

Univerzita Tomáše Bati ve Zlíně
Fakulta technologická
Institut řízení procesů a aplikované informatiky
akademický rok: 2004/2005

ZADÁNÍ DIPLOMOVÉ PRÁCE

(PROJEKTU, UMĚLECKÉHO DÍLA, UMĚLECKÉHO VÝKONU)

Jméno a příjmení: **Radek VICHEREK**
Studijní program: **M2807 Chemické a procesní inženýrství**
Studijní obor: **Automatizace a řídicí technika ve spotřebním průmyslu**

Téma práce: **Modelování tepelných polí v proudícím plynu**

Zásady pro vypracování:

1. Vypracujte rešerši zaměřenou na problematiku šíření tepla v proudícím plynu.
2. Sestavte matematický model, který bude popisovat šíření tepla v proudícím plynu. Pro simulaci proudícího plynu použijte program FEMLAB.
3. Ověřte vlastnosti šíření tepla pro různé velikosti proudících rychlostí. V modelu uvažte proudění působící na polovodičové přechody.
4. Navrhněte možný způsob měření teploty pomocí polovodičových přechodů, zvažte možnosti využití daného modelu pro měření malých průtoků plynů.

Rozsah práce:

Rozsah příloh:

Forma zpracování diplomové práce: **tištěná/elektronická**

Seznam odborné literatury:

1. WEBSTER, J.: The measurement, instrumentation and sensors. Handbook, USA, 1999.
2. LAMMERIMK, T., NIELS, R.: Micro/liquid flow sensor. Sensors and Actuators, 37/38, 1993.
3. ĎADO, S., KREIDL, M.: Senzory a měřicí obvody. ČVUT, Praha, 1999.
4. HEJZLAR, R.: Sdílení tepla. ČVUT, Praha, 1993.
5. KUNEŠ, J.: Modelování tepelných procesů. SNTL, Praha, 1989.
6. KUTATELADZE, S., BOROŠANSKIJ, V.: Příručka sdílení tepla. SNTL, Praha, 1962.
7. MÍKA, V., NEUŽIL, L., VLČEK, J.: Chemicko-inženýrské výpočty. VŠCHT, Praha, 1996.

Vedoucí diplomové práce:

Mgr. Milan Adámek, Ph.D.

Institut řízení procesů a aplikované informatiky

Datum zadání diplomové práce:

21. února 2005

Termín odevzdání diplomové práce:

15. června 2005

Ve Zlíně dne 18. února 2005



prof. Ing. Josef Šimoník, CSc.
děkan



prof. Ing. Vladimír Vašek, CSc.
ředitel institutu

ABSTRACT

The diploma thesis deals with the modeling of heat in a flowing gas. A low flow in the range of 10 to 100 ml/h is being considered.

For that purpose a time-of-flight sensor has been proposed. The sensor consists of a heater and series of semiconductor diodes placed in the pipe's wall. The temperature changes in the flowing gas induce changes of resistance upon the diodes. From the obtained thermal wave profile the temperature maximum position is assessed. The flow volume is then computed from the position shift in time.

To simulate this time-of-flight sensor and the gas flow the Femlab program was used. In the program, the flow velocity influence upon the heat profile of flowing gas was examined. The air, carbon dioxide and nitrogen were the applied gases. Next, the influence of radiation upon the temperature field in the flowing gas was examined.

From the simulations' results the most proper system parameters were determined.

ABSTRAKT

Diplomová práce se zabývá modelováním tepelných polí v proudícím plynu. Zaměřuje se na nízké průtoky v rozsahu 10 až 100 ml/h.

Za tímto účelem byl navržen teplotně-značkovací senzor sestávající ze žhavicího tělesa a řady polovodičových diod umístěných ve stěně trubice. Změna teploty proudícího plynu vyvolá změnu odporu těchto diod. Ze získaného profilu tepelné vlny se určí pozice maximální teploty. Posun maxima v závislosti na čase je úměrný průtoku plynu.

K simulaci tohoto teplotně-značkovacího senzoru a proudění byl z důvodu dostupnosti zvolen program Femlab. V něm byl zkoumán vliv rychlosti proudění na teplotní profil proudícího plynu. Jako médium byl uvažován vzduch, oxid uhličitý a dusík. Dále byl zkoumán vliv radiační složky na šíření tepla v proudícím plynu.

Z výsledků provedených simulací byly stanoveny nejvhodnější parametry celé soustavy.

ACKNOWLEDGEMENTS

I would like to take this opportunity to thank to Mgr. Milan Adámek, Ph. D. for his special guidance, valuable ideas and suggestions, consultations and overall help with various issues that have appeared during the time I worked on this thesis. I appreciate the help of the following people too - Ing. Miroslav Červenka for the time-of-flight sensor suggestion, Ing. Pavel Stříž for moral support and typographic lessons, Karel Bittner for some consultations. Last but not least I wish to thank my family.

Motto:

'E physis te physei terpetai.

'E physis te physei nika.

'E physis te physei kratei.

Othanes

CONTENTS

CONTENTS	5
INTRODUCTION	7
I. THEORETICAL PART	8
1 HEAT TRANSFER	9
1.1 CONDUCTION	10
1.2 CONVECTION	11
1.2.1 Forced convection	12
1.2.2 Free convection	12
1.3 RADIATION	13
1.3.1 Absorption and emissivity	13
1.3.2 Stefan-Boltzmann law	14
2 FLUID FLOW	16
3 LOW-FLOW GAS MEASUREMENT	19
3.1 HIGH-SENSITIVITY AIR FLOW SENSOR	19
3.2 TUBE MICROFLOW SENSOR	20
3.3 2D RESISTOR ARRAY MICROFLOW SENSOR	21
3.4 MICROMACHINED GAS FLOW SENSOR	21
3.5 GAS MICROFLOW SENSOR WITH SELF-HEATING TITANIUM RESISTORS	22
3.6 GAS MICROFLOW SENSOR UTILIZING CAPACITIVE PRESSURE SENSOR	23
3.7 FS1A SENSOR	23
3.8 FS2T SENSOR	24
4 MODELING OF HEAT TRANSFER IN FLUID FLOW	26
II. SIMULATION SOFTWARE	31
5 FEMLAB	32
6 THE FINITE ELEMENT METHOD	36
III. MAIN AIMS	37
IV. EXPERIMENTAL PART	39
7 TIME-OF-FLIGHT SENSOR	40
8 MODIFIED TIME-OF-FLIGHT SENSOR	41
9 MATHEMATICAL MODEL	46
10 APPLICATION MODES IN FEMLAB SIMULATION	49
10.1 CONVECTION AND CONDUCTION	50
10.2 INCOMPRESSIBLE NAVIER-STOKES	55
10.3 HEAT TRANSFER BY CONDUCTION	59

11	MODELS.....	63
11.1	AIR MODELS	65
11.2	CO ₂ MODELS	68
11.3	N ₂ MODELS	69
11.4	STEP CHANGE MODEL	70
	CONCLUSION	72
	BIBLIOGRAPHY	73
	LIST OF SYMBOLS AND OPERATORS.....	76
	LIST OF ABBREVIATIONS	79
	LIST OF FIGURES.....	80
	LIST OF TABLES	82
	LIST OF GRAPHS	83
	LIST OF APPENDICES	84

INTRODUCTION

Driven by the ever-changing requirements of industry and regular introduction of new European standards and legislation, manufacturers of process instrumentation constantly redevelop products that have only been on the market for a few years. Previously, product life cycles may have been in the region of 10 to 12 years, but now 3 to 5 years are not uncommon. For the end user the benefits are huge - modern instrumentation uses the latest microprocessor technology and the needs of the different market sectors can be addressed quickly and effectively. Today, instrumentation must be fit for purpose, modular and cost-effective.

Process level measurement technology has changed greatly in recent years. In the mid-1950s, capacitance systems were commonplace and although still available and widely used today, other techniques are proving more suitable and reliable in specific applications.

Obtaining accurate measurement of low flow rates of air and gas can be difficult. Most flow measurement devices are suited for the measurement of high flow rates, and will only provide reliable measurements above some minimum flow rate.

When it comes to low flow measurement of air and gases, the inherent benefits of thermal dispersion technology are worth reviewing. Traditionally, the most common benefit of thermal dispersion flow measurement is the intrinsic ability to directly measure the mass flow without need for pressure and temperature correction, as required with volumetric gas flow measurement. This not only provides a more useful flow measurement, but also makes thermal cost-effective. Perhaps an equally important and often overlooked benefit is the ability to provide reliable flow measurement at low flow rates, where other technologies are unable to operate.

There are many applications where the flow rate is simply too low, or the turndown requirements too high, to obtain acceptable results with the use of conventional gas flow measurement devices.

Typical applications include: vent lines, vacuum operations, landfill vapor recovery, digester gas flows, aeration, combustion air flow, convection air flow and exhaust gas.

I. THEORETICAL PART

1 HEAT TRANSFER

In the simplest of terms, the discipline of heat transfer is concerned with only two things: *temperature*, and the *flow of heat*. Temperature represents the amount of thermal energy available, whereas heat flow represents the movement of thermal energy from place to place. The second important law of thermodynamics is that heat always travels from a warm object to a colder one. The rate of heat travel is in direct proportion to the temperature difference between the two bodies.

On a microscopic scale, thermal energy is related to the kinetic energy of molecules. The greater a material's temperature, the greater the thermal agitation of its constituent molecules (manifested both in linear motion and vibrational modes). It is natural for regions containing greater molecular kinetic energy to pass this energy to regions with less kinetic energy.

Several material properties serve to modulate the heat transferred between two regions at differing temperatures. Examples include thermal conductivities, specific heats, material densities, fluid velocities, fluid viscosities, surface emissivities, and more. Taken together, these properties serve to make the solution of many heat transfer problems an involved process.

Heat transfer is defined as the movement of energy due to a temperature difference. [4] It is characterized by the following three mechanisms:

- *Conduction* is heat transfer by diffusion in a stationary medium due to a temperature gradient. Regions with greater molecular kinetic energy will pass their thermal energy to regions with less molecular energy through direct molecular collisions, i.e. conduction. The medium can be a solid or a liquid.
- *Convection* is heat transfer between either a hot surface and a cold moving fluid or a cold surface and a hot moving fluid. When heat conducts into a static fluid it leads to a local volumetric expansion. As a result of gravity-induced pressure gradients, the expanded fluid parcel becomes buoyant and displaces, thereby transporting heat by fluid motion (i.e. convection) in addition to conduction. Such heat-induced fluid motion in initially static fluids is known as *free (natural) convection*. For cases where the fluid is already in motion, heat conducted into the fluid will be transported away chiefly by fluid convection. These cases, known as

forced convection, require a pressure gradient to drive the fluid motion, as opposed to a gravity gradient to induce motion through buoyancy. Convection occurs in liquids and gases.

- *Radiation* is heat transfer between Surface A at temperature T_1 and Surface B at temperature T_2 via electromagnetic waves, provided that $T_1 \neq T_2$ and that Surface A is visible to an infinitesimally small observer on Surface B.

1.1 Conduction

When there exists a temperature gradient within a body, heat energy will flow from the region of high temperature to the region of low temperature. This phenomenon is known as conduction heat transfer, and is described by *Fourier's Law* (named after the French physicist Joseph Fourier),

$$\mathbf{q} = -k\nabla T \quad (1)$$

This equation determines the heat flux vector $\mathbf{q}$ for a given temperature profile T and thermal conductivity k . The minus sign ensures that heat flows down the temperature gradient. [6]

The temperature profile within a body depends upon the rate of its internally-generated heat, its capacity to store some of this heat, and its rate of thermal conduction to its boundaries (where the heat is transferred to the surrounding environment). Mathematically this is stated by the *Heat Equation*,

$$\nabla^2 T - \frac{1}{\alpha} \frac{\partial T}{\partial t} = -\frac{1}{k} q_{gen} \quad (2)$$

along with its boundary conditions, equations that prescribe either the temperature T on, or the heat flux q through, all of the body boundaries Ω ,

$$\begin{aligned} T(\Omega_a) &= T_{prescribed} \\ q(\Omega_b) &= q_{prescribed} \\ \Omega_a \cup \Omega_b &= \Omega \end{aligned} \quad (3)$$

In the Heat Equation, the power generated per unit volume is expressed by q_{gen} . The thermal diffusivity α is related to the thermal conductivity k , the specific heat c , and the density ρ by,

$$\alpha = \frac{k}{\rho c} \quad (4)$$

1.2 Convection

Heat energy transfers between a solid and a fluid when there is a temperature difference between the fluid and the solid. This is known as *convection heat transfer*. Generally, convection heat transfer can not be ignored when there is a significant fluid motion around the solid.

The temperature of the solid due to an external field such as fluid buoyancy can induce a fluid motion. This is known as *free convection* and it is a strong function of the temperature difference between the solid and the fluid. Blowing air over the solid by using external devices such as fans and pumps can also generate a fluid motion. This is known as *forced convection*.

Fluid mechanics plays a major role in determining convection heat transfer. For each kind of convection heat transfer, the fluid flow can be either laminar or turbulent. Laminar flow generally occurs in relatively low velocities in a smooth laminar boundary layer over smooth small objects, while turbulent flow forms when the boundary layer is shedding or breaking due to higher velocities or rough geometries. [6]

As is common with fluid mechanics analysis, a number of dimensionless parameters are employed to describe convective heat transfer. A summary of these variables is included in the following tables [5]:

Tab. 1 General convection (forced and free)

Parameter	Formula	Interpretation
Prandtl Number:	$Pr = \frac{\nu}{\alpha} = \frac{c_p \eta}{k}$	Ratio of fluid velocity boundary layer thickness to the fluid temperature boundary layer thickness.

Nusselt Number:	$Nu = \frac{hL}{k}$	Ratio of heat transferred from surface to heat conducted away by fluid.
-----------------	---------------------	---

Tab. 2 Forced convection only

Parameter	Formula	Interpretation
Reynolds Number:	$Re = \frac{wL}{\nu} = \frac{\rho wL}{\eta}$	Ratio of fluid inertia stress to viscous stress (for flow over flat plates).
Stanton Number:	$St = \frac{h}{\rho c_p w} = \frac{Nu}{Re \cdot Pr}$	

Tab. 3 Free convection only

Parameter	Formula	Interpretation
Grashof Number:	$Gr = \frac{g\beta\Delta TL^3}{\nu^2}$	Ratio of fluid buoyancy stress to viscous stress.
Rayleigh Number:	$Ra = Gr \cdot Pr$	

1.2.1 Forced convection

The essential ingredients of forced convection heat transfer analysis are given by *Newton's Law of Cooling*,

$$\dot{Q} = hA(T_w - T_\infty) = hA \cdot \Delta T \quad (5)$$

The rate of heat $\dot{Q}$ transferred to the surrounding fluid is proportional to the object's exposed area A , and the difference between the object temperature T_w and the fluid free-stream temperature T_∞ .

1.2.2 Free convection

Similar to forced convection, heat transfer due to free convection is described by *Newton's Law of Cooling*,

$$\dot{Q} = hA(T_w - T_\infty) = hA \cdot \Delta T \quad (6)$$

The rate of heat $\dot{Q}$ transferred to the surrounding fluid is proportional to the object's exposed area A , and the difference between the object temperature T_w and the fluid free-stream temperature T_∞ . [1]

1.3 Radiation

All materials radiate thermal energy in amounts determined by their temperature, where the energy is carried by photons of light in the infrared and visible portions of the electromagnetic spectrum. When temperatures are uniform, the radiative flux between objects is in equilibrium and no net thermal energy is exchanged. The balance is upset when temperatures are not uniform, and thermal energy is transported from surfaces of higher to surfaces of lower temperature.

Thermal radiation is defined as electromagnetic radiation in the wavelength range of 0.1 to 100 microns (which encompasses the visible light regime), and arises as a result of a temperature difference between 2 bodies.

No medium need exist between the two bodies for heat transfer to take place (as is needed by conduction and convection). Rather, the intermediaries are photons which travel at the speed of light.

The heat transferred into or out of an object by thermal radiation is a function of several components. These include its surface reflectivity, emissivity, surface area, temperature, and geometric orientation with respect to other thermally participating objects. In turn, an object's surface reflectivity and emissivity is a function of its surface conditions (roughness, finish, etc.) and composition.

1.3.1 Absorption and emissivity

Radiation heat transfer must account for both *incoming* and *outgoing* thermal radiation.

Incoming radiation can be either absorbed, reflected, or transmitted. This decomposition can be expressed by the relative fractions,

$$1 = \varepsilon_{\text{reflected}} + \varepsilon_{\text{absorbed}} + \varepsilon_{\text{transmitted}} \quad (7)$$

Since most solid bodies are opaque to thermal radiation, the transmission component can be ignored,

$$1 = \varepsilon_{\text{reflected}} + \varepsilon_{\text{absorbed}} \quad (8)$$

To account for a body's outgoing radiation (or its *emissive power*, defined as the heat flux per unit time), one makes a comparison to a perfect body who emits as much thermal radiation as possible. [9] Such an object is known as a blackbody, and the ratio of the actual emissive power E to the emissive power of a blackbody is defined as the surface emissivity ε ,

$$\varepsilon = \frac{E}{E_{\text{blackbody}}} \quad (9)$$

By stating that a body's surface emissivity is equal to its absorption fraction, *Kirchhoff's Identity* binds incoming and outgoing radiation into a useful dependent relationship,

$$\varepsilon = \varepsilon_{\text{absorbed}} \quad (10)$$

1.3.2 Stefan-Boltzmann law

One of the first to recognize that heat radiation is related to light was the English astronomer William Herschel, who noticed in 1800 that if a thermometer was moved from one end of a prism produced spectrum to the other, the highest temperatures would register below the red band, where no light was visible. Because of this position, this form of radiation is called infrared (*infra* being the Latin word for *below* or *within*).

Stefan-Boltzmann law states that the total energy radiated per unit surface area of a black body in unit time is directly proportional to the fourth power of its thermodynamic temperature T ,

$$E_{\text{blackbody}} = \sigma T^4 \quad (11)$$

The non-fundamental constant of proportionality is called the *Stefan-Boltzmann constant* σ . Its value is

$$\sigma = \frac{2\pi^5 k^4}{15h^3 c^2} = 5.67 \times 10^{-8} \frac{\text{W}}{\text{m}^2 \text{K}^4} \quad (12)$$

where π is an extra factor that converts from a brightness per solid angle to total brightness

c is the speed of light

k is Boltzmann's constant

h is Planck's constant.

[12]

2 FLUID FLOW

By definition, a fluid is a material continuum that is unable to withstand a static shear stress. Unlike an elastic solid which responds to a shear stress with a recoverable deformation,

a fluid responds with an irrecoverable flow.

In order for a fluid to flow from one point to another, there must be a difference in pressure between the two points to cause the flow. With no pressure difference, no flow will occur.

The units used to describe the flow can be of several types. [3] The most common descriptions are the following:

- *Volume flow rate* - expressed as a volume delivered per unit time,

$$Q_v = \lim_{\Delta t \rightarrow 0} \frac{\Delta V}{\Delta t} = \frac{dV}{dt} \quad (13)$$

where ΔV is small unit volume of flowing fluid,

Δt is time interval.

- *Mass flow rate* - expressed as mass flowing per unit time,

$$Q_m = \lim_{\Delta t \rightarrow 0} \frac{\Delta m}{\Delta t} = \frac{dm}{dt} = \rho \frac{dV}{dt} \quad (14)$$

where Δm is small unit mass of flowing fluid,

ρ is density of the fluid.

- *Flow velocity* - expressed as the distance the fluid travels in the carrier per unit time, this is related to the volume and the mass flow rate by,

$$v = \frac{Q_v}{S} = \frac{Q_m}{\rho S} \quad (15)$$

where Q_v is volume flow rate,

Q_m is mass flow rate,

S is cross-sectional area of flow carrier.

Examples of fluids include gases and liquids. Typically, liquids are considered to be incompressible, whereas gases are considered to be compressible. However, there are exceptions in everyday engineering applications.

Fluid flow can be either *laminar* or *turbulent*. The factor that determines which type of flow is present is the ratio of inertia forces to viscous forces within the fluid, expressed by the nondimensional Reynolds Number [8],

$$Re = \frac{\rho w L}{\eta} \quad (16)$$

where w and L are a fluid characteristic velocity and distance. For example, for fluid flowing in a pipe, w could be the average fluid velocity, and L would be the pipe diameter.

Typically, viscous stresses within a fluid tend to stabilize and organize the flow, whereas excessive fluid inertia tends to disrupt organized flow leading to chaotic turbulent behavior.

Fluid flows are laminar for Reynolds Numbers up to 2000. Beyond a Reynolds Number of 4000, the flow is completely turbulent. Between 2000 and 4000, the flow is in transition between laminar and turbulent, and it is possible to find subregions of both flow types within a given flow field.

At low Reynolds Numbers (below 2000), flow is dominated by viscous forces and the velocity profile is (elongated) parabolic. At high Reynolds Numbers (above 20000), the flow is dominated by inertial forces, resulting in a more uniform axial velocity across the flowing stream and a flat velocity profile. [7]

Laminar fluid flow is described by the *Navier-Stokes* equations. For cases of inviscid flow, the Bernoulli equation can be used to describe the flow. When the flow is zero (i.e. statics), the fluid is governed by the laws of fluid statics.

The Navier-Stokes equations for fluid flow,

$$\begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} - \eta \nabla^2 \mathbf{u} + \rho(\mathbf{u} \cdot \nabla) \mathbf{u} + \nabla p &= \mathbf{F} \\ \nabla \cdot \mathbf{u} &= 0 \end{aligned} \quad (17)$$

are deduced for incompressible Newtonian flow, ie. it is assumed that the material's density is constant or almost constant. [13]

Starting with the momentum balance in terms of stresses, the generalized equations in terms of transport properties and velocity gradients are

$$\begin{aligned}\rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot [\eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} + \nabla p &= \mathbf{F} \\ \nabla \cdot \mathbf{u} &= 0\end{aligned}\tag{18}$$

The first equation is the *momentum balance*, and the second is the *equation of continuity* for incompressible fluids. The following variables and parameters appear in the equations:

∇ is the vector differential operator:

$$\nabla = \left(\frac{\partial}{\partial x_1} + \dots + \frac{\partial}{\partial x_n} \right)\tag{19}$$

η is the dynamic viscosity

ρ is the density

$\mathbf{u}$ is the velocity field

p is the pressure

$\mathbf{F}$ is a volume force field such as gravity.

3 LOW-FLOW GAS MEASUREMENT

Recent advances in flow sensing have resulted in more accurate, durable, and economical meters. Increasingly important is the ability to interface meters with a computer for instantaneous flow readout to remotely control flow, or to allow unattended process operation. To meet such demands, new types of flowmeters are being introduced, and older designs are being improved and updated.

Prime considerations when selecting a flow sensor include: the type of fluid being measured, its temperature and pressure, viscosity, conductivity, corrosiveness, and cleanliness. Equally important are the requirements of the sensor itself: flow velocity range, accuracy, ease of installation, and maintenance requirements. Finally, cost can be a major factor in the decision. [2]

The majority of flow sensors can handle high flow rates. For the low flow rates several special flowmeters have been designed. A few examples are being introduced:

3.1 High-sensitivity air flow sensor

The scheme drawing of the micro flow sensor is shown in Fig. 1. The micro flow sensor is a high-sensitivity air flow sensor fabricated using silicon micro machining techniques. The size of the chip is $1.7 \text{ mm} \times 1.7 \text{ mm}$ and the thickness is 0.4 mm. A heater is installed on the center of the chip and two resistance thermometers are installed at the both sides of the heater. The flow velocity can be obtained by detecting the temperature difference with the thermometers when flow passed through the sensor.

The experimental apparatus consists of a standard flow generator and a tested mass flow meter for gases which installed the micro flow sensor on the velocity boundary layer in the pipeline. The standard flow rate is generated controlling the pressure change during air discharge in an isothermal chamber using a servo valve. [24]

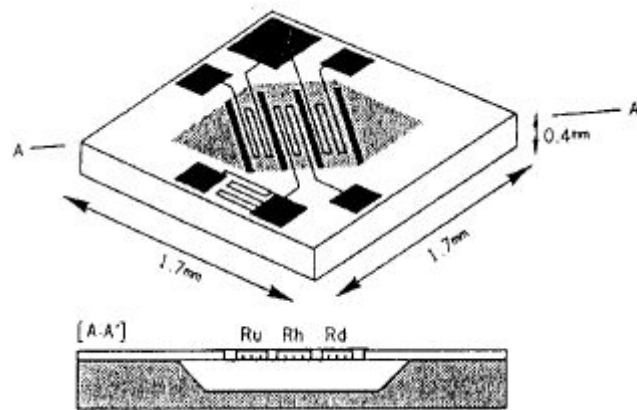


Fig. 1 High-sensitivity air flow sensor

3.2 Tube microflow sensor

The actual flow sensor consists of a straight flow tube with two active elements around it, as shown in Fig. 2. The internal diameter of the flow tube may vary between 20 and 200 μm . Two measurement principles can be distinguished, namely, the constant power (CP) and the constant temperature (CT) method. The CP measurement principle is used for the flow ranges below circa 100 microliter/min. In this case, the two elements are used both as heater and as temperature sensor, as shown in Fig. 2a. Both elements are provided with an equal amount of constant power, the temperature difference ΔT between them is a measure for the flow. The CT measurement principle is used for the flow ranges above circa 100 microliter/min. In this case, the first element acts as temperature sensor, and the second element acts as a heater, as shown in Fig. 2b. The heater is heated to a certain constant temperature ΔT over the medium temperature. The heater power P_{heater} necessary to keep ΔT constant is a measure for the flow. [25]

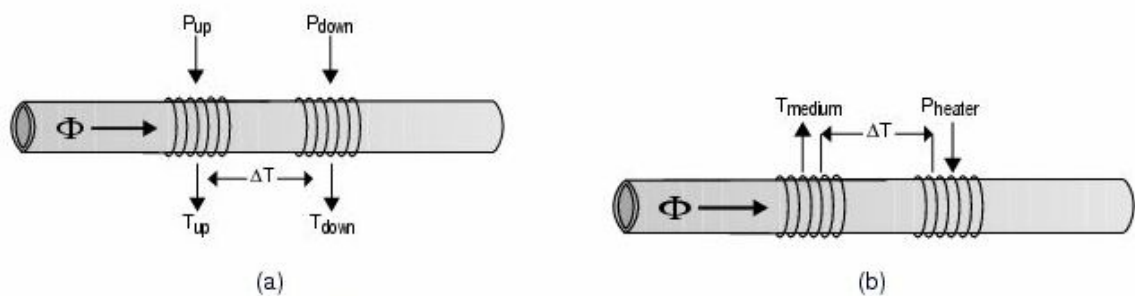


Fig. 2 Tube microflow sensor according to the

(a) constant power and (b) constant temperature measurement principle

3.3 2D resistor array microflow sensor

The flow sensor structure is based on a micro-machined two-dimensional resistor array crossing a flow channel. A segmented beam with platinum heater and voltage contact leads is realized in a micro-machined duct. With this structure the temperature distribution perpendicular to the flow can be measured. From the temperature distribution the flow velocity can be measured independent of the temperature coefficient or resistance of the platinum resistors. A schematic drawing of the structure is given in Fig. 3. Five segmented beams across a channel are presented. Each beam consists of a carrier with a platinum heater and additional voltage leads on top of it. The temperature distribution of the array depends on three aspects: 1. heat transport by conduction through the beams and the fluid, 2. heat transport by forced convection due to the fluid and 3. the spatial and temporal actuation pattern. [26]

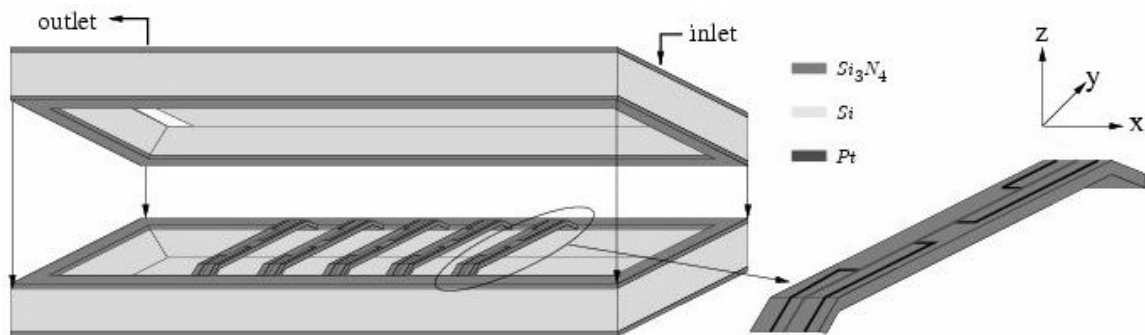


Fig. 3 2D resistor array microflow sensor

3.4 Micromachined gas flow sensor

A cantilever design of a microflow sensor is shown in Fig. 4. The thickness of the cantilever may be as low as 2 μm . It is fabricated in the form of a sandwich consisting of layers of field oxide, chemical vapor deposition oxide, and nitrate. The cantilever sensor is heated by an imbedded resistor with a rate of 26 K/mW of applied electric power, and a typical transfer function of the flow sensor has a negative slope of about 4 mV/m/s. The heat is removed from the sensor by three means: conductance through the cantilever beam, gas flow, and thermal radiation. [1]

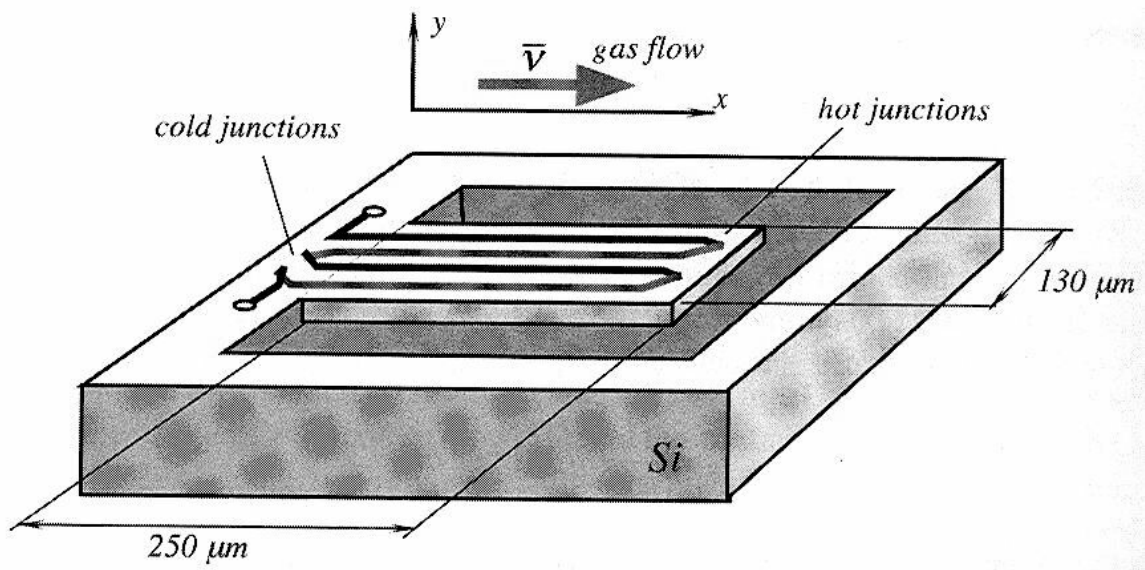


Fig. 4 Micromachined gas flow sensor

3.5 Gas microflow sensor with self-heating titanium resistors

Another design of a thermal transport microsensor is shown in Fig. 5, where titanium films having a thickness of $0.1\ \mu\text{m}$ serve as both the temperature sensors and the heaters. The films are sandwiched between two layers of SiO_2 . Titanium is used because of its high temperature coefficient of resistance and excellent adhesion to SiO_2 . Two microheaters are suspended with four silicon girders at a distance of $20\ \mu\text{m}$ from one another. The titanium film resistance is about $2\ \text{k}\Omega$. [1]

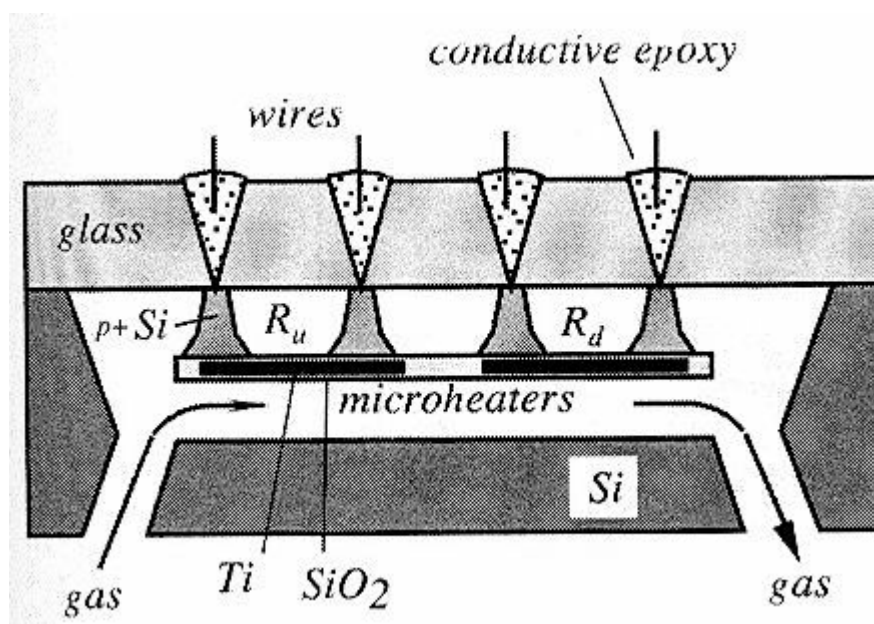


Fig. 5 Gas microflow sensor with self-heating titanium resistors

3.6 Gas microflow sensor utilizing capacitive pressure sensor

A microflow sensor can be constructed by utilizing a capacitive pressure sensor as shown in Fig. 6. An operating principle of the sensor is based on a pressure gradient technique. The sensor is fabricated using silicon micromachining and defused boron etch-stops to define the structure. The gas enters the sensor's housing at pressure P_1 through the inlet, and the same pressure is established around the silicon plate, including the outer side of the etched membrane. The gas flows into the microsensor's cavity through a narrow channel having a relatively high-pressure resistance. As a result, pressure P_2 inside the cavity is lower than P_1 , thus creating a pressure differential across the membrane. [1]

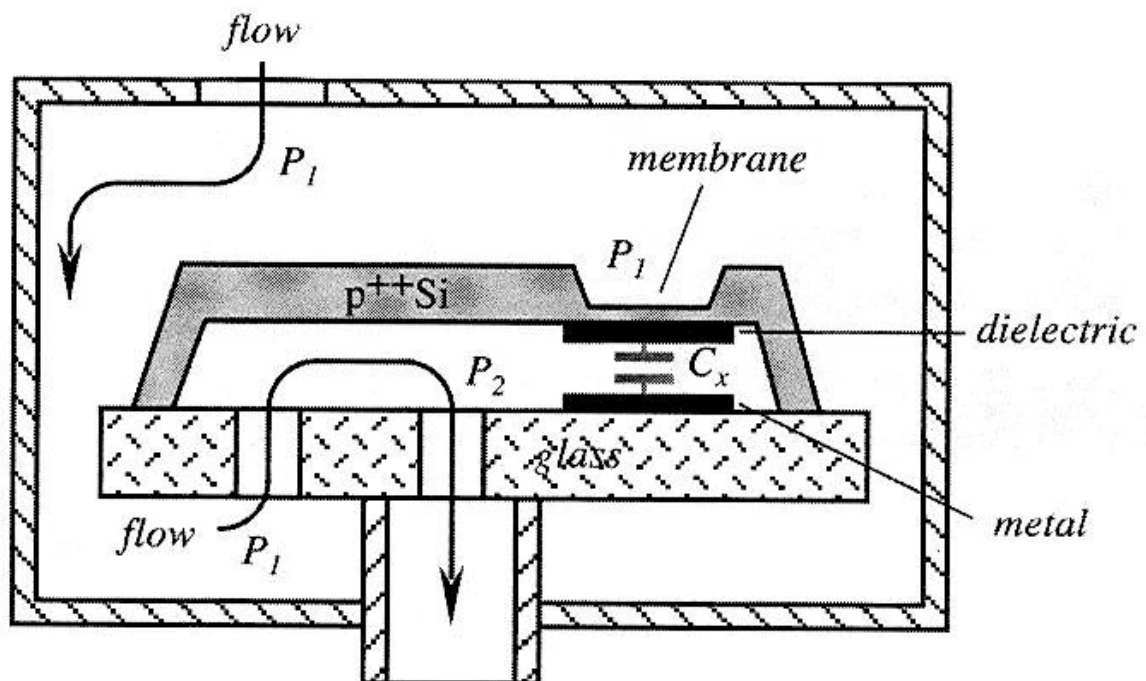


Fig. 6 Gas microflow sensor utilizing capacitive pressure sensor

3.7 FS1A sensor

This element consists of two temperature depending platinum-resistors, both deposited on a substrate, Fig. 7. The low-ohm resistor with a small area is used as a heater, whereas the other high-ohm resistor serves to measure the reference temperature. As a result of his little thermal mass, this sensor has fast heating and cooling response times. The measuring

principle of the sensor can be used for large operation ranges, from 0...0.1 m/s up to 100 m/s.

The sensor is based on the well known calorimetric principle. Using a bridge circuit, the differing resistance value of two elements leads to different (self) heating. The (self) heating is dependent upon the applied voltage and the mass flow of the media in which the sensor is located. Higher voltage increases self heating, a higher flow rate increases cooling, or lowers heating respectively. If the self heating is kept constantly by a suitable controller, the voltage increases with higher flow rates. So the bridge circuit is becoming a measuring unit of the flow velocity.

While a temperature variable resistor (RTD) is heated by an electrical current, the temperature of the heated element (RTD) can be measured. Raising the temperature results in higher resistance, consequently also the cooling of the heated element can be measured. Further more to get an extensively temperature independent measurement which allows for medium temperature and other varying mass properties, an additional temperature variable resistor is structured on the chip. This resistor measures a proportional signal to the ambient respectively to the medium temperature and is used for compensation. [27]

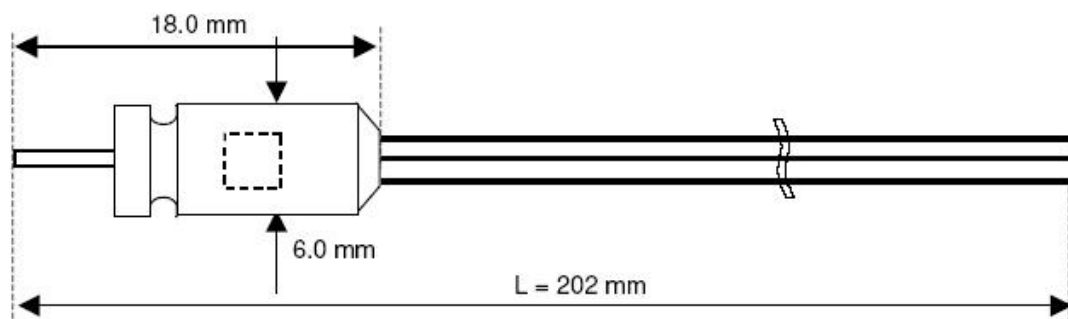


Fig. 7 FS1A sensor

3.8 FS2T sensor

This element consists of three or four temperature depending platinum- resistors, all deposited on one substrate, Fig. 8. The low-ohm resistor with a small area is used as a heater, whereas the two high-ohm resistors on the left and right side are for measuring the mass flow and the direction. A fourth resistor can be used to measure the reference temperature.

As a result of the little thermal mass, this flow sensor has fast heating and cooling response times. This system allows to measure a very small flow volume (0.001 ... 1 m/s) and at the same time can detect the direction of the flow.

Two temperature depending resistors are symmetrical arranged, which are indirectly heated by heat transmission with a heated element in between. If there is no flow, both of the elements are heated equal. Presence of a flow results in a temperature difference of the two elements beside the heater, which is a measure for the flow direction and the mass flow. The measuring element in front of the heater in flow direction is cooled down more than the other. Therefore the heat transmission from heater to the first element is restricted, the transmission to the second element is supported by the flow. [27]

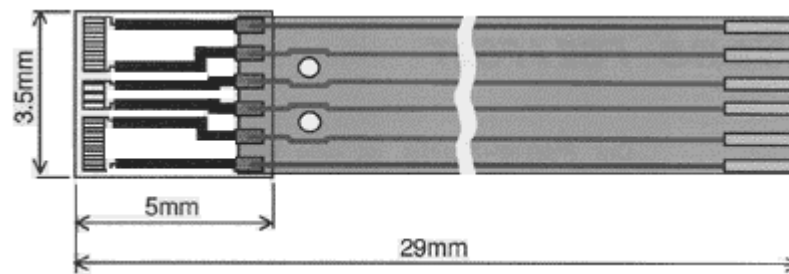


Fig. 8 FS2T sensor

4 MODELING OF HEAT TRANSFER IN FLUID FLOW

The equations of motion in fluid dynamics form a highly complicated coupled system of non-linear partial differential equations of first or second order. [5] As expected from such a complicated system, they turn out to contain lots of interesting physical behaviour: waves, shock waves, turbulent flow, and even chaos, as well as various simpler kinds of motions. As no general solution to this difficult equation system is available, and not even theorems proving existence, uniqueness and stability of solutions to the fully general equations of motions have been established, many situations in fluid dynamics can only be thoroughly studied by the use of computer simulations. Indeed, the modern field of computational fluid dynamics (CFD) is a vast and very active field of research in physics and applied mathematics.

A computer can only handle a problem of finite size. CFD works by dividing the region of interest, the inside of a mixing vessel or the air around the outside of a car for example, into a large number of cells or control volumes (the mesh or grid). In each of these cells, the partial differential equations describing the fluid flow (the Navier-Stokes equations) are rewritten as algebraic equations that relate the pressure, velocity, temperature and other variables, such as species concentrations, to the values in the neighboring cells. These equations are then solved numerically yielding a complete picture of the flow down to the resolution of the grid.

The first stage in any CFD model is to create a geometry which represents the object being modeled. From this point, a mesh is generated which creates the cells or control volumes. Once the mesh is complete, the model input values are specified and the software then solves the equations of state for each cell until an acceptable convergence is achieved. This can be a time consuming process.

When the model has been solved, the results can be analyzed both numerically and graphically.

These techniques date back to the early 1970's, and the first commercial CFD software became available in the early 1980's. Since then, CFD has come a long way - geometric flexibility has increased to the point where there are now very few geometries too complex to be represented accurately; models have been developed for physical phenomena such as turbulence, multiphase flow, chemical reactions and radiative heat transfer; and the usability of software has increased greatly, with powerful pre- and post-processors.

Various software products from the fluid flow area include:

PROFESSIONAL CFD (CORE PACKAGE)

This core package [14] includes steady-state and transient heat transfer analysis; steady and unsteady fluid flow analysis; *FEMPRO*, an easy-to-use, single user interface for finite element modeling, results evaluation and presentation; and a suite of modeling capabilities.

Heat transfer analysis capabilities solve linear and nonlinear thermal designs by considering conduction, convection, heat flux, heat generation, radiation and thermal contact in steady-state or transient analyses.

Fluid flow analysis capabilities solve steady and unsteady flow patterns for incompressible, laminar, Newtonian or non-Newtonian fluid flow. Turbulence capabilities enable the prediction of turbulent flow (large changes in velocity over small distances) and laminar flow (smooth, gradual changes in velocity distribution) at the same time in the same model.

Combined heat transfer and fluid flow analysis capabilities allow engineers to simulate a product's behavior when both thermal and fluid flow factors interact. User-friendly tools and wizards automate the application of results from one type of analysis to another for coupled heat transfer and fluid flow analysis to accurately simulate free, forced or mixed convection.

ANSYS MULTIPHYSICS™

In the past, obtaining all of the simulation capabilities needed for complex and demanding modeling scenarios frequently meant combining several different software packages. *ANSYS Multiphysics* provides the analysis industry's most comprehensive coupled physics tool combining structural, thermal, CFD, acoustic and electromagnetic simulation capabilities into a single software product.

ANSYS Multiphysics integrates the power of direct (matrix) and sequential (load vector) coupling to combine the appropriate "physical fields" required for accurate, reliable simulation results in applications ranging from cooling systems, power generation, to

biotechnology and Micro Electro Mechanical Systems (MEMS). The software easily simulates complex thermal-mechanical, fluid-structural and electrostatic-structural interactions, and includes the complete range of powerful *ANSYS* iterative, direct and eigenvalue matrix solvers. [15]

SINDA/FLUINT

SINDA/FLUINT is a comprehensive finite-difference, lumped parameter (circuit or network analogy) tool for heat transfer design analysis and fluid flow analysis in complex systems.

SINDA/FLUINT is a comprehensive, generalized tool for simulating complex thermal/fluid systems such as those found in the electronics, automotive, petrochemical, and aerospace industries. [16]

FLUENT

Preprocessing is the first step in building and analyzing a flow model. It includes building the model (or importing from a CAD package), applying a mesh, and entering the data. Three preprocessing tools are supplied, *GAMBIT*, *G/Turbo* and *TGrid*. Third-party preprocessing tools can also be used.

After preprocessing, the CFD solver does the calculations and produces the results. Four general-purpose products are provided: *FLUENT*, *FloWizard*, *FIDAP*, and *POLYFLOW*. *FLUENT* is used in most industries. *FloWizard* is the first general-purpose CFD product for designers built by *Fluent*. *FIDAP* and *POLYFLOW* are also used in a wide range of fields, with emphasis in the materials processing industries.

Postprocessing is the final step in CFD analysis, and involves organization and interpretation of the data and images. All *Fluent* software includes full postprocessing capabilities.

Third-party postprocessors can also be used. [17]

ADINA

The *ADINA* system offers a one-system program for comprehensive finite element analyses of structures, fluids, and fluid flows with structural interactions.

The *ADINA-T* program is used for the solution of heat transfer problems in solids and structures. It has powerful features such as radiation between surfaces of arbitrary geometries, element birth-death options and capabilities for highly nonlinear material behavior. [18]

LISA

Multi-physics FEA (finite element analysis) software for analyses: static, dynamic (eigenvalue), dynamic response, steady and transient thermal, fluid (including Navier-Stokes equations), seepage, acoustic, and electro-magnetic. Software provides Active-X port for individual use. [19]

FLUIDYN

Fluidyn-MP (MultiPhysics) is a general purpose CFD software for the simulation of fluid structure coupling. MP simultaneously calculates heat conduction, mechanical stress & vibratory phenomena in structures & flow and heat convection in the fluids in contact. The structure & the fluid are coupled totally by the automatic exchange of boundary conditions. *Fluidyn-MP* offers an innovative strong coupling technique to simultaneously model heat transfer, mechanical deformation and acoustic vibrations.

Modeling heat transfer in fluid / structure include heat conduction modeling in the structure by Finite Elements Method, convection & radiation in the fluid calculated by Finite Volumes Method, Finite Elements Matrix Solution for conduction calculation in structures, automatic calculation of coefficient of heat transfer between the fluid & the wall (Stanton number computation by Reynolds theory). Radiation modeling is developed for transparent media (shadow effect of different obstacles accounted for) or semi-transparent media (six flux model, discrete ordinate model). [20]

PORFLOW™

*PORFLOW*TM is a comprehensive CFD tool developed by *Analytic & Computational Research, Inc., ACRI*, to accurately solve problems involving transient or steady state fluid flow, heat, salinity and mass transport in multi-phase, variably saturated, porous or fractured media with dynamic phase change. The porous/fractured media may be anisotropic and heterogeneous, arbitrary sources (injection or pumping wells) may be present and, chemical reactions or radioactive decay may take place. It accommodates alternate fluid and media property relations and complex and arbitrary boundary conditions. The geometry may be 2D or 3D, Cartesian or Cylindrical and the mesh may be structured or unstructured, giving maximum flexibility to the user. [21]

II. SIMULATION SOFTWARE

5 FEMLAB

Femlab, by Stockholm-based *Comsol*, was one of the earliest integrated Finite Element Method (FEM) solvers (i.e. handling all steps of the FEM analysis). Originally launched in 1985 as a PDE (partial differential equation) Toolbox for *Matlab*, in 1998 it was succeeded by a 2D Femlab, and in 2001 with a 3D Femlab that first stressed the multiphysics capability. In form, these were still *Matlab* Toolboxes. Femlab 3.0 makes a leap to stand-alone operation. It's not a complete divorce: Femlab's storage format is still a *Matlab* structure, so Femlab can be driven direct from the *Matlab* command line or a Femlab session can be saved as an M-file. However, *Matlab* ownership is not mandatory. [22]

Femlab is a powerful, interactive environment for modeling and solving all kinds of scientific and engineering problems based on PDEs. With Femlab can be easily extended conventional models for one type of physics into multiphysics models that solve coupled physics phenomena – and do so simultaneously. Accessing this power does not require an in-depth knowledge of mathematics or numerical analysis. By using the built-in *physics modes* it is possible to build models by defining the relevant physical quantities such as material properties, loads, constraints, sources, and fluxes, rather than by defining the underlying equations. Femlab then internally compiles a set of PDEs representing the entire model.

As noted, the underlying mathematical structure in Femlab is a system of partial differential equations. There are three ways of describing PDEs through the following mathematical application modes:

- *Coefficient form*, suitable for linear or nearly linear models
- *General form*, suitable for nonlinear models
- *Weak form*, for models with PDEs on boundaries, edges, or points, or for models using terms with mixed space and time derivatives.

Using these application modes, various types of analysis can be performed:

- Stationary and time-dependent analysis
- Linear and nonlinear analysis
- Eigenfrequency and modal analysis.

When solving the PDEs, Femlab uses the proven finite element method. The software runs the finite element analysis together with adaptive meshing and error control using a variety of numerical solvers.

PDEs form the basis for the laws of science and provide the foundation for modeling a wide range of scientific and engineering phenomena. Therefore Femlab can be used in many application areas, just a few examples being:

- Acoustics
- Bioscience
- Chemical reactions
- Diffusion
- Electromagnetics
- Fluid dynamics
- Geophysics
- Heat transfer
- Optics
- Porous media flow
- Quantum mechanics
- Semiconductor devices
- Structural mechanics
- Transport phenomena.

[10]

Femlab is a complete package that covers all facets of the modeling process. It contains CAD tools, interfaces for physics and equation specifications, automatic mesh generation, a variety of optimized solvers, as well as visualization and postprocessing tools.

Ready-made application modes encapsulate the key equations and properties for a large number of physical processes and phenomena. Yet, Femlab also furnishes an open and extensible environment that offers access to all stages of the modeling process.

Femlab's interactive modeling environment allows building and analyzing models from start to finish without the need to involve any other software packages. Its integrated tools

allow working efficiently at each step in the process, all within one consistent and easy-to-use graphical environment. It's easy to move back and forth between various stages such as setting up the geometry, defining the physics, creating a mesh, solving the model, and performing postprocessing. Femlab's associative geometry feature preserves any boundary condition or equation even if the geometry is changed.

The modeling procedure typically involves the following steps:

1. CREATE THE GEOMETRY

Femlab provides powerful CAD tools for creating 1D, 2D and 3D geometric objects using solid modeling. In addition, Femlab models can be based on geometric designs created in other software packages. The software also imports 2D images in JPG, TIF and BMP formats and converts them into Femlab geometry objects.

2. DEFINE THE PHYSICS

Femlab makes the modeling of many physical processes and equations effortless. It includes values or expressions for viscosity and density in the Navier-Stokes equations. The properties can be isotropic or anisotropic; they can be functions of the modeled variables, the spatial coordinates and time.

3. GENERATE THE FINITE ELEMENT MESH

Femlab can create triangular or tetrahedral unstructured meshes as well as quadrilateral meshes. Adaptive meshing automatically refines the mesh wherever needed to improve accuracy. The mesh generator can be controlled manually to resolve the geometry.

Femlab's unmatched multiphysics capabilities allow free coupling of different physical phenomena. A Femlab model solves the flow and concentration fields in a micromixer. A passive micromixer works by shredding the flow into thin slices, where the diffusion length is decreased between the two mixtures. The model can be used to calculate the degree of mixing in different designs.

4. COMPUTE THE SOLUTION

Femlab runs time-dependent or stationary simulations for linear and nonlinear systems. They benefit from state-of-the-art direct and iterative methods, multigrid preconditioners, efficient time-stepping algorithms and eigenmode analyses. The solution process can be

automated to solve for different field variables. This feature eliminates the need to stand by and modify solver settings for each iteration.

5. VISUALIZE AND POSTPROCESS THE RESULTS

Femlab provides extensive visualization capabilities, among them:

- Interactive plotting of all field variables and other application-specific properties
- Visualization of several solution properties in all parts of the geometry simultaneously using slices, isosurfaces, contours, streamlines, height and vector-field plots
- High-performance graphics with OpenGL hardware acceleration
- Animation using AVI and QuickTime
- Integration along boundaries and subdomains
- Display results data at any location in the geometry using full numeric precision
- Cross-section and domain plots for projection of solution variables along surfaces and lines as well as plots of variations over time at any location in the geometry
- Report generator for documenting models. The reports can be printed directly and also saved as extensible HTML files.

6. PERFORM OPTIMIZATION AND PARAMETRIC ANALYSIS

The parametric solver in Femlab provides the perfect way for examining a series of conditions. The built-in *Matlab* interface can save Femlab models as M-files for later incorporation as functions into *Matlab* scripts for optimization.

7. UNLIMITED MULTIPHYSICS COMBINATIONS

Femlab provides a multiphysics modeling and simulation environment for the coupling of physics phenomena. Complex models can be built by combining several of the package's integrated, ready-to-use applications modes or using equation based modeling. With Femlab's extended multiphysics features can be freely coupled systems of multiple dimensions or coordinated systems into a single model. [23]

6 THE FINITE ELEMENT METHOD

The Finite Element Method (FEM) is one of the central techniques for physics simulation. It bridges the gap between the late 18th century, when mathematicians such as Lagrange recognised that a variety of physical phenomena were governed by partial differential equations, and our present ability to apply these equations to objects such as engineering components (or even multiphysics problems where linked PDEs operate simultaneously).

The term *finite element method* dates from around 1960, coined by structural engineer Ray Clough of the University of California, but the generally acknowledged originator of the FEM was the German mathematician Richard Courant. Courant used the technique in 1943 to solve a torsion problem on a cylinder, although he first referred to it in a co-written 1922 book on function theory. Courant in turn drew heavily on the work of Lord Rayleigh, Boris Galerkin, and Walter Ritz.

The main reason driving such an approach was the insolubility of PDEs for all but simple geometries. Much simplifying, the FEM is a transformation to discretise the problem. The region under consideration is divided into a number of geometrical elements, in each of which the continuous field of the PDE variables is modelled by a local polynomial approximation controlled by a few coefficients. Linking these elements by the values at shared nodes results in a set of simultaneous algebraic equations that can be solved numerically by optimisation and matrix algorithms.

Generally, FEM modelling operates in standard stages: modelling of the region geometry; *meshing* (creating the framework of element nodes); setup of the PDEs and their boundary conditions; solution; and post-processing for viewing.

Although recent years have brought remarkable developments - particularly the capacity to run, on a PC, models that only a decade ago required a workstation - the finite element method isn't a magical answer to all PDE problems. One limitation is that, as a numerical technique, it can never output a closed-form solution. Another is that it can go badly wrong. Lothar Collatz, in *The Numerical Treatment of Differential Equations*, warns that even for quite simple examples, approximate methods can converge to false solutions. Equation stiffness, approximation errors, bad meshing, and poor user choice of initial conditions can all make this happen with FEM models. But as long as the results are subject to a 'reality check', the FEM is an outstandingly powerful technique. [22]

III. MAIN AIMS

The aim of this thesis is to propose and design a time-of-flight sensor in Femlab environment. Upon the model of the time-of-flight sensor the simulation of the heat phenomena in flowing gas will be performed. For the simulations various gases will be investigated and various low flow rates will be provided to obtain the most proper sensor parameters by comparison upon the results. The influence of radiation on the heat transfer in flowing gas will be examined.

The results presented in this work might be exploited when projecting and constructing a fluid flow meter that should be able to measure the volume flow rate of low flowing gas, record the value along with the measure time and transmit the data to a superior device to be processed and placed on record.

IV. EXPERIMENTAL PART

7 TIME-OF-FLIGHT SENSOR

The time-of-flight sensor consists of a heater and one or more temperature sensors downstream, Fig. 9. The heater is activated by a current pulse. The transport of the generated heat is a combination of diffusion and forced convection. The resulting temperature field can be detected by temperature sensors located downstream. The detected temperature output signal of the temperature sensor is a function of time and flow velocity. The sensor output is the time difference between the starting point of the generated heat pulse and the point in time at which a maximum temperature at the downstream sensor is reached. At the relatively low flow rates, the time difference depends mainly on the diffusivity of the fluid medium. At relatively high flow rates, the time difference tends to relate to the ratio of the heater–sensor distance and the average flow velocity.

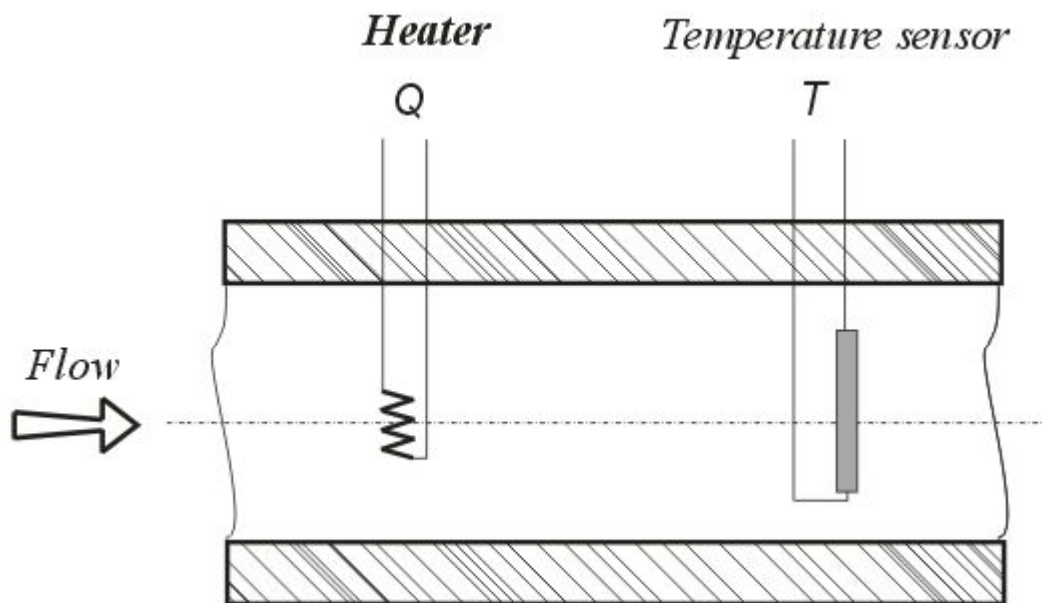


Fig. 9 Time-of-flight sensor

8 MODIFIED TIME-OF-FLIGHT SENSOR

The modified time-of-flight sensor consists of a heater placed on the pipe's floor and series of downstream sensors (semiconductor diodes) placed in the pipe's upper wall. The heater is activated for 1 second by a current pulse. The heat generated by the heater spreads with the flowing gas and leads into resistance changes on the diodes. The resistance change upon each diode is measured as a voltage change by an A/D converter. From the particular points along the pipe is set the wave's heat profile which is scanned for the temperature maximum position. The flow volume is proportional to the position shift in time.

The scheme drawing is shown in Fig. 10.

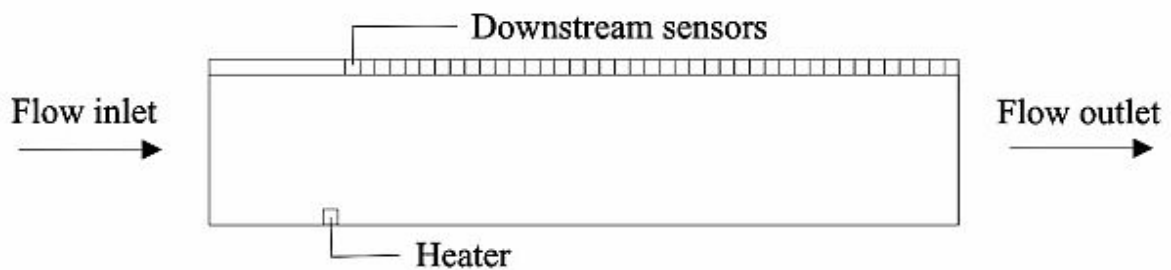


Fig. 10 Modified time-of-flight sensor

The heater is supposed to be an iron wire leaf, the semiconductor array is made of silicon. The whole pipe shall be put into a polystyrene casing to become thermally insulated from the ambient temperature conditions.

The length of the pipe body has been set as 5 mm. Height of the pipe is an unknown parameter to be examined.

Height of the semiconductor layer has been set as 1 μm .

Position of the heater has been set as 7.5 μm downstream of the inlet. Its size has been established as 1 $\mu\text{m} \times 1 \mu\text{m}$.

A block diagram illustrates the internal structure, Fig. 11.

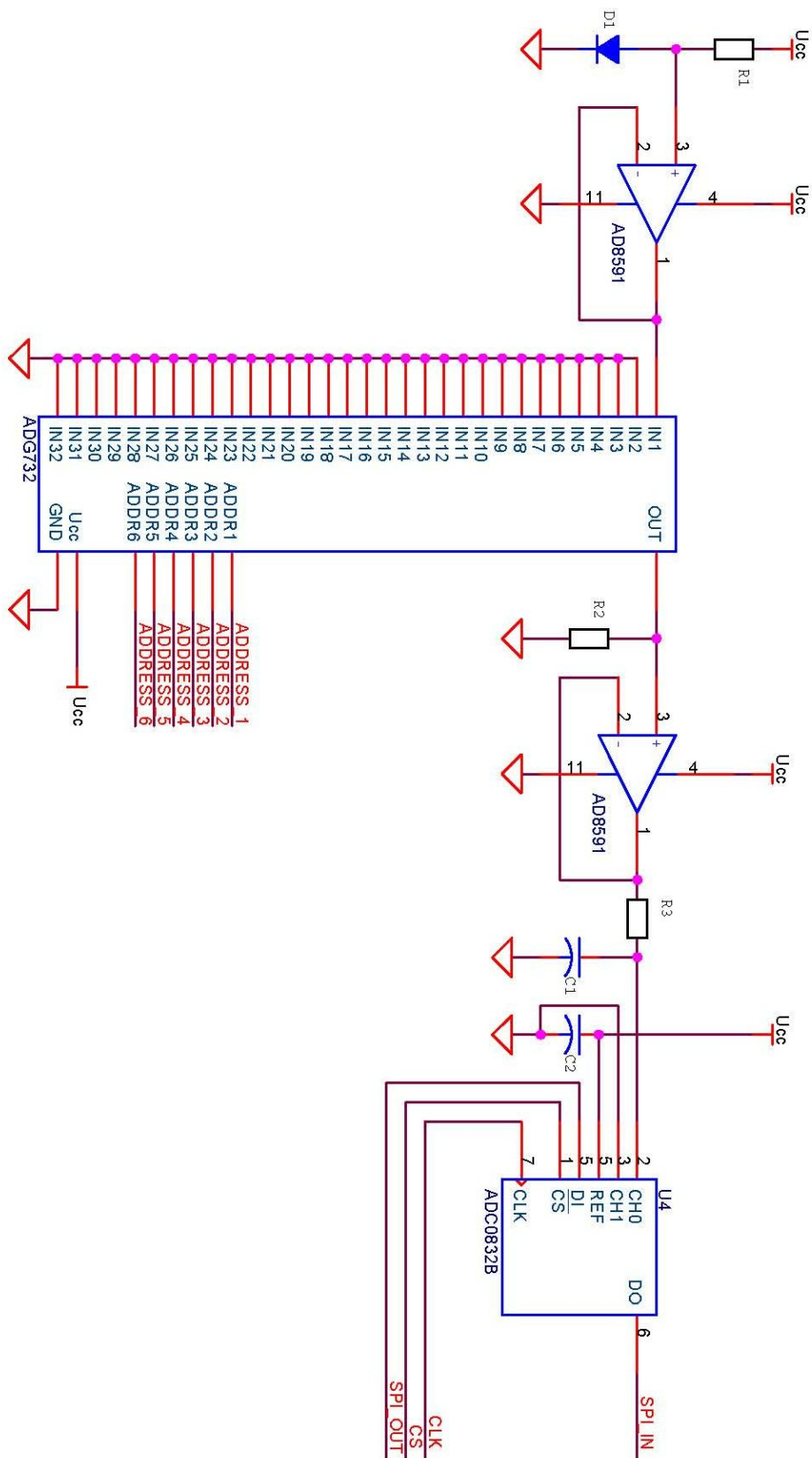


Fig. 11 Block diagram of the modified time-of-flight sensor

The **AD8591** is a single rail-to-rail input and output single supply amplifier featuring 250 mA output drive current and a power saving shutdown mode. The device includes a single master shutdown function that reduces total supply current to less than 1 mA. All amplifier outputs are in a high impedance state when in shutdown mode.

The amplifier has a very low input bias currents. Outputs are stable with virtually any capacitive load. Supply current is less than 750 mA per amplifier in active mode.

It is available in the tiny 6-lead SOT package. The pin configuration is shown in Fig. 12.

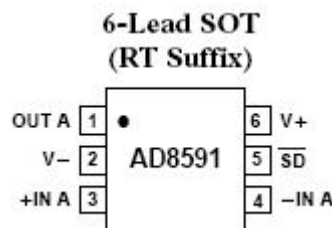


Fig. 12 Pin configuration of the AD8591

The **ADC0832B** device is an 8-bit successive-approximation analog-to-digital converter with multiplexed twin input channels. The serial output is configured to interface with standard shift registers or microprocessors.

The ADC0832 multiplexer is software configured for single-ended or differential inputs. The differential analog voltage input allows for common-mode rejection or offset of the analog zero input voltage value. In addition, the voltage reference input can be adjusted to allow encoding any smaller analog voltage span to the full 8 bits of resolution.

The pin configuration is shown in Fig. 13.

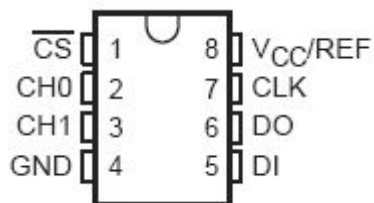


Fig. 13 Pin configuration of the ADC0832B

The **ADG732** is a monolithic CMOS 32-channel analog multiplexer. The device switches one of 32 inputs (S1-S32) to a common output, D, as determined by the 5-bit binary address lines A0, A1, A2, A3, and A4. The functional block diagram is shown in Fig. 14.

The multiplexer is designed on an enhanced submicron process that provides low power dissipation yet gives high switching speed, very low on resistance, and leakage currents. It operates from a single supply of +1.8 V to +5.5 V. On resistance is in the region of a few ohms and is closely matched between switches and very flat over the full signal range. The part can operate equally well as either multiplexer or demultiplexer and has an input signal range that extends to the supplies. In the OFF condition, signal levels up to the supplies are blocked. All channels exhibit break-before-make switching action, preventing momentary shorting when switching channels.

The pin configuration is shown in Fig. 15.

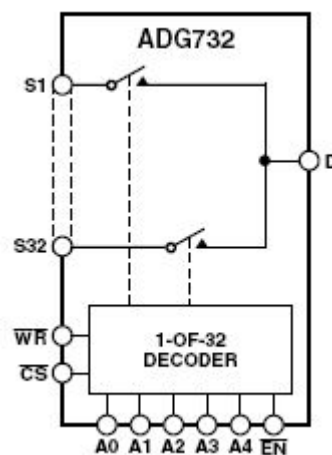


Fig. 14 Functional block diagram of the ADG732

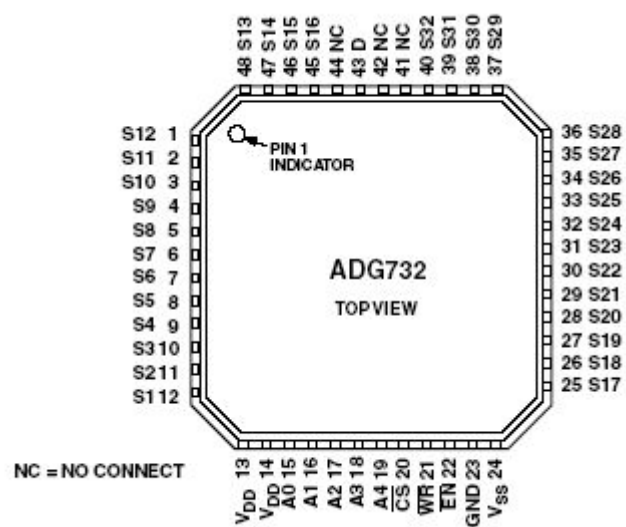


Fig. 15 Pin configuration of the ADG732

9 MATHEMATICAL MODEL

The mathematical model for heat transfer by conduction and convection is the nonconservative heat equation:

$$\delta_{ts} \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = Q - \rho C_p \mathbf{u} \cdot \nabla T \quad (20)$$

where T is temperature

δ_{ts} is a time-scaling coefficient

ρ is the fluid density

C_p is the heat capacity at a constant pressure

k is the thermal conductivity

$\mathbf{u}$ is the velocity field

∇ is the vector differential operator

Q is the heat source.

It assumes an incompressible fluid, which means that $\nabla \cdot \mathbf{u} = 0$. This ensures that no unphysical source term arises from a flow field where the incompressibility constraint, $\nabla \cdot \mathbf{u} = 0$, is not absolutely fulfilled. The nonconservative formulation puts the convective term on the right-hand side of the equation, which implies that setting the temperature gradient to zero directly expresses the convective boundary condition.

The incompressible Navier-Stokes equations from fluid dynamics works together with the heat transfer equation. There are four unknown field variables (dependent variables). The velocity field components, $\mathbf{u}$ and $\mathbf{v}$, the pressure, p and the temperature, T . They are all related through bidirectional multiphysics couplings. The incompressible Navier-Stokes equations (18) consist of a momentum balance (a vector equation) and a mass conservation and incompressibility condition.

The stress tensor is introduced to represent any force that the boundary exerts on the fluid.

The viscous stress tensor τ is defined as:

$$\tau = \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \quad (21)$$

where η is the dynamic viscosity.

In a fluid there can also be internal stresses due to the fluid pressure p in addition to the stress due to viscous forces. The total stress tensor σ is defined as:

$$\sigma = -p\mathbf{I} + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \quad (22)$$

where $\mathbf{I}$ is the identity matrix or unit diagonal matrix.

From these stress tensors, the viscous boundary force per unit area $\mathbf{K}$ is defined as:

$$\mathbf{K} = \boldsymbol{\tau} \mathbf{n} = \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \mathbf{n} \quad (23)$$

The total boundary force per unit area $\mathbf{T}$ is defined as:

$$\mathbf{T} = \sigma \mathbf{n} = (-p\mathbf{I} + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) \mathbf{n} \quad (24)$$

where $\mathbf{n}$ is the outward normal vector on the boundary.

The total stress tensor formulation of (18) is then:

$$\begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot [-p\mathbf{I} + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \rho(\mathbf{u} \cdot \nabla) \mathbf{u} &= \mathbf{F} \\ \nabla \cdot \mathbf{u} &= 0 \end{aligned} \quad (25)$$

where $\mathbf{F}$ is a volume force.

The mathematical model for heat transfer by conduction is the following version of the heat equation (2):

$$\delta_{ts} \rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k \nabla T) = Q \quad (26)$$

The radiation problem is then represented by adding the two terms on the right:

$$\delta_{ts} \rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k \nabla T) = Q + h_{trans} (T_{ext} - T) + C_{trans} (T_{ambtrans}^4 - T^4) \quad (27)$$

where h_{trans} is the transversal convective heat transfer coefficient

T_{ext} is the transversal external temperature

C_{trans} is a constant proportional to the Stefan-Boltzmann constant and emissivity

$T_{ambtrans}$ is the transversal ambient temperature.

To solve the equations (20), (25) and (27) the boundary and initial conditions must be provided. The boundary settings, initial conditions and parametric settings are discussed in the next chapter.

10 APPLICATION MODES IN FEMLAB SIMULATION

The model geometry can be separated into 3 areas, *subdomains*, as shown in Fig. 16.

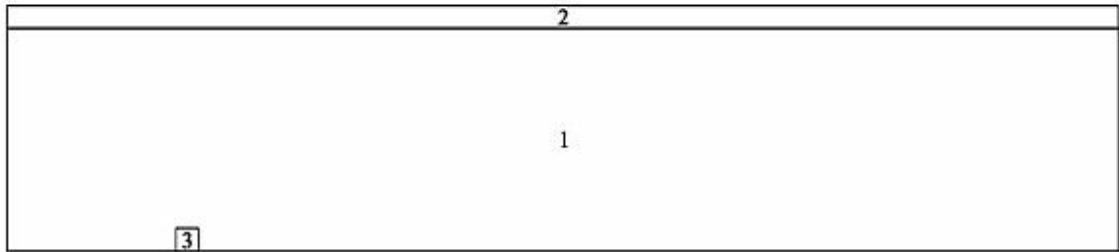


Fig. 16 Model subdomains

The model geometry has 12 *boundaries*, as shown in Fig. 17.

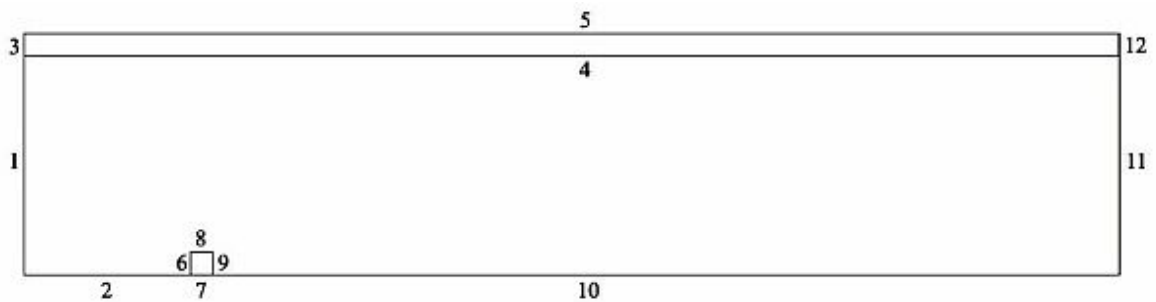


Fig. 17 Model boundaries

This is a multiphysics model because it involves more than one kind of physics.

The following application modes are employed:

- Convection and Conduction
- Incompressible Navier-Stokes
- Heat Transfer by Conduction.

The Femlab model is reduced to a 2D domain. There are presumably no variations in the third dimension, and it can be extrapolated from the solution of a related 2D case.

The requisite application modes are selected from the *Model Navigator*, Fig. 18.

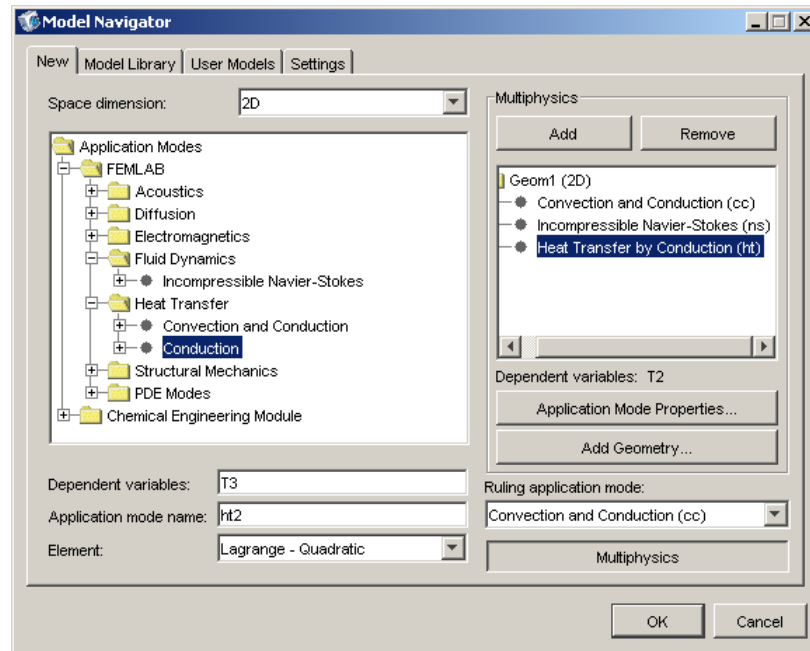


Fig. 18 Model navigator

10.1 Convection and Conduction

The settings of the Convection and Conduction application mode for the subdomain 1 are shown in Fig. 19.

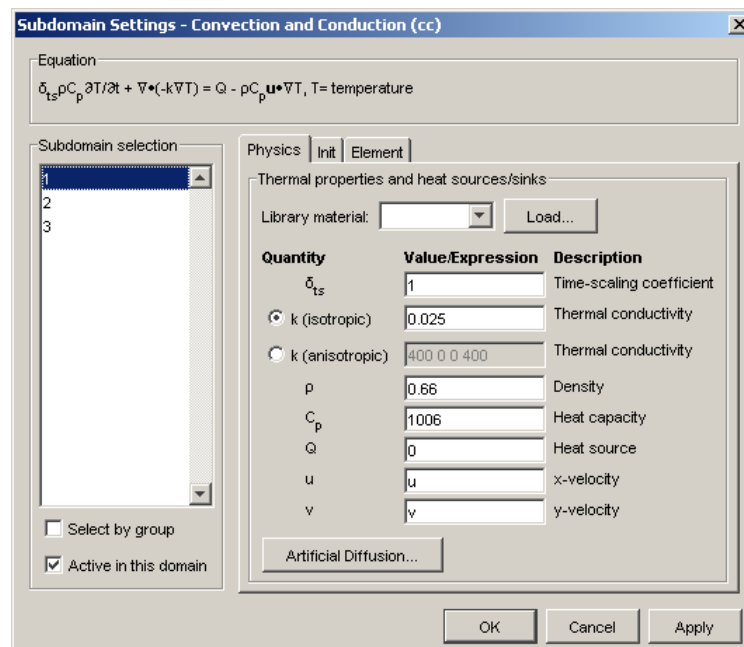


Fig. 19 Convection and Conduction – Subdomain settings – 1

The settings for the subdomain 2 are shown in Fig. 20.

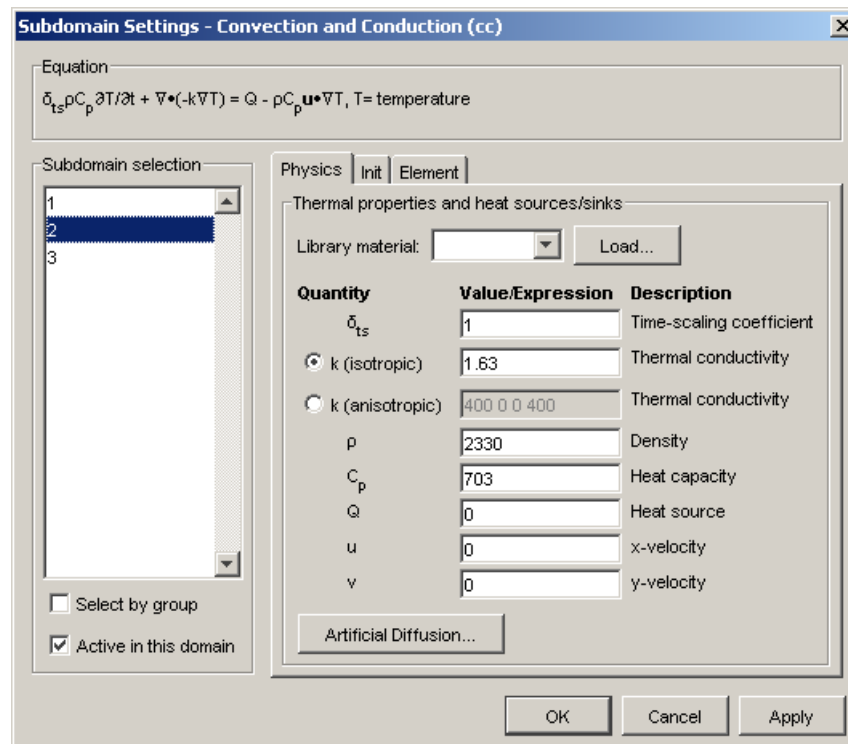


Fig. 20 Convection and Conduction – Subdomain settings – 2

The settings for the subdomain 3 are shown in Fig. 21.

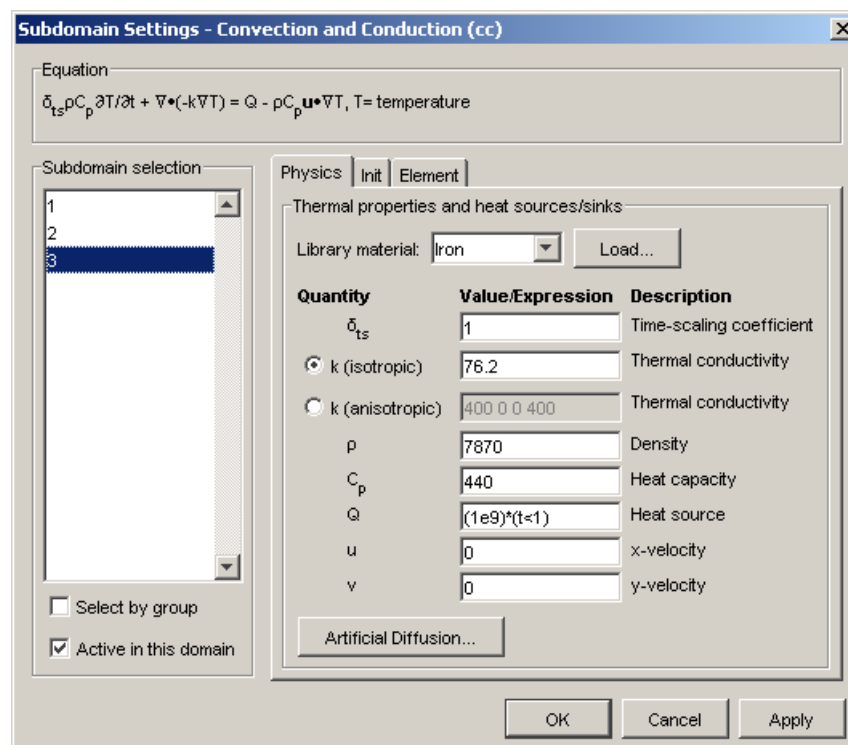


Fig. 21 Convection and Conduction – Subdomain settings – 3

The subdomain quantities are:

Time-scaling coefficient δ_{ts} . This coefficient is normally 1, but the time scale can be changed, for example, from seconds to minutes by setting it to 1/60.

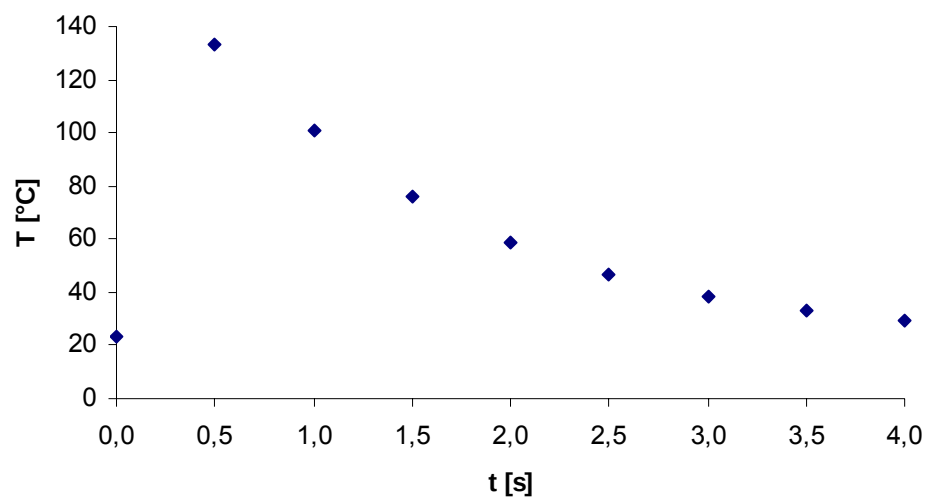
Thermal conductivity k . Thermal conductivity represents the unit power per unit length and unit temperature.

Density ρ . It specifies the material's density as unit mass per unit volume.

Heat capacity C . The heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass.

Heat source Q . The heat source describes heat generation within the domain. It represents the unit power per unit area. The heater generates heat in the time interval $\langle 0,1 \rangle$ seconds.

The temperature change of the heater in time is shown in Graph 1.



Graph 1 Temperature change of the heater in time

The temperature initial value has been set as $T_0 = 23\text{ °C}$ in all subdomains.

The settings of the Convection and Conduction application mode for the boundary 1 are shown in Fig. 22.

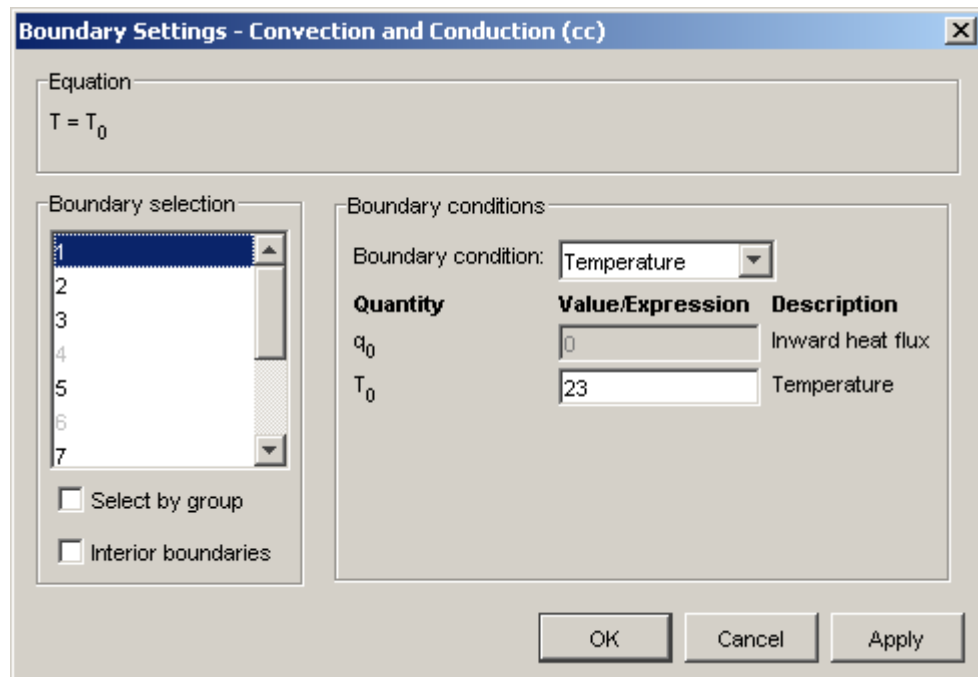


Fig. 22 Convection and Conduction – Boundary settings – 1

The settings for the boundaries 2,3,5,7,10,12 are shown in Fig. 23.

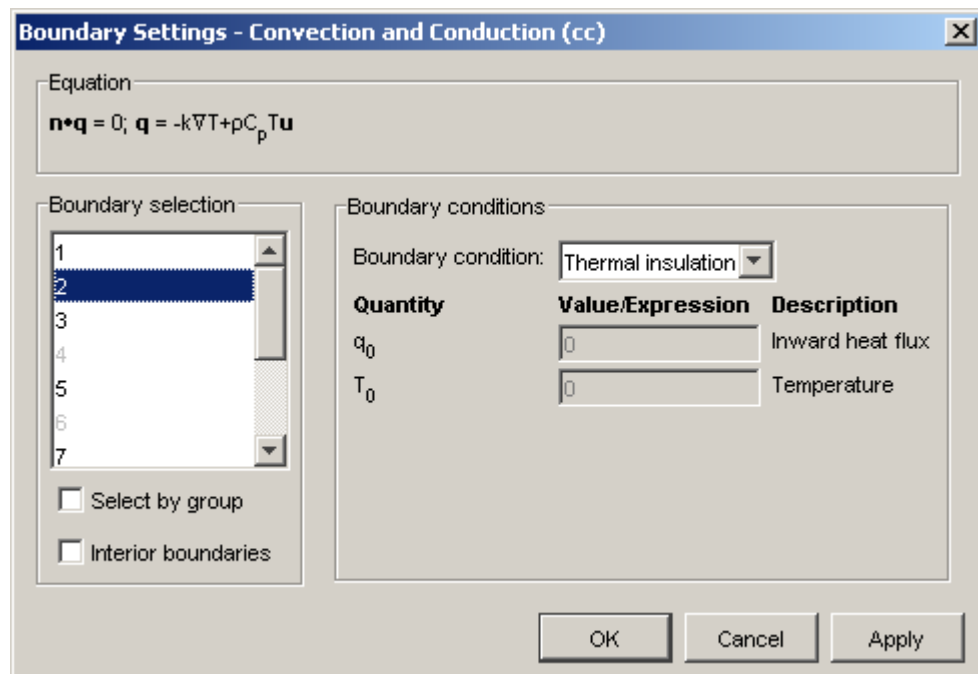


Fig. 23 Convection and Conduction – Boundary settings – 2,3,5,7,10,12

The settings for the boundary 11 are shown in Fig. 24.

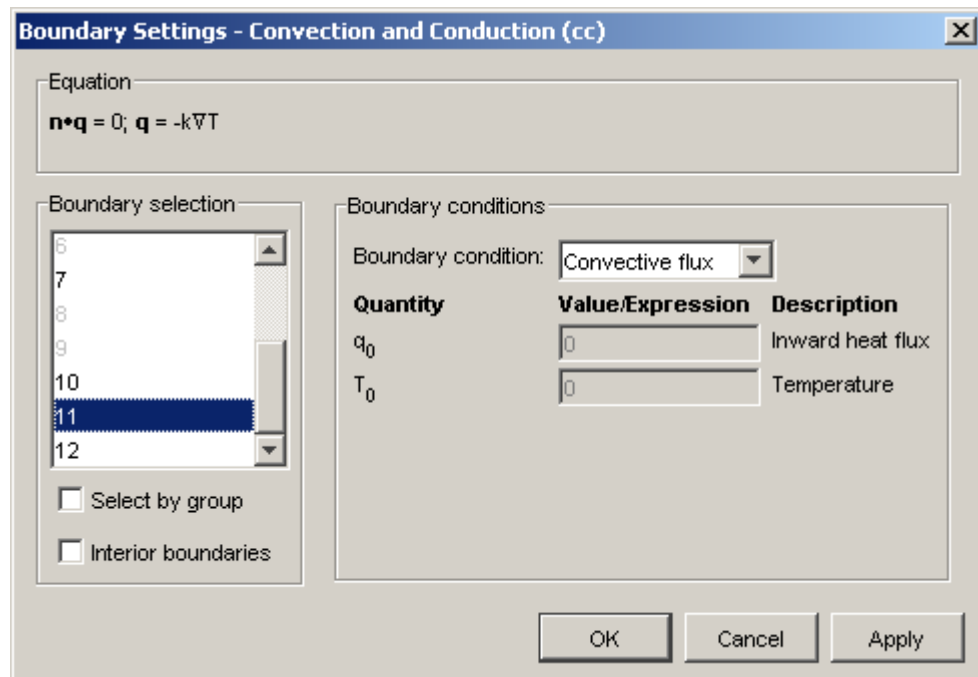


Fig. 24 Convection and Conduction – Boundary settings – 11

The boundaries 4,6,8,9 cannot be set. These boundaries are automatically computed for the heat transfer by conduction.

The boundary conditions are:

Temperature $T = T_0$. This boundary prescribes the temperature T_0 . This condition means that the finite element solution returns a solution in which the condition is either true or minimally approximated.

Thermal insulation $\mathbf{n} \cdot (-k\nabla T + \rho C_p \mathbf{u}T) = 0$. This condition specifies where the domain is well insulated. Intuitively this equation says that the gradient across the boundary must be zero. For this to be true, the temperature on one side of the boundary must equal the temperature on the other side. Because there is no temperature difference across the boundary, heat cannot transfer across it.

Convective flux $\mathbf{n} \cdot (-k\nabla T) = 0$. This condition is in cases where convection across a boundary is much greater than diffusion. It sets the diffusive flux at the boundary to zero, but it allows convective flux to exit the domain.

10.2 Incompressible Navier-Stokes

The settings of the Incompressible Navier-Stokes application mode for the subdomain 1 are shown in Fig. 25.

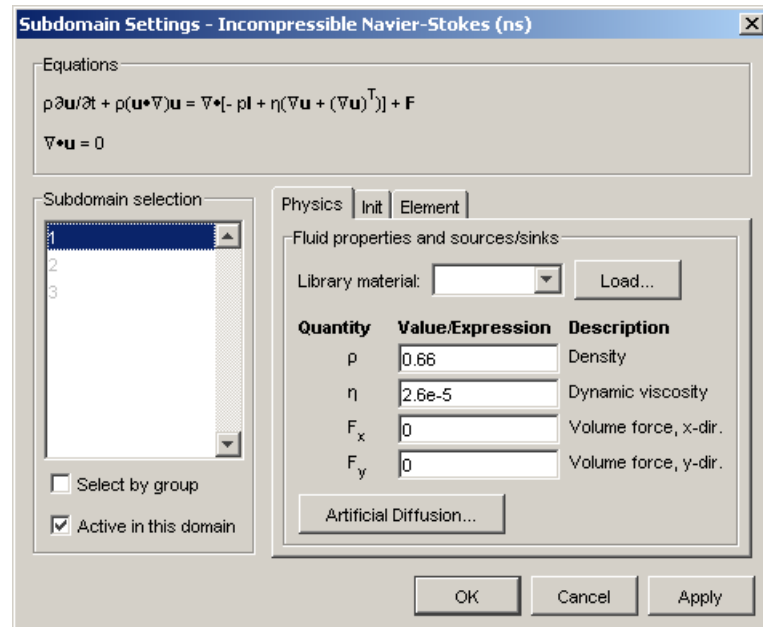


Fig. 25 Incompressible Navier-Stokes – Subdomain settings – 1

The subdomains 2 and 3 are inactive in this domain for there is no flow in them.

The initial value of the flow velocity is set in the *Init* tab of *Subdomain settings*, Fig. 26.

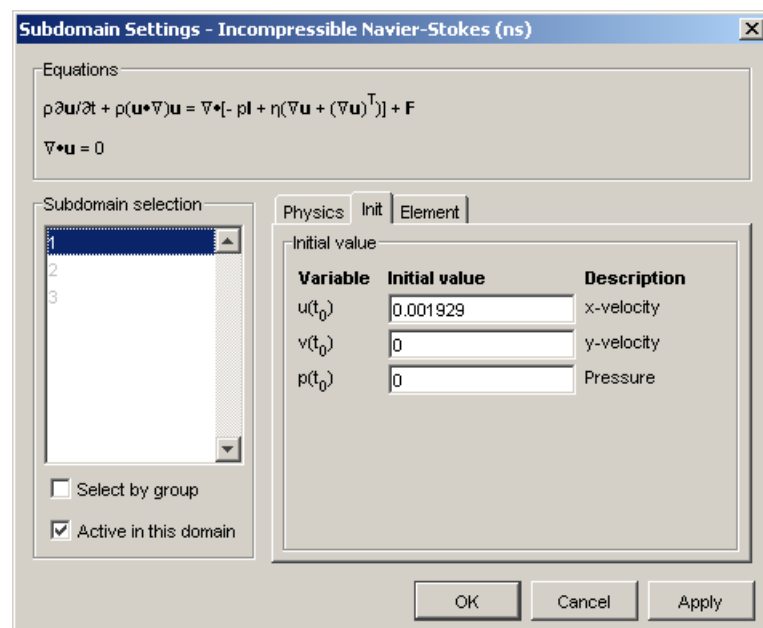


Fig. 26 Incompressible Navier-Stokes – Subdomain settings – Init tab

The subdomain quantities are:

Density ρ . This material property specifies the fluid density.

Dynamic Viscosity η . This term describes the relationship between the shear stresses in a fluid and the shear rate.

Volume Force $\mathbf{F}$. The volume force vector, $\mathbf{F} = (F_x, F_y, F_z)$, describes a distributed force field such as gravity. The unit of the volume force is force/area.

The settings of the Incompressible Navier-Stokes application mode for the boundary 1 are shown in Fig. 27.

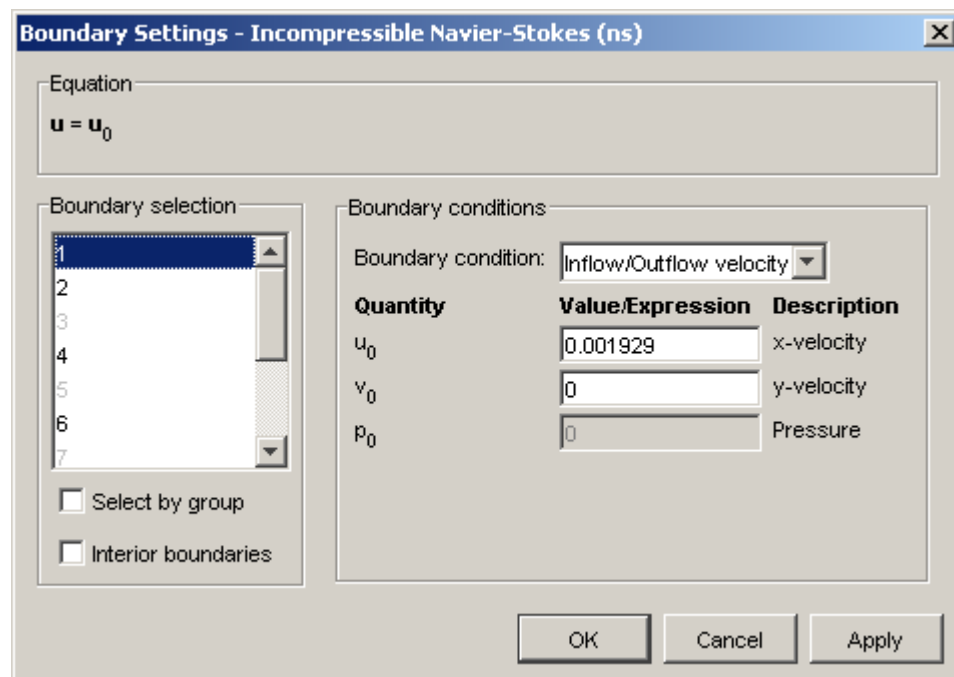


Fig. 27 Incompressible Navier-Stokes – Boundary settings – 1

The settings for the boundaries 2,4,6,8,9,10 are shown in Fig. 28.

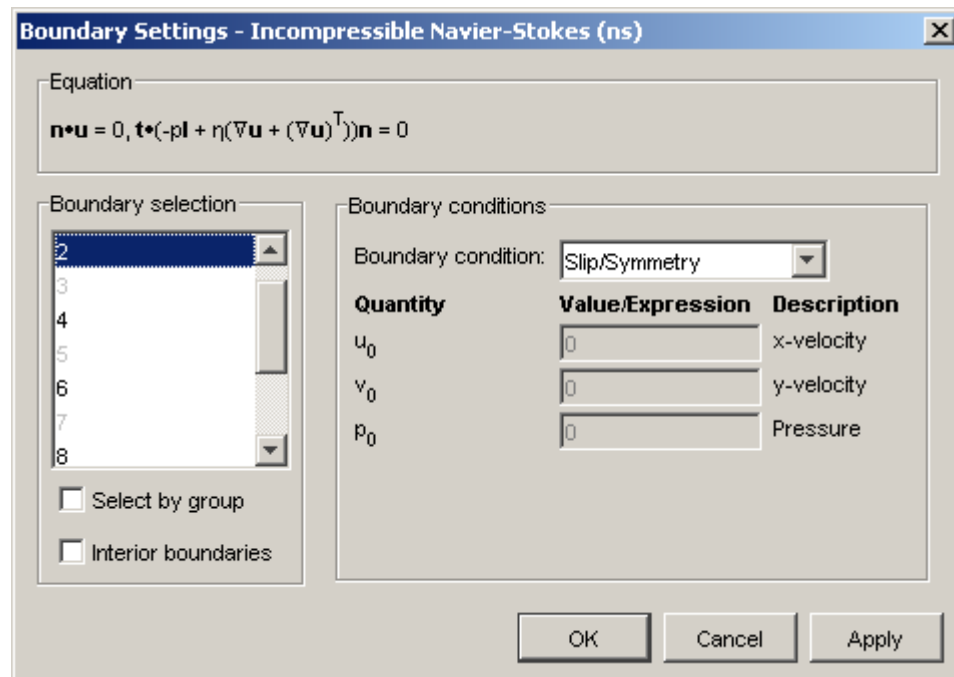


Fig. 28 Incompressible Navier-Stokes – Boundary settings – 2,4,6,8,9,10

The settings for the boundary 11 are shown in Fig. 29.

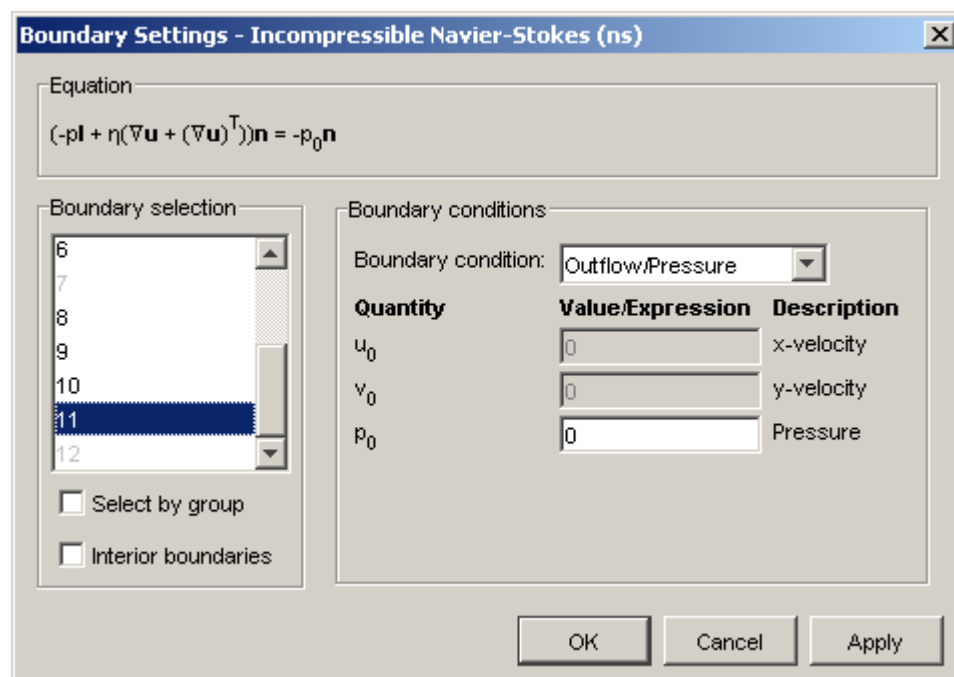


Fig. 29 Incompressible Navier-Stokes – Boundary settings – 11

The boundaries 3,5,7,12 cannot be set because they are in no contact with the fluid.

The boundary conditions for the Incompressible Navier-Stokes application mode are:

Inflow/Outflow velocity $\mathbf{u} = \mathbf{u}_0$. At an inflow or outflow boundary can be specified the fluid's velocity field as $\mathbf{u} = (u, v) = (u_0, v_0)$.

Slip/Symmetry $\mathbf{n} \cdot \mathbf{u} = 0$, $\mathbf{t} \cdot \mathbf{K} = 0$. A slip condition means that the normal component of the velocity is zero $\mathbf{n} \cdot \mathbf{u} = 0$ and that the tangential component of the viscous force vanishes $\mathbf{t} \cdot \mathbf{K} = 0$.

Outflow/Pressure $\mathbf{T} = -p_0 \mathbf{n}$. Using the total stress tensor form, this boundary condition states that the total force on the boundary is a pressure force defined by p_0 : $\mathbf{T} = -p_0 \mathbf{n}$.

Mesh parameters for the solver has been set as shown in Fig. 30.

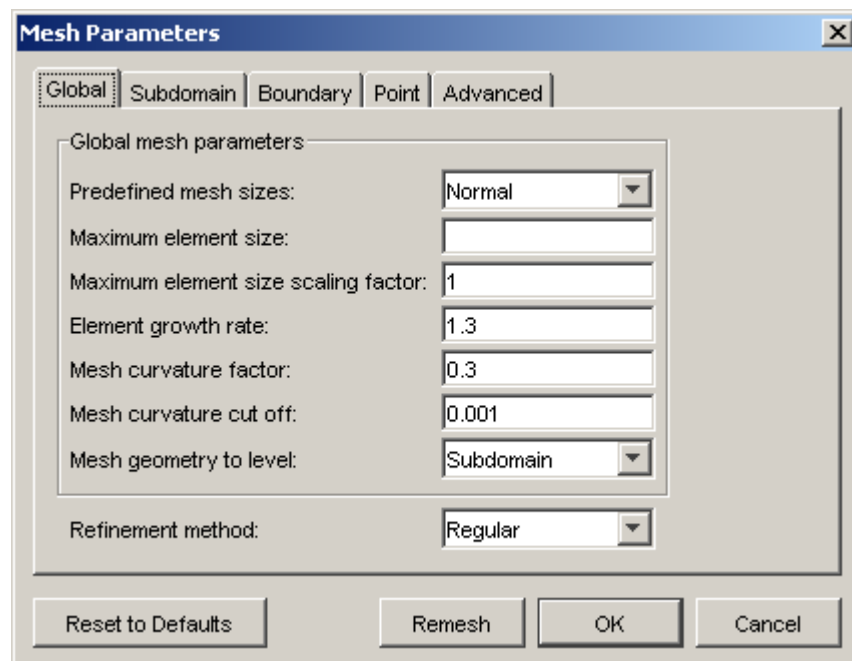


Fig. 30 Mesh parameters

The time-of-flight sensor model is solved as a Time dependent problem. The settings for the Solver parameters are shown in Fig. 31.

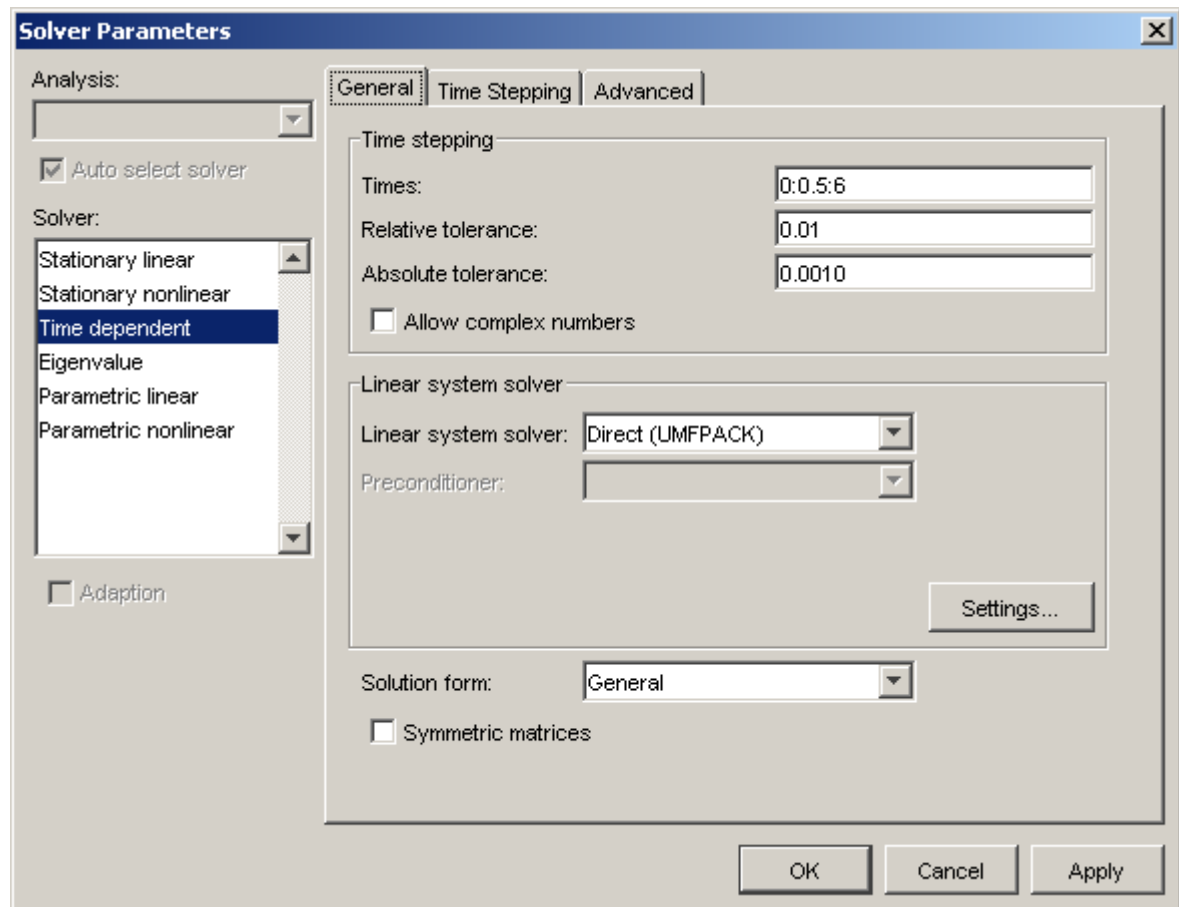


Fig. 31 Solver parameters

10.3 Heat Transfer by Conduction

This application mode models heat transfer by conduction but also includes convection and radiation effects around edges and boundaries.

Emissivity $\varepsilon = 0.64$ and the Stefan-Boltzmann constant $\sigma = 5.67 \times 10^{-8}$ have been entered to the *Constants* box from the *Options* menu.

The settings of the Heat Transfer by Conduction application mode for the subdomain 1 are shown in Fig. 32.

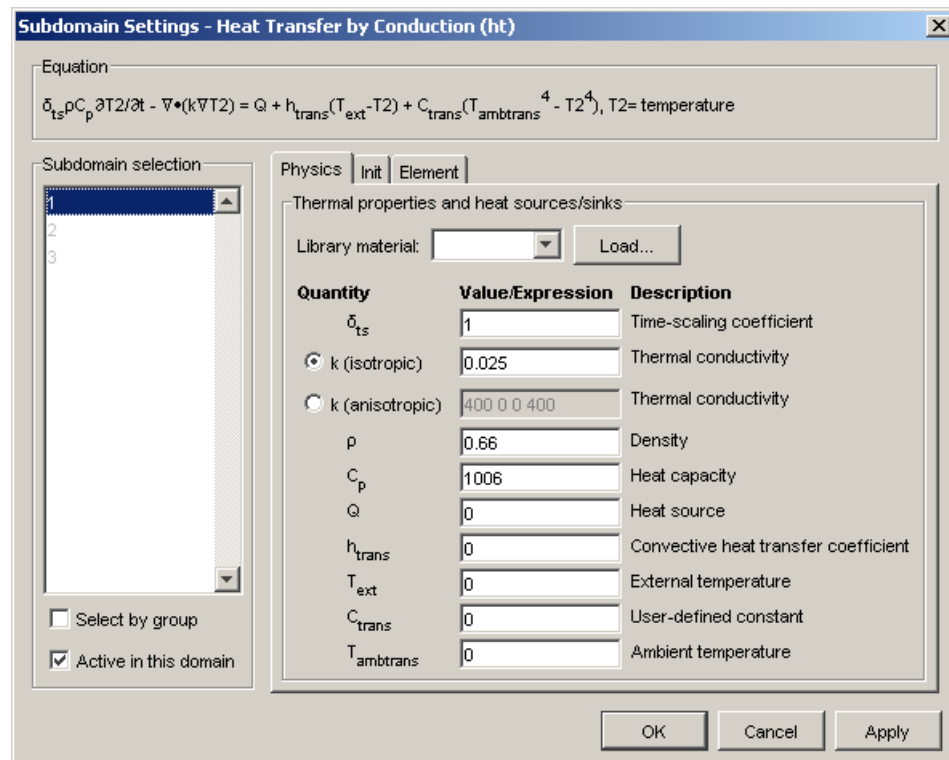


Fig. 32 Heat Transfer by Conduction – Subdomain settings – 1

The subdomains 2 and 3 are inactive in this domain.

The temperature initial value has been set as $T_0 = 23\text{ }^{\circ}\text{C}$.

The settings for the boundaries 1,4,11 are shown in Fig. 33.

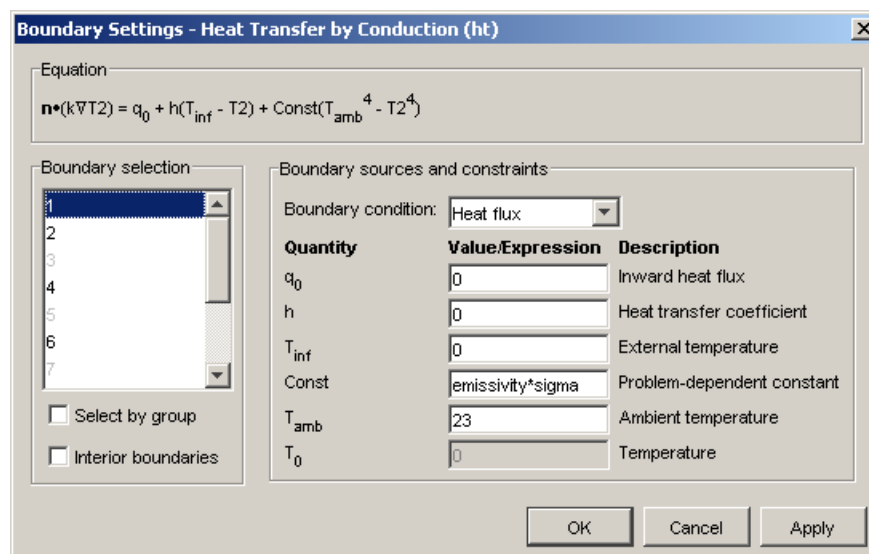


Fig. 33 Heat Transfer by Conduction – Boundary settings – 1,4,11

The settings for the boundaries 2 and 10 are shown in Fig. 34.

The dialog box 'Boundary Settings - Heat Transfer by Conduction (ht)' shows the following settings:

- Equation:** $\mathbf{n} \cdot (k \nabla T_2) = 0$
- Boundary selection:** A list of boundaries 1 through 7. Boundary 2 is selected. Below the list are checkboxes for 'Select by group' and 'Interior boundaries', both of which are unchecked.
- Boundary sources and constraints:**
 - Boundary condition:** Thermal insulation
 - Quantity** | **Value/Expression** | **Description**

q_0	0	Inward heat flux
h	0	Heat transfer coefficient
T_{inf}	0	External temperature
Const	0	Problem-dependent constant
T_{amb}	0	Ambient temperature
T_0	0	Temperature

Buttons at the bottom: OK, Cancel, Apply.

Fig. 34 Heat Transfer by Conduction – Boundary settings – 2,10

The settings for the boundaries 6,8,9 are shown in Fig. 35. These are the boundaries of the heater which is inactive as a subdomain. To follow the generated heat, the approximated temperature function $\frac{100}{t}$ has been provided as the initial temperature estimation.

The dialog box 'Boundary Settings - Heat Transfer by Conduction (ht)' shows the following settings:

- Equation:** $T_2 = T_0$
- Boundary selection:** A list of boundaries 6 through 12. Boundary 6 is selected. Below the list are checkboxes for 'Select by group' and 'Interior boundaries', both of which are unchecked.
- Boundary sources and constraints:**
 - Boundary condition:** Temperature
 - Quantity** | **Value/Expression** | **Description**

q_0	0	Inward heat flux
h	0	Heat transfer coefficient
T_{inf}	0	External temperature
Const	0	Problem-dependent constant
T_{amb}	0	Ambient temperature
T_0	$100/t$	Temperature

Buttons at the bottom: OK, Cancel, Apply.

Fig. 35 Heat Transfer by Conduction – Boundary settings – 6,8,9

The boundaries 3,5,7,12 cannot be set for there is no radiation heat transfer across them (the subdomains 2 and 3 are inactive).

The boundary conditions are:

Heat flux $\mathbf{n} \cdot (k \nabla T) = q_0 + h(T_{\text{inf}} - T) + C_{\text{const}}(T_{\text{amb}}^4 - T^4)$. The application mode interprets the heat flux q_0 in the direction of the inward normal. It interprets the radiation terms in the direction of the outward normal. $h(T_{\text{inf}} - T)$ models convective heat transfer with the surrounding environment, where h is the heat transfer coefficient and T_{inf} is the ambient bulk temperature. $C_{\text{const}}(T_{\text{amb}}^4 - T^4)$ models radiation heat transfer with the surrounding environment. T_{amb} is the temperature of the surrounding radiation environment, which might differ from T_{inf} . C_{const} is the product of the surface emissivity ε and the Stefan-Boltzmann constant σ : $C_{\text{const}} = \varepsilon \sigma$.

Thermal insulation $\mathbf{n} \cdot (k \nabla T) = 0$. This condition specifies where the domain is well insulated. Intuitively this equation says that the gradient across the boundary must be zero. For this to be true, the temperature on one side of the boundary must equal the temperature on the other side. Because there is no temperature difference across the boundary, heat cannot transfer across it.

Temperature $T = T_0$. This boundary prescribes the temperature T_0 . This condition means that the finite element solution returns a solution in which the condition is either true or minimally approximated.

Time stepping in the *Solver Parameters* dialog box has been set to 0.5:0.5:6 because time could not be zero when the prescribed temperature is a function $\frac{100}{t}$. Therefore the process starts in the time $t = 0.5$ and also the heater has been set with 0.5 second delay to supply heat in the time interval $\langle 0.5, 1.5 \rangle$ seconds.

11 MODELS

The gas constants used for the simulations are included in the following table:

Tab. 4 Gas constants

Gas	Density [kg.m ⁻³]	Thermal conductivity [W.m ⁻¹ .K ⁻¹]	Heat capacity [J.kg ⁻¹ .K ⁻¹]	Dynamic viscosity [Pa.s]
O ₂	0.66	0.025	1006	1.60×10 ⁻⁵
CO ₂	1.98	0.016	837	1.48×10 ⁻⁵
N ₂	1.25	0.026	1034	1.75×10 ⁻⁵

The material constants used for the simulations are included in the Tab. 5.

Tab. 5 Material constants

Material	Density [kg.m ⁻³]	Thermal conductivity [W.m ⁻¹ .K ⁻¹]	Heat capacity [J.kg ⁻¹ .K ⁻¹]
Silicon	2330	1.63	703
Iron	7870	76.2	440

The Tab. 6, Tab. 7 and Tab. 8 show the gas flow volumes Q and velocities v for various pipe heights. All sizes in the range <0.1,2.2> mm with the 0.1 mm difference were tested. The heights below 0.1 mm are too thin for the measurement (the flow velocity is high and flushes away the generated heat), the heights above 2.2 mm produce low temperature divergence in the semiconductor array. 10 simulations corresponding to the flow volume range <10,100> ml/h for each pipe height were performed. In each simulation, the maximum temperature value was determined from the temperature wave in the semiconductor domain. The illustration of a temperature wave acquired from the cross-section plot of the subdomain 2 is shown in Appendix I. The temperature capture proceeded in 4 seconds after the heater activation (4.5 seconds in the radiation models). The obtained values of the temperature maximum were diagrammed as a function of flow velocity and the plots revealed the most applicable proportions.

Tab. 6 Gas flow volumes and velocities according to the pipe height 0.3 mm

Q [ml. h ⁻¹]	10	20	30	40	50	60	70	80	90	100
v [m.s ⁻¹]	0.0309	0.0617	0.0926	0.1235	0.1543	0.1852	0.2160	0.2469	0.2778	0.3086

Tab. 7 Gas flow volumes and velocities according to the pipe height 1.2 mm

Q [ml. h ⁻¹]	10	20	30	40	50	60	70	80	90	100
v [m.s ⁻¹]	0.0019	0.0039	0.0058	0.0077	0.0096	0.0116	0.0135	0.0154	0.0174	0.0193

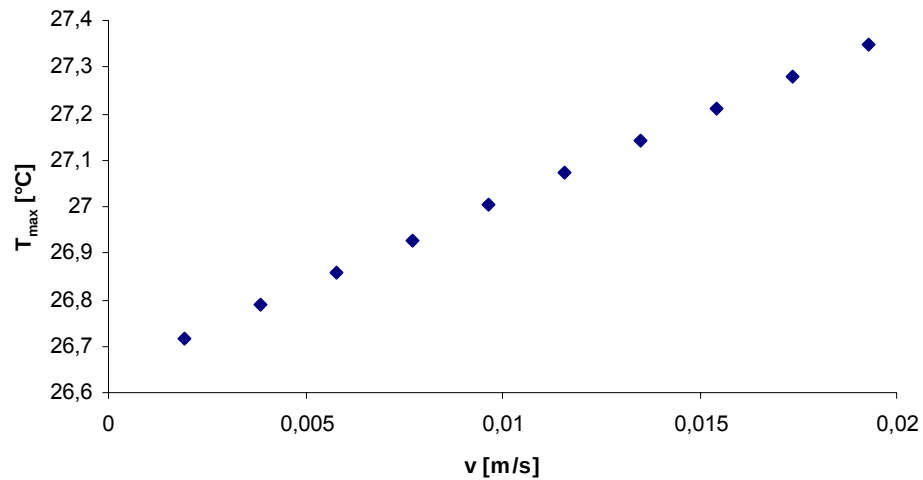
Tab. 8 Gas flow volumes and velocities according to the pipe height 2.2 mm

Q [ml. h ⁻¹]	10	20	30	40	50	60	70	80	90	100
v [m.s ⁻¹]	0.0006	0.0011	0.0017	0.0023	0.0029	0.0034	0.0040	0.0046	0.0052	0.0057

The illustration of a temperature field in the pipe is shown in Appendix II. Femlab also provides visualization of other solution properties, the illustration of the x-velocity field in the pipe is shown in Appendix III.

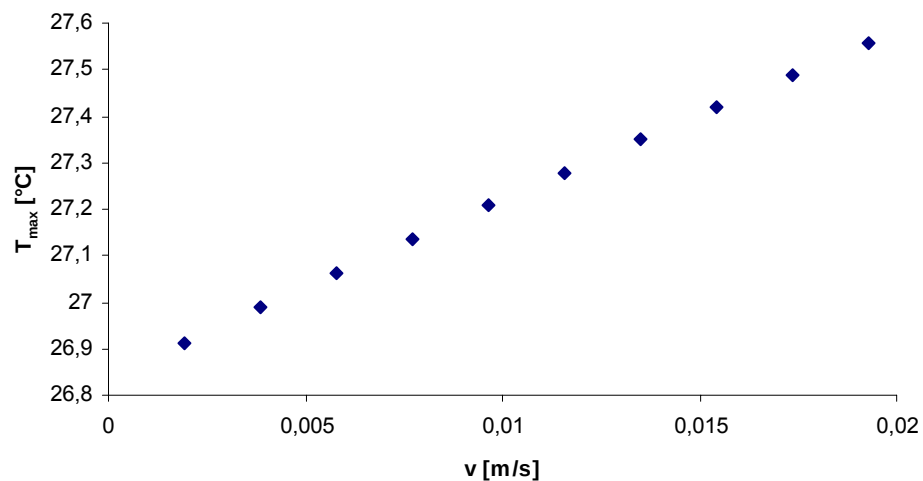
11.1 Air models

The Graph 2 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 1.2 mm.



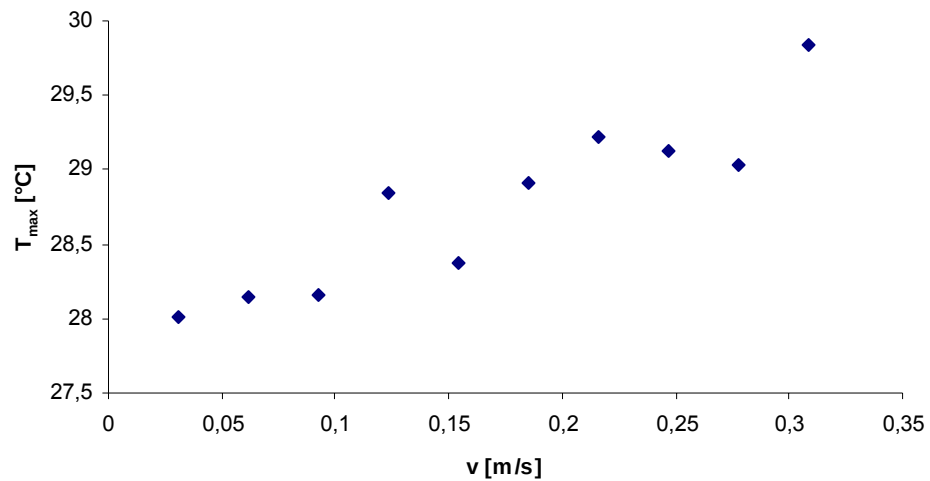
Graph 2 Maximum temperature as a function of velocity – air, 1.2 mm

The Graph 3 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 1.2 mm in a model with radiation.



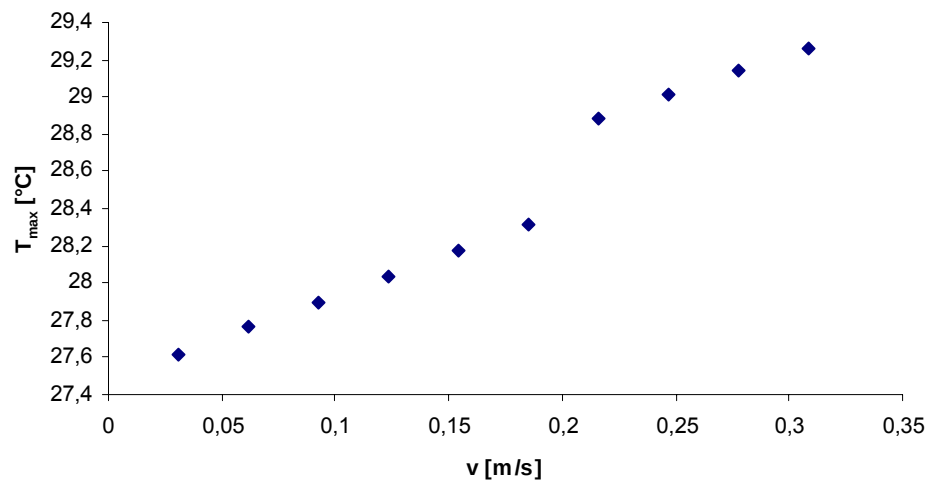
Graph 3 Maximum temperature as a function of velocity – air, 1.2 mm, radiation

The Graph 4 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 0.3 mm.



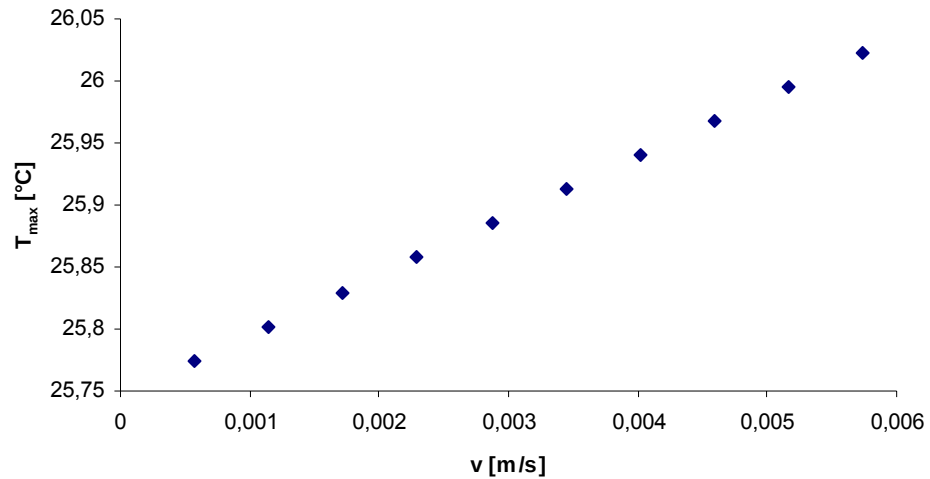
Graph 4 Maximum temperature as a function of velocity – air, 0.3 mm

The Graph 5 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 0.3 mm in a model with radiation.



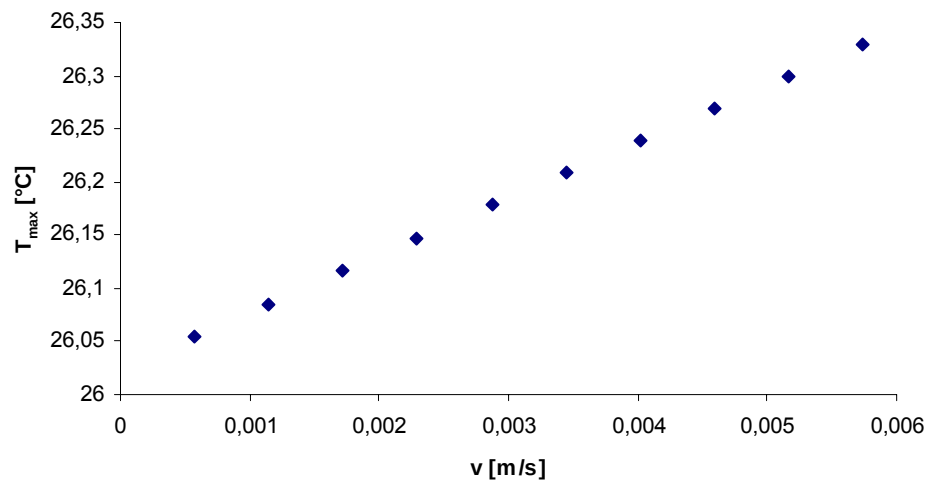
Graph 5 Maximum temperature as a function of velocity – air, 0.3 mm, radiation

The Graph 6 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 2.2 mm.



Graph 6 Maximum temperature as a function of velocity – air, 2.2 mm

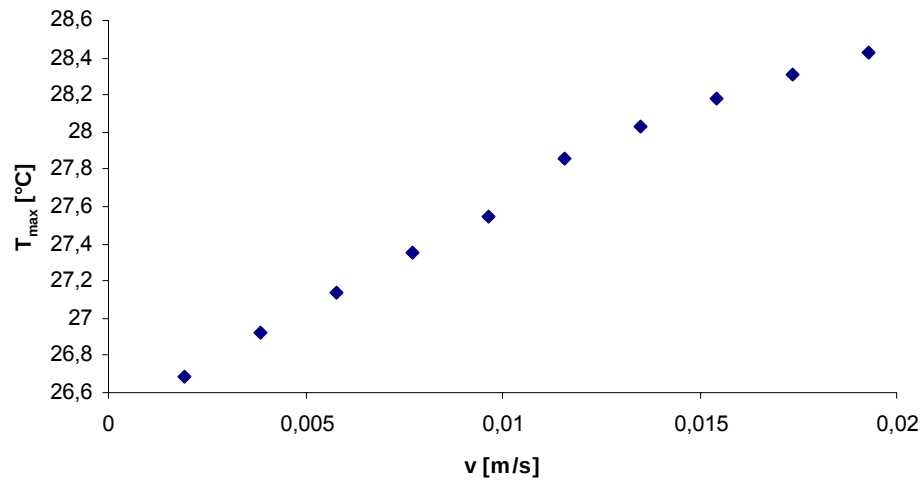
The Graph 7 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 2.2 mm in a model with radiation.



Graph 7 Maximum temperature as a function of velocity – air, 2.2 mm, radiation

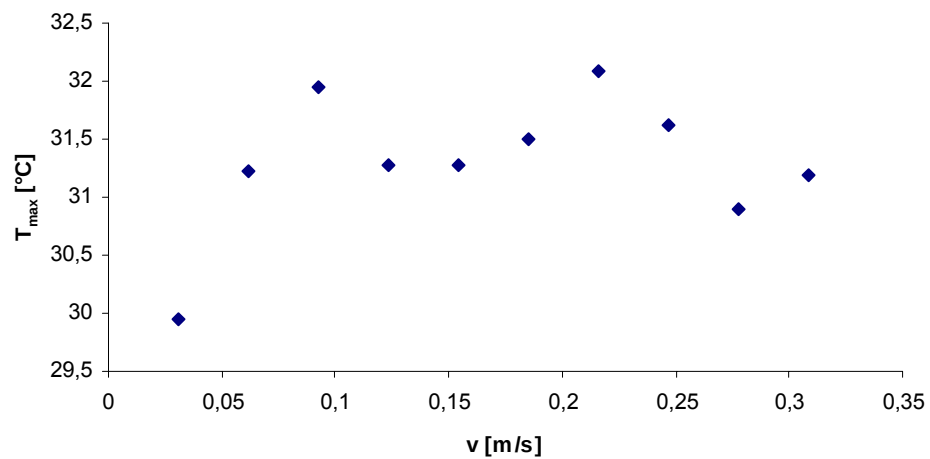
11.2 CO₂ models

The Graph 8 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 1.2 mm in a model with radiation.



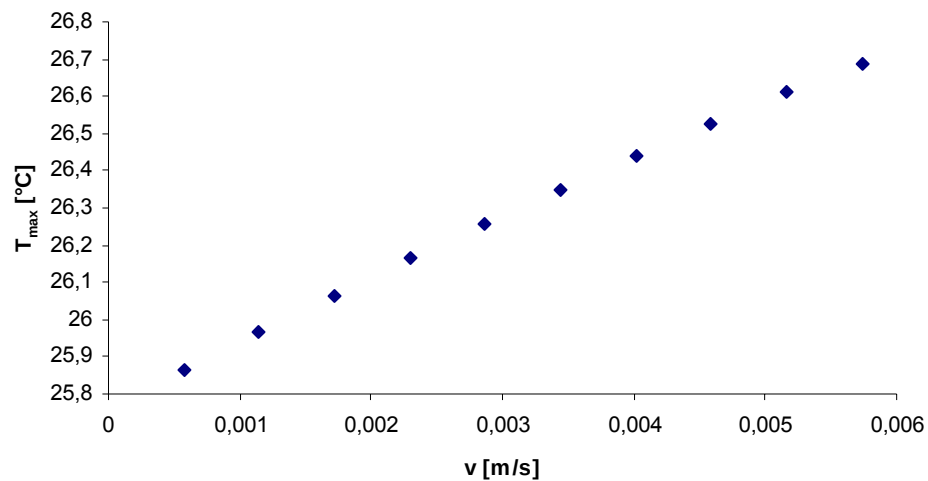
Graph 8 Maximum temperature as a function of velocity – CO₂, 1.2 mm, radiation

The Graph 9 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 0.3 mm in a model with radiation.



Graph 9 Maximum temperature as a function of velocity – CO₂, 0.3 mm, radiation

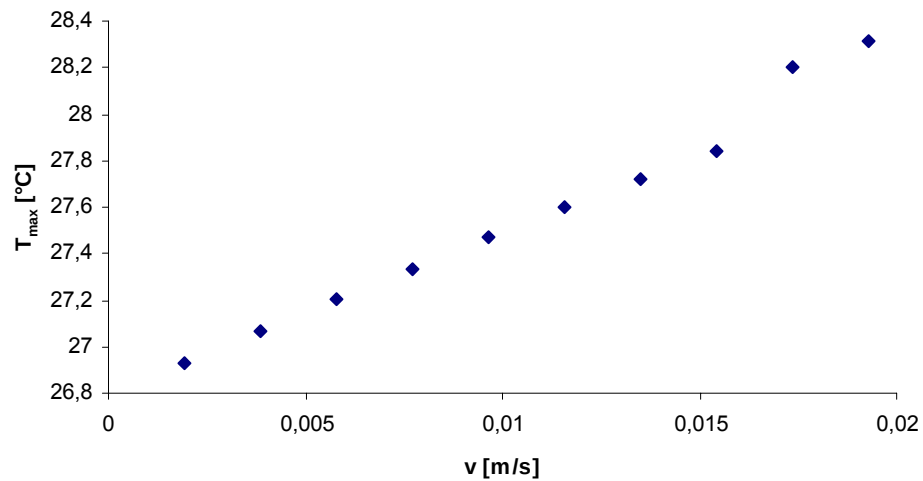
The Graph 10 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 2.2 mm in a model with radiation.



Graph 10 Maximum temperature as a function of velocity – CO_2 , 2.2 mm, radiation

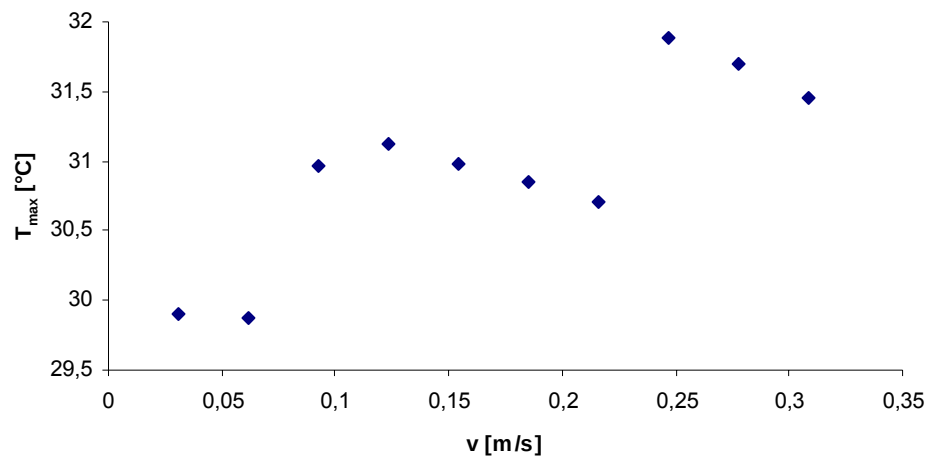
11.3 N_2 models

The Graph 11 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 1.2 mm in a model with radiation.



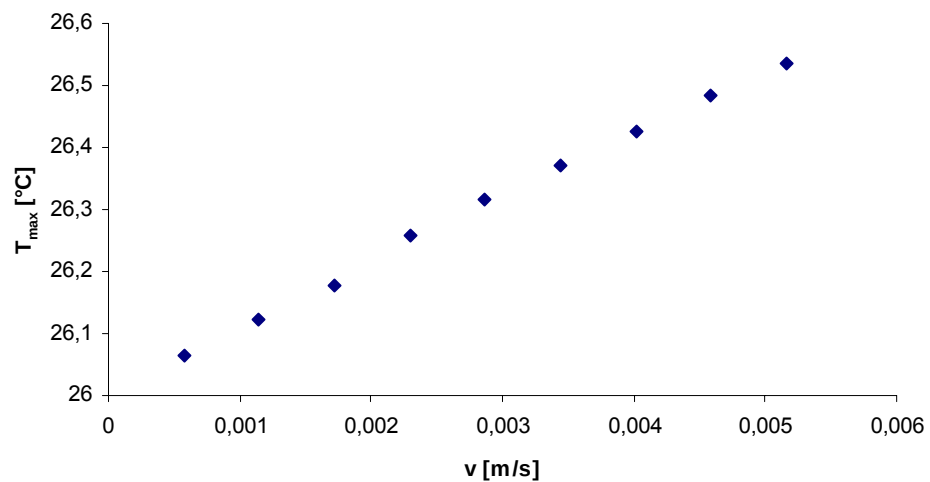
Graph 11 Maximum temperature as a function of velocity – N_2 , 1.2 mm, radiation

The Graph 12 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 0.3 mm in a model with radiation.



Graph 12 Maximum temperature as a function of velocity – N_2 , 0.3 mm, radiation

The Graph 13 shows the maximum temperature T_{max} as a function of flow velocity v for the pipe height 2.2 mm in a model with radiation.



Graph 13 Maximum temperature as a function of velocity – N_2 , 2.2 mm, radiation

11.4 Step change model

A model with a step change of flow velocity was conceived. The medium is air, radiation section is included. To simulate the step change, the function $(t > 3.5) * v_1 + v_2$ was entered to the Incompressible Navier-Stokes mode x-velocity initial value fields, both to the

Subdomain and Boundary settings. It means that the first 3 seconds (the process starts in time $t = 0.5$ second) the fluid flows with the velocity v_2 and in time $t = 3.5$ seconds the flow velocity increases to the velocity $v_1 + v_2$ (the indexes comes from Femlab and they cannot be commuted). The step change corresponds to the flow volume increace from 10 ml/h to 50 ml/h. The maximum temperature value was captured in time $t = 4.5$ seconds. For the pipe height 1.2 mm the comparison between the prime and the step change model is shown in Tab. 9.

Tab. 9 Step change model – 1.2 mm

Q [ml. h ⁻¹]	v [m.s ⁻¹]	T_{max} [°C]
10	0.0019	26.914
10 to 50	0.0019 to 0.0096	26.940

The comparison between the prime and the step change model for the pipe height 2.2 mm is shown in Tab. 10.

Tab. 10 Step change model – 2.2 mm

Q [ml. h ⁻¹]	v [m.s ⁻¹]	T_{max} [°C]
10	0.0006	26.054
10 to 50	0.0006 to 0.0029	26.074

CONCLUSION

The main objective of this work was the modeling of heat transfer in flowing gas.

A modified time-of-flight sensor was proposed to measure low flow rates in the range of 10 to 100 ml/h.

The well-known Femlab modeling toolkit was used for the simulations. The results of the simulations revealed the most applicable proportions of the sensor pipe. Its height is advised to be 1.2 mm with length 5 mm, heater $1\text{ }\mu\text{m} \times 1\text{ }\mu\text{m}$ shall be located $7.5\text{ }\mu\text{m}$ downstream of the inlet, height of the semiconductor array might be $1\text{ }\mu\text{m}$. A thinner pipe results into higher flow volumes that carries away the generated heat. A thick pipe results into unmeasurably low temperature divergence.

The heat spreads easier with CO_2 and N_2 , the air is more resistive. The CO_2 and N_2 models embody higher values of temperature maximum and larger temperature divergence.

The radiation effect added to the convection and conduction causes greater heat transfer in the temperature field which results into difference in tenths of degree Celsius from the model with convection and conduction only. The temperature divergence is identical in both.

The step change models showed that one second after the flow volume step increase from 10 ml/h to 50 ml/h the temperature increased by two hundredths of degree Celsius.

Basically, the simulation is a process of imitating a real phenomenon with a set of mathematical formulas. It is to estimate the true characteristics of the system, where the most problematic parts involve boundary settings and the physical parameters.

The further work might consider including a simulation of the proposed time-of-flight sensor in other software product listed in this thesis to be compared with the Femlab models.

The most efficient would be to set up an approximative physical laboratory model to verify its utility and accuracy.

BIBLIOGRAPHY

- [1] WEBSTER, J.: *The measurement, instrumentation and sensors. Handbook*, USA, 1999. ISBN 3-540-64830-5.
- [2] LAMMERIMK, T., NIELS, R.: *Micro/liquid flow sensors. Sensors and Actuators*, 37/38, 1993. ISSN 0924-4247.
- [3] ĎAĎO, S., KREIDL, M.: *Senzory a měřicí obvody*. ČVUT, Praha, 1999. ISBN 80-01-02057-6.
- [4] HEJZLAR, R.: *Sdílení tepla*. ČVUT, Praha, 1993. ISBN 80-01-01011-2.
- [5] KUNEŠ, J.: *Modelování tepelných procesů*. SNTL, Praha, 1989. ISBN 80-03-00134-X.
- [6] KUTATELADZE, S., BOROŠANSKIJ, V.: *Příručka sdílení tepla*. SNTL, Praha, 1962.
- [7] MÍKA, V., NEUŽIL, L., VLČEK, J.: *Chemicko-inženýrské výpočty*. VŠCHT, Praha, 1996. ISBN 80-7080-255-3.
- [8] NEUŽIL, L., MÍKA, V.: *Chemické inženýrství IB*. VŠCHT, Praha, 1992. ISBN 80-7080-164-6.
- [9] BAFRNEC, M., BÁLEŠ, V., LANGFELDER, I., LONGAUER, J.: *Chemické inženýrstvo I*. MALÉ CENTRUM, Bratislava, 1999. ISBN 80-967064-3-8.
- [10] Documentation manual for FEMLAB 3.1. COMSOL AB, Stockholm, 2004. <<http://www.femlab.com>>.
- [11] *Heat Transfer: Overview* [online]. c2005 [cit. 2005-05-15]. Available from WWW: <http://www.efunda.com/formulae/heat_transfer/home/overview.cfm>.
- [12] ELERT, Glenn. *Radiation* [online]. 1998-2005 [cit. 2005-05-20]. Available from WWW: <<http://hypertextbook.com/physics/thermal/radiation/>>.
- [13] *Fluid Mechanics: Navier-Stokes* [online]. c2005 [cit. 2005-05-23]. Available from WWW: <http://www.efunda.com/formulae/fluids/navier_stokes.cfm>.
- [14] *Algor* [online]. 2005 [cit. 2005-06-02]. Available from WWW: <<http://www.algor.com/products/Profes1751/default.asp>>.

- [15] ANSYS [online]. 2005 [cit. 2005-06-02]. Available from WWW: <<http://www.ansys.com/products/multiphysics.asp>>.
- [16] Cullimore & Ring Technologies [online]. c1999-2005 [cit. 2005-06-02]. Available from WWW: <<http://www.crtech.com/sinda.html>>.
- [17] Fluent [online]. c2005 [cit. 2005-06-02]. Available from WWW: <<http://www.fluent.com/software/index.htm>>.
- [18] ADINA Finite Element System [online]. c2004 [cit. 2005-06-02]. Available from WWW: <<http://www.adina.com/products.shtml>>.
- [19] LISA - Finite Element Technologies [online]. c2005 [cit. 2005-06-02]. Available from WWW: <<http://www.lisa-fet.com/>>.
- [20] Transoft International [online]. 2005 [cit. 2005-06-02]. Available from WWW: <http://www.fluidyn.com/Home_English/Products/MP.htm>.
- [21] PORFLOW -- ACRICFD.COM [online]. c2004 [cit. 2005-06-02]. Available from WWW: <<http://www.acricfd.com/software/porflow/>>.
- [22] GIRVAN, Ray. *Widening the finite element bounds* [online]. 2004 [cit. 2005-06-04]. Available from WWW: <http://www.scientific-computing.com/scwmayjun04review_femlab.html>.
- [23] I-Math [online]. c2005 [cit. 2005-06-05]. Available from WWW: <http://www.i-math.com.my/product/p_fem.htm>.
- [24] KAWASHIMA, Kenji, FUJITA, Toshinori, KAGAWA, Toshinaru. Unsteady Flow Rate Measurement of Gases using a Micro Flow Sensor. In LANEVILLE, André. *Sixth Triennial International Symposium on Fluid Control, Measurement and Visualization*. Canada : 2000. p. 6. Available from WWW: <http://www.callisto.si.usherb.ca/~fluo2000/PDF/FI_027.pdf>. ISBN 2-7622-0126-8.
- [25] LÖTTERS, Joost. NEW GENERATION OF LIQUID FLOW SENSORS FOR THE NANOLITER THROUGH MILLILITER PER MINUTE RANGE WITH EXTREMELY SMALL INTERNAL VOLUME. In *Sensor 03*.

Germany : 2003. Volume II. p. 21-26. Available from WWW:
<<http://www.instrutec.fr/pdf/sensor03.pdf>>.

[26] VAN BAAR, J. J., et al. Sensitive thermal flow sensor based on a micro-machined two dimensional resistor array. In *Transducers' 01*. Germany : 2001. p. 4.

[27] *IST AG - Thermal Based Flow Sensors* [online]. [2005] [cit. 2005-06-06]. Available from WWW:
<<http://www.ist-ag.com/english/products/flow/flow.html>>.

LIST OF SYMBOLS AND OPERATORS

A	m^2	area
c	$\text{J.kg}^{-1}.\text{K}^{-1}$	specific heat
c	m.s^{-1}	speed of light
C_p	$\text{J.kg}^{-1}.\text{K}^{-1}$	heat capacity
C_{trans}	$\text{W.m}^{-2}.\text{K}^{-4}$	radiation constant
E	W.m^{-2}	emissive power
$E_{blackbody}$	W.m^{-2}	emissive power of a blackbody
$\mathbf{F}$	Pa	volume force field
g	m.s^{-2}	acceleration of gravity
Gr	-	Grashof number
h	$\text{W.m}^{-2}.\text{K}^{-1}$	heat transfer coefficient
h	J.s	Planck's constant
h_{trans}	$\text{W.m}^{-2}.\text{K}^{-1}$	transversal convective heat transfer coefficient
$\mathbf{I}$	-	identity matrix
k	$\text{W.m}^{-1}.\text{K}^{-1}$	thermal conductivity
k	J.K^{-1}	Boltzmann's constant
L	m	surface reference length
m	kg	mass
Δm	kg	small unit mass
Nu	-	Nusselt number
p	Pa	pressure
Pr	-	Prandtl number
q_0	W.m^{-2}	inward heat flux
q_{gen}	W.m^{-3}	internal heat generation

$\mathbf{q}$	W.m^{-2}	heat flux
Q	W.m^{-2}	heat source
Q	J	total heat
$\dot{Q}$	W	heat transfer rate
Q_m	kg.s^{-1}	mass flow rate
Q_V	$\text{m}^3.\text{s}^{-1}$	volume flow rate
Ra	-	Rayleigh number
Re	-	Reynolds number
S	m^2	cross-sectional area of flow carrier
St	-	Stanton number
t	s	time
Δt	s	time interval
T	K	temperature
T_{ext}	K	transversal external temperature
T_{amb}	K	ambient temperature
$T_{ambtrans}$	K	transversal ambient temperature
T_{inf}	K	external temperature
T_{max}	K	maximum temperature
T_w	K	object temperature
T_∞	K	free-stream temperature
ΔT	K	temperature difference
$\mathbf{u}$	m.s^{-1}	velocity field
u	m.s^{-1}	x-velocity
v	m.s^{-1}	y-velocity
v	m.s^{-1}	flow velocity

V	m^3	volume
ΔV	m^3	small unit volume
w	m.s^{-1}	free-stream velocity
∂	-	partial derivative operator
∇	-	vector differential operator
α	$\text{m}^2.\text{s}^{-1}$	thermal diffusivity
β	K^{-1}	volumetric thermal expansion coefficient
δ_{ts}	-	time-scaling coefficient
ε	-	emissivity
$\varepsilon_{\text{absorbed}}$	-	absorbability
$\varepsilon_{\text{reflected}}$	-	reflectivity
$\varepsilon_{\text{transmitted}}$	-	transmissivity
η	Pa.s	dynamic viscosity
ν	$\text{m}^2.\text{s}^{-1}$	kinematic viscosity
π	-	Ludolf number
ρ	kg.m^{-3}	density
σ	$\text{W.m}^{-2}.\text{K}^{-4}$	Stefan-Boltzmann constant
σ	Pa	total stress tensor
τ	Pa	viscous stress tensor
Ω_a, Ω_b	-	body boundaries

LIST OF ABBREVIATIONS

A/D	analog/digital
AVI	audio video interleave
BMP	bit map picture
CAD	computer-aided design
CFD	computational fluid dynamics
CMOS	complementary metal oxide semiconductor
CP	constant power
CT	constant temperature
FEA	finite element analysis
FEM	finite element method
HTML	hypertext markup language
JPG	joint photographic group
MEMS	micro electro mechanical systems
MP	multiphysics
OpenGL	open graphics language
PC	personal computer
PDE	partial differential equation
RTD	temperature variable resistor
SOT	small outline transistor
TIF	tagged image file

LIST OF FIGURES

Fig. 1 High-sensitivity air flow sensor	20
Fig. 2 Tube microflow sensor according to the	20
Fig. 3 2D resistor array microflow sensor	21
Fig. 4 Micromachined gas flow sensor	22
Fig. 5 Gas microflow sensor with self-heating titanium resistors	22
Fig. 6 Gas microflow sensor utilizing capacitive pressure sensor	23
Fig. 7 FS1A sensor	24
Fig. 8 FS2T sensor	25
Fig. 9 Time-of-flight sensor	40
Fig. 10 Modified time-of-flight sensor	41
Fig. 11 Block diagram of the modified time-of-flight sensor	42
Fig. 12 Pin configuration of the AD8591	43
Fig. 13 Pin configuration of the ADC0832B	43
Fig. 14 Functional block diagram of the ADG732	44
Fig. 15 Pin configuration of the ADG732	45
Fig. 16 Model subdomains	49
Fig. 17 Model boundaries	49
Fig. 18 Model navigator	50
Fig. 19 Convection and Conduction – Subdomain settings – 1	50
Fig. 20 Convection and Conduction – Subdomain settings – 2	51
Fig. 21 Convection and Conduction – Subdomain settings – 3	51
Fig. 22 Convection and Conduction – Boundary settings – 1	53
Fig. 23 Convection and Conduction – Boundary settings – 2,3,5,7,10,12	53
Fig. 24 Convection and Conduction – Boundary settings – 11	54
Fig. 25 Incompressible Navier-Stokes – Subdomain settings – 1	55
Fig. 26 Incompressible Navier-Stokes – Subdomain settings – Init tab	55
Fig. 27 Incompressible Navier-Stokes – Boundary settings – 1	56
Fig. 28 Incompressible Navier-Stokes – Boundary settings – 2,4,6,8,9,10	57
Fig. 29 Incompressible Navier-Stokes – Boundary settings – 11	57
Fig. 30 Mesh parameters	58
Fig. 31 Solver parameters	59
Fig. 32 Heat Transfer by Conduction – Subdomain settings – 1	60

Fig. 33 Heat Transfer by Conduction – Boundary settings – 1,4,11	60
Fig. 34 Heat Transfer by Conduction – Boundary settings – 2,10	61
Fig. 35 Heat Transfer by Conduction – Boundary settings – 6,8,9	61

LIST OF TABLES

Tab. 1 General convection (forced and free)	11
Tab. 2 Forced convection only	12
Tab. 3 Free convection only	12
Tab. 4 Gas constants	63
Tab. 5 Material constants.....	63
Tab. 6 Gas flow volumes and velocities according to the pipe height 0.3 mm	64
Tab. 7 Gas flow volumes and velocities according to the pipe height 1.2 mm	64
Tab. 8 Gas flow volumes and velocities according to the pipe height 2.2 mm	64
Tab. 9 Step change model – 1.2 mm.....	71
Tab. 10 Step change model – 2.2 mm.....	71

LIST OF GRAPHS

Graph 1 Temperature change of the heater in time	52
Graph 2 Maximum temperature as a function of velocity – air, 1.2 mm.....	65
Graph 3 Maximum temperature as a function of velocity – air, 1.2 mm, radiation	65
Graph 4 Maximum temperature as a function of velocity – air, 0.3 mm.....	66
Graph 5 Maximum temperature as a function of velocity – air, 0.3 mm, radiation	66
Graph 6 Maximum temperature as a function of velocity – air, 2.2 mm.....	67
Graph 7 Maximum temperature as a function of velocity – air, 2.2 mm, radiation	67
Graph 8 Maximum temperature as a function of velocity – CO ₂ , 1.2 mm, radiation.....	68
Graph 9 Maximum temperature as a function of velocity – CO ₂ , 0.3 mm, radiation.....	68
Graph 10 Maximum temperature as a function of velocity – CO ₂ , 2.2 mm, radiation.....	69
Graph 11 Maximum temperature as a function of velocity – N ₂ , 1.2 mm, radiation	69
Graph 12 Maximum temperature as a function of velocity – N ₂ , 0.3 mm, radiation	70
Graph 13 Maximum temperature as a function of velocity – N ₂ , 2.2 mm, radiation	70

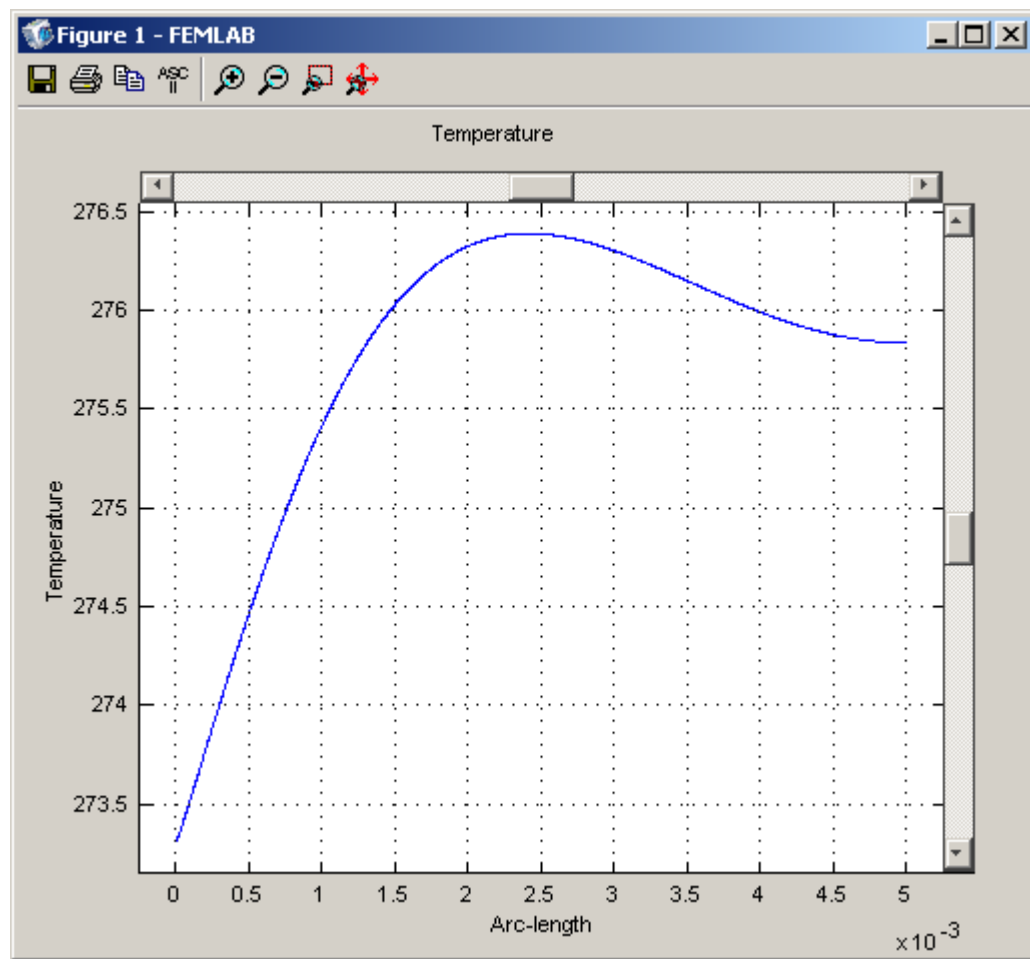
LIST OF APPENDICES

Appendix I Illustration of the temperature wave

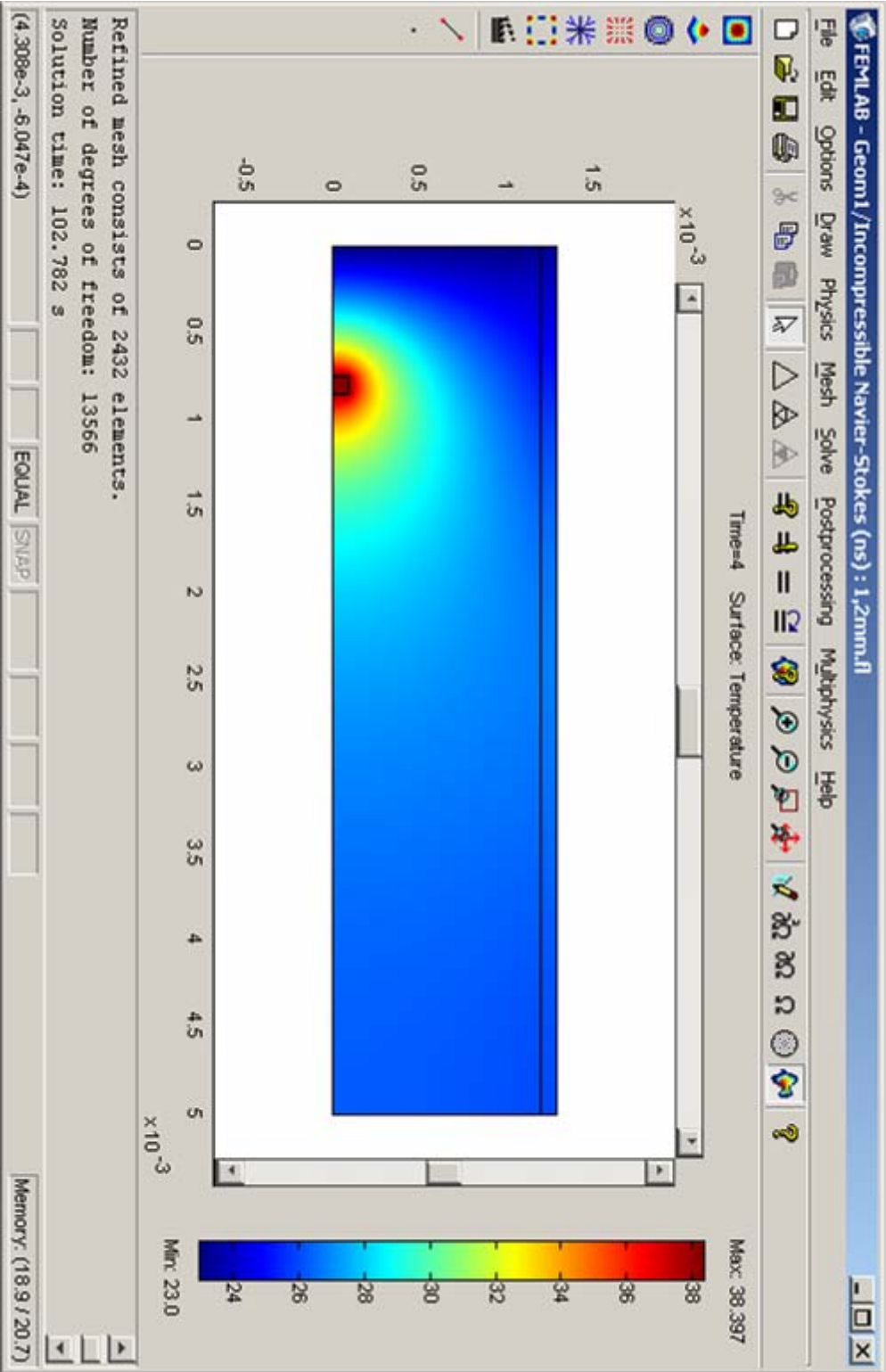
Appendix II Illustration of the temperature field

Appendix III Illustration of the x-velocity field

APPENDIX I: ILLUSTRATION OF THE TEMPERATURE WAVE



APPENDIX II: ILLUSTRATION OF THE TEMPERATURE FIELD



APPENDIX III: ILLUSTRATION OF THE X-VELOCITY FIELD

