MSAS – Assignment #2: Modeling

Lorenzo Cucchi, 221732

Exercise 1

The rocket engine in Figure 1 is fired in laboratory conditions. With reference to Figure 1, the nozzle is made up of an inner lining (k_1), an inner layer having specific heat c_2 and high conductivity k_2, an insulating layer having specific heat c_4 and low conductivity k_4, and an outer coating (k_5). The interface between the conductor and the insulator layers has thermal conductivity k_3.

1.1) Part 1: Parameters definition

Select the materials of which the nozzle is made of*, and therefore determine the values of k_i ($i = 1, \dots, 5$), c_2, and c_4. Assign also the values of ℓ_i (i=1,. . . ,5), L, and A in Figure 1.

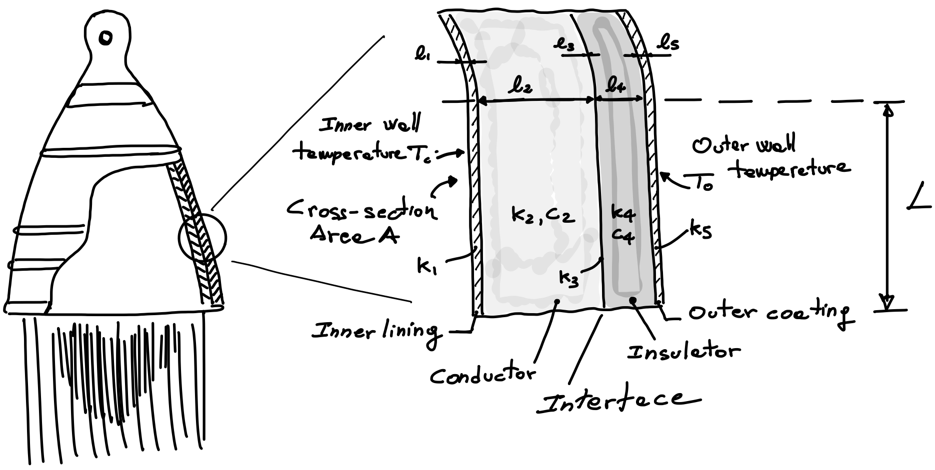

Figure 1: Real thermal system.

1.2) Part 2: Causal modeling

Derive a physical model and the associated mathematical model using one node per each of the five layers and considering that only the conductor and insulator layers have thermal capacitance. The inner wall temperature, T_i, as well as the outer wall temperature, T_o, are assigned. Using the mathematical model, carry out a dynamic simulation in MATLAB to show the temperature profiles across the different sections. At initial time, $T_i(t_0) = T_o(t) = 20$ C°. When the rocket is fired, $T_i(t) = 1000$ C°, $t \in [t_1, t_f]$, following a ramp profile in $[t_0, t_1]$. Integrate the system using $t_1 = 1$ s and $t_f = 60$ s.

1.3) Part 3: Acausal modeling

a) Reproduce in Simscape the physical model derived in Part 2. Run the simulation from $t_0 = 0$ s to $t_f = 60$ s and show the temperature profiles across the different sections. Compare the results with the ones obtained in point 1.2). b) Which solver would you choose? Justify the selection

*The interface layer is not made of a physically existing material, though it produces a thermal resistance. For this layer, the value of the thermal resistance R_3 can be direcly assumed, so avoiding to choose k_3 and ℓ_3.

based on the knowledge acquired from the first part of the course. c) Repeat the simulation in Simscape implementing two nodes for the conductor and insulator layers and show the temperature profiles across the different sections.

(15 points)

The selection of the layers materials has grate effect on the thermal behavior of the system. In particular, the thermal conductivity of the materials is the main parameter that affects the heat transfer between the layers. The thermal conductivity of the materials can be found in literature, the choiche of the material is based on the design requirements assigned. The chosen materials are reported in Table 1.

Layer	**Material**	**k** $[W/(mK)]$	**c** $[J/(KgK)]$	ρ $[kg/m^3]$	**th** [mm]
Inner lining	Carbon-Phenolic comp.	1.5	–	1650	4
Conductor	Graphite G-348	130	750	1830	10
Insulator	Cork	0.07	2100	485	8
Outer coating	Alluminium 6061T6	160	–	2700	2

Table 1: Materials properties

The carbo-phenolic composite is used as inner lining because of its low thermal conductivity and its high thermal resistance, the inner lining is in direct contact with the conductor. Graphite has been chose as the conductor because of its good thermal conductivity and low density compared to metals, moreover its mechanical characteristics don't vary with temperature. The interface between the conductor and the insulator is modelled as a layer with thermal conductivity obtained from literature and derived from the contact resistance $R_\theta = th/(Ak)$ which is approximately $R_\theta = 7 \times 10^{-3} [m^2K/W]$. Not considering an interface cause the temperature to be almost equal at the interface. The insulator is made of cork which has a low thermal conductivity and low density and is in contact with alluminium which act as the outer coating.

The nozzle is modelled as a flat plate, to do this we have to define an exchange area $A = 1$ $[m^2]$ and a length $L = 0.5[m]$ derived from a nozzle with a $D_{exit} = 0.636$ $[m]$.

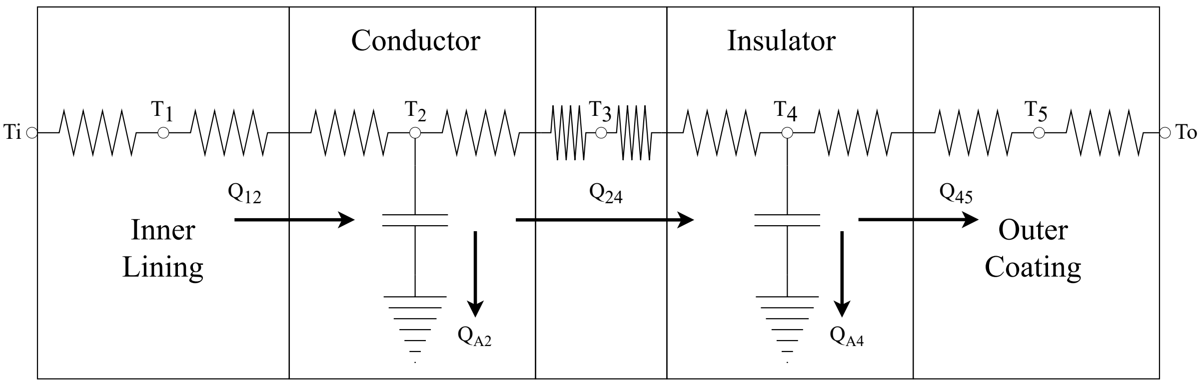

Figure 2: Physical model of the system.

The physical model that approximantes the system uses only one node per layer and it's represented in Figure 2. The thermal capacitance is modelled as a thermal mass $C_i = c_i \rho_i V_i$ the thermal resistance is modelled as $R_i = th_i/(k_i A_i)$, every resistance in Figure 2 is actually divided in half for every layer. Using this definitions it's possible to derive the mathematical model represented from the following equations:

$$\begin{cases} Q_{A2} = Q_{12} - Q_{24} \\ Q_{A4} = Q_{24} - Q_{45} \end{cases} \Rightarrow \begin{cases} C_2 \frac{dT_2}{dt} = \frac{T_i - T_2}{R_1 + R_2/2} - \frac{T_2 - T_4}{R_2/2 + R_3 + R_4/2} \\ C_4 \frac{dT_4}{dt} = \frac{T_2 - T_4}{R_2/2 + R_3 + R_4/2} - \frac{T_4 - T_o}{R_4/2 + R_5} \end{cases} \quad (1)$$

This set of equations can be transformed in a state space model:

$$\dot{\mathbf{x}} = \mathbf{A}\mathbf{x} + \mathbf{B}\mathbf{u} \ , \quad \mathbf{x} = \begin{bmatrix} T_2 \\ T_4 \end{bmatrix} \ , \quad \mathbf{u} = \begin{bmatrix} T_i \\ T_o \end{bmatrix} \tag{2}$$

$$\mathbf{A} = \begin{bmatrix} -\frac{R_1+R_2+R_3+R_4/2}{C_2(R_1+R_2/2)(R_2/2+R_3+R_4/2)} & \frac{1}{C_2(R_2/2+R_3+R_4/2)} \\ \frac{1}{C_4(R_2/2+R_3+R_4/2)} & -\frac{R_2/2+R_3+R_4+R_5}{C_4(R_2/2+R_3+R_4/2)(R_4/2+R_5)} \end{bmatrix} \tag{3}$$

$$\mathbf{B} = \begin{bmatrix} \frac{1}{C_2(R_1+R_2/2)} & 0 \\ 0 & \frac{1}{C_4(R_4/2+R_5)} \end{bmatrix} \tag{4}$$

It's possible to integrate the system using the **ode45** solver in MATLAB, since the eigenvalues of the state matrix $\mathbf{A}$ are $\lambda \approx [-0.029, -0.004]$. After the integration the temperature histories of all nodes are retrieved performing an heat balance of the system and are displayed in Figure 3.

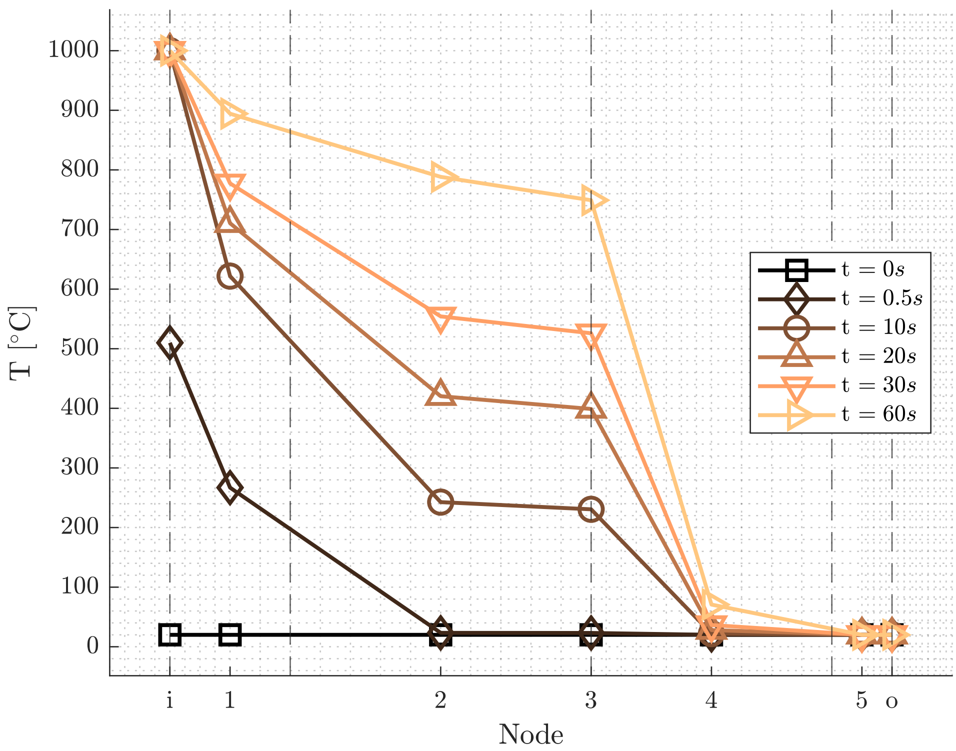

Figure 3: Evolution of temperature in single node casual model.

The acausal model is implemented in Simscape using the thermal library, the temperature source is defined with a ramp block and a saturation one and is connected to a controlled temperature source block. The temperature source also receive in input the ambient temperature which is $20°C$, the temperature source is connected to the inner wall surface of the nozzle. The outer wall surface is connected to the ambient temperature block. To model the inner and outer layers two conductive heat transfer blocks are used for each, the conductive and insulator layers are modelled with a single thermal mass and two conductive heat transfer blocks. The interface layer is modelled with two thermal resistances, this blocks are used since from literature it was possible to obtain only a thermal surface resistance and the layer has no thickness.

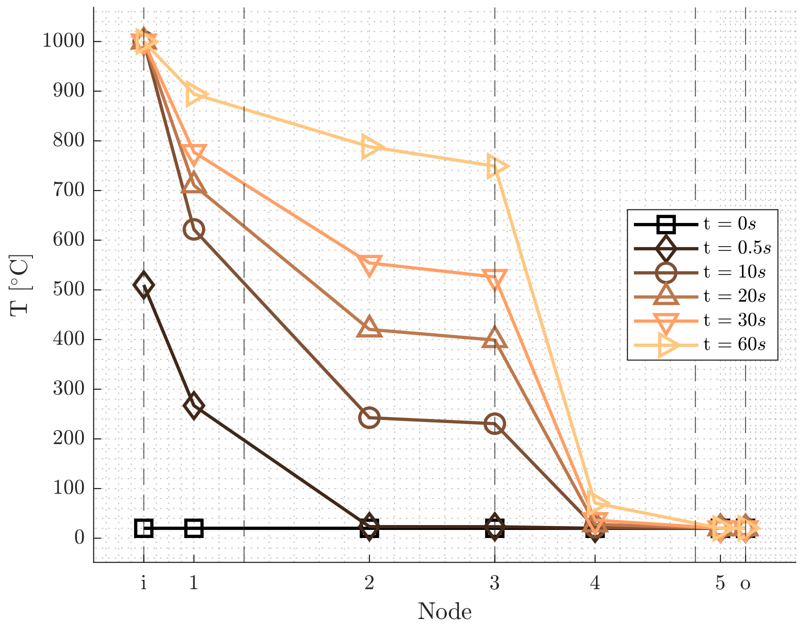

Figure 4: Evolution of temperature in single node acasual model.

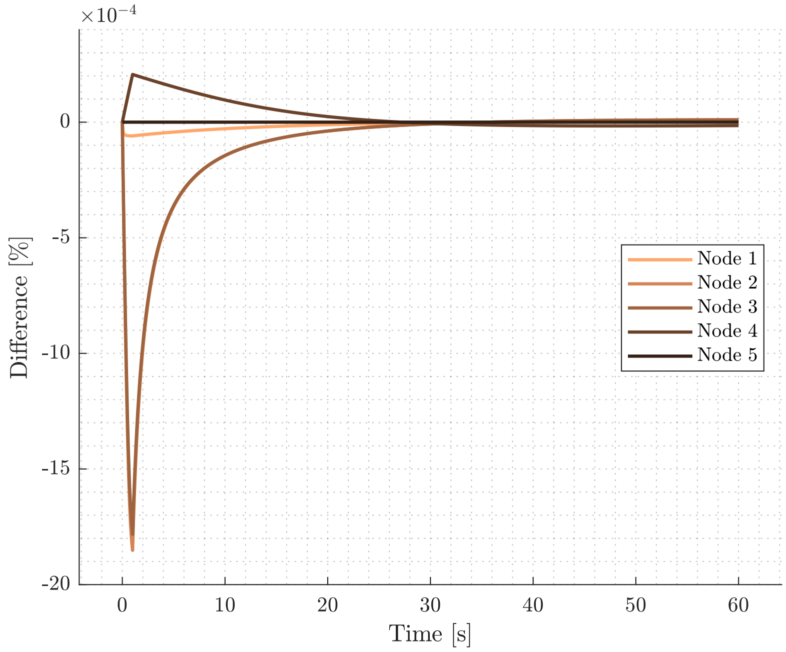

Figure 5: Percentual difference between casual and acasual model.

The simulation is run from $t_0 = 0$ $[s]$ to $t_f = 60$ $[s]$ and the temperature histories are displayed in Figure 4.

The solver used is the Simscape **Trapezoidal Rule** local solver with a fixed time step of 0.1 $[s]$, the solver is chosen since it's A-stable and can handle stiff systems. The simulation results are almost identical to the ones obtained with the **ode45** solver for the casual model, the difference is neglectable and it's shown in Figure 5, node 2 and 3 differences from the casual model are almost identical in the model. Another viable solver is the (ode23t) which is also a trapezoidal rule solver but has variable-time stepping, the result would be more precise near the temperature source step but it wouldn't provide more benefits after the temperature source is stable. The fixed time step is useful in case of real time simulation interfaced with control systems. It's possible to add multiple nodes to the simulation by splitting the thermal mass and the thermal resistance following the position of the additional nodes. In this case the conductor and the insulator will have an additional node each, for both three thermal conductive heat transfer blocks are used each with a third of the layer thickness, the thermal mass is divided in two identical parts. The simulation is run from $t_0 = 0$ $[s]$ to $t_f = 60$ $[s]$ and the temperature history of the nozzle is displayed in Figure 6 and in Figure 7 the temperature history of each node is shown.

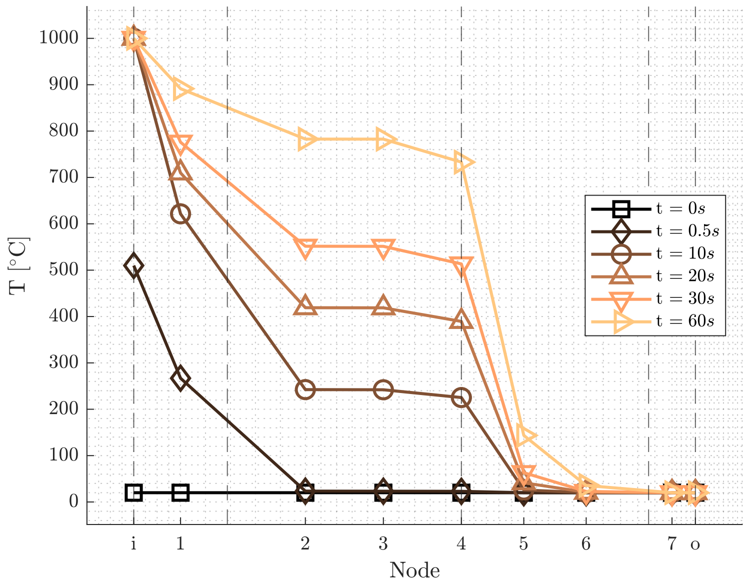

Figure 6: Percentual difference between casual and acasual model.

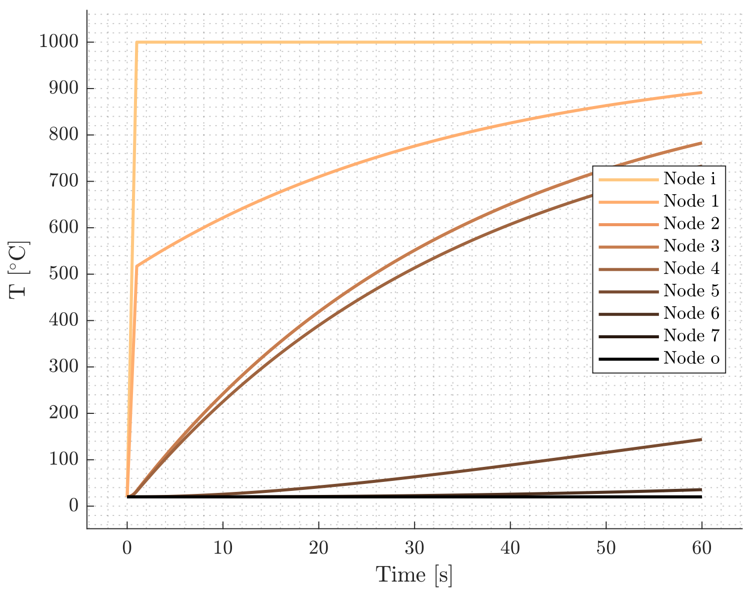

Figure 7: Percentual difference between casual and acasual model.

Exercise 2

The real system of an electric propeller engine is depicted in Figure 8. It is composed by a DC permanent magnet motor which drives a propeller shaft. Between the motor and propeller shaft there is a single stage gear box to regulate the angular speed ratio. Moreover, to avoid overheating of the gear unit, the system is augmented by a cooling system where a fluid exchanges heat with the gear box itself. In Figure 9 a functional breakdown structure of the system is shown.

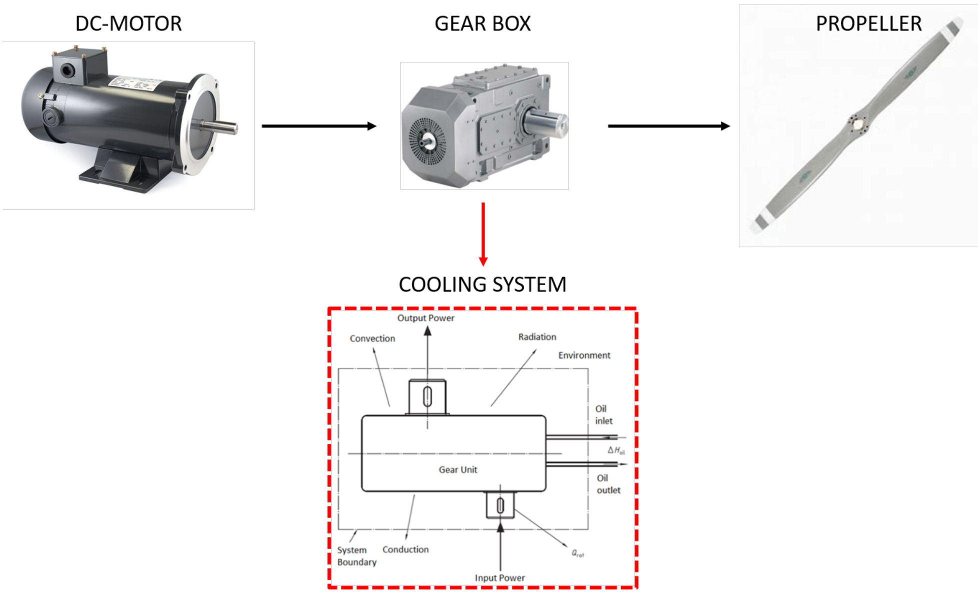

Figure 8: Real system.

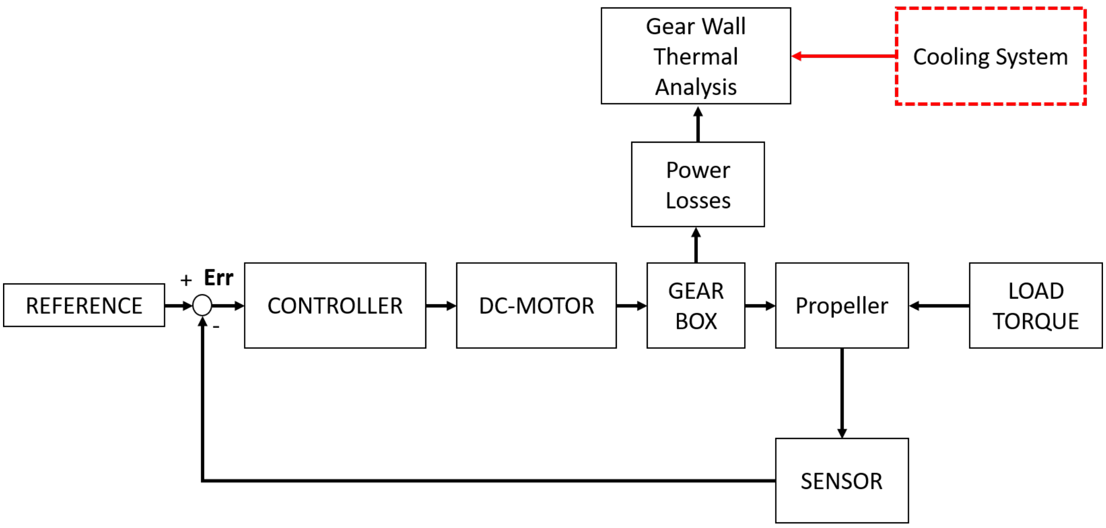

Figure 9: Functional block scheme of the system.

2.1) Part 1: Propeller Electric Engine

Considering the real system in 8 **without** the cooling part, you are asked to:

1. Extract a physical model highlighting assumptions and simplifications.
2. Reproduce the model in acausal manner in Dymola.

3. According to the block scheme in 9, tune a controller (e.g., a PID controller) such that the motor input voltage remains less than 200 V and the error signal **Err** is less than 0.1 rad/s after 10 s.

4. Study the Gear box temperature and heat flux for a simulation time of $\mathbf{t_f} = 120$ s (considering only conduction as heat transfer).

5. Discuss the simulation results and the integration scheme used

For the simulation part, you shall consider: the DC motor data listed in Table 2; the gear box data listed in Table 3, with loss parameters in Table 4; a propeller made of **aluminium** with nominal angular speed $\hat{\omega}$ and a nominal quadratic speed load torque $\hat{T}_{load}$ acting on it (Table 5). The reference angular speed signal to be tracked by the propeller is given in Figure 10.

Table 2: DC motor data

Parameter	**Value**	**Unit**
Coil Resistance	0.1	Ω
Inductance	0.01	H
Motor Inertia	0.001	kg m^2
Motor Constant	0.3	Nm/A

Table 3: Gear Box data

Parameter	**Value**	**Unit**
Mass	3	kg
Gear ratio	2	[-]
Specific heat	1000	J/(kg K)
Thermal Conductivity	100	Wm/K

Table 4: Gear Box Loss Table

Driver angular speed [rad/s]	**Mesh efficiency[-]**	**Bearing friction torque [Nm]**
0	0.99	0
50	0.98	0.5
100	0.97	1
210	0.96	1.5

Table 5: Propeller data

Parameter	**Value**	**Unit**
Diameter	0.8	m
Thickness	0.01	m
$\hat{\omega}$	210	rad/s
$\hat{T}_{load}$	100	Nm

2.2) Part 2: Cooling System

After the previous gear unit thermal analysis, now consider the steady-state condition reached by the propeller engine at the end of the simulation to model and simulate a single **fixed** volume flow rate cooling system (as shown in Figure 8) for the gear unit and considering only **convection** as heat transfer. In particular, you are asked to:

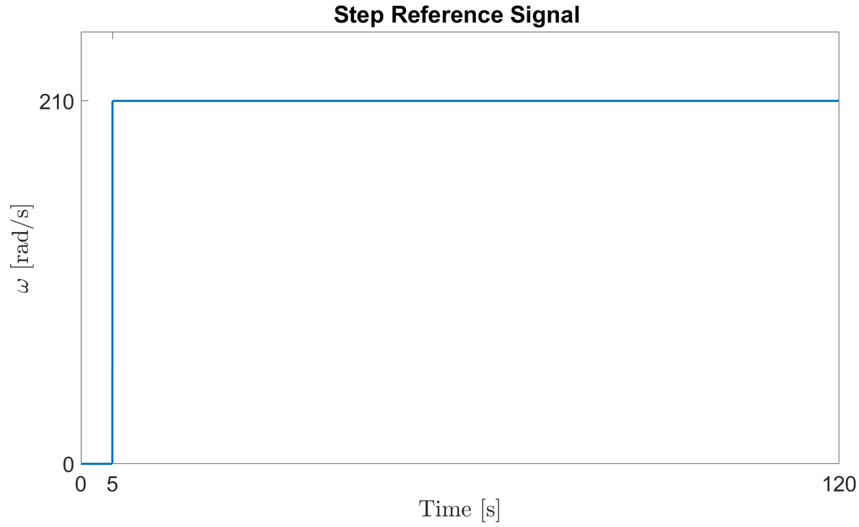

Figure 10: Angular speed reference for the propeller.

1. Derive a physical model highlighting assumptions and simplifications.
2. Reproduce the acausal model in Dymola.
3. Tune the cooling system in terms of volume flow rate, control logics, and initial fluid storage temperature such that:
   (a) the gear unit is kept between 40°C and 60°C.
   (b) the source tank does not get empty before the end simulation time
   (c) the storage tanks have a maximum height of 0.8 m and cross section area of 0.01 m^2
   (d) the system shall have a recirculating capability in order to exploit the outlet fluid for a next cooling process (when the source tank get empty)
   (e) the sink heated fluid is kept between 5°C and 10°C.
   (f) the power consumption of the thermal system shall be no more than 6 kW
4. Discuss the simulation results and the integration scheme used

For the simulation part consider properties of water at 10°C as cooling incompressible fluid (convective thermal conductance λ_{conv} = 300 W/K) and the cylindrical pipe line data listed in Table 6. The simulation shall last at least $\mathbf{t_{sim}}$ = 300 s starting with no water along the pipe.

Table 6: Pipe line properties

Parameter	Value	Unit
Diameter	4	cm
Length	40	cm
Geodetic height	0	m
Friction losses	0	[-]

(15 points)

Write your answer here

- Develop one Matlab script for Exercise 1; name the file `lastname123456_Assign2.m`. If needed, organize the script in sections and use local functions.
- Develop one Simulink model for Exercise 1; name the file `lastname123456_Assign2.slx`.
- Develop two Dymola models for Exercise 2; name the files `lastname123456_Assign2_Part_1.mo` and `lastname123456_Assign2_Part_2.mo`.
- Create a single .zip file containing the report in PDF, the MATLAB file, the Simulink model, and the two Dymola models. The name shall be `lastname123456_Assign2.zip`.
- Red text indicates where answers are needed; be sure there is no red stuff in your report.
- In your answers, be concise: to the point.
- **Deadline for the submission: Dec 18 2023, 23:59**.
- **Load the compressed file to the Homework folder on Webeep**.