CFD MODELING OF ITER CABLE-IN-CONDUIT SUPERCONDUCTORS. PART I: FRICTION IN THE CENTRAL CHANNEL

R. Zanino, S. Giors, and R. Mondino

Dipartimento di Energetica, Politecnico
Torino, 10129, Italy

ABSTRACT

In this paper, the first of a series, we propose a novel approach, based on Computational Fluid Dynamics (CFD), to understand the complex transverse thermal-hydraulic processes in the dual-channel cable-in-conduit conductors (CICC), which are used for the superconducting magnets of the International Thermonuclear Experimental Reactor (ITER). Advanced 2D and 3D CFD, including sophisticated turbulence models, is used to *compute* the mass flow rate corresponding to an imposed pressure drop in rib-roughened pipes, including spirals mimicking the central channel of an ITER CICC and used in several experiments. The results of the calculation are validated against measured data and can be used to *deduce* the friction factor f_H in the central channel, throwing at the same time some light on the role played by the different parameters (Reynolds number, spiral geometry, etc.) in the central channel friction process for an ITER CICC.

KEYWORDS: Fusion reactors, ITER, Superconducting magnets, Computational methods in fluid dynamics, Turbulence simulation and modeling
PACS: 28.52.-s, 84.71.Ba, 47.11.+j, 47.27.Eq, 47.60.+i

INTRODUCTION

In the superconducting magnets of ITER, supercritical helium in forced convection flows both in the voids of the annular region where the cable bundle is present – a porous-medium-like structure, and in the lower hydraulic impedance central channel (hole), delimited by a spiral, see FIG. 1. Assessment of the pressure drop along the CICC is crucial in the determination of the pumping cost of the coolant and this requires suitable correlations for the friction factors in each region. The latter influence also the flow repartition in the CICC, i.e., the fraction of flow directly available for the cooling of the

CP823, *Advances in Cryogenic Engineering: Transactions of the Cryogenic Engineering Conference - CEC, Vol. 51,* edited by J. G. Weisend II

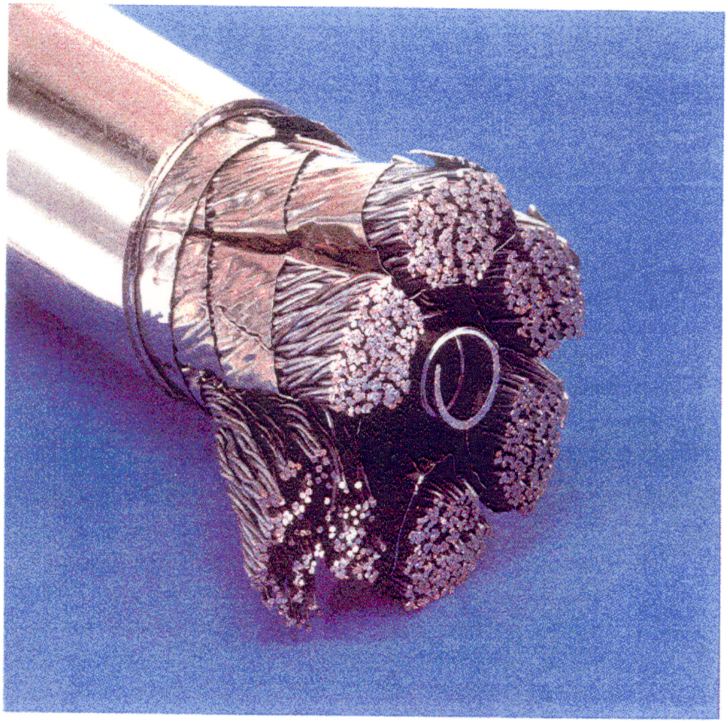

FIGURE 1. Exploded view of an ITER CICC (inner hole diameter is 10 mm in this case).

cable, and are at the same time major input ingredients for the customarily 1D (axial) modeling of ITER CICC's, see, e.g., [1].

The traditional approach to the determination of these friction factors requires at least another independent set of measurements, besides the hydraulic characterization (pressure drop Δp on a pipe length L vs. total mass flow rate) *of the whole* CICC [2], in order to determine the mass flow rate in the central channel *dm/dt*. Either the central channel is blocked, and *dm/dt* is deduced by difference, see, e.g., [3], or a spiral rib-roughened pipe can be used to mimic the real central channel, see, e.g., [4]. The friction factor in the central channel in which we are most interested here, is then defined as $f_H \equiv \frac{\Delta p}{2L}\frac{D_h \rho A^2}{(dm/dt)^2}$, where D_h is the hydraulic diameter, A the flow area [1], ρ is the fluid density. However, notwithstanding their apparent simplicity, these measurements do not always lead to an unequivocal picture [2] so that, in parallel to the experimental efforts conducted worldwide, it may be worthwhile attempting also other approaches.

Here we propose a novel general approach to the thermal-hydraulic modeling of ITER CICC's. While axial processes are fully retained in the typically 1D (axial) models, transverse processes require a lumped parameter description, which is given in terms of friction factors, heat transfer coefficients, etc., since global 3D modeling of the whole

[1] In order to be consistent with the present ITER design criteria, we define here both the hydraulic diameter and the flow area based on the central channel outer diameter D_{out} (i.e., including the spiral thickness).

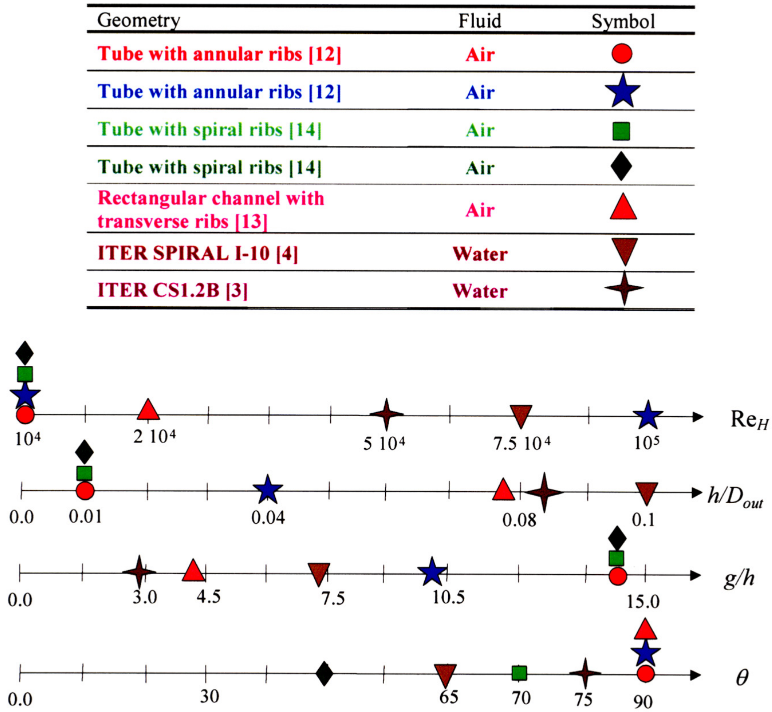

FIGURE 2. Parameters defining the channel geometries (annular, transverse and spiral repeated ribs) used for model validation.

CICC is obviously unpractical. Here we propose to use sophisticated CFD tools (the FLUENT code in the present case [5]) to provide a *local* 2D/3D description of the CICC, from which constitutive relations for the relevant transverse transport fluxes can be first deduced and then applied to *global* 1D modeling.

For the present paper we restrict our attention as a first application of the above approach to friction in the central channel. The role of the central channel geometry (spiral width w, thickness h and gap g, as well as outer diameter D_{out} and spiral angle θ) was relatively early recognized as influencing the pressure drop [6] and will be assessed here.While our approach has to assume a definite geometry of the CICC, it might also be used to computationally study the effect of geometrical irregularities and/or deviations from ideal/design conditions, as they were recently observed, e.g., for some central channel spirals [7], on the thermal-hydraulic performance of the CICC.

MODEL DESCRIPTION

The FLUENT code solves a finite volume approximation of the Navier-Stokes equations in several space dimensions, provided a suitable grid is preliminarily built on the computational domain.

Flow in the central channel is typically turbulent, with Reynolds number $\mathrm{Re}_H \equiv \frac{\rho \bar{u} D_h}{\mu} > 10^4$, μ being the fluid viscosity, $\bar{u}$ the average flow velocity, and incompressible (Mach number < 0.1) for typical ITER coil operating conditions (supercritical He @ temperature $T \sim 5$ K, pressure $p \sim 0.5$ MPa, $dm/dt \sim 10$ g/s, $D_{out} \sim 9$-12 mm). Also, we shall make the assumption (which is always very well verified in the experiments referred to below) that the flow is isothermal, with constant transport properties.

Here we are interested in fully developed turbulent flow in ducts having stream-wise periodic variations of the cross-section (due to the gaps of the spiral, in the case of ITER CICC). Separation and possibly reattachment phenomena occur downstream of the obstacles (e.g., helical ribs) inserted on the inner surface of the duct. This flow feature will be the major item in the choice of the turbulence model below.

Choice of the turbulence model

We do not attempt here a direct simulation of the turbulent flow field. Among FLUENT features we choose a so-called Reynolds-Averaged Navier-Stokes (RANS) approximation to the simulation of turbulence [8]. This approach consists in averaging the Navier-Stokes equation on a period of time much longer than the time scale of turbulent fluctuations, which are filtered away. The filtering procedure results in having additional dependent variables (Reynolds stress tensor), and a proper "closure" (the Boussinesq approximation in our case) must be used to express them as a function of the time-averaged original dependent variables. In the Boussinesq approximation, the so-called

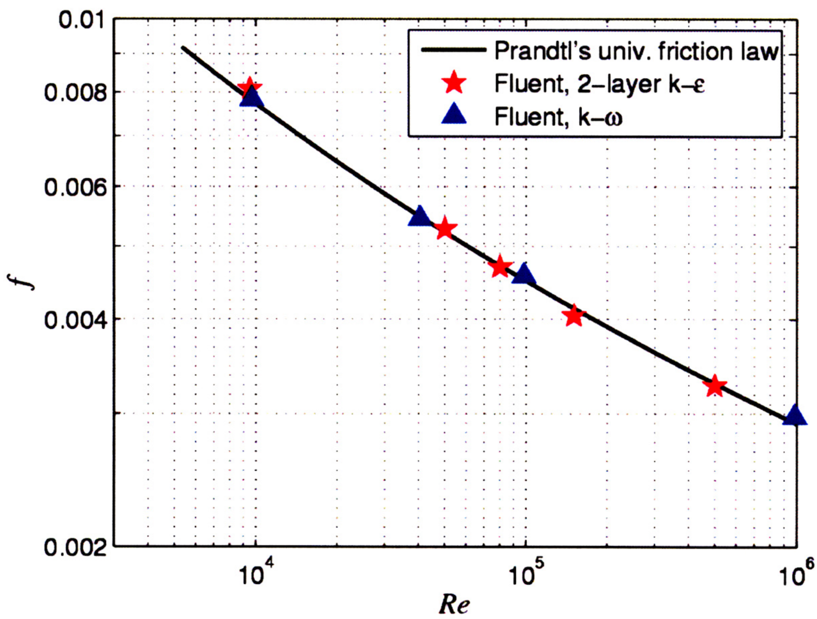

FIGURE 3. Validation of simulations against smooth-tube correlations. Friction factor *f* vs. Reynolds number.

eddy viscosity is introduced, which is a function of the flow field.

Within this framework, different possible options are available to model the eddy-viscosity. Here we choose two which, based on the available literature, appear most promising for problems with flow separation:

- The 2-layer k-ε model of Chen and Patel [9],
- The k-ω model of Wilcox [10],

where k is the turbulent kinetic energy, ε is the turbulent energy dissipation rate and $\omega \propto \varepsilon/k$.

The 2-layer model of Chen and Patel divides the flow domain in two regions: a near-wall region, including the viscous sub-layer, the buffer layer and a small part of the turbulent core, where a one-equation model is solved to compute the eddy viscosity; the rest of the fully developed turbulent core region, where the standard k-ε model is used to calculate the eddy viscosity as a function of k and ε.

The k-ω model of Wilcox is a two-equation model, specifically developed to accurately predict non-equilibrium, adverse pressure gradient flows. Here the eddy viscosity is computed as a function of k and ω.

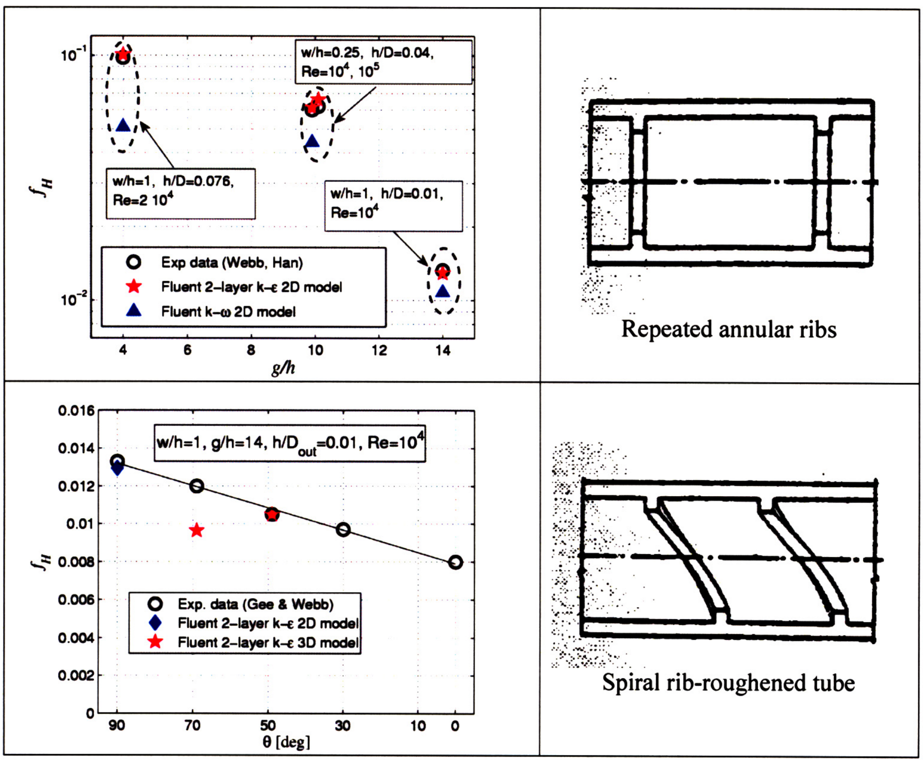

FIGURE 4. Comparison of simulations and measurements in the case of repeated annular ribs (2D) and spiral rib-roughened tubes (3D).

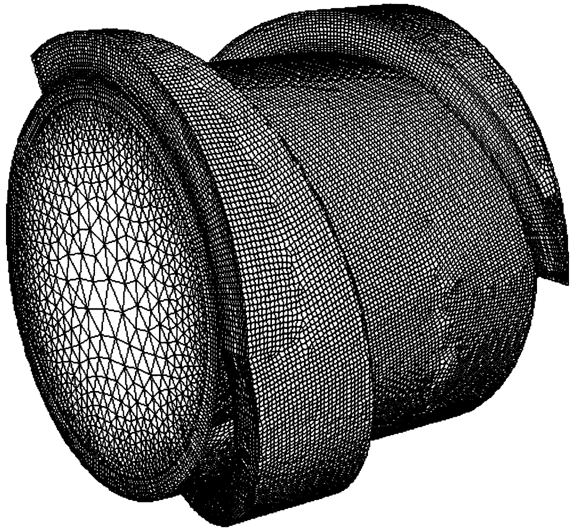

FIGURE 5. Computational 3D grid for FLUENT analysis of CS1.2B spiral. Note the use of hexahedra in the gap and boundary layers, while tetrahedra are used in the core flow region.

MODEL VALIDATION

We perform here a fairly comprehensive set of validation exercises of the FLUENT model, using two selected turbulence models. We start from the smooth tube, for which very accurate correlations are available. We then move to the case of repeated ribs with both annular/transverse (2D) and spiral (3D) geometry, taken from the available (non-ITER) literature (mainly in the field of turbulence promoters for compact heat exchangers) and finally compare the results of our simulations with measurements performed on both an ITER spiral and on the central channel of an actual ITER CICC. The set of geometries considered, together with the respective most relevant parameters, is summarized in FIG. 2. Also, different fluids (air and water) were used in these tests, and we shall assume constant thermodynamic and transport properties.

In all cases, grid-independence of the solution was verified numerically. Grid-independence of 2-layer k-ε and k-ω RANS solutions requires a grid able to resolve both the recirculation regions, where present, and the boundary layer including the viscous sub-layer. This typically requires grids whose first cell adjacent to the wall has $y^+ < 1$ ($y^+ = \rho y V^* / \mu$ is a Reynolds number based on the distance y of the cell center from the wall and the velocity $V^* = \sqrt{\tau_w / \rho}$, τ_w being the wall shear stress).

Smooth tube

This is quite obviously the simplest but already relevant test for our approach. A 2D (axisymmetric) periodic model is built. For any given pressure drop applied on the unit (periodic) length, the corresponding mass flow rate is *computed* by FLUENT and the

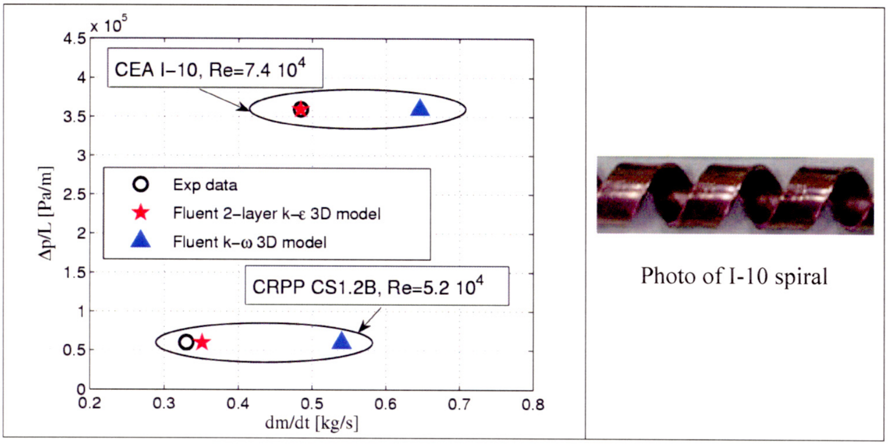

FIGURE 6. Comparison of simulations and measurements in the case of ITER spirals.

friction factor correlation is *deduced* from this computational hydraulic characterization. Results of the simulations are shown in FIG. 3, indicating a very good agreement with Prandtl's universal law of friction for smooth tubes, $\frac{1}{2\sqrt{f}} = 2.0 Log\left(2\,\mathrm{Re}\sqrt{f}\right) - 0.8$, for both turbulence models, in the range $10^4 < \mathrm{Re} < 10^6$.

Repeated ribs (annular and spiral)

In view of their relevance as turbulence promoters, there is a long history of measurements and analysis of repeated ribs, starting from the pioneering work of Webb [12]. Data from [12] and [13] are used here for the case of annular and transverse ribs, respectively, data from [14] for the case of spiral rib-roughened pipes. Simulations, see FIG. 4, show a very good agreement with the measurements in the case of annular ribs, particularly when the 2-layer k-ε turbulence model is used, whereas in the case of the spirals the agreement is very good for the 49 degree spiral case but worse (~ 20 % error) for the 70 degree case (notwithstanding its less 3D nature).

ITER spirals

As last and most ITER-relevant application we consider here two cases: a) the so-called I-10 spiral, recently tested at CEA [4], and b) the CS1.2B conductor, tested at CRPP [3]. An example of the complex, mixed hexahedral-tetrahedral, mesh used for the simulation of the latter is shown in FIG. 5. Results presented in FIG. 6 confirm the very good agreement between simulations and measurements when the 2-layer k-ε turbulence model is used.

CONCLUSIONS AND PERSPECTIVE

A novel approach has been proposed for the hydraulic analysis of ITER CICC, based on Computational Fluid Dynamics. The FLUENT code was applied to the problem of turbulent flow in several geometries relevant to the central channel of ITER conductors. 2D and 3D simulations show very good agreement with measured data in a broad range of operating conditions, when the 2-layer k-ε turbulence model is used.

This approach allows therefore to reliably *compute* the friction factor in the ITER central channel as a function of Reynolds number as well as spiral geometry. In perspective the same approach will be applied to the study of other thermal-hydraulic issues of ITER CICC, e.g., friction in the bundle, heat transfer to the central channel, advection between bundle and central channel.

ACKNOWLEDGEMENTS

This work is supported by EFDA and MIUR.

REFERENCES

1. Zanino, R., DePalo, S., and Bottura, L., "A Two-Fluid Code for the Thermohydraulic Transient Analysis of CICC Superconducting Magnets", *J. Fus. Energy* **14** (1995) 25-40.
2. Zanino, R. and Savoldi Richard, L., "A review of thermal-hydraulic issues in ITER cable-in-conduit conductors", submitted to *Cryogenics* (2005).
3. Bruzzone, P., "Pressure drop and helium inlet in the ITER CS1 conductor", *Fus. Eng. Des.* **58-59** (2001), 211-215.
4. Nicollet, S., et al., "Task CODES: Results of ITER type central spirals friction factor measurements in the OTHELLO facility and application for ITER Coils", CEA Report AIM / NTT- 2003.018.
5. Fluent 6.1 User's Guide, Fluent Inc., Lebanon Pennsylvania, 2003.
6. Zanino, R., Santagati, P., Savoldi, L., Martinez, A., and Nicollet, S., "Friction factor correlation with application to the central cooling channel of cable-in-conduit super-conductors for fusion magnets", *IEEE Trans. Appl. Supercond.* **10** (2000) 1066-1069.
7. Zapretilina, E., "Some results of the TFCI destructive examinations", ITER Conductor Meeting, Muerren, 25-28 January 2005
8. Wilcox, D. C., *Turbulence modeling for CFD*, DCW Industries Inc., La Cañada, 1998, pp. 35-36.
9. Chen, H. C., and Patel, V. C., "Near-Wall Turbulence Models for Complex Flows Including Separation", *AIAA Journal* **26/6,** pp. 641-648 (1988).
10. Wilcox, D. C., "Reassessment of Scale-Determining Equation for Advanced Turbulence Models", *AIAA Journal* **26/11,** pp. 1299-1310 (1988).
11. Driver, D. M., and Seegmiller, H. L., "Features of a Reattaching Turbulent Shear Layer in Divergent Channel Flow", *AIAA Journal* **23/2,** pp. 163-171 (1985).
12. Webb, R.L., Eckert, E. R. G., and Goldstein, R. J...., "Heat transfer and friction in tubes with repeated-rib roughness", *Int. J. Heat Mass Transfer* **14**, pp. 601-617 (1971).
13. Han, J.C., Glicksman, L.R., and Rohsenow, W. M., "An investigation of heat transfer and friction for rib-roughened surfaces", *Int. J. Heat Mass Transfer* **21**, pp. 1143-1156 (1978).
14. Gee, D.L., and Webb, R.L., "Forced convection heat transfer in helically rib-roughened tubes", *Int. J. Heat Mass Transfer* **23**, pp. 1127-1136 (1980).