

|  |  |
| --- | --- |
| Submission Deadline | Marks and Feedback |
| Before 10am on:  22/09/2023 | **20 working days after deadline (L4, 5 and 7) 15 working days after deadline (L6) 10 working days after deadline (block delivery)**  19/10/2023 |





|  |  |
| --- | --- |
| Unit title & code | Applied Heat Transfer, Thermofluids and Computational Fluid Dynamics |
| Assignment number and title | 1 |
| Assignment type | Coursework - Individual report |
| Weighting of assignment | 70 % |
| Size or length of assessment | 3000 word report |
| Unit learning outcomes | 1. Demonstrate the following knowledge and understanding:  Identify, evaluate and optimise applied heat transfer, thermodynamics and thermal systems, fluid dynamics and relevant analysis methodology including FEA (Finite Element Analysis) and CFD (Computational Fluid Dynamics). 2. Demonstrate the following skills and abilities:  Propose, apply and justify the principles and modelling tools of heat transfer, thermodynamics and fluid dynamics to analyse and design various heat exchangers, thermofluid systems and optimise their performance with FEA and CFD. |





|  |
| --- |
| What am I required to do in this assignment? |
| This is an individual report based on project work. Students will need to liaise with the unit coordinator as they progress on their topic. Each individual is expected to produce a word-processed document detailing the work they have done. The document (including diagrams, calculations and discussion) should not exceed the equivalent of 3000 words. The document must include your student IDs. If there is any issue related to the work or the topic, the student should discuss this with the unit co-ordinator so that necessary arrangement should be taken, preferably within the first two weeks of the assignment. The report should include: (a) Mathematical representation of the solutions to the questions using appropriate notations; (b) Results including figures; (c) Discussion of results and recommendations; (d) A recommendation for solving or improving the work done.  **Delivery method**: Submit your report to BREO (Assessment & Feedback / Assignment 1).  **Indicative project plan**:  Week 1 and 3 – Understanding the fundamentals of advanced thermodynamics and thermofluids, FEA, Heat transfer applications, principles systems, Computational Fluid Dynamics (CFD) -principles and methodologies and other relevant application technologies.  Week 3 and 4 - Get to know the assignment, read the questions and plan the progress work. Learn CFD modelling software.  Week 5 to 6 – Complete the modelling exercise and perform calculations necessary for the simulation solution stipulated in this assignment.  Week 7 – Preparation of Report  **Deadline for submitting project documents**: 22nd of September 2023 before 23.30 pm (Submission on BREO).  **The choice of the technique used to study the project is left to the student to decide. However, CFD simulation has to be performed to assess the airflow field in a solar updraft power-plant to complete your project.**  **Solar Updraft Power Plant:**  Solar updraft power plants (SUPP) are known as low temperature solar power plants, which utilise the solar radiation to heat atmosphere air, as working fluid. In solar chimney power plants (SCPP) are a simple configuration of the SUPP. SCPPs covert solar energy in four stages. Solar radiation is converted to thermal energy in the absorbing medium i.e. air, the thermal energy is converted to kinetic energy in the collector passage, the kinetic energy is converted to mechanical energy in the wind rotor, and finally, the mechanical energy is converted to electrical power through the generator [4].  Constructal solar chimney configuration - ScienceDirect  Figure 1:Schematic of a typical solar updraft power plant  **The Problem:**  A Solar Updraft Power Plant (SUPP) consists of three main components: circular solar collector, chimney, and the turbine. The circular solar collector is mounted above a flat ground with a gap between its cover and the soil. The chimney is constructed in the centre of the solar collector that acts similar to a any other chimney. The last component is the turbine which is installed at the lower part of the chimney. The SUPP physical operating principle is very simple. Solar radiation heats air underneath the collector by means of greenhouse effect. When air gets hot, it moves towards the highest point in the system due to convection (buoyancy), which is the chimney. This generates an updraft air momentum in the chimney that propels the turbine generating electricity, located at the chimney base. Fresh air enters the system through the periphery of the collector which is heated inside the collector, and the process continues [1]. Figure 2 shows a schematic of a SCCP cross-section. You may use the given dimensions in modelling your CFD geometry. Make reasonable assumptions for any missing parameters.    Figure 2: Schematic of the solar collector 1  You are required to perform two CFD tasks in this assignment:   1. As the first task, use the available data to model an axi-symmetric CFD simulation of the solar collector configuration as modelled in the paper by Risto V. Filkoski, Filip Stojkovski and Valentino Stojkovski ‘A CFD study of a solar chimney power plant operation’ presented at the 6th International Conference on Sustainable Energy and Environmental Protection SEEP 2013, Maribor, Slovenia. Use the researchers’ data to formulate your CFD problem. 2. As the second task, develop a 3-D CFD model of the same problem using the same conditions as in 1) above. Discuss any advantages of using a 3-D model against using a 2-D model as in 1) above and compare the findings against the 2-D simulation findings. 3. Compare your findings in 1) and 2) above against the findings of Risto V. Filkoski et. al. [1] and discuss any discrepancies. 4. You are expected to perform the CFD exercises outlined in this assignment in order to collect simulation data towards a research publication. Hence, read enough related research material and information outside this assignment brief.   **References**  You are required to submit evidence of your work in electronic format to the Module leader. This can be in the form of either a CD, or one drive folder or a flash memory. (Given that you are following the course online, Onedrive is the default method)   1. Risto V. Filkoski, Filip Stojkovski and Valentino Stojkovski ‘A CFD study of a solar chimney power plant operation’ presented at the 6th International Conference on Sustainable Energy and Environmental Protection SEEP 2013, Maribor, Slovenia. 2. Shadi Kalash, Wajih Naimeh and Salman Ajib, Experimental investigation of the solar collector temperature field of a sloped solar updraft power plant prototype, *Solar Energy,* Volume 98, Part A, December 2013, Pages 70-77. 3. Jing-yin Li, Peng-hua Guo, and Yuan Wang, Effects of collector radius and chimney height on power output of a solar chimney power plant with turbines, *Renewable Energy*, Volume 47, 2012, pp. 21-28 4. Hussain H. Al-Kayiem, Ogboo Chikere Aja, Historic and recent progress in solar chimney power plant enhancing technologies, enewable and Sustainable Energy Reviews, Volume 58, 2016, Pages 1269-1292, ISSN 1364-0321   **Generate a Mesh**: One can generate a good mesh for simulations following few prudent steps. These include creating a basic geometric shape and then using mesh generation tools to create a simplified mesh. One then needs to understand the areas that need fine resolution to capture salient fluid flow features. ANSYS has a good mesh generation tool. Follow the tutorials to learn how to use this tool before attempting the assignment.  **Perform CFD Calculations:**  First, perform **a 2-D CFD simulation** of an updraft power plant. You may use the central vertical plane as your calculation domain.  Then, you are required to perform at least **two 3D-CFD simulations** using models and methods of your choice. (For example, Steady RANS simulation vs Unsteady RANS simulation.) You are required to justify choice giving due reference to past studies of similar nature.  You are required to compare your results in a tabulated form as well as using appropriate graphs.  **Analyse Results.** You are required to analyse the flow field around the updraft power-plant highlighting salient features in aerodynamics and important fluid phenomena. You are expected to show a thorough understanding of the flow field and the underlying physics. Compare results from the simulation with suitable experimental measurements obtained from literature (You are encouraged to find research papers. Use your library account get access to the material). You are expected to adjust necessary parameters (domain, mesh, turbulence model) within the CFD model so that good agreement is obtained with the results from your simulations in comparison with experimental data or published CFD predictions by other researchers.  **Your task in this assignment is to produce a report in a research paper format describing the flow field and energy generation potential.** |
| What do I need to do to pass? (Threshold Expectations from UIF) |
| This assignment is an individual assignment which tests your capabilities and skills in heat transfer, thermofluids and Computational Fluid Dynamics. You have been given a rough geometry of a power-plant. Make assumptions to obtain missing information, if any.You are required to create your own mesh and perform CFD calculations. Use ANSYS (Fluent or CFX) for your calculations.  Assessment load: you are expected to spend 40 hours for preparation and 5 hours for the completion of the assignment. The student report should not exceed the equivalent of 3000 words.  In order to pass this assignment (Assessment 1) you will need to:   * Understand the physics and configuration of the solar updraft power plant. * Create a geometry and an appropriate mesh * Perform CFD calculations (at least using two methods/models) * Compare your results with published data (measurements and simulations) * Produce a report. Make sure you give due reference to the governing equations of CFD whilst producing your report. |
| How do I produce high quality work that merits a good grade? |
| Demonstrate understanding of the concepts and their relevance to MSc level. Your mark will be influenced by your progress and detailed as followed:  **TOTAL GRADE**: 100  Report guidelines (number of words, deadline and presentation) - **5 marks**  content:  Introduction of the project **5 marks**  Theory and technical background and literature review **10 marks**  Commentary about the project **15 marks**  Methodology **15 marks**  Simulations and calculations **20 marks**  Results **10 marks**  Discussion (Analysis of the results and improvements) **15 marks**  Conclusion 5 marks |
| How does this assignment relate to what we are doing in scheduled sessions? |
| The tasks in this assignment are related to what you are learning in Week 1 to Week 6 for ‘Applied Heat Transfer, Thermofluids and Computational Fluid Dynamics’. |



|  |
| --- |
| How will my assignment be marked? |
| Your assignment will be marked according to the threshold expectations and the criteria on the following page.  You can use them to evaluate your own work and consider your grade before you submit. |

|  |  |  |  |  |
| --- | --- | --- | --- | --- |
|  | **3rd Class – 40-49%** | **Lower 2nd – 50-59%** | **Upper 2nd – 60-69%** | **1st Class – 70%+** |
| **50%** | **Solution**  Solution sufficient to make some progress with addressing the problem, but not all features of the problem are included. | **Solution**  Solution is expressed in a good mathematical form including free body diagrams. | **Solution**  Model considers all the relevant details of the case study, and is expressed in a good mathematical form including free body diagrams. | **Solution**  Model considers all the relevant details of the case study, and is expressed in a good mathematical form including free body diagrams. In addition results from the software are correct and well-presented |
| **50%** | **Report**  Report includes the parts listed under “Deliverables”, but some are incomplete or of poor quality. | **Report**  An attractive report that explains what the problem is, how it can be addressed, and how the proposed solution will behave. | **Report**  An attractive report that explains what the problem is, how it can be addressed, and how the proposed solution will behave. | **Report**  An outstanding report, easy to read, well-structured and all (or most) solutions are correct. |