

# FEM Convergence Studies for 2-D and 3-D Elliptic PDEs with Smooth and Non-Smooth Source Terms in COMSOL 5.1

Kourosh M. Kalayeh<sup>1</sup>, Jonathan S. Graf<sup>2</sup>, and Matthias K. Gobbert<sup>2</sup> (gobbert@umbc.edu)

<sup>1</sup>Department of Mechanical Engineering, University of Maryland, Baltimore County

<sup>2</sup>Department of Mathematics and Statistics, University of Maryland, Baltimore County

Technical Report HPCF-2015-19, [hpcf.umbc.edu](http://hpcf.umbc.edu) > Publications

## Abstract

Numerical theory provides the basis for quantification on the accuracy and reliability of a FEM solution by error estimates on the FEM error vs. the mesh spacing of the FEM mesh. This paper presents techniques needed in COMSOL 5.1 to perform computational studies for an elliptic test problem in two and three space dimensions that demonstrate this theory by computing the convergence order of the FEM error. For a PDE with smooth right-hand side, linear Lagrange finite elements exhibit second order convergence for all space dimensions. We also show how to perform these techniques for a problem involving a point source modeled by a Dirac delta distribution as forcing term. This demonstrates that PDE problems with a non-smooth source term necessarily have degraded convergence order and thus can be most efficiently solved by low-order FEM such as linear Lagrange elements. Detailed instructions for obtaining the results are included in an appendix.

**Keywords:** Poisson equation, point source, Dirac delta distribution, convergence study, mesh refinement.

## 1 Introduction

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

$$\begin{aligned} -\Delta u &= f && \text{in } \Omega, \\ u &= r && \text{on } \partial\Omega, \end{aligned} \tag{1}$$

where  $f(\mathbf{x})$  and  $r(\mathbf{x})$  denote given functions on the domain  $\Omega$  and on its boundary  $\partial\Omega$ , respectively. Here, we will consider the domain  $\Omega = (-1, 1)^d \subset \mathbb{R}^d$  is assumed to be a bounded, open, simply connected, convex set in  $d = 2$  or  $3$  space dimensions with a polygonal boundary  $\partial\Omega$ .

The FEM solution  $u_h$  will typically incur an error against the PDE solution  $u$  of the problem (1)–(2). This 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, e.g., [1, Section II.7],

$$\|u - u_h\|_{L^2(\Omega)} \leq C h^q, \quad \text{as } h \rightarrow 0, \tag{3}$$

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  $h \rightarrow 0$ . More realistically, we wish to have for instance  $q = 1$  for linear convergence,  $q = 2$  for quadratic convergence, etc., where higher values yield faster convergence.

The norm  $\|u - u_h\|_{L^2(\Omega)}$  in (3) is the  $L^2$ -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 becoming infinite. The norm is defined concretely as the square root of that integral, namely  $\|v\|_{L^2(\Omega)} := \left(\int v^2(\mathbf{x}) d\mathbf{x}\right)^{1/2}$ .

The result stated in (3) necessitates requirements on the finite element method used as well as on the PDE problem (1)–(2):

- Lagrange finite elements, such as those in COMSOL Multiphysics, approximate the PDE solution  $u$  at several points in each element  $K$  of a mesh  $\mathcal{T}_h$ , such that the restriction of  $u_h$  to each element  $K$  is a polynomial of degree up to  $p$  and  $u_h$  is continuous across all boundaries between neighboring

elements throughout  $\Omega$ . For the case of linear Lagrange elements with piecewise polynomial degree  $p = 1$ , the convergence order is  $q = 2$  in (3), i.e., one higher than the polynomial degree; it also holds under additional assumptions on the PDE problem that for higher-order elements with degree  $p$ , the convergence order in (3) can reach  $q = p + 1$ .

- One necessary assumption on the PDE is that the problem has a solution that is sufficiently regular, as expressed by the number of derivatives it has. In the context of the FEM, it is appropriate to consider weak derivatives [1]. Based on these, we define the Sobolev function spaces  $H^k(\Omega)$  of order  $k$  of all functions on  $\Omega$  that have weak derivatives up to order  $k$  that are square-integrable in the sense of  $L^2(\Omega)$  above. The convergence order  $q$  in (3) of the FEM with Lagrange elements with degree  $p$  is then limited by the regularity order  $k$  of the PDE solution as  $q \leq k$ .

These two facts above can be combined into the formula  $q = \min\{k, p + 1\}$  in (3). Thus, for linear Lagrange elements for instance, we need  $u \in H^k(\Omega)$  with  $k = 2$  to reach the optimal convergence order of  $q = p + 1 = 2$ .

The purpose of this paper is to show how one can demonstrate computationally the convergence order  $q = p + 1 = 2$  for linear Lagrange FEM elements, if the PDE solution  $u$  is smooth enough (i.e.,  $k \geq 2$ ). Moreover, we demonstrate that the convergence order is indeed limited by  $q \leq k$ , if the solution is not smooth enough (i.e.,  $k < 2$ ). This latter situation arises concretely when considering a PDE with one (or more) point sources in the forcing term  $f(\mathbf{x})$  on the right-hand side of (1). The reason is that the mathematical model for point sources is given by the Dirac delta distribution  $\delta(\mathbf{x})$ . This function is not square-integrable, and thus the PDE solution  $u$  is not in  $H^2(\Omega)$ .

In Section 2, we explain the numerical method applied to both a smooth and a non-smooth problem in the following Sections 3 and 4, respectively. Specifically, Section 2 specifies how to initialize convergence studies with the coarsest meshes possible in two and three space dimensions and how to develop a computable estimate  $q^{(\text{est})}$  of the convergence order  $q$  in (3) by regular mesh refinements. Section 3 shows the results for the convergence studies with smooth PDE solutions, which demonstrate that  $q = 2$  in both two and three space dimensions of the PDE domain  $\Omega \subset \mathbb{R}^d$ . By contrast, Section 4 shows that for (1) with  $f = \delta$  as forcing term, the convergence is limited in a dimension-dependent way, namely to  $q = 1.0$  and  $q = 0.5$  in two and three dimensions, respectively. In Appendix A, we provide detailed instructions for obtaining the results of this report in COMSOL 5.1.

## 2 Numerical Method

One well-known, practical test for reliability of a FEM solution is to refine the FEM mesh, compute the solution again on the finer mesh, and compare the solutions on the two meshes qualitatively. The FEM theory provides a quantification of this approach by comparing FEM errors  $u - u_h$  involving the PDE solution  $u$  compared to FEM solutions on meshes, whose mesh spacings  $h$  are related by uniform refinement. We use this theory here for linear Lagrange elements as provided in COMSOL Multiphysics. Since the domain  $\Omega = (-1, 1)^d$  has a polygonal boundary, it can be discretized by the triangular meshes in  $d = 2$  and by tetrahedral meshes in  $d = 3$  dimensions without error. The convergence studies performed rely on a sequence of meshes with mesh spacings  $h$  that are halved in each step. This is accomplished by uniformly refining an initial mesh repeatedly, starting from a minimal initial mesh to allow as many refinements as possible. For the initial mesh, we take advantage of the shape of  $\Omega$  that admits a very coarse, uniform mesh that still includes the origin  $\mathbf{x} = 0$  as a mesh point, which is needed later for the non-smooth problems in Section 4. In  $d = 2$  dimensions, the initial mesh consists of 4 triangles with 5 vertices given by the 4 corners of  $\Omega$  plus the center point. In  $d = 3$  dimensions, the initial mesh has 28 tetrahedra with 15 vertices. Figures 1 (a) and (b) show the initial meshes for the two- and three-dimensional domains, respectively. Figures 2 (a) and (b) show the exploded view of the initial mesh in three-dimensional domain. More specifically, while Figure 2 (a) shows the exploded view of initial mesh for whole domain, Figure 2 (b) shows the exploded view with some elements removed so that we can view the inside elements of the domain and confirm that the origin is indeed a point in the discretization. Specifically, the 16 of the 28 tetrahedra shown in Figure 2 (a) confirm that the origin is indeed a point in the discretization. Instructions for generating these mesh views are included in Appendix A.

The initial meshes are then refined uniformly  $r = 1, 2, \dots$  times. For each of the meshes considered, we track the number of mesh elements  $N_e$ , the degrees of freedom (DOF)  $N$  of the linear nodal elements for

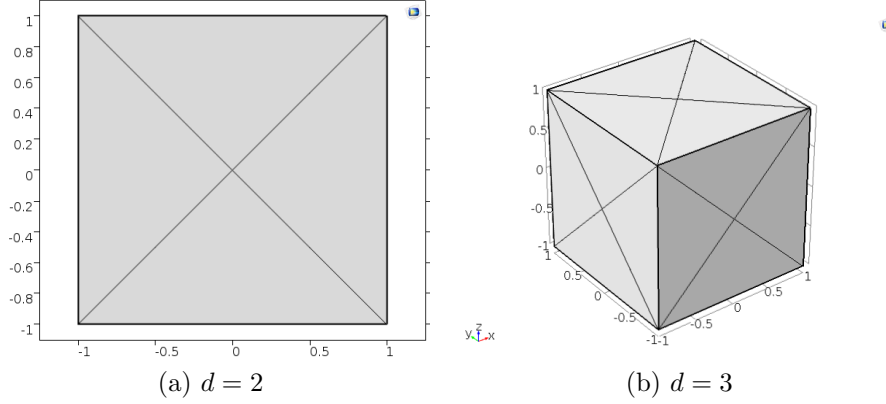


Figure 1: Initial mesh in (a) two dimensions, (b) three dimensions.

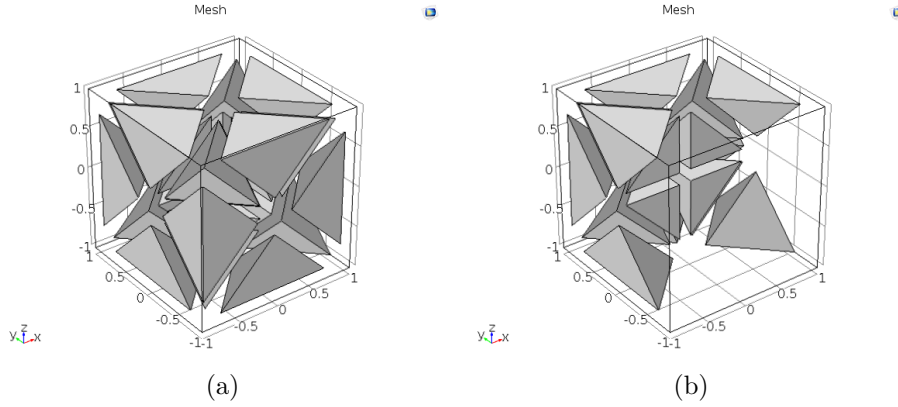


Figure 2: Exploded view of the initial mesh in three dimensions (a) for whole domain, (b) elements on one side removed to show interior elements of the domain.

that mesh, and the mesh spacing  $h$  for each refinement level  $r$  from the initial mesh for  $r = 0$  to the finest mesh explored, as summarized in Tables 1 (a) and (b). The solution plots in the following sections use the mesh with refinement level  $r = 5$ .

All test problems are designed to have a known true PDE solution  $u(\mathbf{x}) = u_{\text{true}}(\mathbf{x})$  to allow for a direct computation of the error  $u - u_h$  against the FEM solution and its norm in (3). 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. If the mesh spacing  $h$  is defined as the maximum side length of all elements, i.e.,  $h := \max_e h_e$ , this procedure halves the value of  $h$  in each refinement. In two dimensions, uniform mesh refinement sub-divides every triangle into 4 triangles. In three dimensions, the same halving of  $h$  occurs, however the uniform refinement sub-divides each tetrahedral element into 8 smaller tetrahedra. The number of elements  $N_e$  as well as the observed mesh spacing  $h$  in Tables 1 (a) and (b) exhibit the expected behavior associated with uniform mesh refinement. Let  $r$  denote the number of refinement levels from the initial mesh and define  $E_r := \|u - u_h\|_{L^2(\Omega)}$  as the error norm on refinement level  $r = 1, 2, \dots$ . Then assuming that  $E_r = C h^q$  according to (3), 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  $q^{(\text{est})} = \lim_{r \rightarrow \infty} Q_r$  for  $q$  in (3) as  $h \rightarrow 0$ .

Table 1: Finite element data for all meshes in dimensions  $d = 2$  and 3 for all refinement levels  $r$ .

$r$	$N_e$	$N = \text{DOF}$	$h$	$r$	$N_e$	$N = \text{DOF}$	$\max_e h_e$
0	4	5	2.000000	0	28	15	2.000000
1	16	13	1.000000	1	224	69	1.000000
2	64	41	0.500000	2	1,792	409	0.500000
3	256	145	0.250000	3	14,336	2,801	0.250000
4	1,024	545	0.125000	4	114,688	20,705	0.125000
5	4,096	2,113	0.062500	5	917,504	159,169	0.062500

(a)  $d = 2$

(b)  $d = 3$

### 3 Smooth Test Problems

For the smooth test problems, the right-hand side of Poisson equation  $f(\mathbf{x})$  in (1) is chosen as

$$f(\mathbf{x}) = \begin{cases} \frac{\pi}{2} \left( \frac{1}{\rho} \sin \frac{\pi \rho}{2} + \frac{\pi}{2} \cos \frac{\pi \rho}{2} \right) & \text{for } d = 2, \\ \frac{\pi}{2} \left( \frac{2}{\rho} \sin \frac{\pi \rho}{2} + \frac{\pi}{2} \cos \frac{\pi \rho}{2} \right) & \text{for } d = 3, \end{cases} \quad (4)$$

where  $\rho = \sqrt{x^2 + y^2}$  in 2-D and  $\rho = \sqrt{x^2 + y^2 + z^2}$  in 3-D. This function satisfies the standard assumption of  $f \in L^2(\Omega)$  using in classical FEM theory [1, Chapter II]. This classical theory provides that  $u$  is two orders smoother, that is,  $u \in H^2(\Omega)$ , since  $L^2(\Omega) \equiv H^0(\Omega)$  formally. The problems are chosen such that we know the true solution  $u_{\text{true}}(\mathbf{x})$ . Using this fact, the Dirichlet boundary condition function  $r(\mathbf{x})$  in (2) is indeed chosen equal to the true solution, thus the equation

$$r(\mathbf{x}) = u_{\text{true}}(\mathbf{x}) = \begin{cases} \cos \frac{\pi \sqrt{x^2 + y^2}}{2} & \text{for } d = 2, \\ \cos \frac{\pi \sqrt{x^2 + y^2 + z^2}}{2} & \text{for } d = 3. \end{cases} \quad (5)$$

lists both functions. Figures 3 (a) and (b) show two views of the FEM solution for mesh refinement  $r = 5$ .

The true PDE solutions  $u_{\text{true}}(\mathbf{x})$  in (5) are infinitely often differentiable in the classical sense, and hence the regularity order  $k$  does not limit the predicted convergence order  $q = \min\{k, p + 1\}$  for any degree  $p$  of the Lagrange elements. Specifically for linear Lagrange elements with degree  $p = 1$ , we expect to see  $q = 2$  as convergence order for all spatial dimensions  $d = 2, 3$ . Table 2 lists for each refinement level  $r$  the error  $E_r = \|u_h(\cdot, t) - u(\cdot, t)\|_{L^2(\Omega)}$  of (3) and in parentheses the estimate  $Q_r$  according to Section 2. We

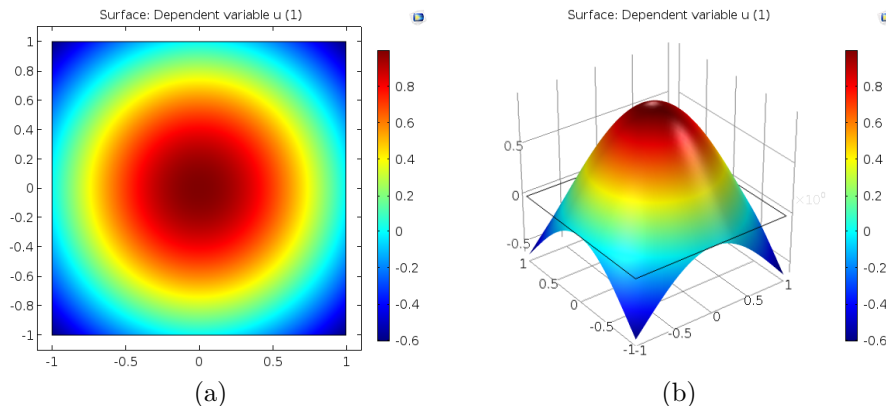


Figure 3: (a) Two-dimensional and (b) three-dimensional view of the FEM solution for the smooth test problem with  $r = 5$ .

Table 2: Convergence studies for the smooth test problem in two and three dimensions.

$r$	$E_r (Q_r)$	$r$	$E_r (Q_r)$
0	1.105	0	1.132
1	3.049e-01 (1.86)	1	3.481e-01 (1.70)
2	8.387e-02 (1.86)	2	9.007e-02 (1.95)
3	2.177e-02 (1.95)	3	2.273e-02 (1.99)
4	5.511e-03 (1.98)	4	5.690e-03 (2.00)
5	1.383e-03 (1.99)	5	1.422e-03 (2.00)

(a)  $d = 2$ 
(b)  $d = 3$

observe that  $Q_r$  approaches the value  $q^{(\text{est})} = 2$ , which is expected for a smooth source term for any spatial dimension  $d = 2, 3$ .

## 4 Non-Smooth Test Problems

For the non-smooth test problems, the forcing term  $f(\mathbf{x})$  in (1) is chosen to model a point source. This is mathematically modeled by setting the forcing term as the Dirac delta distribution  $f(\mathbf{x}) = \delta(\mathbf{x})$ . The Dirac delta distribution models a point source at  $\hat{\mathbf{x}} \in \Omega$  mathematically by requiring  $\delta(\mathbf{x} - \hat{\mathbf{x}}) = 0$  for all  $\mathbf{x} \neq \hat{\mathbf{x}}$ , while simultaneously satisfying  $\int \varphi(\mathbf{x}) \delta(\mathbf{x} - \hat{\mathbf{x}}) d\mathbf{x} = \varphi(\hat{\mathbf{x}})$  for any continuous function  $\varphi(\mathbf{x})$ . Based on the weak formulation of the problem, the finite element method is able to handle the source at  $\hat{\mathbf{x}} = 0$  modeled by  $f(\mathbf{x}) = \delta(\mathbf{x})$  in (1). That is, when the PDE is integrated in the derivation of the FEM with respect to a continuous test function  $v(\mathbf{x})$ , the right-hand side becomes  $\int_{\Omega} v(\mathbf{x}) \delta(\mathbf{x}) d\mathbf{x} = v(0)$ . If the point  $\hat{\mathbf{x}} = 0$  is chosen as a mesh point of the FEM mesh, then the test function of the FEM with Lagrange elements evaluated at 0 in turn will equal 1 for the FEM basis function  $v(\mathbf{x})$  centered at this mesh point and 0 for all others. This is the background behind the instructions in the COMSOL documentation to implement a point source modeled by the Dirac delta distribution by entering 1 as source function value at that mesh point. These instructions can be located by searching for “point source” in the COMSOL documentation.

Also for this problem exists a closed-form true solution  $u_{\text{true}}(\mathbf{x})$  that we again use for  $r(\mathbf{x})$  in the Dirichlet boundary condition in (2), so that

$$r(\mathbf{x}) = u_{\text{true}}(\mathbf{x}) = \begin{cases} \frac{-\ln \sqrt{x^2+y^2}}{2\pi_1} & \text{for } d = 2, \\ \frac{1}{4\pi\sqrt{x^2+y^2+z^2}} & \text{for } d = 3 \end{cases} \quad (6)$$

lists both the boundary function and the true solution. Figures 4 (a) and (b) show two views of the FEM solution for mesh refinement  $r = 5$ . Notice that the true solutions have a singularity at the origin 0, where they tend to infinity. Thus, the solutions are not differentiable everywhere in  $\Omega$  and thus not in any space of continuous or continuously differentiable functions. However, recall the Sobolev Embedding Theorem [2]. Since  $\int \varphi(\mathbf{x}) \delta(\mathbf{x}) d\mathbf{x} = \varphi(0)$  for any continuous function  $\varphi(\mathbf{x})$ , and the Sobolev space  $H^{d/2+\varepsilon}$  is continuously embedded in the space of continuous function  $C^0(\Omega)$  in  $d = 1, 2, 3$  dimensions for any  $\varepsilon > 0$ , one can argue that  $\delta$  is in the dual space of  $\varphi \in H^{d/2+\varepsilon}(\Omega)$ , that is,  $\delta \in H^{-d/2-\varepsilon}(\Omega)$ . Since the solution  $u$  of this second-order elliptic PDE is two orders smoother, we obtain the regularity  $u \in H^{2-d/2-\varepsilon}(\Omega)$  or  $k \approx 2 - d/2$  in (3), which suggests to expect a dimension-dependent convergence order  $q = 2 - d/2$  for  $d = 2, 3$  dimensions [3, 4].

Table 3 presents the error norm  $E_r$  as well as the observed  $Q_r$  in parentheses. For  $d = 2$  dimensions in Table 3 (a), we see that  $Q_r$  approaches the value  $q^{(\text{est})} = 1.0$ , while for  $d = 3$  dimensions in Table 3 (b), it approaches the value 0.5. This is in agreement with the theory which predicts that the value should be  $q = 2 - d/2$  in  $d = 2, 3$  dimensions.

## 5 Conclusions

In this paper, the test problems are Poisson equations with smooth or non-smooth solutions. The domains are chosen as open square in 2-D and cube in 3-D, which possess polygonal boundaries. With true solution

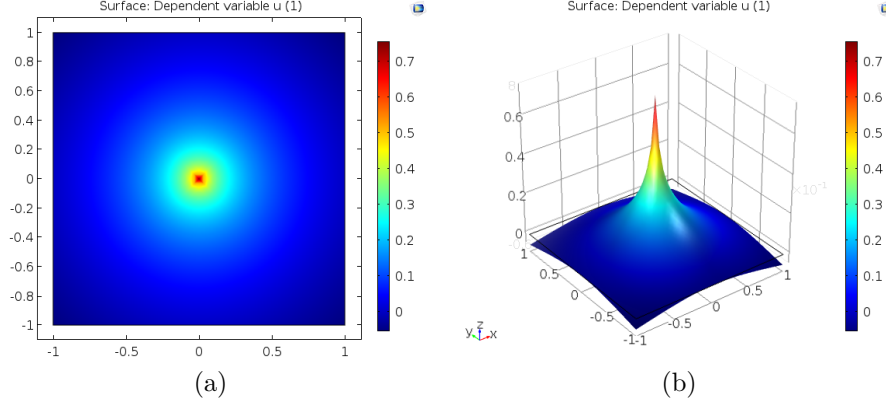


Figure 4: (a) Two-dimensional and (b) three-dimensional view of the FEM solution for the non-smooth test problem with  $r = 5$ .

Table 3: Convergence studies for the non-smooth test problem in two and three dimensions.

$r$	$E_r(Q_r)$	$r$	$E_r(Q_r)$
0	9.332e-02	0	1.026e-01
1	4.589e-02 (1.02)	1	6.990e-02 (0.55)
2	2.468e-02 (0.89)	2	4.842e-02 (0.53)
3	1.256e-02 (0.97)	3	3.410e-02 (0.51)
4	6.311e-03 (0.99)	4	2.410e-02 (0.50)
5	3.160e-03 (1.00)	5	1.704e-02 (0.50)

(a)  $d = 2$                       (b)  $d = 3$

available, we can calculate errors with the FEM solution against the true solution. From the observed data, it is confirmed that the regularity of solution affects the convergence order. Specifically, for the smooth test problems, the convergence order is 2 in all dimensions  $d = 2, 3$ . In the non-smooth test problems, the results agree with the theoretical expectation that convergence order is reduced in a dimension-dependent way to  $2 - d/2$  in  $d = 2, 3$  dimensions.

## Acknowledgments

The hardware used in the computational studies is part of the UMBC High Performance Computing Facility (HPCF). The facility is supported by the U.S. National Science Foundation through the MRI program (grant nos. CNS-0821258 and CNS-1228778) and the SCREMS program (grant no. DMS-0821311), with additional substantial support from the University of Maryland, Baltimore County (UMBC). See [hpcf.umbc.edu](http://hpcf.umbc.edu) for more information on HPCF and the projects using its resources. Co-author Graf acknowledges financial support as HPCF RA.

## References

- [1] Dietrich Braess. *Finite Elements*. Cambridge University Press, third edition, 2007.
- [2] Michael Renardy and Robert C. Rogers. *An Introduction to Partial Differential Equations*, vol. 13 of *Texts in Applied Mathematics*. Springer-Verlag, second edition, 2004.
- [3] Ridgway Scott. Finite element convergence for singular data. *Numer. Math.*, vol. 21, pp. 317–327, 1973.

- [4] Thomas I. Seidman, Matthias K. Gobbert, David W. Trott, and Martin Kružík. Finite element approximation for time-dependent diffusion with measure-valued source. *Numer. Math.*, vol. 122, no. 4, pp. 709–723, 2012.

## A Instructions for Key Steps

This section contains hints and instructions how to obtain the results presented in this report. We assume here that the reader has basic knowledge, such as how to set up and solve the Poisson equation with Dirichlet boundary conditions (2) on a square domain  $\Omega = (-1, 1)^2 \subset \mathbb{R}^2$  in two dimensions.<sup>1</sup> Here, we present key instructions to reproduce the exact results or key steps that can be confusing.

### A.1 Setup of 3-D Problem and Solution

This section gives step-by-step instructions how to setup and solve the 3 dimensional elliptic test problem with smooth source term from Section 3 in COMSOL’s GUI.

1. Once the GUI loads, choose Model Wizard, then choose 3D on the Select a Space Dimension page. The Model Wizard will take you automatically to the Select Physics page.
2. On the Select Physics page, expand the Mathematics branch (by clicking on the arrow to the left of the label) and then the PDE Interfaces branch, and select the **Coefficient Form PDE** node. Click the Add button. 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 Study button (right arrow).
3. Under the Select Study page, select Stationary and click the Done button (checked box) on the bottom 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 Show Menu (eye with a bar symbol) on the toolbar and make sure that Discretization is checked; this is needed to enable user to change element order as will be discussed in step 10 below. 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 **Block** in the Model Builder window. In the settings window for **Block 1**, under the Position section, while Corner is selected for the Base option, change  $(x, y, z)$  to  $(-1, -1, -1)$ . In the same window, under the Size section set Width, Depth, and Height of the block to  $(2, 2, 2)$ . This will generate the desired cubical domain  $\Omega = (-1, 1) \times (-1, 1) \times (-1, 1)$  with one corner of the cube at  $(-1, -1, -1)$ . Select **Build All** under **Geometry 1** to update the geometry.
6. In order to prepare the model for non-smooth case and obtain similar mesh for both smooth and non-smooth cases, we need to introduce a point in the domain which will be used as point source to implement Dirac delta function,  $\delta(x)$ , in COMSOL. To do this, in the model builder window, right click Geometry 1 branch and from More Primitives menu, select **Point**. Leave the default settings for Point 1 to generate a point at position  $(0, 0, 0)$ . Again, build all objects to update the geometry.
7. In this part, we want to set the appropriate source term and Dirichlet boundary condition as stated in Section 3. In doing so, we will use the COMSOL capability in defining parameters and variables as follows.
  - (a) In the model builder, right click Definitions branch and select **Variables**. In the Variables Settings window, in the Variables table enter **rho** in the Name column. Then, in its corresponding column for Expression enter  **$\text{sqrt}(x^2+y^2+z^2)$** .

---

<sup>1</sup>At UMBC, this knowledge may be gained by attending a software workshop offered by the Center for Interdisciplinary Research and Consulting ([circ.umbc.edu](http://circ.umbc.edu)), which provides *A Tutorial Introduction to COMSOL Multiphysics*, version 5.1.

Table 4: Variables table for solving the Poisson problem (1)–(2) with  $\Omega = (-1, 1)^3$ , smooth source term, and known true solution stated in (4) and (5) respectively in COMSOL 5.1.

Name	Expression	Unit	Description
rho	$\text{sqrt}(x^2+y^2+z^2)$	m	
f	$\text{pi}/2*(2/\text{rho}*\sin(\text{pi}*\text{rho}/2)+\text{pi}/2*\cos(\text{pi}*\text{rho}/2))$		
u_true	$\cos(\text{pi}*\text{rho}/2)$		

- (b) Add another row in the Variables table for **f**, the source term as stated in (4). In its expression column enter  $\text{pi}/2*(2/\text{rho}*\sin(\text{pi}*\text{rho}/2)+\text{pi}/2*\cos(\text{pi}*\text{rho}/2))$  which is desired source term for 3-dimensional problem.
- (c) Do the same procedure as above for **u\_true**, the true solution for 3-dimensional Poisson test problem as stated in (5) which is equal to  $\mathbf{r}=\mathbf{u\_true}$ , Dirichlet boundary condition. Your Variables table in COMSOL should look like Table 4.
8. 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 **f** (this was defined in step 7(b)). 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).

9. The desired boundary conditions of the test problem can be generated by right clicking the Coefficient Form PDE node 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 Settings window in the center pane under Boundaries, choose All boundaries under Selection. Then, under the Dirichlet Boundary Condition section, make sure the Prescribed value of  $u$  is checked and enter **u\_true** (which was defined in step 7(c)) for  $r$ .
10. 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 Linear for the Element order. This establishes the degree of the Lagrange elements used. By selecting the element order to be Linear, COMSOL will use linear Lagrange elements in the finite element solution.
11. 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 Tetrahedral** to establish the mesh. On the Free Tetrahedral Settings window in the center pane, under the Domains Selection, select for Geometric entity level the selection Domain. Then, for Selection, choose All domains. Under the **Mesh 1** branch, select the **Size** node and on the Size Settings window under Element Size, choose **Custom**. Under Element Size Parameters, set Maximum element size to 2 (to get coarsest mesh possible). In order to see the mesh being used, right click the **Mesh 1** branch and choose Build All. Figure 1 (b) 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 maximum element size set to 2, the number of triangular elements is 28.
12. 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 blue 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 15 for this coarse mesh using linear Lagrange (element order 1) elements.



## A.2 How to Compute the Error against True Solution in 2-D and 3-D

In this section, we make use of the FEM solution to calculate the  $L_2$ -norm of the error against the true PDE solution  $u_{true}$ .

1. In the model builder window, under the Component 1(comp 1) right click Definitions node. From component couplings menu select **Integration**. In the Integration Settings window, under the Source Selection section, for “Geometric entity level” and “Selection” options choose Domain and All domains respectively. Note that the default name for this operator is `intop1`. From now on, this operator can be used for integrating the desired quantities over the domain. In order to be able to use this operator in post processing you need to rerun your model.
2. After recomputing the FEM solution, in order to compute  $L^2$ -norm, right click Derived Values under the Results branch and choose Global Evaluation. Select **Global Evaluation 1** node, in the setting window, and under Expression section type in `sqrt(intop1((u.true-u)^2))`. Click to check mark the Description and label this quantity by typing E.r to indicate that it is the the norm of the error. Now, right click the **Global Evaluation 1** node on the Model Builder window and select Evaluate and New Table. The result of the computation is shown in the Table 1 tab under the graphics area.

## A.3 How to Refine Mesh Regularly in 2-D and 3-D

In this section, we demonstrate how to refine mesh regularly in COMSOL 5.1 to obtain the results similar to those presented in Table 1.

1. In the model builder window under the Component 1 (comp 1) branch right click Mesh 1 node and from More Operations menu select **Refine**.
2. In the Refine Settings window under Refine Options section, make sure that “Regular refinement” is selected for Refinement method. Then enter 1 for Number of refinements. Rebuild the mesh by right clicking the **Mesh 1** branch and selecting Build All.
3. Again, check the statistics by right clicking the **Mesh 1** branch and selecting **Statistics**. With one refinement, the number of triangular elements has increased by a factor of 4 and 8 for 2-dimensional and 3-dimensional cases respectively.
4. Recompute the FEM solution under this refinement by right clicking the **Study 1** branch and selecting Compute.
5. In order to find the maximum element size,  $max_e h_e$ , as reported in Table 1, after computing the FEM solution, in the Model Builder window under the Results branch, right click the **Derived Values** node. From Maximum menu, select **Volume Maximum**. In the “Volume Maximum” Settings window, under the Selection section choose All domains for Selection option. Under the Expression section, either enter **h** for the Expression or click the Replace Expression on the right corner of the Expression section and choose Mesh and **h-Element size** in turn.

## A.4 How to Obtain Exploded View of Mesh for 3-Dimensional Problem in COMSOL 5.1

In this section, we show how to obtain the exploded view of mesh for three-dimensional problem for the whole domain and also for the interior elements of the domain. The purpose of this section is to obtain figures similar to those reported in Figure 2.

1. After generating **Mesh 1** as explained in Sec. A.1, Step 11, in the Model Builder window, under the Component 1 branch, right click **Mesh 1** and select **Plot**. This will automatically generate data set **Mesh 1** under the Data Sets node and its corresponding 3d plot group, **3D Plot Group 1**, in the Results branch.

2. Make sure that node **Mesh 1** under the 3D Plot Group 1 is selected. In the **Mesh 1** Settings window, locate **Level** section. After expanding Level section, change Level option to **Volume**.
3. Then, locate the **Color** section. Under the Color section, change Element color option to **Gray**.
4. In the same window, under **Shrink Elements** section change Element scale factor to 0.7.
5. Click **Plot** to see exploded view of the mesh. This should look like Figure 2 (a).

In order to see the interior elements of the mesh we need to enable element filters as follows,

1. After obtaining the exploded view of the mesh as above, in Mesh 1 Settings window, under the 3D Plot Group 1 branch, under the results, expand **Element Filter** section.
2. Make sure the **Enable filter** box is checked. Using element filters, you can highlight elements based on, for example, their mesh quality, size, or location. Here, we want to use elements location to filter them. In doing so we make use of **Expression** for a filter criterion. COMSOL will plot fraction of elements closest to where the expression evaluates to the smallest value. For instance, by entering  $-y$  and 0.6 for the expression and fraction, respectively, COMSOL will plot 60% of the elements closest to the  $y = 1$  plane.
3. Under the Element Filter section, for Criterion option choose **Expression**. In the Expression field, enter  $-y$ . In the Fraction field, enter 0.6.
4. Plot this new figure. The generated figure should look like Figure 2 (b).

## A.5 Implementation of a Point Source in 2-D and 3-D

In this section, we modify the models which obtained for 2 and 3 dimensional smooth problems in previous section to get the desired non-smooth problem as stated in section 4.

As described in Section 4, the Dirac delta source function can be implemented in COMSOL5.1 by using the point source and setting the classical source term  $f = 0$ .

1. Load the mph-file for 2 or 3 dimensional smooth problem.
2. Make sure that your geometry includes a point at the center of the square/cube domain i.e., (0,0) and (0,0,0) for 2-d and 3-d cases respectively.
3. Under the Definitions branch in the model builder, select Variables 1 node. Change the expression for **u\_true** according to the true solution for non-smooth case stated in (6). For two- and three-dimensional case the expression for **u\_true** will be  $-\log(\rho)/(2*\pi)$  and  $1/(4*\pi*\rho)$  respectively.
4. Select **Coefficient Form PDE 1** from the Model Builder window. In its Settings window under the Source Term section set  $f$  equal to zero.
5. Select **Dirichlet Boundary Condition 1** under the Coefficient Form PDE (c) branch. In the Settings window under the Dirichlet Boundary Condition make sure that **u\_true** is set for  $r$ .
6. Again in the Model Builder window, right click Coefficient Form PDE (c) branch and from Points menu choose **Point Source**.
7. Choose the desired point (point #5 in this case) under the Point Selection section in the Point Source Settings window.
8. In the same window, under the Source Term section set  $f$  to 1.
9. Under the Mesh 1 branch right click Refine 1 and select Disable.
10. Select Build All to see the mesh. It should be identical to the initial mesh for smooth case.
11. Compute the FE solution.