/\*--------------------------------\*- 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 dictionary;

location "system";

object fvSchemes;

}

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

ddtSchemes

{

default Euler;

}

gradSchemes

{

default Gauss linear;

grad(p) Gauss linear;

}

divSchemes

{

default none;

div(phi,U) Gauss linear;

div(phi,T) Gauss upwind; //注意：コンマと変数の間には空白なし

}

laplacianSchemes

{

default Gauss linear orthogonal;

}

interpolationSchemes

{

default linear;

}

snGradSchemes

{

default orthogonal;

}

fluxRequired

{

default no;

p ;

}

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