Solving the geodetic boundary value problem

Solve the geodetic boundary value problem in the homogenous domain bounded by two spheres with radii 6371km and 6871km, 5° and 50° meridians, and 10° and 50° parallels. The size of the elements is 5° x 5° x 50km. There is the gravitational acceleration applied on the bottom boundary, the zero Neumann boundary condition on the side boundaries and the gravitational potential on the upper spherical boundary. The gravitational constant is GM=398600.4415 km^3.s^(-2), the gravitational potential can be computed as GM/radius, and the formula to calculate the gravitational acceleration can be obtain as derivative of the formula for calculating the gravitational potential.

1. Defining parameters

Ansys Utility Menu > Parameters > Scalar Parameters. In Selection we type GM=398600.4415 and press Accept. In the same way we define Ru=6871, Rd=6371, Lu=50, Ld=5, Bu=50 and Bd=10. All parameters will be seen in section Items in the alphabetical order.

2. Setting the type of the problem

ANSYS Main Menu > Preferences > Thermal > OK.

3. Setting the type of the element

ANSYS Main Menu > Preprocessor > Element Type > Add/Edit/Delete. We click Add..., highlight Thermal solid, then ’Brick 8Node 70’, click OK and close the Element Types window.

4. Setting the material properties

ANSYS Main Menu > Preprocessor > Material Props > Material
Models. We choose Thermal > Conductivity > Isotropic > and in the following window we type KXX 1. We click OK and close the window.

5. Creating of the computational domain

We start by changing the coordinate system to the spherical one:

ANSYS Utility Menu > WorkPlane > Change Active CS to > Global Spherical

the same can be done by typing CSYS, number of the coordinate system:

CSYS, 2

Then we create 8 points that define the whole computational domain:

ANSYS Main Menu > Preprocessor > Modelling > Create > Keypoints > On Working Plane we write RD,LD,BD, click Apply. In the same way we create point with coordinates RD,LD,BU, press Apply, then point with coordinates RD,LU,BU, again click Apply, and the last point on the bottom boundary RD,LU,BD, click Apply. Then we create points on the upper boundary: RU,LD,BD, Apply, RU,LD, BU, Apply, RU,LU, BU, Apply and RU,LU,BD OK. You should see eight points on the screen.

The same can be done by typing K, number of keypoint, radius, longitude, latitude in the command line:

K, , RD,LD, BD,

K, , RD,LD, BU,

K, , RD,LU, BU,

K, , RD,LU, BD,

K, , RU,LD, BD,
The easiest way to create the whole computational domain is to type $V, KP1,KP2,KP3,...$ in the command line:

$V,1,2,3,4,5,6,7,8$

(Com.: in this way we have automatically created lines and areas, so it is very important to type numbers of keypoints in the right logical order, i.e. with lines not crossed.)

6. Setting of the size of the elements and meshing of the domain

To mesh the computational domain we will use the MeshTool:

ANSYS Main Menu > Preprocessor > Meshing > MeshTool

In the Size Controls we click Lines Set. Then using the mouse we click all meridians (4 arcs), confirm the selection by clicking OK and in the Element Sizes on Picked Lines window we type in the second line NDIV No. of element divisions 8 (it corresponds to 5°), press Apply. In the same way we select all parallels and set number of divisions 9. In radial direction we set NDIV 10.

Then we mesh the whole domain using:

ANSYS Main Menu > Preprocessor > Meshing > MeshTool

We check whether Volumes is chosen in the Mesh, then in the Shape we tick Hex and click Mesh button. Then we click on the computational domain and press OK. We see volume being meshed.
7. Setting the boundary conditions

We start by applying the gravitational potential on the upper spherical boundary:

ANSYS Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Areas. We click on the upper spherical boundary, press OK, and in the VALUE Load TEMP value we type $GM/RU$, click OK. We can observe a little yellow arrow on the screen.

Now we will apply the gravitational acceleration on the bottom spherical boundary:

ANSYS Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Heat Flux > On Areas. We click the bottom spherical boundary, press OK, and in VALUE Load HFLUX value we type $GM/(RD*RD)$, press OK. Now we can observe a red mesh on the bottom boundary.

8. Running the solution

ANSYS Main Menu > Solution > Solve > Current LS > OK.

9. Visualizing results

ANSYS Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu. In the Contour Nodal Solution Data window, we highlight Nodal Solution > DOF Solution > Nodal Temperature and click OK.