CAE of Sensors and Actuators
Date: November 2020
Assignment due: assignment is OPTIONAL

Lehrstuhl für Sensorik
Universität Erlangen-Nürnberg
Daniel Kiefer

OPTIONAL - Assignment 1: Weak Form PDE in COMSOL

Note: This assignment is **optional**. You do not have to submit it and it does not count towards the minimum number of assignments that have to be submitted. Nevertheless, you can still submit it if you decide to work on it and you want to have it checked. We **recommend** that you take the opportunity to work on the assignment, as it highlights the strengths of the weak form and how COMSOL uses it.

The aim of this assignment is to solve the following boundary value problem (BVP) using the *Weak Form PDE* interface of COMSOL Multiphysics:

$$
\begin{aligned}
-u''(x) + cu(x) = f(x) \quad &\text{on} \quad x \in [a, b] \subset \mathbb{R} \\
u(a) &= g_a \\
u(b) &= g_b \,,
\end{aligned}
\tag{1}
$$

where the coefficient c is usually called the *absorption* coefficient. Note that you have derived the weak form of a very similar BVP in the exercise class.

General hints and remarks

Please make sure to:

- <u>Save</u> all relevant plots you generate (e.g., as `png`-file) while working on the assignment.

- Always <u>label the axes</u> of the plots and <u>add a legend</u> if necessary.

- Format your plots so that they can be <u>read easily</u> (e.g., appropriate font size).

- Prepare a `Results.pdf` file with the plots and <u>short</u> answers to all questions (label each answer with its task number) for submission.

- If you are unsure on how to perform the tasks, refer to

  - Comsol Tutorial: Electrostatic Field
  - COMSOL Multiphysics Reference Manual

1 Preparation

1.1. Write down the weak form of (1). For this assignment, bring all the terms to one side and choose the signs such that the source term (the one which contains f) is positive. This corresponds to COMSOL's convention.

1.2. Follow the *COMSOL Tutorial*, which you find on StudOn, to get started with using COMSOL Multiphysics. This is a step by step introduction to COMSOL Multiphysics.

2 Basic setup

2.1. Create a new COMSOL project using the Model Wizard. How many dimensions does your problem have and how can you tell? Use *Mathematics→PDE Interfaces→Weak Form PDE* as your *Physics*. This interface allows you to specify a custom weak form. What kind of *Study* do you need to use and why?

2.2. Geometry: (1) shall be solved on the interval $[a = 0, b = 10] \subset \mathbb{R}$. Model your geometry by using an *Interval* node.
Note: it is always a good idea to parameterize your model. Do not specify a and b explicitly in this interval node. Rather define parameters `a` and `b` in the *Global Definitions→Parameters 1* node and use them here.

2.3. Material: do you need to add a *Material*? Why or why not?

2.4. Inspect your physics node, named *Weak Form PDE (w)*. Make sure that the PDE is solved on the entire domain. The unknown variable u is called *Dependent Variable* in COMSOL. Note that the default name for the dependent variable is `u`, you will need this variable name to enter the weak expression. You could change the variable name here or add more dependent variables if you were to model a vector field. Under the *Units* section you could also specify a unit for your field u. Leave it non-dimensional for now. Is the default unit for the *Source term quantity* correct? Why? Consequently, what unit should the coefficient c have?

2.5. Disclose the *Weak Form PDE (w)* node to inspect the *Weak Form PDE 1*. This is where you specify the PDE you want to solve. Disclose the *Equation* section to find out what COMSOL is solving for. This equation contains an expression labeled `weak`. You can specify this expression in the section below, entitled *Weak Expression*. You will see that there is a default expression already specified. The notation used by COMSOL is the following:

description	variable	COMSOL
dependent variable	u	`u`
dependent variable derivative	u'	`ux`
test function	v	`test(u)`
test function derivative	v'	`test(ux)`

What PDE does the default weak expression correspond to? Leave the weak form expression as it is for now. This corresponds to solving (1) for a special value of c and f. What are those values?

2.6. Below the *Weak Form PDE 1* node there is a node labeled *Zero Flux 1*. Select it and inspect the Equation. Press F1 to open the help window for this node and read the text. You will see that the flux Γ is defined as

$$\Gamma = \hat{c}\nabla u + \hat{\alpha}u - \hat{\gamma}. \tag{2}$$

Note that to distinguish the COMSOL coefficients from the ones in (1) we marked them with hats, in special, $\hat{c} \neq c$. Furthermore, the flux Γ is defined in COMSOL to yield the PDE by forming its divergence in the following way:

$$\nabla \cdot \Gamma = F. \tag{3}$$

For the case of (1) , $\Gamma = -u'$ and $F = f - cu$. Comparing this Γ to (2), what are the values of the coefficients $\hat{c}$, $\hat{\alpha}$ and $\hat{\gamma}$? What kind of boundary condition (BC) does the default *Zero Flux* condition represent? Why is this the default BC?

2.7. Go to the *Mesh* node and add an *Edge* node. Right click the new *Edge 1* node and add a *Distribution*. This let's you specify the number of elements you want to have in your modeled interval. Specify 10 elements. How many nodes should this mesh have?

2.8. A study has already been added by the wizard you used to create your COMSOL model. Select it and click on *Compute* on the settings window. Read the error message. Why are you getting the error?

2.9. Go back to the physics node named *Weak Form PDE (w)*. Add a Dirichlet BC and impose $u = 0$ on both edges of the domain.

2.10. Try again to compute the solution. It should work now.

2.11. Plot `u` over `x` (COMSOL will probably do this automatically). Is the solution in accordance to what you have computed in assignment 0?

3   Solving the given boundary value problem

3.1. Add any parameters you need in the *Parameters 1* node. In particular, add a parameter `c` for the c constant in (1) and assign a value of $0.5/\text{m}^2$. Also add a parameter `f_amp` that shall represent the amplitude of the forcing function $f(x)$.

3.2. Modify the *Weak Form PDE 1* node such that the PDE given in (1) is modeled. Use the parameter `c` in the equation and take the right hand side to be of constant value `f_amp`.

3.3. Compute and plot the solution to make sure that you have implemented the new PDE correctly. How did the solution change?

4   Problem variations

4.1. Add a parameter study in the *Study* node and iterate the value of c. Assign the values $0/\text{m}^2, 0.1/\text{m}^2$, $0.5/\text{m}^2$ and $1/\text{m}^2$.

4.2. Compute the above parameter study and plot the result as a series of curves in the same plot. Make sure to add a legend and that everything is labeled correctly. How does the solution change with c?

Now let's consider the following non-constant forcing function:

$$f(x) = \text{rect}(x) = \begin{cases} \texttt{f\_amp} & (a+b)/4 \leq x \leq (a+b)3/4 \\ 0 & \text{otherwise.} \end{cases} \tag{4}$$

4.3. Right click the `Global Definitions` node and go to *Functions*. Add a *Rectangle*-function. Set the parameters of the function as required by (4). In the *Smoothing* section, you can see that the rectangle will not be an ideal rectangle but will have smoothed edges. Click on *Plot* to see how the function looks like.

4.4. Use the newly defined function as forcing function $f(x)$ in the *Weak expression* of the *Weak Form PDE 1* node. The rectangle has unit amplitude, hence, scale it with your parameter `f_amp`.
Note: as you can see in the parenthesis next to the node name of the *Rectangle*, the function is named `rect1`, so you can call the function by typing `rect1(x/1[m])`. In COMSOL `x` denotes per default the coordinate of your 1D geometry in meters. The `rect1` function expects a non-dimensional input.

4.5. Compute the solution for the same values of c as before and plot again the series of curves. For the case $c = 0/\text{m}^2$, how does the solution behave in the different domain regions defined by (4)?

5   Postprocessing and visualization

5.1. Go to the *Line Graph* node under *1D Plot Group* and disclose *Coloring and Style* in the settings window. Keep the solid line but choose a marker of your liking. Position the markers *in the data points*.

5.2. Click plot and inspect the result. Does the number of markers correspond to the number of nodes of your mesh?

5.3. Disclose the *Quality* section. For *Resolution* specify *No refinement*. Click on plot again. How many markers do you have now? Is this what you would have expected? Why?

5.4. The refinement of the plot that was being applied per default was performing a nonlinear interpolation, yielding a smoother and actually more accurate curve. How is this possible? When does it make sense to perform a refinement of the solution before plotting? **Hint:** If you are not able to answer these questions right now, get back to them after finishing section 5.

5.5. Go to the physics node *Weak Form PDE (w)* and in the settings window disclose the *Discretization* section. What type of shape function and polynomial order are you using?

5.6. Change the *Element order* to *Linear*. This is the polynomial order of your basis functions.

5.7. Re-compute the result. How many degrees of freedom did COMSOL solve for? You can find this out in the *Messages* window.

5.8. Change the *Resolution* of the *Line Graph* back to *Normal* to perform an interpolation of the result before plotting. What type of interpolation is being performed now? What element order seems to be giving you a better accuracy of your FE approximation?

5.9. Change the element order back to *Quadratic*, re-compute and re-plot the result without changing any other settings. How many degrees of freedom (DoFs) did COMSOL solve for this time?

Remember: Your Finite Element approximation $u^h(x)$ is not just the solution at the nodal points u_i. It is actually a **continuous function** given by its ansatz, e.g.,

$$u^h(x) = \Sigma_{i=1}^{M} u_i N_i(x). \tag{5}$$

The accuracy of the solution depends on the spacial discretization $h = (b - a)/M$ as well as the polynomial order p of the basis function N_i!

6   One more remark about the weak form

The weak form is an equation in terms of integrals over the whole domain. These are merely scalar or vector *values* rather than functions, e.g., $\int_a^b \sin(x) + 2 \, dx$ is no longer a function of x after performing the integration. No matter how complicated the geometry or the problem is, with the weak form it is always represented in terms of such a scalar or vector valued equation. Every considered effect, domain and boundary condition is included in the equation by **adding** a term to this weak equation. You want to include an additional domain with a different material to your simulation? Add it as an integral term to your weak form. You have been ignoring the effect of convection in your simulation and you want to consider it now? Add it as an integral term to your weak form. Such terms are called *weak contributions* in COMSOL. Try this concept by moving the term $cu(x)$ in (1) form the *Weak Form PDE 1* node into a separate *Weak Contribution* node with the following steps:

6.1. Right click on *Weak Form PDE (w)*. Note that the menu is splitted by horizontal lines. The first section specifies *domain contributions*, the second one *point contributions*. That is why in the latter section you find the boundary conditions (the boundaries are points in 1D). In the domain section go to *More* and select *Weak Contribution*.

6.2. Select the newly created *Weak Contribution* node and make sure that it is being applied to *All domains*. Then go to the *Weak Contribution* section and enter the weak term corresponding to $cu(x)$ as *Weak expression*. Go back to the *Weak Form PDE 1* node and remove this same term from the weak expression.

6.3. Compute and plot the new model. Verify that nothing has changed.

6.4. Deactivate the *Weak Contribution 1* node (select it and press F3). Compute and re-plot your parameter study for c to see the effect. Are your curves independent of c now?

The advantage of splitting the equation into several contributions is that you can separately turn them on and off. In this way you can easily set up different cases of your model and study the influence on your result. In COMSOL the weak expression is in practice always split up into different nodes (physics node, boundary condition nodes, excitation nodes) and this is the reason why we need a sign convention as explained in point 1.1.

The concept of "weak contribution" is very powerful. Whenever you add a sub-node to your *Physics* node in COMSOL, you are in practice adding a weak contribution. Furthermore, you can *always* add a *Weak Contribution* node containing your personal weak expression, even to pre-defined physics

(e.g., electrostatics, mechanics). This empowers you to change the equations and boundary conditions in whatever way you like. On the other hand, any pre-defined physics (PDE), boundary condition or constraint could be specified by a *Weak Contribution* node instead.

7   Submitting your assignment

Hand in your

- COMSOL `mph`-file (clear all meshes and results beforehand, see the COMSOL Tutorial), and

- `Results.pdf` file with the plots and <u>short</u> answers to all questions (label each answer with its task number),

before the due date by uploading to the corresponding StudOn section.

When handing in for the first time you need to <u>create a group</u>. Please add all members when doing so! Once the group has been create, you can and should <u>re-use it</u> for the subsequent submissions.