Solving Poisson’s equation using Lagrange finite elements with Comsol Multiphysics

Michael Neilan

Louisiana State University
Department of Mathematics
Center for Computation & Technology
Poisson’s equation

• Consider Poisson’s equation:

\[-\Delta u = f \quad \text{in } \Omega,\]
\[u = g \quad \text{on } \partial\Omega.\]

• The variational formulation is

\[(\nabla u, \nabla v) = (f, v) \quad \forall v \in V := H^1_0(\Omega),\]

• The corresponding FEM is

\[(\nabla u_h, \nabla v_h) = (f, v_h) \quad \forall v_h \in V^h.\]

Here, \(V^h \subset V\) consists of quadratic piecewise polynomials.

• In this example, we take

\[\Omega = (0,1)^2, \quad f = 2\pi^2 \sin(\pi x) \sin(\pi y), \quad g = 0,\]

so that the exact solution is \(u = \sin(\pi x) \sin(\pi y)\).
Step by step instructions

Step 1: Start the application Comsol Multiphysics. The path to the executable is

/usr/local/packages/comsol35a/bin/comsol
Step by step instructions

Step 2: (a) Select ‘PDE Modes→Weak Form, Subdomain→ Stationary analysis’
(b) Select ‘OK’
Step 3: (a) Select ‘Draw→Specific Object→Square’
(b) Select ‘OK’
Step by step instructions

Step 4: (optional) Click the icon for “Zoom Extents” - it is the icon with a magnifying glass and a red cross
Step by step instructions

Step 5:  
(a) Select ‘Options→Global Expressions’
(b) Enter the following information in the given box:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>f</td>
<td>$2\pi^2 \sin(\pi x) \sin(\pi y)$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>g</td>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>exactsoln</td>
<td>$\sin(\pi x) \sin(\pi y)$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>error</td>
<td>$</td>
<td>u-\text{exactsoln}</td>
<td>$</td>
</tr>
<tr>
<td>l2err</td>
<td>$</td>
<td>\text{error}</td>
<td>^2$</td>
</tr>
</tbody>
</table>

(c) Select ‘Ok’
Step by step instructions

Step 5:  (a) Select ‘Options → Global Expressions’
(b) Enter the following information in the given box:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>f</td>
<td>$2\pi^2 \sin(\pi x) \sin(\pi y)$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>g</td>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>exactsoln</td>
<td>$\sin(\pi x) \sin(\pi y)$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>error</td>
<td>$</td>
<td>u - \text{exactsoln}</td>
<td>$</td>
</tr>
<tr>
<td>l2err</td>
<td>$</td>
<td>\text{error}</td>
<td>^2$</td>
</tr>
</tbody>
</table>

(c) Select ‘Ok’
Step by step instructions

Step 6:  
(a) Select ‘Physics→Subdomain Settings’
(b) Enter the following information in each field:

- weak : $u_x \star \text{test}(u_x) + u_y \star \text{test}(u_y) - f \star \text{test}(u)$
- dweak : 0
- bnd.weak : 0
- constr : 0
- Constraint type : ideal
- constrf : 0

(c) Select ‘OK’
Step by step instructions

Step 6:
Step 7:  
(a) Select ‘Physics→Boundary Settings’
(b) Under the “Weak” tab, select all four boundaries (1,2,3,4) and enter the following information:

- weak : $-\text{test}(u) \times (ux \times nx + uy \times ny)$
- dweak : 0
- constr : $-u + g$
- Constraint type : ideal
- constrf : 0

(c) Select ‘OK’
Step by step instructions

Step 7:

[Image of a software interface showing boundary settings and weak form settings.]
Step by step instructions

Step 8: Select the ‘Solve’ icon (the icon with a plain equal sign)
Step by step instructions

Step 9: To view the error,

(a) Select 'Post Processing→Plot Parameters'
(b) Select the 'Surface tab'
(c) In the 'Surface Data Subtab', enter 'error' in the Expression field
(d) Select the 'Height Data Subtab' and check the box for Height Data
(e) Select ‘Apply’
Step by step instructions

Step 9:
Step by step instructions

Step 9:
Step 10: To calculate the error in the $L^2$ norm,

(a) Select ‘Post Processing→Subdomain Integration’
(b) Enter ‘l2err’ in the Expression field
(c) Select ‘Apply’
(d) The value should appear in the lower left-hand side of the screen (note: this is the quantity $\|u - u_h\|_2^2$ not $\|u - u_h\|_{L^2}$).
Step by step instructions

Further Extensions

1. Change the finite element type by selecting ‘Physics→Subdomain Settings’ and going to the ‘Element’ tab. Many predefined elements are included.

2. Change the mesh parameters by selecting ‘Mesh→Free Mesh Parameters’. Many predefined meshes are already given in the dropped-down tab, ‘Predefined mesh sizes’.

3. Changing the solver by selecting ‘Solve→ Solve Parameters’

4. Changing the dimension. In Step 2, there is a drop-down tab for Space dimension (see picture in Step 2) *(unfortunately, this change can only be changed from the very beginning).*