ICEM Mesh Creation

How-to Guide – Script Bundle

Author: Max Enke

Please note:

This software bundle includes all scripts created for automated ICEM mesh creation. The scripts require the source files “rpl_gen_fnc.py” and “rpl_gen_obj.py” present in the same folder. The bundle is no loinger supported by updates/fixes as of October 2024.

Limitations and Capabilities:

The scripts can be used to automatically generate 2D/3D meshes in ICEM. 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. For every geometry in the table above, different variants can be created, differing in minor geometric features at the film inlet and outlet. An overview over the supported variants gives the following table, also see the documentation of the corresponding geometry.

GEOMETRY	INLET VARIANTS	OUTLET VARIANTS
2D HORIZONTAL	1: simple inlet 2: additional gas space	1: simple outlet (auto select)
2D SMOOTH	1: simple inlet 2: additional gas space	1: simple outlet (auto select)
3D HORIZONTAL	1: simple inlet 2: additional gas space	1: simple outlet (auto select)
3D SMOOTH	1: simple inlet 2: additional gas space	1: simple outlet 2: recessed outlet

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 inlet and outlet variants**
    - Inlet:
        1. Simple inlet
        2. Inlet with additional gas space
    - Outlet:
        1. Simple outlet
        2. Recessed outlet (3D smooth only)
    - (optional) inlet 1 + outlet 1: prepare geometry for periodic boundaries (must be separately defined in ICEM/Fluent)
3. **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.
4. (optional) **Load an existing .conf file for reference**
    - The script will look for the specified file in its root folder.
    - Configuration files created by a different script can be used as well, but may require additional user inputs.
5. **Define geometric parameters** – options:
    - (reference only) Copy geometric parameters from reference file.
    - Manually define geometric parameters.
6. **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. The default cell sizes can be adjusted at the top of the scripts.
    - **Custom Meshing** – Define meshing rule and cell size(s) for every section individually
7. **Output:**
    - A folder with the specified project name containing:
        1. .conf: configuration file containing geometric and meshing parameters.
        2. .rpl: replay file to be loaded into ICEM
8. **Load the .rpl file into ICEM.**
    - Load the .rpl file (File > Replay Scripts > Load script file)
    - Execute all commands (do all)
9. **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).