Cryogenics 50 (2010) 158–166

Contents lists available at ScienceDirect

Cryogenics

journal homepage: www.elsevier.com/locate/cryogenics

CFD model of ITER CICC. Part VI: Heat and mass transfer between cable region and central channel

R. Zanino *, S. Giors, L. Savoldi Richard

Dipartimento di Energetica, Politecnico, Torino, Italy

ARTICLE INFO

Article history:
Received 16 January 2009
Received in revised form 26 October 2009
Accepted 28 November 2009

Keywords:
ITER
Superconducting magnets
Cable-in-conduit conductors
CFD
Friction factor
Heat transfer coefficient

ABSTRACT

Dual-channel cable-in-conduit conductors (CICC) are used in the superconducting magnets for the International Thermonuclear Experimental Reactor (ITER). As the CICC axial/transverse size ratio is typically ~1000, 1D axial models are customarily used for the CICC, but they require constitutive relations for the transverse fluxes. A novel approach, based on Computational Fluid Dynamics (CFD), was recently proposed by these authors to understand the complex transverse thermal–hydraulic processes in an ITER CICC from first principles. Multidimensional (2D, 3D) Reynolds-Averaged Navier–Stokes models implemented in the commercial CFD code FLUENT were validated against compact heat exchanger and ITER-relevant experimental data, and applied to *compute* the friction factor and the heat transfer coefficient in fully turbulent spiral rib-roughened pipes, mimicking the central channel of an ITER CICC. That analysis is extended here to the problem of heat and mass transfer through the perforated spiral separating the central channel from the cable bundle region, by combining the previously developed central channel model with a porous medium model for the cable region. The resulting 2D model is used to analyze several key features of the transport processes occurring between the two regions including the relation between transverse mass transfer and transverse pressure drop, the influence of transverse mass transfer on axial pressure drop, and the heat transfer coefficient between central channel and annular cable bundle region.

1. Introduction

Multi-channel cable-in-conduit conductors (CICC), see Fig. 1, will be used in the superconducting magnets for the International Thermonuclear Experimental Reactor (ITER) [1]. Supercritical helium (SHe) coolant flows both in the annular region, where the cable bundle is present, and in the central channel, delimited by a (perforated) spiral.

As the CICC axial/transverse size ratio is typically $\geqslant 10^3$ in a coil, 1D (axial) models [2], or combinations thereof [3,4] to approximately treat the actually multi-dimensional situation, are customarily used for the sake of sparing CPU time, but they require constitutive relations for the transverse fluxes, including friction factors and heat transfer coefficients.

Unfortunately, however, the wide database available at present on, e.g., friction factors, is neither fully comprehensive nor free of contradictions/ambiguities [5–7].

Therefore, a novel approach, based on Computational Fluid Dynamics (CFD), was recently proposed [7–11] to improve our understanding of the complex *local* transverse thermal–hydraulic processes in an ITER CICC and obtain transverse constitutive relations to feed the *global* 1D codes. So far our modeling was mainly restricted to the central channel region, with the only exception of a study of friction in the simplest possible cable bundle, made of a single triplet [10].

In particular, 2D and 3D simulations were used in [7] to confirm that the 2-layer k–ε turbulence model [12] is the most suitable for the separated flow problems relevant in the central channel case, where detachment occurs at the leading edge of the spiral and reattachment of the flow may occur in the gap, depending on the gap width over height (g/h) ratio. The influence of the different geometrical parameters (g, h, central channel diameter) on friction was studied in [8], with particular reference to the dependence on g/h and to synergistic effects. For the ITER CS and TF relevant 7 mm/9 mm (ID/OD) central channel, a correlation was developed in [9] based on the results of our computational experiments, predicting a lower pressure drop than the present ITER design criteria, for a given total mass flow rate. Finally, in [11] the Colburn analogy between friction and heat transfer was shown *not* to be verified in the case of the CICC central channel and the computed heat transfer coefficient *on the central channel side* turned out to be not much larger than the smooth-pipe Dittus–Boelter value.

* Corresponding author.
E-mail address: roberto.zanino@polito.it (R. Zanino).

0011-2275/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.cryogenics.2009.11.005

Nomenclature

dm/dt, $\dot{m}$	mass flow rate
f	friction factor
g	spiral gap width
h	spiral thickness
k	turbulent kinetic energy
pt	spiral pitch
p	pressure
$\tilde{p}$	pressure modulation
r	radial coordinate
v	flow velocity
v_{mag}	velocity magnitude
w	axial width of the spiral
x	spatial coordinate along conductor axis
A	cross section
C_p	specific heat at constant pressure
D_{in}	inner diameter of the central channel
D_{Jk}	diameter of the cable region (inner diameter of the jacket)
H	heat transfer coefficient
J	inertial constant
K	permeability
L	length along conductor
NP	non periodic model
Nu	Nusselt number
Q	heat source
P	periodic model
Re	Reynolds number
S	porous medium source term in momentum Navier–Stokes equations
SHe	supercritical helium
T	temperature

Greeks

δ_{ij}	Kronecker delta
ε	turbulent dissipation rate
μ	viscosity
ρ	density
θ	normalized temperature
ϕ	arbitrary scalar quantity
Γ	radial mass flux
Δp	pressure difference
$\langle \Delta p \rangle$	volume averaged pressure difference
$\langle \Delta p_0 \rangle$	volume averaged pressure difference obtained from the periodic model

Subscripts

B	bundle
H	central channel (hole)
n	unit vector perpendicular to the control surface
r	radial component
B→H	bundle to hole
In	inlet
Jk	jacket
Out	outlet
tot	total
$\perp$	transverse

The commercial CFD code FLUENT was used for all of these studies and the results of the code were validated showing good agreement against experimental data in many different (2D and 3D) geometries: the computed friction factor in the central channel f_H turned out to be within ~±15% error bar from the measured values, while the computed Nusselt number on the central channel side Nu_{H} resulted within ~±30% error bar from the measured values.

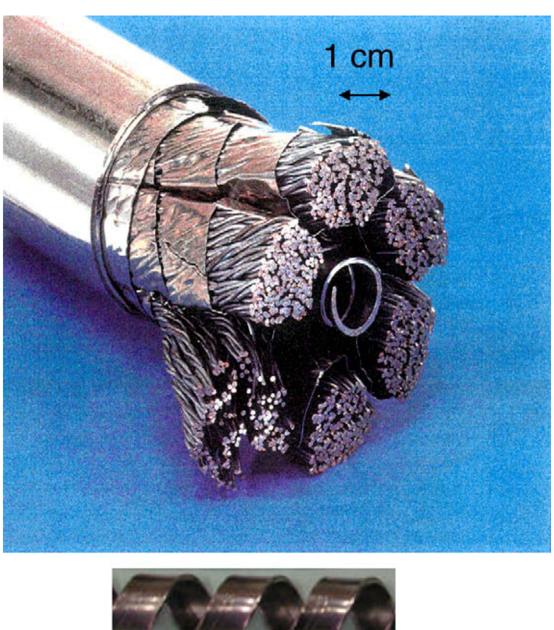

Fig. 1. A typical ITER CICC (top) and the spiral delimiting its central channel (bottom).

Here we extend the approach developed in the previous papers of this series by including two substantially new features: (1) the annular cable region, treated as a porous model, so that the entire CICC cross section available for the SHe flow is now treated by the model; (2) the perforation of the spiral delimiting the central channel, so that mass transfer is now allowed between the two CICC regions. Inclusion of a net mass transfer through the spiral has also a major implication, i.e., the impossibility to assume a periodic model in the axial direction as done so far, see below.

2. Model equations and boundary conditions

The commercial CFD code FLUENT is used in this paper, assuming azimuthal symmetry, i.e., a 2D (axial/radial) cylindrical geometry, which was shown in the past [7–9] to be a very good approximation of the real 3D geometry of an ITER CICC. The entire cross section of the CICC is modeled, see Fig. 1, and the geometrical data used in the simulation are listed in Table I, where g is the gap size, h is the spiral thickness, w is the spiral (axial) width, $pt = g + w$ is the spiral pitch, D_{in} is the inner diameter of the central channel and D_{Jk} is the diameter of the cable region (inner diameter of the jacket).

For the central channel the same model as in the previous papers of this series is used, i.e., the Reynolds-Averaged Navier–Stokes (RANS) with 2-layer k–ε model as turbulence closure.

The bundle region is treated as a porous medium by adding the following source term to the i-th component of the Navier/Stokes momentum equations:

$$S_i = -\left(\sum_{j=1}^{3} D_{ij} \mu v_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho v_{mag} v_j \right) \quad (1)$$