Conducting Finite Element Convergence Studies using COMSOL 4.0

David W. Trott and Matthias K. Gobbert

Department of Mathematics and Statistics, University of Maryland, Baltimore County, 1000 Hilltop Circle, Baltimore, MD 21250, {dtrott1, gobbert}@umbc.edu

Abstract: In order to gauge how reasonable a finite element solution to a partial differential equation is on a given mesh, a common strategy is to refine the mesh, compute the solution on the finer mesh, and use the solutions on the two meshes for a qualitative comparison. The theory of the finite element method (FEM) makes these comparisons quantitative by estimating the convergence order of the FEM error on a sequence of progressively finer meshes. We will show how to carry out convergence studies of this type in the graphical user interface of COMSOL 4.0 on the example of Lagrange elements of varying polynomial degrees, which will also bring out the benefit of using higher order elements.

Key words: Poisson equation, a priori error estimate, convergence study, mesh refinement.

1 Introduction

The finite element method (FEM) is widely used as a numerical method for the solution of PDE problems, especially for elliptic PDEs such as the Poisson equation with Dirichlet boundary conditions

$$-\Delta u = f \qquad \text{in } \Omega, \qquad (1.1)$$
$$u = r \qquad \text{on } \partial\Omega, \qquad (1.2)$$

where $f(x,y)$ and $r(x,y)$ denote given functions on the domain Ω and on its boundary $\partial\Omega$, respectively. Here, the domain $\Omega \subset \mathbb{R}^2$ is assumed to be a bounded, open, simply connected, and convex set in 2 dimensions with piecewise smooth boundary $\partial\Omega$.

The FEM solution u_h will typically incur an error against the PDE solution u of (1.1)–(1.2). This error can be quantified by bounding the norm of the error $u - u_h$ in terms of the mesh spacing h of the finite element mesh. Such estimates have the form $\|u - u_h\| \leq C\,h^q$, where C is a problem-dependent constant independent of h and the constant q indicates the order of convergence of the FEM, as the mesh spacing h decreases. We see from this form of the error estimate that we need $q > 0$ for convergence as $h \to 0$. More realistically, we wish to have for instance $q = 1$ for linear convergence, $q = 2$ for quadratic convergence, or higher values for even faster convergence.

One appropriate norm for FEM errors is the $L^2(\Omega)$-norm associated with the space $L^2(\Omega)$ of square-integrable functions, that is, the space of all functions $v(\mathbf{x})$ whose square $v^2(\mathbf{x})$ can be integrated over all $\mathbf{x} \in \Omega$ without the integral becoming infinite. The norm is defined concretely as the square root of that integral, namely

$$\|v\|_{L^2(\Omega)} := \left(\int v^2(\mathbf{x})\, d\mathbf{x} \right)^{1/2}. \qquad (1.3)$$

Using the L^2-norm to measure the error of the FEM allows the computation of norms of errors also in cases where the solution and its error do not have derivatives. Lagrange finite elements of degree p, such as available in COMSOL with $p = 1, \ldots, 5$, approximate the PDE solution at several points in each element of the mesh such that the restriction of the FEM solution u_h to each element is a polynomial of degree up to p in each spatial variable and u_h is continuous across all boundaries between neighboring mesh elements throughout Ω. For the case of linear (degree $p = 1$) Lagrange elements, we have the well known a priori bound (e.g., [1, Section II.7])

$$\|u - u_h\|_{L^2(\Omega)} \leq C\,h^2$$

provided $u \in H^2(\Omega)$. The assumption on u is ensured if the right-hand side of the PDE (1.1) satisfies $f \in L^2(\Omega)$. We notice that the convergence order is one higher than the polynomial degree used by the Lagrange elements. Analogously, a more general result for using Lagrange elements with degrees $p \geq 1$ is that we can expect an error bound of [5, Section 6.2.1]

$$\|u - u_h\|_{L^2(\Omega)} \leq C\, h^{p+1}. \qquad (1.4)$$

The first purpose of this note is to demonstrate numerically that for an appropriate example this behavior can be observed for the Lagrange elements with all possible orders $p = 1, \ldots, 5$ available in COMSOL; this is the contents of Section 3.3. The second purpose is to explain in detail how to conduct these convergence studies in COMSOL 4.0. An update of this note for COMSOL 4.0a and extension to using LiveLink for MATLAB is the topic of [6].

2 Elliptic Test Problem

In this section, we consider the classical elliptic test problem on a polygonal domain, which can be partitioned into the finite element mesh without error. Specifically, we choose the square domain $\Omega = (0, 1) \times (0, 1) \subset \mathbb{R}^2$ and supply the right-hand side of (1.1) as

$$\begin{aligned} f(x, y) = &-2\pi^2 \cos(2\pi x) \sin^2(\pi y) \\ &-2\pi^2 \sin^2(\pi x) \cos(2\pi y), \end{aligned} \qquad (2.1)$$

and the homogeneous Dirichlet boundary condition of (1.2) as

$$r(x, y) = 0. \qquad (2.2)$$

This problem has been chosen as it has the known PDE solution [4]

$$u(x, y) = \sin^2(\pi x) \sin^2(\pi y). \qquad (2.3)$$

The test problem with (2.1) and (2.2) is appropriate to demonstrate the convergence of the FEM for all possible orders $p = 1, \ldots, 5$ of Lagrange elements in COMSOL, since the right-hand side $f \in L^2(\Omega)$, which guarantees that $u \in H^2(\Omega)$. A larger study that extends convergence studies of this type to non-smooth problems to demonstrate the mathematical assumptions of (1.4) was reported in [3].

By selecting a test problem which has a known PDE solution u, a direct computation of the error $u - u_h$ and its norm in (1.4) can be achieved. The convergence order q is then estimated from these computational results by the following steps: Starting from some initial mesh, we refine it uniformly repeatedly, which subdivides every triangle into four triangles. If h measures the maximum side length of all triangles, this procedure halves the value of h in each refinement. Let r denote the number of refinement levels from the initial mesh and $E_r := \|u - u_h\|_{L^2(\Omega)}$ the error norm on that level. Then assuming that $E_r = C\, h^q$, the error for the next coarser mesh with mesh spacing $2h$ is $E_{r-1} = C\, (2h)^q = 2^q\, C\, h^q$. Their ratio is then $R_r = E_{r-1}/E_r = 2^q$ and $Q_r = \log_2(R_r)$ provides us with a computable estimate for q in (1.4) as $h \to 0$. Notice that the technique described here uses the known PDE solution u; this is in contrast to the technique described in [2] that worked for Lagrange elements with $p = 1$ without knowing the PDE solution u.

In Table 1 we list for each refinement level r, the number of elements N_e in the mesh, the degrees of freedom (DOF), the FEM error $E_r = \|u - u_h\|_{L^2(\Omega)}$, the ratio of error for consecutive refinements $R_r = 2^q$, and the estimate $Q_r = \log_2(R_r)$ for the convergence order computed as described in the Introduction. As expected, the computed error tends to zero as the number of refinements increases. In addition for a given refinement level, the error is smaller for higher order Lagrange elements. We observe that through $p = 1, \ldots, 5$ the convergence order estimate Q_r is consistent with the predicted value $q = p + 1$.

3 Use of COMSOL 4.0

In this section, we demonstrate how to conduct the convergence studies from the previous section using the GUI of COMSOL 4.0. For convenience, the instructions for this process are divided into three subsections. In Subsection 3.1, the test problem will be solved up to the point of the default plot created after solving it. Subsection 3.2 outlines the post-processing of the solution including changing the appearance of the solution plot and computing the FEM error by domain integration. Lastly in Subsection 3.3, we describe how to solve the same problem repeatedly on progressively finer meshes to obtain the convergence study and how to mod-
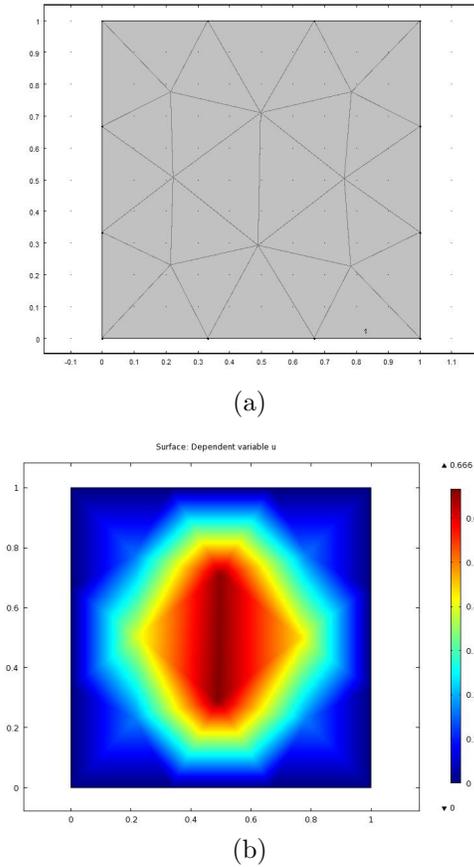
(a)

(b)

Figure 1: Elliptic test problem on polygonal domain: (a) extremely coarse mesh, (b) FEM solution.

ify the solution process to obtain the convergence studies for other degrees of the Lagrange elements. This section also motivates the idea of saving the entire solution process to an mph-file at the appropriate moment, so as to have it available for solving again.

Start the GUI of COMSOL by typing `comsol` at the Linux prompt or double clicking the COMSOL Multiphysics icon on a Windows operating system.

3.1 Solution in the GUI

This section gives step-by-step instructions how to solve the elliptic test problem from Section 2 in COMSOL's GUI.

1. Once the GUI loads, under the Model Wizard Window in the central window pane of the GUI, choose 2D on the Select a Space Di-

mension page. In order to proceed, click the Next button (right arrow) on the toolbar of this page. The Add Physics page replaces the Model Wizard in the center pane.

2. On the Add Physics page, expand the Mathematics branch (by clicking on the down arrow to the left of the label) and then the PDE Interfaces branch, and select the Coefficient Form PDE node. Click the Add Selected button (plus sign below window). By default, the number of dependent variables is one and the variable name is u. Since this is the desired setup for the problem, click the Next button (right arrow).

3. Under the Select Study page, select Stationary and click the Finish button (checkered flag) on the toolbar of this page.

4. Before proceeding to establish the specifics of the test problem, check to ensure that all needed information will easily be displayed. In the Model Builder window in the left pane of the GUI, click the View Menu (upside down triangle) on the toolbar and make sure that Show More Options is properly checked; this setting is saved from one COMSOL session to the next, so once this is selected, COMSOL will retain this selection for future restarts.

5. In order to set up the desired domain, right click **Geometry 1** and select Square in the Model Builder window. By default, this will generate the desired square domain $\Omega = (0,1) \times (0,1)$ with one corner of the square at the origin.

6. In the Model Builder window in the left window pane, the right-hand side of the PDE can be set by expanding the PDE branch and selecting the **Coefficient Form PDE 1** node. The center pane of the GUI specifies the general form of the equation currently selected as

$$e_a \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} + \nabla \cdot (-c\nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + au = f.$$

Under Source Term, enter for f the expression **(-2*pi^2)*(cos(2*pi*x)*sin(pi*y)^2 +sin(pi*x)^2*cos(2*pi*y))**. Also, set the Coefficient d_a to zero to recover a Stationary problem. Leave the other coefficients as their default values in order to establish the Poisson equation of 1.1

3

7. The desired boundary conditions of the test problem can be generated by right clicking the PDE branch in the Model Builder window in the left pane and selecting Dirichlet Boundary Conditions. Select the **Dirichlet Boundary Condition 1** branch in the Model Builder window, then in the Dirichlet Boundary Condition page in the center pane under Boundaries, choose All boundaries under Selection.

8. Again in the Model Builder window in the left pane, select the PDE branch and on the PDE page in the center pane under Discretization (you might have to expand Discretization first), choose 1 for the Element order. This establishes the degree of the Lagrange elements used. By selecting the element order to be 1, COMSOL will use linear Lagrange elements in the finite element solution.

9. In order to generate the FEM mesh that will be used to compute the FEM solution, first right click the **Mesh 1** branch under the Model Builder window and select **Free Triangular** to establish the mesh. On the Free Triangular page in the center pane, under the Domains item, select for Geometric entity level the selection Domain. Under the **Mesh 1** branch, select the **Size** node and on the Size page under Element Size, choose **Extremely coarse** for the Predefined Elements Size. In order to see the mesh being used, right click the **Mesh 1** branch and choose Build All. Figure 1 (a) displays the extremely coarse mesh that will be used to compute the FEM solution. The number of triangular elements used in this mesh can be determined by right clicking the **Mesh 1** branch and choosing **Statistics**. For this domain and extremely coarse mesh, the number of triangular elements are 26.

10. Now, compute the FEM solution by right clicking the **Study 1** branch under the Model Builder window and selecting Compute. Alternatively, one can click the green equal symbol above the Study page on the toolbar. Once the solution is computed, the degrees of freedom which have been solved for can be seen below the Graphics window in the Messages tab, which is 20 for this coarse mesh using linear Lagrange (element order 1) elements.
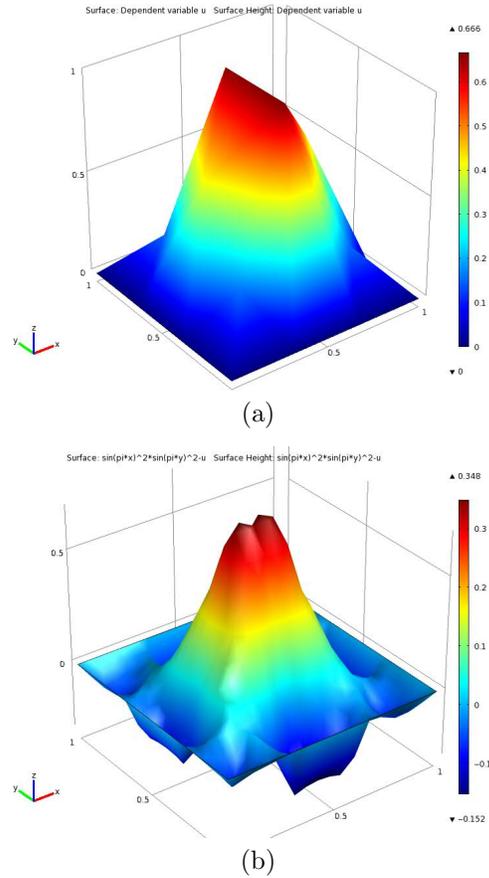

(a)

(b)

Figure 2: Post-processing of FEM solution: (a) three-dimensional view of solution , (b) three-dimensional view of error.

3.2 Post-Processing

Solving the problem in the GUI leads to the solution in a default plot shown in Figure 1 (b). A more conventional way might be to present the solution in a three-dimensional view. This subsection gives instructions on how to post-process the FEM solution obtained in the previous subsection by changing the plot to a three-dimensional view and by computing the FEM error.

1. Under the Results branch of the Model Builder window, expand the **2D Plot Group 1** branch, right click on the **Surface 1** node and choose the Height Expression. This shows a three-dimensional surface and height plot of the FEM solution $u_h(x,y)$. The result is shown in Figure 2 (a). The previous subsection specifically used linear ($p = 1$) Lagrange

elements to solve the problem, which means that the FEM solution $u_h(x, y)$ is a flat patch on each triangle of the mesh. This is clearly visible in Figure 2 (a).

2. In order to construct a plot of the FEM error $u - u_h$, right click the **Results** branch and choose 2D Plot Group. This creates a second plot group called **2D Plot Group 2**. Right Click the **2D Plot Group 2** and select Surface. This creates the node **Surface 1** under the **2D Plot Group 2** branch. Select this **Surface 1** node and on the Surface page under expression, type the formula for the error which is the difference between the PDE solution and the FEM solution: **sin(pi*x)^2*sin(pi*y)^2-u**. Then right click this **Surface 1** node and select the Height Expression. Figure 2 (b) shows a three-dimensional surface and height plot of the error.

3. The convergence studies of the FEM solution rely on the $L^2(\Omega)$-norm $E_r = \|u - u_h\|_{L^2(\Omega)}$ of the FEM error with the norm defined in (1.3) with $v = u - u_h$. COMSOL can compute the integral $\int (u - u_h)^2 \, d\mathbf{x}$ that appears in this norm definition; this integral is then the square E_r^2 of the desired error norm E_r. To compute this integral, right click the **Derived Values** branch on the Model Builder and select Surface Integration. Choose all domains under Selection on the Surface Integration Page. Below expression, type the square $(u - u_h)^2$ of the error as **(sin(pi*x)^2*sin(pi*y)^2-u)^2**. Click to check mark the Description and label this quantity by typing E_r^2 to indicate that it is the square of the norm of the error. Now, right click the **Surface Integration** node on the Model Builder window and select Evaluate and New Table. The result of the computation is 0.0116 and shown in the Results tab under the graphics area.

4. It is useful to save the solution process as a COMSOL mph-file at this stage before mesh refinements to have it available as starting point later when considering higher order Lagrange elements. Under the **File** menu, choose **Save As ...**. This will automatically save as an mph-file and append the extension `mph` to the chosen filename.

3.3 Convergence Studies

In this subsection, we make use of the steps discussed in Subsection 3.2 in order to carry out a convergence study. We repeatedly refine the mesh that was used to compute the FEM solution, recompute the solution and its error norm, and then copy all calculated error norms.

1. Refine the mesh by right clicking the **Mesh 1** branch and under More Operations select Refine. Under refinement, type 1 and rebuild the mesh by right clicking the **Mesh 1** branch and selecting Rebuild All. Again, check the statistics by right clicking the **Mesh 1** branch and selecting **Statistics**. With one refinement, the number of triangular nodes has increased by a factor of 4 to a total of 104 elements. Recompute the FEM solution under this refinement by right clicking the **Study 1** branch and selecting Compute. Once the solution is computed, right click the **Surface Integration** node and select Evaluate and choose **Table 1** to add the result to the previous created table. Continue this process through several refinements. The Results tab under the graphics window accumulates all results for E_r^2 over the course of these refinements.

2. After following the above procedure through 4 consecutive refinements, we can copy the data for the squares E_r^2 of the FEM errors from the table under the Results tab into some other software, such as MATLAB, for further processing. This is how the E_r^2 column in Table 1 was obtained. The remaining columns in this table can be readily computed using the quantites R_r and Q_r defined in Section 1.

Following the previous steps in this section provides a convergence study for the Lagrange elements of order $p = 1$. In order to perform convergence studies for higher order Lagrange elements, start from the mph-file from the end of Section 3.2 that was saved before any mesh refinements. From the **File** menu, choose **Open** to load the file. Once the file is loaded, expand the **Model 1** branch, then select the **PDE** branch, and change the order of the Lagrange element being used under the Discretization to a different order. By retracing the mesh refinement steps of this section, the values shown in Table 1 can be obtained for all degrees $p = 1, \ldots, 5$.

Acknowledgments

References

[1] Dietrich Braess. *Finite Elements*. Cambridge University Press, third edition, 2007.

[2] Matthias K. Gobbert. A technique for the quantitative assessment of the solution quality on particular finite elements in COMSOL Multiphysics. In Vineet Dravid, editor, *Proceedings of the COMSOL Conference 2007, Boston, MA*, pp. 267–272, 2007.

[3] Matthias K. Gobbert and Shiming Yang. Numerical demonstration of finite element convergence for Lagrange elements in COMSOL Multiphysics. In Vineet Dravid, editor, *Proceedings of the COMSOL Conference 2008, Boston, MA*, 2008.

[4] Noemi Petra and Matthias K. Gobbert. Parallel performance studies for COMSOL Multiphysics using scripting and batch processing. In Yeswanth Rao, editor, *Proceedings of the COMSOL Conference 2009, Boston, MA*, 2009.

[5] Alfio Quarteroni and Alberto Valli. *Numerical Approximation of Partial Differential Equations*, vol. 23 of *Springer Series in Computational Mathematics*. Springer-Verlag, 1994.

[6] David W. Trott and Matthias K. Gobbert. Finite element convergence studies using COMSOL 4.0a and LiveLink for MATLAB. Technical Report HPCF–2010–8, UMBC High Performance Computing Facility, University of Maryland, Baltimore County, 2010.

Table 1: Convergence study for the elliptic test problem using Lagrange elements.

(a) Lagrange elements with $p = 1$

r	N_e	DOF	E_r^2	E_r	R_r	Q_r
0	26	20	1.160e–02	1.077e–01	NA	NA
1	104	65	7.033e–04	2.652e–02	4.06	2.02
2	416	233	4.501e–05	6.709e–03	3.95	1.98
3	1664	881	2.836e–06	1.684e–03	3.98	1.99
4	6656	3425	1.775e–07	4.214e–04	3.99	1.99

(b) Lagrange elements with $p = 2$

r	N_e	DOF	E_r^2	E_r	R_r	Q_r
0	26	65	4.351e–05	6.596e–03	NA	NA
1	104	233	1.259e–06	1.122e–03	5.88	2.56
2	416	881	2.076e–08	1.441e–04	7.79	2.96
3	1664	3425	3.294e–10	1.815e–05	7.94	2.99
4	6656	13505	5.180e–12	2.276e–06	7.97	3.00

(c) Lagrange elements with $p = 3$

r	N_e	DOF	E_r^2	E_r	R_r	Q_r
0	26	136	6.991e–06	2.644e–03	NA	NA
1	104	505	2.031e–08	1.425e–04	18.56	4.21
2	416	1945	7.460e–11	8.637e–06	16.50	4.04
3	1664	7633	2.834e–13	5.327e–07	16.22	4.02
4	6656	30241	1.095e–15	3.309e–08	16.10	4.01

(d) Lagrange elements with $p = 4$

r	N_e	DOF	E_r^2	E_r	R_r	Q_r
0	26	233	6.634e–09	8.145e–05	NA	NA
1	104	881	1.467e–11	3.830e–06	21.27	4.41
2	416	3425	1.578e–14	1.256e–07	30.49	4.93
3	1664	13505	1.605e–17	4.006e–09	31.36	4.97
4	6656	53633	1.595e–20	1.263e–10	31.71	4.99

(e) Lagrange elements with $p = 5$

r	N_e	DOF	E_r^2	E_r	R_r	Q_r
0	26	356	7.656e–10	2.767e–05	NA	NA
1	104	1361	1.421e–13	3.770e–07	73.39	6.20
2	416	5321	3.421e–17	5.849e–09	64.45	6.01
3	1664	21041	8.306e–21	9.114e–11	64.174	6.00
4	6656	83681	1.819e–24	1.349e–12	67.58	6.08