/\*--------------------------------\*- C++ -\*----------------------------------\*\

| ========= | |

| \\ / F ield | OpenFOAM: The Open Source CFD Toolbox |

| \\ / O peration | Version: 2.3.0 |

| \\ / A nd | Web: www.OpenFOAM.org |

| \\/ M anipulation | |

\\*---------------------------------------------------------------------------\*/

FoamFile

{

version 2.0;

format ascii;

class volScalarField;

object T;

}

// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //

dimensions [0 0 0 1 0 0 0];

internalField uniform 300;

boundaryField

{

movingWall

{

type fixedValue;

value uniform 350;

}

fixedWalls

{

type fixedValue;

value uniform 300;

}

frontAndBack

{

type empty;

}

}

// \*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\* //