

Application of grid convergence index in FE computation

L. KWAŚNIEWSKI*

Faculty of Civil Engineering, Warsaw University of Technology, 16 Armii Ludowej Ave., 00-637 Warszawa, Poland

Abstract. This paper presents an application of the grid convergence index (GCI) concept based on the Richardson extrapolation to a selected simple problem of a cantilever beam loaded with vertical forces at the tip end. The GCI method, popular in computational fluid dynamics, has been recently recommended for finite element (FE) applications in solid and structural mechanics. Based on the results obtained usually for three meshes, the GCI method enables one to determine, in an objective manner, the order of convergence to estimate the asymptotic solution and the bounds for discretization error. The example shows that the characteristics of the convergence depend on the selection of the quantity of interest, which can be local or a global functional such as the deflection considered here. The results differ for different FE formulations, and the difference is bigger when the nonlinearities (e.g., due to plastic response) are taken into account.

Key words: discretization error, grid convergence index, finite element, mesh refinement, verification.

1. Introduction

Computer simulations and numerical analysis are used in many branches of science and technology, such as aviation, civil engineering, and information technology. The increasing importance of numerical methods reflects the increasing use of the adjective “computational” in the names of various fields, such as computational fluid dynamics (CFD). Individual applications of numerical methods differ in the level of accuracy with which a numerical model is able to reflect the physical process. At one extreme is the mathematical modeling of computer chips, with expected 100% accuracy for all the ones and zeros in the output signal. At the opposite end is a complex mathematical modeling of weather phenomena in the universal conviction of a much lesser ability to predict the real process, especially in the longer term. There are also differing opinions on the usefulness and reliability of such applications in solid and structural mechanics, from very optimistic [1] to extremely skeptical [2]. An analysis with the help of complex numerical models is particularly applicable where there is no closed analytical solution (e.g., because of the complexity of the geometry) and where, for various reasons (e.g., economical), the experiments are impossible or insufficient. In all such cases, the question arises about the ability of such analysis to correctly predict the results of a physical process in question.

This paper is a continuation of the publication [3] focused on applications of nonlinear numerical models in civil engineering and related branches of technology. The work [3] is devoted to the verification and validation procedure of numerical models while stressing the differences between these concepts. Verification and validation is now considered the most objective method of assessing the reliability of nonlinear simulations. The literature contains many inconsistent opinions on the possibility of creating a unified verification and validation procedure defined in isolation from the specifics of

the considered problem. There are also some extremely skeptical opinions that totally negate the possibility of validating numerical models [2, 4].

The current paper is focused on the mesh refinement study as a part of the verification procedure. According to existing standards for verification and validation [5–7], verification, which uses comparison of computational solutions with highly accurate (analytical or numerical) benchmark solutions, should precede validation, which is based on a comparison of the numerical solution with the experimental data [3]. The first recommended procedure within the verification process is the mesh refinement study [7]. Its main objective is to evaluate the error of discretization and to check if the developed mesh is sufficiently refined. The analytical solution for the mathematical model, which would help establish the error of discretization, is usually unknown for practically important problems. The entire process of verification is empirical using the “a posteriori” approach, where the reasoning is based on the experience coming from repeated calculations. The mesh refinement study is conducted based on a comparison of the results for a minimum of two but usually three meshes. Among many estimators offered in the literature (e.g. [8]) there is a consistent method called the grid convergence index (GCI), which is popular in CFD for determining discretization error. In his recent paper, Schwer [9] recommended application of the GCI for finite element (FE) calculations in solid and structural mechanics. The subsequent sections provide a short description of the GCI method and a simple example of its application.

2. Concept of the grid convergence index

Developed by Roache [10], the GCI applies the old concept of the Richardson extrapolation [11]. Based on the results obtained usually for three grids (meshes), the approach applied in the GCI method enables one to determine in an objective

*e-mail: l.kwasniewski@il.pw.edu.pl

manner the order of convergence to estimate the asymptotic solution and the bounds for discretization error. The term “grid” is more common in CFD, whereas the term “mesh” is used in FE analyses. The following section provides a short description of the GCI method, followed by a summary [12].

In CFD, it is assumed that when the grid is successively refined with the number of cells increasing and with the cell dimensions and time step decreasing, the spatial and temporal discretization errors should asymptotically approach zero [12]. The discretization error does not include the computer round-off error, which nowadays is considered small enough to be neglected.

The concept is based on the following assumption [10, 12] about the nature of the discretization error

$$E = f_h - f_{exact} = Ch^p + H.O.T. \quad (1)$$

where E is the discretization error defined as the difference between the result for current mesh density f_h , characterized by parameter h , and the exact solution f_{exact} . On the right side of Eq. (1), C is a constant, the exponent p defines the order of convergence, and $H.O.T.$ means higher-order terms. The parameter h is usually defined by a dimension characterizing the cell (FE) with smaller values for successively finer meshes. The exact solution f_{exact} in practice means an asymptotic solution for a mesh with element dimension h approaching zero. As it is shown later, this solution can differ from the exact solution for the mathematical model due to some approximations and limitations applied in the FE formulation [9] and can be different for different FE formulations [9].

The presented consideration can also be extended to unstructured grids or meshes generated using non integer mesh refinement or irregular coarsening [10]. The publication [12] recommends using unstructured grids for CFD calculations, with an effective grid refinement ratio defined as

$$r_{ef} = \left(\frac{N_1}{N_2} \right)^{\frac{1}{D}}, \quad (2)$$

where N_i is the total number of grid points used for the i -th grid and D is the dimension of the domain. Here, for the sake of simplicity, we consider regular meshes with the same node spacing in all xyz directions. Also, we consider successively finer meshes, which are generated by dividing the node spacing in all directions into halves. In this way, h is divided by two. For example, for a 3D mesh built of solid (brick) elements, every element is divided into eight smaller elements. Such refinement for regular structural meshes can be easily obtained using most of the commercial graphical preprocessors. The following derivation is focused on the typical situation where the error bound is supposed to be determined based on the finest mesh solution. The formula for the estimation based on coarse meshes (which are faster and can be preferably used for repeated calculations) can be found in [12].

A straightforward way for estimating the order of convergence is to read it as a slope on the logarithmic plot of the error versus mesh density parameter ($\log(E)$ vs. $\log(h)$). Neglecting $H.O.T.$ in Eq. (1) and taking the logarithm of both sides gives [12]

$$\log(E) = \log(C) + p \log(h). \quad (3)$$

When using the constant mesh refinement ratio r such that

$$h_i = \frac{h_{i+1}}{r} \quad (4)$$

and where h_i represents the finer mesh, the order of convergence can be estimated directly by obtaining results for three successive meshes f_3 , f_2 , and f_1 . The quantity f is the result of calculation characterizing the response of the system; it can be a local value (e.g., stress) or a global one given by a functional (e.g., displacement). Repeating Eq. (1) for three meshes, we can eliminate the constant C and $H.O.T.$ The order of convergence is given by

$$p = \frac{\ln\left(\frac{f_3 - f_2}{f_2 - f_1}\right)}{\ln(r)}. \quad (5)$$

Verification of the calculations requires that solutions for all considered meshes should be in the asymptotic range of convergence [12]. The asymptotic range of convergence requires that the ratio between the errors E and the mesh spacing h^p is constant

$$C = E/h^p. \quad (6)$$

Checking if the solutions are within the asymptotic range is a part of the GCI procedure.

Following [12], now we introduce the Richardson extrapolation, which here serves as a higher-order estimate of the evaluated quantity. A quantity f calculated for a mesh characterized by parameter (mesh size) h can be expressed using Taylor's theorem as

$$f = f_{h=0} + g_1 h + g_2 h^2 + g_3 h^3 + \dots \quad (7)$$

where $f_{h=0}$ is the asymptotic solution for h approaching 0; the unknown functions g_1 , g_2 , and g_3 are independent of the mesh characteristics h ; and $h^n > h^{n+1}$. Let us assume that f_1 and f_2 are the second-order approximations of $f_{h=0}$ calculated for two mesh characteristics h_1 and h_2 , with the mesh refinement ratio defined by Eq. (4) and h_1 representing finer mesh. The second-order approximation means that $g_1 = 0$ in the expansion (7). Repeating Eq. (7) for two considered meshes h_1 and h_2 and neglecting third-order and higher terms, we obtain the estimate of the asymptotic solution [12]

$$f_{h=0} \cong f_1 + \frac{f_1 - f_2}{r^2 - 1}, \quad (8)$$

where, according to Eq. (4), $r = h_2/h_1$. For example, for the second-order approximation and the mesh refinement ratio $r = 2$

$$f_{h=0} \cong \frac{4}{3}f_1 - \frac{1}{3}f_2. \quad (9)$$

It has been proven that Eqs. (8) and (9) give fourth-order estimates [10, 12] (assuming that f_1 and f_2 are the second-order approximations). In practice, the Richardson extrapolation is generalized for any, also non integer, p -th order approximations and the mesh refinement ratio r

$$f_{h=0} \cong f_1 + \frac{f_1 - f_2}{r^p - 1} \quad (10)$$

and is considered as $p + 1$ order approximation [12]. Moving f_1 to the left side of Eq. (10) and dividing both sides by $f_{h=0}$ gives

$$A_1 = \frac{f_1 - f_{h=0}}{f_{h=0}} \cong \frac{f_1 - f_2}{f_{h=0}} \frac{1}{r^p - 1}, \quad (11)$$

where A_1 defines the relative error for the solution f_1 . Replacing unknown $f_{h=0}$ with calculated f_1 on the right side of Eq. (10), we get an approximation of the relative error A_1

$$A_1 = E_1 + O(h^{p+1}, E_1^2), \quad (12)$$

where $O(h^{p+1}, E_1^2)$ represents *H.O.T.* and E_1 is the estimator of the relative error A_1

$$E_1 = \frac{\varepsilon}{r^p - 1} \quad (13)$$

with the quantity ε defining relative difference between subsequent solutions

$$\varepsilon = \frac{f_1 - f_2}{f_1}. \quad (14)$$

As pointed out in [12], the quantity (14) should not be used directly as an error estimator because it does not take into account r or p . In addition, for example, for the mesh refinement ratio r close to 1.0 it can give a very small, underestimated error.

The GCI is defined as [10, 12]

$$GCI = \frac{F_s |\varepsilon|}{r^p - 1} 100\%, \quad (15)$$

where F_s is a safety factor. The recommended CFD values of the safety factor F_s are [12]

$F_s = 3.0$ when two meshes are considered and

$F_s = 1.25$ for three or more meshes.

Given in a percentage manner, the GCI (15) can be considered as a relative error bound showing how the solution calculated for the finest mesh is far from the asymptotic value. It gives a prediction on how much the solution would change with a further refinement of the mesh. The smaller the value of the GCI, the better. This indicates that the computed solution is within the asymptotic range. The safety factors F_s given above were arbitrarily set based on the accumulated experience on CFD calculations [10]. The safety factor represents 95% confidence for the estimated error bound. That assumption can be expressed as the following statement [9]: There is 95% confidence that the converged solution is within the range $[f_1(1 - GCI_{12}/100\%), f_1(1 + GCI_{12}/100\%)]$.

To properly calculate the GCI, it is important to have all calculations f_i in the asymptotic range of convergence. Based on Eq. (6), it can be checked by comparing if the values of two GCIs computed over three meshes satisfy [12]

$$GCI_{23} = r^p GCI_{12}. \quad (16)$$

3. Example of mesh refinement study

The example study of mesh refinement is presented here for a simple problem depicted in Fig. 1. A cantilever beam with a 100×200 mm rectangular cross-section and 1000 mm length is loaded at the tip end with vertical forces, with the resultant

value P . The loading is distributed uniformly among all the nodes along the beam's edge as shown in Fig. 1. The material is bilinear elastic-plastic with elastic modulus $E = 10$ GPa, Poisson ratio $\nu = 0.3$, yield stress $\sigma_y = 20$ MPa, and tangent modulus $E_T = 2$ GPa. The objective of the calculation is to find the tip-end deflection f for different levels of loading. The problem is solved using commercial FE programs ABAQUS [14] and LS-DYNA [13] and three FE meshes shown in Fig. 2. Each mesh is built of solid FEs with the same spacing in xyz directions. Figure 2 indicates mesh characteristics h_i defined by dimensions of the FEs. The finer meshes are built using constant refinement ratio $r = 2$. Table 1 shows the results of calculations repeated for three levels of loading (10 kN, 20 kN, and 40 kN) and for two FE formulations for eight-node solid elements applied in each of the solvers. The symbols C3D8 and C3D8R indicate eight-node linear bricks applied in program ABAQUS [14], where R means reduced integration with hourglass control. ELFORM 1 and ELFORM 2 indicate similar FE formulations implemented in LS-DYNA, that is, constant stress solid element and fully integrated S/R solid, respectively [13]. The results with asymptotic solutions given in Table 2 are compared in Fig. 3 through 5. According to Fig. 3, which shows deflections for loading $P = 10$ kN, we can see that when the beam is within the elastic range the solutions for finer meshes are very close and that asymptotic solutions are practically identical. For a purely elastic response, we can compare the computed results with the analytical solution for deflection of the cantilever beam due to bending and shear [15]

$$f_{analytical} = \frac{Pl^3}{3EJ_y} \left(1 + \frac{3\beta}{l^2} \right), \quad (17)$$

where $\beta = \alpha \frac{EJ_y}{GA}$, and $\alpha = 1.25$ is the correction coefficient for the rectangular section. The formula (17) gives the value $f_{analytical} = 5.156$ mm for the considered beam, which is very close to the computational solutions obtained for the most refined mesh (see Table 1).

As indicated in Fig. 4 and 5 for loads of 20 kN and 40 kN, respectively, the differences among solutions are bigger when the material nonlinearity is taken into account and there is a plastic zone in the beam, such as shown in Fig. 6.

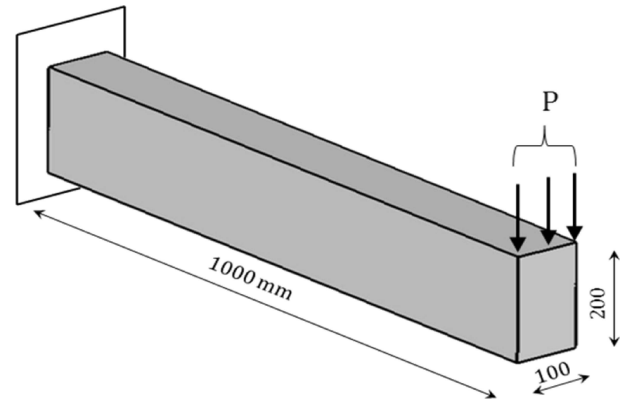


Fig. 1. Example problem – cantilever beam loaded with forces at the tip end

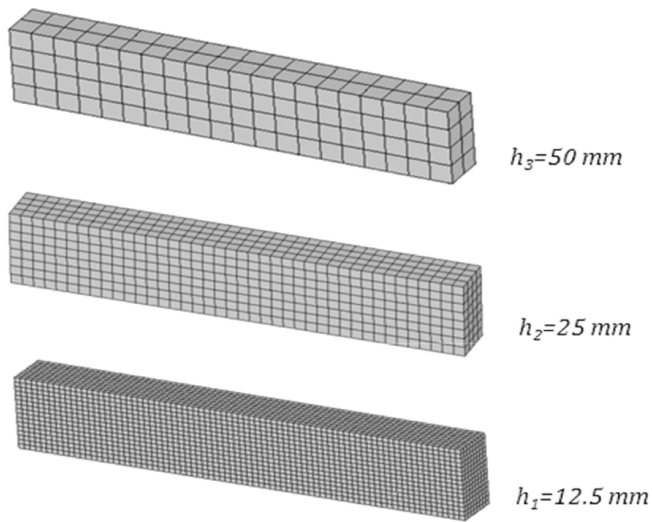
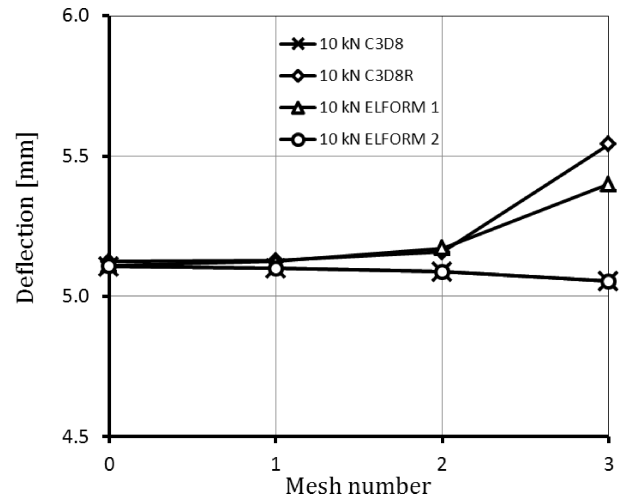
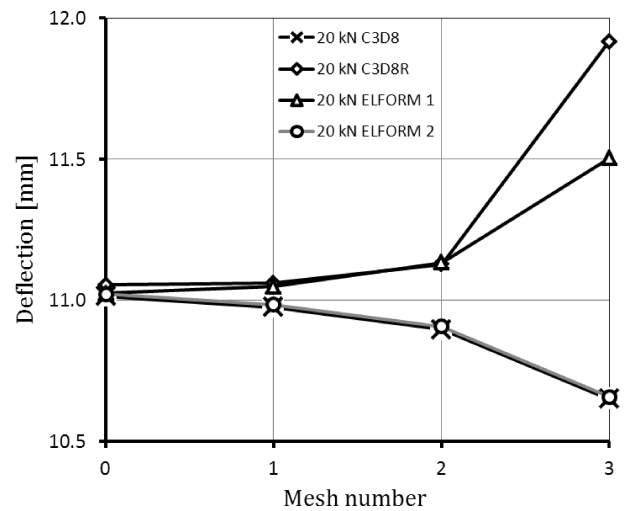


Fig. 2. Three FE meshes considered for mesh refinement study

Table 1

Results of FE calculations for three levels of loading and two FE formulations

Load [kN]	Mesh	h [mm]	Deflection f [mm]			
			C3D8	C3D8R	ELFORM 1	ELFORM 2
10	1	12.5	5.0997	5.1272	5.1238	5.1002
	2	25	5.0875	5.1577	5.1720	5.0880
	3	50	5.0545	5.5422	5.4000	5.0540
20	1	12.5	10.9760	11.0604	11.0490	10.9850
	2	25	10.8967	11.1273	11.1330	10.9060
	3	50	10.6505	11.9177	11.5030	10.6570
40	1	12.5	45.0233	45.5003	45.9750	45.6490
	2	25	44.5698	45.9487	46.1470	45.1870
	3	50	43.1959	48.5291	46.8300	43.7510

Fig. 3. Results for load $P = 10$ kNFig. 4. Results for load $P = 20$ kNTable 2
Calculation of grid convergence index

Load [kN]	FE element formulation	Order of convergence p (5)	Asymptotic solution $f_{h=0}$ (10)	GCI_{12} (15) [%]	GCI_{23} (15) [%]	$GCI_{23}/r^p GCI_{12}$ (16)
10	C3D8	1.431	5.107	0.177	0.478	1.002
	C3D8R	3.656	5.125	0.064	0.803	0.994
	ELFORM 1	2.242	5.111	0.315	1.477	0.991
	ELFORM 2	1.479	5.107	0.167	0.467	1.002
20	C3D8	1.634	11.014	0.429	1.342	1.007
	C3D8R	3.563	11.054	0.070	0.821	0.994
	ELFORM 1	2.139	11.024	0.279	1.220	0.992
	ELFORM 2	1.656	11.022	0.418	1.326	1.007
40	C3D8	1.599	45.247	0.620	1.899	1.010
	C3D8R	2.525	45.406	0.259	1.476	0.990
	ELFORM 1	1.715	45.848	0.248	0.810	0.995
	ELFORM 2	1.684	45.836	0.553	1.795	1.010

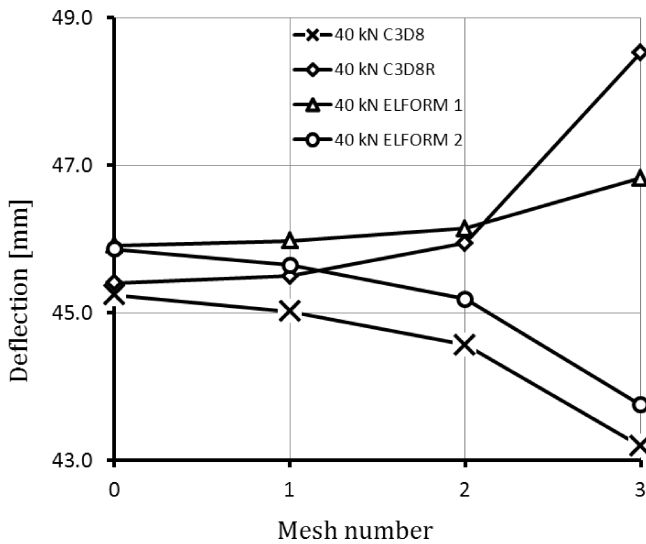


Fig. 5. Results for load $P = 40$ kN

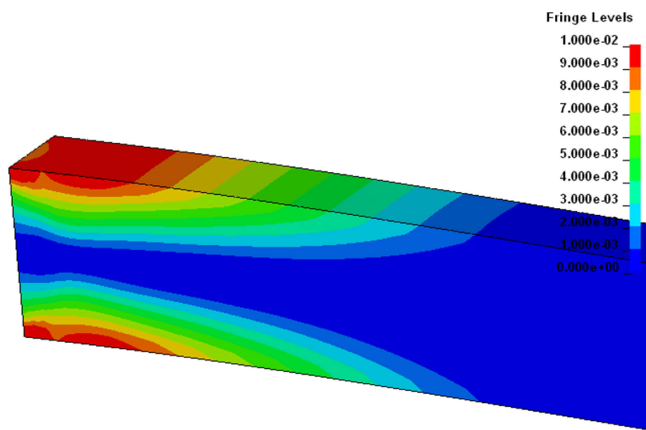


Fig. 6. Contours of effective plastic strain obtained for ELFORM 1 mesh #1 and for load $P = 40$ kN

Table 2 presents convergence parameters calculated based on the data gathered in Table 1. The following are calculated for each mesh, FE formulation, and loading lever: the order of convergence p (5), the asymptotic solution $f_{h=0}$ (10), the GCI for the solutions 1 and 2 GCI_{12} (15) and for the solutions 2 and 3 GCI_{23} (15), and finally the ratio defined by Eq. (16), which provides the check to ensure the calculated solutions are within the asymptotic range. Table 2 shows some clear tendencies. The FE formulations with reduced numerical integration (C3D8R and ELFORM 1) have a higher convergence order and lower GCI. When the material nonlinearity is taken into account and there are plastic strains in the model, all parameters characterizing convergence are worse and there are bigger differences among the solutions obtained for different FE formulations and the FE programs. The fourth column of Table 2 provides asymptotic values $f_{h=0}$, which can be considered as the best approximation, and the fifth column gives GCI_{12} , which can be considered as the error bound. The highest order of convergence $p = 3.656$ and the lowest $GCI_{12} = 0.064$ are for the FE formulation C3D8R and for the elastic response for $P = 10$ kN. The lowest order of

convergence $p = 1.431$ was registered for the FE formulation C3D8 and for the elastic response for $P = 10$ kN. The highest $GCI_{12} = 0.620$ is for the FE formulation C3D8 and for $P = 40$ kN.

4. Conclusions

The mesh refinement study is the first procedure in the verification of numerical models. It should answer the questions of whether the mesh is refined enough and what the error bound is for the quantities of interest. A further analysis is warranted. Many studies have been reported where FE analysis was applied for solid or structural mechanics without any information about the mesh refinement study or discretization error. The situation is much better in the field of CFD, where many of the professional journals (e.g., ASME Fluids Engineering Journal [9]) require discretization error estimation.

This paper presents an application of the CGI concept based on the Richardson extrapolation to a selected simple problem of a cantilever beam loaded with vertical forces at the tip end. The example shows that the characteristics of the convergence (e.g., the order of convergence, asymptotic value, and GCI) depend on the selection of the quantity of interest, which can be local or a global functional such as the deflection considered here. The results differ for different FE formulations, and the difference is bigger when the nonlinearities (e.g., due to plastic response) are taken into account. When there is an elastic response, all FE formulations provide basically the same asymptotic values; however, there are different orders of convergence and error bounds defined by the GCI. When material nonlinearity is also present, the asymptotic solutions differ for different FE formulations. Because the asymptotic solution is influenced inherently by FE formulation, it cannot be considered as the solution of the mathematical model. The computational performance of eight node solids in the inelastic calculations is dependent on the formulation applied in the considered code. This refers to the several aspects such as approximation of volume integration and how locking or zero energy modes are prevented.

The GCI concept presented in this paper provides a comprehensive and objective procedure for discretization error estimation that can be successfully used in structural and solid mechanics applications of FE analysis.

REFERENCES

- [1] J.T. Oden, T. Belytschko, J. Fish, T.J. Hughes, C. Johnson, D. Keyes, A. Laub, L. Petzold, D. Srolovitz, and S. Yip, "Revolutionizing engineering science through simulation", in *National Science Foundation Blue Ribbon Panel Report 65*, National Science Foundation, Arlington, 2006.
- [2] L.F. Konikow and J.D. Bredehoeft, "Ground-water models cannot be validated", *Advances in Water Resources* 15 (1), 75–83 (1992).

- [3] L. Kwaśniewski, "On practical problems with verification and validation of computational models", *Archives of Civil Eng.* LV (3), 323–346 (2009).
- [4] I. Babuska and J.T. Oden, "Verification and validation in computational engineering and science: basic concepts", *Computer Methods in Applied Mechanics and Eng.* 193 (36–38), 4057–4066 (2004).
- [5] U.S. Department of Defense, *Verification, Validation, and Accreditation (VV&A) Recommended Practices Guide*, Defense Modeling and Simulation Office, Office of the Director of Defense Research and Engineering, Virginia, 1996, www.dmsomil/docslib.
- [6] AIAA, *Guide for the Verification and Validation of Computational Fluid Dynamics Simulations*, American Institute of Aeronautics and Astronautics, AIAA-G-077-1998, Reston, 1998.
- [7] ASME, *Guide for Verification and Validation in Computational Solid Mechanics*, American Society of Mechanical Engineers, Virginia, 2006.
- [8] J. Zhu and O. Zienkiewicz, "A posteriori error estimation and three-dimensional automatic mesh generation", in *Finite Elements in Analysis and Design*, vol. 25, no. 1–2, pp. 167–184, Elsevier, London, 1997.
- [9] L.E. Schwer, "Is your mesh refined enough? Estimating discretization error using GCI", *7th LS-DYNA Anwenderforum* I-I-45-54, CD-ROM (2008).
- [10] P.J. Roache, "Verification and validation in computational science and engineering", *Computing in Science Eng.* 1, 8–9 (1998).
- [11] L.F. Richardson, "The approximate arithmetical solution by finite differences of physical problems including differential equations. with an application to the stresses in a masonry dam", *Philosophical Trans. Royal Society of London A* 210 (459–470), 307–357 (1911).
- [12] NASA NPARC Alliance Verification and Validation, *Examining Spatial (Grid) Convergence*, <http://www.grc.nasa.gov/WWW/wind/valid/tutorial/spatconv.html>.
- [13] J.O. Hallquist, *LS-DYNA Keyword User's Manual-Version 971*, Livermore Software Technology Corporation, Livermore, 2007.
- [14] *ABAQUS Users Manual V. 6.10-1*, Dassault Systemes Simulia Corp., Providence, 2010.
- [15] J. Gere and S. Timoshenko, *Mechanics of Materials*, Brooks, Cole Publishing, New York, 2000.