to build a high drainage tunnel on the cracked zone in goaf.

- Goal: to verify the gas extraction effect of high drainage tunnel by CFD simulation.

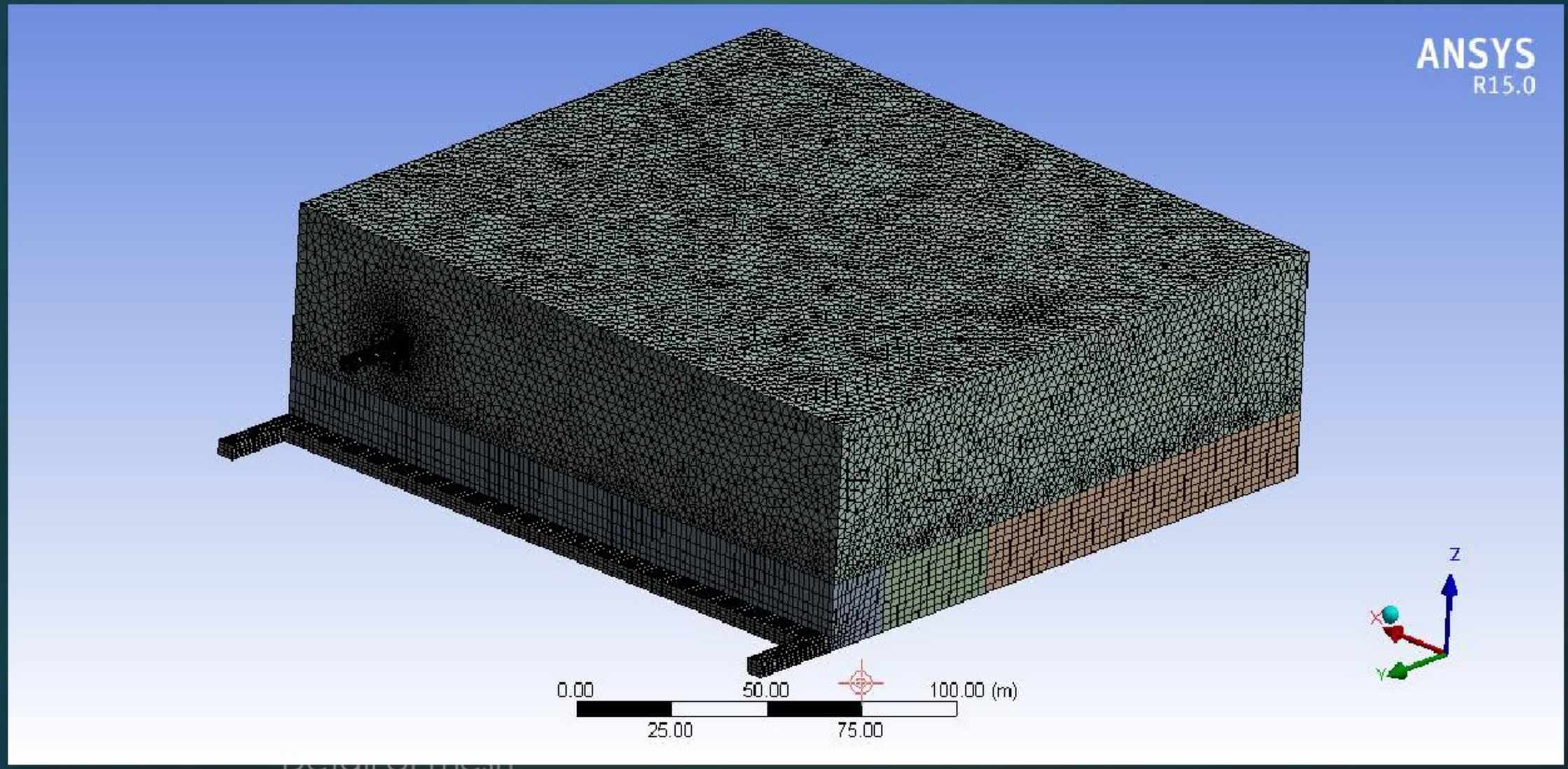


Pre-processing

The physical model of goaf



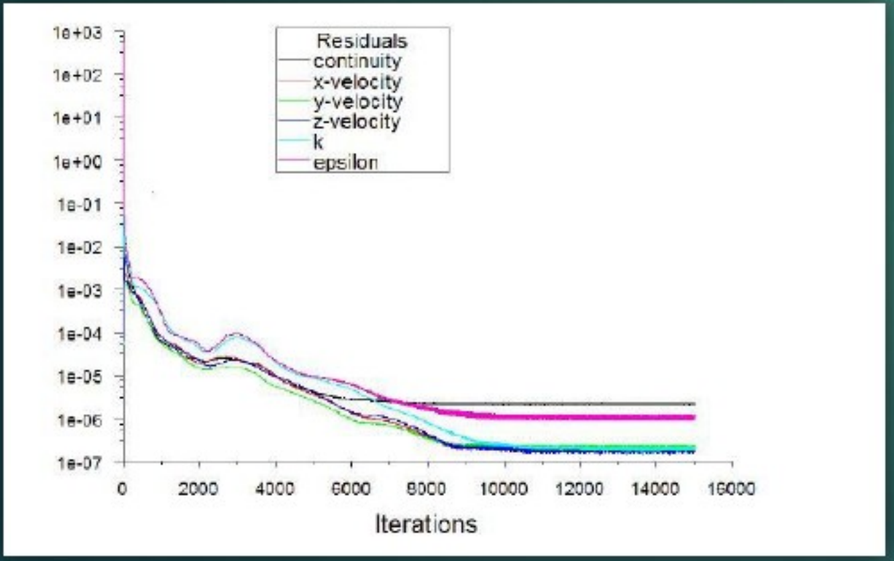
Topology and mesh



Detail of mesh

Solve

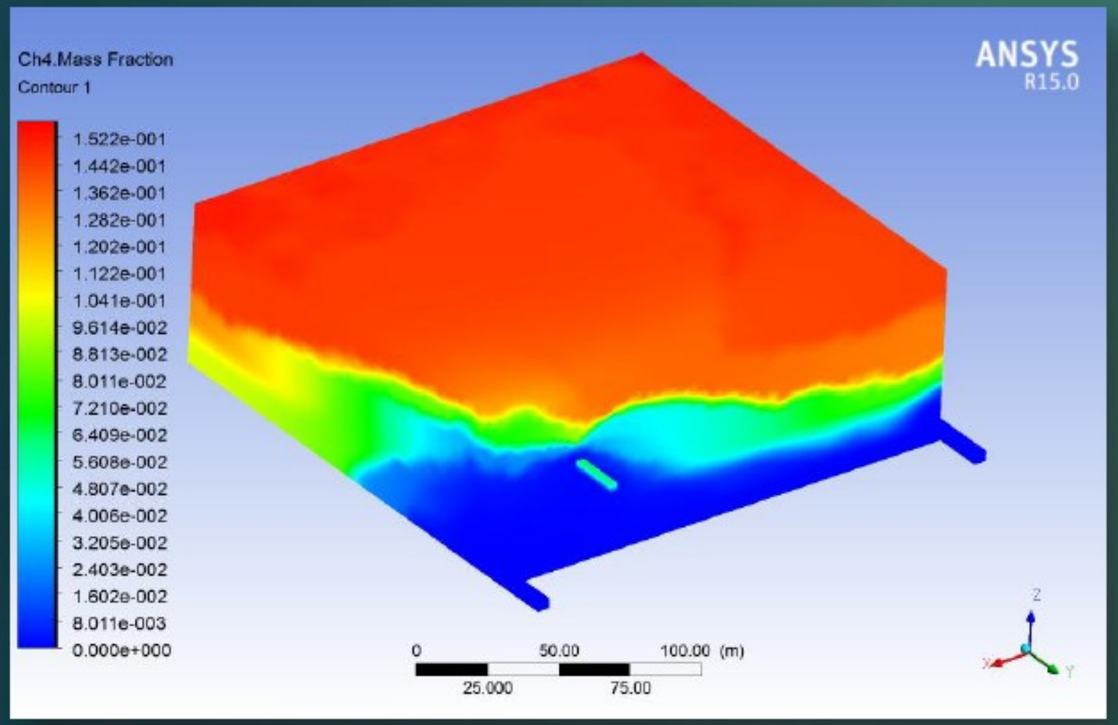
- Boundary conditions:
  - the velocity of inlet(the entrance of fluid) is 2.5m/s
  - the pressure of outlet(the exit of fluid) is 3500pa
  - the calculate model as standard k-ε turbulent model
  - the simulation type is the species transport ,which contains methane and air.



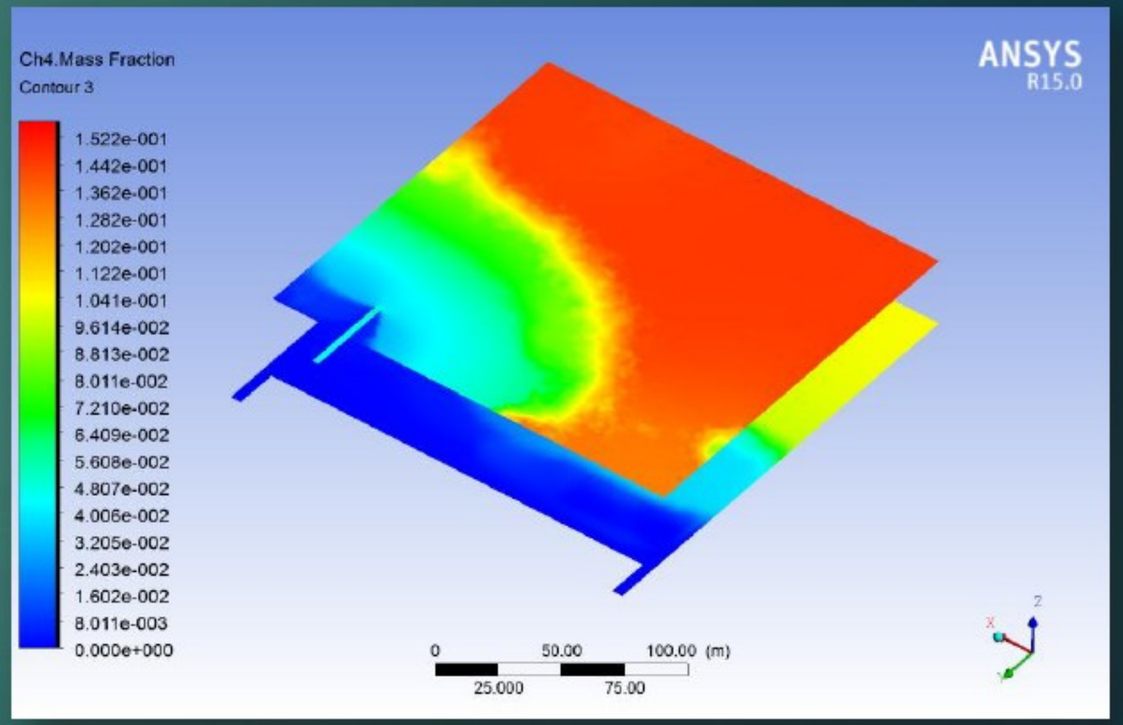
- ✓ solve the conservation equations with iterative computation until they are converged.
- ✓ A converged and mesh- independent solution on a well- posed problem will provide useful engineering results*!*

Post processing

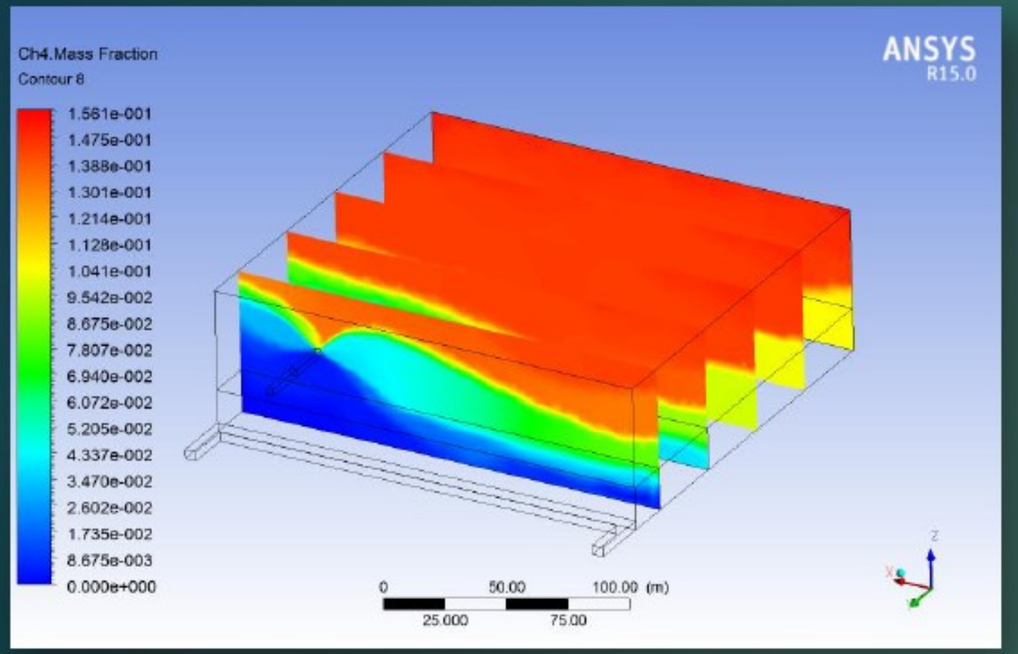
- After the calculation finished, the simulation data will be exported as visible forms.



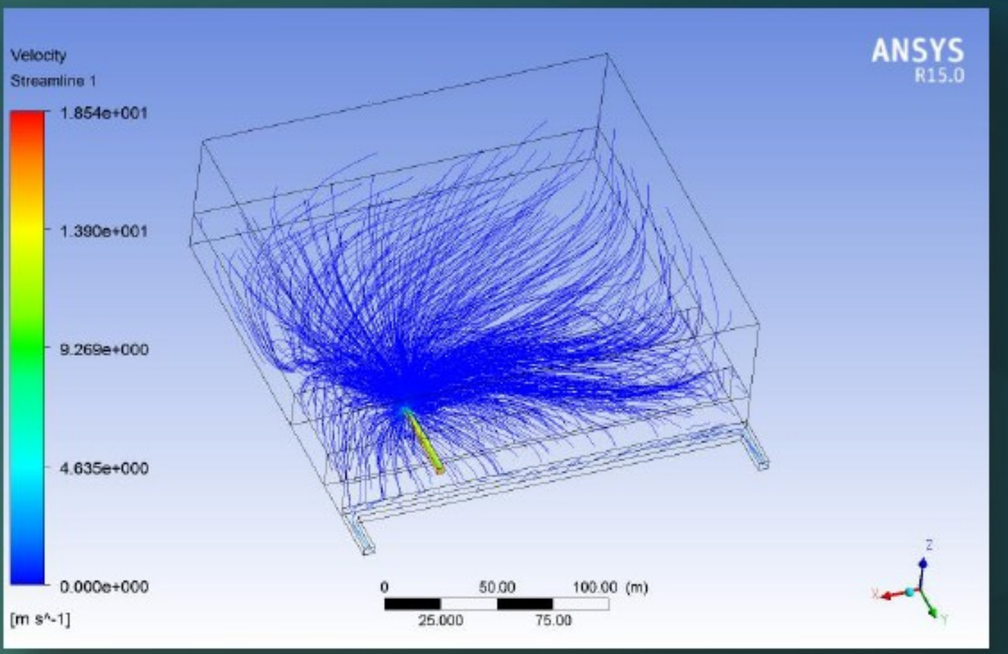
The overall methane（ch4）fraction contour



Methane（ch4）fraction of different height
（Slice contour）



The ch4 mass fraction on strike (slice contour)



The fluid streamline in goaf

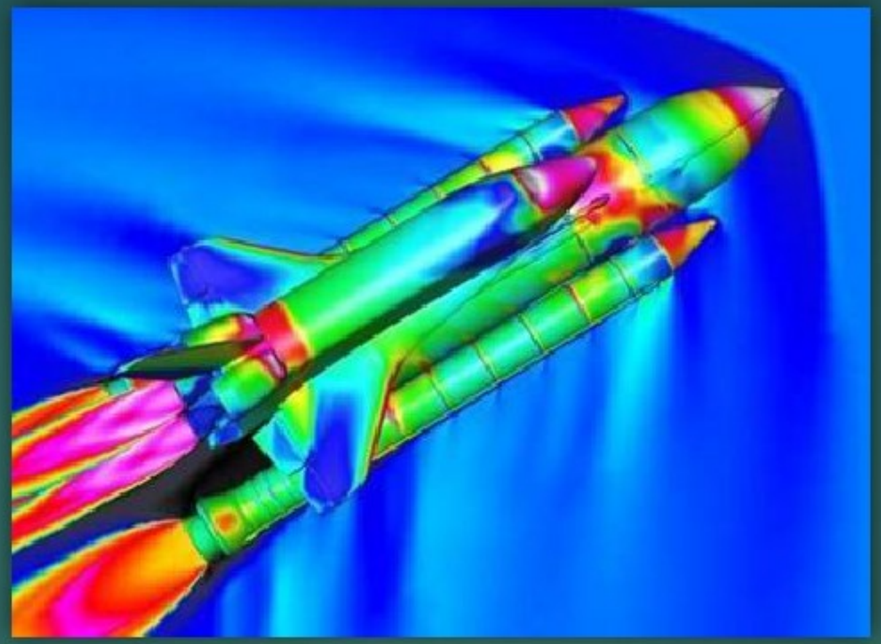
- The high drainage tunnel extracted most of the gas that are emitted from the goaf, decreased the ch4 mass fraction (<0.75%) of working face and upper corner, ensured the efficiency and safety of working face.

Conclusion

- In all of CFD approaches the same basic procedure is followed.

1. During preprocessing The geometry (physical bounds) of the problem is defined.
2. The volume occupied by the fluid is divided into discrete cells (the mesh).
3. The physical modeling and boundary conditions are defined.
4. The simulation is started and the equations are solved iteratively as a steady-state or transient.
5. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

- According to the CFD simulation of goaf gas extraction model, the high accuracy of this technology is obvious, through the CFD numerical simulation we can solve a lot of engineering problems without experiment. With the optimized operation, The CFD technology can be a powerful tool for engineering.



Thank you for watching !