The problem asked to be solved was a hot tap heat transfer problem involving a repair sleeve being welded onto a cylindrical pressurized pipe filled with a flowing process fluid. I was asked to consider a stainless steel pipe with a density $\rho = 0.284\frac{in}{ft^2}$, specific heat capacity, $c_P = 0.119\frac{BTU}{lb-F}$, and thermal conductivity, $k = 31.95\frac{BTU}{hr-ft-F}$. The heat transfer on both the inside and outside of the surfaces operate under a standard Neumann boundary, with heat flux proportional to the difference between the surface temperature and a reference temperature. For the outer G-Ambient surface, the heat transfer coefficient and reference temperature are denoted by $h_{ambient} = 9.0\frac{BTU}{hr-ft^2-F}$, and $T_{ambient} = 70°F$. The G-Process surface is denoted by $h_{process} = 48.0\frac{BTU}{hr-ft^2-F}$ and $T_{process} = 325°F$.

Also involved in the thermal model is a heat source term, $f(r, z, t) = 2700(\frac{1}{2}(1 + \cos(t + 5)(\frac{\pi}{5}))\frac{BTU}{s-in^3}, \forall x \in W$ and $f(r, z, t) = 0\frac{BTU}{s-in^3}, \forall x \notin W$. The problem presents an axisymmetric cross section about a horizontal axis of rotation. Also, the thermal history during welding is determined by using some numerical method for solving the heat equation within the stainless steel pipe.

For solving this problem I started with the 2-Dimensional heat equation with r being the radial coordinate, z the longitudinal coordinate, and t as time. This produced the equation:

$$u_t = \alpha \Delta u = \frac{k}{c\rho}(u_{rr} + u_{zz}) + f(r, z, t). \tag{1}$$

eq. 1

The obvious choice for solving such a problem would be to utilize the finite element method to generate an approximate solution for the heat transfer. I chose to use the PDE toolbox in Matlab to construct and solve, by finite element method, the thermal model. Utilizing the boundary conditions, thermal properties, initial condition, and internal heat source, I was able to generate a solution for the problem. The geometry for the problem is illustrated by the figure below and was the geometry for which the transient thermal model was operating on.

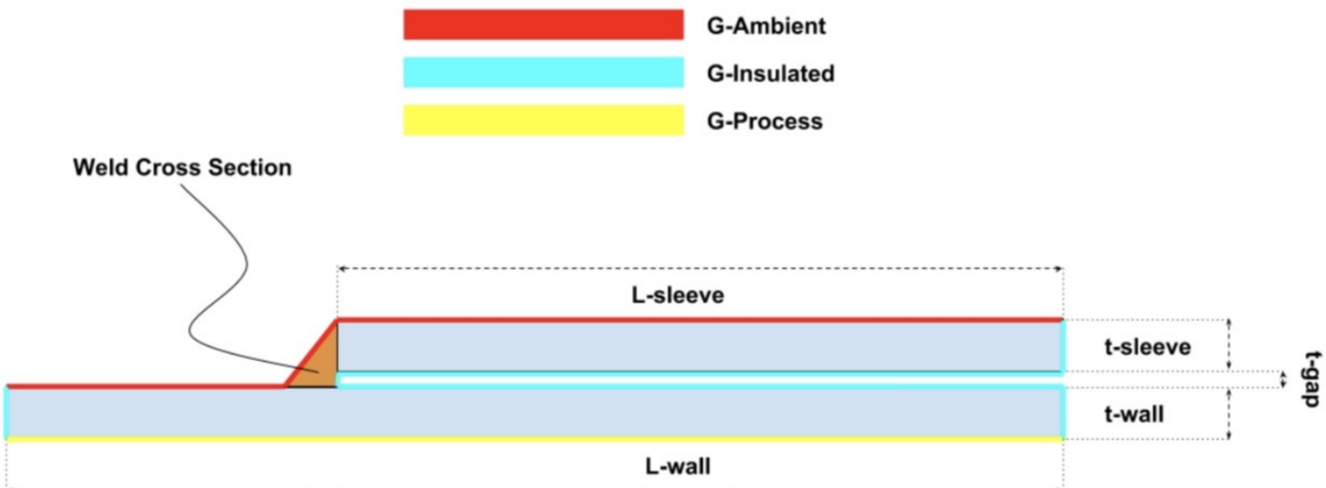

Figure 1: Illustration of the axisymmetric sleeve geometry.

Figure 1: Illustration of the axisymmetric sleeve geometry

The following code was executed to create a solution for the problem

- Thermal Model PDE Toolbox

```
thermalmodel = createpde('thermal','transient-axisymmetric');
pderect([0  2.82  0  0.188])
pderect([0.94  2.82  0.190  0.378])
pderect([0.94  0.95  0.188  0.190])
gd = [pderect  pderect  pderect];
sf = 'pderect+pderect+pderect';
ns = char('pderect','pderect','pderect');
ns = ns';
d1 = descg(gd,sf,ns);
geometryFromEdges(thermalmodel,d1);
thermalProperties(thermalmodel,'ThermalConductivity',31.95,'MassDensity'
thermalBC(thermalmodel,'Edge',[2,3,4,5,6,8],'HeatFlux',0);
thermalBC(thermalmodel,'Edge',1,'ConvectionCoefficient',48,'ProcessTempe
thermalBC(thermalmodel,'Edge',7,'ConvectionCoefficient',9,'AmbientTemper
thermalIC(thermalmodel,0);
internalHeatSource(thermalModel,@heatsource,  'Edge',1)
generateMesh(thermalmodel);

tfinal = 10;
tlist = 0:0.01:tfinal;
result = solve(thermalmodel,tlist);
T = result.Temperature;

figure
subplot(2,1,1)
pdeplot(thermalmodel,'XYData',T(:,6),'Contour','on')
axis equal
title(sprintf('Temperature at %g s',tlist(6)))
subplot(2,1,2)
pdeplot(thermalmodel,'XYData',T(:,end),'Contour','on')
axis equal
title(sprintf('Temperature at %g s',tfinal))


internalHeatSource(thermalModel,@heatsource,  'Edge',4)
function q = heatsource(location,state)
t  = state.time;
```

```
x = location.x;
k = 31.95;
q = 2700*x.*(0.5+0.5*cos((k+5)*(t/5)));
end
```

The preceding code produced the figure:

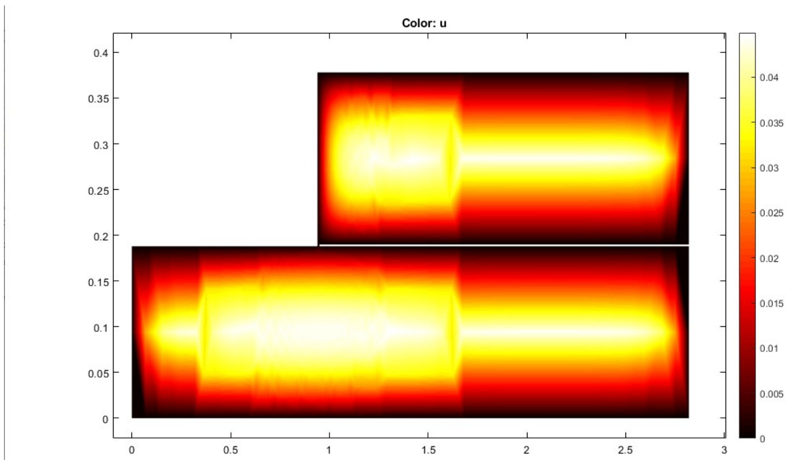

Figure 2: Thermal Flow Produced by Code

Hear we can see heat being transferred throughout the welding sleeve and into the wall. We can see the heat transfer around the insulated boundaries is 0 and the heat increases into the middle of the pipe throughout the 10 seconds being evaluated.