Finite Element Simulation of Induction Heating of a Tubular Geometry

K Madhusoodanan, J.N. Kayal and P.K. Vijayan

Bhabha Atomic Research Centre, Trombay, Mumbai 400 085.

Introduction: 2D axi-symmetric analysis of induction heating of a tubular geometry has been simulated using COMSOL. Split type and pancake type coils along with ferrite flux concentrator has been used. The objective has been to heat the tube with minimum temperature differential.

Material of pressure tube is Zr 2.5 wt% Nb

Variable	Value
Thermal conductivity (W/m.K)	16.64+0.01436xT
Density (kg/m^3)	6500
Specific heat (J/kg. K)	278+0.08205xT
Electrical conductivity (S/m)	10^8/(51.54+0.1384xT)
Relative magnetic permeability	1

Typical split type coil.

Finite element analysis: The option: Electro Thermal Interaction→Azimuthal Induction Heating (Vector potential)→Transient Analysis, available with AC/DC Module of COMSOL has been used for the analysis.
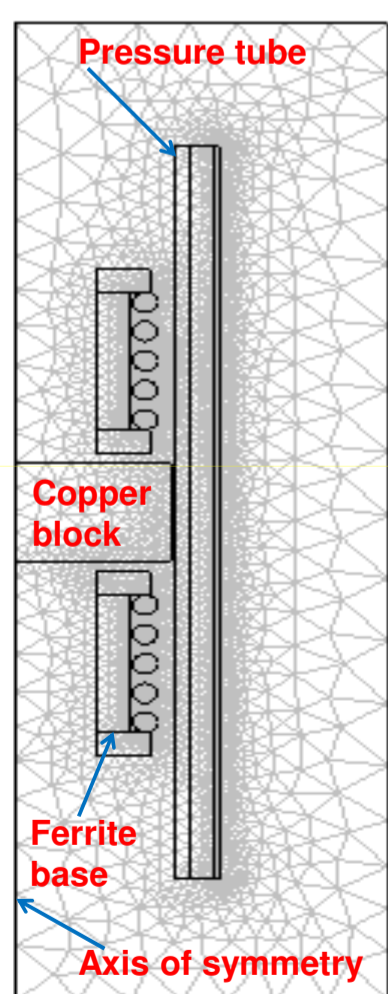
Results: Analysis has been done at 18 kHz frequency: Current flow: 115 A for split type coil and 175 A for pancake type coil.
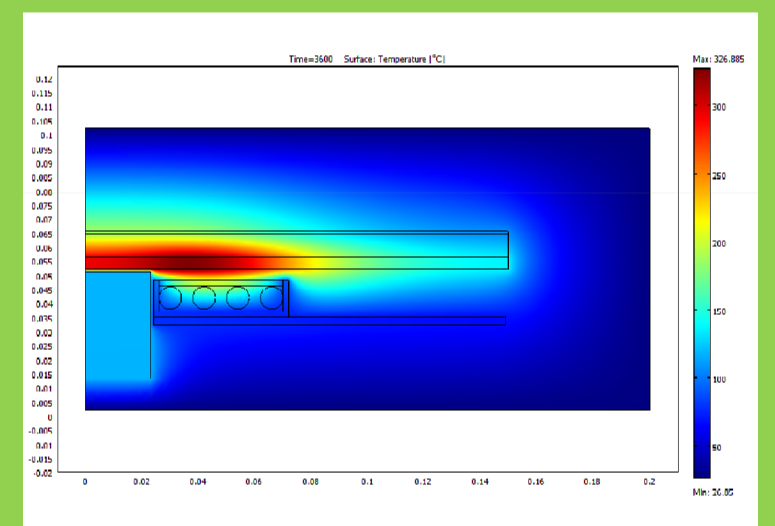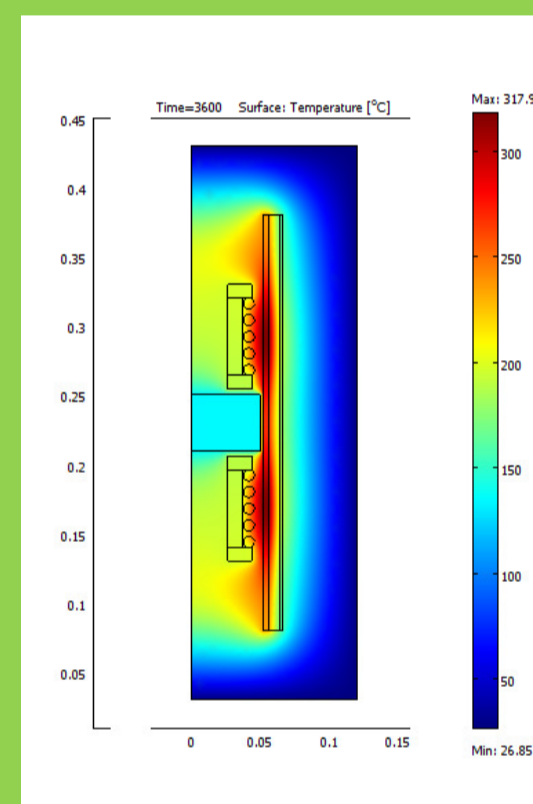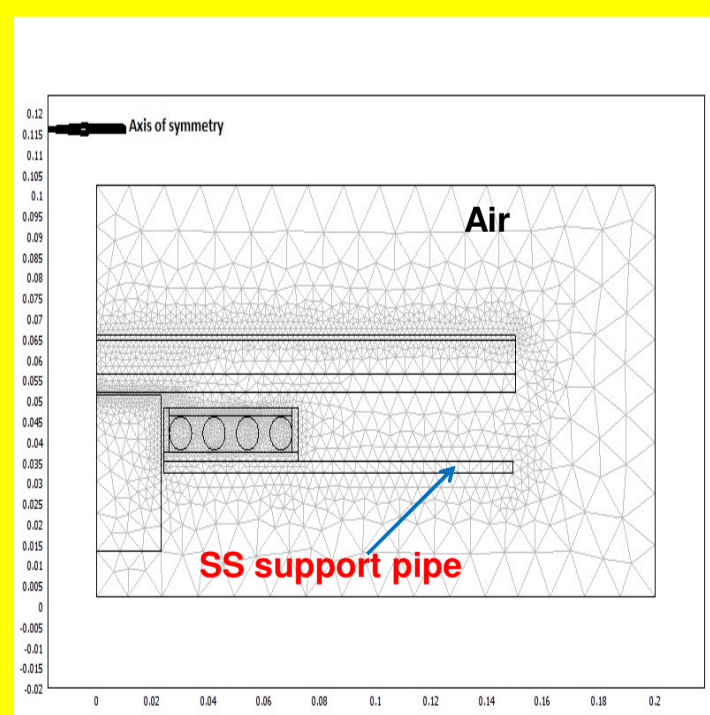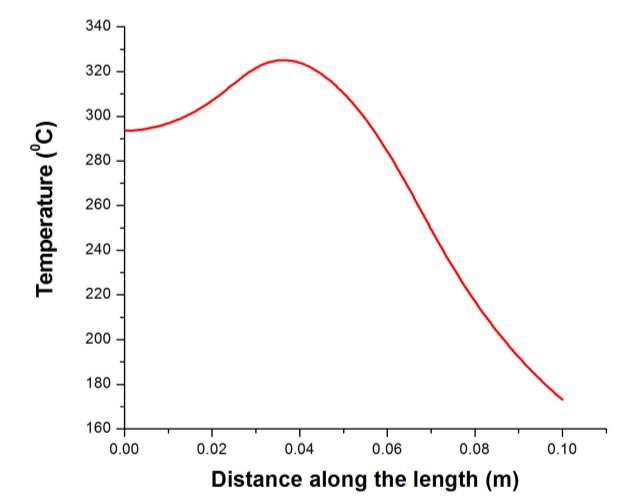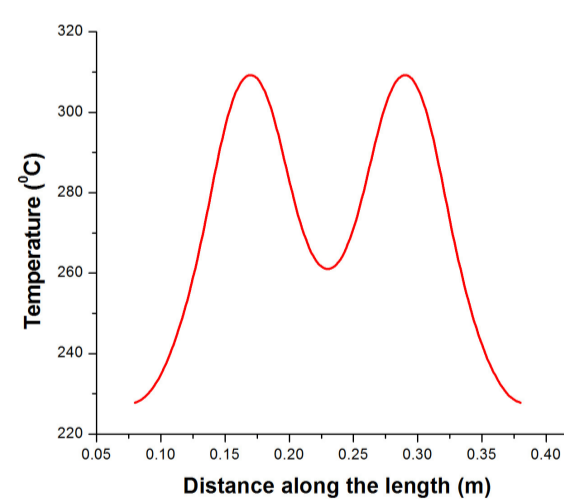

Temperature distribution at the end of one hour

Finite element models

Split type coil

Pancake type coil

Temperature gradient

Conclusions: Parametric studies carried out indicate that pancake type design is better for the application. Use of ferrite base enhances the efficiency of heating considerably and also avoids the heating of the structural pipe and copper block present inside the coil.