

HOMEWORK-1

(Submit it at the start of the next lecture, 5 Oct 2026)

1. Objective

Investigate the numerical and computational differences between structured and unstructured grids. You will implement the numerical method yourself using Python, C/C++, or MATLAB. Commercial CFD packages may be used only for supplementary comparison, not as a substitute for your own solver.

Your investigation must focus on mesh topology, mesh generation, local refinement, cell quality, geometric representation, connectivity, memory requirements, mesh-generation computational cost.

2. Geometry

Consider the two-dimensional computational domain shown schematically below:

$$-10D \leq x \leq 20D, -10D \leq y \leq 10D,$$

containing a circular internal boundary of diameter $D = 1$ centered at $(x_c, y_c) = (0, 0)$.

Thus,

$$(x - x_c)^2 + (y - y_c)^2 = \left(\frac{D}{2}\right)^2$$

defines the circular boundary. The region between the cylinder and the outer rectangular boundary is the computational domain to be meshed.

3. Structured Mesh Generation:

Generate a **body-fitted structured mesh** around the circular boundary. An **O-grid** or **C-grid** topology is recommended. Generate at least three meshes: Coarse, Medium, Fine. The meshes should contain approximately 5,000–10,000, 15,000–30,000, and 40,000–60,000 cells, respectively. The exact number of cells does not have to match these values. Your structured mesh must conform to the circular boundary, contain no negative-area cells, provide increased resolution near the cylinder, gradually increase cell size away from the cylinder, maintain reasonable cell aspect ratios. Implement the mesh generation yourself.

4. Unstructured Mesh Generation:

Generate an **unstructured triangular mesh** for the same computational domain. Again generate at least three meshes: Coarse, Medium, Fine, with approximately the same number of cells as the structured meshes. Your unstructured mesh must conform to the circular boundary, contain no inverted triangles, provide local refinement near the cylinder, have a smooth transition from fine to coarse regions, avoid excessively distorted triangles. You may implement your own triangulation algorithm or use an established triangulation library, but you must document the method used.

Answer the following questions using your generated meshes and results.

Q1. Mesh topology

Explain the main difference between structured and unstructured mesh topology. How are neighboring cells identified in each case?

Q2. Mesh quality

For both mesh types, calculate and compare minimum cell area, maximum cell area, minimum cell angle, maximum cell angle, maximum aspect ratio. Which mesh provides better cell quality, and why?

Q3. Grid refinement

Compare the coarse, medium, and fine meshes. How does increasing the number of cells affect cell size near the cylinder, cell-size distribution, aspect ratio, representation of the circular boundary?

Q4. Local refinement

Why is local refinement near the cylinder useful? Compare the structured and unstructured meshes in terms of their ability to provide local refinement without unnecessarily increasing the number of cells in the far field.

HOMEWORK-1

(Submit it at the start of the next lecture, 5 Oct 2026)

Q5. Geometry representation

For the circular boundary, calculate the maximum radial error:

$$e_{\max} = \max_i \left| \sqrt{(x_i - x_c)^2 + (y_i - y_c)^2} - \frac{D}{2} \right|.$$

Compare this error for the different mesh resolutions.

Q6. Computational cost

Measure the approximate time required to generate each mesh. Plot T_{mesh} versus N_{cells} . Discuss the difference between structured and unstructured mesh-generation cost.

Q7. Overall comparison

Based on your results, discuss the advantages and disadvantages of structured and unstructured meshes for simple geometries, complicated geometries, local refinement, mesh quality, memory/connectivity, mesh-generation effort. Do not simply state that one mesh type is “better.” Explain **which characteristics make each approach advantageous or disadvantageous**.

Report Requirements

Submit only a **printed report** containing:

1. Brief description of your mesh-generation method.
2. The six generated meshes as graphics.
3. Mesh-quality results in tables.
4. Answers to Questions 1–7.
5. A final comparison of structured and unstructured meshes.
6. The main source-code listing or selected important portions of the code.

The report should be concise and should not exceed **10 pages**, excluding the source code. No CFD flow solution is required.

IMPORTANT NOTE: Students are expected to adhere to the principles of academic integrity throughout the course. All reports must represent the student's own work, and all sources must be properly acknowledged. Artificial intelligence (AI) tools (e.g., ChatGPT, Copilot, Gemini) are **not permitted other than plots/graphics/tables generation**. However, any use of AI tools should be appropriately disclosed in **acknowledgements section**, and students remain fully responsible for the accuracy, originality, and integrity of all submitted work. AI-generated content must not replace the student's own understanding, analysis, or original work. Violations of academic integrity will be handled in accordance with the University's academic regulations.