Femlab Introduction as a project of the *Multimediale Lehre*

N.Mayer*
Winfried Kernbichler†

January 29, 2002

Contents

*mayer@itp.tu-graz.ac.at
†Inst. für Theoretische Physik;winfried.kernbichler@itp.tu-graz.ac.at

1 Introduction

In this manual, we will give you a short overview of the possibilities that *FEMLAB* offers and a few general modeling hints.The second part of this manual consists of a few examples which shows how to model using predefined equations.

For more and detailed informations you should use the *FEMLAB HELP SECTION*.

1.1 Overview

- *FEMLAB* integrates the power of *MATLAB* and *SIMULINK* in the form of a powerful simulation package.
- Electromagnetics, structural mechanics, fluid dynamics and chemical engineering are some examples in which *FEMLAB* is used.
- *FEMLAB* solves nonlinear coupled systems of Partial Differential Equations (**PDEs**).

1.2 Strategy in *FEMLAB*

To start *FEMLAB* you first make certain that *MATLAB* is running. From that program's Command Window enter the command: **femlab**

That command loads *FEMLAB* and automatically the **Model Navigator**. *FEMLAB* consists of a number of predefined equations, denoted application modes. The first group of modes are **Physics modes**, are shown in the below figure.

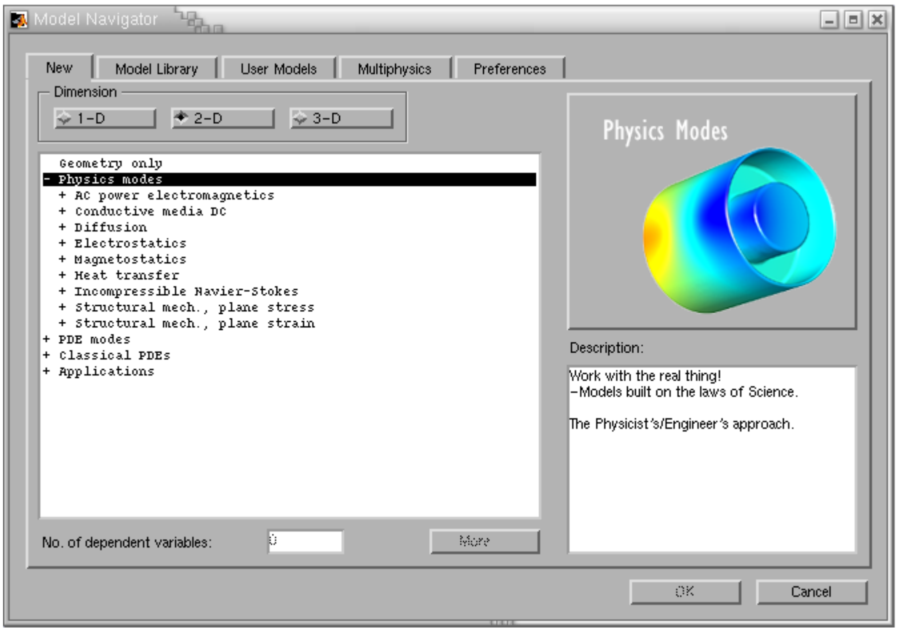

Figure 1: Model Navigator(Physics modes)

The **Physics modes** contain a number of application modes,where specific types of physics are described in PDEs. For the selected physics *FEMLAB* will suggest an equation, or a system of equations. The first step, in setting up a modeling strategy, is to look in **New** tab the **Model Navigator** for a suitable formulation for the problem. As an example you can look into the **Physics modes** and if there is no application for the problem,then you can look into the **Classical PDEs** modes, as shown in the below figure of *FEMLAB*'s **Model Navigator**.

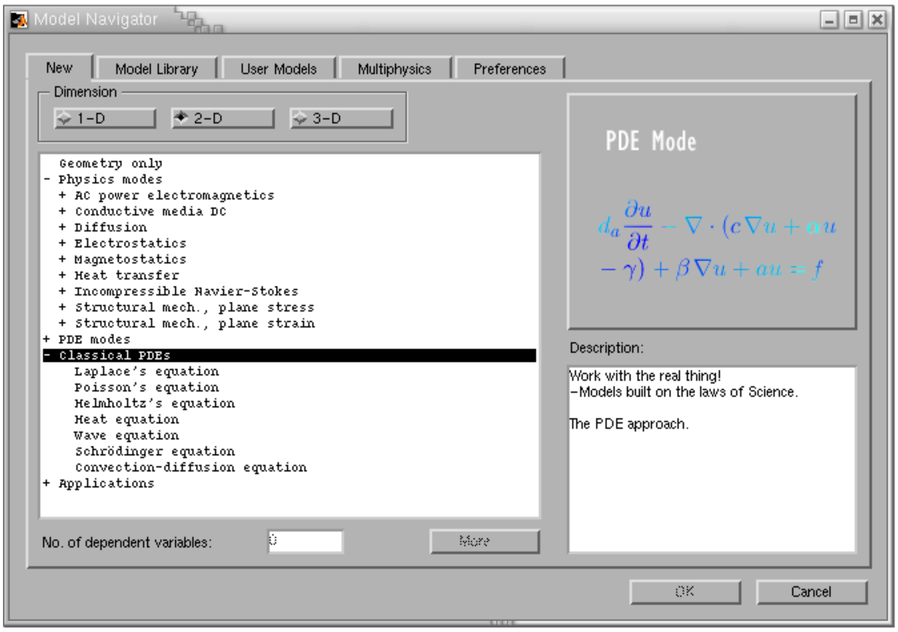

Figure 2: Model Navigator(Classical PDEs modes)

The **Classical PDEs** modes present a number of formulations of classical problems in mathematics and physics. If neither the **PHYSICS modes** nor the **Classical PDEs** modes are suitable formulations, you should try to use a more flexible application mode for example the **PDE modes**. These modes are more general. The **General** mode is suited for nonlinear equations and the **Coefficient** mode is for linear dynamic problems.

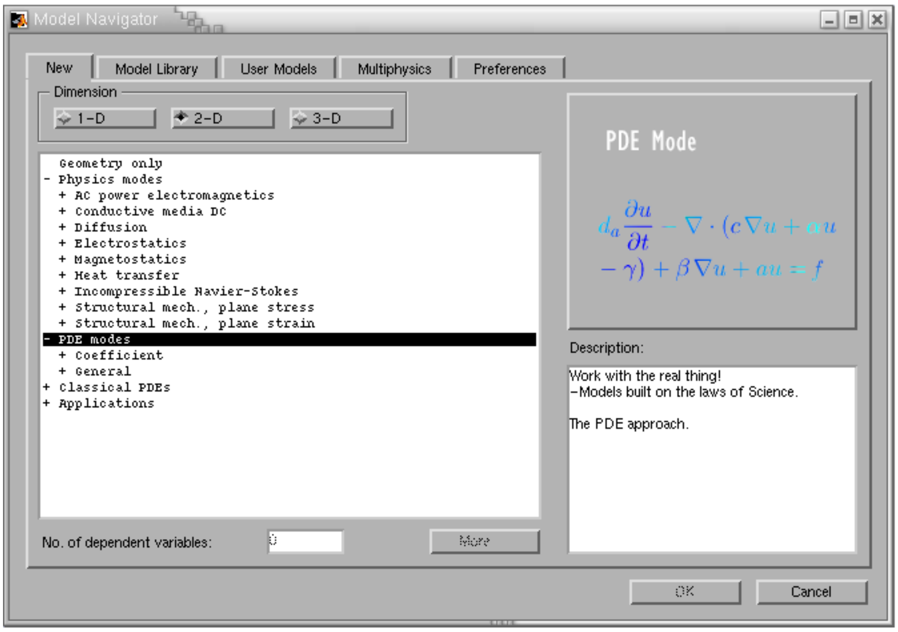

Figure 3: Model Navigator(PDE modes)

1.3 MODELING MULTIPHYSIC PROCESSES

In most processes there are several coupled physical phenomena.If these are found for example in the Physics modes you can use the **Multiphysics** feature in the **Model Navigator**.

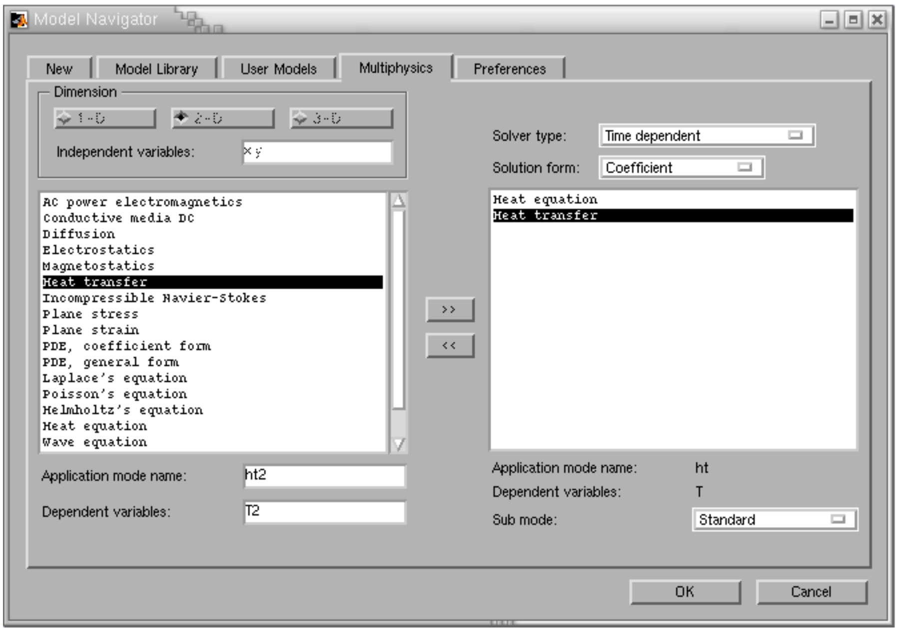

Figure 4: Model Navigator(Multiphsics)

The left side shows the available apllication modes, while the right side shows the modes you have selected. It is also possible to combine a ready-to-use aplication modeans a **PDE mode**. This also creates a system of PDEs but requires that you define the PDEs of the **PDE mode**.

If there are no such apllications for the problem,you can use the **PDE mode** by replacing in the edit field denoted **No. of dependent variables** the default value 1.

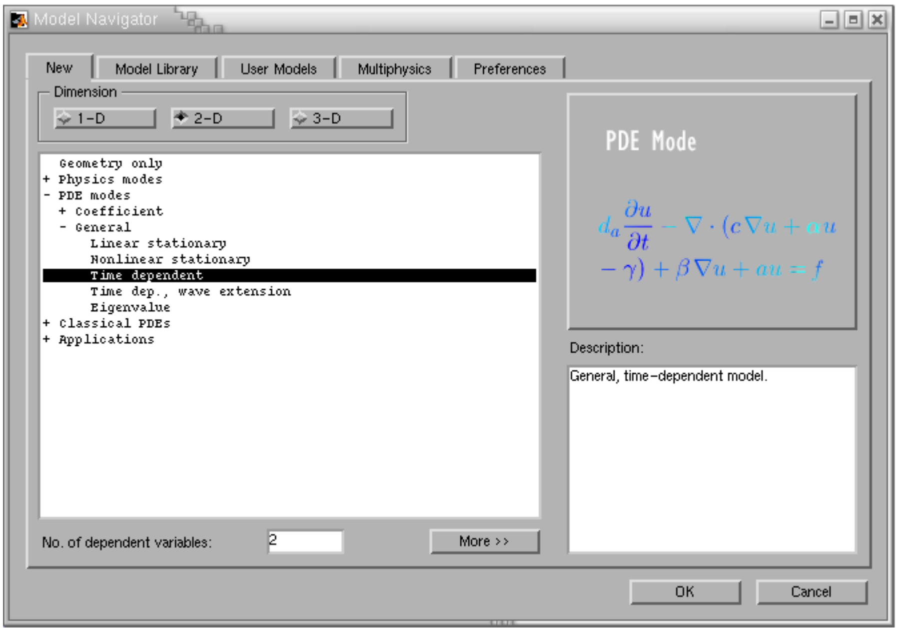

Figure 5: Model Navigator(PDE modes)

We are now ready to summarize the strategy for modeling processes.

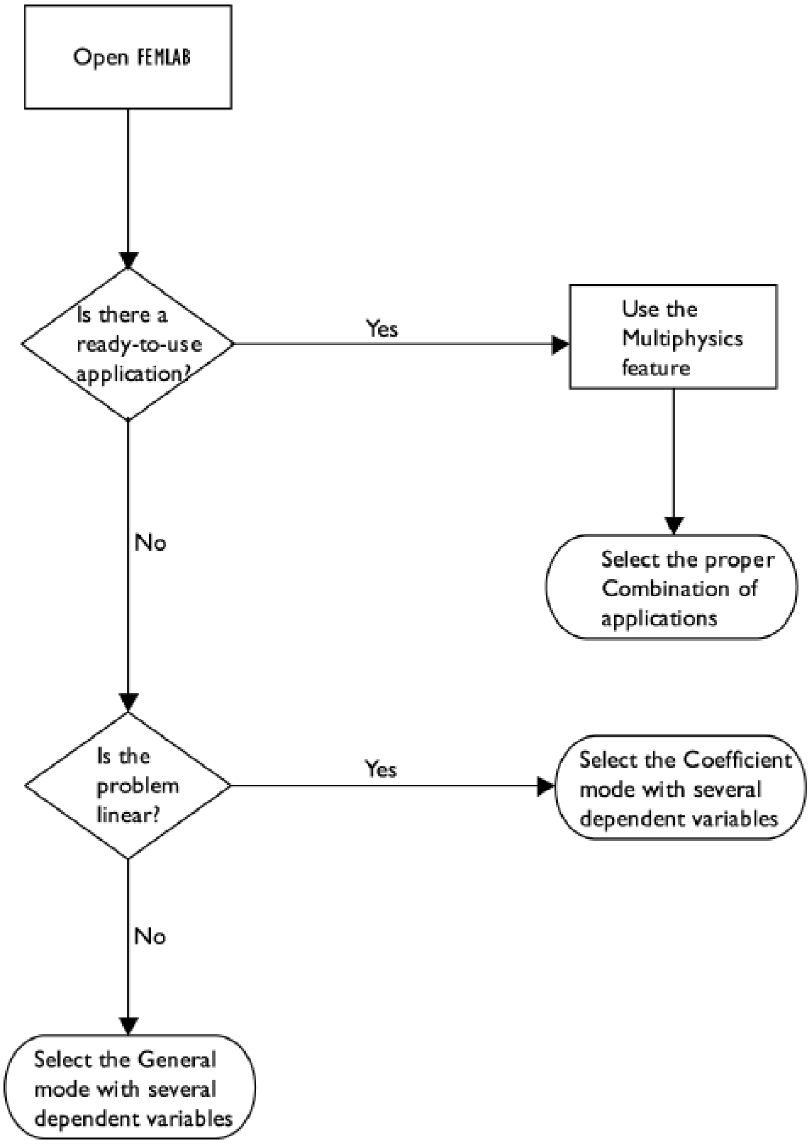

Figure 6: The strategy for modeling processes

2 Tutorial models

2.1 2D MODELING

Model background

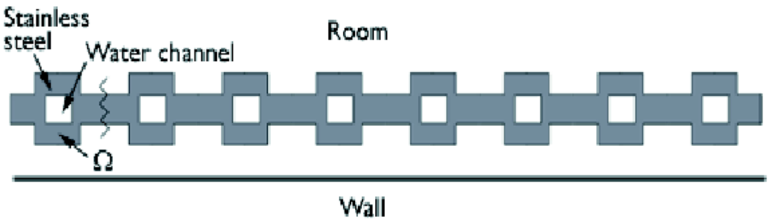

Figure 7: 2D profile of a radiator

A 2D profile of a radiator will be modeled. This example models time dependent heat transfer. Initially the radiator has the same temperature as the surroundings $T_{external} = 293K$. At $t = t_0$ water with the temperature of $T_{in} = 333K$ enters the water channel and heat transfer in the stainless steel profile is studied. Only one tube is modeled due to symmetry.

The time dependent heat transfer is described by:

$$\rho c_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = 0 \qquad in\ \Omega \tag{1}$$

ρ ... density

c ... heat capacity

k ..... Coeff.of heat conduction

T ..... Temperature

The following material properties are used:

- density $\rho = 7800[kg/m^3]$
- heat capacity $c_p = 460[J/kg, K]$
- conductivity $k = 45[W/m, K]$

Convective heat transfer according to

$$(-k\nabla T) \cdot \mathbf{n} = h(T - T_{external}) \tag{2}$$

is used for boundaries facing the wall (red) and the room (blue) in the figure below. The term on the right side of the above equation is the Newton's law of cooling.On the left side is the flux vector projected on the normal unit vector **n** of the surface.

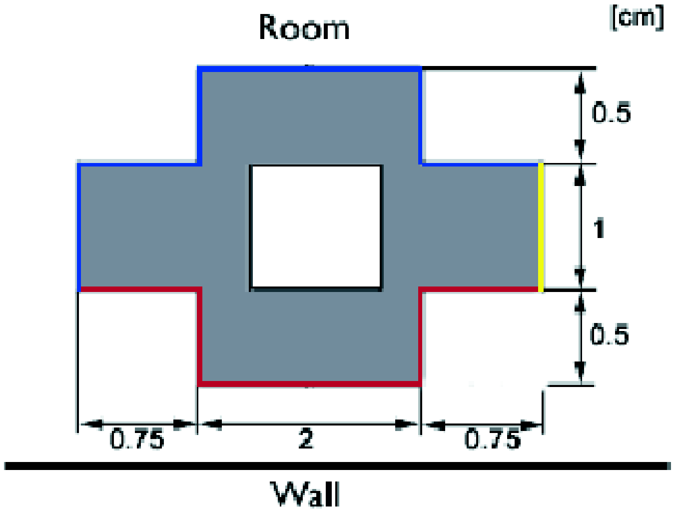

Figure 8: Model of one tube

The six boundaries facing the room (blue) are assumed to have $h = 10[W/m^2, s]$, where h is the heat transfer coefficient between the radiator and the surrounding air. The heat transfer is lower for the five wall boundaries (red) and h is estimated to be $5[W/m^2, s]$ due to lower air circulation.

A symmetry boundary described by

$$(-k\nabla T) \cdot \mathbf{n} = 0 \tag{3}$$

is used for the (yellow) symmetry plane. This means that no net heat flux will go through this boundary.

Constant temperature

$$T = T_0 \tag{4}$$

is assumed for the four internal (black) boundaries of the water channel.

2.2 3D MODELING

Model background

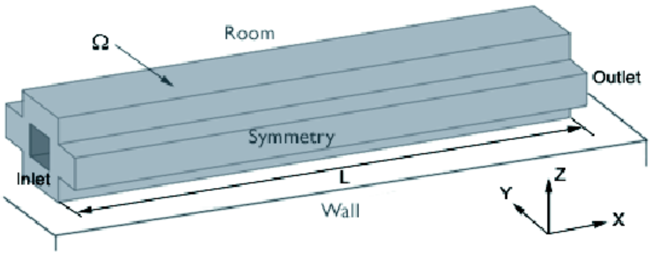

Figure 9: 3D tube of a radiator

The 2D profile is extruded into a 3D tube. Water enters from the top at $T_{in} = 333K$ and is linearly cooled with the traveled distance as it flows downwards, to exit at $T_{out} = 323K$. The example shows a simplified way to model the time dependent heat transfer in the tube.

The time dependent heat transfer is still the same as in (1).

All material properties are the same as in the 2D example.

Boundaries facing the wall and the room is described as in (2) and the heat transfer coefficients are the same as in the 2D example. The symmetry plane is described by (3).

$$T = T_{in} - \frac{T_{in} - T_{out}}{L} \tag{5}$$

for the water channel,where L is the length of the tube.

3 Post Processing

In *FEMLAB* you can save an entire session performed in the graphical interface as a sequence of command-line function calls. This is called a Model M-file.

A Model M-file is useful for many purposes:

- It documents the work you perform within the graphical interface in a human-readable manner.
- Studying the file *FEMLAB* generates can serve as a tool for learning how to work with command-line functions.
- You can save a Model M-file, modify it for optimal design or for creating parameterized models.

- You can insert any *MATLAB* command anywhere in the Model M-file as long as they don't affect the FEM structure in any way inconsistent with the syntactic rules, and reload the model into the GUI.

- When working on a complicated model that requires you to write custom functions, you can start modeling in the graphical interface and then save the Model M-file to expand it. Note again, tough, that if you work outside the syntactic rules, you can run the resulting model only from the command line.

Situations when command-line functions are useful:

- using the *FEMLAB* functions as a function library for creating your own programs or functions

- non-standard boundary conditions

- complicated PDE or boundary condition coefficients

- special solution data processing and presentation

You can always view the help text for the FEM structure by typing: help femstruct at the command line.