This analysis of the engine block is focused on to study the differences between the temperature gradient when the engine is working uncooled and when the engine is liquid cooled at 225°C.

1. **Import Model**:

   _File > open > Engine_Block_Thermal.hm._

   Material properties _**(Thermal conductivity K, E, mu and Rho)**_ and _**Control Cards**_ are already defined in the model.

2. **Creating Interfaces**:

   Heat flux is generated inside the cylinder, and then conducted from inner surface to outer surface of the cylinder block. Hence, _**Interfaces**_ are used to define the surfaces through which Heat Flux passes.

   Goto _**Analysis Panel > Interfaces > create > Name : conduction**_ and _**Type: CONDUCTION**_ click Create. You can see that new _**Group**_ by name _**Conduction**_ is created in the Model browser.
Let us add all the inner surfaces of the four cylinders into the Conduction group. Now in the same sub-panel click add > toggle slave to Face > Solid Elems by sets > C1 (first Cylinder), Choose nodes as shown below and click add.

Now repeat same steps for C2, C3 and C4. Now your Model looks like this.
3. **Defining wall temperature constraints:**

*Right Click* on the model browser and click create *Load collectors*. Name it as *Spc_Thermal*. Now goto *Analysis panel > constraints* and select *nodes by sets > Block_wall*, you can see nodes created on the boundary.

Now, see that all other parameters match the image below. Click Create.

Now your model looks like this.
4. Defining Wall Temp:

In practical cases, the temperature distribution in an engine block is an important factor while Designing Four-Cylinder Engine. The heat flux generated during ignition is 4 W/mm² but the temperature load on the Engine Block wall is 250°C.

To assign the temperature loads on the Engine Block Wall, we use Card edit.

Click on card edit icon, select Config to CONST and Type to SPC. Let the selector be Load and then, left click and drag to select all the loads created on the wall and click Edit > enter the D value = 250.0. Return & Return.

5. Defining Heat Flux:

Here we define the Heat flux 4 W/mm² generated inside the cylinder. Right click on the Model browser and create load collector, name it as Load Thermal.

Goto Analysis panel > Flux > Elements by group, select Conduction and see that rest of the parameters are set as shown in below image. Click create, your model will look like this.

Thermal Analysis – Heat Transfer
6. Creating Load Steps:

Goto Analysis panel > Load steps > Name = Heat_Transfer, Type = Heat transfer (Steady State). Select Spc and assign to SPC_Thermal and Load to Load_Thermal. Click Create and Return.

7. Defining Cooling Loads:
Here we define the Liquid Coolant constraints.

Make Spc_Thermal Load collector as Current. Goto Analysis panel > constraints > select nodes by sets and check cooling and select. See that all other parameters are same as in below image.
Click create. Your model looks like this now.
8. **Defining Cooling Temp:**

Here we define the temperature of the Liquid coolant, used between the Cylinders.

Click **card edit** icon, select only those loads which are created recently and leave rest all the parameters as shown in the image below. Click **edit** and enter \( D = 225.0 \). **Return & Return**
9. Post Processing:

Goto Analysis > optistruct and click save as and save the file in .fem format and click optistruct, you will get pop up window saying Analysis Complete. Now click on Results and it will take you to another new window.

Click on page view layout icon and select this icon. Now you can see that graphic area is split into two. Import result file with cooling load in one Graphic area and without cooling load in other Graphic area. Click on contour plot icon and set the parameters as shown in below image and see the differences in Temperature Gradients between two models.