CFD modelling of an accidental pressurised gas release

POLITECNICO DI TORINO

Gianmario Ledda, Alberto Moscatello, Anna Chiara Uggenti*
Department of Energy – Politecnico di Torino, Italy
**gianmario.ledda@studenti.polito.it*

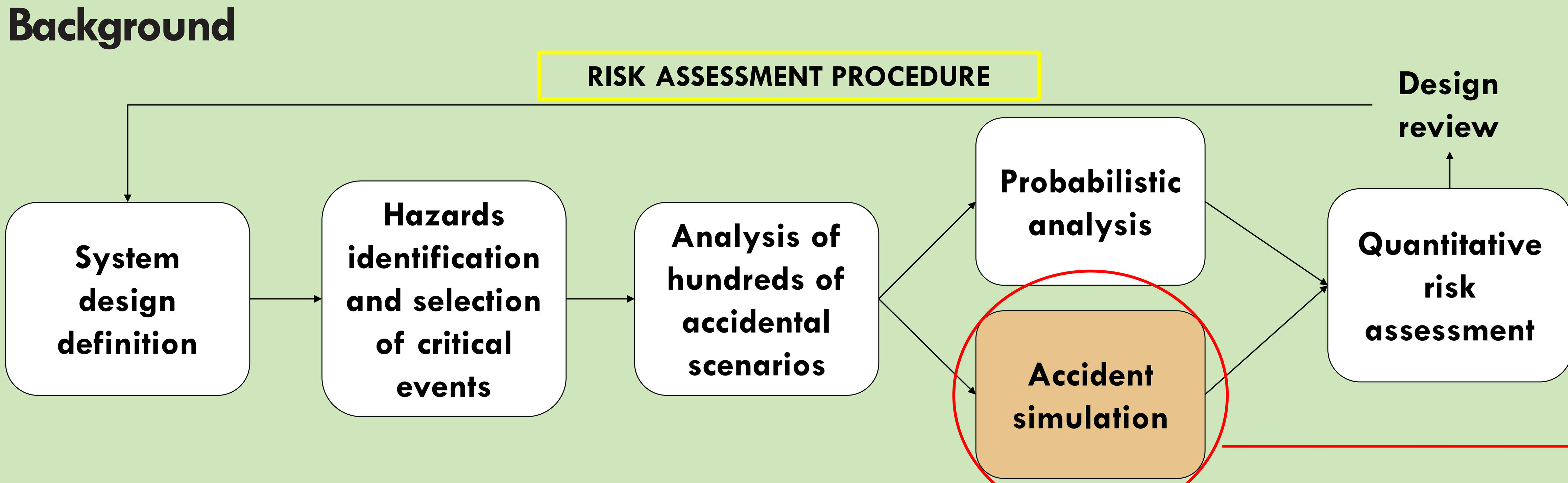
Objective of the project

What	Where	When	How	Why
Damage area for release of hazardous pressurized gases	Industrial plants subject to relevant accident risk	Design and construction phases	Two-Steps CFD model using ANSYS Fluent	Model multiscale and multiphysic phenomena in complex geometries

Risk assessment usually requires to simulate hundreds of different accidental scenarios in order to identify the most potentially critical events.

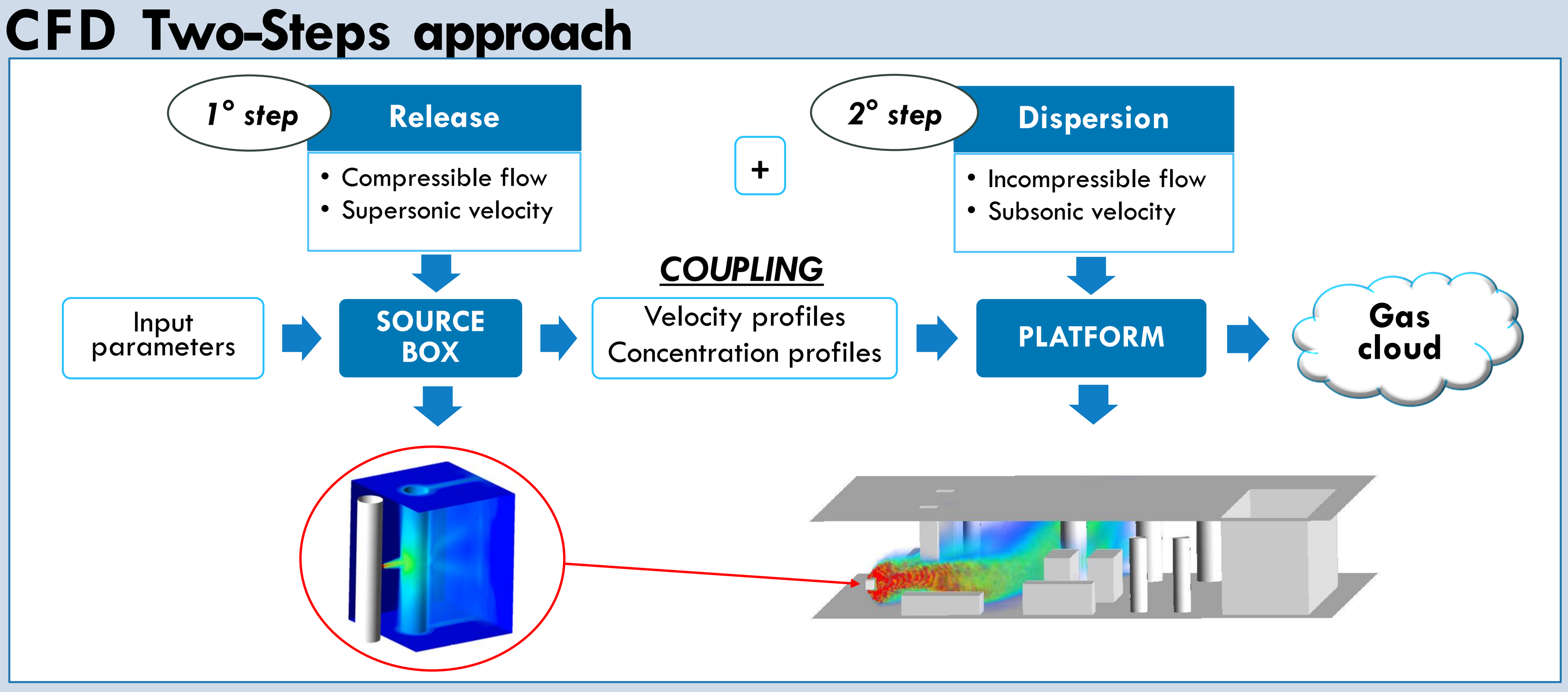
The **objective of this work** is to improve and optimize the use of a ***Two-Steps CFD model*** in order to minimise the number of the needed simulations: this is achieved thanks to a sensitivity analysis on the main parameters characterising a release event in a typical congested industrial environment.

Background



Empirical Methods	CFD
• Simple	• High computational cost
• Low computational cost	• Validation needed
• Highly conservative	• More realistic
• Limited for few cases	• More flexible

CFD Two-Steps approach



In a nutshell: The accidental phenomenon (highly pressurised gas release) is split in two phases → **supersonic release and dispersion.**

Why Two-Steps: Because the two phases involve different spatial and temporal scales → difficult to manage with one-step CFD modelling

Advantages:

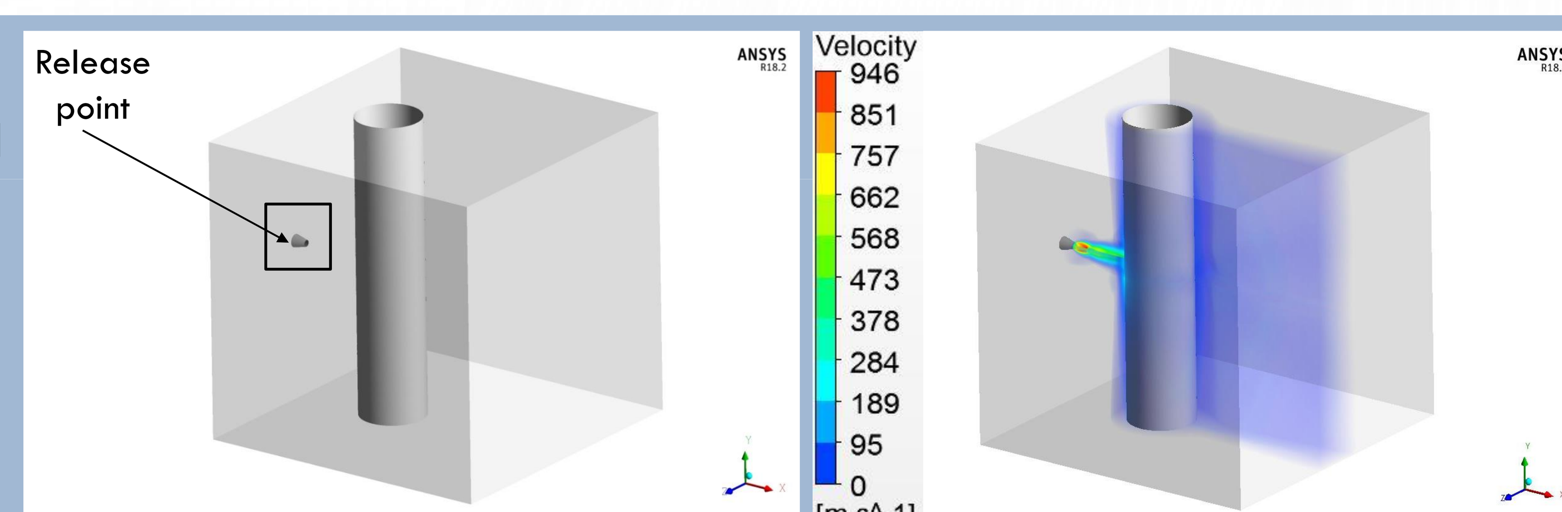
- More flexibility
- Low computational cost
- Good physical modelling of the phenomenon

Results

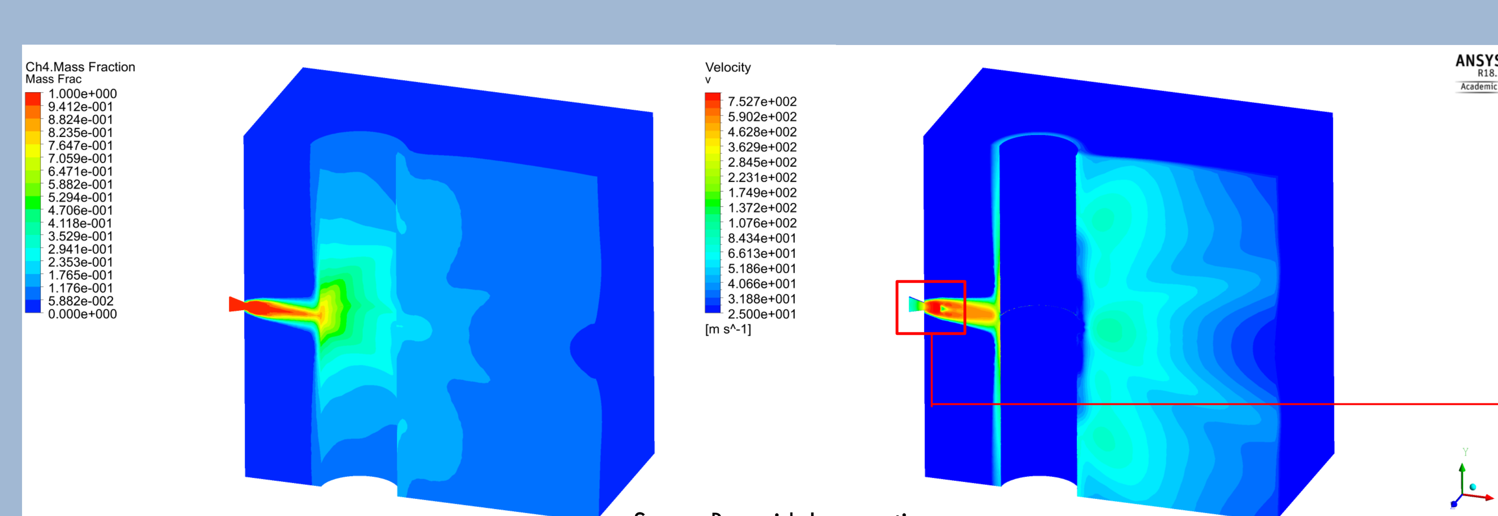
The release phase is studied into a small cubic domain (**Source Box**) containing a cylindrical obstacle near the release point (as it may happen in an industrial congested environment).

The examined **input parameters** are:

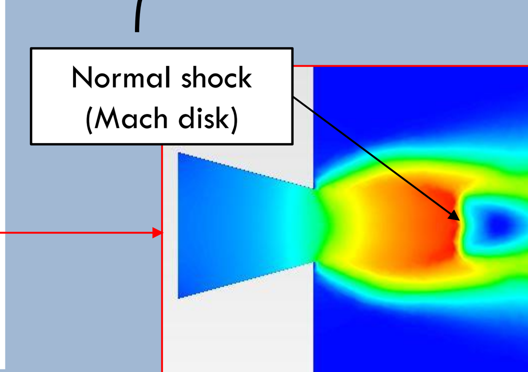
P_{rel}: release pressure;
d_{cyl}: diameter of the obstacle;
l: distance between the rupture and the center of the obstacle.

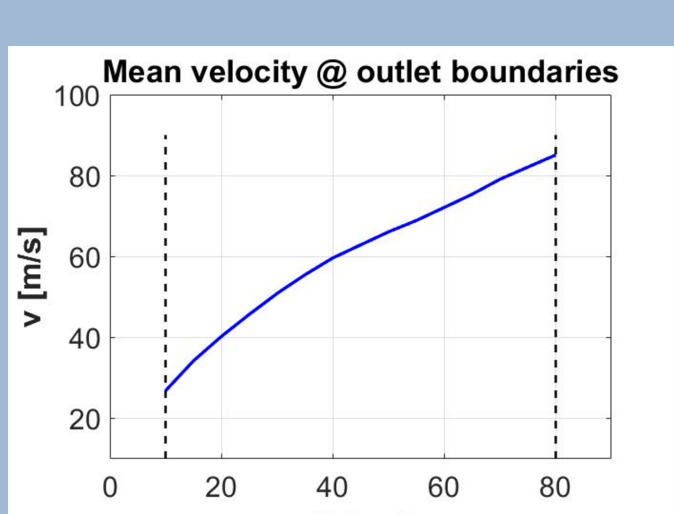


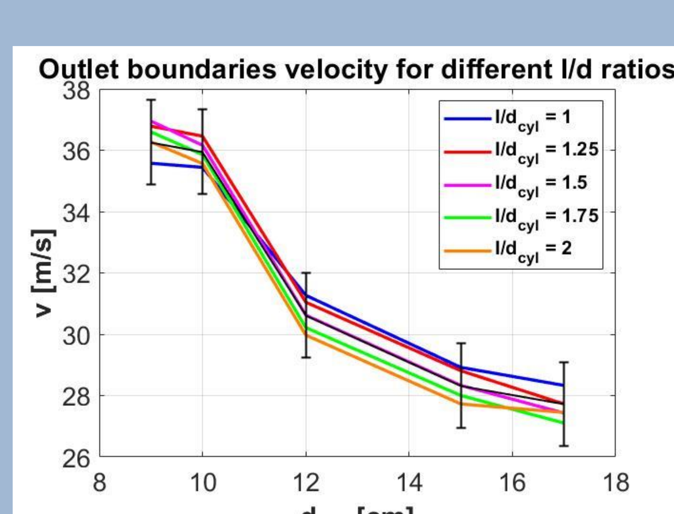
The **velocity flow field** and the CH_4 **mass fraction** @Source Box **outlet faces** are evaluated: these are the input parameters for the dispersion phase simulation.



Typical **barrel-shaped shock** structure of an **highly under-expanded jet**







- The mean velocity at the Source Box outlet boundaries has almost a direct correlation with the release pressure;
- For a certain d_{cyl} value, the variation of the ratio l/d_{cyl} has a negligible effect on the velocity value at the Source Box outlet boundaries.

The **influence of the input parameters** on the Source Box **output values** is defined!

Conclusions

The influence of the different input parameters on the Source Box output values was investigated. Different correlations involving physical quantities (P_{rel}) or geometrical parameters (l, d_{cyl}) were found out, allowing to **reduce drastically the effective number of scenarios to be simulated** and consequently the computational cost.

This work is part of a project financed by the Ministry of Economic Development.
The work was been developed at the Politecnico di Torino, DENERG SEADOG laboratory (Safety & Environmental Analysis Division for Oil & Gas).