

## SUPPLEMENTARY HANDOUT: COMSOL GUIDE FOR 1-D

This handout is designed to highlight some of the key features and settings that can be found in the menus of COMSOL. Remember that you can always get more background information in the “Help Desk”, which can be found in the Help drop down menu.

### 1 Opening a new problem

- For a 1-D problem, select 1-D in the “space dimension” drop down.
- The most general way to construct a 1-D problem is to use PDE Coefficient mode, which can be found under the “Application Mode” → “COMSOL Multiphysics” folders.
- Select either “Stationary” or “Time-dependent analysis”.

### 2 Options drop down menu

- The first item listed is where you define the “Constants”. Here is where you can specify any constants that will remain fixed during your analysis. Simply put the constant name, such as  $k$ , under “Name” and define its numeric value under “Expression”. The value will automatically appear under “Value”. You do not need to use  $=$  to assign constants.
- The second item is where you define “Expressions”. For now we will just focus on “Global Expressions”. This is basically where you can define functions of the space and time variables. Simply put the name of the function under “Name”, and put the actual function itself under “Expression”. For example, if I wanted to define  $f = k^2 e^x$ , where  $k$  has been defined in constants, then it would look like Table 1.
- There are more features in this drop down menu, but they are not needed for your assignments

### 3 Draw drop down menu

- The first item is the “Draw Object” option. If you select “Line”, this allows you to graphically create the geometry, by drawing the line

Name	Expression
f	$k^2 \exp(x)$

Table 1: Defining an expression f.

segment from end point to end point. But you cannot change the length after you create the line. (Very strange, I know)

- The second item is “Specify Objects”. If you select line, this allows you to generate a line based on geometric coordinates. For example, if you specify 0, 1 in the “Coordinates” field, it will generate a single line from 0 to 1. But if you specify 0, .5, 1 in the “Coordinates” field, then it will generate a line that is constructed of a segment that goes from 0 to 0.5 and another that goes from 0.5 to 1. COMSOL knows that these are continuous, so it will only make the right and left most points the boundaries. This is very handy when you have a domain composed of several segments....HINT, HINT.

## 4 Physics drop down menu

- The first item is “Subdomain Settings”. Under the “Coefficients” tab you will specify all of the coefficients and loads that make the PDE take the form of the problem you want to solve. Be aware of minus signs, you may need to adjust the sign of the load to make the problem in COMSOL look like the problem on paper. If you have different coefficients for each subdomain, simply select which domain you want and enter its coefficient values. Under “Init” tab, you can specify any initial conditions that you may have for time-dependent problems. Under the “Element” tab, you can choose which order element you want to solve the problem. Notice that the integration order automatically changes when you change the element order. The integration order values can be changed manually as well if you desire.
- The second item is the “Boundary Settings”. Under the “Coefficients” tab you will specify the Dirichlet boundary conditions (puts condition on the solution field) and the Neumann boundary conditions (puts condition on the derivative of the solution field). Simply fill in the coefficients to match the condition that you wish to specify. For ex-

ample, if we wish to have  $u=1$  at the left boundary, you would select boundary 1 and assign  $h$  to be 1 and  $r$  to be 1.

## 5 Mesh drop down menu

- The main item we want to use, in 1-D, is the “Free Mesh Parameters”. Under the “Global” tab, fill in the “Maximum Element Size” field with the dimension of the elements you want to solve the problem. For example, if we wanted a mesh with 100 elements, we would assign the element size to be 0.01. If you had multiple segments in the domain, you could go to the “Subdomain” tab and assign different element sizes in each segment, if you so desired. You could even adjust how the element size blends as you approach the interface between segments. This is not necessary for your assignments, but it is good to know all of the options.
- Once you have specified the element size, click on “Remesh” to generate the mesh.

## 6 Solve drop down menu

- The main item we want to use is the “Solver Parameters”. Under the “Solver” selection field, you can choose what type of analysis you are conducting. For example you can select “Stationary” for steady state problems, and where all of the parameters are fixed. You can select the “Time dependent” for transient problems, and where all of the parameters are fixed. Finally, you can choose “Parametric”, which is a stationary solver, but you can vary a parameter of interest and have COMSOL solve the problem each parameter all in one simulation. For example, when you are asked to solve the 1-D problem with different  $k$  values, you can put  $k$  in the “Parameter Name” field and then put the values in the “Parameter values” field.
- Under the “General” tab, you can select which type of “Linear system solver” you wish to use. The default UMFPACK is fine for our needs, but notice that you can select other direct and iterative solvers. For example, GMRES, FGMRES, and Conjugate Gradients are all iterative solvers. You can read more about these in the “Help Desk”.

- When you have selected the “Time dependent” analysis, the “General” tab has a section labeled “Time stepping”. In the “Times” field, you assign the start and end times, as well as the time step to be taken. This is entered just like a Matlab command. For example, the default is 0:0.1:1, which means start at 0, end at 1, and move in increments of 0.1.
- Another tab in “Time dependent” analysis, is the “Time Stepping” tab. Under this tab there is a section labeled “General”. You will want to use the default setting in the “Times to store in output” field. The setting is “Specified” times, which means that COMSOL will save the solution it gets at each instant of time that is specified in the “Times” field, under the “General” tab. The default setting in the “Time steps taken by solver” field is Free. This means that COMSOL can take as many steps as it wants between the time increments that you specified, in order to get the solution. This is based on an adaptive time stepping scheme. If you want the times steps to march forward in the exact increments you specified, change the field to “Strict”. With this setting you are in complete control of the time stepping.
- Lastly, there is a section under the “Time Stepping” tab labeled “Advanced”. This is where you choose the actual time integration scheme. COMSOL only uses implicit methods for the time integration. If you change the “Maximum BDF order” to 1, COMSOL will use an implicit (backward) Euler scheme. Higher order BDF methods might not be as stable, so beware when you are choosing a scheme. If you want to read more about these schemes, there is a decent discussion in the “Help Desk”.

## 7 Postprocessing drop down menu

- The first item is the “Plot Parameters”. In 1-D there are only a few tabs that appear in the part of the menu. When we get to 2-D and 3-D problems, there will be many more types of plots that can be generated. In the section labeled “Solution” to use, there is nothing to select if you have used a non-Parametric solver. But if you have used the “Parametric” solver, you can select the solution to whichever parameter you wish. Under the “Line” tab, there is a field labeled “Predefined quantities”. This is where you select what you wish to visualize. For example,  $u$  is the solution itself,  $u_x$  is the first spatial

derivative of the solution,  $u_t$  is the first time derivative of the solution, etc.

- Another item in the menu is for “Data Display”. If you select “Subdomain”, you can get the numerical value of the predefined quantity of interest at a specific coordinate. For example, if you have selected  $u$  as the predefined quantity, an  $x$  coordinate of 0.5, and then click “Apply”, COMSOL will display the value of the solution at that point in the bottom display of the COMSOL window. If you check the “Display results in full precision”, COMSOL will display all the decimal places it has stored.
- Another item in the menu is for “Domain Plot Parameters”. This is one place that you can output the plots to a file. Under the “General” tab, select the “Line/Extrusion plot”. At the bottom of this tab section, you can select “Title/Axis”. This is where you can change the labeling of the axes so they are not blank. Under the “Line/Extrusion” tab you select the predefined quantity you wish to export in the plot. Go into the “Subdomain” selection field and highlight which domains you want, which will usually only be one. Finally, when you click “Apply”, a window will open with the plot and will give you the option to save. Another way to generate less detailed plots is to go to the “File” drop down menu, then go to “Export” → “Image”.