COMSOL
CONFERENCE
2024 FLORENCE

Use lastest developments in COMSOL Multiphysics® to solve eddy current non-destructive testing problems

L. Santandrea[1], Y. Le Bihan[1]
1. GeePs (Group of Electrical Engineering-Paris), CentraleSupélec – CNRS – Université Paris-Saclay – Sorbonne Université, 11 Rue Joliot Curie, 91192 Gif-sur-Yvette, France.

Abstract
As part of its research, the Group of Electrical Engineering-Paris (GeePs) uses COMSOL Multiphysics® software for applications such as multiphysiscal material modeling in electromagnetic compatibility and Eddy current (EC) non-destructive testing (NDT). EC NDT is an easy-to-use and low-cost technique with a wide range of applications, including defect detection, material thickness or electrical conductivity measurement and mechanical stress analysis.
This paper is the continuation of a previous paper published in 2010 [1] carried out under version 3.5 of COMSOL Multiphysics®. Since then, several other papers have been published in connection with the use of COMSOL Multiphysics® in EC NDT, such as [2]. The aim of this contribution is to highlight the new advances in the software in its version 6.2, which allow more efficient and faster resolution of this kind of low-frequency electromagnetic problem.
To solve this kind of problem AC/DC module is used in 3D. The EC NDT problems often involve areas of small thickness (skin depth, thin cracks, small lift-off between probe and specimen to be tested, coating). Generating a mesh in these areas can be tricky. Indeed the quality of the mesh will determine the reliability of the solution and the computation time. Several solutions will be proposed to deal with this kind of zone such as the use of boundary layers, specific boundary conditions, prismatic extruded mesh...
Each case with its proposed solution will be evaluated on benchmark cases or compared to an analytical solution. New solutions to reduce computation time such as distributed computed will be presented.

Keywords: eddy current non-destructive testing, thin layer mesh, harmonic quasi-static fields

Introduction
A typical geometry EC NDT problem is constituted of an electrically conductive specimen that may contain a defect, a probe (one or several coils driven by an excitation current density which may include a magnetic core) and the surrounding air (Figure 1),.

The forward modeling typically consists in the determination of the probe impedance variation varying probe position or probe excitation. The real and imaginary parts of the probe impedance are determined by using numerical computation of the magnetic energy and the power losses, respectively. Both are deduced from the finite element method (FEM) results obtained using the AC/DC Comsol module.

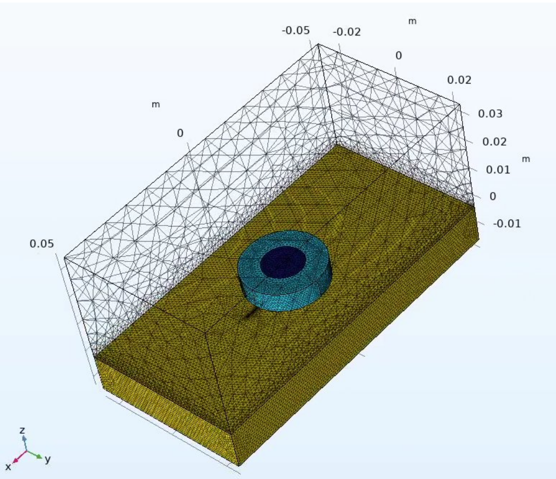

Figure 1. Typical geometry and mesh of an EC NDT problem.

Computing machine configuration

The workstation used for this study is a Dell workstation. (AMD Ryzen Threadripper PRO 5965WX 24-Cores).

Resolution of some typical EC NDT problems

To validate our modeling approach we refer to two benchmark problems containing experimental results.

Benchmark problem Team workshop 15 (TW15)

In this classical EC NDT benchmark problem [3][4] a cylindrical air-cored coil is moved along the length of a rectangular slot made in a conductive plate (Figure 1).
Both frequency and coil lift-off (distance between the coil and the specimen to be tested) are fixed. The objective is to compute the change of the impedance of the coil during its displacement. This value is evaluated by subtracting the values obtained for the specimen (plate) without defect from the values obtained for the plate with defect.
The parameters of the problem are shown in the Table 1.

The coil	
Inner radius *(a2)*	6.15 ± 0.05mm
Outer radius *(a1)*	12.4 ± 0.05mm
Length *(b)*	6.15 ± 0.1mm
Number of turns (*N*)	3790
Lift-off *(l)*	0.88mm
The test specimen	
Conductivity (σ)	3.06 ± 0.02 x 10^7 S/m
Thickness	12.22 ± 0.02mm
The defect	
Length (2c)	12.60 ± 0.02mm
Depth *(h)*	5.00 ± 0.05mm
Width *(w)*	0.28 ± 0.01mm
Other parameters	
Frequency	900 Hz
Skin depth at 900 Hz	3.04mm
Isolated coil inductance	221.8 ± 0.04mH

Table 2: Parameters of Test Experiment No. 2 (see Figure 1)

The coil	
Inner radius *(a2)*	9.34 ± 0.05mm
Outer radius *(a1)*	18.4 ± 0.05mm
Length *(b)*	9.0 ± 0.20mm
Number of turns (*N*)	408
Lift-off *(l)*	2.03 ± 0.05mm
The test specimen	
Conductivity (σ)	3.06 ± 0.02 x 10^7 S/m
Thickness	12.22 ± 0.02mm
The defect	
Length (2c)	12.60 ± 0.02mm
Depth *(h)*	5.00 ± 0.05mm
Width *(w)*	0.28 ± 0.01mm
Other parameters	
Frequency	7000 Hz
Skin depth at 7000 Hz	1.09mm
Isolated coil inductance	3.96 ± 0.1mH

Table 1. Parameters for team Workshop problem 15 [2].

For TW15 the skin depth is equal to 3.04 mm for the first problem (TW15-1) and to 1 mm for the second one (TW15-2).
Both magnetic fields (mf) or magnetic-electric fields (mef) formulations can be used.
Figure 2 gives a mean to obtain resistance and reactance with the software.

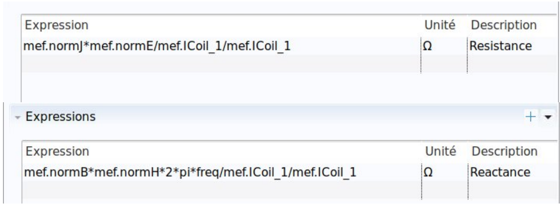

Figure 2. Volume integration for the impedance computation (deduced from magnetic energy and power losses).

The problem was solved using frequency domain resolution with the mf formulation and a tetrahedral mesh (mesh size condition imposed with element edge length < 1mm in the specimen, air-core and coil).

Figure 2 and 3 show a good agreement between experimental and numerical results for both test experiments (TW15-1 and TW15-2). The defect is centered at x = 0 mm. The scan (26 points, 52 FEM resolutions, 6 millions Dofs) is obtained with a very reasonable computing time (about 6h).

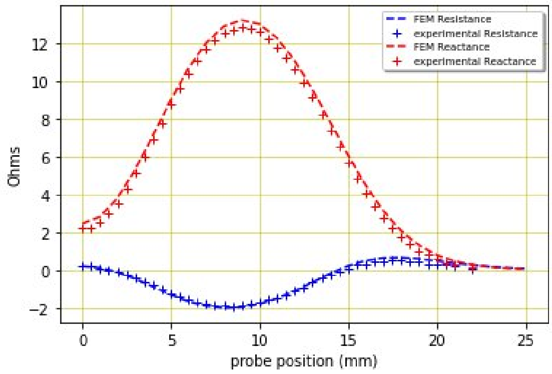

Figure 3. Experimental and numerical resistance and reactance variation for the TW15-1 case.

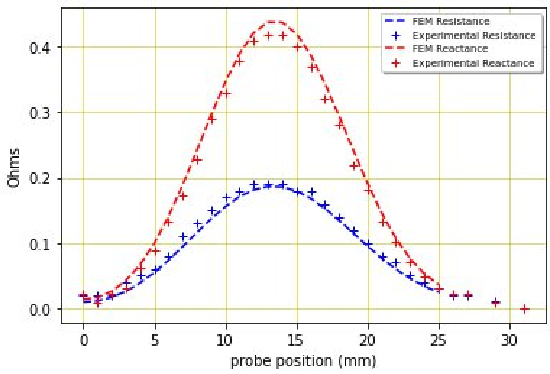

Figure 4. Experimental and numerical resistance and reactance variation for the TW15-2 case.

Benchmark problem JSAEM n° 2-5

The characteristics of this problem are given in Table 2. [5][7] In this case, the skin depth of the electromagnetic field (1.6 mm) is greater than the thickness of the plate (1.25 mm). It is therefore

necessary to add a layer of air underneath the plate. The results obtained using the mf formulation with a tetrahedral meshing are shown in Figure 5.
A good agreement is still obtained between the numerical results and the experimental ones.

	JSAEM 2-5]
Inner radius (mm)	0.6
Outer radius (mm)	1.6
Length (mm)	0.8
Relative permeability	1
Number of turns	140
Lift-off (mm)	0.5
Frequency (Hz)	150×10^3
Conductivity (S/m)	1.0×10^6
Thickness (mm)	1.25
Length (mm)	10.0
Depth (mm)	0.75
Width (mm)	0.21

Table 2. Parameters for JSAEM problem 2.5.

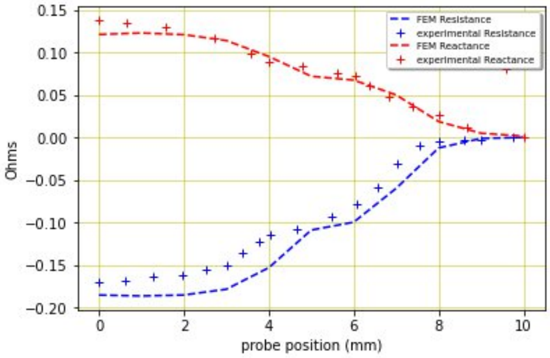

Figure 5. Experimental and numerical resistance and reactance variation for the JSAEM benchmark

Thin domain treatment

One difficulty with EC NDT problem could be the presence of thin domains to mesh (thin lift-off, skin depth, thin flaw, flat coil,..).

Mesh solution for small skin depth

As a first example, we will consider a problem with a small skin depth. When the frequency of the excitation current or the magnetic permeability of the material increases, the zone in which EC develop (skin depth, noted δ) decrease according to:

$$\delta = \frac{1}{\sqrt{\pi \mu \sigma f}}$$

With µ the magnetic permeability, σ the electric conductivity and f the coil excitation frequency.

In order to illustrate the consequences of this decrease and to have an analytical solution to validate our numerical results, we will vary δ on a defect-free plate by increasing the excitation frequency and then compare our results with the analytical model proposed by Dodd and Deed [6]. We use the geometry of the TW15-1 test case. Remember that COMSOL Multiphysics® uses second-order elements by default.
The use of tetrahedral elements can prove costly when the quality of elements required for good convergence has to be maintained. Indeed, a decrease of δ due to frequency increase imply to reduce the size of the elements, leading to an increase of the number of degrees of freedom (DOF), of the memory size and of the computation time, as can be seen in Table 3.

Element edge size (mm)	**Number of DOF (memory size)**	**Computation time**	**R (Ω)**
1	9 682 525 (38 Gb)	2 mins 24 s	3875
0.5	68 968 410 (247 Gb)	17 mins 37 s	3464
0.4	133 331 917 (470 Gb)	36 mins 13 s	3407

Table 3. FEM resistance versus plate tetrahedral mesh size

If the element size of the mesh (Figure 6) is fixed and not adapted to the skin depth decrease when the frequency increases (δ decreases shown in Figure 7)), we observe in figure 8 that the FEM solution obtained with the tetrahedral meshing starts to diverge at about 150 kHz.
If the number of tetrahedrons is adapted to the decreasing value of δ by adjusting the element size, above 100 kHz (skin depth δ = 0.4 mm), the calculation becomes impossible due to insufficient memory resources.

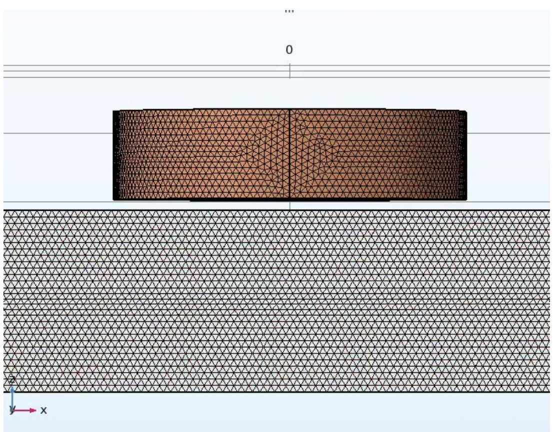

Figure 6. Tetrahedral mesh of the TW15-1 problem.

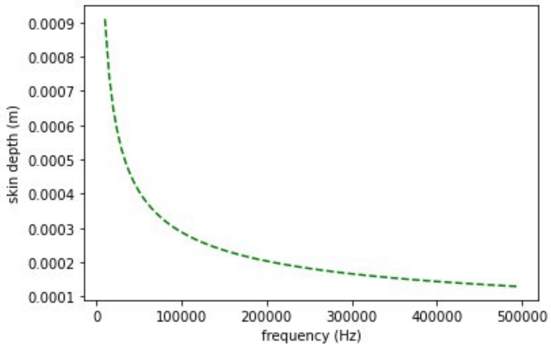

Figure 7. Skin depth evolution versus frequency.

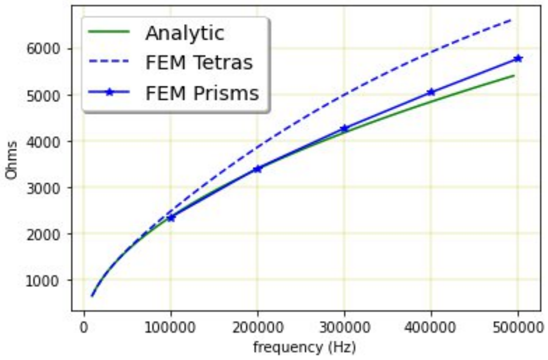

Figure 8. FEM resistance computation with uniform tetrahedral mesh (maximum edge size of 1 mm) and with prismatic mesh.

A solution to avoid this problem is to use prismatic elements to mesh the plate where the skin effect occurs. This kind of elements is more tolerant to a high dimension ratio between the thickness of the prism and its two other dimensions. Prismatic mesh is obtained by performing a swept meshing from a triangle mesh surface under the coil (figure 9). The mesh size can be controlled using an exponential or a linear progression using the sub-menu "size". Figure 8 shows the good values of the resistance obtained using this mesh strategy versus increase of the frequency. Table 4 gives the computational time for an excitation frequency equal to 200 kHz (skin depth = 0.2 mm). The analytic reference resistance value is 3377 Ω.

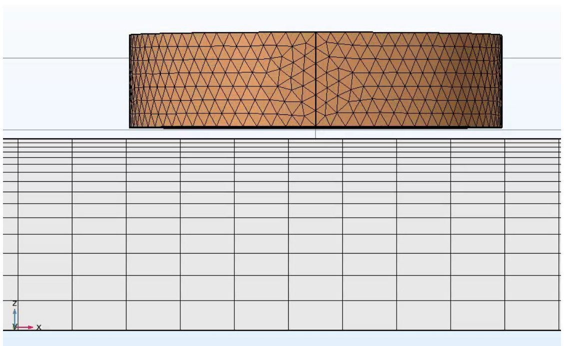

Figure 9. Mesh of the TW15-1 problem with prismatic elements in the plate.

Number of DOF	**Computation time**	**R (Ω)**
5303554 (24 Gb)	1 mins 45 s	3377

Table 4. FEM resistance obtained using prismatic mesh in the plate (200 kHz).

It can be noted that 'Boundary layers' can be a also a good alternative solution to mesh thin area such as skin depth, lift-off or multiple thin layers.

Zero thickness approach

When the thickness of the defect is very small compared to its other dimensions, the defect can be considered as an electrically insulating surface [7].
Comsol Magnetic and Electric Fields (mef) physics is used for this purpose. It offers the possibility of imposing an electrical insulation on the surface defining the defect (figure 10). This approach was applied to the TW15-1 test case with the sole modification that the defect is now considered infinitely thin. Figure 11 shows the distribution of the current density in the plate around the thin defect when the insulating condition is applied to this surface. A good agreement is obtained between the numerical results and the experimental ones (Figure 12), the difference lying in the effect of the thickness of the real volumetric defect. It follows that this method can be useful by avoiding mesh problems of very thin defects (cracks) which otherwise leads to an increase in the number of elements or convergence problems due to poor mesh quality.
Note : A word of caution when using this condition : the interior Electric Insulation condition is not applied to the edges of the surface. With this in mind, a correct surface representation of the thin defect is obtained by extending the surface of the electrical insulation to a small distance in air above the conductive plate.

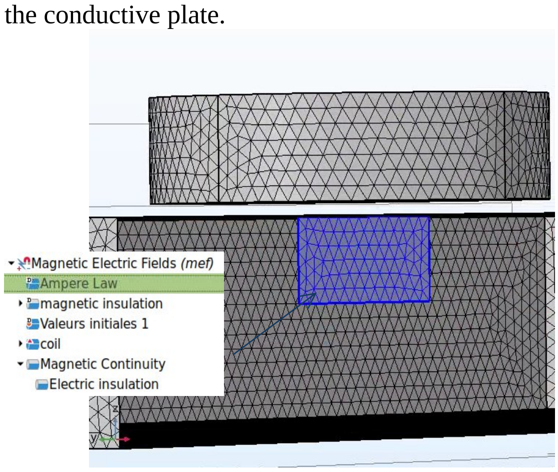

Figure 10. Imposition of the electrical insulation on the defect considered as a surface.

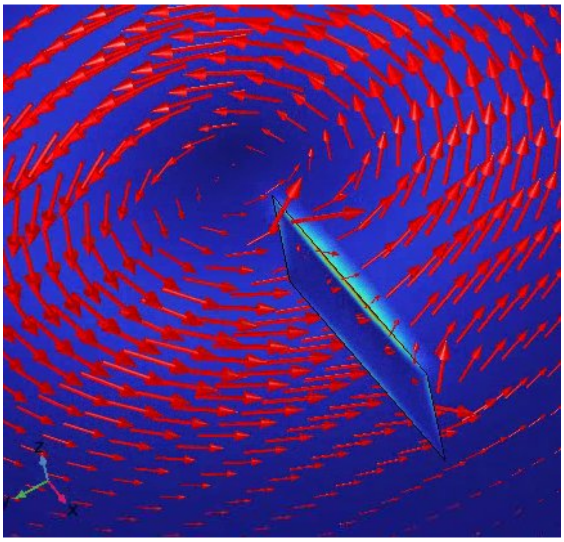

Figure 11. Eddy current density distribution around the defect.

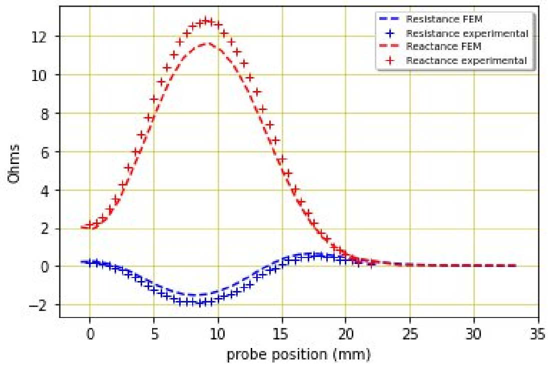

Figure 12. Numerical results for the TW15-1 test case with zero thickness defect.

Evaluation of different parallelization strategies on a cluster for parametric sweep in EC NDT

In EC NDT problems, the same resolution is often reproduced with one or more changes to the physical or geometrical parameters (e.g. variation of the position of the probe or of the defect, of the excitation frequency of the coil or of the electrical conductivity of the specimen). To speedup the solving, it could be interesting to use a cluster to parallelize the computations with a parametric swept resolution.

The application to illustrate this part concerns the identification of 2D mechanical stresses in ferromagnetic materials using EC NDT method [6]. The inversion problem consists in identifying the relevant impedance results on maps obtained by numerical simulations. COMSOL Multiphysics® enables us to generate these impedance maps as a function of the stress state along x and y and the orientation of the sensor in this plane. It has been established that for ferromagnetic materials, the magnetic permeability of the magnetic specimen depends on the stress state [8]. To take into account of this magneto-elastic behavior, a multi-scale model is used. For each state of mechanical stress in xy plane. This model returns a permeability tensor for the material, which is then entered into the COMSOL Multiphysics® software. A 3D magnetodynamic frequency domain model is implemented (AC/DC Module - mf model). To generate impedance maps, parameter swept process is used. That means that the same model is running for different samples of permeability tensors. To speedup the computation, the Ruche Cluster of Mesocentre of Paris-Saclay is used. The scheduler uses by this Cluster is SLURM. First, the scalability of the solution is evaluated with the geometry of the TW15-1 case with no defect.

Two possible ways are possible to improve the scalability of the resolution for a parametric swept study. The first one is to use an Open-MP process and several cores to solve a FEM resolution for one sample of parameters. The parameters used are ***Cpu-per task*** for the SLURM directive and ***np*** in the COMSOL batch command line.

The second one is to use MPI process to run several samples of parameters simultaneously. The parameters are ***ntasks-per-node*** for the SLURM directive and ***nn*** for the COMSOL batch command line.

To well understand the speedup, the same sample of parameters is uses for 20 FEM resolutions. In absolute terms, the speedup should be proportional to the number of cores used.

To observe the influence of the problem size two different meshes are considered. A normal mesh : about 190 000 degrees of freedom (1 parameter : 54 s for the resolution) and a fine mesh with about 1 million degrees of freedom.

The results of Figure 13 confirms a well-known Comsol guideline, namely that there is an optimal number of cores for a finite element resolution (between 4 and 8 cores).

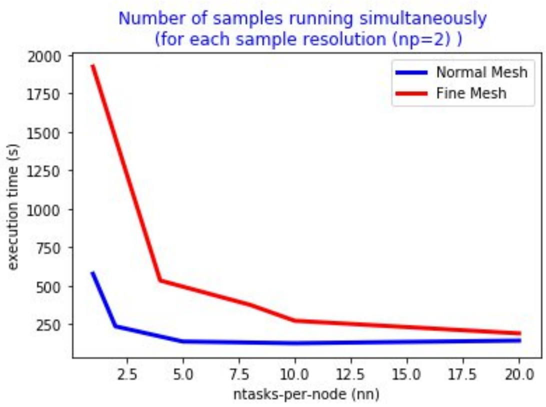

Figure 13. Numerical results for the TW15-1 case.

Figure 14 shows that the gain is important at the beginning and next a threshold is observed. This the consequence of the fork process treatment depending directly of the problem size. The speedup is more important when the mesh size is important.

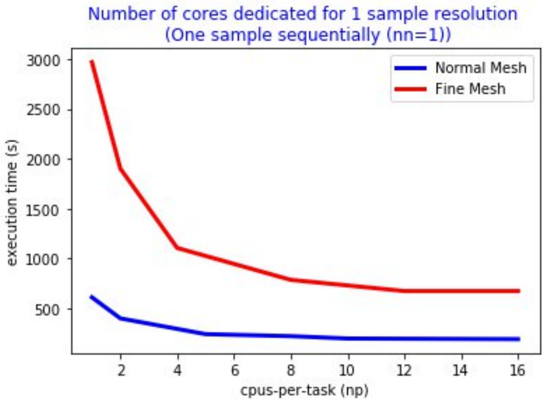

Figure 14. Numerical results for the TW15-1 case.

Conclusions

Beyond the usual capabilities of a conventional FEM software, COMSOL Multiphysics® exhibits several key specifities allowing to considerably simplify the analysis of EC NDT problems.
These interesting features can be also useful to solve low-frequency EMC problems, such as the study of electromagnetic shielding.

References

[1] L. Santandrea and Y. Le Bihan, “Using COMSOL-multiphysics in an eddy current non-destructive testing context”, *Comsol Conference*, Paris, 2010.

[2] A. Kyrgiazoglou, T. Theodoulidis, “Simulation of Eddy Current Non Destructive Testing using COMSOL Multiphysics”, *Comsol Conference*, Rotterdam, 2017.

[3] S. K. Burke, "A Benchmark Problem for Computation of ΔZ in Eddy-Current Nondestructive Evaluation (NDE)," J. Nondestructive Evaluation, pp. 35-41, 1988.

[4] Team Workshop Problem 15, https://www.compumag.org/wp/wp-content/uploads/2018/06/problem15.pdf

[5] P. Burrascano, E. Cardelli, A. Faba, S. Fiori and A. Massinelli, “Numerical Analysis of Eddy Current Non Destructive Testing (JSAEM Benchmark Problem 6- Cracks with Different Shapes)”, *ENDE Conference*, Budapest, 2000.

[6] C.V. Dodd and W.E. Deeds, “Analytical Solutions to Eddy-Current Probe-Coil Problems”, Journal of Applied Physics, 1968.

[7] Y. Choua, L. Santandrea, Y. Le Bihan and C. Marchand, “Thin crack modeling in ECT with combined potential formulations”, IEEE Transactions on Magnetics, 2006.

[8] S. Bouterfas, Y. Le Bihan, L. Santandrea and L. Daniel, “Mechanical Stress Identification Method in Anisotropic Ferromagnetic Materials Using Eddy Current Testing”, *IEEE Transactions on Magnetics*, 2023.

Acknowledgements

This work was performed using computational resources from the“Mésocentre” computing center of Université Paris-Saclay, CentraleSupélec and École Normale Supérieure Paris-Saclay supported by CNRS and Région Île-de-France (https://mesocentre.universite-paris-saclay.fr/).
The authors thank COMSOL support for his reactivity and his efficient help.