ANSYS - 3D Conduction

Author: Samuel Smith, Cornell University

3D Conduction

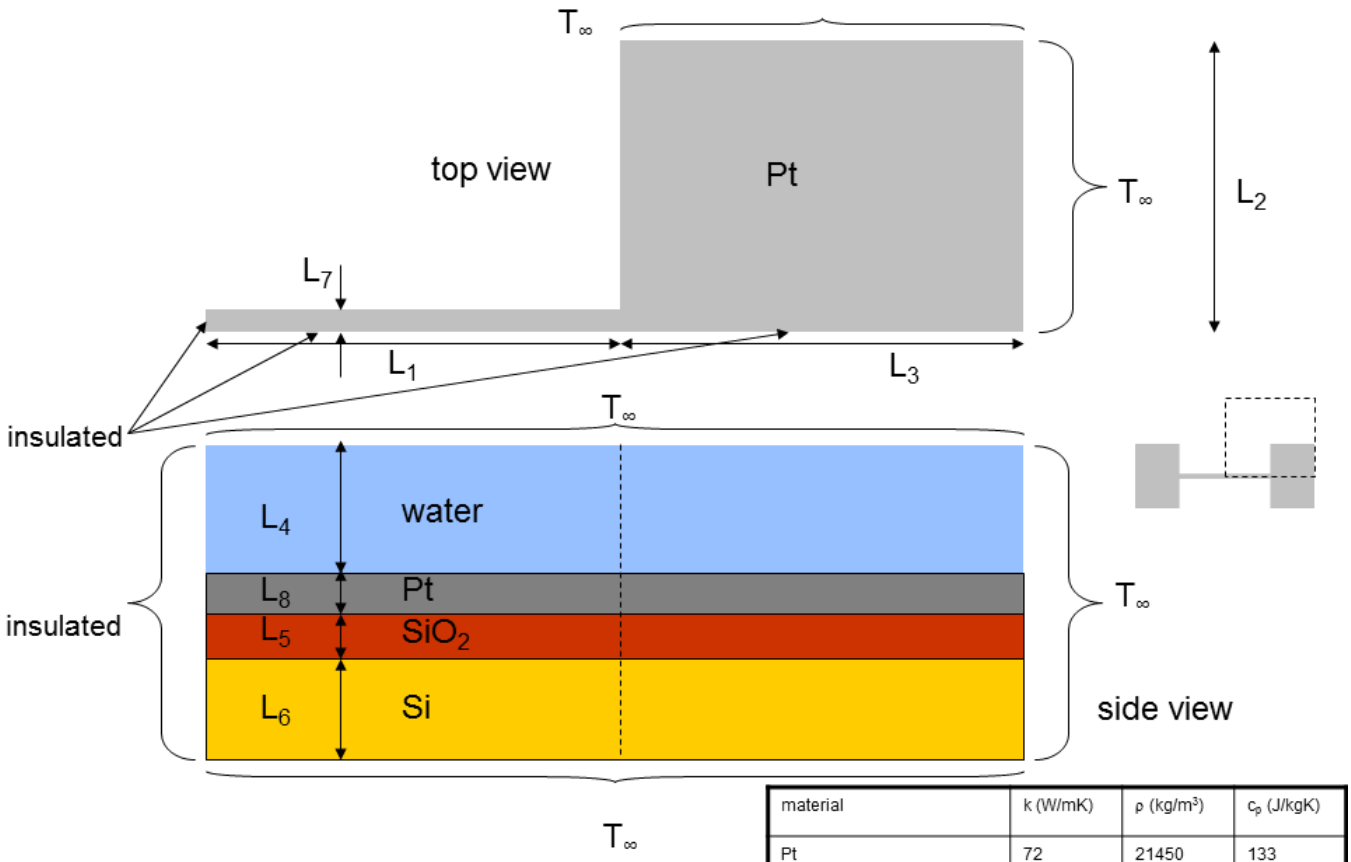
Created using ANSYS 13.0

Problem Specification

This is a design-problem simulation. We have a Platinum micrometer bridge deposited onto a Silicon wafer (with a Silicon Dioxide insulation layer) submersed in a pool of stagnant water.

The small platinum bridge will have electricity passing through it, generating 4 Watts of heat uniformly across it. We need to determine how long we can keep the electricity active before the hottest portion of the platinum becomes 570 K.

Notice that this bridge has 2 planes of symmetry, thus we need only model 1/4 of the body! This is definitely something we need to take advantage of to ease the computationally intensive project.



material	k (W/mK)	ρ (kg/m³)	c_p (J/kgK)
Pt	72	21450	133
SiO_2	6	2650	745
Si	148	2330	712
Water @ 440 K	0.682	900	4360

NOT TO SCALE
L_1=100 µm;$L_{2,3}$=250µm; L_4=variable; L_5=200nm; L_6=300µm
L_7=1.5 µm; L_8=200nmT_∞=300K; P=1W; @ t=0, T=T_∞ everywhere

Go to Step 1: Pre-Analysis & Start-Up

Go to all ANSYS Learning Modules