Thermal Professional Training Exam
(100 points)

This exam covers the topics that you learned in the Professional Track 3: Thermal SImulations Course. SimScale certification with be awarded with a score of 70% or higher.

What do I need for the exam?

1. In order to be fully prepared for the exam, it is recommended that you complete all of the course video lectures, exercises, and quizzes prior to starting this exam

2. You will need access to a computer with an internet connection to access your SimScale account.

3. For some exercise(s), you have the option to use a CAD tool to modify the provided geometry before performing a specified simulation.

4. You will require a calculator for some minor calculations with regard to mesh generation.

5. As you may need to switch back and forth between this document and SimScale workbench, it is recommended that you open this document separately in either a new window or download it to your PC.

6. DO NOT record your answers directly on this exam document. Please use the Exam Answer Form provided in the course curriculum to submit your answers.

This exam consists of:

- 15 Multiple choice questions on the basics of thermal analysis and simulation practices on SimScale.

- 2 Exercises about the following topics:

    - Meshing:
        - Meshing studies
        - Generating consistent mesh using the tetrahedral and hex-dominant approaches
    - Analysis:
        - Uncoupled thermomechanical analysis
        - Conjugate heat transfer analysis
    - Materials:
        - Defining simple elastic material

        - Defining fluid and solid materials separately for conjugate heat transfer case
    - Initial condition:
        - Defining a sub-domain based condition
    - Boundary Conditions:
        - Applying basic constraints, loads, temperature and heat flux loads
        - Defining symmetry constraints
        - Specifying inlet, outlets and special wall conditions
    - Numerics:
        - Modifying solution schemes to obtain a stable solution
    - Results:
        - Getting result data at a point
        - Getting specific result data across a cross-section or a volume

Note:

The exercises involves generating mesh and setting up the simulation using SimScale. The 90 minute duration is the time taken to construct the cases, while the mesh generation and simulation run could take time to complete. Hence a suggestion would be to start with the Exercise 2, then Exercise 1 and finally with the multiple choice questions. Nevertheless, you can prioritize the sequence as per your convenience.

MULTIPLE CHOICE [30 points (2 points each)]

Q1	What is the first step in conducting the finite element analysis?		
A	Select interpolation functions	**B**	Assemble the element equations
C	Find the element properties	**D**	Discretize the continuum

Q2	A steel plate has a fixed temperature on side A and a convective heat flux on side B. What type of boundary conditions are these two?		
A	Neumann on side A and Dirichlet on side B	**B**	Dirichlet on side A and Neumann on side B
C	Neumann on both A and B	**D**	Dirichlet on both A and B

Q3	A simple static uncoupled thermomechanical analysis of a multibody structure was performed. All the solids were made of same material and bonded contacts were defined between them. Even being a simple analysis, it took lot of time to solve the case. What can be done to make it faster without compromising on result quality?		
A	The bonded contacts can be removed	**B**	All the solids can be merged into one
C	Geometric nonlinearity can be turned off in case it was turned on initially	**D**	All of the above

Q4	In case of a heat transfer analysis, any kind of nonlinearity can be resolved by:		
A	Performing a steady state analysis	**B**	Performing both steady state and transient analysis
C	Performing a transient analysis	**D**	None of the above

Q5	After a tiring day Jack decided to cook dinner. While placing a steel pan on the stove Jack dozed off! He wakes up middle of the night to find that the pan has deformed due to overheating. The next day Jack decides to analyse and understand how this deformation occurred. Which analysis type should Jack use to solve the case in SimScale?		
A	Heat transfer based on FEA approach	**B**	Conjugate heat transfer
C	Uncoupled thermomechanical	**D**	None of the above

Q6	In the standard CFD methodology, where are the velocity, pressure, temperature fields stored		
A	Centroid of the control volume	**B**	At the discretized nodes
C	Throughout the continuous domain	**D**	Centroid of the faces

Q7	The experimental temperature values at the inlet and outlet fluids and the solid pipes of heat exchanger are obtained at various time intervals. What type of analysis be carried out in order to validate the experimental results?		
A	Steady state Convective Heat transfer	**B**	Transient Convective Heat Transfer
C	Transient Conjugate Heat Transfer	**D**	Steady state Conjugate Heat transfer

Q8	To which case/s can Boussinesq approximation ideally be applied?		
A	All applications of heat transfer involving gases	**B**	Problems where the expected changes in density are small
C	When there are sizable changes in temperature or fluid properties	**D**	All of the above

Q9	What is a prerequisite for setting up a CHT simulation?		
A	Solid and fluid volumes material definition	**B**	Assigning topological entity sets
C	Multi-region mesh	**D**	Both A and C

Q10	Which of the following is correct regarding turbulence?		
A	It increases with increase in flow rate	**B**	It decreases with increase in viscosity
C	It increases with increase in the length scale	**D**	All of the above

Q11	Natural convection is characterized by which of the following?		
A	Schmidt number	**B**	Rayleigh number
C	Peclet number	**D**	Stokes number

Q12	The partitioned approach in OpenFOAM CHT multi-region has		
A	Same governing equations for all regions	**B**	Single coupled matrix equation system
C	Different governing equations for the regions	**D**	Both A and B

Q13	What is NOT true about coupled interface patches in a multi-region mesh?		
A	Coincident boundary patches should be available, one patch associated with each region	**B**	The patches can only be flat surfaces
C	The interface solution is based on Dirichlet-Neumann partitioning	**D**	Both A and C

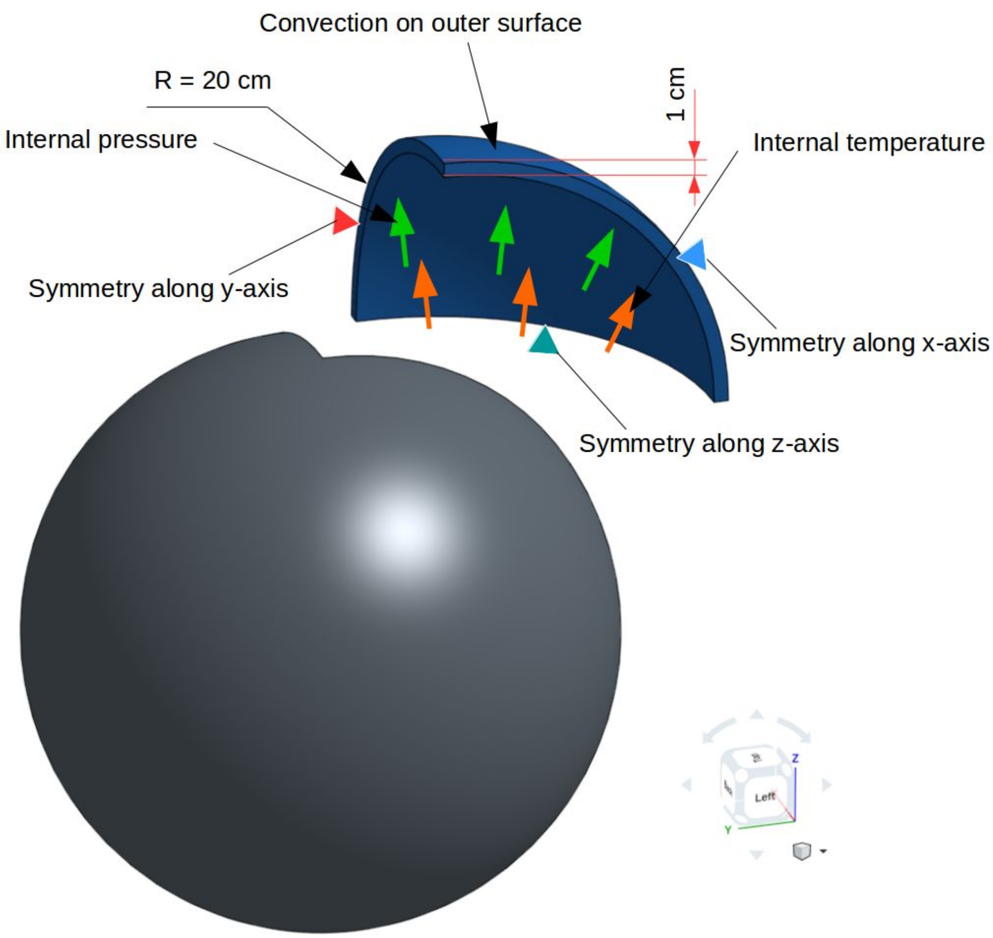
Q14	What is the basic principle of CHT interface coupling in OpenFOAM?		
A	Momentum conservation	**B**	Energy conservation
C	Newton-Raphson Theory	**D**	Mass conservation

Q15	A hex-dominant mesh is generated to perform external flow analysis of an object. The bounding box (base mesh box/external flow domain) has a value 500 cm and number of cells being 100 in that particular direction. What should be the cell size (in a particular direction) for a minimum and maximum refinement of level 2 on a surface of the body?		
A	1.25 m	**B**	0.125 m
C	0.0125 m	**D**	12.5 m

EXERCISES

E1 | Thermostructural analysis of a Pressure Vessel [30 points]

Let's consider a simple spherical hollow pressure vessel:



Due to symmetry, only 1/8th of the spherical vessel is taken into consideration. The applied boundary conditions will be discussed shortly under 'Task 2'. First import the project to your workspace and then perform the following tasks:

Import the project
Thermal Exam - Exercise 1

Task 1 (Meshing): (8 points)

Create a first order manual tet-dominant mesh such that you have at least 6 to 8 elements over the vessel thickness.

Task 2 (Simulation): (14 points)

1. Create a new simulation and set it up as following:

    a. **Analysis Type:**

    Uncoupled thermomechanical analysis - advanced (**Transient - Static - Linear)**

    b. **Domain:** Created mesh in Task 1

    c. **Geometry Primitives:** Create the following points under geometry primitives

    Point 1: (0,0,0.190)
    Point 2: (0,0,0.195)
    Point 3: (0,0,0.199)

    d. **Material:** Stainless Steel

    Material behavior: Elastic
    Young's modulus [N/m²]: 190 GPa
    Poisson's ratio [-]: 0.305
    Density [kg/m³]: 7750
    Kappa [W/(m K)]: 14
    Expansion coefficient [1/K]: 9.7e-6
    Reference temperature [K]: 300
    Specific heat [J/(kg K)]: 486

    e. **Initial Conditions:** Temperature [K] = 300

    f. **Boundary Conditions:**

    Constraints:
    Please see the above image
    Load:
    Internal pressure according to following function: P = 0.2e6*t
    (where t is time)
    Temperature Loads:
    Internal temperature according to following function: T = 40*t+300
    (where t is time)
    Heat Flux Loads:
    Convective heat flux on outer surface at Reference temperature [K] = 300 and h [W/(m² K)] = 90

g. **Simulation Control:**

Simulation interval [s]: 5
Time step length [s]: 0.2
Number of computing cores: 8
Maximum runtime [s]: 14400

h. **Result Control:**

i. Create temperature point data on all the created points in step 'c' above separately (i.e. total three point data)

ii. Create vonMises stress point data on all the created points in step 'c' above separately (i.e. total three point data)

i. Create a new simulation run and start

2. Change the **Kappa** from 14 to 20, 26 and 32 and rerun. Thus making total of 4 simulation runs.

Task 3 (Results + Questions): (8 points)

ANALYTICAL DATA

Point	Temperature [K]	vonMises stress [MPa]
(0,0,0.190)	500.00	247.56
(0,0,0.195)	359.66	57.11
(0,0,0.199)	327.33	137.35

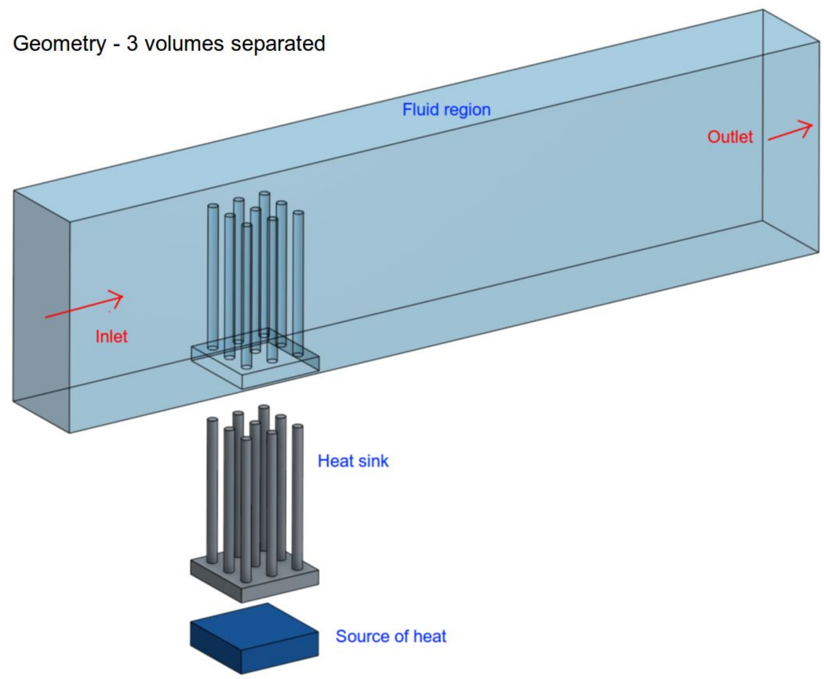
Considering the above analytical data[1], please answer the following question (Please record your answers directly on the Exam Answer Form).

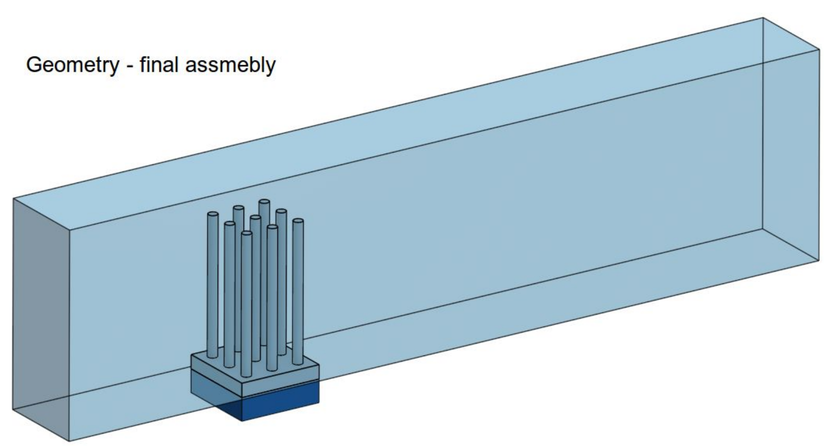
3.1 At final simulation interval, which Kappa gives you the most accurate temperature and vonMises stress values over these points in comparison to the above analytical data?

[1] Afkar, Amir, Majid Nouri Camari, and Amin Paykani. "Design and analysis of a spherical pressure vessel using finite element method." World Journal of Modelling and Simulation 10.2 (2014): 126-135.

E2 | Conjugate heat transfer analysis of a heat sink (40 points)

This exercise includes an analysis of heat sink which tries to transfer the heat from a source to the external fluid. The geometry consists of 3 volumes - 2 solids and 1 fluid, which are not overlapping but the faces touching each other.





The fluid region is to be defined as ***air***, the heat sink as ***aluminium*** and heat source as ***copper***. Import the project to your workspace and then perform the following tasks:

Import the project
Thermal Exam - Exercise 2

Task 1 (Meshing): (14 points)

1. Create a hex-dominant parametric mesh in order to perform a conjugate heat transfer analysis.
2. The number of cells in each direction and bounding box dimensions can be left default, the ones which are generated when parametric mesh option is selected.

Number of cells - bounding box discretization

Direction	Number of cells
x	146
y	17
z	41

Background Mesh Box dimension (in meters)

Direction	Minimum	Maximum
x	-0.05875	0.1338
y	-0.011	0.011
z	-0.007475	0.04698

3. Define refinements as per the following criteria.
   All the dimensions mentioned is with regard to only the x-direction, i.e. the size mentioned is in x-direction for the particular refinement level. The values are specified in *meters*. Calculate the refinement levels using the formula and enter these during the mesh refinement definition. During calculations round off the refinement level to the nearest whole number.
   a. Surface refinement for the fluid region should have maximum and minimum cell sizes 6.64e-4 and 1.65e-4
   b. Surface refinement for the heat sink should have maximum and minimum cell sizes 3.3e-4 and 1.65e-4
   c. Surface refinement for the heat source should have maximum and minimum cell sizes 6.64e-4 and 3.3e-4
   d. The feature refinement is defined with cell size 1.65e-4 for a distance 0.001
   e. Add boundary layers to faces of the fluid region, except the inlet and outlet.

Number of layers	2
Expansion ratio for layer cell thickness	1.3
Thickness of the final layer	0.3
Minimum overall layer thickness	0.01

Goals:

- Make sure all surfaces are well resolved using reasonable refinement levels and settings. Hint: Check the mesh log to avoid illegal cells and also a quick visual inspection if the geometric features are retained.
- Try to keep the mesh size under 1.5 million 3D elements (volumes).

Task 2 (Simulation): (18 points)

1. Use the mesh created from **Task 1** to set up the simulation. In case you were not able to generate a mesh successfully then please import this project to your dashboard and start setting up the simulation, as defined below.

   Analysis Type: K-omega SST, Steady-state

   Model: Gravity 9.81 m/s^2 in negative z direction

   Material: The materials are chosen from the library and assigned to the corresponding mesh volumes

   Initial Conditions: Temperature definition - Sub-domain based

Fluid	: 298 K
Heat sink	: 318 K
Heat source	: 343 K

   Boundary Conditions:

Air inlet	: Velocity inlet - Fixed Value - File upload type (Click to download the .csv file) Upload this file during inlet definition Set the parameters with respect to time only Column index of time & value are **1** & **2** respectively Inlet temperature - Fixed value **293 K**
Air outlet	: Pressure outlet - Fixed value - **101325 Pa**
Heated surface	: Fixed value temperature **343 K** Assigned to bottom surface of the heat source
Adiabatic slip walls	: Slip wall condition (Wall function treatment) Zero gradient temperature Assigned to the sides and top wall of fluid
Adiabatic no-slip walls	: No-slip wall condition (Wall function treatment)

Zero gradient temperature
Assigned to all remaining walls, except interfaces

Numerics:

Under Solver settings change the **Relative tolerance** of *all* solvers to **0.01**
Change all possible **Divergence schemes** to **bounded Gauss upwind**

Simulation Control:

Simulation interval [s] : 0 to 4000 [s]
Time step length [s] : 1
Write interval : 2000
Number of computing cores : 16
Maximum runtime [s] : 12000

Result Control:

The outlet air temperature is to be noted - hence **area average** result controls needs to be created at inlet and outlet.

Simulation run:

Create a new simulation run and start

The csv file contains the velocity in only x direction with a value 1 m/s. Kindly refer this link to understand how such a file is generated. Change the x-direction **Velocity** values as 0.5 and 4 m/s, upload the new files for inlet velocity definition and rerun. This makes a total of **3 simulation runs** (0.5, 1 and 4 m/s velocity values in the x-direction).

Task 3 (Questions): (8 points)

3.1 Which velocity has a higher average fluid temperature at the outlet, and what is the approximate value?

3.2 Please post the project public link, with a screenshot of slice section temperature plot (across the fluid and solid volumes) stored in the post-processor