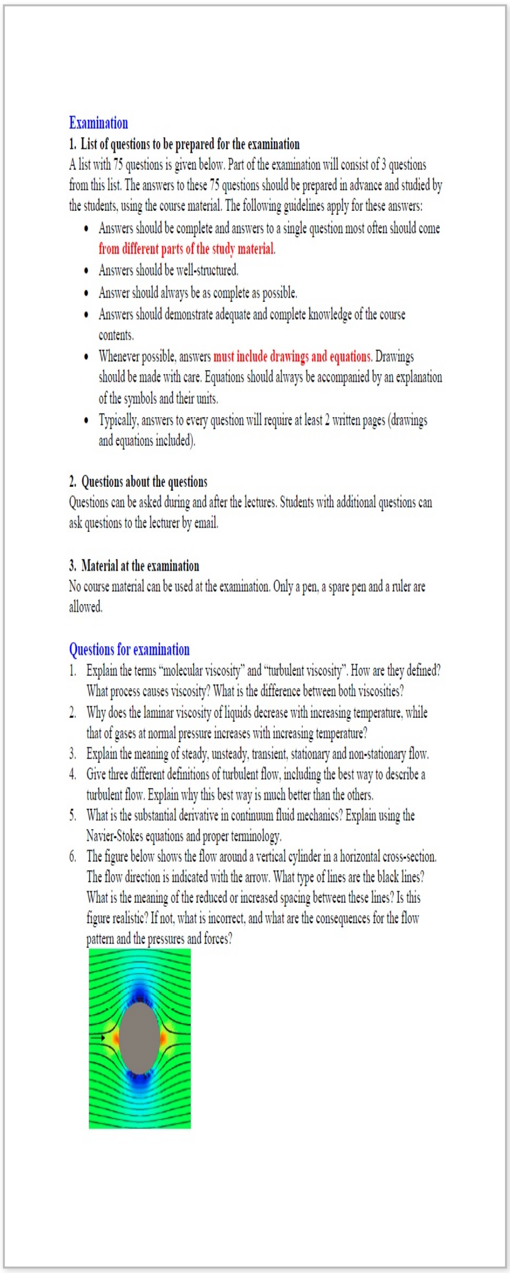
如何入门计算流体力学？

匿名用户

题主你好，正处于流体力学的期末挣扎之中。不知道你以后想学的是什么领域，我目前在学 CFD 城市风工程 / 环境。最近在上的一门课刚好是入门课，老师给了 75 个问题，如果能够回答出来，就算是入门了吧。



Examination

1. List of questions to be prepared for the examination

A list with 75 questions is given below. Part of the examination will consist of 3 questions from this list. The answers to these 75 questions should be prepared in advance and studied by the students, using the course material. The following guidelines apply for these answers:

- Answers should be complete and answers to a single question most often should come **from different parts of the study material**.
- Answers should be well-structured.
- Answer should always be as complete as possible.
- Answers should demonstrate adequate and complete knowledge of the course contents.
- Whenever possible, answers **must include drawings and equations**. Drawings should be made with care. Equations should always be accompanied by an explanation of the symbols and their units.
- Typically, answers to every question will require at least 2 written pages (drawings and equations included).

2. Questions about the questions

Questions can be asked during and after the lectures. Students with additional questions can ask questions to the lecturer by email.

3. Material at the examination

No course material can be used at the examination. Only a pen, a spare pen and a ruler are allowed.

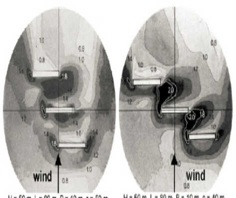
Questions for examination

1. Explain the terms "molecular viscosity" and "turbulent viscosity". How are they defined? What process causes viscosity? What is the difference between both viscosities?
2. Why does the laminar viscosity of liquids decrease with increasing temperature, while that of gases at normal pressure increases with increasing temperature?
3. Explain the meaning of steady, unsteady, transient, stationary and non-stationary flow.
4. Give three different definitions of turbulent flow, including the best way to describe a turbulent flow. Explain why this best way is much better than the others.
5. What is the substantial derivative in continuum fluid mechanics? Explain using the Navier-Stokes equations and proper terminology.
6. The figure below shows the flow around a vertical cylinder in a horizontal cross-section. The flow direction is indicated with the arrow. What type of lines are the black lines? What is the meaning of the reduced or increased spacing between these lines? Is this figure realistic? If not, what is incorrect, and what are the consequences for the flow pattern and the pressures and forces?

7. What is a boundary layer? What is flow separation? What is the reason for flow separation? Using a drawing, explain at which positions of the surfaces of a building flow separation occurs.
8. Explain in detail the potential effects of surface roughness on the aerodynamic drag of a bluff body.
9. What is the main contribution of the German engineer Ludwig Prandtl to the field of fluid mechanics? Why is his work important for building aerodynamics and urban physics?
10. Explain the different characteristics of the wind flow in the atmospheric boundary layer compared to the free atmosphere.
11. Give the equations and a graph for the log law and the power law. When are these laws valid, and what are the limitations?
12. What are the advantages of wind-tunnel testing? Compare in detail with field (on-site) measurements and numerical simulation with Computational Fluid Dynamics. Give an example of a study that can only be performed with wind-tunnel testing.
13. Draw the two types of open-circuit wind tunnel with all necessary components to perform high-quality atmospheric boundary layer wind-tunnel testing. Explain in detail.
14. Draw a closed-circuit wind tunnel with all necessary components to perform high-quality atmospheric boundary layer wind-tunnel testing. Explain in detail.
15. Draw two closed-circuit wind tunnels: a typical aeronautical wind tunnel and a typical atmospheric boundary layer wind tunnel. Indicate the names and functions of the different components and explain in detail.
16. Give the advantages and disadvantages of open-circuit and closed-circuit wind tunnels.
17. Explain the advantages and disadvantages of hot-wire anemometry, Irwin probes, laser-Doppler anemometry and particle image velocimetry.
18. What is similarity? Explain the concept of similarity with the example of natural ventilation of the Amsterdam ArenA football stadium.
19. What is computational fluid dynamics (CFD)? What are the main components of a CFD simulation?
20. Describe the procedure of a CFD simulation in Sports & Building Aerodynamics, from pre-processing to postprocessing.
21. Explain in detail the acronyms DNS, LES, and RANS. What are the differences between these approaches? What are their advantages and disadvantages?
22. Explain the train analogy for Reynolds stresses.
23. What is hybrid URANS/LES? Explain advantages and disadvantages and compare with steady RANS and LES.
24. Describe closure in turbulence modeling, using detailed equations.
25. What are the main advantages and disadvantages of the standard k-ε turbulence model in modeling flow around a building?
26. Explain the main characteristics of the finite difference method, the finite volume method and the finite element method. Which of these methods can be used for studying the drag of a cyclist?
27. What is grid convergence? What is Richardson extrapolation? Why are these concepts important in CFD?
28. Explain grid convergence and the grid convergence index.
29. How can boundary layers be modeled in CFD? Explain using drawings and equations.
30. Draw and explain the universal law of the wall. What is the significance of this law in CFD simulations?
31. Can convective heat transfer from a rough wall be accurately modeled with CFD? Why or why not?
32. Why is modeling surface convective heat transfer in CFD so difficult?

2

33. Describe the potential errors in a CFD simulation. Explain the concepts of verification and validation, and their link to the potential errors.
34. What is a blockage ratio? Describe the blockage ratio guidelines for wind flow around buildings.
35. Given an isolated building with dimensions length x width x height = 30 m x 15 m x 45 m. If wind flow around this building needs to be simulated accurately from all wind directions, what should be the smallest acceptable size of the computational domain?
36. Explain the guidelines for high-quality grid generation. Provide drawings and examples from the literature / course material.
37. For wind flow around a cubic building, draw the computational domain and a high-quality computational grid on the surfaces of the building and the ground surface. Explain why this grid is a high-quality grid.
38. What is the difference between aerodynamic roughness length and roughness height? When do you need to use the one, and when the other? Explain with drawings and practical examples.
39. Make a drawing of the three-dimensional wind-flow pattern around a high-rise rectangular building and indicate and explain the main flow features. Which flow features are important for pedestrian wind conditions?
40. Explain the following concepts: corner stream, standing vortex, stagnation point. Provide drawings of these components in the wind-flow pattern around a high-rise building.
41. Explain the concept of flow separation. Why does flow separation always occur at sharp corners of an object? What is the physical reason?
42. What is the definition of the Venturi effect in urban aerodynamics? Does the Venturi effect exist in passages between perpendicular buildings in a converging arrangement? Explain why or why not.
43. Which three typical building configurations will almost always cause wind comfort problems? Provide a drawing and an explanation of how these configurations cause high wind speed at pedestrian level. What can be done to limit wind discomfort problems in each of these cases?
44. Which are the five main tips to avoid wind discomfort and wind danger? Give advantages and disadvantages of these tips or remedies (including drawings).
45. Describe in detail the figure below. How is this type of figure obtained? What are the numbers in this figure? Explain the difference between both figures, and the reasons for this difference.

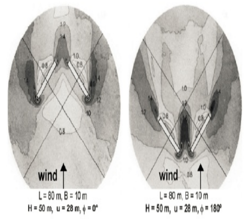


H = 50 m, L = 80 m, B = 10 m, s = 50 m
$\phi = 0°$ - buildings shifted 40 m

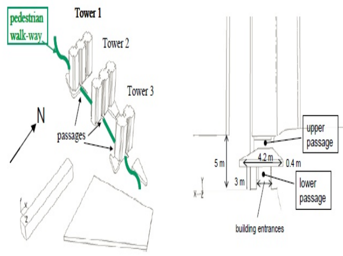
H = 50 m, L = 80 m, B = 10 m, s = 40 m
$\phi = 0°$ - buildings shifted 80 m

46. Describe in detail the figure below. How is this type of figure obtained? What are the numbers in this figure? Explain the difference between both figures, and the reasons for this difference.

3

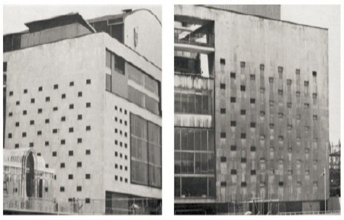


47. Is the work of Giovanni Batista Venturi (1799) relevant for building aerodynamics and urban physics? Explain in detail.
48. Give a definition for pedestrian wind discomfort and for pedestrian wind danger. Explain why high-rise building can cause wind discomfort and wind danger. What can be done to solve such problems, or at least to limit them?
49. Explain the three parts of information that are needed to perform a wind comfort study.
50. What is subconfiguration validation? What are its advantages and disadvantages? Why is it sometimes used?
51. Give and explain the comfort and danger criteria in the Dutch wind nuisance standard. What were the main goals for establishing this standard?
52. The figure below shows the Silvertop Towers. In these towers, through-passages were designed. Why will this lead to wind discomfort? Describe possible solutions and advantages and disadvantages of each of these solutions.

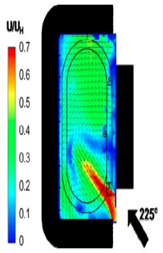


53. Which methodologies can be applied to study natural ventilation of buildings? Describe in detail their advantages and disadvantages.
54. Describe the problem with natural ventilation during concerts in the Amsterdam ArenA stadium. What causes these problems? What can be done to solve these problems?
55. What is wind-driven rain? Why is it important in building physics?
56. The figure below shows the Royal Festival Hall in Londen, after completion (left) and after a year of weather exposure (right). Explain in detail what hapenned here to cause this difference in appearance. Could this have been avoided, and if so, why, or why not?

4



57. Which assessment methods are available for wind-driven rain on buildings? What are their advantages and disadvantages?
58. Briefly explain the five steps that are used for CFD simulation of wind-driven rain on buildings.
59. What is the wind-blocking effect in wind-driven rain? How can it be explained?
60. What is the most beneficial type of a venturi-shaped roof for wind energy harvesting in the built environment? Explain why this type is better than other types. How much energy could be yearly obtained from a single wind turbine in such a roof (e.g. for Eindhoven location)? How much could be obtained from 10 wind turbines in the same roof? Is the latter 10 times as large as the yield of a single turbine, or not? And if so, why or why not?
61. Is the design of the Bahrain World Trade Center optimal from wind energy point of view? Explain your answer in detail by providing evidence whether or not it is optimal.
62. What information is needed to estimate the wind energy yield of a wind turbine integrated in a building? How should this information be obtained?
63. Describe the four phases of the 100 m dash. How does the drag area of the runner change during these phases?
64. Describe the potential effect of wind and altitude on sprint records using equations (back-of-the-envelope equation). How does the IAAF take effect of wind and altitude into account?
65. What are the main specifications of the wind measurement of the IAAF? Explain and comment on these specifications.
66. Explain in detail how stadium wind flow patterns can affect records in the 100 m sprint. Can these effects be both positive and negative? If so, why? If not, why not?
67. How would you design a stadium to limit effects on sprint records? Consider different stadium geometries, and explain their advantages and disadvantages.
68. What is shown in the figure below? How was this figure obtained? Explain in detail what happens if a sprinter runs from S to F.



69. What is shown in the figure below? How was this figure obtained? Explain in detail what happens if a sprinter runs from S to F.
70. Draw and explain the forces acting on a cyclist riding on level road in still air. Given the equation for the drag force and explain the variables in this equation. What are typical values for each variable?
71. Draw the flow field around a time-trial cyclist. What are the most important components of this flow field? Indicate the areas where wind speed is highest and where static pressure on the body is lowest.
72. Given and explain all the advantages and disadvantages of testing with real cyclists in a wind tunnel, with dummy cyclists in a wind tunnel or using numerical simulation with CFD.
73. What is the typical size of a grid cell near the body of a cyclist in a high-quality CFD simulation in which the laminar sublayer is resolved? How could this size change if the cyclist would be riding at half speed?
74. Why does a cyclist riding behind another cyclist experience a reduction in drag? Why does a cyclist riding in front of another cyclist also experience a reduction in drag? Explain the physical mechanism using drawings.
75. Consider identical cyclists in a team time trial in the Tour de France. If four cyclists are riding behind each other, which one has the highest and which one has the lowest aerodynamic resistance? Same question, but for six cyclists and for eight cyclists. Explain the physical mechanisms causing these results using drawings.

Good luck!

Bert Blocken, 13 December 2015

6

虽然和专业学 CFD 的硕博们有一定差距，但是能够回答出来这 75 个问题，应该可以初步使用 CFD 这一工具服务于其他研究了吧。

计算流体力学，是需要编程的，看似很难，实则并没有想象的那么难。

萧忆南

1、首先要深刻理解各种方法编程的核心思

想。掌握有限差分、有限元、有限体积、边界元、格子玻尔兹曼、蒙特卡罗、粒子法等等的核心张量（数组、矩阵，叫什么不重要）数学

表达式，难点在上、下标的几何意义与物理意义的理解。

其中，有限元与边界元方法要懂加权余量法基本知识。明确什么样的偏微分方程方程可以使用加权余量法。这样核心问题就解决了。

2、对于域内网格的算法，要懂结构化及非结构化网格的构造方法。这里要对各种网格生成准则有所了解。

3、程序编辑能力，框图一定要有，明确总体思维和局部网格思维的数据传递形式。这个心中要有数。

完成以上三点，恭喜你，你可以编出一个小程序了。此后，该程序也许尚无法完成很多计算。在程序编辑没有错的情况下

1、考察微分方程的解理论上是否收敛。只需要看泛函及 pde 相关章节就好，无需全学。

2、考察数值耗散，看数值分析等，同样无需全学。

3、考察程序稳定性问题，看李雅普诺夫理论或者 von numann 理论。

4、看工况的湍流模型与离散方法是否使用得当（靠经验）

就这样你的计算流体力学就算可以了。这个过程看似复杂，其实不难 慢慢来。。会好的，有什么问题可以留言讨论。。

CFD 理论比较艰深，按部就班先学习理论未必是一个好办法。

从实践入手，牵引理论学习

- 看看最简单的周期性一维线性对流方程，推导它的离散形式（考虑光滑和非光滑），编写一个小程序。你就能够了解掌握空间离散（光滑场离散与间断场离散）与时间离散了，测试各种格式与初始条件，你可以认识到所谓色散、耗散、精度阶数等数学概念。
- 再推广到一维 Burgers 方程，你就知道非线性问题的空间离散了。
- 再推广到一维 Euler 方程，你就知道非线性耦合问题的求解了，取决于你采用 FDM 还是 FVM，其细节有所不同。
- 再推广到伪一维 Euler 方程求解管道流动，你需要考虑入口出口边界条件了，了解到特征速度与边界处理的理论。
- 再推广到二维 Euler 方程，你就掌握多维问题的求解了，涉及到多维的空间离散、边界条件和求解方法。
- 再推广到隐式求解，可了解牛顿迭代法和各种线性方程组松弛技术在 CFD 中的应用
- 再推广到二维 NS 方程求解层流问题，了解扩散项的离散格式
- 再加入一个代数湍流模型，求解湍流

边界层，你就知道湍流建模的基本点。

这时候你基本上就打好基础了。这个单子还可以列下去，取决于你的具体研究领域。

全文完