16.90: Project #2
Heat Transfer Simulation of a Turbine Blade Trailing edge

1 Background

In the first stages of a turbine, the turbine blades are subjected to a high temperature flow due to the hot gas produced in the combustor. As a result, turbine blades are actively cooled by pumping lower temperature air through the blades in passages. One particular passage goes through the trailing edge of the turbine blade. Cooling air flows from the inside of the blade, towards and then out of the trailing edge. Figure 1 shows the geometry.
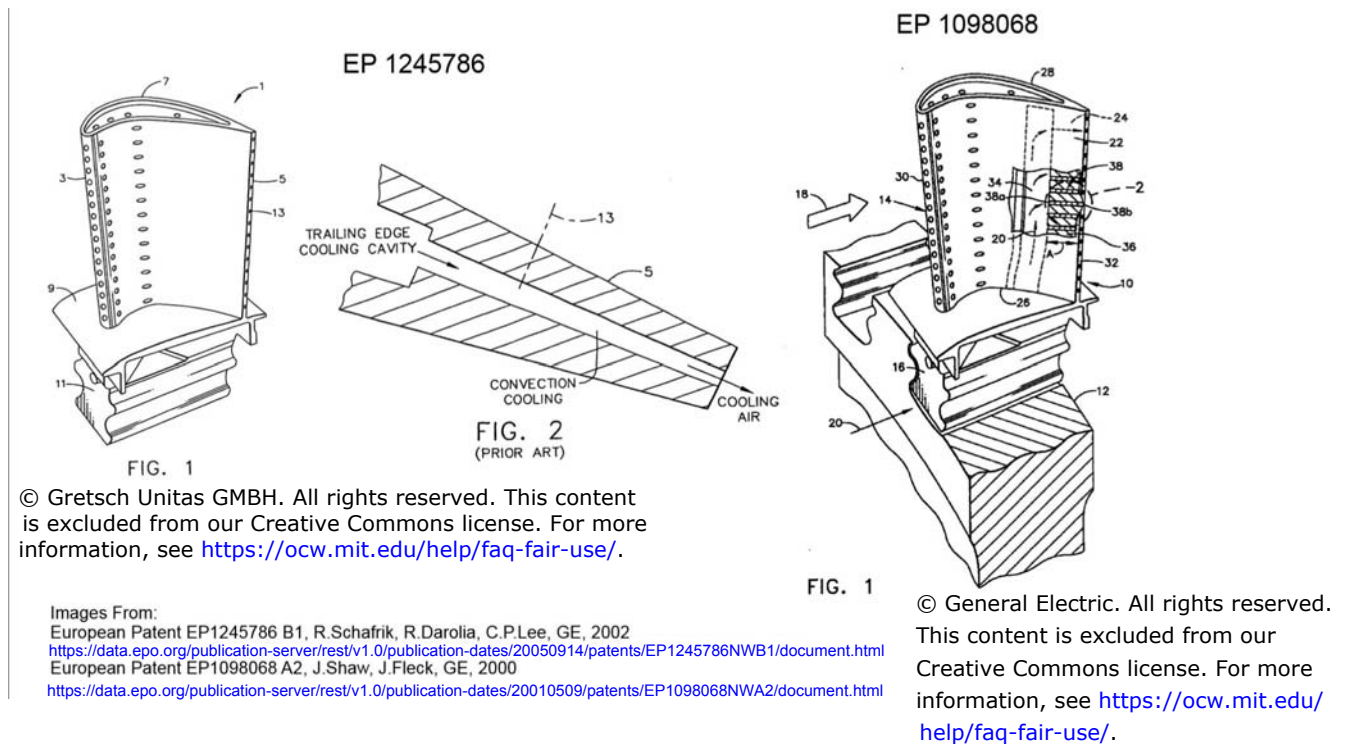
Figure 1: Trailing edge with cooling passages

In this project, we will simulate the heat transfer of an internally cooled turbine blade using a finite element discretization of the blade, and finite difference discretization of the cooling flow. The mesh and geometry is shown in Figure 2. The geometry is two-dimensional, assuming the blade geometry varies slowly in the spanwise direction.

Inside the turbine blade, the heat transfer is only due to conduction. Therefore, the steady state temperature distribution inside the turbine blade can be modeled using the Laplace equation in two-dimensions,

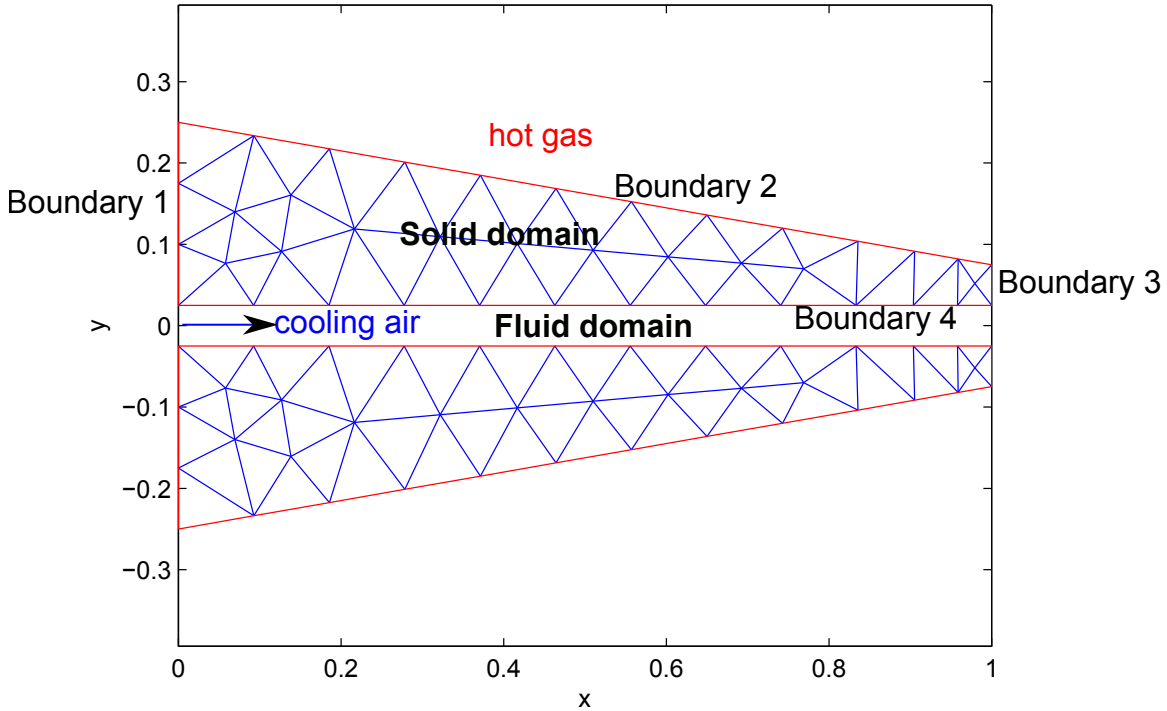$$0 = \nabla \cdot (k \nabla T), \tag{1}$$

Figure 2: Mesh of blade trailing edge with cooling passages (**in centimeters**). Due to symmetry, we only need to solve the heat equation in the upper half of the trailing edge.

where $k = 30\,W/(m\,K)$ is the thermal conductivity of the blade metal. We assume there is no heat transfer through the left boundary of the domain (Boundary 1). We have convective heating due to the hot exterior flow on the external boundaries (Boundary 2 and Boundary 3), and convective cooling due to the cooler interior flow along the internal boundaries (Boundary 4). Therefore, the boundary conditions on all other surfaces can be modeled using Newton's law of cooling. Recall that the conductive heat transfer rate is given by,

$$\vec{q} = -k\nabla T. \tag{2}$$

where k is the thermal conductivity of the blade material. For all surfaces (both internal and external to the blade), the heat flux out of the blade will be given by,

$$\vec{q} \cdot \vec{n} = h\,(T - T_t)\,, \tag{3}$$

where $h = 1500\,W/(m^2K)$ is the convective heat transfer coefficient and T_t is the stagnation temperature of gas away from the surface. $\vec{n}$ is a normal pointing out of the solid domain. For the outer surface of the blade, the stagnation temperature will be assumed to be,

$$T_{t,\mathrm{hot}} = 1400\,K. \tag{4}$$

The temperature inside the internal cooling passage depends on many factors. But the most important factors are the rate at which the gas is pumped, its temperature at the inlet

2

and heat capacity of the air. The air flows from left to right in the cooling passage in the figure shown. Assuming that heat transfer is fast in the direction normal to the flow, so that the temperature is uniform in the y-direction, the temperature distribution can be modeled by the one-dimensional energy conservation equation

$$c_p \dot{m} \frac{dT_{t,\text{cool}}}{dx} = 2\vec{q} \cdot \vec{n} \tag{5}$$

where $c_p = 1000 J/(kgK)$ is the heat capacitance of the cooling flow, $\dot{m}$ is cooling flow rate per unit span. The inlet of the cooling flow (on the left side of the domain at $x = 0$) has stagnation temperature of $800K$.

2 Tasks

2.1 Fluid Part

1. Write Equation (5) in the form of a conservation law. What is the flux function? What is the source?

2. Discretize Equation (5) using a finite volume scheme. In this question, the domain of $x \in [0, 1cm]$ is discretized into N uniform control volumes. Use upwind flux

$$F_{i+\frac{1}{2}} = F_i$$

3. Solve Equation (5) using finite volume, with the mass flow rate $\dot{m} = 0.05 kg/(ms)$, the boundary condition $T_{t,\text{cool}}(x = 0) = 800K$ and source term $\vec{q} \cdot \vec{n} = (1 - x/(1cm))MW/m^2$. Derive an analytic solution and compare against your finite volume solution. What is the order of convergence? What is the number of control volumes N necessary to compute a temperature distribution to an accuracy of $5K$?

4. Describe how your finite volume method would be different if the control volumes are non-uniform, and if the heat flux is a piecewise linear function (linear over each control volume.)

2.2 Thermal Part

5. Utilize Equations (2)-(3) to derive the boundary conditions of Equation (1) on all 4 sides of the solid domain. What is the type of the boundary condition on each side?

6. Write Equation (1) in a weak form. Specifically,

   (a) Derive the bilinear form from the differential equation.

   (b) Incorporate the desired boundary conditions into the bilinear form, and derive the boundary integrals, if any, resulting from the boundary conditions.

   (c) With nodal basis functions, derive the boundary integrals on each side of the reference triangular element (three integrals).

(d) Specify which terms you derived in (a), (b) and (c) will go into the stiffness matrix and which terms will go into the right hand side vector.

7. Utilize the skeleton MATLAB®code implement a finite element scheme to solve Equation (1) using nodal basis on a triangular mesh. Use the boundary conditions given by $T_{t,\text{hot}}$ = $1400\,K$, $T_{t,\text{cool}} = 800\,K$.. What is the temperature distribution on the side of the domain exposed to the cooling air? What is the heat flux $\dot{q}(x)$ on the side of the domain exposed to the cooling air? How does your answer depend on the mesh resolution?

2.3 Thermal Fluid Coupling

8. Solve Equation (5) with the $\dot{q}$ computed from the last part. Here you need to modify your code to deal with non-uniform control volumes given by the finite element mesh, piecewise linear heat flux $\dot{q}$ given by the finite element solution. What is the stagnation temperature distribution of the cooling air $T_{t,\text{cool}}(x)$?

9. Solve Equation (1) again with the temperature distribution $T_{t,\text{cool}}(x)$ calculated in the last question, which gives updated values for the boundary conditions. Now what is the heat flux $\dot{q}(x)$ on the side of the domain exposed to the cooling air? How different is it from the $\dot{q}(x)$ computed in Part 2?

10. Repeat Tasks 8 and 9 until $\dot{q}(x)$ converges. **Graph** the temperature distribution of the blade. What is the maximum temperature on the blade? How does it depend on the mesh resolution?

11. How does the maximum temperature of the blade vary as the cooling flow rate $\dot{m}$ vary from 0.01 to $0.05 kg/m$? Please plot the maximum temperature as a function of $\dot{m}$. What is the accuracy of your result?

16.90 Computational Methods in Aerospace Engineering

Spring 2014