ICEM Mesh Creation

How-to Guide – Standalone Scripts

Author: Max Enke

Please note:

These are the standalone versions of the .rpl generation scripts, which do not need a source file to work. All standalone scripts can only be used to create geometry variant 2, with an additional gas space above the film inlet (inlet variant 2) and a simple film outlet (outlet variant 1). They are no longer supported by updates/fixes as of October 2024. In case of an error, please use the scripts of the bundled version first.

Limitations and Capabilities:

The scripts can be used to generate 2D/3D meshes in ICEM automatically. Currently supported geometries:

GEOMETRY	DESCRIPTION
2D HORIZONTAL	generate a 2D mesh with rectangular, horizontal* structures
2D SMOOTH	generate a 2D mesh without structures and a smooth reactor wall
3D HORIZONTAL	generate a 3D mesh with rectangular, horizontal* structures
3D SMOOTH	generate a 3D mesh without structures and a smooth reactor wall

*horizontal = structures orthogonal to main flow direction

All scripts can only create a predefined geometry with certain geometric features. These features can vary in their absolute dimensions, but are always present in every created mesh.

How to use the Scripts:

For clarification of the geometric and meshing parameters, please refer to the provided documents outlining variable names and meshing zones.

1. **Execute the .py file** (ensure, that a suitable python environment is installed)
2. **Define project name**
    - The project name will be the name of all output files and the output folder, in which the output files are saved to.
3. (optional) **Load an existing .conf file for reference**
    - The script will look for the specified file in its root folder and in the folder with the same name as the specified reverence file.
    - Only .conf files created through the current script can be used for reference.
4. **Define geometric parameters** – options:
    - (reference only) Copy geometric parameters from reference file.
    - Manually define geometric parameters.
5. **Define meshing parameters** – options:
    - (reference only) **Reference Meshing** – use mesh parameters as defined in reference file.
        - A refinement factor can be defined, which will coarsen/refine the mesh. Its default value is 1.0, resulting in a mesh topology identical to the one defined in the reference file.
        - values > 1.0 lead to coarsening, values < 1.0 lead to refining
        - The factor will be applied to all absolute cell dimensions defined in the reference file.
    - **Default Meshing** – Will mesh geometry automatically with default settings
    - **Custom Meshing** – Define meshing rule and cell size(s) for every section individually
6. **Output:**
    - A folder with the specified project name containing:
        - .conf file with geometric and meshing parameters
        - .rpl file to be loaded into ICEM
7. **Load the .rpl file into ICEM.**
    1. Load the .rpl file (File > Replay Scripts > Load script file)
    2. Execute all commands (do all)
8. **Check mesh configuration and export**
    - Always check you geometry and mesh if all parameters have been applied as specified.
    - Consider running a mesh check before exporting to Fluent/CFX…
    - When exporting a mesh to Fluent/CFX, select the correct type of geometry (2D/3D).