



11/1/2021

CATS User Guide

Manual for Users and Developers

Austin Ladshaw

OAK RIDGE NATIONAL LABORATORY

Contents

Mass Balances	8
Separated Phase Mass Balances	8
Combined Phase Mass Balances	9
Energy Balances	12
Separated Phase Energy Balances	12
Combined Phase Energy Balances	14
Navier-Stokes	16
Incompressible Flow - Built-in MOOSE Module	16
Compressible Flow - CATS Methods	17
Best Solver Options	18
Executioner	18
Time Schemes	18
Solver Type	18
PETSc Options	18
LU and ILU Methods	21
GASM and ASM Methods	21
GMRES (and GCR) Methods	23
Debugging Issues with Linear Solvers and Preconditioners	23
Line Search	24
Tolerances	24
Time Stepper	25
Preconditioning	25
Finite Difference Preconditioner	26
Single Matrix Preconditioner	26
Example Options	26
Example 1: GMRES with ASM preconditioner and BJacobi sub-preconditioner	27
Example 2: FMGRES with GMRES preconditioner and LU sub-preconditioner	27
Example 3: GMRES with ILU preconditioner and increased factor levels	28
Example 4: Setting up all options	29
Model Outputs	31
Output Options	31
Postprocessors	32

ElementVariableValue	32
SideAverageValue	32
ElementAverageValue.....	33
AreaPostprocessor	33
VolumePostprocessor	33
Exodus Files.....	34
Input Files.....	35
GlobalParams	35
Mesh	36
GeneratedMeshGenerator	36
FileMeshGenerator	37
Problem.....	37
Functions.....	37
Variables	38
AuxVariables	40
Kernels	40
DGKernels	40
AuxKernels	40
BCs.....	41
InterfaceKernels.....	41
Materials	41
Postprocessors.....	42
Preconditioning.....	42
Executioner	42
Outputs	42
Simulation Examples.....	44
Conservation of Mass	44
The Input File	45
Conservation of Energy.....	51
Coupling Mass and Energy.....	53
Adding Gas Properties	53
Coupling Mass and Energy with Simple Gas Properties	55
MOOSE CG Incompressible Navier-Stokes.....	56

INS Cartesian - Fully Bounded	56
INS Cartesian - Semi-Bounded	59
INS RZ	61
CATS DG Navier-Stokes	61
Variables.....	61
Kernels/DGKernels	62
Boundary Conditions.....	64
Other Information	66
Coupling Navier-Stokes, Mass, and Energy.....	67
Simulations with Numerous Reactions	68
Example 1: Multi-Reaction Adsorption	68
Example 2: Multi-Reaction Adsorption with Bulk Mass Balance Coupling	70
Simulations with Subdomains	71
Simulations with Hybrid FD/FE Microscale Diffusion	74
Packed-bed Adsorption.....	75
Packed-bed Adsorption (same problem, without Microscale)	77
Monolith CO Light-off	78
Monolith CO Light-off with True Wall Thickness	79
Doing Everything: Fully Coupled Mass-Energy with Microscale Physics	79
Running the Code.....	83
Unit Tests	83
Single Core Command Line	84
Multi-Core Command Line.....	84
To Do List.....	85
Specific Kernels	85
High Advection Stabilization	85
Adding Capabilities for Darcy Flow or Porous Flow	85
Finite Volumes CFD	85
Setting BCs based on ppm values	85
Kernels	86
ArrheniusEquilibriumReaction	86
ArrheniusEquilibriumReactionEnergyTransfer	87
ArrheniusReaction	88

ArrheniusReactionEnergyTransfer	89
ConstMassTransfer	90
ConstReaction	91
CoupledCoeffTimeDerivative	92
CoupledPorePhaseTransfer	93
CoupledSumFunction	94
DivergenceFreeCondition	95
EquilibriumReaction	95
ExtendedLangmuirFunction	96
ExtendedLangmuirModel	98
FilmMassTransfer	98
GAdvection (DGAdvection)	100
GAnisotropicDiffusion (DGAnisotropicDiffusion)	101
GConcentrationAdvection (DGConcentrationAdvection)	103
GNernstPlanckDiffusion (DGNernstPlanckDiffusion)	104
GNSMomentumAdvection (DGNMomentumAdvection)	106
GNSViscousVelocityDivergence (DGNViscousVelocityDivergence)	109
GPhaseThermalConductivity (DGPhaseThermalConductivity)	113
GPoreConcAdvection (DGPoreConcAdvection)	114
GThermalConductivity (DGThermalConductivity)	115
GVarPoreDiffusion (DGVarPoreDiffusion)	117
GVariableDiffusion (DGVariableDiffusion)	118
InhibitedArrheniusReaction	119
InhibitedArrheniusReactionEnergyTransfer	120
InhibitionProducts	121
LangmuirInhibition	122
MaterialBalance	124
MicroscaleCoefTimeDerivative	125
MicroscaleCoupledCoefTimeDerivative	127
MicroscaleCoupledVariableCoefTimeDerivative	128
MicroscaleDiffusion	129
MicroscaleDiffusionInnerBC	131
MicroscaleDiffusionOuterBC	132

MicroscaleScaledWeightedCoupledSumFunction	133
MicroscaleVariableCoefTimeDerivative.....	135
MicroscaleVariableDiffusion	136
MicroscaleVariableDiffusionInnerBC	137
MicroscaleVariableDiffusionOuterBC	138
PairedLangmuirInhibition	139
PhaseEnergyTransfer	140
PhaseTemperature.....	141
Reaction	142
ScaledWeightedCoupledSumFunction.....	143
TimeDerivative	144
VarSiteDensityExtLangModel.....	145
VariableCoefTimeDerivative	146
VariableCoupledCoefTimeDerivative.....	146
VectorCoupledGradient	147
VariableVectorCoupledGradient.....	149
WeightedCoupledSumFunction	150
Boundary Conditions.....	153
DGConcFluxLimitedStepwiseBC	153
DGConcFluxStepwiseBC	154
DGConcentrationFluxBC	155
DGConcentrationFluxLimitedBC	155
DGFlowEnergyFluxBC.....	156
DGFluxBC.....	157
DGFluxLimitedBC.....	158
DGFluxLimitedStepwiseBC	160
DGFluxStepwiseBC	161
DGNSMomentumOutflowBC	161
DGPoreConcFluxBC_ppm.....	162
DGPoreConcFluxBC	164
DGPoreConcFluxStepwiseBC.....	165
DGPoreDiffFluxLimitedBC	165
DGPoreDiffFluxLimitedStepwiseBC.....	167

DGVarVelDiffFluxLimitedBC	168
DGVarVelDiffFluxLimitedStepwiseBC	168
DGWallEnergyFluxBC	169
DirichletBC	170
INSNormalFlowBC	171
PenaltyDirichletBC	172
Interface Kernels	174
InterfaceConstReaction	175
InterfaceEnergyTransfer	176
InterfaceMassTransfer	177
Initial Conditions	179
InitialInhibitionProducts	179
InitialLangmuirInhibition.....	180
InitialPhaseEnergy.....	181
Auxiliary Kernels.....	182
AuxAvgLinearVelocity	182
AuxErgunPressure	183
GasDensity	184
GasEffectiveThermalConductivity.....	185
GasPropertiesBase	186
GasSolidHeatTransferCoef.....	187
GasSpecHeat	189
GasSpeciesAxialDispersion.....	189
GasSpeciesDiffusion	191
GasSpeciesEffectiveTransferCoef	192
GasSpeciesKnudsenDiffusionCorrection.....	193
GasSpeciesMassTransCoef.....	194
GasSpeciesPoreDiffusion	195
GasThermalConductivity.....	195
GasVelocityCylindricalReactor	196
GasViscosity	198
GasVolSpecHeat.....	199
LinearChangeInTime	200

MicroscaleIntegralAvg	200
MicroscaleIntegralTotal	201
MicroscalePoreVolumePerTotalVolume.....	202
MonolithAreaVolumeRatio	203
MonolithMicroscaleTotalThickness	204
MonolithHydraulicDiameter	205
SimpleGasPropertiesBase	206
SimpleGasMonolithMassTransCoef.....	208
SimpleGasMonolithHeatTransCoef.....	209
SimpleGasSphericalMassTransCoef.....	210
SimpleGasSphericalHeatTransCoef.....	212
SimpleGasCylinderWallHeatTransCoef	213
SimpleGasEffectivePoreDiffusivity.....	214
SimpleGasPoreDiffusivity.....	215
SimpleGasEffectiveKnudsenDiffusivity	216
SimpleGasKnudsenDiffusivity	217
SimpleGasDispersion	219
SimpleGasDensity	220
SimpleGasViscosity	221
SimpleGasIsobaricHeatCapacity	222
SimpleGasThermalConductivity.....	222
SolidsVolumeFraction	223
SphericalAreaVolumeRatio	224
VoidsVolumeFraction.....	225
Materials	227
INSFluid	227
Utilities	228
Egret.....	228
Error	228
Macaw.....	228

Mass Balances

In CATS, mass is balanced on a mass of a chemical species per total volume per time basis. In principle, if you do not need to couple the mass balance to an energy balance, then you can use any units of mass, volume, and time you want. However, once you couple with the energy balances, your time units must be in seconds and your volume units should be in cubic meters. This means your mass units can typically be in either moles or kilograms, whatever is most convenient. Just make sure the units work out for all the balances. For the rest of this discussion, we will assume the mass balances are to be done in mol/m³/s for consistency.

Separated Phase Mass Balances

If your domain does not contain a second phase (such as solid particles or air bubbles), or if there are two phases separated by boundaries and separated subdomains defined in your mesh, then the mass balances will not involve any porosity or void-volume correction terms.

For instance, consider the mass balance of a chemical species (C_i in mol/m³) in an open-channel of a monolith catalyst.

$$\frac{\partial C_i}{\partial t} + \nabla \cdot (\mathbf{v} \cdot C_i) = \nabla \cdot (D \cdot \nabla C_i) + \sum_{\forall j} r_j$$

In this formulation, we also represent a series of reactions (r_j) that involve C_i and occur in the bulk fluid phase. The sign of the specific reaction (r_j) will be negative for a loss of C_i and positive for a gain in C_i . Units of r_j are presumed here to be in mol/m³/s, same as C_i . The velocity vector (\mathbf{v} in m/s) and diffusion coefficient (D in m²/s) simulate the transport of the material through the channel.

Additionally, we can include the transfer of mass to another phase (i.e., walls of the channel) through an [Interface Kernel](#) (which behaves similar to a boundary condition). Here, $C_{w,i}$ (in mol/m³) represents the concentration of the i^{th} species in the pore spaces of the wall and k (in m/s) represents the mass transfer rate. Note that this kernel yields combined units of mol/m²/s, instead of mol/m³/s. This is because interface kernels (and [Boundary Conditions](#) in DG methods) are integrated over the area of the elements to produce a total mass flux.

$$\text{Mass Flux @ the channel walls: } k(C_i - C_{w,i})$$

The other boundary conditions for the mass balances in the open-channel are based on advective flux into and out of the simulation domain. The normal vector (\mathbf{n}) is known internally and is even used to determine whether the given boundary is the outlet or inlet boundary (see [DGFluxBC](#)). This is sufficient to describe the physics of mass balance in the open-channel.

$$\text{Inlet Boundary: } (\mathbf{v} \cdot \mathbf{n}) \cdot C_{inlet,i}$$

$$\text{Outlet Boundary: } (\mathbf{v} \cdot \mathbf{n}) \cdot C_i$$

For a mass balances in the separate subdomain for the channel walls, we can derive a mass balance in terms of the pore-concentration and pore-diffusion through the washcoat matrix.

$$\varepsilon_w \frac{\partial C_{w,i}}{\partial t} = \nabla \cdot (\varepsilon_w D_p \cdot \nabla C_{w,i}) + \varepsilon_w \sum_{\forall j} r_{p,j} + (1 - \varepsilon_w) \sum_{\forall k} r_{s,k}$$

In this formulation, ε_w represents the volume of washcoat-pores per total washcoat volume, $C_{w,i}$ is the pore-space concentration (which is specifically moles per volume of washcoat-pores in mol/m³), $r_{p,j}$ represent reactions that can take place in the pore-spaces (thus, is assumed to have same units as $C_{w,i}$ and need the ε_w conversion), and the $r_{s,k}$ represents surface/adsorption reactions in units of moles per volume of only the solids, i.e., mol/m³/s (which is why there is the (1- ε_w) unit conversion).

For the washcoat, there is no advective flux boundary condition, but there is the mass flux from the channel wall. This is implemented as an Interface Kernel and does not need to be repeated here or in the input file. There is no other boundary condition to consider for the washcoat since the natural boundary condition is the so-called No Flux boundary.

Lastly, for the immobile species (i.e., surface/adsorbed species) in the system, their own mass balances are based solely on the surface reactions taking place and a site/material balance to account for the losses in reaction site availability as the reactions progress.

$$\frac{dq_i}{dt} = \sum_{\forall k} r_{s,k}$$

$$q_{max} = \sum_{\forall j} n_j q_j + q_e$$

In this representation, the adsorbed concentration q_i (in mol/m³) is based on moles of surface species per volume of only solids, i.e., the same unit basis as $r_{s,k}$. The site balance is closed by invoking a kernel like [MaterialBalance](#) to ensure that the total number of occupied and unoccupied sites matches the theoretical total site density. In the above balance, the q_{max} represents the maximum sites (in mol/m³), n_j represents the number of sites occupied by q_j , and q_j is representative adsorbed species (in mol/m³) at those sites, and q_e represents empty sites (in mol/m³).

NOTE: The above is only an example of mass balances. Your problem may vary. For instance, $r_{s,k}$ may be represented in units of mol/m²/s for adsorption or catalytic reactions. In that case, there is an additional unit conversion term that goes in front (A_s) to represent the ratio of the total reactive surface area per unit of solids volume (in m⁻¹).

PAY ATTENTION TO YOUR UNITS AND DERIVE YOUR OWN BALANCES!

Combined Phase Mass Balances

In some situations, it may be more computationally efficient or more convenient to keep your simulation domain as one continuous mesh, rather than dividing into subdomains for different phases. For instance, if simulating adsorption or catalysis in a packed bed, it is nearly impossible to fully separate the solids from the gases in your grid because the packing of the particles would make the mesh far too

complex to construct and divided into all the subdomains for each individual particle. Thus, you would instead have a single simulation domain and “separate” the variables that are in solids or gases by using bulk unit conversion factors that represent a volume average of particle count or void space in each finite mesh element.

For starters, consider the transport of mass through a packed column. We will start by considering the mass balance on the gas species concentration. Just like in the [Separated Phase Mass Balances](#), we will represent the gas-phase concentration with C_i in units of moles per gas volume (mol/m^3). However, for the mass balance on this species, we need to now add a unit conversion factor (ε) that represents the average gas volume per total volume in the mesh.

$$\varepsilon \frac{\partial C_i}{\partial t} + \nabla \cdot (\varepsilon \mathbf{v} \cdot C_i) = \nabla \cdot (\varepsilon D \cdot \nabla C_i) + \varepsilon \sum_{\forall j} r_j - (1 - \varepsilon) \cdot k \cdot A_o (C_i - C_{w,i})$$

Just like before, we have our velocity vector (\mathbf{v} in m/s), a diffusion parameter (D in m^2/s), and reactions that can occur in the gas-phase (r_j) with same units as C_i (moles per volume of gas). The concentration term $C_{w,i}$ represents the concentration in the pore-spaces of the solid particles.

NOTE: Often times you will see this type of porous flow use a superficial gas velocity (\mathbf{v}_s). Here, the velocity vector \mathbf{v} is representative of the average velocity of fluid elements in the domain. By definition, the product of average velocity and porosity (ε) is the superficial velocity: $\varepsilon \mathbf{v} = \mathbf{v}_s$.

The new feature in the mass balance is the addition of the mass transfer into the pore-spaces of the solids. Previously, this was introduced as an integrated mass transfer boundary, however, here it must be included in the mass balance since there is no interface boundary on the domain. In addition, since it is not an integrated mass flux term, the regular mass transfer also includes 2 unit conversions: (i) A_o , which represents the outer shell surface area of particles per hard shell volume of the particles in the domain and (ii) $(1 - \varepsilon)$, which represents the volume of solids per total volume. The culmination of these conversions ensure that each term in the balance has units of moles of gas species per total volume per time.

The boundary conditions for the gas-phase concentrations are essentially the same as before, but we replace the actual velocity with the superficial velocity.

$$\text{Inlet Boundary:} \quad (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot C_{inlet,i}$$

$$\text{Outlet Boundary:} \quad (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot C_i$$

Although there is no longer a specified subdomain for the pore-space concentration ($C_{w,i}$), we still need to provide a mass balance for this species. The major difference here is that we would not include diffusion through the pore-space because there is not defined grid or mesh subdomain for it to diffuse through.

$$\varepsilon_w \frac{\partial C_{w,i}}{\partial t} = -k \cdot A_o (C_{w,i} - C_i) + \varepsilon_w \sum_{\forall j} r_{p,j} + (1 - \varepsilon_w) \sum_{\forall k} r_{s,k}$$

In this formulation, the mass balance is done in units of moles of gas species per total volume of solids per time. Unit conversion factor ε_w is the solid pore volume per bulk solid volume. To convert further to moles per total volume, we would multiple all terms by $(1-\varepsilon)$, i.e., bulk solids volume per total volume, thus we can forgo that step since it would factor out of each. The $r_{s,k}$ factor has units of moles per volume of solids (not bulk volume of solids), which is why it gets a factor of $(1-\varepsilon_w)$.

The immobile species in this example (q_i) is formulated exactly the same as in the [Separated Phase Mass Balances](#).

$$\frac{dq_i}{dt} = \sum_{\forall k} r_{s,k}$$

$$q_{max} = \sum_{\forall j} n_j q_j + q_e$$

In this representation, the adsorbed concentration q_i (in mol/m^3) is based on moles of surface species per volume of only solids, i.e., the same unit basis as $r_{s,k}$. The site balance is closed by invoking a kernel like [MaterialBalance](#) to ensure that the total number of occupied and unoccupied sites matches the theoretical total site density. In the above balance, the q_{max} represents the maximum sites (in mol/m^3), n_j represents the number of sites occupied by q_j , and q_j is representative adsorbed species (in mol/m^3) at those sites, and q_e represents empty sites (in mol/m^3).

NOTE: Microscale diffusion simulations for [Combined Phase Mass Balances](#) are being developed using a hybridization of finite differences for the microscale regime and finite elements for the macroscale mesh. See [MicroscaleDiffusion](#) for computation details. Still under construction!

AGAIN: NOTE: The above is only an example of mass balances. Your problem may vary. For instance, $r_{s,k}$ may be represented in units of $\text{mol/m}^2/\text{s}$ for adsorption or catalytic reactions. In that case, there is an additional unit conversion term that goes in front (A_s) to represent the ratio of the total reactive surface area per unit of solids volume (in m^{-1}).

AGAIN: PAY ATTENTION TO YOUR UNITS AND DERIVE YOUR OWN BALANCES!

Energy Balances

In CATS, energy balances are done in terms of energy per unit volume per time, specific with each term having the units of J/m³/s (or W/m³). The variables being balanced are those of internal energy density (E) and NOT the temperature of a given phase (T). This is the most conservative form of the energy balance as it will not only capture how temperature variations impact system energy, but also how density and specific heat variations impact that energy density as well. In the below sections, we outline and detail the strong forms of energy balances for [Separated Phase Energy Balances](#) and [Combined Phase Energy Balances](#).

NOTE: If you use the new [SimpleGasProperties](#) system, then you are **NO LONGER** forced to use this unit basis. Users can use any length, time, energy, and mass units (within the limits outlined in [SimpleGasProperties](#)). An example of this can be found [here](#).

Separated Phase Energy Balances

By separated phase, we mean that these are energy balances that are on a clearly divided subdomain of a mesh that represents a gas, liquid, or solid phase energy balance. These energy balances would not apply to a domain that does not have a clear subdivision between solids and fluids (such as a packed bed reactor), but would apply to a monolith catalyst wherein the empty channels of the monolith are 1 subdomain and the washcoat is another, separated subdomain.

For instance, consider an energy balance for the fluid phase in the monolith channels.

$$\frac{\partial E_f}{\partial t} + \nabla \cdot (\mathbf{v} \cdot E_f) = \nabla \cdot (K_f \cdot \nabla T_f) + \sum_{\forall j} (-\Delta H_j) r_j$$

In this model equation, E_f (J/m³) is the energy density variable for the fluid phase, \mathbf{v} (m/s) is the velocity vector for the fluid elements, K_f (W/m/K) is the fluid thermal conductivity, T_f (K) is the temperature of the fluid phase, r_j (mol/m³/s) are a series of chemical reactions that may occur in the fluid phase, and ΔH_j (J/mol) are the enthalpies of those reactions. Note that if you need to couple the energy balance with a mass balance, your concentrations in each respective phase should have units of mol/m³. Otherwise, you will need additional unit conversions to include the heats of reaction.

The energy balance equation for the fluid phase must be coupled to the temperature variable of the fluid phase (and the concentrations/reactions in the fluid phase if they are present). The temperature of the fluid phase must be included in your problem as its own equation and residual (see [PhaseTemperature](#)). That temperature will be a function of the energy density, fluid density (ρ_f in kg/m³), and fluid heat capacity (c_{pf} in J/kg/K) as follows:

$$E_f = \rho_f c_{pf} T_f$$

The boundary conditions for the energy balance of the fluid phase are similar to those of the mass balances (see [Separated Phase Mass Balances](#)). Since the fluid phase is in contact with walls of the channel, energy can be transferred from one phase to another (i.e., from wall to fluid or from washcoat to fluid depending on your specific solution domain). That energy transfer will have a very similar form

to the mass transfer at the washcoat walls. Here, h is the heat transfer coefficient ($\text{W}/\text{m}^2/\text{K}$) and T_w (in K) is the temperature of the washcoat subdomain (or the reactor walls depending on implementation).

$$\text{Energy Transfer @ washcoat/walls: } h(T_f - T_w)$$

The above expression would be implemented as an [InterfaceKernel](#) (in the case of the washcoat) or as a [BoundaryCondition](#) (in the case of the system/reactor walls). Recall that here that this residual yields combined units of $\text{J}/\text{m}^2/\text{s}$, instead of $\text{J}/\text{m}^3/\text{s}$. This is because interface kernels (and [Boundary Conditions](#) in DG methods) are integrated over the area of the elements to produce a total energy flux.

For the other boundary conditions, we need to consider the flux of energy carried into and out of our domain by the advective fluid velocities. Most commonly, we would know what the fluid temperature, density, and specific heat are at the inlet boundary, but not necessarily the outlet boundary. There for, the mathematical representation for each will be different. However, they are both implemented in the same boundary condition kernel ([DGFlowEnergyFluxBC](#)) in CATS. This is because we would implicitly know whether or not to consider the boundary an exit or entrance based solely on the inner product between the velocity vector and the normal vector to the mesh boundary surface.

$$\text{Inlet Boundary: } (\mathbf{v} \cdot \mathbf{n}) \cdot \rho_f c_{pf} T_{f,inlet}$$

$$\text{Outlet Boundary: } (\mathbf{v} \cdot \mathbf{n}) \cdot E_f$$

In this monolith channel energy balance example, we also must consider the energy balance in the washcoat areas of the monolith. For simplification, we will assume the gases in the pore spaces of the washcoat are the same temperature as the solids of the washcoat, thus we need to only consider the bulk energy density of the washcoat (E_w in J/m^3) as our variable.

$$\frac{\partial E_w}{\partial t} = \nabla \cdot (K_w \cdot \nabla T_w) + \varepsilon_w \sum_{\forall j} (-\Delta H_j) r_{p,j} + (1 - \varepsilon_w) \sum_{\forall k} (-\Delta H_k) r_{s,k}$$

In this formulation, K_w (in $\text{W}/\text{m}/\text{K}$) is either the solids thermal conductivity or an effective washcoat conductivity (see [GasEffectiveThermalConductivity](#)), $r_{p,j}$ (in $\text{mol}/\text{m}^3/\text{s}$) are the gas reactions taking place in the pore-spaces with enthalpy of ΔH_j (J/mol), $r_{s,k}$ (in $\text{mol}/\text{m}^3/\text{s}$) are the surface/solids reactions taking place with enthalpy of ΔH_k (J/mol), and ε_w is the porosity of the washcoat. Note that each term here has units of energy per washcoat volume per time. The ε_w factors are used because the concentrations in the pore-space are units of mass per pore-volume and/or mass per only solid-volume. If your reaction or concentration terms have different units, then this form of the energy balance will be slightly different.

To recover the bulk temperature of the washcoat (T_w in K), you again use the [PhaseTemperature](#) kernel, but with bulk washcoat density (ρ_w in kg/m^3) and bulk washcoat specific heat (c_{pw} in $\text{J}/\text{kg}/\text{K}$).

$$E_w = \rho_w c_{pw} T_w$$

NOTE: The above is only an example of energy balances. Your problem may vary. For instance, $r_{s,k}$ may be represented in units of $\text{mol}/\text{m}^2/\text{s}$ for adsorption or catalytic reactions. In that case, there is an

additional unit conversion term that goes in front (A_s) to represent the ratio of the total reactive surface area per unit of solids volume (in m^{-1}).

PAY ATTENTION TO YOUR UNITS AND DERIVE YOUR OWN BALANCES!

Combined Phase Energy Balances

Following along with the derivation from [Combined Phase Mass Balances](#), there are situations in which your domain has two distinct phases (i.e., solids and fluids), but your mesh may be incapable of clearly and efficiently subdividing those phases into separated subdomains. In this case, we need to derive energy balances for the fluids and solids on an averaged basis using parameters that represent the average volume of solids or fluids within each element of the domain.

For starters, consider the transport of energy through a packed column. We will start by considering the energy balance on the fluid phase. Just like in the [Separated Phase Energy Balances](#), we will represent the fluid energy density with E_f in units of energy per fluid volume (J/m^3). However, for the energy balance on this variable, we need to now add a unit conversion factor (ε) that represents the average gas volume per total volume in the mesh.

$$\varepsilon \frac{\partial E_f}{\partial t} + \nabla \cdot (\varepsilon \mathbf{v} \cdot E_f) = \nabla \cdot (\varepsilon K_f \cdot \nabla T_f) + \varepsilon \sum_{\forall j} (-\Delta H_j) r_j - (1 - \varepsilon) \cdot h \cdot A_o (T_f - T_s)$$

Just like before, we have our velocity vector (\mathbf{v} in m/s), a fluid conductivity parameter (K_f in $\text{W}/\text{m}/\text{K}$), and reactions that can occur in the fluid (r_j) with their respective enthalpies ΔH_j in J/mol . The new coupled variable is for the temperature of the solid particles (T_s in K).

NOTE: Often times you will see this type of porous flow use a superficial gas velocity (\mathbf{v}_s). Here, the velocity vector \mathbf{v} is representative of the average velocity of fluid elements in the domain. By definition, the product of average velocity and porosity (ε) is the superficial velocity: $\varepsilon \mathbf{v} = \mathbf{v}_s$.

The new feature in the energy balance is the addition of the energy transfer with the solid particles. Previously, this was introduced as an integrated energy transfer boundary, however, here it must be included in the energy balance since there is no interface boundary on the domain. In addition, since it is not an integrated energy flux term, the regular energy transfer also includes 2 unit conversions: (i) A_o , which represents the outer shell surface area of particles per hard shell volume of the particles in the domain and (ii) $(1 - \varepsilon)$, which represents the volume of solids per total volume. The culmination of these conversions ensure that each term in the balance has units of energy of the fluid per total volume per time.

The temperature of the fluid phase can be recovered using the [PhaseTemperature](#) kernel, with a slightly different equation, which includes the ε factor for.

$$E_f = \rho_f c_{pf} T_f$$

The boundary conditions for the fluid energy flux at the open boundaries are essentially the same as before, but we replace the actual velocity with the superficial velocity.

$$\text{Inlet Boundary:} \quad (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot \rho_f c_{pf} T_{f,inlet}$$

$$\text{Outlet Boundary:} \quad (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot E_f$$

In addition to the energy flow boundary conditions, you will also have energy flux boundary conditions at the walls of the domain. We represent that boundary as a [DGWallEnergyFluxBC](#) where T_w is the temperature of the wall (in K), h_w is the heat transfer coefficient (in W/m²/K), and ε accounts for the fraction of the fluid phase that is actually in contact with the wall surface area.

$$\text{Wall Boundary:} \quad h_w \cdot \varepsilon \cdot (T_f - T_w)$$

For the energy balance in the solid particles, we include the energy transfer from the fluid phase along with the solids thermal conductivity or *effective* solids conductivity (K_s in W/m/K). This conductivity term is included such that we can simulate the impact of neighboring particles exchanging energy with each other. We also include heat of reactions for reactions in the pore-spaces ($r_{p,j}$ in mol/m³/s) and reactions on the solids/surfaces ($r_{s,k}$ in mol/m³/s), along with the respective enthalpies for each. The ε_s parameter is the porosity of the solid particles themselves. Note that each term in this energy balance has units of energy density per unit volume of bulk solids per time.

$$\frac{\partial E_s}{\partial t} = \nabla \cdot (K_s \cdot \nabla T_s) - h \cdot A_o (T_s - T_f) + \varepsilon_s \sum_{\forall j} (-\Delta H_j) r_{p,j} + (1 - \varepsilon_s) \sum_{\forall k} (-\Delta H_k) r_{s,k}$$

There is not energy exchange boundary condition needed at the open ends of the channel, since the solid particles are assumed stationary in the domain and we already account for energy exchange with the fluid phase volumetrically. However, we do need to include a boundary condition for energy exchange at the walls of the reactor system since the particles are in direct contact with the walls due to the tight packing of the particles. Just like with the fluid phase wall boundary, the solid phase wall boundary will involve a contact fraction ($1 - \varepsilon$) to represent the fraction of the area at the wall that the solids are in contact with.

$$\text{Wall Boundary:} \quad h_w \cdot (1 - \varepsilon) \cdot (T_s - T_w)$$

To recover the bulk temperature of the solids (T_s in K), you again use the [PhaseTemperature](#) kernel, but with bulk particle density (ρ_s in kg/m³) and bulk particle specific heat (c_{ps} in J/kg/K).

$$E_s = \rho_s c_{ps} T_s$$

NOTE: The above is only an example of energy balances. Your problem may vary. For instance, $r_{s,k}$ may be represented in units of mol/m²/s for adsorption or catalytic reactions. In that case, there is an additional unit conversion term that goes in front (A_s) to represent the ratio of the total reactive surface area per unit of solids volume (in m⁻¹).

PAY ATTENTION TO YOUR UNITS AND DERIVE YOUR OWN BALANCES!

Navier-Stokes

The CATS module does contain some kernels and methods ([DGNSMomentumAdvection](#), [DGNSMomentumOutflowBC](#), and [VectorCoupledGradient](#)) to create a DG implementation of Navier-Stokes equations. However, it can also be linked and/or coupled with the Navier-Stokes module in MOOSE built-in with the MOOSE framework (mooseframework.org/modules/navier_stokes/). Provided below are some brief details and instructions for invoking the pre-built Navier-Stokes solvers. Example input files to run simulations are provided under the [Examples](#) section. An example specifically showing the utilization of our own custom DG implementation of incompressible and compressible Navier-Stokes can be seen [here](#).

Incompressible Flow - Built-in MOOSE Module

For systems in which the density variations are very small (< %10) and/or systems involving Mach numbers well below 0.3 (i.e., velocities less than 100 m/s), it is generally acceptable to assume that the fluid phase is incompressible. This assumption greatly simplifies and stabilizes the simulation of the conservation of momentum. The incompressible, convective form of the Navier-Stokes equations are represented by the following 2 equations:

$$\text{Momentum:} \quad \rho_f \frac{\partial \mathbf{v}}{\partial t} + \rho_f (\mathbf{v} \cdot \nabla) \mathbf{v} = -\nabla \bar{p} + \mu_f \nabla^2 \mathbf{v} + \rho_f \mathbf{g} + \mathbf{f}$$

$$\text{Continuity:} \quad (\nabla \cdot \mathbf{v}) = 0$$

In the above equations, ρ_f is the density of the fluid (in kg/m³), μ_f is the viscosity of the fluid (in kg/m/s), \mathbf{v} is the velocity field vector (in m/s), \mathbf{g} is a gravitational constant (in m/s²), \mathbf{f} is a body force vector function (in N/m³), and \bar{p} is the dynamic pressure (in Pa) caused by the fluid motion. Note that since this representation only involves dynamic pressure, recovering the total pressure in a domain would require some other kernels or auxiliary system (see [AuxErgunPressure](#) for a pressure calculation in packed columns or monolith structures.)

Solving the incompressible Navier-Stokes equations requires both the momentum and continuity equations, as well as a set of variables for each velocity component (x, y, and z) and the dynamic pressure. Dynamic pressure is resolved through the continuity equation. However, the continuity equation itself is insufficient for determining the total pressure variations in the domain. It is only sufficient for resolving the gradients of dynamic pressure needed in the Navier-Stokes incompressible momentum equation.

In addition to the above equations for the interior of the domain, you must provide boundary conditions for the velocity components. For the incompressible Navier-Stokes module in MOOSE, you will need to provide some form of inlet/outlet flow condition for open boundaries and a “no slip” condition for closed boundaries or walls. For details on boundary condition options, see [PenaltyDirichletBC](#), [DirichletBC](#), and/or [INSNormalFlowBC](#). In general, you will use [PenaltyDirichletBC](#) or [DirichletBC](#) to force 0 velocity for all velocity components at walls (i.e., the “no slip” condition) and you will use [INSNormalFlowBC](#), [PenaltyDirichletBC](#), or [DirichletBC](#) to specify a specific velocity flux or velocity profile at the inlet or outlet boundaries.

Compressible Flow - CATS Methods

Also in CATS is an implementation of both compressible and incompressible Navier-Stokes flow using DG methods. The CATS Navier-Stokes kernels were developed in such that they would be applicable to both incompressible and compressible simulation conditions. Thus, it is generally advised to always assume your simulation involves a compressible fluid. In this way, your fluid flow field will adjust to changes in fluid density during the simulation.

The formulation for compressible Navier-Stokes flow is as follows:

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \frac{1}{3} \mu \nabla (\nabla \cdot \mathbf{u})$$
$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

In most situation, you can assume that the fluid mass exists at a pseudo-steady-state. In such a case, the time derivative of density becomes zero and the equations simplify to:

$$\rho \frac{\partial(\mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \frac{1}{3} \mu \nabla (\nabla \cdot \mathbf{u})$$
$$\nabla \cdot (\rho \mathbf{u}) = 0$$

In CATS, we implement these equations in a piecewise manner, such that each component of velocity is its own variable and has its own kernels. This would then transform our conservation of momentum equation into the following:

$$\rho \frac{\partial u_i}{\partial t} + \nabla \cdot (\rho \mathbf{u} \cdot u_i) = -\mathbf{n}_i \cdot \nabla p + \mu \nabla^2 u_i + \frac{1}{3} \mu \nabla (\mathbf{D}_{ix} \cdot \nabla u_x + \mathbf{D}_{iy} \cdot \nabla u_y + \mathbf{D}_{iz} \cdot \nabla u_z)$$

Where i represents the i -th velocity component (x, y, or z). As such, \mathbf{n}_i is a unit vector in the i -th direction, and \mathbf{D}_{ix} , \mathbf{D}_{iy} , and \mathbf{D}_{iz} are tensor matrices whose entries are all 0, except for entry at the i -th row and x (or y or z) column, which has a 1.

The continuity equation is used to resolve the pressure gradients in the flow field. To solve this equation, you must enforce a pressure value at the exits/outlets of the domain. The standard is to specify that the exits/outlets of the domain have 0 pressure. In this way, the pressure calculated inside the domain represents only the dynamic pressure (and not absolute pressure).

The BCs for the velocity variables should be setup to give a 'No Slip' condition at the walls of the domain and an outflow of momentum flux at each exit/outlet of the domain. Additional external forces, such as gravity, can be added to this problem as needed through other kernels acting on each component of velocity. An example of the usage of the CATS Navier-Stokes kernels can be found [here](#).

Best Solver Options

MOOSE allows the users to specify a plethora of solver options for (i) the time integration scheme, (ii) the non-linear solver type, (iii) the linear solver type, (iv) the type of preconditioner, (v) the solver tolerances, etc. Given these abundant options it can be overwhelming to determine what works best or what solvers should be invoked. This section talks generally about which solvers and options have had the most efficient and accurate results in the testing of CATS.

Executioner

The 'Executioner' block of the input file is where you would specify most of your solver options except for your preconditioner. The most import options you will want to customize and control are provided below.

Time Schemes

The 'scheme' option is where you direct MOOSE to use a specific time integration scheme for the simulation. MOOSE supports numerous integration schemes for both implicit and explicit methods. However, for CATS it is highly recommended that you only use implicit methods for stability and efficiency. While implicit methods are more complex than explicit methods, with proper preconditioning they can be more computationally efficient by allowing for larger time steps to be taken resulting in fewer overall steps needed to complete a simulation.

Most commonly, CATS uses the standard 'implicit-euler' method since it is the most efficient and most stable scheme. However, if you desire higher levels of computational accuracy, you can specify either the 'bdf2' or 'crank-nicolson' methods. Of those two schemes, 'bdf2' is usually the best, especially for any simulations utilizing the Navier-Stokes module or simulations with advectively dominant physics.

Solver Type

In MOOSE, the 'solve_type' option is used to denote the non-linear solver method the user wants to deploy. Again, there are many different options available, but for CATS it is recommended you use 1 of 2 possible methods: (i) 'newton' or (ii) 'pjfnk'. If you are running a somewhat small simulation on a single core or personal computer, often times the 'newton' method will be faster. However, for large scale simulations or simulations on a computer cluster, it is highly recommended you use 'pjfnk'. If you use the 'newton' method you will not need to provide any preconditioner to the solver, but your solve will not scale well as you request more computer cores or resources. Using 'pjfnk' will allow the simulation to scale better on multiple cores, but will REQUIRE a preconditioner to be specified in order for the solve to be efficient (see [Preconditioning](#) for details).

PETSc Options

The linear solvers in MOOSE are directed and handled by the PETSc library. Thus, you will need to specify a set of options for the linear solvers to be used in conjunction with your simulation. These options are specified in 3 tags: (i) 'petsc_options', (ii) 'petsc_options_iname', and (iii) 'petsc_options_value'. Each tag accepts a list of arguments. The number of arguments in 'petsc_options_iname' must be the same number of arguments in 'petsc_options_value' because the

‘iname’ represents the name of the PETSc argument and the corresponding ‘value’ represents what you are setting that argument to.

Generally, it is a good idea to always set ‘petsc_options’ to ‘-snes_converged_reason’ as this will print out to the console window the reason why a particular time step converged or failed to converge. You can use this information to try and determine whether or not other solver options need to be changed or modified to improve convergence. For instance, if the program reports that a time step did not converge due to DIVERGED_LINESEARCH or something similar, then you may need to change line searching options. There are other arguments you can place here as well, which will be discussed more in later sections.

The other tags (‘petsc_options_iname’ and ‘petsc_options_value’) are used to specify the linear solver you want to use, the preconditioning algorithm to apply to your preconditioning matrix, and some other special options specific to your linear solver. Due to the complexity and breadth of these options, we cannot go over everything in this document. Instead, we will just outline some of the more popular options you can invoke.

petsc_options_iname → -ksp_type : petsc_options_value → gmres / fgmres / gcr / cgs / bcgs

This is the name of the option for the Krylov subspace method to be used as the iterative linear solver. Many linear solvers may work for this option, but we have outlined 4 options that seem to give the best result: (i) gmres, (ii) fgmres, (iii) gcr, (iv) cgs, and (v) bcgs. The methods ‘gmres’, ‘fgmres’, and ‘gcr’ are the most stable solvers, but are computationally more expensive. Note that ‘gmres’ is actually the default solver if no option is specified. They should be used if you notice that the linear iterations struggle to converge, as they are almost guaranteed to converge (with proper preconditioning). Both cgs and bcgs are much more efficient solvers, but do not guarantee convergence, even with preconditioning.

NOTE: If you are planning to apply ‘ksp’ as the ‘-pc_type’ (see below), then you **NEED** to use ‘fgmres’ as the ‘-ksp_type’ here. Otherwise, convergence may not occur.

petsc_options_iname → -pc_type : petsc_options_value → asm / gasm / bjacobi

The ‘-pc_type’ tag is used to specify the name of the primary preconditioning operator to use with the given linear solver. From the range of options tested in CATS, it was found that (i) asm, (ii) gasm, and (iii) bjacobi provide the best efficiencies and residual reduction. Both ‘asm’ and ‘gasm’ use the additive Schwarz method. The difference between ‘asm’ and ‘gasm’ is whether or not a single matrix block can be shared across multiple processors (as in ‘gasm’) or whether a processor is assigned a block or blocks in whole (as in ‘asm’). The ‘bjacobi’ method applies a Jacobi preconditioning operation to each block the problem is divided into. As such, ‘bjacobi’ is much more parallelizable than either ‘asm’ or ‘gasm’ and may scale better with many CPUs and/or GPUs. However, ‘bjacobi’ is generally a worse preconditioner and should only be used if you need the simulation to be massively parallel or when wanting to utilize GPU acceleration.

In addition to providing a standard preconditioning type, users can also give 'ksp' for the argument '-pc_type'. When doing so, you are specifying that you want to precondition with another 'ksp' method. The outer 'ksp' method is dictated by '-ksp_type' (see above), then the inner 'ksp' method is set by a new argument '-ksp_ksp_type'. This argument can be any of the '-ksp_type' arguments from before (see below for options).

If '-pc_type' == 'ksp', then...

petsc_options_iname → -ksp_ksp_type: petsc_options_value → gmres / fgmr / gcr / cgs / bcgs

If no '-ksp_ksp_type' is given, then it will default to 'gmres'. Users will also need to provide an argument for '-ksp_pc_type' for the terminal preconditioner instead of '-sub_pc_type' as shown below. An example of this can be found in this [example](#).

NOTE: If you are planning to apply 'redundant' as the '-ksp_pc_type' (see below), then you **NEED** to use 'fgmr' as the '-ksp_ksp_type' here. Otherwise, convergence may not occur.

(NOTE: Currently, CATS does not support GPU acceleration, but that may change.)

petsc_options_iname → -sub_pc_type : petsc_options_value → lu / ilu / bjacobi / asm

The 'sub_pc_type' tag is used to specify the terminal preconditioning operation to be applied to individual blocks of the matrix when divided into subdomain regions by the primary preconditioner. The best option in terms of convergence is the 'lu' solver, which essentially applies a direct solve to the sub-block of the matrix. For greater scalability and parallelization, you can use either 'ilu', 'bjacobi', or 'asm' for the terminal preconditioner. If you are running on a personal computer, just use 'lu' for best performance.

(NOTE: If your simulation involves the MOOSE Navier-Stokes module, you **MUST** use 'lu' for this option)

If the '-pc_type' was set to 'ksp', then this argument does nothing. You must instead set an argument for '-ksp_pc_type' as shown below.

petsc_options_iname → -ksp_pc_type : petsc_options_value → lu / ilu / bjacobi / asm
gasm / pbjacobi / redundant

Generally, 'lu' gives the best convergence behavior, but does not scale well. For better scalability, it is recommended to use 'ilu', 'asm', 'gasm', 'bjacobi', 'pbjacobi', or 'redundant'. The 'redundant' method will run an additional 'ksp' method with 'gmres' as the terminal preconditioner and applies 'lu' on a series of sub-blocks divided among a number of MPI processes. This might be overkill for most problems, but is a method that scales relatively well and has good convergence.

More information on PETSc solvers and options can be found at <https://www.mcs.anl.gov/petsc/>.

LU and ILU Methods

In the above section, we discussed some of the most common PETSc arguments you can use with a focus on the manner in which specific types of solvers are invoked. However, each of those solver methods have their own options to play with as well. For instance, the LU and ILU preconditioners have numerous options available to help stabilize the solve. If you begin to see error message such as 'SUBPC_ERROR', then maybe you need to change these options.

On thing you should try to always have active, is a 'NONZERO' shift type for LU and ILU methods. You can specify this option as follows:

petsc options iname → **-pc factor shift type** : **petsc options value** → **NONZERO**

petsc options iname → **-sub pc factor shift type** : **petsc options value** → **NONZERO**

petsc options iname → **-ksp pc factor shift type** : **petsc options value** → **NONZERO**

NOTE: You MUST supply this argument corresponding to the correct usage of LU/ILU in your PETSc options.

For instance, if you set LU/ILU as the '-sub_pc_type', then you must set the '-sub_pc_factor_shift_type' as the 'NONZERO' method. Pay close attention to the prefixes on these options

Specifically for ILU, you can also set the factorization levels to augment the amount of 'fill' in the incomplete factorization procedure. By default, the 'fill' level is ILU(1), which allows for an amount of matrix sparsity equal to A^2 , where A is the original matrix. Making this value higher will increase the amount of 'fill' and make the factorization more complete (e.g., ILU(2) corresponds to a sparsity of A^3). You can set this value with the following arguments:

petsc options iname → **-pc factor levels** : **petsc options value** → **(# > 1)**

petsc options iname → **-sub pc factor levels** : **petsc options value** → **(# > 1)**

petsc options iname → **-ksp pc factor levels** : **petsc options value** → **(# > 1)**

NOTE: You MUST supply this argument corresponding to the correct usage of ILU in your PETSc options.

For instance, if you set ILU as the '-sub_pc_type', then you must set the '-sub_pc_factor_levels' to the specified value you desire. Pay close attention to the prefixes on these options

GASM and ASM Methods

The difference between 'gasm' and 'asm' is that 'gasm' can share blocks/subdomains among a set of processors/cores, where as in 'asm' each processor/core has its own set of blocks/subdomains to

work on and does not share that information with any other processor. As a result, 'asm' is more memory efficient than 'gasm', but 'gasm' is often a *better* preconditioner.

Each of these solvers can change the number of blocks/subdomains that each processor works on, as well as an amount of 'overlap' allowed between those blocks/subdomains. In general, the more overlap you have, the more memory is needed, but better convergence can be achieved. Thus, with each of these methods you are constantly balancing convergence vs memory efficiency.

In general, I have found that the default number of blocks/subdomains per processor (which defaults to 1) is usually sufficient, and improvements can be made by increasing the levels of overlap between 10 to 100. However, this may be very problem specific. Below shows how to set these options.

For ASM:

petsc options iname → -pc asm blocks : petsc options value → (# > 1)

petsc options iname → -sub pc asm blocks : petsc options value → (# > 1)

petsc options iname → -ksp pc asm blocks : petsc options value → (# > 1)

petsc options iname → -pc asm overlap : petsc options value → (# 10 - 100)

petsc options iname → -sub pc asm overlap: petsc options value → (# > 10 - 100)

petsc options iname → -ksp pc asm overlap: petsc options value → (# > 10 - 100)

NOTE: You MUST supply this argument corresponding to the correct usage of ASM in your PETSc options. Pay close attention to the prefixes in each argument.

For instance, if you set ASM as the '-sub_pc_type', then you must set the '-sub_pc_asm_overlap' (etc) to the specified value you desire.

For GASM:

petsc options iname → -pc gasm subdomains : petsc options value → (# > 1)

petsc options iname → -sub pc gasm subdomains : petsc options value → (# > 1)

petsc options iname → -ksp pc gasm subdomains : petsc options value → (# > 1)

petsc options iname → -pc gasm overlap : petsc options value → (# 10 - 100)

petsc options iname → -sub pc gasm overlap: petsc options value → (# > 10 - 100)

petsc options iname → -ksp pc gasm overlap: petsc options value → (# > 10 - 100)

NOTE: You MUST supply this argument corresponding to the correct usage of GASM in your PETSc options. Pay close attention to the prefixes in each argument.

For instance, if you set GASM as the ‘-sub_pc_type’, then you must set the ‘-sub_pc_gasm_overlap’ (etc) to the specified value you desire.

GMRES (and GCR) Methods

Both ‘gmres’ (and ‘fgmres’) and ‘gcr’ are “restarted” subspace methods. These methods build a growing subspace of search vectors and hold each vector in memory to build the next step. Because of this, in order for them to be practical the vector space has to be periodically flushed to free up memory. By default, PETSc will only hold 30 vectors in the subspace before flushing and restarting. For very large problems with many mesh elements, 30 vectors may not be enough to hold on to. To change this options you must also have either ‘-ksp_gmres_restart’ or ‘-ksp_gcr_restart’ as a ‘petsc_options_iname’ tag to be changed. Ideally, you would choose a large number that represents a significant fraction of your ‘degrees of freedom’ in the problem, which is related to the number of elements in the mesh and the number of non-linear variables being solved for. If the supplied number is too large, then it may tax your computer memory. You invoke these arguments as shown below.

petsc_options_iname → -ksp_gmres_restart : **petsc_options_value → (large #)**

petsc_options_iname → -ksp_gcr_restart : **petsc_options_value → (large #)**

NOTE: Whether you are using ‘fgmres’ or ‘gmres’, the restart argument is always by the same name, so the options shown above is valid for either method.

In addition to setting the restart value, you may also need to specify an orthogonalization procedure for ‘fgmres’ or ‘gmres’. By default, both methods use a ‘classical Gram-Schmidt’ orthogonalization procedure, which is a fast algorithm, but can be unstable. You can change this to the ‘modified Gram-Schmidt’ method by adding the following argument to ‘petsc_options’.

petsc_options → -ksp_gmres_modifiedgramschmidt

Debugging Issues with Linear Solvers and Preconditioners

Whenever you are having solver issues, you can view what the solvers are doing, how they are invoked, and the options they were invoked with by adding a couple of additional arguments to the ‘petsc_options’ list. By default, I always recommend leaving the ‘-snes_converged_reason’ option active. This gives minimal reporting and will simply tell you why the linear (and non-linear) solvers have stopped. This is done by reporting the PETSc flags, which will report why it converged or why it didn’t converge.

For a more advanced view of the linear solver actions, you can also add ‘-ksp_view’ option to the ‘petsc_options’ argument. This will report a significant amount of information to the console including what linear solvers, preconditions, and sub-preconditioners are being called, where they are called in the solve, and the options they were called with.

<u><code>petsc_options → -ksp_view</code></u>	:	All linear solver information
<u><code>petsc_options → -snes_converged_reason</code></u>	:	Reason the solver stopped

Line Search

Line searching is a method that applies to the non-linear iterations in order to smooth out the residuals at each iteration. By default, the ‘line_search’ option is set to ‘none’ (i.e., to not apply line searching). Other options include: (i) ‘bt’, (ii) ‘l2’, and (iii) ‘basic’. Generally, adding line search to the simulation will very slightly increase the computation time at each non-linear step. Thus, it is recommended to not use line searching if it is not needed. However, some very difficult simulations with poor initial conditions or rapidly changing non-linear variables may struggle to converge without it. In these situations, the ‘bt’ line search method has been shown to be very effective and is the most stable line search method.

NOTE: Line searching will not always help a simulation converge and can occasionally have the opposite effect if the initial condition or prior time step is no where near the current solution. Sometimes a better option is to just take smaller time steps with no line searching.

Tolerances

In MOOSE, you have the ability to change and adjust all the tolerances you want. In some cases, you may need to adjust the solution tolerances from their default values, especially if you are simulating a transient problem to the steady-state solution. The following list of tolerances and iteration limits have been found to work well.

NOTE: tolerances can also be directly setup in the PETSc options (see [examples](#) below).

nl_rel_tol → from 1e-6 to 1e-8

nl_abs_tol → from 1e-6 to 1e-8

nl_rel_step_tol → from 1e-10 to 1e-16

nl_abs_step_tol → from 1e-10 to 1e-16

l_tol → from 1e-4 to 1e-6

nl_max_its → from 10 to 30

l_max_its → from 100 to 500 (really depends on the number of elements in the mesh)

Time Stepper

In addition to specifying a time scheme, the user needs to direct MOOSE how to choose its next time step, when to start and stop the simulation, and specify a cap to the time step size. The start time, end time, and maximum step size are given under the Executioner block, while the time step method is given in a TimeStepper subblock. In the TimeStepper subblock, the user specifies the ‘type’ of stepper algorithm to use, as well as the initial time step to take. Below is an example of how the times and time stepper are specified in the Executioner block.

```
start_time = 0
end_time = 10
dtmax = 0.5

[./TimeStepper]
    type = ConstantDT
    dt = 0.1
[../]
```

The TimeStepper ‘type’ will usually be either ‘ConstantDT’ to specify a constant time step to take at each solve or ‘SolutionTimeAdaptiveDT’ which will adjust the time step size up or down after each solve based on how well or how quickly the previous solve went. In general, the method for ‘SolutionTimeAdaptiveDT’ should be used with a small initial time step and will increase the time step size to a maximum to accelerate the simulation to complete in a more timely manner. If you are so inclined, you can also create custom Time Steppers to use in CATS.

Preconditioning

In order for the linear iterations to converge efficiently, the user is REQUIRED to provide some form of preconditioner. Otherwise, convergence is extraordinarily slow or sometimes impossible. However, if the ‘solve_type’ was set to ‘newton’, then there is no need for a preconditioner. For any large scale simulations on multiple processors or computer clusters, you should be using ‘pjfnk’ as your ‘solve_type’ and MUST provide a preconditioner.

Each residual contribution in CATS is fully developed with appropriate Jacobian elements and Off-Diagonal Jacobian elements to make preconditioning very simple. This relieves the user from having to employ some of the more complex preconditioning options, such as the physics-based MOOSE preconditioner, which are extremely tedious to setup in the simulations. Instead, you will only ever need to use the [Single Matrix Preconditioner](#) method. This will be by far the most efficient method to use in CATS. However, we will also discuss the [Finite Difference Preconditioner](#), which may be useful for debugging issues with developed residuals or solvers.

Finite Difference Preconditioner

This preconditioning option is highly inefficient, but is useful for occasionally tracking down problems that may persist in the developed modules and kernels. Essentially, the Finite Difference Preconditioner is a matrix preconditioner constructed from a full finite differences sweep of all residuals invoked in a simulation. Thus, it creates a “perfect” preconditioner that should theoretically solve the linear subproblem in a single iteration. If you use this option and see that the linear steps do NOT converge within 1 to 2 steps, then there is likely a problem in the residuals. To invoke the Finite Difference Preconditioner, you add the following blocks to your input file:

```
[Preconditioning]
  [./FDP_method]
    type = FDP
    full = true
  [../]
[]
```

Single Matrix Preconditioner

This is the preconditioner option that every simulation case in CATS should have for any realistic problem you want to simulate. It should be extremely efficient due to the numerous Jacobian and Off-Diagonal Jacobian functions that have been manually coded into each and every customized CATS kernels. When used correctly, it should create a nearly perfect, sparse preconditioning matrix. You invoke the use of this option by including the following blocks in your input file.

```
[Preconditioning]
  [./SMP_method]
    type = SMP
    full = true
  [../]
[]
```

Example Options

This section provides a sampling of how these solver options are structured in the CATS input files. Again, this section and example is not exhaustive, but demonstrates how to invoke some of the most common solver options that have resulted in the most accurate and efficient simulations.

For problems that are ‘difficult’ to converge, it is generally recommended to use ‘LU’ or ‘ILU’ as your terminal preconditioner at some point in the solve. This is because ‘LU’ and ‘ILU’ are currently the best preconditioning methods.

For problems that are VERY ‘difficult’, I recommend using ‘fgmres’ with a ‘ksp’ preconditioner that uses either ‘fgmres’ or ‘gmres’ followed by a sub-preconditioner of ‘lu’ or ‘ilu’. When using both ‘lu’ and ‘ilu’, be sure to always set the shift to ‘NONZERO’. For ‘ilu’, you may need to increase the ‘factor levels’ to make a good preconditioner.

Example 1: GMRES with ASM preconditioner and BJacobi sub-preconditioner

[Executioner]

```
type = Transient
scheme = bdf2
solve_type = pjfnk
petsc_options = '-snes_converged_reason'
petsc_options_iname = '-ksp_type -ksp_gmres_restart -pc_type -sub_pc_type'
petsc_options_value = 'gmres      100          asm      bjacobi'
line_search = none
nl_rel_tol = 1e-8
nl_abs_tol = 1e-6
nl_rel_step_tol = 1e-10
nl_abs_step_tol = 1e-10
l_tol = 1e-6
nl_max_its = 10
l_max_its = 100
start_time = 0
end_time = 10
dtmax = 0.5
[./TimeStepper]
    type = SolutionTimeAdaptiveDT
    dt = 0.1
[../]
```

[]

Example 2: FMGRES with GMRES preconditioner and LU sub-preconditioner

[Executioner]

```
type = Transient
scheme = implicit_euler
solve_type = pjfnk
petsc_options = '-snes_converged_reason'
                '-ksp_gmres_modifiedgramschmidt'

petsc_options_iname = '-ksp_type
                    -ksp_gmres_restart
                    -pc_type
                    -ksp_ksp_type
                    -ksp_pc_type
                    -ksp_pc_factor_shift_type'

petsc_options_value = 'fgmres
                    100
                    ksp
                    gmres'
```

```

                                lu
                                NONZERO'
line_search = none
nl_rel_tol = 1e-8
nl_abs_tol = 1e-6
nl_rel_step_tol = 1e-10
nl_abs_step_tol = 1e-10
l_tol = 1e-6
nl_max_its = 10
l_max_its = 100
start_time = 0
end_time = 10
dtmax = 0.5
[./TimeStepper]
    type = ConstantDT
    dt = 0.5
[./]
[]

```

Example 3: GMRES with ILU preconditioner and increased factor levels

[Executioner]

```

type = Steady
solve_type = pjfnk
petsc_options = '-snes_converged_reason
                -ksp_gmres_modifiedgramschmidt'

petsc_options_iname = '-ksp_type
                    -ksp_gmres_restart
                    -pc_type
                    -pc_factor_shift_type
                    -pc_factor_levels'

petsc_options_value = 'gmres
                    100
                    ilu
                    NONZERO
                    3'

line_search = none
nl_rel_tol = 1e-8
nl_abs_tol = 1e-6
nl_rel_step_tol = 1e-10
nl_abs_step_tol = 1e-10
l_tol = 1e-6
nl_max_its = 10

```

```
l_max_its = 100
[]
```

Example 4: Setting up all options

When options are passed to PETSc from your input file, PETSc will only use options that it needs. As a result, it may be to your benefit to set ALL available options in your input files. This way, you can simply toggle a few values at the input to change the solvers, rather than have to write a completely new set of options.

Additionally, you can directly set some of the tolerances, maximum iterations, or other options directly in PETSc. Those options are also demonstrated below. When these options are set in the PETSc options, they will override any options given below them in the MOOSE Executioner block.

NOTE: Not a FULLY exhaustive list, but the most comprehensive you will likely need.

[Executioner]

```
type = Steady
solve_type = pjfnk
petsc_options = '-snes_converged_reason
                 -ksp_gmres_modifiedgramschmidt
                 -ksp_view'

petsc_options_iname = '-ksp_type
                      -ksp_gmres_restart   -ksp_gcr_restart

                      -pc_type             -sub_pc_type
                      -ksp_ksp_type        -ksp_pc_type

                      -pc_factor_shift_type -pc_factor_levels
                      -sub_pc_factor_shift_type -sub_pc_factor_levels
                      -ksp_pc_factor_shift_type -ksp_pc_factor_levels

                      -pc_asm_blocks       -pc_asm_overlap
                      -sub_pc_asm_blocks   -sub_pc_asm_overlap
                      -ksp_pc_asm_blocks   -ksp_pc_asm_overlap

                      -pc_gasm_subdomains   -pc_gasm_overlap
                      -sub_pc_gasm_subdomains -sub_pc_gasm_overlap
                      -ksp_pc_gasm_subdomains -ksp_pc_gasm_overlap

                      -snes_max_it -snes_atol -snes_rtol -snes_stol
                      -ksp_max_it  -ksp_atol  -ksp_rtol'
```

```

petsc_options_value = 'fgmres
                        100  100

                        ksp    lu
                        gmres  lu

                        NONZERO 3
                        NONZERO 3
                        NONZERO 3

                        3  50
                        3  50
                        3  50

                        3  50
                        3  50
                        3  50

                        20  1e-8  1e-10 1e-10
                        100 1e-6  1e-6'

line_search = none
[]

```

Model Outputs

The standard output for the MOOSE model includes (i) CSV files and (ii) Exodus files, however, no outputs are setup by default. You must turn on output options from the 'Outputs' block in the input file. This section discusses some of the basic output, how to request specific types of outputs, and how to interpret results of the output.

Output Options

For CATS, there are 4 primary output flags you should know about: (i) 'exodus', (ii) 'csv', (iii) 'interval', and (iv) 'print_linear_residuals'. Each option is as follows:

exodus : **true / false**

The 'exodus' tag is used to determine whether or not you wish to have an Exodus file produced during the simulation. An Exodus file is a binary file that contains all information about the simulation domain and the variables on that domain. It is the primary file you will want to use for visualizing the results from CATS. However, it may be unnecessary if your domain is 0D or 1D. For more information on Exodus files, see [Exodus Files](#) below.

csv : **true / false**

The 'csv' tag is used to determine whether or not you wish to have a .csv file created during the simulation of the values of the computed [Postprocessors](#). You can import or open .csv files directly into MS Excel (or another spreadsheet software) to create plots of specific data or variables as a function of simulation time. The [Postprocessors](#) are MOOSE calculation objects that are used to estimate variable values at specific locations in the mesh, integrals of variable values over the domain, or average values of variables at a boundary or interface in the domain. For more information on [Postprocessors](#), see the next section below.

interval : **(some #)**

The 'interval' tag is used to tell MOOSE how often you want output from the model to be created or displayed. By default, this value is set to 1, which means that at the end of every time step, solution values are recorded to the console window and/or to the various output files. For very long simulations with many time steps, it may be helpful to set this to a larger number so that some output gets suppressed and the output files don't become too large. For instance, if you set this value to 10, then MOOSE will only record the output of every 10th time step.

print_linear_residuals : **true / false**

The 'print_linear_residuals' tag is used to determine whether or not you want MOOSE to report the residuals at each linear iteration. It is usually not necessary to set this to 'true', but may be useful for

debugging or testing the code. When you see that the linear residuals are not reducing very quickly, that can be an indication that the preconditioning or the chosen linear solver is inappropriate.

Postprocessors

Postprocessors are a set of built-in calculation features in MOOSE that allow the user to: (i) probe the values of variables in specific elements, (ii) obtain average values for variables in the domain or at a boundary, and (iii) obtain integral values for variables in the domain or at a boundary. Below are some of the most common Postprocessors and how they are invoked inside the 'Postprocessors' block of the input file.

ElementVariableValue

This postprocessor is used to probe the value of a variable at a specific element in the domain. We generally do not use this postprocessor simply because it is relatively difficult to use as you have to provide the actual element id you want to probe and not just a location in special dimensions. You can invoke this option as shown below.

```
# Probing the value of variable 'u' at element '1'
[/u_at_1]
  type = ElementVariableValue
  variable = u
  elementid = 1
  execute_on = 'initial timestep_end'
[../]
```

SideAverageValue

This postprocessor is used to compute the average value of a variable at a boundary or interface of the domain. We commonly use this postprocessor to observe changes in outlet conditions of a domain for a particular variable. For example, probing the average value of the value of a concentration variable at the 'outlet' of the domain gives us the "breakthrough" data from the simulation. Note that you must provide the name of the boundary for which this postprocessor applies, as well as the name of the variable to probe.

```
# Probing the value of variable 'C' a boundary named 'exit'
[/C_exit]
  type = SideAverageValue
  variable = C
  boundary = 'exit'
  execute_on = 'initial timestep_end'
[../]
```

ElementAverageValue

This postprocessor is used to formulate an average value of a variable in the entire domain or subdomain. If your mesh involves subdomains, then you are required to provide the name of the subdomain for which this average will apply. Each subdomain will either be named by the user or automatically given a number by MOOSE depending on how the mesh was constructed. This postprocessor is commonly used to provide estimates for average values of non-mobile variables, like adsorption capacities, in a domain.

```
# Computing the average value of adsorption variable 'q' in subdomain 'washcoat'
[/q_avg]
  type = ElementAverageValue
  variable = q
  block = 'washcoat'
  execute_on = 'initial timestep_end'
[/./]
```

AreaPostprocessor

This postprocessor is used to compute the actual surface area (on a 3D mesh), or length (on a 2D mesh) of a particular boundary or interface. In most cases, you will not need to use this postprocessor, but it is occasionally useful if you are checking mass/energy balances and need an accurate account of the size of the inlet/outlet of the domain. No variable needs to be given, just the name of the boundary.

```
# Calculating the area of the 'exit' of the domain
[/area_exit]
  type = AreaPostprocessor
  boundary = 'exit'
  execute_on = 'initial timestep_end'
[/./]
```

VolumePostprocessor

This postprocessor is used to compute the actual volume (on a 3D mesh), or area (on a 2D mesh), of a particular subdomain or the entire domain. Much like the AreaPostprocessor, this is not necessarily needed, but is occasionally useful for checking mass/energy balances when you need an accurate account of the size of each subdomain.

```
# Calculating the volume of the 'washcoat' of subdomain
[/vol_wash]
  type = VolumePostprocessor
  block = 'washcoat'
  execute_on = 'initial timestep_end'
[/./]
```

Exodus Files

The [Postprocessors](#) can only yield specific information at a given location or some average values, but all of the simulate results are available in the Exodus file. Since the Exodus file is a binary file, you will need a specific application to read the data. Cubit is a common application that can read Exodus files, but it is only available for Government work. You can also use an open source software such as ParaView to read Exodus files.

More information on Cubit can be found at <https://cubit.sandia.gov/>.

More information on ParaView can be found at <https://www.paraview.org/>.

Input Files

In this section, we generically discuss the structure of MOOSE/CATS input files. Each input file is structured in a manner reminiscent of html or xml. It is a “hierarchical input text format” where each block of the input starts with the block name in square brackets (e.g., [Variables]) and that block ends when an empty set of square brackets is encountered (e.g., []). Inside each block there may be subblocks. All subblocks are started by [./subblock_name] and ended by [../]. Inside the blocks and subblocks are where all parametric, variable, function, and other arguments are placed. Any line in the input file preceded by a ‘#’ is a comment and will not be read in by MOOSE. The example below just demonstrates this structure. The arguments with “ marks can denote string arguments or a list of arguments, depending on the context.

```
[Block_1]
  param = 13
  bool_arg = true
  # Comment about Subblock_1
  [./Subblock_1]
    another_arg = 'something'
    list_of_args = '1 2 3'
  [../]
[]
```

Provided below in the next few sections are examples and discussion on the many types of blocks and subblocks commonly used in CATS. Note that this list is not exhaustive for the MOOSE framework. For more information on input files, visit <https://mooseframework.org/>.

GlobalParams

The GlobalParams block is a place where you can put any arguments you want to be used by all other blocks, unless specifically overridden within that particular block. It is very useful for reducing some redundancy in the input file structure. Often times, multiple kernels will need the same arguments to be declared each time they are invoked. Thus, if you put those arguments into GlobalParams, then those arguments will remain consistent throughout all the kernels that use them.

NOTE: When MOOSE searches for input arguments, it will ALWAYS check the local kernel blocks FIRST for the needed arguments. If they are not found locally, then it checks GlobalParams.

A great example of when and why to use GlobalParams comes from how we invoke the Navier-Stokes module in MOOSE. Each Navier-Stokes kernel requires a specific set of stabilization and other arguments to be shared among each invocation of the kernels. Therefore, instead of repeating this information in each Navier-Stokes specific kernel block, we can put them all into GlobalParams as follows.

```
[GlobalParams]
  gravity = '0 0 0'
  integrate_p_by_parts = true
```

```

    supg = true
    pspg = true
    alpha = 0.5
    laplace = true
    convective_term = true
    transient_term = true
  []

```

For more detailed information on these arguments, see [Incompressible Navier-Stokes](#) example.

Mesh

The Mesh block is where you direct MOOSE to either: (i) construct a mesh domain and subdomains to simulate on or (ii) read in a mesh file that represents your simulation domain. There are many different ways in which MOOSE can generate and use meshes, so we will only discuss some of the basics here. For more information, go to <https://www.mooseframework.org/syntax/index.html>.

GeneratedMeshGenerator

This subblock allows the user to very quickly and easily create lines, rectangles, and cubic domains to simulate on. In concert with the RZ-Cylindrical option in the [Problem](#) block, we can also create cylindrical domains with this subblock. The users provide the minimum and maximum x, y, and z points for the domain and the number of elements to divide each dimension into. By default, the domain 'block' ID number is set to 0 and the boundaries of the mesh get set to 'left right' for the x-direction ('left' → minimum x and 'right' → maximum x), 'bottom top' for the y-direction ('bottom' → minimum y and 'top' → maximum y), and 'front back' for the z-direction ('front' → z minimum and 'back' → z maximum).

In the example below, we use the GeneratedMeshGenerator to create a 1-by-5 sized 2D mesh in the xy plane with 10 subdivisions in each the x and y directions.

```

[Mesh]
  [./my_mesh]
    type = GeneratedMeshGenerator
    dim = 2
    xmin = 0
    xmax = 1
    ymin = 0
    ymax = 5
    nx = 10
    ny = 10
  [../]
[]

```

FileMeshGenerator

For more complex mesh structures, you can direct MOOSE to simply read in a mesh file. MOOSE supports numerous mesh file types. For a complete list of supported mesh formats, check out <https://www.mooseframework.org/source/meshgenerators/FileMeshGenerator.html>. In CATS, we most commonly use the '.unv' and '.msh' type meshes. One major advantage of these mesh formats is they allow for the naming of blocks, boundaries, and interfaces. Thus, this allows us to create complex meshes with various subdomains and unusual boundaries that other kernels can identify by name.

In the example below, we use FileMeshGenerator to read in a 2D mesh representing a monolith channel. That mesh has 2 blocks named 'washcoat' and 'channel', as well as having boundaries identified as 'inlet' and 'outlet' (i.e., the open flow boundaries) and 'interface' the interface boundary between the 'washcoat' and 'channel' subdomains. Note that here we do not need to declare any other arguments to set these names. The names come from the mesh file and can be referenced later in the input file.

```
[Mesh]
  [./2D_monolith]
    type = FileMeshGenerator
    file = 2DChannel.msh
  [./]
[]
```

Problem

In CATS, there is only a single purpose to the Problem block that is applicable which is to direct a coordinate system transformation from XYZ-Cartesian to RZ-Cylindrical. We use the RZ-Cylindrical domain when we want to simulate a 3D column in a 2D domain. It provides a convenient way to simulate gas flow through a cylindrical shape without needing to create and mesh an actual column. If your problem is not RZ-Cylindrical, then you do not need to include this block in your input file.

When we want a 2D cylindrical domain, we only need to add the Problem block and specify the coordinate system as 'RZ' as shown below. Note that in this transformation of the coordinate system, the x-direction becomes the radial coordinate R and the y-direction becomes the axial coordinate Z. Thus, if this block were included in concert with the [GeneratedMeshGenerator](#) object from before, we would create a 2 diameter cylinder (e.g., x length = 1 → radius of 1) with a 5 unit length.

```
[Problem]
  coord_type = RZ
[]
```

Functions

The Functions block on its own doesn't actually add anything to the simulation, but can be used in coordination with other blocks to setup custom initial conditions or boundary conditions. There are many different built-in functions in MOOSE (<https://www.mooseframework.org/syntax/index.html>), but

the most commonly used function is the `ParsedFunction` object. It is very easy to use this object to create custom initial conditions for variables if we want that variable to initially have some spatial variation in our domain.

The `ParsedFunction` object reads in a user argument that represents a mathematical expression using parameters for 'x', 'y', 'z', and 't' to update the value of the function based on the specific location in the domain and/or a given point in time. The operators that are recognized by the MOOSE parser include, but are not limited to: (i) '+', (ii) '-', (iii) '*', (iv) '/', and (v) 'exp(..)'. More options can be found at <https://www.mooseframework.org/source/functions/MooseParsedFunction.html>.

Provided below is an example of a parsed function we could use to set a custom initial condition for a variable in our domain as a function of the z-location in the domain. Use " marks around the function to encapsulate the whole expression.

```
[Functions]
  [./q_ic]
    type = ParsedFunction
    value = '4.01E-4 * -1.8779 * exp( z / 10) / (1 - exp(10))'
  [../]
[]
```

Variables

The Variables block is where you put down all the non-linear variables that you plan to use the CATS physics/chemistry kernels to solve. Each variable name is given by the user and is interpreted as the name given as the subblock in the input file for that variable. Within each variable subblock, the user must specify: (i) the 'order' of the variable, (ii) the 'family' of the variable, and (iii) an initial condition, which can be done as a constant value, a user function, or some other custom kernel. Additionally, if your mesh involves subdomains, then you need to provide the 'block' name or ID that the variable exists on. Providing nothing will assign the variable to all blocks.

The 'order' of the variable can be either 'CONSTANT', 'FIRST', or 'SECOND'. In general, you should make all variables 'FIRST' order, however, if simulations of concentration or energy are oscillatory, you can lower the order to 'CONSTANT' to maximize simulation stability. The 'family' of the variable must be 'MONOMIAL' for mass balance variables and energy balance variables, but must be 'LAGRANGE' for Navier-Stokes variables. Temperature can also be a 'LAGRANGE' family.

To set a constant initial condition, you just add the 'initial_condition' argument to the subblock and give a value to set the variable to. For any other initial condition, you must add another subblock named 'InitialCondition' and any arguments that that particular custom initial condition argument may need.

In the below examples, we create variables for concentration (C), adsorption (q), energy (E), and velocity in the x-direction (vel_x). Each variable is on block 0. We show how to set the variable on the block, but this line is unnecessary if there is only 1 block. Only the velocity needs to be made a 'LAGRANGE' variable, while the others are 'MONOMIAL'. Variables C and vel_x get set to a constant initial condition, while q and E get set to a function and custom CATS initial condition kernel

([InitialPhaseEnergy](#)), respectively. The function for the initial condition of q comes from the [Functions](#) block (see above). The purpose is just to provide a demonstration of the various options for setting up non-linear variables in CATS.

```
[Variables]
    [./C]
        order = FIRST
        family = MONOMIAL
        initial_condition = 1e-9
        block = 0
    [../]

    [./q]
        order = FIRST
        family = MONOMIAL
        block = 0
        [./InitialCondition]
            type = FunctionIC
            #below sets the name of the function object in Functions block
            function = q_ic
        [../]
    [../]

    [./E]
        order = FIRST
        family = MONOMIAL
        block = 0
        [./InitialCondition]
            type = InitialPhaseEnergy
            specific_heat = 1000
            density = 1.2
            temperature = 298
        [../]
    [../]

    [./vel_x]
        order = FIRST
        family = LAGRANGE
        initial_condition = 0
        block = 0
    [../]

[]
```


AuxVariables

The AuxVariables block is formulated identically to that of the [Variables](#) block. Each subblock is the name of an auxiliary variable, each auxiliary variable must have an 'order' and a 'family', and each can have constant initial conditions or custom/function initial conditions. The only difference is that an auxiliary variable does not have any associated kernels, however, they can have an associated [AuxKernel](#). Often times, CATS will use auxiliary variable even without an AuxKernel just so that parametric constants can be referenced to by name in other blocks, rather than repeating the values in those blocks. See [Variables](#) for examples on how to structure this block.

Kernels

The Kernels block is where different pieces of physics and chemistry are invoked. This will likely be the largest block in the input file since every variable needs multiple kernels to describe all the physics. Each kernel will have its own subblock and each subblock must at least give (i) the 'type' of kernel, (ii) the 'variable' the kernel acts on, and (iii) the 'block' or subdomain name/ID that the kernel applies to. Note that the 'block' argument is only needed if there are multiple subdomains in the mesh. Examples of how to invoke each kernel in CATS is provided in significant detail throughout the main [Kernels](#) section of this guide.

DGKernels

One of the major features in CATS is in how the transport equations for mass and energy are handled. For these physics, we use Discontinuous Galerkin (DG) methods, which are significantly more stable and more accurate from problems involving conservation laws compared to the standard Galerkin methods. In CATS, we have developed a host of custom advective and diffusive transport kernels to take advantage of this methodology. Whenever your physics involve these types of transport mechanism, you must use the corresponding [DGKernels](#). In addition, each DG kernel must also be accompanied by a corresponding standard Galerkin kernel in the MOOSE system to complete the residual formation. More discussion of this can be found [here](#) and examples of how to invoke the DG kernels is provided in all the [Simulation Examples](#).

AuxKernels

The AuxKernels block are for providing a calculation for a specific [AuxVariable](#). Each auxiliary variable is only allowed to have one auxiliary kernel. In CATS, the auxiliary kernels are most commonly used to set specific gas properties, rate parameters, and/or transport parameters. However, you can use the auxiliary system for a wide range of calculations. It should be noted though that you should not use auxiliary kernels for variables such as concentration/mass or energy densities. These should be treated as non-linear variables and coupled fully and implicitly. The auxiliary kernels and variables are only loosely coupled with the non-linear variables. Usage examples and all CATS auxiliary kernels can be found in more detail in the main [AuxKernels](#) section.

BCs

The BCs block is used to set boundary conditions for any variable that needs them. By default, if no boundary condition is provided, then MOOSE applies a so-called “natural” boundary condition, which essentially would enforce a zero slope at a boundary. This is sufficient for any “closed” boundaries on the domain or for any variable that represents an immobile phase, such as an adsorption variable. For mobile variables, you will need to provide some specific boundary conditions.

For the conservation of mass and energy problems, you cannot actually provide any true Dirichlet type boundary conditions because solutions are not defined at nodes, they are defined at elements. Thus, in CATS, there are a host of integrated boundary conditions you can apply for phenomena such as inflow/outflow and wall energy transfer, as well as an emulated or penalty based pseudo-Dirichlet boundary condition. For the Navier-Stokes module, you can use either the [DirichletBC](#), [PenaltyDirichletBC](#), or [INSNormalFlowBC](#) options. More information on all boundary conditions can be found in the main [Boundary Conditions](#) section.

InterfaceKernels

The InterfaceKernels are essentially boundary conditions on mesh elements that represent the interface between 2 subdomains in the global mesh. You will only ever need to invoke these kernels when your mesh contains more than 1 ‘block’ ID. We use these kernels to facilitate the transfer of mass and energy from one domain to another. When invoked, you must provide the name of the variable on the primary mesh/subdomain, as well as the ‘neighbor’ variable that the other subdomain. Examples of this can be found in the main [Interface Kernels](#) section.

Materials

The Materials block is a section of the input file where certain properties can be set for the various subdomains of a mesh. When you declare any materials in your input file, all blocks must invoke a material, regardless of whether or not those properties are needed on each subdomain. An individual material may be invoked on multiple blocks if necessary. Often times this is the simplest way to resolve any MOOSE errors stemming from missing materials.

In CATS, we don’t really use the Materials system for handling parameters. Typically, all of our parameters may be set as constants or may be specified as [AuxVariables](#). The reasoning for this distinction in our framework boils down to code modularity and ease-of-use. When a particular property or parameter is declared in MOOSE as a “Material Property”, the kernels that expect that property will only function correctly if the specific material object, which calculates or declares that property by name, is invoked. On the other hand, if that property is declared a variable, then MOOSE will accept any argument in the kernel that is a variable, an auxiliary variable, or a specified constant. Thus, making the same bit of code significantly more modular. In addition, since the CATS kernels are designed specifically to anticipate parameters as variables, all the necessary off-diagonal Jacobian elements that the MOOSE system would need to resolve the coupling have already been pre-programmed into those kernels. If we instead used the Materials system, then we would have to update the entire kernel base if we wanted to include those material properties as actual variables at a later point in time.

Although CATS does not make use of the Materials system, the built-in MOOSE Navier-Stokes module does. Therefore, we must discuss here how a user wanting to invoke Navier-Stokes in their simulation can do so in CATS. We have created a custom Material object called [INSFluid](#), which provides a simple way for CATS properties to be linked to the incompressible Navier-Stokes module while still taking advantage of the CATS method of coupling variables and parameters. Thus, in any simulation where you use the incompressible Navier-Stokes module, you should create a Materials block in your input file as shown below. Note that the expected arguments for ‘density’ and ‘viscosity’ are allowed to be the names of CATS variables (or auxiliary variables) or may be given constant values. See [INSFluid](#) for more details on usage and unit conventions.

```
[Materials]
  [./ins_mat]
    type = INSFluid
    block = 0
    density = 1.225
    viscosity = 1.81E-5
  [../]
[]
```

Postprocessors

The Postprocessors block of the input file is used for calculating specific integrated variable values, boundary values, and/or probing variables in the mesh at specific elements. See the main [Postprocessors](#) section above for details on commonly used types of postprocessors. To invoke any of the above calculations, simply add the subblocks detailed in the subsections of [Postprocessors](#) into a ‘[Postprocessors]’ block in your input file.

Preconditioning

Unless you are using the ‘Newton’ solver in the [Executioner](#) block, you will need to provide some form of preconditioning matrix. The Preconditioning block of the input file tells MOOSE how to construct a preconditioning matrix. For more details, see the main [Preconditioning](#) section above.

Executioner

The Executioner block tells MOOSE how to solve each time step, advance each time step, the linear and non-linear solvers to use, and the tolerances of those solvers. Without an Executioner block, your simulation will not run. The [Best Solver Options](#) section above talks at length about what all needs to be in the Executioner block and provides a [simple example](#).

Outputs

The last block of the input file is the Outputs block, which directs how often MOOSE should produce an output and what that output should be. Realistically, there are only 4 output options that

you will invoke in CATS. Those options are detailed in the main [Outputs](#) section above. Below provides a brief example of how those options are invoked in the input file.

```
[Outputs]
  print_linear_residuals = true
  exodus = true
  csv = true
  interval = 10
[]
```

Simulation Examples

This section will provide simple demonstrations of how to use CATS to solve some a variety of problems. You can use the examples provided here to help you learn your way around the modeling framework and test out developing your own simulation cases. All simulations provided below are for demonstration purposes only. Your parameters, domains, variables, units, and objects may vary.

Conservation of Mass

The most common type of problem you may seek to solve involves a conservation of mass, wherein you track the fluid phase concentration evolution in a domain over time, as well as the mass transfer of the fluid phase to a washcoat or particle pore-space, then track adsorption or other reactions as well. For the purpose of this first demonstration to familiarize you with the input file syntax and basics of invoking kernels, let's first consider a simple, generic material balance as follows.

Consider the adsorption of a gas species in a column packed with solid particles. This model requires: (i) a mass balance on the gas species in the bulk spaces, (ii) a mass balance on the gas species in the pore-spaces of the particle, (iii) an adsorption reaction inside the particle, and (iv) a mass balance on the available surface sites. Thus, it will involve 4 primary equations each to solve for a different non-linear variable: (i) C for concentration in bulk, (ii) C_p for concentration in pore-spaces, (iii) q for adsorption, and (iv) S for surface sites. We also need to think about the units for each variable, as that will dictate how the equations should be formed. Most commonly: (i) C will have units of moles per cubic meter of gas, (ii) C_p will have moles per cubic meter of gas, (iii) q will have units of moles per kg of particle, and (iv) S will have units of moles per kg of particle. Thus, our equations might be as follows:

$$C: \quad \varepsilon \frac{\partial C}{\partial t} + \nabla \cdot (\varepsilon \mathbf{v} \cdot C) = \nabla \cdot (\varepsilon D \cdot \nabla C) - (1 - \varepsilon) \cdot k \cdot A_o (C - C_p)$$

$$C_p: \quad \varepsilon_p \frac{\partial C_p}{\partial t} = -k \cdot A_o (C_p - C) - \rho_p \frac{dq}{dt}$$

$$q: \quad \frac{dq}{dt} = k_f C_p S - k_r q$$

$$S: \quad S = S_{max} - q$$

In this representation, ε is the bulk porosity, \mathbf{v} is the linear velocity, D is the dispersion coefficient, k is the mass transfer rate, A_o is the ratio of particle outer surface area to particle volume (for spherical particles it would be $3/r$ where r is particle radius), ε_p is the porosity of the particles, ρ_p is the particle bulk density, k_f and k_r are forward and reverse reaction rates, and S_{max} is the maximum available adsorption sites.

To solve this problem, there are actually many different combinations of kernels that you can use to invoke each piece of physics for each equation. It largely depends on what is most convenient for your usage, but here we will using the following kernels:

[VariableCoefTimeDerivative](#): $\varepsilon \frac{\partial C}{\partial t}$ and $\varepsilon_p \frac{\partial C_p}{\partial t}$

[TimeDerivative](#): $\frac{dq}{dt}$

$$\text{CoupledCoeffTimeDerivative: } -\rho_p \frac{dq}{dt}$$

$$\text{GPoreConcAdvection (DGPoreConcAdvection): } \nabla \cdot (\varepsilon \mathbf{v} \cdot C)$$

$$\text{GVarPoreDiffusion (DGVarPoreDiffusion): } \nabla \cdot (\varepsilon D \cdot \nabla C)$$

$$\text{FilmMassTransfer: } -(1 - \varepsilon) \cdot k \cdot A_o (C - C_p) \quad \text{and} \quad -k \cdot A_o (C_p - C)$$

$$\text{ConstReaction: } k_f C_p S - k_r q$$

$$\text{MaterialBalance: } S = S_{max} - q$$

For more information on each of these kernels, click on the respective kernel names. It is worth noting that some of these kernels accept variable arguments for their parameters, while others expect given constant values. In addition, the $(1-\varepsilon)$ term for the first [FilmMassTransfer](#) kernel is not actually explicitly available in that given kernel. Thus, it may be convenient to create a new, compound rate parameter that is the product of $(1-\varepsilon)$ and k and pass that product to the kernel as the k value. As this project continues to grow and evolve, more and more specific kernels will be added as necessary in order to prevent these kinds of special treatment you may need to apply to your problems.

In addition to the kernels needed to simulate each piece of physics in the 4 equations, we need to include boundary conditions for the first equation, since it is a PDE. For this problem, we will have a boundary condition kernel at the inlet and a boundary condition at the outlet. Because our bulk concentration variable C involves a porosity term, we need to use the [DGPoreConcFluxBC](#) kernel. This kernel can be applied to the inlet and outlet boundaries simultaneously or separately because it uses the velocity vector and normal vectors at the boundary to naturally distinguish between inputs and outputs. At the inlet boundary, we need to specify a value for the concentration entering the domain.

$$\text{DGPoreConcFluxBC: } (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot C_{inlet} \quad \text{and} \quad (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot C$$

Now that we have identified all the kernels and boundary conditions, we can begin to construct the input file to simulate this case. For detailed information on each block of the input file and input file structure in general, review the [Input Files](#) section. In the below section, we discuss each block of the input file for this particular example and provide our reasoning for each specific parameter choice and/or kernel usage. The full input file will be available in the CATS code repository (https://github.com/aladshaw3/cats/tree/master/user_examples).

The Input File

First, we will add blocks to the input file to represent the physical domain we want to simulate on. In this example, we want to simulate adsorption in a packed column and thus our domain will be made cylindrical. To make an axis-symmetric 2D domain, we must specify a '[Problem]' block with 'coord_type' set to 'RZ' and then use the [GeneratedMeshGenerator](#) in the mesh block as shown below.

```
[Problem]
    coord_type = RZ
[]
```

```

[Mesh]
    [./my_mesh]
        type = GeneratedMeshGenerator
        dim = 2
        nx = 5
        ny = 20
        xmin = 0
        xmax = 0.05
        ymin = 0
        ymax = 0.1
    [../]
[]

```

The sections above create a cylindrical mesh with a radius of 0.05 m (x-direction → radius) and length of 0.1 m (y-direction → length). The domain is meshed into 5x20 elements and identified as a 2 dimensional structure.

After setting up the simulation domain, we need to declare what variable and auxiliary variables are involved with this simulation. The variables are the values that the model is implicitly solving for: (i) C, (ii) C_p , (iii) q, and (iv) S. Auxiliary variables can be specific parameters that are solved for independently or are just place holders for specific constant values we give them. In this example, we will just use constants as our auxiliary variables and declare their values to be their initial conditions.

```

[Variables]
    [./C]
        order = FIRST
        family = MONOMIAL
        initial_condition = 0
    [../]
    [./Cp]
        order = FIRST
        family = MONOMIAL
        initial_condition = 0
    [../]
    [./q]
        order = FIRST
        family = MONOMIAL
        initial_condition = 0
    [../]
    [./S]
        order = FIRST
        family = MONOMIAL
        initial_condition = 0
    [../]
[]

```

```

[AuxVariables]
    # porosity
    [./eps]
        order = FIRST
        family = LAGRANGE
        initial_condition = 0.5
    [../]
    [./vel_x]
        order = FIRST
        family = LAGRANGE
        initial_condition = 0
    [../]
    [./vel_y]
        order = FIRST
        family = LAGRANGE
        initial_condition = 3
    [../]
    [./vel_z]
        order = FIRST
        family = LAGRANGE
        initial_condition = 0
    [../]
    [./D]
        order = FIRST
        family = LAGRANGE
        initial_condition = 2.5E-5
    [../]
    [./k]
        order = FIRST
        family = LAGRANGE
        initial_condition = 1
    [../]
    [./k_eps]
        # Represents  $(1 - \text{eps}) * k$ 
        order = FIRST
        family = LAGRANGE
        initial_condition = 0.5
    [../]
    [./eps_p]
        order = FIRST
        family = LAGRANGE
        initial_condition = 0.25
    [../]

```



```

[./S_max]
    order = FIRST
    family = LAGRANGE
    initial_condition = 1
[../]
[]

```

Note that the AuxVariables do not cover all the parameters. Some kernels we use are looking for parameters in their subblocks as specific constant values, so we do not place those here as an AuxVariable. In future, newer kernels may have more flexibility for accepting both variable arguments and constant arguments.

Next, we need to invoke the Kernels, DGKernels, and BCs to invoke all the chemistry and physics that governs our model. We then pass to those kernels the variables, constant values, and other arguments that those kernels need to build the total model. Also note that every 'G' or 'DG' prefixed kernel or DG kernel must be paired with the same input arguments. The combinations of these two sets of kernels are needed to fully describe transport physics with DG methods. See [here](#) for more information.

```

[Kernels]
#Conservation of mass on C
[./C_dot]
    type = VariableCoefTimeDerivative
    variable = C
    coupled_coef = eps
[../]
[./C_gadv]
    type = GPoreConcAdvection
    variable = C
    porosity = eps
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]
[./C_gdiff]
    type = GVarPoreDiffusion
    variable = C
    porosity = eps
    Dx = D
    Dy = D
    Dz = D
[../]
[./C_trans_Cp]
    type = FilmMassTransfer
    variable = C

```

```

        coupled = Cp
        rate_variable = k_eps
        # Area to volume ratio: Ao
        av_ratio = 5000
[../]

#Conservation of mass on Cp
[./Cp_dot]
    type = VariableCoeftTimeDerivative
    variable = Cp
    coupled_coef = eps_p
[../]
[./Cp_trans_C]
    type = FilmMassTransfer
    variable = Cp
    coupled = C
    rate_variable = k
    # Area to volume ratio: Ao
    av_ratio = 5000
[../]
[./ads_q]
    type = CoupledCoeftTimeDerivative
    variable = Cp
    coupled = q
    time_coeff = 1500
[../]

# Conservation of mass for q
[./q_dot]
    type = TimeDerivative
    variable = q
[../]
[./q_rxn]
    type = ConstReaction
    variable = q
    this_variable = q
    forward_rate = 2
    reverse_rate = 0.5
    scale = 1
    reactants = 'Cp S'
    reactant_stoich = '1 1'
    products = 'q'
    product_stoich = '1'
[../]

```

```

# Conservation of mass for S
[./S_bal]
    type = MaterialBalance
    variable = S
    this_variable = S
    coupled_list = 'S q'
    weights = '1 1'
    total_material = S_max
[../]

[]

[DGKernels]
[./C_dgadv]
    type = DGPoreConcAdvection
    variable = C
    porosity = eps
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]
[./C_dgdiff]
    type = DGVarPoreDiffusion
    variable = C
    porosity = eps
    Dx = D
    Dy = D
    Dz = D
[../]

[]

[BCs]
[./C_FluxIn]
    type = DGPoreConcFluxBC
    variable = C
    boundary = 'bottom'
    u_input = 1
    porosity = eps
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]

```

```

[./C_FluxOut]
    type = DGPoreConcFluxBC
    variable = C
    boundary = 'top'
    porosity = eps
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]
[]

```

Next, for this demonstration, we want to monitor some specific simulation results as Postprocessors. Of particular interest is (i) the exit concentration of C to yield a breakthrough curve and (ii) the average adsorption (q) in the column. Since breakthrough occurs at a boundary, we want to use the [SideAverageValue](#) object at the 'top' boundary. This will yield an integral average of C in the top-most set of elements at that boundary.

```

[Postprocessors]
  [./C_exit]
    type = SideAverageValue
    boundary = 'top'
    variable = C
    execute_on = 'initial timestep_end'
  [../]
  [./q_avg]
    type = ElementAverageValue
    variable = q
    execute_on = 'initial timestep_end'
  [../]
[]

```

Lastly, we need to setup the Preconditioning, Executioner, and Outputs. Examples for how to setup these blocks were provided earlier ([Preconditioning and Executioner](#) and [Outputs](#)). To view all of these put together, check out the actual input file for this example in the CATS code repository at (https://github.com/aladshaw3/cats/tree/master/user_examples).

Conservation of Energy

This example will continue from where the last one left off. Here, we will consider an energy balance inside a packed column. In this scenario, we have the energy density of the fluid phase (E_f) and solid phases (E_s) to consider. Those energy densities are related to the temperatures of each phase (T_f and T_s) through their respective densities and heat capacities. In the absence of reactions and adsorption, those energy balances and temperature relationships could be represented as follows.

$$E_f: \quad \varepsilon \frac{\partial E_f}{\partial t} + \nabla \cdot (\varepsilon \mathbf{v} \cdot E_f) = \nabla \cdot (\varepsilon K_f \cdot \nabla T_f) - (1 - \varepsilon) \cdot h \cdot A_o (T_f - T_s)$$

$$E_s: \quad \frac{\partial E_s}{\partial t} = \nabla \cdot (K_s \cdot \nabla T_s) - h \cdot A_o (T_s - T_f)$$

$$T_f: \quad E_f = \rho_f c_{pf} T_f$$

$$T_s: \quad E_s = \rho_p c_{ps} T_s$$

Note that the units from above are as follows: E_f and E_s (J/m³), K_f and K_s (W/m/K), h (W/m²/K), A_o (m⁻¹), T_f and T_s (K), ρ_f and ρ_p (kg/m³), and c_{pf} and c_{ps} (J/kg/K). The energy density of the fluid is per volume of fluid and the energy density of the solid is per bulk volume of solids. As a consequence, the solid properties must also be per bulk volume (at least for this example).

The kernels that correspond to the above physics are shown below:

[VariableCoefTimeDerivative](#): $\varepsilon \frac{\partial E_f}{\partial t}$

[TimeDerivative](#): $\frac{\partial E_s}{\partial t}$

[GPoreConcAdvection \(DGPoreConcAdvection\)](#): $\nabla \cdot (\varepsilon \mathbf{v} \cdot E_f)$

[GPhaseThermalConductivity \(DGPhaseThermalConductivity\)](#): $\nabla \cdot (\varepsilon K_f \cdot \nabla T_f)$ and $\nabla \cdot (K_s \cdot \nabla T_s)$

[PhaseEnergyTransfer](#): $-(1 - \varepsilon) \cdot h \cdot A_o (T_f - T_s)$ and $-h \cdot A_o (T_s - T_f)$

[PhaseTemperature](#): $E_f = \rho_f c_{pf} T_f$ and $E_s = \rho_p c_{ps} T_s$

In addition to the above system of equations for the interior of the domain, we must include boundary conditions for E_f and E_s , since these are the variables that are represented by PDEs. As with the conservation of mass, the mobile fluid phase will have boundary conditions at the inlet and outlet of the column where the gas flows into and out of the system. Those boundary condition kernels are provided below.

[DGFlowEnergyFluxBC](#): $(\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot \rho_f c_{pf} T_{f,inlet}$ and $(\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot E_f$

However, since energy can also be exchanged across the walls of the domain, we must also include a boundary condition for that exchange of energy for both the fluid and solid phases. The reasoning behind including this energy term for both phases is due to the fact that in a packed column, but phases will be in contact with the wall. Each may have a different representative area for contact with the wall represented by the ε and $(1 - \varepsilon)$ factors, respectively.

[DGWallEnergyFluxBC](#): $h_w \cdot \varepsilon \cdot (T_f - T_w)$ and $h_w \cdot (1 - \varepsilon) \cdot (T_s - T_w)$

Since we have already covered the input file structure in great detail in the [above example](#), as well as in the main section on [input files](#), we will not rework through all those details here. Instead, just see the sample input file provided (https://github.com/aladshaw3/cats/tree/master/user_examples).

Coupling Mass and Energy

Up to this point, we have demonstrated both a mass and energy balance separately, however, realistic simulations will involve the coupling of mass and energy together. To accomplish this, we need to make a minor kernel change to the mass balances and add a new kernel to the energy balances. For the mass balance, we need to update the [ConstReaction](#) kernel to the [ArrheniusReaction](#) or [ArrheniusEquilibriumReaction](#) kernel to couple the forward and reverse rates with temperature. Since the reaction in the mass balance is an adsorption/surface reaction, it should be coupled with the solid temperature and not the gas temperature. The new kernel also has different parameters to set and those parameters will be common with the corresponding [ArrheniusReactionEnergyTransfer](#) or [ArrheniusEquilibriumReactionEnergyTransfer](#) kernels. Thus, it is advantageous to place those common parameters under the [GlobalParams](#) block of the input file as shown below.

```
[GlobalParams]
  forward_activation_energy = 0
  forward_pre_exponential = 2
  reverse_activation_energy = 2305
  reverse_pre_exponential = 1
  enthalpy = -2305
[]
```

In the Kernels block of the input file, the [ConstReaction](#) kernel for variable q is replaced with [ArrheniusReaction](#). Since the forward and reverse rates are calculated as a function of temperature, we can replace the input arguments for the rates with just the coupled temperature of the solid (T_s). For the energy balance of the solid phase, we need to add a new kernel to account for the energy transfer caused by the reaction. Thus, the new energy balance would be as shown below.

$$E_s: \quad \frac{\partial E_s}{\partial t} = \nabla \cdot (K_s \cdot \nabla T_s) - h \cdot A_o (T_s - T_f) + \rho_p (-\Delta H) [k_f C_p S - k_r q]$$

The new term involves the enthalpy of the reaction, the reaction rate expression, and a unit conversion factor, which in this case is the particle bulk density. This ensures that the units of the new term are in energy per bulk volume of solids. In the [ArrheniusReactionEnergyTransfer](#) kernel, the density of the particles is passed as the 'volume_frac' argument. The enthalpy was already declared in the GlobalParams block, so it does not need to be repeated here. To look at the actual input file for this coupled simulation case, check out (https://github.com/aladshaw3/cats/tree/master/user_examples).

Adding Gas Properties

Up to this point, we have simulated processes with constant coefficients. However, rate parameters such as the diffusion/dispersion coefficients and film mass transfer rates are themselves functions of temperature and other factors based on kinetic theory of gases. If we want properties such as these to be calculate automatically, we can invoke the various [Auxiliary Kernels](#) for those parameter calculations to use in our simulation.

In this demo, we will use the auxiliary system to calculate (i) pressure drop across the column, (ii) dispersion coefficients, (iii) film mass transfer rates, (iv) gas density, (v) gas viscosity, (vi) gas thermal

conductivity, and (vii) gas specific heat. To help accommodate this, we will also add another auxiliary variable for the ‘carrier’ gas, since our concentration variable C is too dilute to account for total gas density. In addition, previously we had introduced an auxiliary variable ‘ k_eps ’ to represent a conversion factor of the true k value by the $(1 - \epsilon)$ factor. Now, we need to move that unit conversion into the kernel and apply it to the ‘ av_ratio ’ to correct the kernel so that the ‘ k ’ value can come exclusively from the auxiliary kernels.

Below are the auxiliary kernels we need to invoke and what they apply to:

AuxErgunPressure:	New auxiliary variable P (in Pa)
GasSpeciesAxialDispersion:	Parameter D (in m^2/s)
GasSpeciesMassTransCoef:	Parameter k (in m/s)
GasDensity:	Parameter ρ_f (in kg/m^3)
GasViscosity:	New auxiliary variable μ_f (in $kg/m/s$)
GasThermalConductivity:	Parameter K_f (in $W/m/K$)
GasSpecHeat:	Parameter c_{pf} (in $J/kg/K$)

For this demo, our carrier gas variable will be ‘N2’ (i.e., nitrogen gas) and we will give that a concentration of $40 \text{ mol}/m^3$, which will roughly correspond to density of air at standard temperature and pressure. Then, in the GlobalParams block, we add a set of input arguments that are common to all the auxiliary kernels we are invoking in this simulation. This includes the list of gases, their corresponding molecular weights, and their Sutherland’s constants. In this example, our gas concentration variable C will represent the concentration of water vapor, so we will use the molecular weight and Sutherland information for water. Therefore, we add the following lines to the GlobalParams block. For more information on these input arguments, see [GasPropertiesBase](#).

```
gases = 'N2 C'
molar_weights = '28 18'
sutherland_temp = '300.55 298.16'
sutherland_const = '111 784.72'
spec_heat = '1.04 1.97'
is_ideal_gas = false
execute_on = 'initial timestep_end'
```

Lastly, add the auxiliary kernels list above to the AuxKernels block of the input file. Each auxiliary kernel may require some additional variable arguments, such as temperature and pressure. For details on each input argument, click on the links/names of the auxiliary kernels named above. To view the input file for this simulation, check out (https://github.com/aladshaw3/cats/tree/master/user_examples).

NOTE: You will need to change some of the auxiliary variables from LAGRANGE to MONOMIAL families. In general, only velocities and temperatures should be LAGRANGE.

NOTE: Simulation results for temperature will be very different, because of the new properties.

Coupling Mass and Energy with Simple Gas Properties

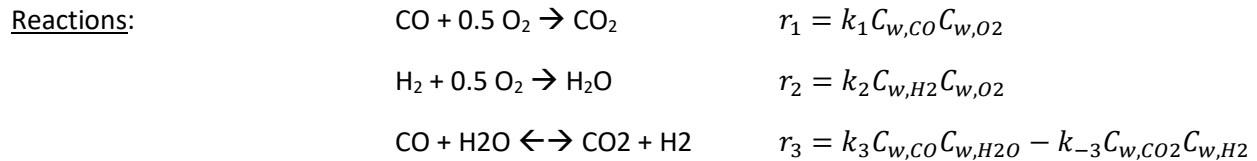
In the discussion above, we talked about using the [GasProperties](#) system to calculate properties associated with mass and energy balances. In this older properties system, a user was forced into using a specific unit basis for their problems. This can be very inconvenient for systems that have widely varied length and time scales. In addition, these more involved property calculations requires very specific information on individual gas species properties that may be difficult to find. To get around both these issues, a newer [SimpleGasProperties](#) system was developed that makes a few simplifying assumptions and relationships to calculate system properties with limited parameter knowledge. This newer system also allows the user to specify parameter units on input and desired units on output, thus allowing a user to perform balances on any unit basis for both mass and energy balances.

An example of using this system can be found in the CATS repository located under 'user_examples/simple_coupling_mass_and_energy.i'. In this example, we solve a fully coupled mass and energy system for the light-off of CO and H₂ in a three-way catalyst. We can represent the problem mathematically as follows...

MASS BALANCES

$$\text{Fluid:} \quad \varepsilon_b \frac{\partial C_i}{\partial t} + \nabla \cdot (\varepsilon_b \mathbf{v} \cdot C_i) = \nabla \cdot (\varepsilon_b D \cdot \nabla C_i) - (1 - \varepsilon_b) \cdot A_o \cdot k_m (C_i - C_{w,i})$$

$$\text{Solid:} \quad \varepsilon_w (1 - \varepsilon_b) \frac{\partial C_{w,i}}{\partial t} = (1 - \varepsilon_b) \cdot A_o \cdot k_m (C_i - C_{w,i}) + (1 - \varepsilon_b) \cdot \sum u_i r_i$$



ENERGY BALANCES

$$\text{Fluid:} \quad \varepsilon_b \frac{\partial E_f}{\partial t} + \nabla \cdot (\varepsilon_b \mathbf{v} \cdot E_f) = \nabla \cdot (\varepsilon_b K_f \cdot \nabla T_f) - (1 - \varepsilon_b) \cdot h_s \cdot A_o (T_f - T_s)$$

$$\text{Solid:} \quad (1 - \varepsilon_b) \frac{\partial E_s}{\partial t} = \nabla \cdot ((1 - \varepsilon_b) K_s \cdot \nabla T_s) - (1 - \varepsilon_b) \cdot h_s \cdot A_o (T_s - T_f) + (1 - \varepsilon_b) \cdot \sum (-\Delta H_i) r_i$$

$$\text{Fluid Temperature:} \quad E_f = \rho_f c_{pf} T_f$$

$$\text{Solid Temperature:} \quad E_s = \rho_p c_{ps} T_s$$

In this example, the spatial dimensions are in cm, the time dimension is in min, our concentrations (C_i and $C_{w,i}$) are in units of mol/L, and our energy densities (E_f and E_s) are in J/cm³. Because of our unit convention for concentrations, the reaction variables (r_i) have units of mol/L/min, where the volume is on a solids volume basis. Thus, when those reaction kernels are put into both the mass and energy balances, we need the scale factor of $(1 - \varepsilon_b)$ to put the balances on the basis of total volume (rather than just solids volume). In addition, this means that the units of reaction enthalpies (ΔH_i) must include a conversion from volume as L to volume as cm³ to account for the fact that out

volume units on energy density are different from out volume units on concentration. As a result, the units for ΔH_i are $\text{J}^*\text{L}/\text{mol}/\text{cm}^3$ (which is equivalent to kJ/mol).

Also note, that the sign of ΔH_i used in the kernels is the opposite sign of the actual ΔH_i values. This is because of the implicit negative sign in the energy balance. Thus, if ΔH_i is $-283 \text{ kJ}/\text{mol}$, you supply the value as $+283$.

The kernels invoked are mostly the same sets of kernels discussed above for [mass](#) and [energy](#) balances independently, with some minor differences for coupling reactions. Both the reaction coupling in the mass and reaction coupling in the energy balance use the same kernel (see below).

[ScaledWeightedCoupledSumFunction](#): $(1 - \varepsilon_b) \cdot \sum u_i r_j$ and $(1 - \varepsilon_b) \cdot \sum (-\Delta H_i) r_j$

In this example, we also use the recursive Krylov subspace methods to precondition linear steps with other Krylov subspace methods. Details of this, and other example details are written in the comments of the simulation run file ('user_examples/simple_coupling_mass_and_energy.i'). See that input file for additional details.

MOOSE CG Incompressible Navier-Stokes

The incompressible Navier-Stokes module is not developed in CATS, it is a part of the MOOSE framework. However, we can utilize this module inside of CATS to resolve approximations to the flow field in a domain. In this section, we will discuss 2 methods to invoke the Navier-Stokes module: (i) in cartesian coordinates and (ii) in RZ-cylindrical coordinates. The simulations in cartesian coordinates will be further subdivided into fully bounded and semi-bounded domains. Information on each case is provided in the sections below.

INS Cartesian - Fully Bounded

In a fully bounded domain, there will be walls to constrain and constrict the flow such that only the truly open boundaries will allow for flux. At the walls of the system, we will apply a "no slip" boundary condition, which is a standard wall condition for Navier-Stokes problems. Then, we only need to define either the inlet flux or outlet flux for the open boundaries. The reason we only need to specify one of the two open boundaries is because the continuity equation will inherently enforce a total flow balance such that the flow leaving the domain will equal the flow entering the domain.

For the purpose of this demonstration, we will use a 2D flow domain of effectively the same size as the mesh from the [previous examples](#). However, the previous mesh was in 'RZ' cylindrical coordinates. Here, we are in cartesian coordinates, so we will remove the 'RZ' designation in the Problem block of the input file. In addition, we will modify the mesh such that the total x-direction length would span the 'diameter' of the column from the previous example. The reason for this is so we can get an approximate flow cross section of the column as if the column were a cubic block rather than a cylinder. Since we are extended the length of the x-direction, we also need to increase the number of elements in the x-direction. The new mesh block can be assembled as follows.

```

[Mesh]
  [./my_mesh]
    type = GeneratedMeshGenerator
    dim = 2
    nx = 10
    ny = 20
    xmin = -0.05
    xmax = 0.05
    ymin = 0
    ymax = 0.1
  [../]
[]

```

We also need to add several input arguments into the GlobalParams block for some stabilization options for the incompressible Navier-Stokes module. It is highly recommended that you always use the same options provided below, except for the 'gravity' and 'alpha' arguments. These can be changed to fit your specific problem or needed level of stabilization. The 'gravity' argument is a vector of acceleration constants you can provide for x, y, and z, respectively. In this case, we specify a gravitational acceleration constant of 9.8 m/s^2 in the negative y-direction. The 'alpha' option can be any number between 0.1 and 2, where 1 seems to give the best results. See the example file for more details (https://github.com/aladshaw3/cats/tree/master/user_examples).

```

[GlobalParams]
  gravity = '0 -9.8 0'
  alpha = 1
  integrate_p_by_parts = true
  supg = true
  pspg = true
  laplace = true
  convective_term = true
  transient_term = true
[]

```

To use the incompressible Navier-Stokes module, we are required to provide a [Materials](#) block that gives the fluid phase density (in kg/m^3) and viscosity (in kg/m/s). For this, there is a custom Materials object in CATS called [INSFluid](#) that we will invoke. The input arguments for density and viscosity may be given as constants or as auxiliary variables. For the purpose of this demo, we will give them constants that emulate the approximate values of density and viscosity of air under our [simulation conditions from before](#). The Materials block for this input file is structured as below.

```

[Materials]
  [./ins_material]
    type = INSFluid
    density = 1.13

```

```

        viscosity = 2.25E-5
    [../]
[]

```

In simulations of incompressible Navier-Stokes, the velocity variable is introduced as it's vector components (i.e., v_x , v_y , and v_z) and a variable for dynamic pressure (p) must be included for the continuity equation. For 2D simulation, we still include v_z , but only as an auxiliary variable whose value is set to 0. All of the variables for Navier-Stokes must be of the LAGRANGE family. See the example files for samples of inputs (https://github.com/aladshaw3/cats/tree/master/user_examples).

The kernels that need to be invoked for the incompressible Navier-Stokes module are formulated as follows. Note that there are no DG kernels to invoke here because the MOOSE Navier-Stokes module uses a different stabilization technique for transport.

```

[Kernels]
  #Continuity Equation
  [./mass]
    type = INSMass
    variable = p
    u = vel_x
    v = vel_y
    w = vel_z
    p = p
  [../]

  # Conservation of x-momentum
  [./x_dot]
    type = INSMomentumTimeDerivative
    variable = vel_x
  [../]
  [./x_ins]
    type = INSMomentumLaplaceForm
    variable = vel_x
    u = vel_x
    v = vel_y
    w = vel_z
    p = p
    component = 0
  [../]

  # Conservation of y-momentum
  [./y_dot]
    type = INSMomentumTimeDerivative
    variable = vel_y
  [../]

```

```

[./y_ins]
    type = INSMomentumLaplaceForm
    variable = vel_y
    u = vel_x
    v = vel_y
    w = vel_z
    p = p
    component = 1
[../]

[]

```

The most important components for the incompressible Navier-Stokes simulations are the boundary conditions. Due to how the module was constructed in MOOSE, the kernels will always be the same. Thus, the various boundary conditions you apply are really what controls the flow profile. Review the [incompressible Navier-Stokes](#) section for a listing of the most common boundaries. In this example, we have 2 wall boundaries (i.e., 'left' and 'right' that correspond to the 'xmin' and 'xmax' values in the mesh) and 2 open boundaries (i.e., 'bottom' and 'top' that correspond to 'ymin' and 'ymax' values in the mesh). At the 'bottom' boundary we define our inflow conditions using the [INSNormalFlowBC](#), which dictates the flux normal to the boundary for a given variable. We do not need to specify anything for the 'top' boundary as the continuity equation will inherently enforce the continuity of the flow. For the 'left' and 'right' boundaries, we need to apply the [PenaltyDirichletBC](#) for both 'vel_x' and 'vel_y' to enforce the "no slip" boundary conditions at the walls. The BCs block of the input file can be viewed from the example files (https://github.com/aladshaw3/cats/tree/master/user_examples).

Lastly, we may want to use some custom Postprocessors as a way to check and make sure the continuity equation and our boundary conditions are conserving the flow. To check this, we can use the VolumetricFlowRate postprocessor that is built into the MOOSE module to compute the flow rate at the 2 open boundaries. If the flow is conserved, then the exit flow rate will approximately equal to the exit flow rate. Note that depending on your tolerances for the solver, these values may be slightly different, but they should be very close to each other. Be sure to check out the actual example input files for these simulation cases (https://github.com/aladshaw3/cats/tree/master/user_examples).

NOTE: The incompressible Navier-Stokes module has very poor convergence if you do not use 'lu' as one of your preconditioning options. See [PETSc options](#) for more details.

INS Cartesian - Semi-Bounded

Occasionally, your simulation domain may not be fully bounded (i.e., does not fully confine the flow or have walls at all boundaries) or you may want to simulate a case where there is expected to be symmetry of the flow profile at a boundary. In these cases, everything from the [above example](#) will be the same, but we must change the boundary conditions that we specify. In this example, let's consider the same domain as [above](#), but because the flow profile will be symmetric about $x=0$, we will only mesh half the domain and invoke some custom boundaries to create that flow symmetry.

The new mesh would be as follows:

```
[Mesh]
    ./my_mesh
        type = GeneratedMeshGenerator
        dim = 2
        nx = 5
        ny = 20
        xmin = 0
        xmax = 0.05
        ymin = 0
        ymax = 0.1
    [../]
[]
```

Now, the ‘left’ boundary (corresponding to ‘xmin’) of the domain is our axis of symmetry. We will have the same inflow and ‘right’ boundary conditions (i.e., “no slip” at the walls) as the [previous example](#), but the ‘left’ boundary will be different for ‘vel_x’ and ‘vel_y’. For ‘vel_x’ at the ‘left’ boundary, we must enforce a zero velocity condition, which is essentially the same as the “no slip” boundary. The reason for this is we must not allow velocities to “flux” across a boundary normal to our axis of symmetry. If flux is allowed, then we essentially have a “vacuum” condition at that boundary that would allow flow to be lost to outside our domain. So we use the [PenaltyDirichletBC](#) to ensure there is no flux in the x-direction at the axis of symmetry.

```
    ./x_center
        type = PenaltyDirichletBC
        variable = vel_x
        boundary = 'left'
        value = 0
        penalty = 1000
    [../]
```

For the ‘vel_y’ at the ‘left’ boundary, we must use the NeumannBC to enforce the slope of the velocity in the y-direction is zero. Note that you can also invoke this boundary condition by doing nothing in the input file as this is the “natural” boundary condition for standard finite elements. The physical interpretation of this boundary condition is that we are enforcing that the velocity in elements just outside the domain must be the same as the velocity for elements just inside the domain (i.e., the velocity profile is symmetric about the domain).

```
    ./y_center
        type = NeumannBC
        variable = vel_y
        boundary = 'left'
        value = 0
    [../]
```

For additional details, check out (https://github.com/aladshaw3/cats/tree/master/user_examples). Note that the simulation results from the fully bounded and semi-bounded domains will produce the same velocity profiles from $x=0$ to $x=x_{\max}$, thus demonstrating that we can reduce complexity of simulations by taking advantage of symmetry where possible.

INS RZ

The last incompressible Navier-Stokes example is for building on prior [mass](#) and [energy](#) examples in our previously cylindrical domain. What we did above ([fully bounded](#) and [semi-bounded](#)) was to try to estimate a flow profile for a 2D slice of our column, but in cartesian coordinates. All dimensions were the same as the cylindrical column, but the spatial representation was not accurate. As a result, the cartesian examples provided an effective flow profile for a cubic conduit and not a cylinder. To get the flow profile in a cylinder, we must use some custom 'RZ' Navier-Stokes kernels that apply coordinate system transformations necessary to simulate a more accurate flow profile.

NOTE: These kernels are significantly less efficient and may change in future.

There are only 3 changes that need to be made to our previous example simulations to account for the coordinate changes: (i) the Problem block needs to define the 'coord_type' as 'RZ', (ii) the 'INS' kernels, except for the time derivatives, need to be suffixed with 'RZ' (i.e., INSMass \rightarrow INSMassRZ), and (iii) the boundary conditions need to be updated. Boundary condition at the inflow boundary for the 'vel_y' variable is the same as before, but now the "no slip" conditions need to be applied to 'vel_x' at both the 'left' and 'right' boundaries (i.e., center line and the wall) while the "no slip" condition for 'vel_y' is only applied at the 'right' boundary (i.e., the wall). Check out the example files for all Navier-Stokes cases at (https://github.com/aladshaw3/cats/tree/master/user_examples).

CATS DG Navier-Stokes

Within CATS, we now have a working implementation of both incompressible and compressible Navier-Stokes using DG methods. This implementation solves the continuity and momentum conservation equations (outlined [here](#)) by performing integration-by-parts for each velocity component and by applying penalty-based pseudo-Dirichlet boundary conditions to enforce a 'No Slip' condition at the walls or obstructions in a mesh.

NOTE: A consequence of integrating-by-parts is that this formulation is valid ONLY in Cartesian coordinates. Thus, you cannot use these methods currently for 'RZ' coordinate systems.

Variables

To use our Navier-Stokes module, you will need to define a 'pressure' variable and a 'velocity' variable for each direction (i.e., x, y, and z directions). **NOTE:** Any velocity component is technically optional, as you can replace one velocity term with a 0 if you don't need that dimension.

In this example, we will create variables for 'pressure' and 'vel_x' and 'vel_y' because we will do a 2D Navier-Stokes simulation. To do this, we MUST define 'pressure' as a 'FIRST' order 'LAGRANGE'

variable and each velocity as a 'SECOND' order 'MONOMIAL' variable. For velocity in the z-direction, we create an 'AuxVariable' and default its value to 0.

NOTE: In general, you should provide variables for all velocity components, even if the simulation is only 2D. This is because how some of the 'DG' kernels are implemented, you are required to supply velocity terms as variables and cannot just use default coupling.

```
[Variables]
  [./pressure]
    order = FIRST
    family = LAGRANGE
    initial_condition = 0.0
  [../]
  [./vel_x]
    order = SECOND
    family = MONOMIAL
    initial_condition = 0.0
  [../]

  [./vel_y]
    order = SECOND
    family = MONOMIAL
    initial_condition = 0.0
  [../]
[]

[AuxVariables]
  [./vel_z]
    order = SECOND
    family = MONOMIAL
    initial_condition = 0.0
  [../]
[]
```

Kernels/DGKernels

Next, we need to add kernels to describe the Navier-Stokes equations and integrate the solution by parts. This is accomplished by using a combination of some of the existing DG kernels discussed in other examples, as well as combining those kernels with new kernels designed specifically for the Navier-Stokes equations. Below gives a brief example of how to invoke the kernels we need, and what equations they solve.

```
[Kernels]
#### Enforce Divergence of velocity = 0 ###
### NOTE: If 'rho' is constant, then this will be incompressible
###       but if 'rho' is not constant, this will account for pressure accordingly
[./continuity]
```

```

        type = DivergenceFreeCondition
        variable = pressure
        ux = vel_x
        uy = vel_y
        uz = vel_z
        coupled_scalar = rho
[../]

#### Conservation of x-momentum ####
#   This example only include 'internal' forces
#   (i.e., the viscous forces and pressure forces)

# rho* d(vel_x)/dt
[./x_dot]
        type = VariableCoefTimeDerivative
        variable = vel_x
        coupled_coef = rho
[../]
# -grad(P)_x
[./x_press]
        type = VectorCoupledGradient
        variable = vel_x
        coupled = pressure
        vx = 1
[../]
# Div*(mu*grad(vel_x))
[./x_gdiff]
        type = GVariableDiffusion
        variable = vel_x
        Dx = mu
        Dy = mu
        Dz = mu
[../]
# Div*(rho*vel*vel_x)
[./x_gadv]
        type = GNSMomentumAdvection
        variable = vel_x
        this_variable = vel_x
        density = rho
        ux = vel_x
        uy = vel_y
        uz = vel_z
[../]
# (1/3)*mu* (grad(Div*vel)_x)
[./x_gvisdiv]
        type = GNSViscousVelocityDivergence
        variable = vel_x

```



```

        this_variable = vel_x
        viscosity = mu
        ux = vel_x
        uy = vel_y
        uz = vel_z
    [../]

[]

# Recall: ANY 'G' prefixed kernel MUST have an equivalent 'DG' kernel active
[DGKernels]
# Div*(mu*grad(vel_x))
[./x_dgdiff]
    type = DGVariableDiffusion
    variable = vel_x
    Dx = mu
    Dy = mu
    Dz = mu
[../]
# Div*(rho*vel*vel_x)
[./x_dgadv]
    type = DGNSMomentumAdvection
    variable = vel_x
    this_variable = vel_x
    density = rho
    ux = vel_x
    uy = vel_y
    uz = 0
[../]
# (1/3)*mu* (grad(Div*vel)_x)
[./x_gvisdiv]
    type = DGNSViscousVelocityDivergence
    variable = vel_x
    this_variable = vel_x
    viscosity = mu
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]

[]

```

Boundary Conditions

Lastly, we need to enforce proper boundary conditions on the system. For the pressure variable, the simplest boundary condition is to enforce that the pressure at any outlet to the system is 0. In this way, you are actually solving for the 'dynamic' or 'gage' pressure in the system rather than the absolute pressure. However, in principle, you can supply any Dirichlet condition you want for pressure at the

outlet. **It should also be noted, the units of pressure that get solved for are entirely dependent on the units you give for density and viscosity of the fluid.**

```
[BCs]
# Zero pressure at exit
[./press_at_exit]
    type = DirichletBC
    variable = pressure
    boundary = 'outlet'
    value = 0.0
[../]
[]
```

The boundary conditions for velocity should come in 3 parts: (i) some inlet velocity for flux into the domain, (ii) some conservation of momentum statement leaving the domain, and (iii) some form of 'No Slip' condition at the walls or at any obstructions in the domain. For BCs (i) and (iii), we will make use of the 'PenaltyDirichletBC' system in MOOSE. In most cases tested, a 'penalty' value of about 300 seems to be the best compromise between accuracy and efficiency. There are several flavors of these types of boundary conditions, so have a look at this link for more options (<https://mooseframework.inl.gov/syntax/index.html>).

For BC of type (ii), we have developed a conservation of momentum equation for the outflow boundary of any domain ([DGNSMomentumOutflowBC](#)). This would be applied to all velocity components at all outlets of a domain.

```
[BCs]
# Inlet velocity at boundary
[./vel_x_inlet]
    type = FunctionPenaltyDirichletBC
    variable = vel_x
    boundary = 'inlet'
    penalty = 3e2
    function = '0.1*t'
[../]

### Momentum Flux Out of Domain ###
# in x-direction
[./vel_x_outlet]
    type = DGNSMomentumOutflowBC
    variable = vel_x
    this_variable = vel_x
    boundary = 'outlet'
    density = rho
    ux = vel_x
    uy = vel_y
```

```

        uz = 0
    [../]
    # in y-direction
    [./vel_y_outlet]
        type = DGNSMomentumOutflowBC
        variable = vel_y
        this_variable = vel_y
        boundary = 'outlet'
        density = rho
        ux = vel_x
        uy = vel_y
        uz = 0
    [../]

    ### No Slip Conditions at the Walls ###
    # in x-direction
    [./vel_x_obj]
        type = PenaltyDirichletBC
        variable = vel_x
        boundary = 'top bottom object'
        value = 0.0
        penalty = 300
    [../]

    # in y-direction
    [./vel_y_obj]
        type = PenaltyDirichletBC
        variable = vel_y
        boundary = 'top bottom object'
        value = 0.0
        penalty = 300
    [../]
[]

```

Other Information

The simulation run file for the above example also includes a conservative ‘tracer’ to view how the simulated flow field carries a mass through the domain and around an obstruction. The file to run is found in ‘user_examples’ under the file name ‘dg_incompressible_navier_stokes_implementation.i’. The compressible version is ‘dg_compressible_navier_stokes_implementation.i’. When you run the incompressible file and simulate a flow field, you will get a flow field that looks something Figure 1 below (NOTE: The compressible example involves a linearly decreasing density in the x-direction, and so velocity increases to ensure conservation of fluid mass when this occurs). This image shows the relative magnitudes of the velocity vector, where blue indicates a velocity of 0 (i.e., no slip at the walls or the object) and the darker red indicates high velocities.

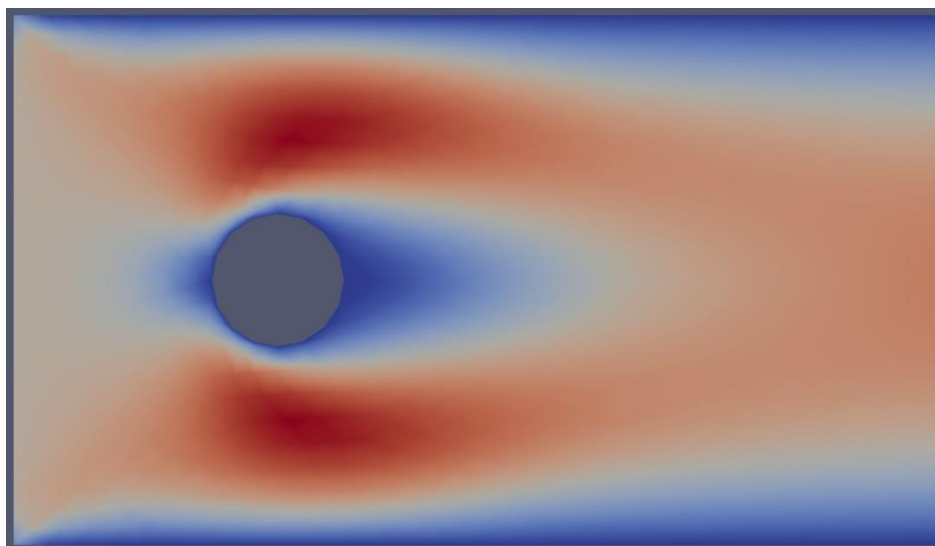


Figure 1: Incompressible flow field around an obstruction. Velocity at the inlet and exit are roughly the same because the flow is incompressible and the aperture for the inlet and outlet are the same size.

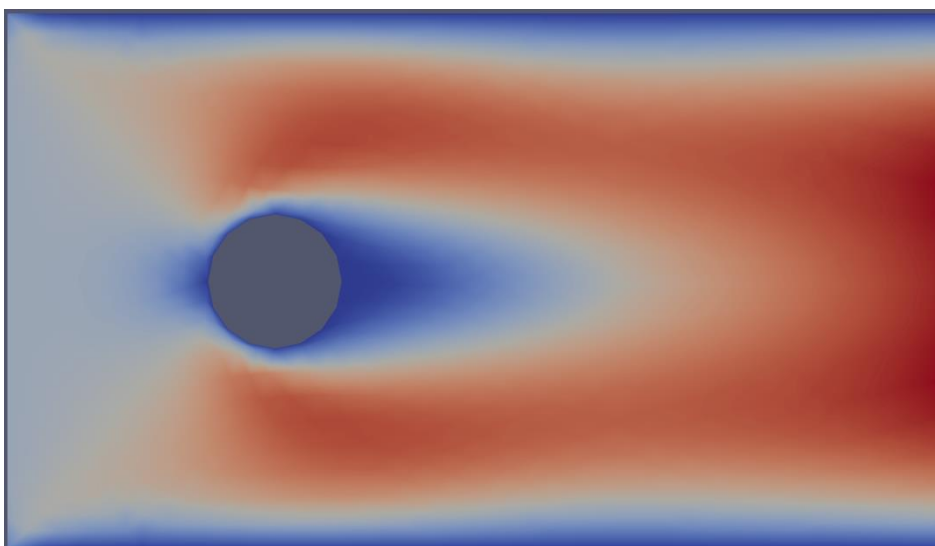


Figure 2: Compressible flow field around an obstruction. The density in this flow field varies linearly in the x-direction from 1 to 0.5 (left to right). As such, the velocities get higher towards the exit of the domain to maintain the total fluid mass flux.

Coupling Navier-Stokes, Mass, and Energy

The culmination of all prior examples would be to couple together all physics of flow fields, mass balances, and energy balances. To keep things simple, we will build upon the input file developed for the [“Adding Gas Properties”](#) example and simply add in the [incompressible Navier-Stokes example for RZ coordinates](#). Note that the RZ formulation of the incompressible Navier-Stokes is not very computationally efficient, so in this input file many of the solver options were changed to gain greater efficiency and stability. The end result from this simulation is remarkably similar to that of the [“Adding Gas Properties”](#) example results. This is mainly because in a packed column the flow profiles are fairly uniform, so adding Navier-Stokes to this simulation did not add much more unique information. In

general, for packed columns in RZ coordinates, it is usually unnecessary to invoke Navier-Stokes kernels. Instead, you can produce a reasonable approximation by just using an average linear velocity based on the total volumetric flow rate and the cross-sectional void area of the column. Have a look at the sample input file for more details (https://github.com/aladshaw3/cats/tree/master/user_examples).

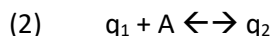
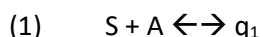
Simulations with Numerous Reactions

So far we have discussed how to setup relatively simple problems that involve only a handful of reactions. However, real chemical systems will be extraordinarily complex and might involve many, many reactions. While you can use everything we have discussed previously to solve these problems, it may quickly become cumbersome to redefine a reaction again and again when it shows up in multiple mass balances or equations.

To eliminate redundancy around this issue, you can define reactions and their associated rates as non-linear variable themselves, then couple those variables representing those rate functions into other non-linear variables using the kernels for [WeightedCoupledSumFunction](#) and [ScaledWeightedCoupledSumFunction](#).

Example 1: Multi-Reaction Adsorption

Consider the following set of reactions:



In this example, we want to define accumulation terms (or time derivatives) to solve for the adsorbed amounts of each surface species (q_1 and q_2). Mathematically, if we can define each reaction rate as:

$$r_1 = k_{f,1} * S * A - k_{r,1} * q_1 \quad \& \quad r_2 = k_{f,2} * q_1 * A - k_{r,2} * q_2$$

Then, we can define each rate of accumulation of q_1 and q_2 as:

$$dq_1/dt = r_1 - r_2 \quad \& \quad dq_2/dt = r_2$$

While you could redefine these in terms of the variables for S , A , and each q_i variable, it is most convenient to use a summation of rate terms, i.e., define them by r_i .

To do this, we introduce variables for each standard components (A , S , q_1 , and q_2) in the [Variables](#) block, but also define variables for r_1 and r_2 . Then, in the Kernels block, we create residuals to reflect functions denoting the representation of those rates (as seen below). Once those rate variables are defined, we then can reference them in the [WeightedCoupledSumFunction](#) kernel and use that in conjunction with the [TimeDerivative](#) kernel to define the adsorption for q_1 and q_2 (as seen below).

```
[Kernels]
#r1 residual contributions
[./r1_val]
type = Reaction
```

```

        variable = r1
[../]
[./r1_rate]
    type = ConstReaction
    variable = r1
    this_variable = r1
    forward_rate = 1
    reactants = 'S A'
    reactant_stoich = '1 1'
    reverse_rate = 0.1
    products = 'q1'
    product_stoich = '1'
[../]

#r2 residual contributions
[./r2_val]
    type = Reaction
    variable = r2
[../]
[./r2_rate]
    type = ConstReaction
    variable = r2
    this_variable = r2
    forward_rate = 2
    reactants = 'q1 A'
    reactant_stoich = '1 1'
    reverse_rate = 0.01
    products = 'q2'
    product_stoich = '1'
[../]

#q1 residual contributions:  $dq_1/dt = r_1 - r_2$ 
[./q1_dot]
    type = TimeDerivative
    variable = q1
[../]
[./q1_rate_sum]
    type = WeightedCoupledSumFunction
    variable = q1
    coupled_list = 'r1 r2'
    weights = '1 -1'
[../]

#q2 residual contributions:  $dq_2/dt = r_2$ 

```

```

[./q2_dot]
    type = TimeDerivative
    variable = q2
[./]
[./q2_rate_sum]
    type = WeightedCoupledSumFunction
    variable = q2
    coupled_list = 'r2'
    weights = '1'
[./]

[./]

```

Check out [WeightedCoupledSumFunction](#) for more details on this method.

Example 2: Multi-Reaction Adsorption with Bulk Mass Balance Coupling

In the [previous example](#), we only demonstrated how to use reaction rate variables to evaluate some basic time derivative functions. However, the process will likely need to be coupled with the bulk phase variable 'A' since as these reactions occur they will pull 'A' from the bulk phase. While you can use the [WeightedCoupledSumFunction](#) as we did for 'q', you can use [ScaledWeightedCoupledSumFunction](#) instead, which gives you a bit more flexibility for cross-coupling between phases by introducing a scaling factor that is often used as a unit conversion from solid to bulk concentrations.

As a demonstration, let's consider that the disappearance of 'A' from the bulk phase due to the reactions from the [prior example](#) would take the following functional form:

$$\varepsilon_b \frac{\partial A}{\partial t} = (1 - \varepsilon_b) \sum_{\forall j} w_j r_j \implies \varepsilon_b \frac{\partial A}{\partial t} = (1 - \varepsilon_b)[-r_1 - r_2]$$

The weights (w_j) for each reaction would be negative for 'A' because 'A' is being consumed in each reaction. The 'scaling factor' for this example would be represented by $(1 - \varepsilon_b)$, which could be given as a constant or another non-linear variable. In this case, ε_b represents the pore-to-volume ratio and $(1 - \varepsilon_b)$ would represent the solids-to-volume ratio for the system.

Setting up this problem to solve would involve first creating the variables and kernels for r_1 and r_2 (see [previous example](#)), then creating the kernels for the bulk concentration of A using the [VariableCoefTimeDerivative](#) and [ScaledWeightedCoupledSumFunction](#) kernels (see below).

```

[Kernels]
#A residual contributions: 0.25*dA/dt = 0.75*(-r1 - r2)
[./A_dot]
    type = VariableCoefTimeDerivative
    variable = A
    coupled_coef = 0.25
[./]

```

```

[./A_rate_sum]
    type = ScaledWeightedCoupledSumFunction
    variable = A
    coupled_list = 'r1 r2'
    weights = '-1 -1'
    scale = 0.75
[../]

[../]

```

Checkout [ScaledWeightedCoupledSumFunction](#) for more details on this method.

Simulations with Subdomains

All of the examples above were created on meshes that were not divisible into subdomains. This is common for packed columns, since there are far too many particles to individually mesh within the global domain. However, for more structured domains such as a monolith catalyst, we may want to actually mesh the open channels of the monolith separate from the porous-washcoat materials. In this case, we will need to construct a mesh with a 3rd party meshing software first or using some mesh breaking functions in MOOSE. For simplicity, these examples here will use meshes created in Gmsh (<http://gmsh.info/>), an open source meshing program. In addition, we will only develop these examples for a mass balance, however, these instructions will transfer over to energy balances and only the specific kernels you invoke will be different.

In this example, we will investigate simulating the transfer of mass through a single monolith channel and the mass transfer and adsorption that occurs at the boundaries of the washcoat and in the washcoat itself. The formulation of the mass balance follows the [Separated Phase Mass Balances](#) equation derivations. For the concentration of mass in the channel (C), we track that concentration based on a standard advection-diffusion equation. Mass transfer (MT) into the washcoat is a function applied only at the physical interface that exists between the channel-space variable (C) and the washcoat-space variable (C_w). This function essentially replaces what was the [FilmMassTransfer](#) kernel from the [first mass balance example](#). Once mass transfers across that boundary, it can then travel through the washcoat via diffusion and can undergo adsorption (q). In this example, the units of adsorption are in moles adsorbed per volume of washcoat, thus there is no additional unit conversion term needed on the time derivative. The adsorption process itself will then follow the same Langmuir like reaction we specified in the [first example](#). For your particular problem, the units may vary and thus you would include some conversion factors.

$$C \text{ (in mol/m}^3\text{):} \quad \frac{\partial C}{\partial t} + \nabla \cdot (\mathbf{v} \cdot C) = \nabla \cdot (D \cdot \nabla C)$$

$$\text{MT @ interface:} \quad k(C - C_w)$$

$$C_w \text{ (in mol/m}^3\text{):} \quad \varepsilon_w \frac{\partial C_w}{\partial t} = \nabla \cdot (\varepsilon_w D_w \cdot \nabla C_w) - \frac{dq}{dt}$$

$$q \text{ (in mol/m}^3\text{):} \quad \frac{dq}{dt} = k_f C_p S - k_r q$$

$$S \text{ (in mol/m}^3\text{):} \quad S = S_{max} - q$$

The kernels we need to invoke for each piece of physics are as shown below. You can click on the links to get additional details on what each kernel is doing and what the input arguments are.

$$\text{VariableCoefTimeDerivative:} \quad \varepsilon_w \frac{\partial C_w}{\partial t}$$

$$\text{TimeDerivative:} \quad \frac{dq}{dt} \quad \text{and} \quad \frac{\partial C}{\partial t}$$

$$\text{CoupledCoeffTimeDerivative:} \quad -\frac{dq}{dt}$$

$$\text{GConcentrationAdvection (DGConcentrationAdvection):} \quad \nabla \cdot (\mathbf{v} \cdot C)$$

$$\text{GVariableDiffusion (DGVariableDiffusion):} \quad \nabla \cdot (D \cdot \nabla C)$$

$$\text{GVarPoreDiffusion (DGVarPoreDiffusion):} \quad \nabla \cdot (\varepsilon_w D_w \cdot \nabla C_w)$$

$$\text{ConstReaction:} \quad k_f C_p S - k_r q$$

$$\text{MaterialBalance:} \quad S = S_{max} - q$$

Just like before, we need to include boundary conditions for the mobile phases. For the channel space concentration (C), the boundary conditions are essentially the same as they were before, but without the porosity term. The boundary condition for the washcoat concentration (C_w) actually comes from the interface kernel to facilitate transfer from the channel to the washcoat. Note that whenever an interface kernel is invoked, it only needs to be invoked once and will be automatically applied to both domains that the interface acts on.

$$\text{DGConcentrationFluxBC:} \quad (\mathbf{v} \cdot \mathbf{n}) \cdot C_{inlet} \quad \text{and} \quad (\mathbf{v} \cdot \mathbf{n}) \cdot C$$

$$\text{InterfaceMassTransfer:} \quad k(C - C_w)$$

The mesh we use for this simulation is a 2D representation of a single monolith channel that has a gap in the middle between each layer of washcoat. This gap will help us simulate what would occur if the washcoat were not applied in a single, continuous layer of even thickness. The length of the channel is 5 cm and the total channel diameter is 0.127 cm with a washcoat layer thickness of about 0.027 cm. For your reference, both the .geo script file and .msh file for the mesh are provided (2DChannel.geo and 2DChannel.msh) in (https://github.com/aladshaw3/cats/tree/master/user_examples). We direct MOOSE/CATS to read in this object in the Mesh block as follows. Note that this mesh file identifies each subdomain as 'channel' and 'washcoat' and also identifies the boundaries as 'inlet', 'outlet', and 'inner_walls' for the interface between the washcoat and channel.

```
[Mesh]
  [./mesh_file]
    type = FileMeshGenerator
    file = 2DChannel.msh
  [../]
[]
```

Because our simulation involves subdomains, each variable and kernel must not identify the ‘block’ (i.e., subdomain) that that variable or kernel acts on. We will follow the same variable declarations from the [first example](#), but will now include a ‘block’ designation for each variable as follows.

```
[Variables]
  [./C]
    order = FIRST
    family = MONOMIAL
    initial_condition = 0
    block = 'channel'

  [../]
  [./Cw]
    order = FIRST
    family = MONOMIAL
    initial_condition = 0
    block = 'washcoat'

  [../]
  [./q]
    order = FIRST
    family = MONOMIAL
    initial_condition = 0
    block = 'washcoat'

  [../]
  [./S]
    order = FIRST
    family = MONOMIAL
    initial_condition = 0
    block = 'washcoat'

  [../]
[]
```

As with the variables block, any auxiliary variables we want to use as parameters must also declare what ‘block’ (i.e., subdomain) those auxiliary variables apply to, and the same goes for any auxiliary kernels that calculate the values for those auxiliary variables. In this example, we will use auxiliary variables to just hold parametric information and will not calculate those value as was done in the “[Adding Gas Properties](#)” example.

When invoking the physics/chemistry kernels for our variables, we again must include that ‘block’ (i.e., subdomain) designation for each kernel. Below, we show an example of the input syntax for doing so. For the full input file, see (https://github.com/aladshaw3/cats/tree/master/user_examples).

```
#Calling the material balance in the washcoat for S
```

```
[./mat_bal]
    type = MaterialBalance
    variable = S
    this_variable = S
    coupled_list = 'S q'
    weights = '1 1'
    total_material = S_max
    block = 'washcoat'

[../]
```

The boundary conditions and interface kernels do not need the 'block' argument set. This is because the 'boundary' argument is sufficient for instructing MOOSE when and where these kernels apply to the full domain. Interface kernels only need to be invoked once for each boundary/physics they apply to and they will automatically apply residuals for both variables that get coupled. Below shows an example of how to invoke the interface kernel for this simulation. For the full input file, see the example file in (https://github.com/aladshaw3/cats/tree/master/user_examples).

```
[InterfaceKernels]
    type = InterfaceMassTransfer
    variable = C
    neighbor_var = Cw
    boundary = 'inner_walls'
    transfer_rate = 2

[]
```

NOTE: In this simulation example, the tolerances for the solver option had to be significantly tightened and we changed to a direct linear solver (i.e., 'solve_type = newton'). This was done to improve convergence for a very fine mesh with very small elements. The mesh used here was done in units of meters, but given the size of the channel, it may be more appropriate to use centimeters. If you choose to change unit basis on the mesh, make sure all your parameters align with the new units.

Simulations with Hybrid FD/FE Microscale Diffusion

CATS has developed within it a hybrid Finite-Difference/Finite-Element method for simulating intraparticle or intralayer diffusion of a microscale and couple those physics fully with the macroscale physics. The advantage of using this approach over using sub-domains is to simplify the mesh in the model and reduce problem dimensionality. Doing so will lead to a reduction in computation resources needed to resolve the full problem, without making significant concessions in model applicability. To see details on the microscale diffusion kernels developed in CATS, check out all kernels prefixed with 'Microscale' starting from [here](#).

Packed-bed Adsorption

The hybrid FD/FE method was originally developed for resolving the microscale diffusion physics for columns packed with particles. This is because a domain such as that cannot actually use a sub-domain approach to resolve the microscale (at least not in any efficient way). Thus, if we discretize the microscale using finite differences, then formulate a “weak form” of that discretization on the macroscale grid, we can then still solve for intraparticle diffusion in a column packed with thousands or millions of particles.

This example seeks to simulate the adsorption and desorption behavior of water vapor on spherical zeolite pellets. The equations we will solve in this example are as follows:

Macroscale: $\varepsilon_b \frac{\partial C_{H_2O}}{\partial t} + \nabla \cdot (\varepsilon_b \mathbf{v} \cdot C_{H_2O}) = \nabla \cdot (\varepsilon_b D \cdot \nabla C_{H_2O}) - (1 - \varepsilon_b) \cdot G_a \cdot k_m (C_{H_2O} - C_{H_2O,p})$

Microscale: $r^2 \varepsilon_p (1 - \varepsilon_b) \frac{\partial C_{H_2O,p}}{\partial t} = \frac{\partial}{\partial r} \left(r^2 \varepsilon_p (1 - \varepsilon_b) D_p \frac{\partial C_{H_2O,p}}{\partial r} \right) + r^2 (1 - \varepsilon_b) \cdot (-1) \cdot r_{H_2O}$

Adsorption Reaction: $H_2O + S \leftrightarrow q \quad r_{H_2O} = k_f C_{H_2O,p} S - k_r q$

Microscale Mass Transfer BC: $\varepsilon_p (1 - \varepsilon_b) D_p \frac{\partial C_{H_2O,p}}{\partial r} = k_m (C_{H_2O} - C_{H_2O,p})$

Microscale Interior BC: $\frac{\partial C_{H_2O,p}}{\partial r} = 0$

Surface ODEs: $\frac{dS}{dt} = -r_{H_2O} \quad \frac{dq}{dt} = r_{H_2O}$

Both the macroscale mass balance and microscale mass balance are on a per total volume basis, that is what all the scale factors with the various ε_i values are doing. The reaction is introduced via a non-linear variable (r_{H_2O}), which can be applied to the microscale balance and equations for available sites (S) and adsorbed water (q). This makes the coupling simpler from the user’s perspective. All of the parameters for this example will be automatically calculated using the [SimpleGasProperties](#) interface through a series of auxiliary kernels. Temperature will be assumed isothermal for simplicity, but we include a linear temperature ramp to create a Temperature Programmed Desorption (TPD) curve towards the end of the simulation.

The kernels we will be invoking are as follows:

[VariableCoefTimeDerivative:](#) $\varepsilon_b \frac{\partial C_{H_2O}}{\partial t}$

[GPoreConcAdvection \(DGPoreConcAdvection\):](#) $\nabla \cdot (\varepsilon_b \mathbf{v} \cdot C_{H_2O})$

[GVarPoreDiffusion \(DGVarPoreDiffusion\):](#) $\nabla \cdot (\varepsilon_b D \cdot \nabla C_{H_2O})$

[FilmMassTransfer:](#) $(1 - \varepsilon_b) \cdot G_a \cdot k_m (C_{H_2O} - C_{H_2O,p})$

[MicroscaleVariableCoefTimeDerivative:](#) $r^2 \varepsilon_p (1 - \varepsilon_b) \frac{\partial C_{H_2O,p}}{\partial t}$

[MicroscaleVariableDiffusion:](#) $\frac{\partial}{\partial r} \left(r^2 \varepsilon_p (1 - \varepsilon_b) D_p \frac{\partial C_{H_2O,p}}{\partial r} \right)$

MicroscaleScaledWeightedCoupledSumFunction:	$r^2(1 - \varepsilon_b) \cdot (-1) \cdot r_{H_2O}$
ArrheniusEquilibriumReaction (with Reaction):	$r_{H_2O} = k_f C_{H_2O,p} S - k_r q$
MicroscaleVariableDiffusionOuterBC:	$\varepsilon_p(1 - \varepsilon_b) D_p \frac{\partial C_{H_2O,p}}{\partial r} = k_m (C_{H_2O} - C_{H_2O,p})$
MicroscaleVariableDiffusionInnerBC:	$\frac{\partial C_{H_2O,p}}{\partial r} = 0$

WeightedCoupledSumFunction (with TimeDerivative):	$\frac{dS}{dt} = -r_{H_2O}$	$\frac{dq}{dt} = r_{H_2O}$
--	-----------------------------	----------------------------

In order to resolve the microscale domain, for this example we will divide the microscale into 5 nodes (or 5 discrete interior pellet volumes). We will denote those nodes as (0, 1, 2, 3, 4), where 0 represents the inner most position (i.e., center of the particle) and 4 represents the outer most position (i.e., where particle is in contact with bulk gases). When we do this, we must also create variable IDs for each interior variable at each position. Thus, instead of adding a variable for interior pellet concentration as 'H2Op', we create a series of interior concentration variables as 'H2Op_0', 'H2Op_1', 'H2Op_2', 'H2Op_3' and 'H2Op_4'. The same is applied for each reaction variable (r_0 through r_4) and each surface species (S_0 through S_4 and q_0 through q_4).

Kernels for the microscale physics must be repeated for each nodal position. The only difference will be at the BC nodes of the microscale, which replace the [MicroscaleVariableDiffusion](#) kernel with the [MicroscaleVariableDiffusionOuterBC](#) kernel (for node 4) and the [MicroscaleVariableDiffusionInnerBC](#) kernel (for node 0). This process is somewhat tedious, but is necessary to create the full “weak form” of the microscale on the macroscale mesh domain. To see details on the microscale diffusion kernels developed in CATS, check out all kernels prefixed with 'Microscale' starting from [here](#).

The parameters for each coefficient in this model will be automatically calculated using the following auxiliary kernels:

VoidsVolumeFraction:	ε_b
GasVelocityCylindricalReactor:	$\mathbf{v} \rightarrow$ here, we are only calculating the y-component of velocity to get an average linear velocity in the packed column.
SphericalAreaVolumeRatio:	G_a
SolidsVolumeFraction:	$(1 - \varepsilon_b)$
MicroscalePoreVolumePerTotalVolume:	$\varepsilon_p(1 - \varepsilon_b)$
SimpleGasSphericalMassTransCoef:	k_m
SimpleGasEffectivePoreDiffusivity:	$\varepsilon_p(1 - \varepsilon_b) D_p$
SimpleGasDispersion:	D

Additionally, there are a number of other auxiliary kernels we are using:

LinearChangeInTime: To automatically raise temperature linearly as a function of time, give some starting time, ending time, and temperature target. This is used to emulate a TPD.

MicroscaleIntegralAvg: To automatically calculate average interior concentrations of $C_{H_2O,p}$, S , and q . This will integrate over the values of the individual nodal variables in the microscale domain and produce average amounts of adsorption, free sites, or other variables in that space of interest.

A full run down of the input file to run these simulations, as well as some develop notes and comments, can be found under the 'user_examples' folder filename 'simulations_with_spherical_microscale_diffusion.i'

(https://github.com/aladshaw3/cats/blob/master/user_examples/simulations_with_spherical_microscale_diffusion.i).

Packed-bed Adsorption (same problem, without Microscale)

This example shows how to setup a simulation to model the same problem as the example above, but not include the microscale physics inside the particles. The purpose of this example is to show how much of a difference may be when you include microscale diffusion or not. In this particular case, the inclusion of microscale diffusion does not have a significant impact. Whether or not it is needed depends on how small the diffusion coefficient is relative to the particle sizes.

Even if we do not explicitly discretize the microscale, we can still have variables represent what is happening in the microscale, we just won't be able to model diffusion through that microscale. Instead, all microscale variables will be representative of the average levels found in that subdomain. Thus, our mathematical description will be as follows:

Macroscale:
$$\epsilon_b \frac{\partial C_{H_2O}}{\partial t} + \nabla \cdot (\epsilon_b \mathbf{v} \cdot C_{H_2O}) = \nabla \cdot (\epsilon_b D \cdot \nabla C_{H_2O}) - (1 - \epsilon_b) \cdot G_a \cdot k_m (C_{H_2O} - C_{H_2O,p})$$

Microscale:
$$\epsilon_p (1 - \epsilon_b) \frac{\partial C_{H_2O,p}}{\partial t} = (1 - \epsilon_b) \cdot G_a \cdot k_m (C_{H_2O} - C_{H_2O,p}) + (1 - \epsilon_b) \cdot (-1) \cdot r_{H_2O}$$

Adsorption Reaction:
$$H_2O + S \leftrightarrow q \quad r_{H_2O} = k_f C_{H_2O,p} S - k_r q$$

Surface ODEs:
$$\frac{dS}{dt} = -r_{H_2O} \quad \frac{dq}{dt} = r_{H_2O}$$

As you can see, the only difference is that the intraparticle diffusion (as associated outer boundary condition) is essentially just replaced with the mass-transfer statement (same statement from the macroscale balance). Since we have already gone through all the kernels in the previous example, we will not do a detailed run down here. A full run down of the input file to run these simulations, as well as some develop notes and comments, can be found under the 'user_examples' folder filename 'equivalent_spherical_particle_adsorption_without_microscale.i'

(https://github.com/aladshaw3/cats/blob/master/user_examples/equivalent_spherical_particle_adsorption_without_microscale.i).

Monolith CO Light-off

In addition to using the hybrid FD/FE method for resolving microscale diffusion in particles, you can use this method to also simulate the microscale diffusion inside of monolith walls/washcoating. This can be accomplished in the same way it was done in the [example above](#). However, in a monolith, we have diffusion through a wall and there may be many walls within a given channel that we diffuse through. For each wall you have, you will need to identify new microscale variables for that wall. For instance, if you have square channels, then you would need a variable that represents the concentration of a species in each wall at a given nodal position (e.g., variable C_{w0_n4} could be a designation for concentration in wall 0 at nodal position 4 of the microscale). See [next example](#) for details.

This would be even more tedious than the prior example, but is a way to also explore the possibility of different coatings on different walls in a given channel. Instead of showing an example of this, we will show a simplified example of microscale diffusion in washcoated monoliths using a calculation of an 'effective total monolith thickness' for the microscale. More information on how this thickness is calculated can be found here ([MonolithMicroscaleTotalThickness](#)). Note that using this thickness will only yield an approximate intralayer diffusion effect. It is more accurate to use a true thickness and a set of microscale physics for each wall. This can be done with this same modeling paradigm simply by expanding the microscale variables to include wall IDs and node IDs, duplicating the microscale kernels for each wall, and providing true wall/washcoat thickness.

The physics for this problem are represented mathematically as follows:

Macroscale: $\varepsilon_b \frac{\partial C_i}{\partial t} + \nabla \cdot (\varepsilon_b \mathbf{v} \cdot C_i) = \nabla \cdot (\varepsilon_b D \cdot \nabla C_i) - (1 - \varepsilon_b) \cdot G_a \cdot k_m (C_i - C_{w,i})$

Microscale: $\varepsilon_w (1 - \varepsilon_b) \frac{\partial C_{w,i}}{\partial t} = \frac{\partial}{\partial r} \left(\varepsilon_w (1 - \varepsilon_b) D_p \frac{\partial C_{w,i}}{\partial r} \right) + (1 - \varepsilon_b) \cdot (u_i) \cdot r_{lightoff}$

Lightoff Reaction: $\text{CO} + 0.5 \text{O}_2 \rightarrow \text{CO}_2$ $r_{lightoff} = k_f C_{w,\text{CO}} C_{w,\text{O}_2}$

Microscale Mass Transfer BC: $\varepsilon_w (1 - \varepsilon_b) D_p \frac{\partial C_{w,i}}{\partial r} = k_m (C_i - C_{w,i})$

Microscale Interior BC: $\frac{\partial C_{w,i}}{\partial r} = 0$

In this formulation, each index i represents either CO, O₂, or CO₂. The parameter u_i is the molar impact that the lightoff reaction would have on a given gas species. This value is -1 for $i = \text{CO}$, -0.5 for $i = \text{O}_2$, and +1 for $i = \text{CO}_2$.

This example will use all the same kernels from the [previous example](#), so we will not repeat that information here. Just like before, all the parameters are calculated using the [SimpleGasProperties](#) auxiliary system. Most of those calculations are the same as the prior example, so we will just summarize the ones below that are different.

[MonolithAreaVolumeRatio:](#) G_a

[MonolithHydraulicDiameter:](#) $d_h \rightarrow$ not directly used in the above kernels, but is used in the calculation of other auxiliary properties.

[SimpleGasMonolithMassTransCoef:](#) k_m

MonolithMicroscaleTotalThickness: This for the calculation of the ‘effective total thickness’ for the microscale dimension of the monolith. This should be calculated ahead of time and its value should be used as the ‘micro_length’ parameter for the microscale kernels.

Additionally, in this example we use the auxiliary system to setup the lightoff temperatures based on a set of data (rather than simulating those temperatures for simplicity). We can do this by making use of the ‘PiecewiseMultilinear’ function built into MOOSE (see <https://mooseframework.inl.gov/source/functions/PiecewiseMultilinear.html>). Combined with the auxiliary kernel ‘FunctionAux’ these features will take a data file with time-position temperature information and setup a simulation that contains those exact temperature gradients (with some linear interpolations for time-position data that are missing).

A full run down of the input file to run these simulations, as well as some develop notes and comments, can be found under the ‘user_examples’ folder filename ‘simulations_with_monolith_microscale_diffusion.i’ (https://github.com/aladshaw3/cats/blob/master/user_examples/simulations_with_monolith_microscale_diffusion.i).

Monolith CO Light-off with True Wall Thickness

This example does the same calculations from [above](#), but separates out the diffusion into the individual walls of the monolith channel (denoted with *_w0, *_w1, *_w2, and *_w3 for each wall). Since this example is solving the same problem, we will not go through everything here. Instead, a full run down of the input file to run these simulations, as well as some develop notes and comments, can be found under the ‘user_examples’ folder filename ‘simulations_with_monolith_microscale_diffusion_multiwalls.i’ (https://github.com/aladshaw3/cats/blob/master/user_examples/simulations_with_monolith_microscale_diffusion_multiwalls.i).

Doing Everything: Fully Coupled Mass-Energy with Microscale Physics

In this final example, we use the hybrid Finite-Difference/Finite-Element methods for simulating intraparticle diffusion in packed columns and build upon all prior examples to simulate a more realistic and relevant simulation case.

For this example, we will simulate the coupling of mass and energy in a packed cylindrical column with particles containing ‘coke’ (i.e., unburned carbon). Then, we introduce oxygen, in a nitrogen carrier gas, into the column to oxidize the carbon inside of the particles creating CO₂ and a significant increase in system heat due to the exothermic nature of the reaction. The mass and energy balances for the macroscale are as follows:

$$\begin{aligned} \text{O}_2: \quad \varepsilon \frac{\partial C_{\text{O}_2}}{\partial t} + \nabla \cdot (\varepsilon \mathbf{v} \cdot C_{\text{O}_2}) &= \nabla \cdot (\varepsilon D_{e,\text{O}_2} \cdot \nabla C_{\text{O}_2}) - (1 - \varepsilon) \cdot k_{\text{O}_2} \cdot A_o (C_{\text{O}_2} - C_{p,\text{O}_2}) \\ \text{CO}_2: \quad \varepsilon \frac{\partial C_{\text{CO}_2}}{\partial t} + \nabla \cdot (\varepsilon \mathbf{v} \cdot C_{\text{CO}_2}) &= \nabla \cdot (\varepsilon D_{e,\text{CO}_2} \cdot \nabla C_{\text{CO}_2}) - (1 - \varepsilon) \cdot k_{\text{CO}_2} \cdot A_o (C_{\text{CO}_2} - C_{p,\text{CO}_2}) \end{aligned}$$

$$E_f: \quad \varepsilon \frac{\partial E_f}{\partial t} + \nabla \cdot (\varepsilon \mathbf{v} \cdot E_f) = \nabla \cdot (\varepsilon K_f \cdot \nabla T_f) - (1 - \varepsilon) \cdot h_s \cdot A_o (T_f - T_s)$$

$$E_s: \quad (1 - \varepsilon) \frac{\partial E_s}{\partial t} = \nabla \cdot ((1 - \varepsilon) \cdot K_s \cdot \nabla T_s) - (1 - \varepsilon) \cdot h_s \cdot A_o (T_s - T_f) \\ + (1 - \varepsilon) \cdot \sum_{\forall j} (-\Delta H_j) A_{s,j} r_{s,j}$$

$$T_f: \quad E_f = \rho_f c_{pf} T_f$$

$$T_s: \quad E_s = \rho_p c_{ps} T_s$$

Since nitrogen gas is assumed ‘inert’ as the carrier gas, we do not include physics for this species and instead just treat it as an auxiliary variable held to a constant value. All the parameters for this model are resolved using the auxiliary system. Since our focus here is on the microscale physics, we do not discuss all aspects of the variable, kernel, and auxiliary blocks of the input file. Check out the full input file at (https://github.com/aladshaw3/cats/tree/master/user_examples) for additional details.

Many of the above equations should be familiar right now, but let’s take a moment to discuss some of the key unique features. Each energy balance is done in terms of energy per total volume, which is why each has a different unit conversion factor associated with it to convert energy per phase volume to energy per total volume. The energy balance for the solids (E_s) involves a summation over a set of surface reactions ($r_{s,j}$) that take place at specific locations (j) within the particles. Since this is a surface based reaction, the factor $A_{s,j}$ represents the approximate reactive surface area at location j inside the particles. Thus, the sum of all representative areas $A_{s,j}$ would yield the total reactive surface area of the particles.

The reaction rate ($r_{s,j}$) then represents the actual carbon oxidation reaction that occurs at the j location inside of a particle. We would represent this reaction as shown below:



Where $q_{c,j}$ represents the surface concentration of active carbon at the j location in the particle and $C_{p,\text{O}_2,j}$ represents the pore-space concentration of oxygen at the j location in the particle. Note that here we represent this reaction as reversible, so the pore-space concentration of carbon-dioxide ($C_{p,\text{CO}_2,j}$) does not show up in the rate expression. However, the formation of carbon-dioxide will be a function of the above rate expression.

To resolve the intraparticle diffusion of oxygen and carbon-dioxide, we need to introduce 3 more mass balances that all act on the microscale domain of the individual particles. Those base equations would be as follows:

$$C_{p,\text{O}_2}: \quad \varepsilon_p \frac{\partial C_{p,\text{O}_2}}{\partial t} = \nabla \cdot (\varepsilon_p D_{p,\text{O}_2} \cdot \nabla C_{p,\text{O}_2}) - A_s r_s$$

$$C_{p,\text{CO}_2}: \quad \varepsilon_p \frac{\partial C_{p,\text{CO}_2}}{\partial t} = \nabla \cdot (\varepsilon_p D_{p,\text{CO}_2} \cdot \nabla C_{p,\text{CO}_2}) + A_s r_s$$

$$q_c: \quad \frac{dq_c}{dt} = -r_s$$

These equations are written in their respective “strong forms” and as such they are not currently discretized into the j intraparticle positions, as we showed for the energy balance. The r_s term is the same rate expression that we showed earlier, but without the j subscripts. For details on how these equations get transformed and semi-discretized into the “weak forms”, take a look at the ‘Microscale’ kernels starting [here](#).

In addition to the above expressions, the intraparticle mobile phase also requires boundary conditions. These boundary conditions are how we couple together the macro- and micro-scales of the simulation. Currently, CATS only offers a [Cauchy type boundary condition](#) for this coupling. That coupled expression, in its strong form, would be as follows:

$$C_{p,O_2}: \quad \varepsilon_p D_{p,O_2} \frac{\partial C_{p,O_2}}{\partial r} = k_{O_2} (C_{O_2} - C_{p,O_2})$$

$$C_{p,CO_2}: \quad \varepsilon_p D_{p,CO_2} \frac{\partial C_{p,CO_2}}{\partial r} = k_{CO_2} (C_{CO_2} - C_{p,CO_2})$$

The interior boundary condition enforces a “no flux” or “r-symmetric” condition for the interior most location of the particles (i.e., at the 0 radius of the particles). More information on this type of boundary condition can be found [here](#).

For this example, the microscale will be discretized into 10 j locations. Each j location needs its own variable for q_c , C_{p,O_2} , and C_{p,CO_2} . Typically, we will identify each j location variable by suffixing a variable name with an id number. For instance, in this example the variable named ‘qc0’ represents the surface carbon concentration at the 0th location in the particles, where the 0th location is at a radius of 0 (i.e., the center of the particle). Then, the outer most location of the particle will be denoted as ‘qc9’ since there are only 10 locations (nodes 0 through 9) we divide the microscale space into. Setting up the input file for these types of simulations is very tedious and long, so we will not write down all the details here. Additional instructions on how to structure input files for microscale simulations can be found in the ‘Microscale’ kernels section starting [here](#). Additionally, the input file containing all these kernels and expressions is available at (https://github.com/aladshaw3/cats/tree/master/user_examples).

Lastly, for convenience, it will be advantageous to introduce auxiliary variables to represent the average pore-space concentrations at the microscale. Since we have divided up the microscale into 3 sets of 10 separate variables, it will be difficult to interpret exactly how much material is inside of the particles and/or the extent of the reactions inside the particles. To facilitate the need to interpret this information, provided in CATS are auxiliary kernels for ‘[MicroscaleIntegralAvg](#)’ and ‘[MicroscaleIntegralTotal](#)’. These kernels perform an integration over the microscale domain to average or total the amount of material inside of the particles. The integration uses a trapezoid rule for accuracy, but these integrals will not be exact. As a consequence, it will not be uncommon to see some small mass errors (0.5 - 2 %) if you are looking to account for all mass in the system.

As a final note, we did not go over line-by-line each argument in the input file for this example due to how large this case is (the input file requires over 2000 lines of arguments). Instead, you should use the prior, simpler examples to build from first. After you become more familiar with the input file system and how to call specific kernels for specific purposes, then understanding this input file and simulation case should be a bit easier. Be sure to read through the ‘Microscale’ kernels ([here](#)) if you get lost on why certain input arguments are being invoked and check out the full input file for this simulation case at (https://github.com/aladshaw3/cats/tree/master/user_examples). Throughout that

input file are multiple developer comments which discuss what the kernels and input arguments are doing and why they are there.

Running the Code

CATS requires the MOOSE framework to function. Before attempting to run simulations in CATS, you must download and install MOOSE. Follow the instructions on the MOOSE website to get started (<https://mooseframework.org/>). After you have installed MOOSE, you can use the 'git' commands to download and install the CATS module. The commands below need to be run in the same directory that the 'moose' folder is located in.

```
git clone https://github.com/aladshaw3/cats.git
cd cats/
make -j4
```

The 'make -j4' command will instruct your computer to build the CATS executable from the source files using 4 cores. After these instructions complete, and assuming no errors occurred, you should now see an executable called 'cats-opt' in the 'cats/' directory. This is the program you will invoke to run simulations. However, before get started creating and running simulations, you should test the CATS build on your machine to ensure everything is working as intended. The next section discusses how to run the unit tests.

Unit Tests

Unit tests are small scale simulations run on individual kernels or sets of kernels. Each test case has a 'gold' set of output files that will be used for comparison. In general, after you install the CATS executable, you should always run the unit tests and check for any errors. To run the unit tests, use the following command from within the 'cats/' directory.

```
./run_tests -j4
```

If there are no errors, then all tests will pass and show a green 'OK' or 'PASSED' message. Note that some tests may be skipped depending on specific test or computer conditions. This is normal. When the tests fail, there are 2 modes of failure: (i) yellow 'EXODIFF' failure and (ii) red 'CRASH' failure. The 'EXODIFF' message occurs when the test ran to completion, but the end results of the simulation were not within a specified level of tolerance when compared to the 'gold' solution. This can sometimes occur if your PETSc version is different than the version that was originally used to create the test. In general, if these errors occur, you either need to update the 'gold' files or update MOOSE/PETSc.

If you get a red 'CRASH' error, or something similar, then this is a situation in which the code could not run at all. When this happens, it could mean (i) you are missing MOOSE libraries, (ii) you are missing PETSc or libmesh libraries, and/or (iii) there is an underlying problem in some CATS kernels (this would be rare if you are on the 'master' branch. In general, should you encounter this problem, you should record your console output detailing all the error messages and report that to a CATS developer for debugging. You can report problems on the GitHub CATS page (<https://github.com/aladshaw3/cats>) or by emailing a developer list on the 'README' file.

Single Core Command Line

To run a simulation in CATS uses the command line. The basic structure for calling a simulation is as follows:

```
./cats-opt -i path/to/input.i
```

The above command calls the 'cats-opt' executable, denotes that you are giving an input file with the '-i' designation, then provides the input file to run along with the path to that file from the current directory.

As an additional argument, you can also suffix the above command with option '-log_view' that will print out some information after a completed run that can be used to see the statistics of the solvers (i.e., amount of time spent in each solver, percent of time in PETSc vs MOOSE, number of CPUs and flops per CPU, etc). This can be vital information when performing diagnostics checks. To invoke this option, your command line argument would be as follows:

```
./cats-opt -i path/to/input.i -log_view
```

Multi-Core Command Line

One of the major advantages of the MOOSE modeling framework is how easy it is to run simulations with multiple CPUs. MOOSE uses the Message Passing Interface (MPI) to distribute memory and operations among the cores of your computer. To use multiple cores to run a simulation, you simply prefix your command line argument with 'mpiexec --n 4' where the number 4 would represent the number of CPU cores to use. See the example below.

```
mpiexec --n 4 ./cats-opt -i path/to/input.i
```

Just like with single core simulations, you can invoke '-log_view' to get a diagnostics report for the multicore simulations. See the example below.

```
mpiexec --n 4 ./cats-opt -i path/to/input.i -log_view
```

To Do List

This section of the user guide is dedicated to items that are not yet developed and plan to be developed. It keeps a running list of things in the code base that needs changing or updating. Assume that any item discussed and/or listed here is not yet fully functional.

Specific Kernels

High Advection Stabilization

In recent testing, results have shown that even using DG methods that some oscillations in the solution wave front is still apparent under extreme circumstances, such as convergent fronts or fast-moving upwind fronts with slow moving downwind fronts. Thus, it may be necessary to investigate additional kernels that can be invoked to add stabilization for extreme cases. Such stabilization will need to be done by either (i) invoking a form of slope limiter or (ii) adding dispersion in those areas for numerical stabilization.

Adding Capabilities for Darcy Flow or Porous Flow

In CATS, all the transport kernels are developed such that they can couple with variables for velocity. As such, they are valid for any given flow field, however, CATS generally does not seek to solve for the flow field itself, and instead relies on a flow field being given up front or calculated through other MOOSE modules. However, there are now a base set of kernels ([VectorCoupledGradient](#)) in CATS that allow a user to define a flow field via Darcy Law (or purely laminar flow). These need to be expanded on. User examples that demonstrate this can be found in the 'user_examples' folder of the repository.

Finite Volumes CFD

The MOOSE framework has an experimental version of the Navier-Stokes equations using a finite volumes method built out of DG kernels. This will prove very useful for our purposes, thus, there is a desire to build our tests and examples around this new MOOSE framework capability. However, at current, this is low on the priorities list for this project.

Setting BCs based on ppm values

As a matter of convenience, setting input boundary values based on ppm values, instead of actual concentration units, would be very beneficial to simulations of light-off and light-down of catalysts. Currently, only one such BC is available ([DGPoreConcFluxBC_ppm](#)). Each other BC needs an equivalent ppm conversion kernel for setting BCs.

Kernels

Kernels are the primary physics/chemistry portions of the CATS model framework. Each kernel provides a single residual to the overall problem you are seeking to solve. Simply invoke new kernels to add a single piece of physics or chemistry to your simulation. The best approach to developing your specific simulation is to first write out the strong form of the physics, then move all pieces of physics to the same side of the equation as your variable's time derivative (if there is one), then multiply through by a test function (φ). Search through the Residual Formulations below to see if that piece of physics exists in the framework and how to invoke it.

For example, let's say you want to simulate a simple advection diffusion problem. Mathematically, that would be represented by:

$$\text{"Strong Form"} \quad \frac{\partial u}{\partial t} + \nabla \cdot (v \cdot u) = \nabla \cdot (D \cdot \nabla u)$$

Move all physics to the left-hand side and multiply by φ .

$$\text{"Weak Form"} \quad \varphi \frac{\partial u}{\partial t} + \varphi \nabla \cdot (v \cdot u) - \varphi \nabla \cdot (D \cdot \nabla u) = 0$$

Search through the kernels listed below to find residual contributions for each piece of physics:

$$\text{TimeDerivative} \rightarrow \varphi \frac{\partial u}{\partial t}$$

$$\text{GAdvection (DGAdvection)} \rightarrow \varphi \nabla \cdot (v \cdot u)$$

$$\text{GAnisotropicDiffusion (DGAnisotropicDiffusion)} \rightarrow -\varphi \nabla \cdot (D \cdot \nabla u)$$

ArrheniusEquilibriumReaction

Inheritance → [ArrheniusReaction](#)

Notes → This kernel uses existing residuals from [ArrheniusReaction](#), but calculates the rate terms for the reverse half of the reaction based on the enthalpies (ΔH) and entropies (ΔS) of the reaction. User must still provide the forward activation energy (E_f) and forward pre-exponential term (A_f).

Residual Formulation

(See [ArrheniusReaction](#) for residual formulation)

Additional Computations

$$k_f = A_f T^{\beta_f} \exp \left[-\frac{E_f}{RT} \right] \quad k_r = A_r T^{\beta_r} \exp \left[-\frac{E_r}{RT} \right]$$

- Both β are assumed 0 for this kernel.
- All reverse components are calculated from forward components with ΔH & ΔS

$$E_r = E_f - \Delta H \quad A_r = A_f \cdot \exp \left\{ -\frac{\Delta S}{R} \right\}$$

ΔH = reaction enthalpy (J/mol)

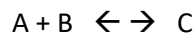
ΔS = reaction entropy (J/K/mol)

- The units for A_f and A_r must have the same units as k_f and k_r
- The units for E_f and E_r must be J/mol (same as ΔH)

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Code below creates a residual for species A for the following reaction:



```
[./rxn]
  type = ArrheniusEquilibriumReaction
  variable = A
  this_variable = A
  temperature = T
  scale = -1
  forward_activation_energy = 5E4
  forward_pre_exponential = 25
  enthalpy = -3.9E5
  entropy = 0
  reactants = 'A B'
  reactant_stoich = '1 1'
  products = 'C'
  product_stoich = '1'
[../]
```

ArrheniusEquilibriumReactionEnergyTransfer

Inheritance → [ArrheniusReactionEnergyTransfer](#)

Notes → This kernel using existing residuals from [ArrheniusReactionEnergyTransfer](#), but calculates the rate terms for the reverse half of the reaction based on the enthalpies (ΔH) and entropies (ΔS) of the reaction. User must still provide the forward activation energy (E_f) and forward pre-exponential term (A_f).

Residual Formulation

(See [ArrheniusReactionEnergyTransfer](#) for residual formulation)

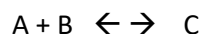
Additional Computations

The forward and reverse rate components are calculated the same way as [ArrheniusEquilibriumReaction](#). See that kernel for details.

Usage

Generally used in conjunction with an energy balance set of kernels to add the physics of energy changes caused by a chemical reaction in the system. The usage also includes terms to convert the volumetric or per reactive area terms into the proper energy density residual.

Code below creates a residual for energy exchange via the following surface reaction:



[./rxn]

```
type = ArrheniusEquilibriumReactionEnergyTransfer
variable = E          # E = Energy density variable
this_variable = E
temperature = T
volume_frac = s_frac # Volume of solids per Total Volume
specific_area = 8.6E7 # Reaction area of solid per volume of solid
forward_activation_energy = 5E4
forward_pre_exponential = 25
enthalpy = -3.9E5
entropy = 0
reactants = 'A B'
reactant_stoich = '1 1'
products = 'C'
product_stoich = '1'
```

[../]

ArrheniusReaction

Inheritance → [ConstReaction](#)

Notes → This kernel has all the same residuals from [ConstReaction](#), but calculates the forward and reverse rates of that reaction based on the activation energies (E), pre-exponential factors (A), and beta terms (β) for both the forward and/or reverse halves of the reaction. All of those terms are optional. User only needs to provide the terms that the kernel actually needs.

Residual Formulation

(See [ConstReaction](#) for residual formulation)

Additional Computations

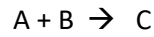
$$k_f = A_f T^{\beta_f} \exp\left[-\frac{E_f}{RT}\right] \quad k_r = A_r T^{\beta_r} \exp\left[-\frac{E_r}{RT}\right]$$

- The units for A_f and A_r must have the same units as k_f and k_r
- The units for E_f and E_r must be J/mol

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Code below creates a residual for species A for the following reaction:



```
[./rxn]
  type = ArrheniusReaction
  variable = A
  this_variable = A
  temperature = T
  scale = -1
  forward_activation_energy = 5E4
  forward_pre_exponential = 25
  reactants = 'A B'
  reactant_stoich = '1 1'
  products = 'C'
  product_stoich = '1'
[../]
```

ArrheniusReactionEnergyTransfer

Inheritance → [ArrheniusReaction](#)

Notes → Although this kernel is used in energy balances and not mass balances, the kernels are mathematically very similar. Thus, this kernel uses the residual computed from [ArrheniusReaction](#), but applies the enthalpy (ΔH) and 2 unit conversion factors to modify the scaling parameter of [ArrheniusReaction](#). The net effect is to convert the existing reaction rate function from a mass term to an energy term.

Residual Formulation

The baseline residual comes from [ArrheniusReaction](#). In this formulation, we will represent that portion of the residual as r (i.e., the rate of the reaction) with a scaling parameter of 1. See [ArrheniusReaction](#) for more details on scaling parameters. Then, this kernel uses that reaction residual (r) and applies a custom scaling based on ΔH and other unit conversions.

$$\text{Residual (J/m}^3\text{/s)} = -\Delta H \cdot f_v \cdot A_s \cdot r$$

f_v = volume fraction (i.e., volume of solids per total volume)

→ Can be set to 1 if no such conversion is needed

A_s = Specific reaction area per volume of solids (for a surface reaction)

→ Can be set to 1 if no such conversion is needed

ΔH = enthalpy of the reaction (J/mol)

r = reaction rate (mol/m³/s) or (mol/m²/s) [depends on reaction type]

→ Use f_v and A_s to convert units such that the units for the residual return as energy rate per total volume (J/m³/s)

Additional Computations

As noted above, the scaling parameter from the reaction is overridden to always be 1.

Usage

Generally used in conjunction with an energy balance set of kernels to add the physics of energy changes caused by a chemical reaction in the system. The usage also includes terms to convert the volumetric or per reactive area terms into the proper energy density residual.

Code below creates a residual for energy exchange via the following surface reaction:



[./rxn]

```
type = ArrheniusReactionEnergyTransfer
variable = E          # E = Energy density variable
this_variable = E
temperature = T
volume_frac = s_frac  # Volume of solids per Total Volume
specific_area = 8.6E7  # Reaction area of solid per volume of solid
forward_activation_energy = 5E4
forward_pre_exponential = 25
enthalpy = -3.9E5
reactants = 'A B'
reactant_stoich = '1 1'
products = 'C'
product_stoich = '1'
```

[./]

ConstMassTransfer

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel can be used to create a generic exchange of mass (or energy) between 2 different non-linear variables (which may represent two phases of matter). For instance, a common usage would be to add the physics of mass transfer from a liquid phase to a gaseous phase inside bubbles in a CO₂ absorption column.

Residual Formulation

$$\text{Residual (mass / volume / time)} = \varphi \cdot k \cdot (u - v)$$

φ = MOOSE variable test function (for FE formulation)

k = transfer rate (time⁻¹)

u = this object's non-linear variable (mass / volume)

v = a coupled non-linear variable (mass / volume)

Additional Computations → None

Usage

Generally used in conjunction with other physics or chemistry, this kernel facilitates the coupling of 2 different variables that exchange mass with each other at a given rate at all locations in the domain of interest.

```
[./mass_trans]
  type = ConstMassTransfer
  variable = u
  coupled = v
  transfer_rate = 1
[../]
```

ConstReaction

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel creates a generic residual for a single reaction. That reaction is assumed to be mechanistic (i.e., the stoichiometric coefficients are also the powers of the variables in the rate expression). The formulation of the kernel is such that it is valid for both reversible and irreversible reactions. A scaling parameter is provided to account for the molar amounts of a species either formed or consumed during the reaction. Multiple instances of this object can be invoked to create residuals for a series of reactions that all contribute to the losses/gains in a specific chemical species.

Residual Formulation

Residual (mass / volume (or area) / time)

$$= - \left(a k_f \cdot \prod_{react} C_i^{v_i} - a k_r \cdot \prod_{prod} C_j^{v_j} \right) \cdot \varphi$$

φ = MOOSE variable test function (for FE formulation)

a = scaling parameter

- Used as a conversion for the rates of a particular species
- Additionally, the scaling parameter is used to determine whether a species is lost or gained in a reaction. Negative scaling indicates the species is lost. Positive scaling indicates the species is gained. (i.e., if using this kernel to add a reaction rate for a species whose stoichiometry in the reaction is 2 and is lost in the reaction, then the scaling $a = -2$).

k_f = forward rate constant (units depend on number of reactants)

k_r = reverse rate constant (units depend on number of products)

C_i = the i^{th} reactant species variable

C_j = the j^{th} product species variable

v_i = the i^{th} reactant species' stoichiometry

v_j = the j^{th} product species' stoichiometry

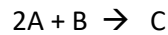
- Both the product and reaction species and stoichiometries will be given as a list of variables and values in the input file. The stoichiometry values given MUST be in the same order that the variable names are given in.

Additional Computations → None

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoeffTimeDerivative](#) kernel.

Code below creates a residual for species A for the following reaction:



```
[./rxn]
  type = ConstReaction
  variable = A
  this_variable = A
  scale = -2
  forward_rate = 1
  reactants = 'A B'
  reactant_stoich = '2 1'
  products = 'C'
  product_stoich = '1'
[../]
```

CoupledCoeffTimeDerivative

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel is used to couple the time derivative of another non-linear variable to this variable's kernel set. It provides another means to couple the effects of mass or energy transfer, if the physics of transfer is directly related to the rate of change of another variable. For example, the mass transfer term in adsorption often is coupled to the bulk phase concentration as a sink term that is a direct function of the rate of adsorption ($-a \cdot dq/dt$).

Residual Formulation

$$\text{Residual (mass / volume / time)} = a \cdot \frac{dv}{dt} \cdot \varphi$$

φ = MOOSE variable test function (for FE formulation)

a = time coefficient (useful for unit conversions or scaling)

- If a is a positive number, then this is a sink term to the main variable
→ If a is a negative number, then this is a source term

v = coupled non-linear variable

(*Deprecated option*: “gaining” – Boolean used to determine whether or not to treat this as a source or sink term. Just change the sign of a to change from sink term to a source term).

Additional Computations → None

Usage

Must be used in conjunction with a set of kernels for the main variable and the coupled variable.

Code below would create a sink term for the variable u based on the rate of change of v:

```
[./coupled_dv_dt]
  type = CoupledCoeffTimeDerivative
  variable = u
  coupled = v
  time_coeff = 1
[../]
```

CoupledPorePhaseTransfer

Inheritance → [CoupledCoeffTimeDerivative](#)

Notes → This kernel creates a specific form of [CoupledCoeffTimeDerivative](#) wherein the concentration of a bulk fluid species is coupled to the time derivative of a solid or surface species in a particle. The time coefficient is the porosity of the bulk phase, which is a common defining parameter for packed bed reactors. That porosity is used as a unit conversion from mass per volume of solids for the coupled variable to mass per total volume in the system.

Residual Formulation

(See [CoupledCoeffTimeDerivative](#) for residual formulation)

Additional Computations

The time coefficient (a) for the [CoupledCoeffTimeDerivative](#) is overridden to be as follows:

$$a = (1 - \varepsilon) \quad \text{where } \varepsilon \text{ is the bulk porosity in the domain (volume voids / total volume)}$$

This kernel still uses the “gaining” Boolean argument to determine whether or not to treat this transfer as a source or sink term. That option is deprecated, so you do not need to use it.

Usage

Must be used in conjunction with a set of kernels for the bulk variable and the coupled variable.

Code below would create a sink term for the variable Cb based on the rate of change of q:

```
[./coupled_dq_dt]
  type = CoupledPorePhaseTransfer
```

```

    variable = Cb
    coupled = q
    porosity = eps
[../]

```

CoupledSumFunction

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel creates a residual based on a summation of other non-linear variables. It is intended to create a residual for a non-linear variable (not in the list of summed variables) to enforce the constraint that the value of the primary variable be a sum of other variables. For example, you can use this to establish the total amount of a material adsorbed as the sum of different adsorbed species that contribute to the total.

Residual Formulation

$$\text{Residual (same units as the variables given)} = -\varphi \cdot \sum_{\forall i} u_i$$

φ = MOOSE variable test function (for FE formulation)

u_i = i^{th} coupled non-linear variable

Additional Computations → None

Usage

Generally used in conjunction with the built-in Reaction kernel (MOOSE framework kernel) in order to complete the summation expression that one non-linear variable be made as a sum of other non-linear variables.

In the example code below, this kernel is being used to sum up the adsorption from 3 species (q1, q2, and q3) to form the total adsorption (qT). Note that here we also must invoke the Reaction kernel on qT to force the finished summation expression.

```

[./qT_res]
    type = Reaction
    variable = qT
[../]
[./qT_sum]
    type = CoupledSumFunction
    variable = qT
    coupled_list = 'q1 q2 q3'
[../]

```

DivergenceFreeCondition

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel creates a residual based on a coupled divergence function. Generally, this kernel would be used to resolve pressure and/or potential gradients for the resolution of a flow field. This particular formulation is valid ONLY in cartesian coordinates, since the divergence condition is done in a piece-wise fashion. An optional variable for coupling a non-linear scalar variable is also provided within this formulation. As such, this can be used for both Incompressible and Compressible flows.

Residual Formulation

$$\text{Residual (depends on given variables)} = \varphi \cdot (\nabla \cdot \rho \mathbf{v})$$

φ = MOOSE variable test function (for FE formulation)

ρ = scalar coupled variable (such as density in Navier-Stokes)

\mathbf{v} = coupled vector of variables in Cartesian coordinates (ux, uy, uz)

Additional Computations → None

Usage

Generally used to enforce the continuity equation for fluid mass (or a continuity equation for current/electroneutrality). By default, the ' ρ ' parameter will be 1, in which case this simplifies to just the divergence of the given vector of variables.

In the demonstration below, we use this as the sole kernel acting on a pressure variable to solve the Navier-Stokes equations. In this formulation, we also give the 'coupled_scalar' as a 'rho' variable so that this is a valid expression in both incompressible and compressible flow.

```
[./continuity]
  type = DivergenceFreeCondition
  variable = pressure
  coupled_scalar = rho
  ux = vel_x
  uy = vel_y
  uz = vel_z
[../]
```

EquilibriumReaction

Inheritance → [ConstReaction](#)

Notes → This kernel uses the residual calculations from [ConstReaction](#), but calculates the equilibrium constant for the reaction from a given ΔH and ΔS for the reaction. The scaling parameter from [ConstReaction](#) is overridden to 1, since it is useless for an equilibrium reaction, and the reverse rate is overridden to 1, then the forward rate forced to become the equilibrium

constant. Use this kernel when you want to specify some local equilibrium between reactants and products, or when rate constants are unknown, but ΔH and ΔS are known.

Residual Formulation

(See [ConstReaction](#) for residual formulation)

Additional Computations

The value of the scaling parameter (a) from ConstReaction is overridden to be 1.

The value of the reverse rate (k_r) from ConstReaction is overridden to be 1.

The value of the forward rate (k_f) from ConstReaction is transformed into the equilibrium constant for this reaction through the van't Hoff relationship:

$$k_f = K = \exp \left\{ -\frac{\Delta H}{RT} + \frac{\Delta S}{R} \right\}$$

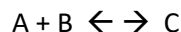
ΔH = reaction enthalpy (J/mol)

ΔS = reaction entropy (J/K/mol)

Usage

Generally, this kernel is intended to be used by itself to establish local equilibrium for a chemical species in the simulation domain. However, it can also be used in conjunction with other kernels on that variable for motion or transport of that species.

Example code below applies local equilibrium to chemical species A given the reaction:



[./rxn]

```
type = EquilibriumReaction
variable = A
this_variable = A
temperature = T
enthalpy = -3.9E5
entropy = 0
reactants = 'A B'
reactant_stoich = '1 1'
products = 'C'
product_stoich = '1'
```

[../]

ExtendedLangmuirFunction

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel creates a residual for the function side of the Extended Langmuir function for competitive adsorption with ideal surface behavior. While it may be tempting to use this kernel as a quick and easy way to establish adsorption behavior, this kernel is less computationally efficient than representing adsorption through a series of reactions using the

[ArrheniusReaction](#), [ConstReaction](#), [ArrheniusEquilibriumReaction](#), or other such combinations of reaction kernels. It is also worth noting that this kernel only provides the right-hand side of the Extended Langmuir function and thus needs to be combined with the standard MOOSE Reaction kernel to finalize the representation. You can also combine multiple instances of this kernel to create a Multi-site or Heterogeneous Extended Langmuir type of function.

Residual Formulation

$$\text{Residual (units of adsorbed species)} = -\varphi \cdot q_{\max} \cdot \frac{K_i C_i}{1 + \sum_j K_j C_j}$$

φ = MOOSE variable test function (for FE formulation)

q_{\max} = theoretical maximum adsorption for this site (site density)

C_i = concentration of the primary bulk species adsorbing

K_i = equilibrium Langmuir constant for C_i (inverse units of C_i)

C_j = concentrations of all species j involved in reaction (including C_i)

K_j = equilibrium Langmuir constant for C_j (inverse units of C_j)

Additional Computations → None

Usage

Generally used in conjunction with the built-in Reaction kernel (MOOSE framework kernel) in order to complete the adsorption expression.

Note that the list of adsorbing species MUST align in order with the list of Langmuir constants for those species.

Example below calculates the adsorption of species A to form q in the presence of species B, which is competing for the same adsorption sites as A. The Langmuir constant for A is 0.25 and for B is 0.75. The maximum site density (q_{\max}) is 0.5.

```
[./q_res]
  type = Reaction
  variable = q
[../]
[./q_lang]
  type = ExtendedLangmuirFunction
  variable = q
  site_density = 0.5
  main_coupled = A
  coupled_list = 'A B'
  langmuir_coeff = '0.25 0.75'
[../]
```

ExtendedLangmuirModel

Inheritance → [ExtendedLangmuirFunction](#)

Notes → This kernel uses the same residuals from [ExtendedLangmuirFunction](#), but calculates the Langmuir coefficients from the DH and DS of each species' site reaction based on the van't Hoff relationship. All other notes from [ExtendedLangmuirFunction](#) also apply to this kernel.

Residual Formulation

(See [ExtendedLangmuirFunction](#) for residual formulation)

Additional Computations

The Langmuir coefficients for each species involved in the reactions is calculated from the van't Hoff expression. See [EquilibriumReaction](#) for additional computation details. Must provide reaction enthalpies and entropies.

ΔH = reaction enthalpy (J/mol)

ΔS = reaction entropy (J/K/mol)

Usage

Usage is essentially the same as [ExtendedLangmuirFunction](#), but user provides enthalpies and entropies instead of Langmuir coefficients (example below).

Example below calculates the adsorption of species A to form q in the presence of species B, which is competing for the same adsorption sites as A.

```
[./q_res]
  type = Reaction
  variable = q
[../]
[./q_lang]
  type = ExtendedLangmuirModel
  variable = q
  site_density = 0.5
  main_coupled = A
  coupled_list = 'A B'
  coupled_temp = T
  enthalpies = '-5000 -4000'
  entropies = '100 50'
[../]
```

FilmMassTransfer

Inheritance → [ConstMassTransfer](#)

Notes → This kernel uses the residual calculations from [ConstMassTransfer](#), but allows for a variable rate transfer rate coefficient (k_m) and applies a conversion factor (A_s) to account for the

transfer of mass across a specific area of a film layer of another phase (i.e., such as a bubble or solid particle).

Residual Formulation

(See [ConstMassTransfer](#) for residual formulation)

Additional Computations

The original rate parameter (k) from [ConstMassTransfer](#) is overridden in this kernel prior to calling the base kernel residual function.

$$k = k_m A_s v_f$$

k_m = variable rate of film mass transfer (m/s)

A_s = film layer area per volume of other phase (m^{-1})

v_f = volume fraction (used to convert units from one volume basis to another)

Usage

This kernel has the same usage as its base class [ConstMassTransfer](#). The only difference is the inclusion of the A_s and v_f coupled variables, and coupling with the k_m variable.

The below example is for the transfer of mass from the bulk phase (C_b) to the pore-space (C_p) of a porous material with rate variable of k_m and area-to-volume ratio of $1000 m^{-1}$. Note that to balance the mass between the 2 phases, you must include this kernel for both the bulk phase (C_b) and the pore-space (C_p) separately, and also use in conjunction with other kernels describing the transport or changes in those variables respectively. Additionally, we use a variable named (solids_frac) to apply the volume fraction unit conversion to each kernel. Note that this factor is optional. The default value in the kernel is set to 1.

```
[./mass_trans_from_bulk_to_pores]
  type = FilmMassTransfer
  variable = Cb
  coupled = Cp
  av_ratio = 1000
  rate_variable = km
  volume_frac = solids_frac
[../]
```

```
[./mass_trans_from_pores_to_bulk]
  type = FilmMassTransfer
  variable = Cp
  coupled = Cb
  av_ratio = 1000
  rate_variable = km
  volume_frac = solids_frac
[../]
```

GAdvection (DGAdvection)

Inheritance: GAdvection → Kernel & DGAdvection → DGKernel

Special Notes

→ All kernels prefixed with ‘G’ and ‘DG’ must be used together (Except for Boundary Conditions). Combined they fully describe 1 piece of transport physics. You **CANNOT** use either ‘G’ or ‘DG’ kernels without also invoking the other. If you do, your simulation will likely result in errors in conservation of mass or energy.

→ Any variable in the MOOSE system that uses ‘DG’ kernels must also either use MONOMIAL or L2_LAGRANGE shape functions (i.e., family). These are invoked when creating the variables under the variables block (see below).

```
[Variables]
  [./u]
    order = FIRST
    family = MONOMIAL
  [./]
[]
```

→ It is recommended that you use a variable ‘order’ as either FIRST or SECOND for accuracy. However, if solutions are unstable, then reduce the ‘order’ to CONSTANT.

Notes → This set of kernels formulates the Discontinuous Galerkin form for conservative advection in a simulation domain. This is the base class for advective physics in the DG system of kernels. All other advection kernels will inherit from this kernel. The velocities are assumed to be predetermined constants for the base class.

Residual Formulation

Due to the complexity of forming transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (mass / volume / time)} = \varphi \cdot \nabla \cdot (\mathbf{v}u)$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector (m/s)

u = a conserved quantity per volume (mass / volume) or (energy / volume)

Additional Computations → None

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative advection of u with a constant velocity in the x-direction.

```
[Kernels]
  [./g_adv_u]
    type = GAdvection
    variable = u
    vx = 1
    vy = 0
    vz = 0
  [../]
[]

[DGKernels]
  [./dg_adv_u]
    type = DGAdvection
    variable = u
    vx = 1
    vy = 0
    vz = 0
  [../]
[]
```

GAnisotropicDiffusion (DGAnisotropicDiffusion)

Inheritance: GAnisotropicDiffusion → Kernel & DGAnisotropicDiffusion → DGKernel

Special Notes

→ All kernels prefixed with 'G' and 'DG' must be used together (Except for Boundary Conditions). Combined they fully describe 1 piece of transport physics. You **CANNOT** use either 'G' or 'DG' kernels without also invoking the other. If you do, your simulation will likely result in errors in conservation of mass or energy.

→ Any variable in the MOOSE system that uses 'DG' kernels must also either use MONOMIAL or L2_LAGRANGE shape functions (i.e., family). These are invoked when creating the variables under the variables block (see below).

```
[Variables]
  [./u]
    order = FIRST
    family = MONOMIAL
  [../]
[]
```

→ It is recommended that you use a variable 'order' as either FIRST or SECOND for accuracy. However, if solutions are unstable, then reduce the 'order' to CONSTANT.

→ **NOTE:** If you use a **CONSTANT** order for the variables, then this set of kernels will provide almost no value to the simulation. This is because diffusion is based on the local gradients of the variables and **CONSTANT** order variables have no gradients on a given element in the domain. However, it is sometimes necessary to invoke this order anyway for maximum stability.

→ Note: DG diffusion type kernels all require additional parametric options to penalize jumps in discontinuity between neighboring variable values. These include a 'sigma' parameter and a 'dg_scheme' parameter.

dg_scheme → Can be either 'nipg', 'sipg', or 'iipg'. Default is 'nipg'.

More information on dg_scheme can be found in the corresponding source code files.

sigma → penalty term which should be ≥ 0 . Default is 10.

As this penalty term gets higher, it convergence suffers.

Generally will want to put these options under 'GlobalParameters'

```
[GlobalParameters]
  dg_scheme = nipg
  sigma = 10
[]
```

Notes → This set of kernels formulates the Discontinuous Galerkin form for conservative diffusion or dispersion in a simulation domain. This is the base class for diffusion physics in the DG system of kernels. All other diffusion kernels will inherit from this kernel. The diffusivity is taken as a tensor matrix such that the kernel can simulate transport in a medium that has anisotropic properties. However, you can just given the same diffusivities for the diagonal elements of the tensor to simulate isotropic diffusion transport with this kernel as well.

Residual Formulation

Due to the complexity of forming transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the "strong form" of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (mass / volume / time)} = -\phi \cdot \nabla \cdot (\mathbf{D} \cdot \nabla u)$$

ϕ = MOOSE variable test function (for FE formulation)

\mathbf{D} = diffusion tensor (m^2/s)

→ The tensor is given element-by-element in input values named as D_{ij} where i is the row and j is the column of the tensor

→ Example: D_{xy} = xy -component of diffusion (which can be different from D_{yx} if desired)

➔ For isotropic diffusion, simply provide only Dxx, Dyy, and Dzz with the same values (off-diagonals are assumed 0).

u = a conserved quantity per volume (mass / volume) or (energy / volume)

Additional Computations ➔ None

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative diffusion of u with a constant, isotropic diffusion.

```
[Kernels]
  ./g_diff_u
    type = GAnisotropicDiffusion
    variable = u
    Dxx = 1
    Dyy = 1
    Dzz = 1
  [../]
[]

[DGKernels]
  ./dg_diff_u
    type = DGAnisotropicDiffusion
    variable = u
    Dxx = 1
    Dyy = 1
    Dzz = 1
  [../]
[]
```

GConcentrationAdvection (DGConcentrationAdvection)

Inheritance:

GConcentrationAdvection ➔ [GAdvection](#)

DGConcentrationAdvection ➔ [DGAdvection](#)

Special Notes ➔ (See [GAdvection \(DGAdvection\)](#) for special notes)

Notes ➔ This kernel set uses the residuals from [GAdvection \(DGAdvection\)](#), but allows for a variable velocity, which is useful for coupling mass transfer to a velocity field that comes from other conservation of momentum simulations (i.e., Navier-Stokes modules).

Residual Formulation ➔ (See [GAdvection \(DGAdvection\)](#) for residual formulation)

Additional Computations → The velocity vector variable from [GAdvection \(DGAdvection\)](#) is overridden with the corresponding variable values for each velocity vector component.

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative advection of u with a variable velocity vector constructed from vector components.

```
[Kernels]
  ./g_adv_u
    type = GConcentrationAdvection
    variable = u
    ux = vel_x
    uy = vel_y
    uz = vel_z
  [../]
[]

[DGKernels]
  ./dg_adv_u
    type = DGConcentrationAdvection
    variable = u
    ux = vel_x
    uy = vel_y
    uz = vel_z
  [../]
[]
```

[GNernstPlanckDiffusion \(DGNernstPlanckDiffusion\)](#)

Inheritance:

GVarPoreDiffusion → [GVariableDiffusion](#)

DGVarPoreDiffusion → [DGVariableDiffusion](#)

Special Notes → This kernel acts on a concentration of ions in solution, but couples that diffusion with an electric potential gradient to base the diffusive flux off of.

NOTE: This kernel ONLY includes diffusive flux based on an electric potential gradient. It should be combined with another diffusion kernel (such as [GVariableDiffusion \(DGVariableDiffusion\)](#)) or similar to represent the full flux term.

Notes → This set of kernels calculates the diffusive flux of ions caused by an electric potential gradient. If there is no electric potential gradient, then this term does not introduce any flux. Users must also provide the valence of the ions (which will help to dictate both the magnitude and direction of ionic flux). This kernel includes 2 constants (Faraday’s constant and the Gas Law constant) that the user may provide other values for other than their defaults. This may be useful for converting this kernel to use a different unit basis in the overall ion mass balance.

Residual Formulation

Due to the complexity of forming transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (mass / volume / time)} = -\varphi \cdot \nabla \cdot \left(\frac{zF}{RT} \mathbf{D} \cdot \varepsilon \cdot c \nabla \phi \right)$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{D} = diffusion tensor (m²/s)

→ See [GVariableDiffusion \(DGVariableDiffusion\)](#) for more notes

c = a conserved ion quantity per volume (mass / volume)

ε = porosity of the domain

z = valence of the ions (negative values for anions, positive values for cations)

T = coupled temperature variable (in K)

ϕ = electric potential variable in domain (units depend on F and R)

[Default units would be V or J/C]

F = Faraday’s constant (default = 96,485.3 C/mol)

R = Gas Law constant (default = 8.314462 J/K/mol)

Additional Computations → None

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the ‘G’ and ‘DG’ kernels matching by name and have matching parameters. ‘G’ kernels are placed under the [Kernels] block and ‘DG’ kernels are placed under the [DGKernels] block in the input files.

Below example is for diffusion of positive ions in a porous media with the Nernst-Planck diffusion added to the standard diffusion. The porosity value is 0.5 and each kernel is coupled with variable for diffusivity (D), temperature (temp), and electric potential (phi).

```
[Kernels]
  ./g_diff_u]
```

```

        type = GVarPoreDiffusion
        variable = u
        porosity = 0.5
        Dx = D
        Dy = D
        Dz = D
    [../]
    [./g_np_diff_u]
        type = GNernstPlanckDiffusion
        variable = u
        porosity = 0.5
        temperature = temp
        electric_potential = phi
        valence = 1
        Dx = D
        Dy = D
        Dz = D
    [../]
[]

[DGKernels]
    [./dg_diff_u]
        type = DGVarPoreDiffusion
        variable = u
        porosity = 0.5
        Dx = D
        Dy = D
        Dz = D
    [../]
    [./dg_np_diff_u]
        type = DGNernstPlanckDiffusion
        variable = u
        porosity = 0.5
        temperature = temp
        electric_potential = phi
        valence = 1
        Dx = D
        Dy = D
        Dz = D
    [../]
[]

```

GNSMomentumAdvection (DGNSMomentumAdvection)

Inheritance: GNSMomentumAdvection → [GConcentrationAdvection](#)

DGNSMomentumAdvection → [DGConcentrationAdvection](#)

Special Notes

- All kernels prefixed with 'G' and 'DG' must be used together (Except for Boundary Conditions). Combined they fully describe 1 piece of transport physics. You **CANNOT** use either 'G' or 'DG' kernels without also invoking the other. If you do, your simulation will likely result in errors in conservation of mass or energy.
- Any variable in the MOOSE system that uses 'DG' kernels must also either use MONOMIAL or L2_LAGRANGE shape functions (i.e., family). These are invoked when creating the variables under the variables block (see below).
- Specifically for Navier-Stokes equations with DG methods. Users should use 'SECOND' order functions for each velocity term.
- Navier-Stokes equations also require the usage of a pressure variable. That variable MUST BE 'FIRST' order 'LAGRANGE' functions for improved stability and evaluations of boundary pressure.

```
[Variables]
  [./vel_x]
    order = SECOND
    family = MONOMIAL
  [../]
  [./vel_y]
    order = SECOND
    family = MONOMIAL
  [../]
  [./pressure]
    order = FIRST
    family = LAGRANGE
  [../]
[]
```

Notes → This set of kernels formulates the Discontinuous Galerkin form for conservative momentum advection in a simulation domain. These kernels act on velocity components in a piece-wise manner. Other sections of the Navier-Stokes equations are satisfied from other existing kernels such as [GVariableDiffusion \(DGVariableDiffusion\)](#) and [VectorCoupledGradient](#).

Residual Formulation

Due to the complexity of forming Navier-Stokes transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (momentum / volume / time)} = \varphi \cdot \nabla \cdot (\rho \mathbf{v} \cdot \mathbf{u}_i)$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector (length / time)

[NOTE: this vector is built-up from each u_i velocity variable component]

ρ = density variable (mass / volume)

u_i = velocity variable in the i-th direction (where i = x, y, or z)

Additional Computations → None

Usage

This kernel set is designed specifically for the use in the Navier-Stokes equations to formulate a DG implementation and solution to that system of equations. By itself, this kernel set does not do much. It should be combined with a [VariableCoefTimeDerivative](#), [VectorCoupledGradient](#), and [GVariableDiffusion \(DGVariableDiffusion\)](#) to complete the Navier-Stokes residuals. In addition, you must use these in conjunction with [VectorCoupledGradient](#) and/or [DivergenceFreeCondition](#) objects that act on the pressure variable to resolve the continuity equation portion of the Navier-Stokes equation set.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below is an example for how to setup the momentum balance from Navier-Stokes for only the conservation of momentum in the x-direction. (**NOTE:** This example does not show the continuity equation. See the example in [VectorCoupledGradient](#) and/or [DivergenceFreeCondition](#) for that formulation).

```
[Kernels]
# rho * d(vel_x)/dt
[./x_dot]
    type = VariableCoefTimeDerivative
    variable = vel_x
    coupled_coef = rho

[../]
# -grad(P)_x
[./x_press]
    type = VectorCoupledGradient
    variable = vel_x
    coupled = pressure
    vx = 1

[../]
# Div*(mu*grad(vel_x))
[./x_gvis]
    type = GVariableDiffusion
    variable = vel_x
    Dx = mu
    Dy = mu
    Dz = mu

[../]
# Div*(rho*vel*vel_x)
[./x_gadv]
```

```

        type = GNSMomentumAdvection
        variable = vel_x
        this_variable = vel_x
        density = rho
        ux = vel_x
        uy = vel_y
        uz = vel_z
    [../]
[]

[DGKernels]
    # Div*(mu*grad(vel_x))
    [./x_dgvis]
        type = DGVariableDiffusion
        variable = vel_x
        Dx = mu
        Dy = mu
        Dz = mu
    [../]
    # Div*(rho*vel*vel_x)
    [./x_dgadv]
        type = DGNSMomentumAdvection
        variable = vel_x
        this_variable = vel_x
        density = rho
        ux = vel_x
        uy = vel_y
        uz = vel_z
    [../]
[]

```

GNSViscousVelocityDivergence (DGNSViscousVelocityDivergence)

Inheritance: GNSMomentumAdvection → [GConcentrationAdvection](#)

DGNSMomentumAdvection → [DGConcentrationAdvection](#)

Special Notes

→ All kernels prefixed with 'G' and 'DG' must be used together (Except for Boundary Conditions). Combined they fully describe 1 piece of transport physics. You **CANNOT** use either 'G' or 'DG' kernels without also invoking the other. If you do, your simulation will likely result in errors in conservation of mass or energy.

→ Any variable in the MOOSE system that uses 'DG' kernels must also either use MONOMIAL or L2_LAGRANGE shape functions (i.e., family). These are invoked when creating the variables under the variables block (see below).

➔ Specifically for Navier-Stokes equations with DG methods. Users should use 'SECOND' order functions for each velocity term.

➔ Navier-Stokes equations also require the usage of a pressure variable. That variable MUST BE 'FIRST' order 'LAGRANGE' functions for improved stability and evaluations of boundary pressure.

```
[Variables]
  [./vel_x]
    order = SECOND
    family = MONOMIAL
  [./]
  [./vel_y]
    order = SECOND
    family = MONOMIAL
  [./]
  [./pressure]
    order = FIRST
    family = LAGRANGE
  [./]
[]
```

Notes ➔ This set of kernels formulates the Discontinuous Galerkin form for conservative momentum advection in a simulation domain. These kernels act on velocity components in a piece-wise manner. Other sections of the Navier-Stokes equations are satisfied from other existing kernels such as [GVariableDiffusion \(DGVariableDiffusion\)](#) and [VectorCoupledGradient](#).

Residual Formulation

Due to the complexity of forming Navier-Stokes transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (momentum / volume / time)} = \varphi \cdot \frac{1}{3} \mu \nabla (\nabla \cdot \mathbf{v})$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector (length / time)

[NOTE: this vector is built-up from each u_i velocity variable component]

μ = viscosity variable (mass / length / time)

NOTE: When doing Navier-Stokes in a piece-wise fashion, each velocity component is its own non-linear variable and each will have an instance of this kernel. When doing so, the above residual formulation breaks down into the following:

$$\text{Residual} = \frac{1}{3} \mu \cdot [\mathbf{D}_{ix} \cdot \nabla u_x + \mathbf{D}_{iy} \cdot \nabla u_y + \mathbf{D}_{iz} \cdot \nabla u_z]$$

\mathbf{D}_{ix} = Tensor matrix whose entries are all 0 except $D(i,x)$, which has value of 1.

D_{iy} = Tensor matrix whose entries are all 0 except $D(i,y)$, which has value of 1.

D_{iz} = Tensor matrix whose entries are all 0 except $D(i,z)$, which has value of 1.

i = Cartesian direction of the velocity variable that this kernel acts on

Additional Computations → None

Usage

This kernel set is designed specifically for the use in the Navier-Stokes equations to formulate a DG implementation and solution to that system of equations. By itself, this kernel set does not do much. It should be combined with a [VariableCoefTimeDerivative](#), [VectorCoupledGradient](#), and [GVariableDiffusion \(DGVariableDiffusion\)](#) to complete the Navier-Stokes residuals. In addition, you must use these in conjunction with [VectorCoupledGradient](#) and/or [DivergenceFreeCondition](#) objects that act on the pressure variable to resolve the continuity equation portion of the Navier-Stokes equation set.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below is an example for how to setup the momentum balance from Navier-Stokes for only the conservation of momentum in the x-direction. (**NOTE:** This example does not show the continuity equation. See the example in [VectorCoupledGradient](#) and/or [DivergenceFreeCondition](#) for that formulation).

NOTE: This kernel is technically ONLY needed for compressible flow.

[Kernels]

```
# rho * d(vel_x)/dt
[./x_dot]
    type = VariableCoefTimeDerivative
    variable = vel_x
    coupled_coef = rho

[./]
# -grad(P)_x
[./x_press]
    type = VectorCoupledGradient
    variable = vel_x
    coupled = pressure
    vx = 1

[./]
# Div*(mu*grad(vel_x))
[./x_gvis]
    type = GVariableDiffusion
    variable = vel_x
    Dx = mu
    Dy = mu
    Dz = mu
```



```

[../]
# Div*(rho*vel*vel_x)
[./x_gadv]
    type = GNSMomentumAdvection
    variable = vel_x
    this_variable = vel_x
    density = rho
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]
# (1/3)*mu*(grad(Div*vel)_x)
[./x_gvisdiv]
    type = GNSViscousVelocityDivergence
    variable = vel_x
    this_variable = vel_x
    viscosity = mu
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]

[]
[DGKernels]
# Div*(mu*grad(vel_x))
[./x_dgvis]
    type = DGVariableDiffusion
    variable = vel_x
    Dx = mu
    Dy = mu
    Dz = mu
[../]
# Div*(rho*vel*vel_x)
[./x_dgadv]
    type = DGNSMomentumAdvection
    variable = vel_x
    this_variable = vel_x
    density = rho
    ux = vel_x
    uy = vel_y
    uz = vel_z
[../]
# (1/3)*mu*(grad(Div*vel)_x)
[./x_dgvisdiv]
    type = DGNSViscousVelocityDivergence
    variable = vel_x
    this_variable = vel_x
    viscosity = mu

```

```

ux = vel_x
uy = vel_y
uz = vel_z
[./]

```

[]

GPhaseThermalConductivity (DGPhaseThermalConductivity)

Inheritance

GPhaseThermalConductivity → [GThermalConductivity](#)

DGPhaseThermalConductivity → [DGThermalConductivity](#)

Special Notes → (See [GThermalConductivity \(DGThermalConductivity\)](#) for special notes)

Notes → This kernel uses the same residuals from [GThermalConductivity \(DGThermalConductivity\)](#), but scales the conductivity transport of energy by a volume fraction that represents the fraction of the domain the represents the specific phase holding the energy. For instance, in the case of porous flow, you may have 1 energy density in the solids and another energy density in the fluid phases. Thus, the volume fraction would be representative of the solids-to-total volume or the fluid-to-total volume ratio.

Residual Formulation

Due to the complexity of forming transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (J / m}^3 \text{ / s)} = -\varphi \cdot \nabla \cdot (\mathbf{K} \cdot f_v \cdot \nabla T)$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{K} = thermal conductivity tensor (W/m/K)

→ See [GThermalConductivity \(DGThermalConductivity\)](#) for more notes

T = temperature of the phase (K)

f_v = volume fraction of the phase (e.g., volume voids / total volume)

Additional Computations → None

Usage

NOTE: This kernel acts on an energy variable, but is coupled to the temperature variable

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the ‘G’ and ‘DG’ kernels matching by name and have matching parameters (i.e., same velocities). ‘G’ kernels are placed under the [Kernels] block and ‘DG’ kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative thermal conductivity of fluid phase energy (Ef) in a system with a porosity of 0.5 and an isotropic conductivity behavior in x, y, and z directions with a variable conductivity (Kg).

```
[Kernels]
  [./g_cond_T]
    type = GPhaseThermalConductivity
    variable = Ef
    temperature = T
    volume_frac = 0.5
    Dx = Kg
    Dy = Kg
    Dz = Kg
  [../]
[]

[DGKernels]
  [./dg_cond_T]
    type = DGPhaseThermalConductivity
    variable = Ef
    temperature = T
    volume_frac = 0.5
    Dx = Kg
    Dy = Kg
    Dz = Kg
  [../]
[]
```

GPOreConcAdvection (DGPOreConcAdvection)

Inheritance:

GPOreConcAdvection → [GConcentrationAdvection](#)

DGPOreConcAdvection → [DGConcentrationAdvection](#)

Special Notes → (See [GConcentrationAdvection \(DGConcentrationAdvection\)](#) for special notes)

Notes → This kernel set uses the residuals from [GConcentrationAdvection \(DGConcentrationAdvection\)](#), but scales the advective flow by a volume fraction or a porosity variable to represent the fraction of the total domain that is open for advective transport. This is most commonly used for the advective flow of mass or energy in packed beds or porous media.

Residual Formulation

Due to the complexity of forming transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (mass / volume / time)} = \varphi \cdot \nabla \cdot (\mathbf{v} \cdot \varepsilon u)$$

ϕ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector (m/s)

u = a conserved quantity per volume (mass / volume) or (energy / volume)

ε = volume fraction or porosity variable for the void spaces

Additional Computations → None

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative advection of fluid energy (Ef) with a variable velocity vector constructed from vector components and a variable porosity (eps).

```
[Kernels]
  [./g_adv_E]
    type = GPoreConcAdvection
    variable = Ef
    porosity = eps
    ux = vel_x
    uy = vel_y
    uz = vel_z
  [../]
[]

[DGKernels]
  [./dg_adv_E]
    type = DGPoreConcAdvection
    variable = Ef
    porosity = eps
    ux = vel_x
    uy = vel_y
    uz = vel_z
  [../]
[]
```

[GThermalConductivity](#) ([DGThermalConductivity](#))

Inheritance

GThermalConductivity → [GVariableDiffusion](#)

DGThermalConductivity → [DGVariableDiffusion](#)

Special Notes → (See [GAnisotropicDiffusion](#) ([DGAnisotropicDiffusion](#)) for special notes)

→ While the physics of thermal conductivity and mass diffusion are extraordinarily similar, there is a major difference in their usages. This is because an energy balance's variable is energy density and not temperature. However, the conductivity of energy is driven entirely by temperature, which itself is a variable. Thus, this kernel is not a function of the energy variable.

Notes → This kernel uses the same basic strong form residuals from [GVariableDiffusion](#) ([DGVariableDiffusion](#)), which is also the same as [GAnisotropicDiffusion](#) ([DGAnisotropicDiffusion](#)), but acts on a variable that is not the primary variable. Thus, this kernel did have to be specially constructed for our usage. Here, the diffusion tensor (**D**) is replaced with a conductivity tensor (**K**) and the variable gradient is no longer with respect to the conserved quantity, but the temperature (T) of the media.

Residual Formulation

Due to the complexity of forming transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (J / m}^3 \text{ / s)} = -\varphi \cdot \nabla \cdot (\mathbf{K} \cdot \nabla T)$$

φ = MOOSE variable test function (for FE formulation)

K = thermal conductivity tensor (W/m/K)

→ See [GThermalConductivity](#) ([DGThermalConductivity](#)) for more notes

T = temperature of the phase (K)

Additional Computations → None

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the ‘G’ and ‘DG’ kernels matching by name and have matching parameters (i.e., same velocities). ‘G’ kernels are placed under the [Kernels] block and ‘DG’ kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative thermal conductivity of fluid phase energy (Ef) in a system with an isotropic conductivity behavior in x, y, and z directions with a variable conductivity (Kg).

```
[Kernels]
  [./g_cond_T]
    type = GThermalConductivity
    variable = Ef
    temperature = T
    Dx = Kg
    Dy = Kg
```

```

        Dz = Kg
    [../]
[]

[DGKernels]
    [./dg_cond_T]
        type = DGThermalConductivity
        variable = Ef
        temperature = T
        Dx = Kg
        Dy = Kg
        Dz = Kg
    [../]
[]

```

GVarPoreDiffusion (DGVarPoreDiffusion)

Inheritance:

GVarPoreDiffusion → [GVariableDiffusion](#)

DGVarPoreDiffusion → [DGVariableDiffusion](#)

Special Notes → (See [GAnisotropicDiffusion \(DGAnisotropicDiffusion\)](#) for special notes)

Notes → This set of kernels uses the same residual calculation from [GVariableDiffusion \(DGVariableDiffusion\)](#), but scales the diffusive transport flux by the system porosity. Thus, representing only the diffusion that occurs through the pore spaces of a media. That porosity is allowed to be either a constant or a variable itself.

Residual Formulation

Due to the complexity of forming transport physics in the DG system of kernels, the exact formulations are not provided here. Instead, only the “strong form” of the physics is provided for all DG and associated kernels.

$$\text{Strong Form Residual (mass / volume / time)} = -\varphi \cdot \nabla \cdot (\mathbf{D} \cdot \varepsilon \nabla u)$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{D} = diffusion tensor (m²/s)

→ See [GVariableDiffusion \(DGVariableDiffusion\)](#) for more notes

u = a conserved quantity per volume (mass / volume)

ε = porosity of the domain

Additional Computations → None

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative diffusion of u with a variable, anisotropic diffusion in x, y, and z, as well as a constant porosity of 0.5.

```
[Kernels]
  [./g_diff_u]
    type = GVarPoreDiffusion
    variable = u
    porosity = 0.5
    Dx = Diff_x
    Dy = Diff_y
    Dz = Diff_z
  [../]
[]

[DGKernels]
  [./dg_diff_u]
    type = DGVarPoreDiffusion
    variable = u
    porosity = 0.5
    Dx = Diff_x
    Dy = Diff_y
    Dz = Diff_z
  [../]
[]
```

GVariableDiffusion (DGVariableDiffusion)

Inheritance:

GVariableDiffusion → [GAnisotropicDiffusion](#)

DGVariableDiffusion → [DGAnisotropicDiffusion](#)

Special Notes → (See [GAnisotropicDiffusion \(DGAnisotropicDiffusion\)](#) for special notes)

Notes → This set of kernels uses the same residual calculation from [GAnisotropicDiffusion \(DGAnisotropicDiffusion\)](#), but overrides the diffusion tensor (**D**) with given variables for diffusion in x, y, and z directions. From those three variables, the diffusion tensor is constructed

Residual Formulation → (See [GAnisotropicDiffusion \(DGAnisotropicDiffusion\)](#) for residual)

Additional Computations → The diffusion tensor (**D**) is overridden with diffusion variable components D_x , D_y , and D_z .

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Must always have the 'G' and 'DG' kernels matching by name and have matching parameters (i.e., same velocities). 'G' kernels are placed under the [Kernels] block and 'DG' kernels are placed under the [DGKernels] block in the input files.

Below example is for conservative diffusion of u with a variable, isotropic diffusion.

```
[Kernels]
  ./g_diff_u
      type = GVariableDiffusion
      variable = u
      Dx = Diff
      Dy = Diff
      Dz = Diff
  [../]
[]

[DGKernels]
  ./dg_diff_u
      type = DGVariableDiffusion
      variable = u
      Dx = Diff
      Dy = Diff
      Dz = Diff
  [../]
[]
```

InhibitedArrheniusReaction

Inheritance → [ArrheniusReaction](#)

Notes → This kernel has all the same residuals from [ArrheniusReaction](#), but changes the effective forward and reverse rates by dividing each of those rates by non-linear variables that will represent the inhibition term for each half of the reaction. The inhibition term can be given as a constant, but it is more useful to calculate that term in another kernel, or set of kernels, and couple it to this kernel. Some currently available inhibition kernels include [LangmuirInhibition](#), [PairedLangmuirInhibition](#), and [InhibitionProducts](#).

Residual Formulation

(See [ArrheniusReaction](#) for residual formulation)

Additional Computations

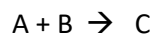
$$k_f = \frac{A_f T^{\beta_f} \exp\left[-\frac{E_f}{RT}\right]}{R_f} \quad k_r = \frac{A_r T^{\beta_r} \exp\left[-\frac{E_r}{RT}\right]}{R_r}$$

- The units for A_f and A_r must have the same units as k_f and k_r
- The units for E_f and E_r must be J/mol
 - The above parameters come from [ArrheniusReaction](#)
- R_f and R_r are non-linear variables that represent an inhibition term

Usage

Generally used in conjunction with a [TimeDerivative](#) or [VariableCoefTimeDerivative](#) kernel.

Code below creates a residual for species A for the following reaction:



That reaction rate is inhibited by a variable R, which needs its own kernel elsewhere.

```
[./rxn]
  type = InhibitedArrheniusReaction
  variable = A
  this_variable = A
  temperature = T
  scale = -1
  forward_activation_energy = 5E4
  forward_pre_exponential = 25
  forward_inhibition = R
  reactants = 'A B'
  reactant_stoich = '1 1'
  products = 'C'
  product_stoich = '1'
[../]
```

InhibitedArrheniusReactionEnergyTransfer

Inheritance → [InhibitedArrheniusReaction](#)

Notes → Although this kernel is used in energy balances and not mass balances, the kernels are mathematically very similar. Thus, this kernel uses the residual computed from [InhibitedArrheniusReaction](#), but applies the enthalpy (ΔH) and 2 unit conversion factors to modify the scaling parameter of [InhibitedArrheniusReaction](#). The net effect is to convert the existing reaction rate function from a mass term to an energy term. In addition, the forward and reverse rates are divided by the inhibition terms, which can be passed as non-linear variables or given constant values (see [InhibitedArrheniusReaction](#) for more details).

Residual Formulation

The baseline residual comes from [InhibitedArrheniusReaction](#). In this formulation, we will represent that portion of the residual as r (i.e., the rate of the reaction) with a scaling parameter of 1. See [InhibitedArrheniusReaction](#) for more details on scaling parameters. Then, this kernel uses that reaction residual (r) and applies a custom scaling based on ΔH and other unit conversions.

$$\text{Residual (J/m}^3\text{/s)} = -\Delta H \cdot f_v \cdot A_s \cdot r$$

f_v = volume fraction (i.e., volume of solids per total volume)

→ Can be set to 1 if no such conversion is needed

A_s = Specific reaction area per volume of solids (for a surface reaction)

→ Can be set to 1 if no such conversion is needed

ΔH = enthalpy of the reaction (J/mol)

r = reaction rate (mol/m³/s) or (mol/m²/s) [depends on reaction type]

→ Use f_v and A_s to convert units such that the units for the residual return as energy rate per total volume (J/m³/s)

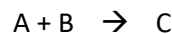
Additional Computations

As noted above, the scaling parameter from the reaction is overridden to always be 1.

Usage

Generally used in conjunction with an energy balance set of kernels to add the physics of energy changes caused by a chemical reaction in the system. The usage also includes terms to convert the volumetric or per reactive area terms into the proper energy density residual.

Code below creates a residual for energy exchange via the following surface reaction:



That reaction rate is inhibited by a variable R, which needs its own kernel elsewhere.

```
[./rxn]
  type = ArrheniusReactionEnergyTransfer
  variable = E          # E = Energy density variable
  this_variable = E
  temperature = T
  volume_frac = s_frac  # Volume of solids per Total Volume
  specific_area = 8.6E7  # Reaction area of solid per volume of solid
  forward_activation_energy = 5E4
  forward_pre_exponential = 25
  forward_inhibition = R
  enthalpy = -3.9E5
  reactants = 'A B'
  reactant_stoich = '1 1'
  products = 'C'
  product_stoich = '1'
[./]
```

InhibitionProducts

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel is used to create a residual to represent an inhibition term for [InhibitedArrheniusReaction](#). The inhibition term is its own non-linear variable and formulated as a product function of other inhibition terms. As such, this kernel must be used in conjunction with a set of other inhibition variables, whose values are fulfilled by other inhibition models such as [LangmuirInhibition](#) and [PairedLangmuirInhibition](#).

Residual Formulation

The residual is formed from a list of coupled inhibition terms and a list of powers that is applied to each term.

$$\text{Residual (-)} = -\varphi \cdot (\prod_i R_i^{p_i})$$

φ = MOOSE variable test function (for FE formulation)

R_i = i-th inhibition variable

p_i = power that the i-th inhibition term is raised to

Additional Computations → None

Usage

MUST be used in conjunction with a “Reaction” kernel from the MOOSE base system of kernels in order to fully describe the inhibition term (R) for a reaction.

Code below creates an inhibition term that is a function of inhibition terms RA and RB. Those other inhibition terms must be defined elsewhere in the input file.

```
[./R_equ]
  type = Reaction
  variable = R
[./]
[./R_prod]
  type = InhibitionProducts
  variable = R
  coupled_list = 'RA RB'
  power_list = '1 1'
[./]
```

LangmuirInhibition

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel is used to create a residual to represent an inhibition term for [InhibitedArrheniusReaction](#). The inhibition term is its own non-linear variable and is used in conjunction with other inhibition terms or a “Reaction” kernel to finish the full definition of the term in the MOOSE residual functions (see **Usage** below).

Residual Formulation

The residual is formed from a list of coupled concentration terms and a coupled temperature. The inhibition term does not have any units.

$$\text{Residual } (-) = -\varphi \cdot (1 + \sum_{\forall i} K_i C_i)$$

φ = MOOSE variable test function (for FE formulation)

C_i = concentration variable for species i

K_i = Langmuir coefficient for the i -th species in the list

Additional Computations

The Langmuir coefficients are calculated from the same type of expression as the Arrhenius reaction term.

$$K_i = A_i T^{\beta_i} \exp \left[-\frac{E_i}{RT} \right]$$

- The units for A_i must have the same units as K_i
- K_i must have units of inverse concentration C_i
- The units for E_i must be J/mol
- β_i are powers on temperature (which default to 0)

Usage

MUST be used in conjunction with a “Reaction” kernel from the MOOSE base system of kernels in order to fully describe the inhibition term (R) for a reaction.

Code below creates an inhibition term that is a function of concentrations for A and B. The coefficients (K) in the inhibition term are each set to a value of 1. This is accomplished by providing 0s for beta and activation energy terms, which removes the temperature dependence for those parameters.

```
[./R_equ]
  type = Reaction
  variable = R
[../]
[./R_lang]
  type = LangmuirInhibition
  variable = R
  temperature = T
  coupled_list = 'A B'
  pre_exponentials = '1 1'
  betas = '0 0'
  activation_energies = '0 0'
[../]
```

MaterialBalance

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel creates a residual for a generic mass/material balance on a set of non-linear variables. The point of this kernel is to reduce complexities in solving chemical reaction problems by closing the system of equations with a single kernel instead of having to create several sets of rate or equilibrium kernels for all chemical reactions. For instance, in the case of a simple adsorption reaction, the rate of adsorption is a function of the concentration of available sites. Those available sites will decrease overtime as adsorption continues. You can account for this decrease in site availability by invoking a set of rate kernels for both the adsorbed species and the surface sites. However, you can also just invoke rates for the adsorbed species, then close the system by invoking this material balance kernel to specific that the amount of available sites is a function of the initial site availability minus the amount of sites used by adsorption.

Residual Formulation

$$\text{Residual (same units for all coupled variables)} = \varphi \cdot (C_T - \sum_i w_i C_i)$$

φ = MOOSE variable test function (for FE formulation)

C_T = total concentration variable

C_i = concentration variable for species i

w_i = weight factor for amount of C_i that contributes to C_T

Additional Computations → None

Usage

Generally, this kernel is used in combination with other reaction or transport kernels on the variables of interest. You use this kernel to completely solve for 1 other variable in your domain that you do not have any other kernels for.

For example, consider an adsorption process wherein we have solved for the concentration of the adsorbate (C) and adsorbed concentration (q) with other sets of kernels. The rates of adsorption depends on a site density variable (S). We know that the maximum or total site density is S_{max} , so we can use MaterialBalance to solve for S . Here, we state that 1 mole of q adsorbed uses 2 reaction sites of S_{max} and S represents 1 mole of total sites.

```
[./site_dens]
  type = MaterialBalance
  variable = S
  this_variable = S
  total_material = Smax
  coupled_list = 'S q'
  weights = '1 2'
[../]
```

MicroscaleCoefTimeDerivative

Inheritance → [TimeDerivative](#)

Special Notes → All kernels preceded with a 'Microscale' prefix are experimental kernels developed specifically for setting up and solving a hybrid Finite Difference/Finite Element problem involving micro-scale subdomains in a macro-scale domain. The general idea is to discretize the micro-scale domain with finite differences, then use that discretization to create a finite element weak form in MOOSE. This then allows us to represent the residuals in MOOSE as a set of ODEs instead of needing to explicitly create small subdomains or MultiApp grids in MOOSE to solve micro-scale physics alongside the macro-scale domain.

Mathematical Expression of the Microscale (not all physics shown)

$$r^d R \frac{\partial u}{\partial t} = \frac{\partial}{\partial r} \left(r^d D \frac{\partial u}{\partial r} \right) - r^d K \frac{\partial v}{\partial t} + r^d \lambda \sum w_i r_i$$

r = dimension of the micro-scale (i.e., radius of a pellet)

d = conversion factor between coordinate systems for micro-scale

$d = 0$ → z-cartesian

$d = 1$ → r-cylindrical

$d = 2$ → r-spherical

R = time derivative coefficient

D = diffusion coefficient

K = coupled time derivative coefficient

λ = scale factor for coupled sum

w_i = weight factors in coupled sum

r_i = variable in coupled sum (generally represents a reaction rate)

u = primary variable (i.e., pore-space concentration)

v = coupled variable or physics (i.e., surface rxn)

→ All 'Microscale' kernels require the following parameters...

'micro_length' (i.e., length of the micro-scale domain, such as pellet radius)

'num_nodes' (i.e., the number of nodes to divide the micro-scale into)

'coord_id' (i.e., the conversion factor that is 0 for cartesian, 1 for cylindrical pellets, or 2 for spherical pellets)

The above parameters can be defined individually in each kernel or defined as GlobalParams, which is generally more appropriate.

[GlobalParams]

```

        micro_length = 1
        num_nodes = 3
        coord_id = 2
    []

```

Notes → This kernel is for the hybrid FD/FE representation of the time derivatives at all nodal locations in the micro-scale. Every node in the micro-scale requires an instance of this kernel (i.e., if you discretize the micro-scale into 10 nodes, you need 10 of these kernels).

Residual Formulation

$$\text{Residual (mass / volume / time)} = \varphi \cdot r_l^d R_l \frac{\partial(u_l)}{\partial t}$$

φ = MOOSE variable test function (for FE formulation)

r_l = nodal position in the micro-scale that this kernel acts on

R_l = nodal time coefficient for this kernel

l = node id for this position

u_l = conserved quantity variable at this nodal position

d = coordinate id (0 = z-cartesian, 1 = r-cylindrical, 2 = r-spherical)

Additional Computations → None

Usage

All ‘Microscale’ kernels must have an instance of themselves for each node you choose to divide the micro-scale into. Generally, you would use this in conjunction with a series of other ‘Microscale’ kernels with all the same parameters, but at different nodes.

Each different node requires a different nodal variable to be declared. For instance, if your total micro-scale quantity is being represented by u , and if you divide the micro-scale into 3 nodes, then your variables might be named u_0 , u_1 , and u_2 (i.e., value of u at node 0, value of u at node 1, and value of u at node 2).

Code example below is for the time derivatives of a micro-scale problem inside spherical pellets whose micro-scale domain was broken up into 3 nodes (0, 1, and 2). All time coefficients are assumed 1 at each nodal position.

```

[Kernels]
  [./u0_dot]
    type = MicroscaleCoefTimeDerivative
    variable = u0
    nodal_time_coef = 1
    node_id = 0
  [../]
  [./u1_dot]

```

```

        type = MicroscaleCoefTimeDerivative
        variable = u1
        nodal_time_coef = 1
        node_id = 1
    [../]
    [./u2_dot]
        type = MicroscaleCoefTimeDerivative
        variable = u2
        nodal_time_coef = 1
        node_id = 2
    [../]
[]

```

MicroscaleCoupledCoefTimeDerivative

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel is for introducing a coupled time derivative for mass transfer within a specified particle nodal location for the hybrid FD/FE method. Just like the primary micro-scale variable, all coupled micro-scale variables must also have a variable for each nodal position.

Residual Formulation

$$\text{Residual (mass / volume / time)} = \varphi \cdot r_l^d K_l \frac{\partial(v_l)}{\partial t}$$

φ = MOOSE variable test function (for FE formulation)

r_l = nodal position in the micro-scale that this kernel acts on

K_l = nodal time coefficient for the coupled time derivative

l = node id for this position

v_l = coupled variable at this nodal position

d = coordinate id (0 = z-cartesian, 1 = r-cylindrical, 2 = r-spherical)

Additional Computations → None

Usage

All ‘Microscale’ kernels must have an instance of themselves for each node you choose to divide the micro-scale into. Generally, you would use this in conjunction with a series of other ‘Microscale’ kernels with all the same parameters, but at different nodes.

Each different node requires a different nodal variable to be declared. For instance, if your total micro-scale quantity is being represented by u , and if you divide the micro-scale into 3 nodes, then your variables might be named u_0 , u_1 , and u_2 (i.e., value of u at node 0, etc).

The coupled time derivatives are kernels that act on your primary variable (in this example: u0, u1, and u2), but are coupled to the time derivatives of other variables in the micro-scale subdomain (in this example: v0, v1, and v2). MAKE SURE the correct variables are coupled together (e.g., in this example: u0 couples to v0, not any other v).

Code example below is for the coupled time derivatives of a micro-scale problem inside spherical pellets whose micro-scale domain was broken up into 3 nodes (0, 1, and 2). All time coefficients are assumed 1 at each nodal position.

```
[Kernels]
  [./v0_trans]
    type = MicroscaleCoupledCoefTimeDerivative
    variable = u0
    coupled_at_node = v0
    nodal_time_coef = 1
    node_id = 0
  [../]
  [./v1_trans]
    type = MicroscaleCoupledCoefTimeDerivative
    variable = u1
    coupled_at_node = v1
    nodal_time_coef = 1
    node_id = 1
  [../]
  [./v2_trans]
    type = MicroscaleCoupledCoefTimeDerivative
    variable = u2
    coupled_at_node = v2
    nodal_time_coef = 1
    node_id = 2
  [../]
[]
```

MicroscaleCoupledVariableCoefTimeDerivative

Inheritance → [MicroscaleCoupledCoefTimeDerivative](#)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel is for introducing a coupled time derivative for mass transfer within a specified particle nodal location for the hybrid FD/FE method. Just like the primary micro-scale variable, all coupled micro-scale variables must also have a variable for each nodal position. The parameter K_i from [MicroscaleCoupledCoefTimeDerivative](#) is now a non-linear variable, rather than a given constant.

Residual Formulation → (See [MicroscaleCoupledCoefTimeDerivative](#) for residual formulation)

Additional Computations → K_i is replaced with a non-linear variable coefficient

Usage

All ‘Microscale’ kernels must have an instance of themselves for each node you choose to divide the micro-scale into. Generally, you would use this in conjunction with a series of other ‘Microscale’ kernels with all the same parameters, but at different nodes.

Each different node requires a different nodal variable to be declared. For instance, if your total micro-scale quantity is being represented by u , and if you divide the micro-scale into 3 nodes, then your variables might be named u_0 , u_1 , and u_2 (i.e., value of u at node 0, etc).

The coupled time derivatives are kernels that act on your primary variable (in this example: u_0 , u_1 , and u_2), but are coupled to the time derivatives of other variables in the micro-scale subdomain (in this example: v_0 , v_1 , and v_2). MAKE SURE the correct variables are coupled together (e.g., in this example: u_0 couples to v_0 , not any other v).

Code example below is for the coupled time derivatives of a micro-scale problem inside spherical pellets whose micro-scale domain was broken up into 3 nodes (0, 1, and 2). The time coefficients are nodal variables named K_0 , K_1 , and K_2 at each node.

```
[Kernels]
  [./v0_trans]
    type = MicroscaleCoupledVariableCoefTimeDerivative
    variable = u0
    coupled_at_node = v0
    nodal_time_var = K0
    node_id = 0
  [../]
  [./v1_trans]
    type = MicroscaleCoupledVariableCoefTimeDerivative
    variable = u1
    coupled_at_node = v1
    nodal_time_var = K1
    node_id = 1
  [../]
  [./v2_trans]
    type = MicroscaleCoupledVariableCoefTimeDerivative
    variable = u2
    coupled_at_node = v2
    nodal_time_var = K2
    node_id = 2
  [../]
[]
```

MicroscaleDiffusion

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel sets up the diffusion portion of the residuals in the micro-scale using a Monotonic Upstream-centered Scheme for Conservation Laws (MUSCL) method. User must

invoke this kernel for each interior node in the micro-scale subdomain. For instance, if you have 3 nodes for the micro-scale, then there is 1 interior node and 2 boundary nodes. Boundary nodes must use the [MicroscaleDiffusionInnerBC](#) or [MicroscaleDiffusionOuterBC](#) kernels.

Residual Formulation

Residual (mass / volume / time)

$$= \varphi \cdot \left[\frac{r_{l+1/2}^d D_{l+1/2}}{(\Delta r)^2} \right] (u_l - u_{l+1}) + \varphi \cdot \left[\frac{r_{l-1/2}^d D_{l-1/2}}{(\Delta r)^2} \right] (u_l - u_{l-1})$$

φ = MOOSE variable test function (for FE formulation)

r_l = nodal position in the micro-scale that this kernel acts on

D_l = nodal diffusion coefficient for this kernel

l = node id for this position

u_l = conserved quantity variable at the l nodal position

u_{l-1} = conserved quantity variable at the $l-1$ nodal position

u_{l+1} = conserved quantity variable at the $l+1$ nodal position

d = coordinate id (0 = z-cartesian, 1 = r-cylindrical, 2 = r-spherical)

Additional Computations → None

Usage

Each interior node of the micro-scale needs a MicroscaleDiffusion kernel to represent the diffusive transport between the neighboring nodes in the micro-scale domain. The primary variable for a given MicroscaleDiffusion kernel should be the center node, then you couple to the other variables that represent the upper and lower neighbors respectively. Generally, you should order the nodal variables such that higher numbers are upper neighbors and lower numbers are lower neighbors.

Code example below is for the micro-scale diffusion occurring at node 1 in a 3 node system. The variables are u_0 , u_1 , and u_2 . We only invoke one instance of MicroscaleDiffusion since node 1 is the only interior node for this example. The diffusion coefficient is a single given constant.

This kernel must also be using in conjunction with the MicroscaleCoefTimeDerivative and/or MicroscaleCoupledCoefTimeDerivative for the u_1 variable.

[Kernels]

[./u1_diff]

type = MicroscaleDiffusion

variable = u1

node_id = 1

upper_neighbor = u2

lower_neighbor = u0

```

diffusion_const = 1
[../]
[]

```

MicroscaleDiffusionInnerBC

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel creates a combined residual for MicroscaleDiffusion in addition to applying a zero-flux boundary condition for the inner boundary in the micro-scale particles. The zero-flux boundary condition is a standard inner boundary condition for all micro-scale transport physics.

Mathematical Representation: $D \frac{\partial u}{\partial r} = 0$

Residual Formulation

Residual (mass / volume / time)

$$= \varphi \cdot \left[\frac{r_0^d D_0}{(\Delta r)^2} + \frac{r_{1/2}^d D_{1/2}}{(\Delta r)^2} \right] (u_0 - u_1)$$

φ = MOOSE variable test function (for FE formulation)

r_1 = nodal position in the micro-scale that this kernel acts on

D_1 = nodal diffusion coefficient for this kernel

0 = node id for this position

u_0 = conserved quantity variable at the 0 nodal position (boundary node)

u_1 = conserved quantity variable at the 1 nodal position (1st interior node)

d = coordinate id (0 = z-cartesian, 1 = r-cylindrical, 2 = r-spherical)

Additional Computations → None

Usage

The inner boundary node must use this kernel instead of the MicroscaleDiffusion kernel used for the interior nodes.

Code example below is for the micro-scale diffusion occurring at node 0 (i.e., the inner boundary node) in a 3 node system. The variables are u_0 , u_1 , and u_2 . This kernel only couples u_0 and u_1 since there is no lower neighbor at the interior.

This kernel must also be using in conjunction with the MicroscaleCoefTimeDerivative and/or MicroscaleCoupledCoefTimeDerivative for the u_1 variable.

```

[Kernels]
  [./u0_diff]

```

```

type = MicroscaleDiffusionInnerBC
variable = u0
node_id = 0
upper_neighbor = u1
diffusion_const = 1

[../]

[]

```

MicroscaleDiffusionOuterBC

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel creates a combined residual for MicroscaleDiffusion in addition to applying a mass-transfer flux boundary condition for the outer boundary in the micro-scale particles. The mass-transfer flux boundary condition is a standard outer boundary condition for all micro-scale transport physics. Dirichlet boundaries are not implemented because they are not relevant.

Mathematical Representation: $D \frac{\partial u}{\partial r} = k_m(u_b - u_L)$

Residual Formulation

Residual (mass / volume / time)

$$= \varphi \cdot \left[\frac{r_{L+1/2}^d 2k_m}{(\Delta r)} \right] (u_L - u_b) + \varphi \cdot \left[\frac{r_{L-1/2}^d D_{L-1/2}}{(\Delta r)^2} + \frac{r_{L+1/2}^d D_L}{(\Delta r)^2} \right] (u_L - u_{L-1})$$

φ = MOOSE variable test function (for FE formulation)

r_l = nodal position in the micro-scale that this kernel acts on

D_l = nodal diffusion coefficient for this kernel

L = node id for this position

u_b = conserved quantify in the macro-scale domain (i.e., bulk concentration)

u_L = conserved quantity variable at the L nodal position (boundary node)

u_{L-1} = conserved quantity variable at the $L-1$ nodal position ($L-1$ interior node)

d = coordinate id (0 = z-cartesian, 1 = r-cylindrical, 2 = r-spherical)

Additional Computations → None

Usage

The outer boundary node must use this kernel instead of the MicroscaleDiffusion kernel used for the interior nodes.

Code example below is for the micro-scale diffusion occurring at node 2 (i.e., the outer boundary node) in a 3 node system. The variables are u0, u1, and u2. This kernel only couples

u1, u2, and ub, where ub represents the coupled variable in the macro-scale to simulate transfer of mass from the macro-scale to the micro-scale.

This kernel must also be using in conjunction with the MicroscaleCoefTimeDerivative and/or MicroscaleCoupledCoefTimeDerivative for the u2 variable.

```
[Kernels]
  [./u2_diff]
    type = MicroscaleDiffusionOuterBC
    variable = u2
    node_id = 2
    macro_variable = ub
    lower_neighbor = u1
    diffusion_const = 1
  [../]
[]
```

MicroscaleScaledWeightedCoupledSumFunction

Inheritance → [ScaledWeightedCoupledSumFunction](#)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel is for introducing a scaled, weighted sum of non-linear variables into microscale kernel set. This generally will be used to introduce reaction variables into the microscale. It is much more convenient to use this to couple multiple reactions rather than to use several difference [MicroscaleCoupledCoefTimeDerivative](#) kernels to cross-coupled numerous reactions.

Residual Formulation

$$\text{Residual (mass / volume / time)} = -\varphi \cdot r_l^d \cdot \lambda \cdot \sum_{v_i} w_i v_i$$

φ = MOOSE variable test function (for FE formulation)

r_l = nodal position in the micro-scale that this kernel acts on

l = node id for this position

w_i = weight factor for the i-th non-linear variable

λ = coupled scaling factor for the variable sum

v_i = i-th coupled variable at this nodal position

d = coordinate id (0 = z-cartesian, 1 = r-cylindrical, 2 = r-spherical)

Additional Computations → None

Usage

All 'Microscale' kernels must have an instance of themselves for each node you choose to divide the micro-scale into. Generally, you would use this in conjunction with a series of other 'Microscale' kernels with all the same parameters, but at different nodes.

Each different node requires a different nodal variable to be declared. For instance, if your total micro-scale quantity is being represented by u , and if you divide the micro-scale into 3 nodes, then your variables might be named u_0 , u_1 , and u_2 (i.e., value of u at node 0, etc).

The scaled, weighted sum function is used to couple together a list of non-linear variables, that all exist on the microscale, to add them into the current variables' residual function. Every node in your microscale will likely need an instance of this sum kernel and the `node_id` values should line up for all non-linear variables.

In this example, we are coupling a reaction (r) at each node id (0, 1, 2) to the total residual of a non-linear variable (u). This is needed at each node and each reaction variable (r) is also discretized by microscale location (r_0 , r_1 , r_2). You can use the 'scale' as a constant or a non-linear variable to represent a unit conversion.

[Kernels]

`./u0_rxn`

```
        type = MicroscaleScaledWeightedCoupledSumFunction
        variable = u0
        node_id = 0
        coupled_list = 'r_0'
        weights = '-1'
        scale = 1
```

`[../]`

`./u1_rxn`

```
        type = MicroscaleScaledWeightedCoupledSumFunction
        variable = u1
        node_id = 1
        coupled_list = 'r_1'
        weights = '-1'
        scale = 1
```

`[../]`

`./u2_rxn`

```
        type = MicroscaleScaledWeightedCoupledSumFunction
        variable = u2
        node_id = 2
        coupled_list = 'r_2'
        weights = '-1'
        scale = 1
```

`[../]`

[]

MicroscaleVariableCoefTimeDerivative

Inheritance → [MicroscaleCoefTimeDerivative](#)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel is for introducing a time derivative for a microscale mass balance within a specified particle nodal location for the hybrid FD/FE method. The parameter R_i from [MicroscaleCoefTimeDerivative](#) is now a non-linear variable, rather than a given constant.

Residual Formulation → (See [MicroscaleCoefTimeDerivative](#) for residual formulation)

Additional Computations → R_i is replaced with a non-linear variable coefficient

Usage

All 'Microscale' kernels must have an instance of themselves for each node you choose to divide the micro-scale into. Generally, you would use this in conjunction with a series of other 'Microscale' kernels with all the same parameters, but at different nodes.

Each different node requires a different nodal variable to be declared. For instance, if your total micro-scale quantity is being represented by u , and if you divide the micro-scale into 3 nodes, then your variables might be named u_0 , u_1 , and u_2 (i.e., value of u at node 0, etc).

Code example below is for the variable time derivatives of a micro-scale problem inside spherical pellets whose micro-scale domain was broken up into 3 nodes (0, 1, and 2). The time coefficients are nodal variables named R_0 , R_1 , and R_2 at each node.

```
[Kernels]
  [./u0_dot]
    type = MicroscaleVariableCoefTimeDerivative
    variable = u0
    nodal_time_var = R0
    node_id = 0
  [../]
  [./u1_dot]
    type = MicroscaleVariableCoefTimeDerivative
    variable = u1
    nodal_time_var = R1
    node_id = 1
  [../]
  [./u2_dot]
    type = MicroscaleVariableCoefTimeDerivative
    variable = u2
    nodal_time_var = R2
    node_id = 2
  [../]
[]
```


MicroscaleVariableDiffusion

Inheritance → [MicroscaleDiffusion](#)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel sets up the diffusion portion of the residuals in the micro-scale using a Monotonic Upstream-centered Scheme for Conservation Laws (MUSCL) method. User must invoke this kernel for each interior node in the micro-scale subdomain. For instance, if you have 3 nodes for the micro-scale, then there is 1 interior node and 2 boundary nodes. Boundary nodes must use the [MicroscaleVariableDiffusionInnerBC](#) or [MicroscaleVariableDiffusionOuterBC](#) kernels. This version of the microscale diffusion kernel replaces the constant diffusion value from [MicroscaleDiffusion](#) with a set of variables representing the diffusion coefficients associated with this and all neighboring nodes in the microscale domain.

Residual Formulation → (See [MicroscaleDiffusion](#) for residual formulation)

Additional Computations

$$D_{l+1/2} = 0.5(D_{l+1} + D_l) \quad D_{l-1/2} = 0.5(D_{l-1} + D_l)$$

D_{l+1} = upper diffusion coefficient

D_l = current diffusion coefficient

D_{l-1} = lower diffusion coefficient

Usage

Each interior node of the micro-scale needs a MicroscaleVariableDiffusion kernel to represent the diffusive transport between the neighboring nodes in the micro-scale domain. The primary variable for a given MicroscaleVariableDiffusion kernel should be the center node, then you couple to the other variables that represent the upper and lower neighbors respectively. Generally, you should order the nodal variables such that higher numbers are upper neighbors and lower numbers are lower neighbors.

Code example below is for the micro-scale diffusion occurring at node 1 in a 3 node system. The variables are u0, u1, and u2. We only invoke one instance of MicroscaleDiffusion since node 1 is the only interior node for this example. The diffusion coefficients are given as D0, D1, and D3 for the variation in diffusion at their respective nodes.

This kernel must also be using in conjunction with the MicroscaleVariableCoefTimeDerivative and/or MicroscaleCoupledVariableCoefTimeDerivative for the u1 variable.

[Kernels]

```
[./u1_diff]
  type = MicroscaleVariableDiffusion
  variable = u1
  node_id = 1
  upper_neighbor = u2
```

```

        lower_neighbor = u0
        current_diff = D1
        upper_diff = D2
        lower_diff = D0
    [../]
[]

```

MicroscaleVariableDiffusionInnerBC

Inheritance → [MicroscaleDiffusionInnerBC](#)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel creates a combined residual for MicroscaleDiffusion in addition to applying a zero-flux boundary condition for the inner boundary in the micro-scale particles. The zero-flux boundary condition is a standard inner boundary condition for all micro-scale transport physics. The diffusion coefficients are replaced with diffusion variables for the nodes.

(See [MicroscaleDiffusionInnerBC](#) for additional notes)

Residual Formulation → (See [MicroscaleDiffusionInnerBC](#) for residual formulation)

Additional Computations

$$D_{1/2} = 0.5(D_1 + D_0)$$

D_1 = upper diffusion coefficient

D_0 = current diffusion coefficient

Usage

The inner boundary node must use this kernel instead of the MicroscaleVariableDiffusion kernel used for the interior nodes.

Code example below is for the micro-scale diffusion occurring at node 0 (i.e., the inner boundary node) in a 3 node system. The variables are u0, u1, and u2. This kernel only couples u0 and u1 since there is no lower neighbor at the interior.

This kernel must also be using in conjunction with the MicroscaleVariableCoefTimeDerivative and/or MicroscaleCoupledVariableCoefTimeDerivative for the u1 variable.

```

[Kernels]
  [./u0_diff]
    type = MicroscaleDiffusionInnerBC
    variable = u0
    node_id = 0
    upper_neighbor = u1
    current_diff = D0
    upper_diff = D1
  [../]
[]

```

MicroscaleVariableDiffusionOuterBC

Inheritance → [MicroscaleDiffusionOuterBC](#)

Special Notes → (See [MicroscaleCoefTimeDerivative](#) for special notes)

Notes → This kernel creates a combined residual for MicroscaleVariableDiffusion in addition to applying a mass-transfer flux boundary condition for the outer boundary in the micro-scale particles. The mass-transfer flux boundary condition is a standard outer boundary condition for all micro-scale transport physics. Dirichlet boundaries are not implemented because they are not relevant. Both the diffusion coefficients and mass transfer coefficients are variables instead of just constants as they were in the base class.

(See [MicroscaleDiffusionOuterBC](#) for additional notes)

Residual Formulation → (See [MicroscaleDiffusionOuterBC](#) for residual formulation)

Additional Computations → k_m is replaced with a non-linear variable

$$D_{L-1/2} = 0.5(D_{L-1} + D_L)$$

D_L = current diffusion coefficient

D_{L-1} = lower diffusion coefficient

Usage

The outer boundary node must use this kernel instead of the MicroscaleVariableDiffusion kernel used for the interior nodes.

Code example below is for the micro-scale diffusion occurring at node 2 (i.e., the outer boundary node) in a 3 node system. The variables are u_0 , u_1 , and u_2 . This kernel only couples u_1 , u_2 , and u_b , where u_b represents the coupled variable in the macro-scale to simulate transfer of mass from the macro-scale to the micro-scale.

Note: When coupling with the macro-scale, you would couple the outer boundary variable (u_2) with the bulk parameter (u_b) through the [FilmMassTransfer](#) kernel using the same variable for the mass transfer coefficient ($\text{rate_variable} = k_m$).

This kernel must also be using in conjunction with the MicroscaleVariableCoefTimeDerivative and/or MicroscaleCoupledVariableCoefTimeDerivative for the u_2 variable.

[Kernels]

[./u2_diff]

type = MicroscaleVariableDiffusionOuterBC

variable = u2

node_id = 2

macro_variable = u_b

lower_neighbor = u1

current_diff = D2

lower_diff = D1

rate_variable = k_m

[../]

[]

PairedLangmuirInhibition

Inheritance → [LangmuirInhibition](#)

Notes → This kernel is used to create a residual to represent an inhibition term for [InhibitedArrheniusReaction](#). The inhibition term is its own non-linear variable and is used in conjunction with other inhibition terms or a “Reaction” kernel to finish the full definition of the term in the MOOSE residual functions (see **Usage** below). The difference between this kernel and [LangmuirInhibition](#) is the inclusion of 2nd order coefficients and binary interactions between concentrations of species.

Residual Formulation

The residual is formed from a list of coupled concentration terms and a coupled temperature. The inhibition term does not have any units. In addition, lists of “paired” species (C_i and C_j) are given with associated “paired” Langmuir coefficients (K_{ij}).

$$\text{Residual } (-) = -\phi \cdot \left(1 + \sum_i K_i C_i + \sum_{i,j} K_{ij} C_i C_j \right)$$

ϕ = MOOSE variable test function (for FE formulation)

C_i = concentration variable for species i

K_i = Langmuir coefficient for the i -th species in the list

C_i, C_j = concentration variables for a pairing of i and j species

K_{ij} = associated Langmuir coefficients for the i, j pairs

Additional Computations

The Langmuir coefficients are calculated from the same expression in [LangmuirInhibition](#). In addition, the paired Langmuir coefficients (K_{ij}) are calculated with a similar expression.

$$K_{ij} = A_{ij} T^{\beta_{ij}} \exp \left[-\frac{E_{ij}}{RT} \right]$$

- The units for A_{ij} must have the same units as K_{ij}
- K_{ij} must have inverse units of the products of concentration C_i and C_j
- The units for E_{ij} must be J/mol
- β_{ij} are powers on temperature (which default to 0)

Usage

MUST be used in conjunction with a “Reaction” kernel from the MOOSE base system of kernels in order to fully describe the inhibition term (R) for a reaction.

Code below creates an inhibition term that is a function of concentrations for A and B, as well as a pairing of A and B. The coefficients (K) in the inhibition term are each set to a value of 1. This is accomplished by providing 0s for beta and activation energy terms, which removes the temperature dependence for those parameters.

```
[./R_equ]
    type = Reaction
    variable = R

[../]
[./R_lang]
    type = PairedLangmuirInhibition
    variable = R
    temperature = T
    coupled_list = 'A B'
    pre_exponentials = '1 1'
    betas = '0 0'
    activation_energies = '0 0'
    coupled_i_list = 'A'
    coupled_j_list = 'B'
    binary_pre_exp = '1'
    binary_betas = '0'
    binary_energies = '0'

[../]
```

PhaseEnergyTransfer

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes

→ This kernel facilitates the transfer of energy from one system phase to another. For instance, in a packed bed there is a solid phase and a fluid phase. If those two phases cannot be assumed locally isothermal, then each will have a different temperature and must transfer energy between each other through a heat transfer coefficient and effective area of contact. That area of contact will be a function of the volume fraction in the domain between the two phases and the specific area per volume of the phases.

→ Each phase will need an instance of this kernel to complete the energy transfer expression for each phase. In addition, this kernel is intended to act upon the energy density variable of each phase and not the temperature of each phase. It is coupled to temperature, but used to solve for internal energy.

Residual Formulation

$$\text{Residual (J / m}^3 \text{ / s)} = \varphi \cdot h \cdot f_v \cdot A_s \cdot (T_a - T_b)$$

φ = MOOSE variable test function (for FE formulation)

h = heat transfer rate (W/m²/K)

f_v = volume fraction between the phases (i.e., volume solids / total volume)

A_s = specific area of contact per volume of a phase (m⁻¹)

(i.e., surface area per volume of solids)

T_a = temperature of this phase energy variable (K)

T_b = temperature of the other phase energy variable (K)

Additional Computations → None

Usage

Generally used in conjunction with other kernels describing the conservation of energy in different phases. Each phase that is transferring energy needs to invoke this kernel in their respective energy balances. Note that the variable this kernel acts on is the energy density of the given phase, NOT the temperature of the phase. Temperature is a coupled variable.

Example below is for transfer of energy to the fluid phase energy density (E_f) from the solid phase. The parameters for heat transfer, specific area, and volume fraction are all constants. The temperature of this phase is the temperature of the fluid (T_f) and the other temperature is the temperature variable for the solid (T_s).

```
[./energy_trans]
  type = PhaseEnergyTransfer
  variable = Ef
  this_phase_temp = Tf
  other_phase_temp = Ts
  transfer_coef = 10
  volume_frac = 0.5
  specific_area = 1000
[../]
```

PhaseTemperature

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel is used in conjunction with an energy density variable and kernel set in order to calculate the temperature of the phase that the energy density variable represents. This is done because the energy balance is most logically performed on the energy density and not the temperature. Thus, it is necessary to add a kernel that can calculate the temperature for the phase, which is used in other computations. Temperature of a phase is based variables for the density of the phase, the specific heat of the phase, and the volume fraction of the phase.

Residual Formulation

$$\text{Residual (J / m}^3\text{)} = \varphi \cdot (\rho c T - E)$$

φ = MOOSE variable test function (for FE formulation)

ρ = density of the phase (kg/m³)

c = specific heat of the phase (J/kg/K)

T = temperature of the phase (K)

E = energy density of the phase (J/m^3)

Additional Computations → None

Usage

Generally used in conjunction with kernel that calculation energy density of a phase from conservation laws. Can also be combined with variables or auxiliary variables for the other parameters, such as density. The temperature is the primary variable for this kernel.

Example below calculates the temperature (T) of a phase based on the energy density (E) of that phase, as well as variables for all other properties of the phase.

```
[./phase_temp]
  type = PhaseTemperature
  variable = T
  energy = E
  specific_heat = cp
  density = rho
[../]
```

Reaction

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel is actually part of the MOOSE framework, but is discussed here since it is used to create and evaluate some reaction rate variables for complex reaction systems. It provides a residual for a basic, linear reaction with an optional coefficient (which we will not be using for most of our problems).

Residual Formulation

$$\text{Residual (mass / volume / time)} = \varphi \cdot \lambda u$$

φ = MOOSE variable test function (for FE formulation)

u = a non-linear variable (which will represent a reaction rate in our models)

λ = (optional) rate parameter/coefficient [we won't use this]

Additional Computations → None

Usage

Used whenever we want to create a non-linear variable for a reaction rate. Will be used in conjunction with a rate function such as [ArrheniusReaction](#) or [ConstReaction](#). For example, here we create non-linear variable 'r1' and specify that that reaction rate is evaluated as a simple forward reaction, e.g., $r1 = k_f \cdot C$, where $k_f = 1$.

That reaction rate can then be directly coupled into other mass balances or rate expressions.

```

[./r1_rxn]
    type = Reaction
    variable = r1
[./]
[./r1_rate]
    type = ConstReaction
    variable = r1
    this_variable = r1
    scale = 1
    forward_rate = 1
    reactants = 'C'
    reactant_stoich = '1'
[./]

```

ScaledWeightedCoupledSumFunction

Inheritance → [WeightedCoupledSumFunction](#)

Notes → This kernel inherits from [WeightedCoupledSumFunction](#) and can be used in the exact same way if needed. However, it adds to the summation a 'scale' factor variable that can be used as a way to convert units and/or couple between mass balances.

For example, in surface reaction problems, the binding of bulk phase molecules to a surface removes those molecules from that bulk phase. However, the amount removed is not directly a function of the surface reactions, but a 'scaled' representation of those reactions, wherein the scaling factor is based on a surface-to-volume or solids-to-volume ratio.

Example:
$$\varepsilon_b \frac{\partial C_{b,i}}{\partial t} + \nabla \cdot (\varepsilon_b v C_{b,i}) + (1 - \varepsilon_b) \sum_j w_j r_j = 0$$

In the above formulation, the factor of $(1 - \varepsilon_b)$ would be our 'scaling' factor that converts the rate of reactions from surface concentrations to bulk concentrations to finish the mass balance. The r_j are the reaction variables that represent each surface reaction and w_j are the weights (usually molar amounts) that that reaction contributes to the total effect on the bulk species.

Residual Formulation

Residual (same units as the variables given) $= -\varphi \cdot \lambda \cdot \sum_i w_i v_i$

φ = MOOSE variable test function (for FE formulation)

v_i = i^{th} coupled non-linear variable

w_i = i^{th} weight coefficient for the sum (can be positive or negative)

λ = scaling factor (can be a non-linear variable or a constant)

Additional Computations → None

Usage

Our example here is a continuation of the example from [WeightedCoupledSumFunction](#), wherein we have 3 reactions that contribute to a surface concentration 'q', which is resolved in the previous example. Here, we build on that example and demonstrate how to use [ScaledWeightedCoupledSumFunction](#) to account for the disappearance of bulk species 'A' and 'B' from those reactions. This kernel must be used in conjunction with other kernels describing the bulk behaviors of 'A' and 'B'.

```
# This kernel defines disappearance of 'A' from 'r1'
# Weights are negative to represent that 'A' is lost during these reactions
[./A_losses]
  type = ScaledWeightedCoupledSumFunction
  variable = A
  coupled_list = 'r1'
  weights = '-1'
  scale = 0.5
[../]

# This kernel defines disappearance of 'B' from 'r2'
# Weights are negative to represent that 'B' is lost during these reactions
[./B_losses]
  type = ScaledWeightedCoupledSumFunction
  variable = B
  coupled_list = 'r2'
  weights = '-1'
  scale = 0.5
[../]
```

TimeDerivative

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel is actually part of the MOOSE framework, but is discussed here since it is commonly used in all MOOSE modules/simulations. It provides a residual for a basic time derivative of the non-linear variable.

Residual Formulation

$$\text{Residual (mass / volume / time)} = \varphi \cdot \frac{\partial u}{\partial t}$$

φ = MOOSE variable test function (for FE formulation)

u = a non-linear variable that changes in time

Additional Computations → None

Usage

Used whenever a non-linear variable varies in time. Generally needs to be accompanied by other kernels that describe how the variable changes in time.

```
[./u_dot]
    type = TimeDerivative
    variable = u
[../]
```

VarSiteDensityExtLangModel

Inheritance → [ExtendedLangmuirModel](#)

Notes → This kernel uses the residuals from [ExtendedLangmuirModel](#), but allows the maximum adsorption capacity for the Langmuir model to be a non-linear variable. This way that maximum capacity can change as some other function of space and time. This is useful if we need to simulate aging effects that reduce the maximum site densities overtime. HOWEVER, this kernel is unnecessary as it is more efficient to just use a series of reaction kernels (such as [ArrheniusReaction](#) or [ArrheniusEquilibriumReaction](#) or even [MaterialBalance](#)) to account for variations in site density from aging.

Residual Formulation → (See [ExtendedLangmuirModel](#) for residual formulation)

Additional Computations → The maximum capacity parameter from [ExtendedLangmuirModel](#) is overridden to be the value of another non-linear variable provided.

Usage

Usage is essentially the same as [ExtendedLangmuirModel](#), but user also provides a non-linear variable for site density instead of a constant for maximum capacity.

Example below calculates the adsorption of species A to form q in the presence of species B, which is competing for the same adsorption sites as A.

```
[./q_res]
    type = Reaction
    variable = q
[../]
[./q_lang]
    type = VarSiteDensityExtLangModel
    variable = q
    coupled_site_density = qmax
    main_coupled = A
    coupled_list = 'A B'
    coupled_temp = T
    enthalpies = '-5000 -4000'
    entropies = '100 50'
[../]
```

VariableCoefTimeDerivative

Inheritance → CoefTimeDerivative (MOOSE framework kernel)

Notes → This kernel uses the existing residual calculations from CoefTimeDerivative (in the MOOSE framework), but overrides the time coefficient of that kernel to be another non-linear variable. This is useful for simulations that involve a time coefficient that might vary in space (such as a variable porosity in a porous media).

Residual Formulation

$$\text{Residual (mass / volume / time)} = \varphi \cdot R \frac{\partial u}{\partial t}$$

φ = MOOSE variable test function (for FE formulation)

u = a non-linear variable that changes in time

R = a non-linear variable for the time coefficient

Additional Computations → Overrides the base kernel parameter for the time coefficient

Usage

Used whenever a non-linear variable varies in time and has a non-linear variable for a time coefficient that varies in space (but not in time). Generally needs to be accompanied by other kernels that describe how the variable changes in time.

[./u_dot]

type = VariableCoefTimeDerivative

variable = u

coupled_coef = eps

[../]

VariableCoupledCoefTimeDerivative

Inheritance → [CoupledCoefTimeDerivative](#)

Notes → This kernel creates a specific form of [CoupledCoefTimeDerivative](#) wherein the concentration of a bulk fluid species is coupled to the time derivative of a solid or surface species in a particle. The time coefficient is the bulk density of solids in the column, which is a common defining parameter for packed bed reactors. That bulk density is used as a unit conversion from mass per volume of solids for the coupled variable to mass per volume of gases in the system.

Residual Formulation

(See [CoupledCoefTimeDerivative](#) for residual formulation)

Additional Computations

The time coefficient (a) for the [CoupledCoefTimeDerivative](#) is overridden to be as follows:

$a = \rho_b$ where ρ_b is the bulk density in the domain (mass solids / total volume)

This kernel still uses the “gaining” Boolean argument to determine whether or not to treat this transfer as a source or sink term. That option is depreciated, so you do not need to use it.

Usage

Must be used in conjunction with a set of kernels for the bulk variable and the coupled variable.

Code below would create a sink term for the variable Cb based on the rate of change of q:

```
[./coupled_dq_dt]
  type = VariableCoupledCoeffTimeDerivative
  variable = Cb
  coupled = q
  var_time_coeff = rho_b
[../]
```

VectorCoupledGradient

Inheritance → Kernel (MOOSE framework base kernel)

Notes → This kernel creates a way to couple one variable to the gradient of another variable through a vector dot product operation. The most basic way to use this kernel would be as a means to enforce continuity on a fluid (i.e., $\nabla \cdot \mathbf{u} = 0$, where \mathbf{u} is the velocity vector) or to calculate velocity based on a pressure gradient as in Darcy’s Law.

Residual Formulation

$$\text{Residual (mass / volume / time)} = \phi \cdot (\mathbf{v} \cdot \nabla p)$$

ϕ = MOOSE variable test function (for FE formulation)

p = a non-linear variable whose gradient we couple to

\mathbf{v} = a constant vector that is dotted with the coupled gradient

$$= \langle v_x, v_y, v_z \rangle$$

NOTE: User should supply each vector component in the input file.

Additional Computations → None

Usage

One such usage of this kernel is to enforce the condition of the continuity of flow in an incompressible fluid. We would do this in a piecewise manner and couple with each individual velocity variable component and give unit vectors for the specific direction that an individual piece is integrating on. In this example, the kernels act on the pressure variable and couple with velocity variables in each direction

Example 1: Continuity in 2D

```
[./p_x]
    type = VectorCoupledGradient
    variable = pressure
    coupled = vel_x
    vx = 1
    vy = 0
    vz = 0
```

```
[../]
[./p_y]
    type = VectorCoupledGradient
    variable = pressure
    coupled = vel_y
    vx = 0
    vy = 1
    vz = 0
```

```
[../]
```

Another way to use this kernel is Darcy's Law. According to Darcy's Law, the velocity in a porous media can be calculated as a function of the pressure gradient multiplied by a permeability coefficient (K).

$$\text{Darcy's Law: } \mathbf{u} = -K\nabla p$$

This statement is then completed by also providing pressure at the inlet and outlet of a domain and enforcing linearity of the pressure gradient (i.e., $\nabla^2 p = 0$).

NOTE: In the kernel, the velocity vector components (vx, vy, vz) would be a scalar multiple of the unit vector in the direction of interest. For example, if my permeability is 5, then replace 'vx' with 5 for the velocity in the x-direction and make all other velocity components 0.

Example 2: Darcy Flow in 2D

Force linear pressure gradient (no other active kernels for pressure)

```
[./p_lin]
    type = Diffusion
    variable = pressure
[../]
```

Weak form statement for velocity in x-direction

```
[./vel_x_equ]
    type = Reaction
    variable = vel_x
[../]
[./vel_x_Darcy]
    type = VectorCoupledGradient
    variable = vel_x
    coupled = pressure
    vx = 5
    vy = 0
```

```

        vz = 0
[../]

# Weak form statement for velocity in y-direction
[./vel_y_equ]
    type = Reaction
    variable = vel_y
[../]
[./vel_y_Darcy]
    type = VectorCoupledGradient
    variable = vel_y
    coupled = pressure
    vx = 0
    vy = 5
    vz = 0
[../]

```

VariableVectorCoupledGradient

Inheritance → [VectorCoupledGradient](#)

Notes → This kernel creates a way to couple one variable to the gradient of another variable through a vector dot product operation. The most basic way to use this kernel would be as a means to enforce continuity on a fluid (i.e., $\nabla \cdot \mathbf{u} = 0$, where \mathbf{u} is the velocity vector) or to calculate velocity based on a pressure gradient as in Darcy's Law.

Residual Formulation

$$\text{Residual (mass / volume / time)} = \varphi \cdot (\mathbf{v} \cdot \nabla p)$$

φ = MOOSE variable test function (for FE formulation)

p = a non-linear variable whose gradient we couple to

\mathbf{v} = a variable vector that is dotted with the coupled gradient

= $\langle u_x, u_y, u_z \rangle$ where each u_i can be its own non-linear variable.

NOTE: User should supply each vector component variable in the input file.

Additional Computations → None

Usage

The primary way to use this kernel is Darcy's Law. According to Darcy's Law, the velocity in a porous media can be calculated as a function of the pressure gradient multiplied by a permeability coefficient (K).

$$\text{Darcy's Law: } \mathbf{u} = -K \nabla p$$

This statement is then completed by also providing pressure at the inlet and outlet of a domain and enforcing linearity of the pressure gradient (i.e., $\nabla^2 p = 0$).

NOTE: In the kernel, the velocity vector components (ux, uy, uz) are variables that would represent permeability (K). For each velocity direction, provide 'K' in ONLY the direction that the kernel acts on (e.g., for velocity in x-direction, supply 'ux = K' and each other component, uy and uz, would be zero).

Example: Darcy Flow in 2D

```
# Force linear pressure gradient (no other active kernels for pressure)
```

```
[./p_lin]
```

```
    type = Diffusion
```

```
    variable = pressure
```

```
[../]
```

```
# Weak form statement for velocity in x-direction
```

```
[./vel_x_equ]
```

```
    type = Reaction
```

```
    variable = vel_x
```

```
[../]
```

```
[./vel_x_Darcy]
```

```
    type = VariableVectorCoupledGradient
```

```
    variable = vel_x
```

```
    coupled = pressure
```

```
    ux = K
```

```
    uy = 0
```

```
    uz = 0
```

```
[../]
```

```
# Weak form statement for velocity in y-direction
```

```
[./vel_y_equ]
```

```
    type = Reaction
```

```
    variable = vel_y
```

```
[../]
```

```
[./vel_y_Darcy]
```

```
    type = VariableVectorCoupledGradient
```

```
    variable = vel_y
```

```
    coupled = pressure
```

```
    ux = 0
```

```
    uy = K
```

```
    uz = 0
```

```
[../]
```

WeightedCoupledSumFunction

Inheritance → Kernel (i.e., the MOOSE Kernel base system)

Notes → This kernel is similar to [CoupledSumFunction](#), but does not inherit from it. It creates a residual for a weighted sum of other coupled variables. The variable that this kernel acts on should NOT be in the list of other coupled variables. Most common usage of this kernel would be in conjunction with the MOOSE standard [Reaction](#) kernel, [TimeDerivative](#) kernel, or [VariableCoefTimeDerivative](#) kernel. The culmination of this kernel with one of the previous will create an equation such as shown below.

Example Problem: $dq/dt = r_1 + r_2 - r_3$

To setup and solve this problem, we create variables for 'r1', 'r2', 'r3', and 'q'. Each reaction variable is setup using the [Reaction](#) kernel with an associated [ArrheniusReaction](#), [ConstReaction](#), and/or [ArrheniusEquilibriumReaction](#). Then, the 'q' variable is setup as a combination of a [TimeDerivative](#) kernel and this kernel, the [WeightedCoupledSumFunction](#). See [Usage](#) below for input file formatting.

Residual Formulation

Residual (same units as the variables given) $= -\phi \cdot \sum_i w_i v_i$

ϕ = MOOSE variable test function (for FE formulation)

v_i = i^{th} coupled non-linear variable

w_i = i^{th} weight coefficient for the sum (can be positive or negative)

Additional Computations → None

Usage

Our example here highlights how to setup the example problem described above in [Notes](#). The rate functions for each reaction will be [ConstReaction](#) time kernels just to keep the example relatively simple.

```
# Define r1: r1 = 1*C
[/r1_rxn]
  type = Reaction
  variable = r1
[../]
[/r1_rate]
  type = ConstReaction
  variable = r1
  this_variable = r1
  scale = 1
  forward_rate = 1
  reactants = 'C'
  reactant_stoich = '1'
[../]

# Define r2: r2 = 0.5*B
[/r2_rxn]
```



```

        type = Reaction
        variable = r2
[../]
[./r2_rate]
    type = ConstReaction
    variable = r2
    this_variable = r2
    scale = 1
    forward_rate = 0.5
    reactants = 'B'
    reactant_stoich = '1'
[../]

# Define r3:  r3 = 2.5*q
[./r3_rxn]
    type = Reaction
    variable = r3
[../]
[./r3_rate]
    type = ConstReaction
    variable = r3
    this_variable = r3
    scale = 1
    forward_rate = 2.5
    reactants = 'q'
    reactant_stoich = '1'
[../]

# Define q:      dq/dt = r1 + r2 - r3 == 1*C + 0.5*B - 2.5*q
[./q_dot]
    type = TimeDerivative
    variable = q
[../]
[./q_rate_sum]
    type = WeightedCoupledSumFunction
    variable = q
    coupled_list = 'r1 r2 r3'
    weights = '1 1 -1'
[../]

```

Boundary Conditions

Boundary conditions are required for any physics that involves either motion or spatially variant non-linear residual kernels (such as diffusion or advection kernels). In DG methods, it is impossible to provide Dirichlet type boundary conditions because solutions are not defined on nodes, but elements instead. However, we can emulate Dirichlet type boundary conditions if necessary.

In CATS, there are 2 types of boundaries: (i) 'Flux' boundary and (ii) 'FluxLimited' boundary. The 'Flux' boundaries should be employed for any "open" type of boundary conditions where energy or mass is flowing into the domain carried by a velocity (or heat transfer coefficient). The 'FluxLimited' boundaries use the same parametric information as their 'Flux' counter-parts, but are used to emulate a Dirichlet type boundary. In general, it is recommended to use the 'Flux' type boundary conditions unless you absolutely need a Dirichlet boundary condition or if simulations are oscillatory.

All specialized boundary conditions inherit from either DGFluxBC or DGFluxLimitedBC. Those are the most generic of the boundaries. The other kernels create more specific conditions at the boundaries for a variety of different situations such as variable velocities or porosities, etc.

DGConcFluxLimitedStepwiseBC

Inheritance → [DGConcentrationFluxLimitedBC](#)

Notes → This kernel uses the residuals form [DGConcentrationFluxLimitedBC](#), but changes the `u_input` term in a stepwise fashion based on a list of values to update it to and a corresponding list of times when it gets updated. Optionally, the user can provide another list of arguments for the span of time over which the stepwise increase occurs. This allows for the simulation to slowly ramp up the inlet values from its previous time value to its future time value.

(See [DGConcentrationFluxLimitedBC](#) for additional notes)

Residual Formulation → (See [DGConcentrationFluxLimitedBC](#) for residual formulation)

Additional Computations → Value of `u_input` is updated at specified times from the user.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (`u`) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to. Since this is a 'FluxLimited' boundary, you must also provide diffusion values to fill out the penalty terms.

The value of `u_input` will start out as 1, then be stepped down to 0.5 at a time of 1, then down to 0.25 at a time of 2. During stepdown, the `u_input` will gradually change for a period of 0.1 time units centered around each time when the step changes occur.

```
[./flux]
    type = DGConcFluxLimitedStepwiseBC
    variable = u
    boundary = 'inlet outlet'
    u_input = 1
    input_vals = '0.5 0.25'
    input_times = '1 2'
    time_spans = '0.1 0.1'
    ux = vel_x
    uy = vel_y
    uz = vel_z
    Dxx = 1
    Dyy = 1
    Dzz = 1
[../]
```

DGConcFluxStepwiseBC

Inheritance → [DGConcentrationFluxBC](#)

Notes → This kernel uses the residuals form [DGConcentrationFluxBC](#), but changes the `u_input` term in a stepwise fashion based on a list of values to update it to and a corresponding list of times when it gets updated. Optionally, the user can provide another list of arguments for the span of time over which the stepwise increase occurs. This allows for the simulation to slowly ramp up the inlet values from its previous time value to its future time value.

(See [DGConcentrationFluxBC](#) for additional notes)

Residual Formulation → (See [DGConcentrationFluxBC](#) for residual formulation)

Additional Computations → Value of `u_input` is updated at specified times from the user.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (`u`) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to.

The value of `u_input` will start out as 1, then be stepped down to 0.5 at a time of 1, then down to 0.25 at a time of 2. During stepdown, the `u_input` will gradually change for a period of 0.1 time units centered around each time when the step changes occur.

```
[./flux]
    type = DGFluxStepwiseBC
    variable = u
    boundary = 'inlet outlet'
    u_input = 1
```

```

input_vals = '0.5 0.25'
input_times = '1 2'
time_spans = '0.1 0.1'
ux = vel_x
uy = vel_y
uz = vel_z

[../]

```

DGConcentrationFluxBC

Inheritance → [DGFluxBC](#)

Notes → This kernel uses the residuals form [DGFluxBC](#), but overrides the velocity vector with variables for the velocity in each component direction. This is most useful when the velocity field is calculated by other kernels or MOOSE modules, such as Navier-Stokes.

(See [DGFluxBC](#) for additional notes)

Residual Formulation → (See [DGFluxBC](#) for residual formulation)

Additional Computations → The velocity vector is overridden with new variable components.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to.

```

[./flux]
  type = DGConcentrationFluxBC
  variable = u
  boundary = 'inlet outlet'
  u_input = 1
  ux = vel_x
  uy = vel_y
  uz = vel_z

[../]

```

DGConcentrationFluxLimitedBC

Inheritance → [DGFluxLimitedBC](#)

Notes → This kernel uses the residuals form [DGFluxLimitedBC](#), but overrides the velocity vector with variables for the velocity in each component direction. This is most useful when the velocity field is calculated by other kernels or MOOSE modules, such as Navier-Stokes.

(See [DGFluxLimitedBC](#) for additional notes)

Residual Formulation → (See [DGFluxLimitedBC](#) for residual formulation)

Additional Computations → The velocity vector is overridden with new variable components.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to. Since this is a 'FluxLimited' boundary, you must also provide diffusion values to fill out the penalty terms.

```
[./flux]
  type = DGConcentrationFluxLimitedBC
  variable = u
  boundary = 'inlet outlet'
  u_input = 1
  ux = vel_x
  uy = vel_y
  uz = vel_z
  Dxx = 1
  Dyy = 1
  Dzz = 1
[../]
```

DGFlowEnergyFluxBC

Inheritance → [DGConcentrationFluxBC](#)

Notes → This kernel uses the residuals form [DGConcentrationFluxBC](#), but is specific for an energy balance where the conserved quantity is the energy density of a specific phase in the domain. The flux of energy into an open boundary is dictated by the velocity variable that carries the fluid phase into the domain. Then, the energy density that is carried into, or out of, the domain is a function of the variables for phase density, phase porosity, phase specific heat, and phase temperature.

(See [DGConcentrationFluxBC](#) for additional notes)

Residual Formulation

Note: The residuals are integrated across the area of the boundary.

Residual (J / m² / s) =

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) > 0 \quad \varphi \cdot (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot E$$

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) \leq 0 \quad \varphi \cdot (\varepsilon \mathbf{v} \cdot \mathbf{n}) \cdot (\rho c T_{inlet})$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector at the boundaries

\mathbf{n} = normal vector at the boundaries

E = non-linear variable for energy density (J/m^3)

ε = domain porosity (volume voids per total volume)

ρ = phase density (kg/m^3)

c = phase specific heat (J/kg/m^3)

T_{inlet} = input value for the temperature at the boundary (K)

Additional Computations → None

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the energy density variable (E) is calculated at the inlet base on an inlet temperature (T_{in}) of 298 K along with the porosity, density, and specific heats given at the inlet. The user must provide a list of the names of the boundaries that this kernel applies to.

```
[./flux]
  type = DGFlowEnergyFluxBC
  variable = E
  boundary = 'inlet outlet'
  porosity = 0.5
  specific_heat = 1000
  density = 1
  inlet_temp = 298
  ux = vel_x
  uy = vel_y
  uz = vel_z
[./]
```

DGFluxBC

Inheritance → IntegratedBC (i.e., the MOOSE base kernel for flux-based boundaries)

Notes → This kernel is used for both inlet and outlet boundaries for the flow of mass or energy carried into and out of a domain through an advective process. It should be invoked at all “open” boundaries. It uses the velocity vector and the normal vector to the boundary to determine whether or not mass/energy is flowing into or out of the system. When it finds that the boundary is an outlet, it uses the interior values of the non-linear variable to establish exit flux. When it finds that the boundary is an inlet, it uses the user specified inlet value to establish flux of mass/energy into the domain.

Residual Formulation

Note: The residuals are integrated across the area of the boundary.

Residual (mass / area / time) =

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) > 0 \quad \varphi \cdot (\mathbf{v} \cdot \mathbf{n}) \cdot u$$

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) \leq 0 \quad \varphi \cdot (\mathbf{v} \cdot \mathbf{n}) \cdot u_{inlet}$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector at the boundaries

\mathbf{n} = normal vector at the boundaries

u = non-linear variable at the interior of the domain

u_{inlet} = input value for the variable at an inlet

Additional Computations → None

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries (i.e., any open boundaries in the domain). The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a constant velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to.

```
[./flux]
  type = DGFluxBC
  variable = u
  boundary = 'inlet outlet'
  u_input = 1
  vx = 1
  vy = 0
  vz = 0
[../]
```

DGFluxLimitedBC

Inheritance → IntegratedBC (i.e., the MOOSE base kernel for flux-based boundaries)

Notes → This kernel is used for both inlet and outlet boundaries for the flow of mass or energy carried into and out of a domain through an advective process. The outlet boundary will inherently use advective flux at the exit, but the inlet boundary will attempt to emulate a Dirichlet type boundary condition. This emulation requires the same parameter arguments from [GAnisotropicDiffusion](#) ([DGAnisotropicDiffusion](#)) in order to penalize variation at the inlet from

the given inlet value. Thus, the kernel attempts to enforce the non-linear variable to a given constant inlet value based on the DG formulation and penalty terms.

(See [GAnisotropicDiffusion \(DGAnisotropicDiffusion\)](#) for special notes on penalty terms)

Residual Formulation

Note: The residuals are integrated across the area of the boundary.

Note: The formulation of the inlet boundary is very complex and long, so the full residual expression is not show here.

Residual (mass / area / time) =

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) > 0 \quad \varphi \cdot (\mathbf{v} \cdot \mathbf{n}) \cdot u$$

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) \leq 0 \quad \approx \varphi \cdot p \cdot (u - u_{inlet})$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector at the boundaries

\mathbf{n} = normal vector at the boundaries

p = penalty term based on diffusion coefficients and other

[DGAnisotropicDiffusion](#) parameter arguments

u = non-linear variable at the interior of the domain

u_{inlet} = input value for the variable at an inlet

Additional Computations → None

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a constant velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to. Since this is a 'FluxLimited' boundary, you must also provide diffusion values to fill out the penalty terms.

[./flux]

```
type = DGFluxLimitedBC
variable = u
boundary = 'inlet outlet'
u_input = 1
vx = 1
vy = 0
vz = 0
Dxx = 1
```



```

        Dyy = 1
        Dzz = 1
[../]

```

DGFluxLimitedStepwiseBC

Inheritance → [DGFluxLimitedBC](#)

Notes → This kernel uses the residuals form [DGFluxLimitedBC](#), but changes the `u_input` term in a stepwise fashion based on a list of values to update it to and a corresponding list of times when it gets updated. Optionally, the user can provide another list of arguments for the span of time over which the stepwise increase occurs. This allows for the simulation to slowly ramp up the inlet values from its previous time value to its future time value.

(See [DGFluxLimitedBC](#) for additional notes)

Residual Formulation → (See [DGFluxLimitedBC](#) for residual formulation)

Additional Computations → Value of `u_input` is updated at specified times from the user.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a constant velocity vector (given component-wise) where the non-linear variable (`u`) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to. Since this is a 'FluxLimited' boundary, you must also provide diffusion values to fill out the penalty terms.

The value of `u_input` will start out as 1, then be stepped down to 0.5 at a time of 1, then down to 0.25 at a time of 2. During stepdown, the `u_input` will gradually change for a period of 0.1 time units centered around each time when the step changes occur.

```

[./flux]
    type = DGFluxLimitedStepwiseBC
    variable = u
    boundary = 'inlet outlet'
    u_input = 1
    input_vals = '0.5 0.25'
    input_times = '1 2'
    time_spans = '0.1 0.1'
    vx = 1
    vy = 0
    vz = 0
    Dxx = 1
    Dyy = 1
    Dzz = 1
[../]

```

DGFluxStepwiseBC

Inheritance → [DGFluxBC](#)

Notes → This kernel uses the residuals form [DGFluxBC](#), but changes the `u_input` term in a stepwise fashion based on a list of values to update it to and a corresponding list of times when it gets updated. Optionally, the user can provide another list of arguments for the span of time over which the stepwise increase occurs. This allows for the simulation to slowly ramp up the inlet values from its previous time value to its future time value.

(See [DGFluxBC](#) for additional notes)

Residual Formulation → (See [DGFluxBC](#) for residual formulation)

Additional Computations → Value of `u_input` is updated at specified times from the user.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a constant velocity vector (given component-wise) where the non-linear variable (`u`) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to.

The value of `u_input` will start out as 1, then be stepped down to 0.5 at a time of 1, then down to 0.25 at a time of 2. During stepdown, the `u_input` will gradually change for a period of 0.1 time units centered around each time when the step changes occur.

```
[./flux]
    type = DGFluxStepwiseBC
    variable = u
    boundary = 'inlet outlet'
    u_input = 1
    input_vals = '0.5 0.25'
    input_times = '1 2'
    time_spans = '0.1 0.1'
    vx = 1
    vy = 0
    vz = 0
[../]
```

DGNSMomentumOutflowBC

Inheritance → [DGConcentrationFluxBC](#)

Notes → This kernel uses the residuals form [DGConcentrationFluxBC](#), but scales the flux by the density of the domain and modifies the Jacobian elements to account for the fact that the variable we are coupling to is also a variable in the velocity vector. This is a boundary condition that should apply to ANY outlet/open boundary for conservation of momentum using the DG implementation of Incompressible Navier-Stokes equations.

(See [DGConcentrationFluxBC](#) for additional notes)

Residual Formulation

Note: The residuals are integrated across the area of the boundary.

Residual (momentum / area / time) =

for $(\mathbf{v} \cdot \mathbf{n}) > 0$ $residual = \varphi \cdot (\mathbf{v} \cdot \mathbf{n}) \cdot (\rho u_i)$

for $(\mathbf{v} \cdot \mathbf{n}) \leq 0$ $residual = 0$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector at the boundaries

[This vector will hold variables for velocity in each direction]

\mathbf{n} = normal vector at the boundaries

ρ = fluid density (mass / volume)

u_i = non-linear variable for velocity in the i-th direction

where i = x, y, or z

Additional Computations → None

Usage

Generally, whenever you use the DG implementation of Navier-Stokes equations, you need to allow the momentum in the domain to leave out the open boundary. This kernel will apply that boundary condition to each velocity variable (one at a time). Thus, for each velocity variable, you MUST invoke this BC kernel at EACH open boundary.

```
[./momentum_flux_in_x]
  type = DGNSMomentumOutflowBC
  variable = vel_x
  this_variable = vel_x
  boundary = 'outlet'
  density = rho
  ux = vel_x
  uy = vel_y
  uz = vel_z
[./]
```

[DGPoreConcFluxBC_ppm](#)

Inheritance → [DGPoreConcFluxBC](#)

Notes → This kernel uses the residuals form [DGPoreConcFluxBC](#), but changes the inlet values from a user provided value in ppm to a moles per liter (mol/L) inlet concentration using ideal gas law and coupled variables for pressure and temperature at the boundary. This kernel is very

convenient for simulations in which your inlet temperatures and/or pressure are changing, but you want to specify a constant mole-fraction or volume-per-volume inlet as constant.

(See [DGPoreConcFluxBC](#) for additional notes)

Residual Formulation

Note: The residuals are integrated across the area of the boundary.

Residual (mass / area / time) = See [DGPoreConcFluxBC](#)

Additional Computations → The value of u_{inlet} (see [DGPoreConcFluxBC](#)) is calculated from the user provided inlet in ppm as well as variables for temperature (T) and pressure (P) using ideal gas law.

$$u_{inlet} = \frac{P \cdot (y_{ppm}/10^6)}{RT}$$

P = inlet pressure (in kPa)

T = inlet temperature (in K)

R = gas law constant = 8.3145 L*kPa/mol/K

y_{ppm} = inlet concentration in ppm

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1000 ppmv. In addition, the porosity of the domain is a variable named eps and the temperature is a variable named temperature. The user must provide a list of the names of the boundaries that this kernel applies to.

```
[./flux]
  type = DGPoreConcFluxBC
  variable = u
  boundary = 'inlet outlet'
  porosity = eps
  ux = vel_x
  uy = vel_y
  uz = vel_z
  pressure = 101.35
  temperature = temp
  inlet_ppm = 1000
[../]
```

DGPoreConcFluxBC

Inheritance → [DGConcentrationFluxBC](#)

Notes → This kernel uses the residuals form [DGConcentrationFluxBC](#), but scales the flux by the porosity of the domain. Thus, this kernel is most appropriate for boundary conditions for porous flow or packed bed domains.

(See [DGConcentrationFluxBC](#) for additional notes)

Residual Formulation

Note: The residuals are integrated across the area of the boundary.

Residual (mass / area / time) =

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) > 0 \quad \varphi \cdot (\mathbf{v} \cdot \mathbf{n}) \cdot (\varepsilon u)$$

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) \leq 0 \quad \varphi \cdot (\mathbf{v} \cdot \mathbf{n}) \cdot (\varepsilon u_{inlet})$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector at the boundaries

\mathbf{n} = normal vector at the boundaries

ε = domain porosity (volume voids per total volume)

u = non-linear variable for concentration (mass/volume)

u_{inlet} = input value for concentration (mass/volume)

Additional Computations → None

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1. In addition, the porosity of the domain is a variable named eps . The user must provide a list of the names of the boundaries that this kernel applies to.

```
[./flux]
  type = DGPoreConcFluxBC
  variable = u
  boundary = 'inlet outlet'
  porosity = eps
  u_input = 1
  ux = vel_x
  uy = vel_y
  uz = vel_z
[../]
```

DGPoreConcFluxStepwiseBC

Inheritance → [DGPoreConcFluxBC](#)

Notes → This kernel uses the residuals form [DGPoreConcFluxBC](#), but changes the `u_input` term in a stepwise fashion based on a list of values to update it to and a corresponding list of times when it gets updated. Optionally, the user can provide another list of arguments for the span of time over which the stepwise increase occurs. This allows for the simulation to slowly ramp up the inlet values from its previous time value to its future time value.

(See [DGPoreConcFluxBC](#) for additional notes)

Residual Formulation → (See [DGPoreConcFluxBC](#) for residual formulation)

Additional Computations → Value of `u_input` is updated at specified times from the user.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (`u`) has an inlet value of 1. In addition, the porosity of the domain is a variable named `eps`. The user must provide a list of the names of the boundaries that this kernel applies to.

The value of `u_input` will start out as 1, then be stepped down to 0.5 at a time of 1, then down to 0.25 at a time of 2. During stepdown, the `u_input` will gradually change for a period of 0.1 time units centered around each time when the step changes occur.

```
[./flux]
    type = DGPoreConcFluxStepwiseBC
    variable = u
    porosity = eps
    boundary = 'inlet outlet'
    u_input = 1
    input_vals = '0.5 0.25'
    input_times = '1 2'
    time_spans = '0.1 0.1'
    ux = vel_x
    uy = vel_y
    uz = vel_z
[./]
```

DGPoreDiffFluxLimitedBC

Inheritance → [DGVarVelDiffFluxLimitedBC](#)

Notes → This kernel uses the residuals form [DGVarVelDiffFluxLimitedBC](#), but scales the flux by the porosity of the domain. Thus, this kernel is most appropriate for boundary conditions for porous flow or packed bed domains. Note that this is the version of the flux that emulates the Dirichlet boundary condition.

(See [DGVarVelDiffFluxLimitedBC](#) for additional notes)

(See [GAnisotropicDiffusion \(DGAnisotropicDiffusion\)](#) for notes on penalty terms)

Residual Formulation

Note: The residuals are integrated across the area of the boundary.

Residual (mass / area / time) =

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) > 0 \quad \varphi \cdot (\mathbf{v} \cdot \mathbf{n}) \cdot (\varepsilon u)$$

$$\text{for } (\mathbf{v} \cdot \mathbf{n}) \leq 0 \quad \approx \varphi \cdot p \cdot (u - u_{inlet})$$

φ = MOOSE variable test function (for FE formulation)

\mathbf{v} = velocity vector at the boundaries

\mathbf{n} = normal vector at the boundaries

ε = domain porosity (volume voids per total volume)

p = penalty term applied to emulate Dirichlet boundary condition

u = non-linear variable for concentration (mass/volume)

u_{inlet} = input value for concentration (mass/volume)

Additional Computations → None

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1. In addition, the porosity of the domain is a variable named eps . The user must provide a list of the names of the boundaries that this kernel applies to.

```
[./flux]
  type = DGPoreDiffFluxLimitedBC
  variable = u
  boundary = 'inlet outlet'
  porosity = eps
  u_input = 1
  ux = vel_x
  uy = vel_y
  uz = vel_z
  Dx = Diff
  Dy = Diff
  Dz = Diff
[../]
```

DGPoreDiffFluxLimitedStepwiseBC

Inheritance → [DGPoreDiffFluxLimitedBC](#)

Notes → This kernel uses the residuals form [DGPoreDiffFluxLimitedBC](#), but changes the `u_input` term in a stepwise fashion based on a list of values to update it to and a corresponding list of times when it gets updated. Optionally, the user can provide another list of arguments for the span of time over which the stepwise increase occurs. This allows for the simulation to slowly ramp up the inlet values from its previous time value to its future time value.

(See [DGPoreDiffFluxLimitedBC](#) for additional notes)

Residual Formulation → (See [DGPoreDiffFluxLimitedBC](#) for residual formulation)

Additional Computations → Value of `u_input` is updated at specified times from the user.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a constant velocity vector (given component-wise) where the non-linear variable (`u`) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to. In addition, the porosity of the domain is a variable named `eps`. Since this is a 'FluxLimited' boundary, you must also provide diffusion variables to fill out the penalty terms.

The value of `u_input` will start out as 1, then be stepped down to 0.5 at a time of 1, then down to 0.25 at a time of 2. During stepdown, the `u_input` will gradually change for a period of 0.1 time units centered around each time when the step changes occur.

```
[./flux]
    type = DGPoreDiffFluxLimitedStepwiseBC
    variable = u
    porosity = eps
    boundary = 'inlet outlet'
    u_input = 1
    input_vals = '0.5 0.25'
    input_times = '1 2'
    time_spans = '0.1 0.1'
    ux = vel_x
    uy = vel_y
    uz = vel_z
    Dx = Diff
    Dy = Diff
    Dz = Diff
[../]
```


DGVarVelDiffFluxLimitedBC

Inheritance → [DGConcentrationFluxLimitedBC](#)

Notes → This kernel uses the residuals form [DGConcentrationFluxLimitedBC](#), but allows the diffusion parameter to be a function of other non-linear variables for each x, y, z direction.

(See [DGConcentrationFluxLimitedBC](#) for additional notes)

(See [GVariableDiffusion \(DGVariableDiffusion\)](#) for notes on variable diffusion)

Residual Formulation → (See [DGConcentrationFluxLimitedBC](#) for residual formulation)

Additional Computations → Value of diffusion tensor replaced with variables Dx, Dy, and Dz.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (u) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to. Since this is a 'FluxLimited' boundary, you must also provide diffusion values to fill out the penalty terms. In this formulation, the diffusion values are other variables.

```
[./flux]
  type = DGVarVelDiffFluxLimitedBC
  variable = u
  boundary = 'inlet outlet'
  u_input = 1
  ux = vel_x
  uy = vel_y
  uz = vel_z
  Dx = Diff
  Dy = Diff
  Dz = Diff
[../]
```

DGVarVelDiffFluxLimitedStepwiseBC

Inheritance → [DGVarVelDiffFluxLimitedBC](#)

Notes → This kernel uses the residuals form [DGVarVelDiffFluxLimitedBC](#), but changes the u_input term in a stepwise fashion based on a list of values to update it to and a corresponding list of times when it gets updated. Optionally, the user can provide another list of arguments for the span of time over which the stepwise increase occurs. This allows for the simulation to slowly ramp up the inlet values from its previous time value to its future time value.

(See [DGVarVelDiffFluxLimitedBC](#) for additional notes)

Residual Formulation → (See [DGVarVelDiffFluxLimitedBC](#) for residual formulation)

Additional Computations → Value of `u_input` is updated at specified times from the user.

Usage

Generally you invoke this same kernel for both inlet and outlet boundaries. The kernel knows implicitly whether the boundary is an inlet or outlet based on the velocity vector given.

This example is for a variable velocity vector (given component-wise) where the non-linear variable (`u`) has an inlet value of 1. The user must provide a list of the names of the boundaries that this kernel applies to. Since this is a 'FluxLimited' boundary, you must also provide diffusion values to fill out the penalty terms. In this formulation, the diffusion values are other variables.

The value of `u_input` will start out as 1, then be stepped down to 0.5 at a time of 1, then down to 0.25 at a time of 2. During stepdown, the `u_input` will gradually change for a period of 0.1 time units centered around each time when the step changes occur.

```
[./flux]
  type = DGVarVelDiffFluxLimitedStepwiseBC
  variable = u
  boundary = 'inlet outlet'
  u_input = 1
  input_vals = '0.5 0.25'
  input_times = '1 2'
  time_spans = '0.1 0.1'
  ux = vel_x
  uy = vel_y
  uz = vel_z
  Dx = Diff
  Dy = Diff
  Dz = Diff
[../]
```

DGWallEnergyFluxBC

Inheritance → IntegratedBC (i.e., the MOOSE base kernel for flux-based boundaries)

Notes → This kernel provides residuals for a closed boundary wherein energy is transferred into the domain via a heat transfer coefficient and direct contact with an entity outside the domain. The residual is also scaled by an area fraction in the case of a multi-phase domain wherein only part of the phase is in direct contact with the wall. Both the fluid phase and solid phase would need this kernel if simulating energy density in 2 phases. Also note that this kernel acts on the energy density variable and not the phase temperature.

Residual Formulation

$$\text{Residual (J / m}^2 \text{ / s)} = \varphi \cdot h \cdot f_a \cdot (T - T_w)$$

φ = MOOSE variable test function (for FE formulation)

h = heat transfer coefficient at the wall (W/m²/K)

f_a = area fraction for the phase in contact with the wall

T = variable for phase temperature at the interior of the domain (K)

T_w = variable for temperature of the wall (K)

Additional Computations → None

Usage

Used to create the transfer of energy via contact of a phase with the wall. The variable this kernel acts on is the energy density, not the temperature.

In this example, the energy transfer at a wall boundary is dictated by a variable wall temperature (T_w) with a constant area fraction of 1 and a constant heat transfer of 50 W/m²/K. The internal temperature variable is T and contributes to the residual for internal energy (E).

```
[./wallflux]
  type = DGWallEnergyFluxBC
  variable = E
  boundary = 'wall1 wall2'
  transfer_coef = 50
  area_frac = 1
  wall_temp = Tw
  temperature = T
[../]
```

DirichletBC

Inheritance → NodalBC (i.e., the MOOSE base kernel for flux-based boundaries)

Notes → This kernel provides residuals for a generic Dirichlet type boundary condition in the MOOSE system. **NOTE:** Since the majority of CATS uses DG methods for mass and energy balances, this kernel would NEVER be used for any of those balances and is ONLY valid for the incompressible Navier-Stokes module that CATS can be linked to. CATS uses this kernel only to specify a velocity value at a boundary for incompressible Navier-Stokes. You can use it to specify a value at an inlet/outlet boundary, however, it is most commonly used to specify a “no slip” boundary condition at the walls of a domain. For specification of inflow/outflow, you should use the [INSNormalFlowBC](#) or [PenaltyDirichletBC](#).

Residual Formulation

$$\text{Residual (any units)} = \varphi \cdot g$$

φ = MOOSE variable test function (for FE formulation)

g = expected value at a boundary

Additional Computations → None

Usage

Here, we impose that at the ‘top’ and ‘bottom’ boundary, the velocity in the y-direction should be 0 (i.e., “no slip”).

```
[./strong_no_slip_y]
  type = DirichletBC
  variable = vel_y
  boundary = 'top bottom'
  value = 0
[./]
```

INSNormalFlowBC

Inheritance → IntegratedBC (i.e., the MOOSE base kernel for flux-based boundaries)

Notes → This kernel provides residuals for an open-boundary for the incompressible Navier-Stokes module to weakly specify that the flux of flow across the given boundary must equation a given scalar for the result of the dot product between the actual velocity vector and a normal vector for this boundary. In the case of an inflow boundary condition, this dot product will be a negative value. In the case of an outflow boundary condition, this dot product will be a positive value. Depending on the orientation of the boundary, you may need to provide this type of boundary condition for each component of velocity (i.e., any non-zero velocity component at this particular boundary).

Residual Formulation

$$\text{Residual (m / s)} = \varphi \cdot p \cdot ((\mathbf{v} \cdot \mathbf{n}) - a)$$

φ = MOOSE variable test function (for FE formulation)

p = a penalty term for deviations from this relationship

NOTE: Should provide a large enough value (i.e., 1e6)

\mathbf{v} = velocity vector at the boundary (m/s)

\mathbf{n} = normal vector at the boundary

a = expected value for the dot product (m/s)

Additional Computations → None

Usage

All velocities should also be accompanied by “no slip” boundary conditions at the walls. Those conditions are necessary to complete the Navier-Stokes formulation (at least for the incompressible flow). The “no slip” boundary condition can be applied in a “strong” form using [DirichletBC](#) or in a “weak” form using [PenaltyDirichletBC](#).

Here, we impose that at the ‘left’ boundary, the dot product between the velocity and the boundary normal should result in -1.15 m/s and the resulting velocity should act solely in the y-direction. Since this value is negative, this boundary would naturally represent an inflow

condition. User MUST provide all other velocity component variables and the direction that this boundary residual acts on (0 = x, 1 = y, and 2 = z).

```
[./normal_flow]
  type = INSNormalFlowBC
  variable = vel_y
  direction = 1
  boundary = 'left'
  u_dot_n = -1.15
  ux = vel_x
  uy = vel_y
  uz = vel_z
  penalty = 1e6
[../]
```

PenaltyDirichletBC

Inheritance → NodalBC (i.e., the MOOSE base kernel for flux-based boundaries)

Notes → This kernel provides residuals for a generic penalty based, Dirichlet type boundary condition in the MOOSE system. **NOTE:** Since the majority of CATS uses DG methods for mass and energy balances, this kernel would NEVER be used for any of those balances and is ONLY valid for the incompressible Navier-Stokes module that CATS can be linked to. CATS uses this kernel only to specify a velocity value at a boundary for incompressible Navier-Stokes. It is most commonly used to weakly specify a “no slip” boundary condition at the walls of a domain, however, you can also use it to specify a specific velocity at inflow/outflow boundaries. The advantage of specifying this boundary weakly is improved convergence.

Residual Formulation

$$\text{Residual (any units)} = \phi \cdot p \cdot (u - g)$$

ϕ = MOOSE variable test function (for FE formulation)

p = a penalty term for deviations from this relationship

NOTE: Should provide a large enough value (i.e., 1e6), but should be smaller than the penalty for [INSNormalFlowBC](#)

g = expected value at a boundary

u = non-linear nodal variable

Additional Computations → None

Usage

Here, we impose that at the ‘top’ and ‘bottom’ boundary, the velocity in the y-direction should be very close to 0 (i.e., “no slip”).

```
[./weak_no_slip_y]
```

```
type = PenaltyDirichletBC
variable = vel_y
boundary = 'top bottom'
value = 0
penalty = 1000
[../]
```

Interface Kernels

These kernels are essentially like boundary conditions, but at boundaries that are internal to the overall simulation domain. When the simulation domain needs to be divided into subdomains, the interface kernels define how mass and energy is transferred to different variables in each subdomain respectively. This is particularly useful for 3D modeling of catalysts, as you can create a physical space to represent the open-air channels in a monolith catalyst as well as a separated domain for the solid, porous washcoat where the catalytic reactions take place.

Each subdomain can have its own set of non-linear variables, boundary conditions, parameters, properties, auxiliary variables, etc. Subdomains can be created using the MeshGenerator systems in MOOSE or can be created in other CAD software and imported into MOOSE. Below is an example of how to use the MeshGenerator system to create two different subdomains. In this example, we are creating a cylindrical shaped open channel (block 0) that is coated with a porous washcoat (block 1). We then specify that the side set of nodes between those 2 blocks as an interface boundary and name it so it can be referenced later in the interface kernels.

[Problem]

#NOTE: For RZ coordinates, x ==> R and y ==> Z (and z ==> nothing)

coord_type = RZ

[] #END Problem

[Mesh]

Master block (entire domain) with default block_id = 0

[gen]

type = GeneratedMeshGenerator

dim = 2

nx = 7

ny = 10

xmin = 0.0

xmax = 0.1015 # m radius

ymin = 0.0

ymax = 0.1346 # m length

[]

#Create a bounding box from the entire domain to span the new subdomain (block = 1)

[./subdomain1]

input = gen

type = SubdomainBoundingBoxGenerator

bottom_left = '0.0726 0 0'

top_right = '0.1015 0.1346 0'

block_id = 1

[./]

#Designate a new boundary as the side sets that are shared between block 0 and block 1

The new boundary is now labeled and can be used in boundary conditions or InterfaceKernels

[./interface]

type = SideSetsBetweenSubdomainsGenerator

input = subdomain1

master_block = '0'

```

        paired_block = '1'
        new_boundary = 'master0_interface'
    [../]
    #Break up the original boundaries (left right top bottom) to create separate boundaries
    # for each subdomain new boundary names are (old_name)_to_(block_id)
    # For example, two new left side boundary names: left_to_0 and left_to_1
    # left_to_0 is the new left side boundary that is a part of block 0
    [./break_boundary]
        input = interface
        type = BreakBoundaryOnSubdomainGenerator
    [../]
[]

# Variables must be defined to exist on one domain or the other (or multiple domains)
[Variables]
    [./u]
        order = FIRST
        family = MONOMIAL
        block = 0          #domain on which the variable is defined
    [../]
    [./v]
        order = FIRST
        family = MONOMIAL
        block = 1          #domain on which the variable is defined
    [../]
[]

```

InterfaceConstReaction

Inheritance → InterfaceKernel (i.e., the MOOSE base class for interface kernels)

Notes → This kernel creates residuals for the master block (block 0) variable and the neighbor block (block 1) variable. It represents the physics of a simple chemical reaction at the boundary that results in a phase change. For instance, this kernel is analogous to a Henry's Law type of reaction for the transfer of mass from the gas phase to a liquid phase (or solid phase) at a specific interior boundary.

Residual Formulation

Residual (mass / area / time) =

master variable (u): $\varphi \cdot (k_u u - k_v v)$

neighbor variable (v): $-\varphi \cdot (k_u u - k_v v)$

φ = MOOSE variable test function (for FE formulation)

k_u = reaction rate coefficient for master variable

k_v = reaction rate coefficient for the neighbor variable

u = master variable on the master block (block 0)

v = neighbor variable on the neighbor block (block 1)

Note: each subdomain is not necessarily named block 0 and block 1

Additional Computations → None

Usage

All interface kernels only need to be invoked once for each interface and they will automatically provide the residuals for both the master and neighbor variables.

In this example, the physics is for a reaction that occurs at the surface of a boundary named master0_interface. The master variable is u and the neighbor variable is v . The rate parameters for each variable are 1 and 2, respectively.

```
[InterfaceKernels]
  [./interface]
    type = InterfaceConstReaction
    variable = u           #variable must be the variable in the master block
    neighbor_var = v       #neighbor_var must be the variable in the neighbor block
    boundary = master0_interface #name of the interface boundary
    variable_rate = 1
    neighbor_rate = 2
  [../]
[] #END InterfaceKernels
```

InterfaceEnergyTransfer

Inheritance → InterfaceKernel (i.e., the MOOSE base class for interface kernels)

Notes → This kernel creates residuals for the master block (block 0) variable and the neighbor block (block 1) variable. It represents the physics of energy transfer between the two different subdomains of different energy density variables with different phase temperatures. They transfer energy through a temperature difference at the point of contact with a heat transfer coefficient and contact area fraction, similar to the [DGWallEnergyFluxBC](#).

Residual Formulation

Residual ($J / m^2 / s$) =

master variable (Eu): $\varphi \cdot h \cdot f_a (T_u - T_v)$

neighbor variable (Ev): $-\varphi \cdot h \cdot f_a (T_u - T_v)$

φ = MOOSE variable test function (for FE formulation)

h = heat transfer coefficient ($W/m^2/K$)

f_a = contact area fraction (if not the full boundary area)

T_u = temperature for master energy density variable

T_v = temperature for the neighbor energy density variable

E_u = master energy variable on the master block (block 0)

E_v = neighbor energy variable on the neighbor block (block 1)

Note: each subdomain is not necessarily named block 0 and block 1

Additional Computations → None

Usage

All interface kernels only need to be invoked once for each interface and they will automatically provide the residuals for both the master and neighbor variables.

In this example, the physics is for a transfer of energy that occurs at the surface of a boundary named master0_interface. The master variable is E_u and the neighbor variable is E_v . The temperatures variables for each are T_u and T_v , respectively. The area fraction is 1 and the heat transfer coefficient is 50.

```
[InterfaceKernels]
  [./interface]
    type = InterfaceEnergyTransfer
    variable = Eu           #variable must be the variable in the master block
    neighbor_var = Ev       #neighbor_var must the variable in the neighbor block
    boundary = master0_interface #name of the interface boundary
    transfer_coef = 50
    area_frac = 1
    master_temp = Tu
    neighbor_temp = Tv
  [../]
[] #END InterfaceKernels
```

InterfaceMassTransfer

Inheritance → InterfaceKernel (i.e., the MOOSE base class for interface kernels)

Notes → This kernel creates residuals for the master block (block 0) variable and the neighbor block (block 1) variable. It represents the physics of mass transfer between the two different subdomains. They transfer energy through a concentration difference at the point of contact with a constant mass transfer rate.

Residual Formulation

Residual (mass / area / time) =

$$\text{master variable (Eu): } \varphi \cdot k(u - v)$$

neighbor variable (Ev): $-\varphi \cdot k(u - v)$

φ = MOOSE variable test function (for FE formulation)

u = master concentration variable on the master block (block 0)

v = neighbor concentration variable on the neighbor block (block 1)

Note: each subdomain is not necessarily named block 0 and block 1

Additional Computations → None

Usage

All interface kernels only need to be invoked once for each interface and they will automatically provide the residuals for both the master and neighbor variables.

In this example, the physics is for a transfer of mass that occurs at the surface of a boundary named master0_interface. The master variable is u and the neighbor variable is v. The mass transfer rate is 1.

```
[InterfaceKernels]
  [./interface]
    type = InterfaceMassTransfer
    variable = u           #variable must be the variable in the master block
    neighbor_var = v       #neighbor_var must be the variable in the neighbor block
    boundary = master0_interface #name of the interface boundary
    transfer_rate = 1
  [./]
[] #END InterfaceKernels
```

Initial Conditions

These are kernels that do not create residuals, but instead create initial conditions for non-linear variables in the domain. There are many, many built-in initial condition kernels in MOOSE including just setting variables to a constant or setting variables to values specified by a generic function of space (in x, y, z coordinates). Discussed here are some custom initial condition kernels specific to our problems of interest.

InitialInhibitionProducts

Inheritance → InitialCondition (i.e., MOOSE base kernel for initial conditions)

Notes → This kernel is designed to automate the process by which initial conditions are set for the inhibition product variables (see [InhibitionProducts](#) kernel for more details). This is useful for improving convergence of problems that use inhibition terms for chemical reactions. In this particular case, this form of the inhibition term is made up of a product of other terms (presumably other Langmuir inhibition terms).

Computation

$$\text{Inhibition variable (unitless)} = \left(\prod_{i=1} R_i^{p_i} \right)$$

R_i = i-th inhibition variable

p_i = power that the i-th inhibition term is raised to

Usage

Typically, the user would provide the initial condition kernel underneath the definition of the non-linear variable.

This kernel allows all the parameters that it depends on to be non-linear variables, thus the initial inhibition product variable can vary depending on how the other parameters are defined.

Note that the usage and parameter/variable names should always be the same as they are defined in the [InhibitionProducts](#) kernel.

```
[Variables]
  [./R]
    order = FIRST
    family = MONOMIAL
  [./InitialCondition]
    type = InitialInhibitionProducts
    coupled_list = 'R1 R2'
    power_list = '1 1'
  [../]
[../]
[]
```

InitialLangmuirInhibition

Inheritance → InitialCondition (i.e., MOOSE base kernel for initial conditions)

Notes → This kernel is designed to automate the process by which initial conditions are set for the Langmuir inhibition variables (see [LangmuirInhibition](#) kernel for more details). This is useful for improving convergence of problems that use inhibition terms for chemical reactions. In this particular case, this form of the inhibition term is made up of a sum of Langmuir terms that is used to represent surface coverage for a catalytic reaction.

Computation

$$\text{Inhibition variable (unitless)} = (1 + \sum_i K_i C_i)$$

C_i = concentration variable for species i

K_i = Langmuir coefficient for the i -th species in the list

Additional Computations

The Langmuir coefficients are calculated from the same type of expression as the Arrhenius reaction term.

$$K_i = A_i T^{\beta_i} \exp \left[-\frac{E_i}{RT} \right]$$

- The units for A_i must have the same units as K_i
- K_i must have units of inverse concentration C_i
- The units for E_i must be J/mol
- β_i are powers on temperature (which default to 0)

Usage

Typically, the user would provide the initial condition kernel underneath the definition of the non-linear variable.

This kernel allows all the parameters that it depends on to be non-linear variables, thus the initial Langmuir Inhibition variable can vary depending on other parameters/variables.

Note that the usage and parameter/variable names should always be the same as they are defined in the [LangmuirInhibition](#) kernel.

The “temperature” argument can be a value or a variable. Same with the concentrations in the “coupled_list” argument.

```
[Variables]
  [./R1]
    order = FIRST
    family = MONOMIAL
  [./InitialCondition]
    type = InitialLangmuirInhibition
    coupled_list = 'C1 C2'
    temperature = 298
```

```

pre_exponentials = '1 2'
activation_energies = '-20000 10000'
betas = '0 0'

[../]

[../]

[]

```

InitialPhaseEnergy

Inheritance → InitialCondition (i.e., MOOSE base kernel for initial conditions)

Notes → Generally, we do not know what the total internal energy is for a system. Instead, we would more typically know what the initial temperatures, densities, and other properties are in the domain. Thus, this kernel can be invoked to calculate the initial internal energy of a phase in the domain as a function of those known domain properties. Whenever we are performing an energy balance based on the energy density of a phase, we are required to provide an initial condition to that energy density. This kernel facilitates those calculations.

Computation

$$\text{Initial Energy Density (J/m}^3\text{)} = \rho c T$$

ρ = initial density of the phase in the domain (kg/m³)

c = initial specific heat of the phase in the domain (J/kg/K)

T = initial temperature of the phase in the domain (K)

Usage

Typically, the user would provide the initial condition kernel underneath the definition of the non-linear variable.

This kernel allows all the parameters that it depends on to be non-linear variables, thus the initial energy density can vary in space depending on how the other parameters are defined.

```

[Variables]
[../Ef]
    order = FIRST
    family = MONOMIAL
[../InitialCondition]
    type = InitialPhaseEnergy
    specific_heat = cpg
    density = rho
    temperature = Tf
[../]

[../]

[]

```

Auxiliary Kernels

The auxiliary system is used to calculate material properties needed for the simulations. We put these calculations in the auxiliary system, rather than the material properties system, because this will allow us to later move the calculation of these parameters to the kernel system if need be. For instance, if we want pressure or density to be a non-linear variable, rather than a parameter or property, it is easier to declare them as auxiliary variables now. This will allow all kernels that depend on those properties to be valid whether they are constants, auxiliary variables, or other non-linear variables. On the other hand, if we had declared those parameters as material properties, then we would have to build new kernels to use them if and when we decided to move those properties to the kernel system.

In addition to calculation of system properties, the auxiliary system can also be used to update system parameters as generic functions of space and time. This is particularly useful for simulating isothermal processes that do experience a set temperature change, like in the case of Temperature Programmed Desorption (TPD) experiments. For such a simulation, we need to include temperature as a variable into the calculations, but since the reactor system is kept isothermal, it is unnecessary to invoke a full energy balance. Instead, we can declare temperature as an auxiliary variable and write an auxiliary function to represent what the average experimental temperature is at any point in time.

AuxAvgLinearVelocity

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel can be used to calculate the average linear velocity in a packed column or through the channels of a monolith catalyst. For simulations that do not have a set velocity field, or do not use Navier-Stokes modules to produce flow fields, you can invoke this kernel for the primary flow direction and estimate the gas velocity in that direction based on the total volumetric flow rate, total cross-sectional area, and/or the porosity or bulk void space of the domain. This only estimates 1 velocity term, not a velocity vector. Thus, it would then presume that all velocity moves in the direction perpendicular to the cross-sectional area of the domain.

Computation

$$\text{Average linear velocity (m/s)} = \frac{Q}{\varepsilon A}$$

Q = total volumetric flow rate (m³/s)

A = total cross-sectional area (m²)

ε = domain average void space or porosity

Usage

All auxiliary kernels must provide an argument for option 'execute_on' that is used to tell the software when to compute this value. You MUST ALWAYS specify to execute on 'initial' and 'timestep_end' only!

In the below example, we are calculating the velocity in the y-direction (vel_y) as a function of a constant flow rate and a variable cross-sectional area (A) with a porosity of 1.

```
[./vel_calc]
    type = AuxAvgLinearVelocity
    variable = vel_y
    execute_on = 'initial timestep_end'
    flow_rate = 0.5
    xsec_area = A
    porosity = 1
[../]
```

AuxErgunPressure

Inheritance → [GasPropertiesBase](#)

Notes → This kernel uses the Ergun equation to estimate pressure drop axially in a packed bed or any straight segment of a reactor. User must provide a direction in which the pressure drop applies (0=x, 1=y, 2=z) and a pressure value or variable representing either a known pressure at the inlet or outlet of the domain. If the outlet pressure is provided, then the user must also specify some parameters defining the length of the column/domain (i.e., the 'start_point' in meters and the 'end_point' in meters).

(See [GasPropertiesBase](#) for additional notes)

Computation

$$\text{Ergun Equation for Pressure Drop} \rightarrow -\frac{\Delta P}{\Delta z} = 150 \frac{\mu(1-\varepsilon)^2}{\varepsilon^3 d^2} \varepsilon v + 1.75 \frac{(1-\varepsilon)}{\varepsilon^3 d} \rho \varepsilon^2 v^2$$

If given inlet pressure...

$$P \text{ (Pa)} = P_{in} - \left[150 \frac{\mu(1-\varepsilon)^2}{\varepsilon^3 d^2} \varepsilon v + 1.75 \frac{(1-\varepsilon)}{\varepsilon^3 d} \rho \varepsilon^2 v^2 \right] \cdot (z - z_s)$$

If given outlet pressure...

$$P \text{ (Pa)} = P_{out} + \left[150 \frac{\mu(1-\varepsilon)^2}{\varepsilon^3 d^2} \varepsilon v + 1.75 \frac{(1-\varepsilon)}{\varepsilon^3 d} \rho \varepsilon^2 v^2 \right] \cdot (z_e - z)$$

z = axial position in the system (m)

z_s = starting point in the domain (m)

z_e = ending point in the domain (m)

d = particle diameter or hydraulic diameter (m)

ε = porosity in the domain (constant or variable)

v = average linear velocity (m/s)

→ Calculated from the velocity vector components

μ = gas viscosity (See [GasViscosity](#) for calculation)

ρ = gas density (See [GasDensity](#) for calculation)

P = calculated pressure at all axial positions (Pa)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the pressure drop in a packed column along the y-direction given a reference pressure variable (Po) that represents the outlet pressure. The hydraulic diameter is the size of the particles in the system, which is a constant 0.01 m. The size of the column is indicated by the starting and ending y-points of 0 and 0.13 m, respectively.

```
[./press_calc]
    type = AuxErgunPressure
    variable = P
    direction = 1
    porosity = 0.5
    temperature = T
    pressure = Po
    is_inlet_press = false
    start_point = 0
    end_point = 0.13
    hydraulic_diameter = 0.01
    uy = vel_y
[../]
```

GasDensity

Inheritance → [GasPropertiesBase](#)

Notes → The density of the gas phase is computed from the known concentrations of species in the gas phase. In this kernel, it is NOT a function of ideal gas law, since the domain is assumed to be confined. If the user desires to have the gas density a function of ideal gas law, then a new kernel will need to be added. As an optional argument, the user may provide a variable or value representing the concentration of a carrier gas and its respective molecular weight.

(See [GasPropertiesBase](#) for additional notes)

Computation

$$\text{Gas density (kg/m}^3\text{)} = \sum_i C_i \frac{MW_i}{1000} + C_{\text{carrier}} \frac{MW_{\text{carrier}}}{1000}$$

C_i = molar concentration of the i th gas species (mol/m³)

MW_i = molar weight of the i th gas species (g/mol)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the density in the domain given an N2 carrier gas with a molecular weight of 28 g/mol.

NOTE: If N2 is from the original list of gas species, then DO NOT provide here. This will cause errors in the calculation of density.

```
[./dens_calc]
    type = GasDensity
    variable = rho
    carrier_gas = N2
    carrier_gas_mw = 28
    temperature = T           #Note: not used, but a required argument
    pressure = P              #Note: not used, but a required argument
    hydraulic_diameter = d    #Note: not used, but a required argument
    uy = vel_y                #Note: not used, but a required argument
[../]
```

GasEffectiveThermalConductivity

Inheritance → [GasPropertiesBase](#)

Notes → This kernel is used to calculate an effective thermal conductivity that is to be a volume averaged conductivity between the conductivity of 2 different phases in a domain. For instance, in the case of packed beds, the energy density and temperatures are usually solved in a single combined term which assumes local isothermal equilibria between the solid and fluid. Thus, it is necessary to create an effective conductivity parameter that averages the fluid and solid conductivities based on the domain porosity.

(See [GasPropertiesBase](#) for additional notes)

Computation

$$\text{Effective Conductivity (W/m/K)} = (1 - \varepsilon) \cdot K_s + \varepsilon \cdot K_g$$

ε = porosity in the domain (constant or variable)

K_s = solid phase thermal conductivity (W/m/K)

K_g = gas phase thermal conductivity (See [GasThermalConductivity](#) for details)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the effective thermal conductivity of a packed column. The porosity in the system is 0.5 and the average linear velocity is calculated from the given velocity vectors. The 'heat_cap_ratio' is an optional parameter that represents the ratio of the constant pressure specific heat to the constant volume specific heat (i.e., c_p/c_v). Typical values for this ratio is about 1.4. It can range between 0.56 and 1.67. Some arguments are required by the base class object [GasPropertiesBase](#), but are not actually used in the calculations.

```
[./Ke_calc]
  type = GasEffectiveThermalConductivity
  variable = Ke
  temperature = T
  pressure = P
  hydraulic_diameter = d      #Note: not used, but a required argument
  uy = vel_y                  #Note: not used, but a required argument
  heat_cap_ratio = 1.4
  solid_conductivity = Ks
  porosity = 0.5
[../]
```

GasPropertiesBase

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This is the base object that defines how all ‘Gas’ auxiliary values are calculated. It sets up the variables needed and provides a way to interface with the [Egret](#) utilities subroutines. Because of this, all the ‘Gas’ auxiliary kernels require a common set of input parameters, although some do require additional arguments beyond the base set of arguments. Since all those input arguments are common to all other ‘Gas’ auxiliary kernels, you should put those arguments in the ‘GlobalParams’ block of you input file. See the example below...

```
[GlobalParams]
  # Always have the arguments below in GlobalParams
  gases = 'N2 O2 CO2'
  molar_weights = '28 32 44'
  sutherland_temp = '300.55 292.25 293.15'
  sutherland_const = '111 127 240'
  sutherland_vis = '0.0001781 0.0002018 0.000148'
  spec_heat = '1.04 0.919 0.846'
  execute_on = 'initial timestep_end'

  # Arguments in the below list can also go in GlobalParams
  #       or defined locally in other auxiliary kernels if desired
  #       NOTE: it is usually recommended to provide these locally
  temperature = T
  ux = vel_x
  uy = vel_y
  uz = vel_z

  # Arguments below are optional arguments to refine how gas
  #       properties are calculated (also used in GasDensity calculation)
  carrier_gas = Air
  carrier_gas_mw = 28.8
  is_ideal_gas = false
[]
```

gases = list of gas concentration variables (mol/m³)

carrier_gas = gas concentration variable/constant for carrier gas (mol/m³)

→ **NOTE:** DO NOT put a gas species concentration here that already exists in the list of 'gases'. If all gases are accounted for in 'gases', then you can leave this optional parameter out or set to 0.

molar_weights = list of molecular weights of corresponding gas species (g/mol)

carrier_gas_mw = molecular weight of carrier gas (g/mol)

sutherland_temp = list of Sutherland's temperatures for each gas species (K)

sutherland_const = list of Sutherland's constants for each gas species (K)

sutherland_vis = list of Sutherland's viscosities for each gas species (g/cm/s)

spec_heat = list of specific heat capacities at constant pressure for each gas (J/g/k)

execute_on = when to calculate the properties (ALWAYS 'initial timestep_end')

temperature = temperature variable for the gas (K)

ux, uy, uz = velocity component variables in each cardinal direction (m/s)

is_ideal_gas = Boolean (true/false) used to determine whether or not kinetic theory of gases will use the Ideal Gas Law to estimate gas densities or just use the given gas concentrations. If true, then ideal gas law is assumed for gas properties.

Computation → None, this is a place holder object to prepare calculations.

Usage → None, this object should not be invoked in the input files.

GasSolidHeatTransferCoef

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates the gas-to-solid heat transfer coefficient (h_s) in W/m²/K. This is a parameter that the user can provide as a constant for [PhaseEnergyTransfer](#) or can use this kernel to approximate that parameter from the Prandtl and Reynolds numbers, as well as an approximated gas thermal conductivity. The calculation is not a fundamental calculation, but instead is an empirical estimation based a relationship between the Prandtl and Reynolds numbers, as well as the thermal conductivity of the solids.

(See [GasPropertiesBase](#) for additional notes)

Computation

$$\text{Prandtl Number (Pr)} = \frac{\mu c_p}{K_g}$$

$$\text{Reynolds Number (Re)} = \frac{\varepsilon v d \rho}{\mu}$$

$$\text{Gas-solid Heat Transfer (W/m}^2\text{/K)} = \frac{K_s}{500 \cdot d} [2 + 1.1 \cdot Re^{0.6} \cdot Pr^{0.3}]$$

ε = porosity in the domain (constant or variable)

v = average linear velocity (m/s)

→ Calculated from the velocity vector components

μ = gas viscosity (See [GasViscosity](#) for calculation)

ρ = gas density (See [GasDensity](#) for calculation)

d = particle diameter or hydraulic diameter (m)

c_p = specific heat of the gas at constant pressure (J/kg/m³)

K_s = solids thermal conductivity (W/m/K)

K_g = gas thermal conductivity (See [GasThermalConductivity](#) for calculation)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the heat transfer coefficient for solids with diameter of 0.01 m and a variable conductivity (Ks). The porosity in the system is 0.5 and the average linear velocity is calculated from the given velocity vectors. The 'heat_cap_ratio' is an optional parameter that represents the ratio of the constant pressure specific heat to the constant volume specific heat (i.e., c_p/c_v). Typical values for this ratio is about 1.4. It can range between 0.56 and 1.67.

```
[./hs_calc]
  type = GasSolidHeatTransferCoef
  variable = hs
  temperature = T
  pressure = P
  hydraulic_diameter = 0.01
  ux = vel_x
  uy = vel_y
  uz = vel_z
  heat_cap_ratio = 1.4
  solid_conductivity = Ks
  porosity = 0.5
[../]
```

GasSpecHeat

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates the specific heat of the gas (at constant pressure: c_p) in J/kg/K. That value is computed as a weighted average of the specific heats of each gas species given. Recall that [GasPropertiesBase](#) requires you give the list of gas species and their respective standard state specific heats.

(See [GasPropertiesBase](#) for additional notes)

Computation

$$\text{Specific heat at constant pressure (J/kg/K)} = \sum_i y_i c_{p,i}$$

y_i = mole fraction of gas species i

→ mole fractions are calculated from pressure, temperature, and mass concentrations for each species

$c_{p,i}$ = specific heat of gas species i

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the specific heat of the gas in the domain given variables for temperature and pressure in that domain. Because this kernel inherits from [GasPropertiesBase](#), there are some arguments required that are not actually used. That is why it is recommended to push some of those arguments to GlobalParams, so you reduce redundancy in the input files.

```
[./cp_calc]
  type = GasSpecHeat
  variable = cp
  temperature = T
  pressure = P
  hydraulic_diameter = d          #Note: not used, but a required argument
  uy = vel_y                     #Note: not used, but a required argument
[../]
```

GasSpeciesAxialDispersion

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates an effective dispersion coefficient (m^2/s) for a gas species as a function of temperature, pressure, and concentrations of all gas species of interest. That dispersion variable can then be used in [GVariableDiffusion \(DGVariableDiffusion\)](#) or [GVarPoreDiffusion \(DGVarPoreDiffusion\)](#) to update the diffusive flux based on dispersion in a packed column. It is a function of the true molecular diffusivity of the gas species, as well as a correction for mechanical mixing that occurs in the tortuous path the molecules take in the

domain. Those corrections are based on empirical relationships with the Reynolds and Schmidt numbers, as well as the overall diameter or hydraulic diameter of the domain.

(See [GasPropertiesBase](#) for additional notes)

Computation

$$\text{Schmidt Number (Sc)} = \frac{\mu}{\rho D_{m,i}}$$

$$\text{Reynolds Number (Re)} = \frac{v D \rho}{\mu}$$

$$\text{Effective Dispersion of Species (m}^2\text{/s)} = v \cdot d \cdot \left(0.5 + \frac{20}{Re \cdot Sc} \right)$$

μ = gas viscosity (See [GasViscosity](#) for calculation)

ρ = gas density (See [GasDensity](#) for calculation)

d = particle diameter or hydraulic diameter (m)

D = bed diameter domain width variable (m)

$D_{m,i}$ = molecular diffusivity of species i (See [GasSpeciesDiffusion](#) for calculation)

v = average linear velocity (m/s)

→ Calculated from the velocity vector components

i = gas species index

→ This value must be provided by user and should correspond to the position of the gas species of interest in the 'gases' list from [GasPropertiesBase](#) (indexing starts from 0)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the effective diffusivity of O₂ gas in the domain given variables for temperature and pressure in that domain. The column diameter is a variable named Dia and the particle diameter is another parameter named d.

[./De_calc]

```

type = GasSpeciesAxialDispersion
variable = De
gases = 'N2 O2 CO2'
species_index = 1          #1 – Corresponds to O2 in the 'gases' list
macroscale_diameter = Dia
temperature = T
pressure = P
hydraulic_diameter = d
uy = vel_y

```

[../]

GasSpeciesDiffusion

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates the gas phase molecular diffusivity of a given gas species based on kinetic theory of gases, the Sutherland's model of gas viscosity, and Ideal Gas Law. User needs to provide the temperature, pressure, and index of the gas species of interest from the 'gases' list in [GasPropertiesBase](#).

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Molecular Diffusion of Species (m}^2\text{/s)} = \frac{(1 - y_i)}{\left(\sum_{j \neq i} \frac{y_j}{D_{ij}}\right)}$$

$$D_{ij} = \frac{(4/\sqrt{2})\sqrt{(MW_i^{-1} + MW_j^{-1})}}{\left[\left(\frac{\rho_i^2}{\{1.92\mu_i^2 MW_i\}}\right)^{0.25} + \left(\frac{\rho_j^2}{\{1.92\mu_j^2 MW_j\}}\right)^{0.25}\right]^2}$$

MW_i = molecular weights of ith species in gas (g/mol)

μ_i = partial viscosity of the ith species (See [GasViscosity](#) for calculation)

y_i = mole fraction of gas species i

→ mole fractions are calculated from pressure, temperature, and mass concentrations for each species

P = gas pressure variable (Pa)

T = gas temperature variable (K)

R = gas law constant (8.3144621 L*kPa/K/mol)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the molecular diffusivity of O₂ gas in the domain given variables for temperature and pressure in that domain.

[./D_calc]

type = GasSpeciesDiffusion

variable = D

gases = 'N2 O2 CO2'

species_index = 1

#1 – Corresponds to O2 in the 'gases' list

temperature = T

pressure = P

hydraulic_diameter = d

#Note: not used, but a required argument

uy = vel_y

#Note: not used, but a required argument

[../]

GasSpeciesEffectiveTransferCoef

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates the effective mass transfer rate for a gas species. The effective rate of mass transfer is calculated in a very similar manner to [GasSpeciesMassTransCoef](#), however, it uses corrections for the impact of pore-diffusion. You would typically use this kernel over [GasSpeciesMassTransCoef](#) when pore-diffusion is an important factor in the transfer of mass from gas to solid spaces, but do not want to have to simulate the diffusion inside the pore space explicitly with a hybrid FD/FE subdomain approach as described in [MicroscaleDiffusion](#) (and related) kernels.

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Schmidt Number (Sc)} = \frac{\mu}{\rho D_{p,i}}$$

$$\text{Reynolds Number (Re)} = \frac{v d \rho}{\mu}$$

$$\text{Effective Mass Transfer (m/s)} = \frac{D_{p,i}}{d} [2 + 1.1 \cdot \text{Re}^{0.6} \cdot \text{Sc}^{0.3}]$$

ϵ_p = porosity in the particle (constant or variable) [Required for Pore Diffusion]

$D_{p,i}$ = pore-space diffusivity (See [GasSpeciesPoreDiffusion](#) for calculation)

v = average linear velocity (m/s)

→ Calculated from the velocity vector components

μ = gas viscosity (See [GasViscosity](#) for calculation)

ρ = gas density (See [GasDensity](#) for calculation)

d = particle diameter or hydraulic diameter (m)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the effective mass transfer of O₂ gas in the domain given variables for temperature and pressure in that domain. The particle porosity has a value of 0.2 and the particle as a diameter of d.

[./ke_calc]

type = GasSpeciesEffectiveTransferCoef

variable = ke

gases = 'N2 O2 CO2'

species_index = 1

#1 – Corresponds to O2 in the 'gases' list

micro_porosity = 0.2

temperature = T

pressure = P

```

        hydraulic_diameter = d
        uy = vel_y
[../]

```

GasSpeciesKnudsenDiffusionCorrection

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates and effective pore-diffusivity inside of micro-porous region of a particle using a correction for the effects of Knudsen diffusion. User needs to provide the gas species index for the species of interest as well as providing the micro-scale porosity and nominal/average micro-pore radius.

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Effective Pore Diffusivity of Species (m}^2\text{/s)} = [D_{p,i}^{-1} + D_{k,i}^{-1}]^{-1}$$

$$D_{k,i} = 9700 \cdot r_p \left(\frac{T}{MW_i} \right)^{0.5}$$

ε_p = porosity in the particle (constant or variable) [Required for Pore Diffusion]

$D_{p,i}$ = pore-space diffusivity (See [GasSpeciesPoreDiffusion](#) for calculation)

MW_i = molecular weights of ith species in gas (g/mol)

r_p = micro-pore radius (m)

T = gas temperature variable (K)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the effective pore-diffusion of O₂ gas in the domain given variables for temperature and pressure in that domain. The particle porosity has a value of 0.2 and the particle as a diameter of d. The nominal pore radius is 1E-7 m.

```

[../Deff_calc]
    type = GasSpeciesKnudsenDiffusionCorrection
    variable = Deff
    gases = 'N2 O2 CO2'
    species_index = 1          #1 – Corresponds to O2 in the 'gases' list
    micro_porosity = 0.2
    micro_pore_radius = 1E-7
    temperature = T
    pressure = P
    hydraulic_diameter = d     #Note: not used, but a required argument
[../]

```

GasSpeciesMassTransCoef

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates the film mass transfer coefficient for a gas species. The film mass transfer coefficient can be used in conjunction with [FilmMassTransfer](#) (or similar) kernels to facilitate the simulation of mass transfer from the bulk gas to the gases in the pore-space of particles. It would most appropriately be used with the [MicroscaleDiffusion](#) type of kernels that also simulate pore-diffusion in porous particles or materials.

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Schmidt Number (Sc)} = \frac{\mu}{\rho D_{m,i}}$$

$$\text{Reynolds Number (Re)} = \frac{vd\rho}{\mu}$$

$$\text{Mass Transfer Rate (m/s)} = \frac{D_{m,i}}{d} [2 + 1.1 \cdot Re^{0.6} \cdot Sc^{0.3}]$$

$D_{m,i}$ = molecular diffusivity (See [GasSpeciesDiffusion](#) for calculation)

v = average linear velocity (m/s)

→ Calculated from the velocity vector components

μ = gas viscosity (See [GasViscosity](#) for calculation)

ρ = gas density (See [GasDensity](#) for calculation)

d = particle diameter or hydraulic diameter (m)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the mass transfer of O₂ gas in the domain given variables for temperature and pressure in that domain. The particle porosity has a diameter of d .

[./k_calc]

type = GasSpeciesMassTransCoef

variable = k

gases = 'N2 O2 CO2'

species_index = 1

#1 – Corresponds to O2 in the 'gases' list

temperature = T

pressure = P

hydraulic_diameter = d

uy = vel_y

[../]

GasSpeciesPoreDiffusion

Inheritance → [GasPropertiesBase](#)

Notes → This kernel estimates the pore-diffusion coefficient from the molecular diffusivity of a species and the micro-porosity of the particles. Pore diffusion is a function of the tortuosity of the path the molecules take through the pore spaces, but tortuosity is a measure that is difficult to quantify. Here, the tortuosity is approximated as $1/\varepsilon_p$, where ε_p is the particle porosity.

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Pore Diffusivity (m}^2\text{/s)} = \frac{\varepsilon_p D_{m,i}}{\tau}$$

$D_{m,i}$ = molecular diffusivity (See [GasSpeciesDiffusion](#) for calculation)

ε_p = porosity in the particle (constant or variable)

τ = tortuosity for the particles $\sim 1/\varepsilon_p$

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the effective pore-diffusion of O₂ gas in the domain given variables for temperature and pressure in that domain. The particle porosity has a value of 0.2.

```
[./Dp_calc]
  type = GasSpeciesPoreDiffusion
  variable = Dp
  gases = 'N2 O2 CO2'
  species_index = 1           #1 – Corresponds to O2 in the 'gases' list
  micro_porosity = 0.2
  temperature = T
  pressure = P
  hydraulic_diameter = d      #Note: not used, but a required argument
[../]
```

GasThermalConductivity

Inheritance → [GasPropertiesBase](#)

Notes → This kernel approximates the thermal conductivity of the gas phase based on the specific heat at constant pressure, the ratio of the specific heats at constant pressure and temperature (i.e., c_p/c_v), and the viscosity of the gas. The specific heat ratio is an optional parameter that defaults to 1.4. Typical range is between 0.56 and 1.67.

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Gas Thermal Conductivity (W/m/K)} = 0.25 \cdot \left(9 \left[\frac{c_p}{c_v} \right] - 5 \right) \cdot \mu \cdot c_v$$

c_p = specific heat of the gas at constant pressure (J/kg/m³)

→ See [GasSpecHeat](#) for calculation

c_v = specific heat of the gas at constant volume (J/kg/m³)

μ = gas viscosity (See [GasViscosity](#) for calculation)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the thermal conductivity of the gas phase. The 'heat_cap_ratio' is an optional parameter that represents the ratio of the constant pressure specific heat to the constant volume specific heat (i.e., c_p/c_v). Typical values for this ratio is about 1.4. It can range between 0.56 and 1.67.

[./Kg_calc]

type = GasThermalConductivity

variable = Kg

temperature = T

pressure = P

hydraulic_diameter = d

#Note: not used, but a required argument

uy = vel_y

#Note: not used, but a required argument

heat_cap_ratio = 1.4

[./]

GasVelocityCylindricalReactor

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel can be used to calculate the average linear velocity in a packed column or through the channels of a monolith catalyst. For simulations that do not have a set velocity field, or do not use Navier-Stokes modules to produce flow fields, you can invoke this kernel for the primary flow direction and estimate the gas velocity in that direction based on the space-velocity at a given reference state, dimensions of the reactor, and the porosity or bulk void space of the domain. This only estimates 1 velocity term, not a velocity vector. Thus, it would then presume that all velocity moves in the direction perpendicular to the cross-sectional area of the domain. This computation differs from [AuxAvgLinearVelocity](#) in that it will account for variations in average velocity with changes in inlet temperatures and pressures using an ideal gas assumption.

Computation

$$\text{Average linear velocity (length/time)} = \frac{Q_{true}}{\varepsilon A}$$

Q_{true} = true total volumetric flow rate (volume/time)

$$= Q_{\text{ref}} * (P_{\text{ref}}/P_{\text{in}}) * (T_{\text{in}}/T_{\text{ref}})$$

$P_{\text{ref/in}}$ = pressure terms (kPa)

$T_{\text{ref/in}}$ = temperature terms (K)

Q_{ref} = reference state volumetric flow rate (volume/time)

$$= (SV)*V$$

SV = space-velocity at the reference state (reactor volumes / time)

V = total reactor volume (volume)

$$= A*L$$

L = length of the reactor (length)

A = total cross-sectional area (area)

$$=\pi r^2$$

r = radius of cylindrical reactor (length)

ε = domain average void space or porosity

Usage

All auxiliary kernels must provide an argument for option 'execute_on' that is used to tell the software when to compute this value. You MUST ALWAYS specify to execute on 'initial' and 'timestep_end' only!

In the below example, we are calculating the velocity in the y-direction (vel_y) as a function of the given space-velocity, variables for inlet pressure and temperature ($press$ and $temp$), and the given reactor dimension information (porosity, length, and radius)

```
[./vel_calc]
  type = GasVelocityCylindricalReactor
  variable = vel_y
  execute_on = 'initial timestep_end'
  space_velocity = 500 #volumes / min
  ref_temperature = 298
  inlet_temperature = temp
  ref_pressure = 100
  inlet_pressure = press
  porosity = 0.33
  radius = 1
  length = 5
[../]
```

GasViscosity

Inheritance → [GasPropertiesBase](#)

Notes → This kernel calculates the gas viscosity based on kinetic theory of gases and the Sutherland's model for the viscosities of pure gas species.

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Gas Viscosity (kg/m/s)} = \sum_i \left\{ \frac{\mu_i}{\left(1 + \frac{113.65 \cdot \chi \mu_i T}{y_i MW_i \left[\sum_{j \neq i} y_j / D'_{ij} \right]} \right)} \right\}$$

$$\chi = 0.873143 + (7.23875 \cdot 10^{-5})T$$

$$P_o D'_{ij} = P D_{ij} \quad \mu_i = \mu_i^o \frac{T_i^o + C_i}{T + C_i} \left(\frac{T}{T_i^o} \right)^{1.5}$$

D_{ij} = binary diffusion parameter (See [GasSpeciesDiffusion](#) for calculation)

P_o = standard state pressure (100,000 Pa)

T = gas temperature variable (K)

P = gas pressure variable (Pa)

MW_i = molecular weights of i th species in gas (g/mol)

y_i = mole fraction of gas species i

→ Calculated from the velocity vector components

μ_i^o = Sutherland's reference viscosity for species i

→ See [GasPropertiesBase](#) for notes on input values

T_i^o = Sutherland's reference temperature for species i

→ See [GasPropertiesBase](#) for notes on input values

C_i = Sutherland's constant for species i

→ See [GasPropertiesBase](#) for notes on input values

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the viscosity in the domain given variables for temperature and pressure in that domain. Because this kernel inherits from [GasPropertiesBase](#), there are some

arguments required that are not actually used. That is why it is recommended to push some of those arguments to GlobalParams, so you reduce redundancy in the input files.

```
[./mu_calc]
    type = GasViscosity
    variable = mu
    temperature = T
    pressure = P
    hydraulic_diameter = d      #Note: not used, but a required argument
    uy = vel_y                 #Note: not used, but a required argument
[../]
```

GasVolSpecHeat

Inheritance → [GasPropertiesBase](#)

Notes → This kernel estimates the specific heat at constant volume for the gas. The calculation uses a given ratio of specific heat at constant pressure and constant volume (i.e., c_p/c_v) to return the value of c_v based on the calculated c_p value from [GasSpecHeat](#).

(See [GasPropertiesBase](#) for additional notes and parameter requirements)

Computation

$$\text{Specific Heat at Constant Volume (J/kg/K)} = \frac{c_p}{f}$$

c_p = specific heat at constant pressure (See [GasSpecHeat](#) for calculation)

f = ratio of c_p/c_v given as a constant between 0.56 and 1.67 (Default = 1.4)

Usage

Recall that there are arguments for this kernel pushed to GlobalParams (See [GasPropertiesBase](#))

Example below calculates the specific heat at constant volume for the gas. The 'heat_cap_ratio' is an optional parameter that represents the ratio of the constant pressure specific heat to the constant volume specific heat (i.e., c_p/c_v). Typical values for this ratio is about 1.4. It can range between 0.56 and 1.67.

```
[./cv_calc]
    type = GasVolSpecHeat
    variable = cv
    temperature = T
    pressure = P
    hydraulic_diameter = d      #Note: not used, but a required argument
    uy = vel_y                 #Note: not used, but a required argument
    heat_cap_ratio = 1.4
[../]
```


LinearChangeInTime

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is used to change the value of an auxiliary variable linearly in time. The user gives a simulation time to both start and end the linear increases, as well as a target value to have the auxiliary variable end on. This is particularly useful for simulations of Temperature Programmed Desorption (TPD) curves wherein the temperature in the catalyst system is increased isothermally and linearly over a specified time period.

Computation [Pseudo Code]

```
if (time < start_time)          value = initial_value

if (start_time <= time <= end_time)

    slope = (final_value - initial_value) / (end_time - start_time)
    value = initial_value + slope*(time - start_time)

if (time > end_time)           value = final_value
```

Usage

In the example below, we are simulating a TPD wherein the temperature (T) raises inside the catalyst starting at a time of 13,530 seconds and continuing until 18,300 seconds and reaches a final temperature of 810 K. The initial value (423 K) for the temperature needs to be declared during the creation of the variable under the AuxVariables block in the input file.

```
[AuxVariables]
  [./T]
    order = FIRST
    family = LAGRANGE
    initial_condition = 423
  [../]
[]
[AuxKernels]
  [./TPD]
    type = LinearChangeInTime
    variable = T
    start_time = 13530
    end_time = 18300
    end_value = 810
    execute_on = 'initial timestep_end'
  [../]
[]
```

MicroscaleIntegralAvg

Inheritance → [MicroscaleIntegralTotal](#)

Notes → This kernel inherits from [MicroscaleIntegralTotal](#) to first compute the total space integral of the microscale, then divide that result by the total microscale volume (of a single pellet), thus providing the average concentration of a species contained within the particle.

(See [MicroscaleIntegralTotal](#) for special notes and other notes)

Computation → See [MicroscaleIntegralTotal](#) for calculation information

Usage

Recall that the [MicroscaleDiffusion](#) kernel set requires specific GlobalParams to be defined. It is recommended that you define these parameters as global to reduce input code redundancy.

The example below is for calculation of the average concentration (uAvg) inside of the pellets based on a [MicroscaleDiffusion](#) problem that was divided into 10 nodal variables (u0 through u9). The first node ID is 0 and the space_factor is 1, because the particle is spherical.

```
[GlobalParams]
    micro_length = 1
    num_nodes = 10
    coord_id = 2
[]

[AuxKernels]
    [./avg]
        type = MicroscaleIntegralAvg
        variable = uAvg
        space_factor = 1
        first_node = 0
        micro_vars = 'u0 u1 u2 u3 u4 u5 u6 u7 u8 u9'
        execute_on = 'initial timestep_end'
    [../]
[]
```

MicroscaleIntegralTotal

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is to be used specifically in conjunction with [MicroscaleDiffusion](#) kernels in the hybrid FD/FE method for multiscale mass conservation physics. The user must provide the list of microscale variables at all the nodes in the microscale for a particular conserved quantity (i.e., u0 through uL or v0 through vL). The list of those variables MUST be given in ascending order as well as the node ID for the first node in that list (otherwise result will be erroneous). User must also provide the same specified GlobalParams from the [MicroscaleDiffusion](#) notes and special notes. Optionally, the user may want to specify a 'space_factor' that is used to complete the total integral.

- space_factor → if coord_id = 0, space_factor is cross-sectional area
- if coord_id = 1, space_factor is the length of a cylindrical particle

→ if coord_id = 2, then space_factor is not needed

(See [MicroscaleDiffusion](#) for special notes and other notes)

Computation

Calculation of the integral is carried out using the trapezoid rule for discrete spatial integration of the non-linear variable. The units of the integral will be that of the total mass of material in the pellets (not the mass per volume). To get the mass per volume in the pellets, use [MicroscaleIntegralAvg](#).

Usage

Recall that the [MicroscaleDiffusion](#) kernel set requires specific GlobalParams to be defined. It is recommended that you define these parameters as global to reduce input code redundancy.

The example below is for calculation of the total mass (uTotal) inside of the pellets based on a [MicroscaleDiffusion](#) problem that was divided into 10 nodal variables (u0 through u9). The first node ID is 0 and the space_factor is 1, because the particle is spherical.

```
[GlobalParams]
  micro_length = 1
  num_nodes = 10
  coord_id = 2
[]

[AuxKernels]
  [./total]
    type = MicroscaleIntegralTotal
    variable = uTotal
    space_factor = 1
    first_node = 0
    micro_vars = 'u0 u1 u2 u3 u4 u5 u6 u7 u8 u9'
    execute_on = 'initial timestep_end'
  [../]
[]
```

[MicroscalePoreVolumePerTotalVolume](#)

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is to be used to calculate an effective microscale pore volume per total reactor volume ratio. This is a specific ratio that you generally only see used in the accumulation term, or time derivative term, that represents the microscale or pore-space concentration in a material balance. You would use this kernel if you microscale or pore-space mass balance was based on the total volume instead of on the microscale volume. This is most commonly the case when you model the microscale on an average basis instead of discretizing the microscale.

Computation

Micropore Volume per total volume (volume of micropores / total volume) = $\varepsilon_p(1 - \varepsilon_b)$

ε_p = porosity of the microscale (i.e., particle/pellet/washcoat porosity)

ε_b = porosity of the macroscale (i.e., bulk bed porosity)

Usage

Couple with variables for microscale and macroscale porosity to formulate an auxiliary variable and corresponding calculation for the micro-pore volume to total volume ratio. In this example, 'total_pore' is the variable representing the total micro-porosity per total volume ratio. The porosity is represented by 'eps_b' and the particle/micro-scale porosity is 'eps_p'.

```
[AuxKernels]
  [./total_pore_calc]
    type = MicroscalePoreVolumePerTotalVolume
    variable = total_pore
    porosity = eps_b
    microscale_porosity = eps_p
    execute_on = 'initial timestep_end'
  [../]
[]
```

MonolithAreaVolumeRatio

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is used to calculate the ratio of monolith surface area to monolith volume (or to the total volume depending on user input and application). User must provide the cell density of the monolith (# cells per area) and the bulk voids fraction (volume of open channels per total reactor volume) for the monolith structure in question. By default, the calculation will return the ratio of monolith area to monolith volume. However, the user can specify that they would like to return the ratio of monolith area to total volume instead. Units on return will be in the same unit basis as the cell density (i.e., if the cell density is given in cells per square cm, then the return will be cm^{-1}).

Computation

Area-volume Ratio of Monolith (surface area / volume of monolith) = G_a

If (want area per total volume)

$$G_a = 4 \cdot (cd) \cdot d_h$$

If (want area per monolith volume) - Default

$$G_a = 4 \cdot (cd) \cdot d_h / (1 - \varepsilon_b)$$

ϵ_b = bulk channel voids to total volume ratio

(cd) = cell density (# cells per face area of monolith)

d_h = hydraulic diameter of open channels (as square with rounded corners)

$$= 0.5(d_c + d_s)$$

$$d_s = \left(\frac{\epsilon_b}{(cd)} \right)^{1/2} \quad d_c = 2 \left(\frac{\epsilon_b}{\pi(cd)} \right)^{1/2}$$

Usage

Couple with a variable for the channel volume ratio (eps_b), user provides the cell density and (optionally) a Boolean for whether to calculate on the ratio of the monolith/solids volume. In this example, the user wants to calculate on a total volume ratio, so that option is set to 'false'.

```
[AuxKernels]
  [./Ga_calc]
    type = MonolithAreaVolumeRatio
    variable = Ga
    channel_vol_ratio = eps_b
    cell_density = 50
    per_solids_volume = false
    execute_on = 'initial timestep_end'
  [../]
[]
```

MonolithMicroscaleTotalThickness

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is used to calculate the total thickness to be used as the 'micro_length' parameter for setting up a coupled Microscale simulation for a washcoated monolith. Due to how the Microscale kernels are implemented (see [MicroscaleCoefTimeDerivative](#)), this parameter cannot be automatically set by this kernel (i.e., you cannot set the 'micro_length' as an auxiliary variable). Thus, it is recommended that you calculate this value in an input file first, then input the calculated value as a constant into 'micro_length'. Units on output are of the same basis of the units on input (i.e., if cell density is in cm^{-2} , then thickness comes out in cm).

Why not just use washcoat thickness for 'micro_length'?

The manner in which the Microscale kernels work is to discretize the microscale with finite differences and apply diffusion kernels onto those microscale variables. In a monolith system, you are diffusing into more than 1 wall/washcoat at any location in the macroscale (e.g., if you have square channels, you have 4 walls to diffuse into). Therefore, if you only provide the washcoat thickness for 'micro_length', then you will effectively be only allowing diffusion into 1 of those 4 walls. This will result in a significant underestimation of the level of mass transfer occurring in the system. The 'MonolithMicroscaleTotalThickness' is a singular calculation for

thickness that will implicitly include all walls (regardless of the number of walls) by creating an effective thickness to account for the total volume of washcoating.

Alternatively, you could use multiple sets of microscale variables to represent the concentration of the microscale for each monolith wall separately, then apply separate mass-transfer and diffusion kernels for each of those sets of concentrations. Doing it in this way would allow you to use the true washcoat thickness for the microscale, but requires significantly more work to setup in the input file. That is why we generally recommend just using this simple calculation to get an effective thickness. **NOTE:** This kernel can also be used to estimate the actual washcoat thickness by providing an options 'wall_factor' value that will divide the total thickness by this factor (i.e., a representation of the number of walls). See this [example](#) for details.

Computation

$$\text{Effective Washcoat Thickness (units of length)} = \left[\frac{1}{(cd)} - d_h^2 \right]^{1/2} / n_w$$

$$d_h = 0.5(d_c + d_s)$$

$$d_s = \left(\frac{\varepsilon_b}{(cd)} \right)^{1/2} \quad d_c = 2 \left(\frac{\varepsilon_b}{\pi(cd)} \right)^{1/2}$$

ε_b = bulk channel voids to total volume ratio

(cd) = cell density (# cells per face area of monolith)

n_w = (optional) effective number of walls

[Default value = 1] Only give the number of walls if you plan to expand out all the microscale diffusion kernels for each individual wall.

Usage

Couple with a variable for the channel volume ratio (eps_b) and user provides the cell density.

```
[AuxKernels]
  [./wt_calc]
    type = MonolithMicroscaleTotalThickness
    variable = wt
    channel_vol_ratio = eps_b
    cell_density = 50
    execute_on = 'initial timestep_end'
  [../]
[]
```

MonolithHydraulicDiameter

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is used to calculate the hydraulic diameter of the channels in a monolith. This is not commonly used in individual kernels, but is a calculation that may be used elsewhere

in the auxiliary system (see [SimpleGasPropertiesBase](#)). We do not provide an equivalent calculation for the hydraulic diameter of particles since that diameter is usually just the diameter of the particle itself, and thus, there is nothing to calculate. Units on return will be in the same unit basis as the cell density (i.e., if the cell density is given in cells per square cm, then the return will be cm^{-1}).

Computation

Hydraulic Diameter of Channel (units of length) = d_h

d_h = hydraulic diameter of open channels (as square with rounded corners)

$$= 0.5(d_c + d_s)$$

$$d_s = \left(\frac{\varepsilon_b}{(cd)} \right)^{1/2} \quad d_c = 2 \left(\frac{\varepsilon_b}{\pi(cd)} \right)^{1/2}$$

ε_b = bulk channel voids to total volume ratio

(cd) = cell density (# cells per face area of monolith)

Usage

Couple with a variable for the channel volume ratio (eps_b) and user provides the cell density.

```
[AuxKernels]
  [./dh_calc]
    type = MonolithHydraulicDiameter
    variable = dh
    channel_vol_ratio = eps_b
    cell_density = 50
    execute_on = 'initial timestep_end'
  [../]
[]
```

[SimpleGasPropertiesBase](#)

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This is the base object that defines how all ‘SimpleGas’ auxiliary values are calculated. It sets up the variables needed and provides a way to interface with some utilities and subroutines for performing unit conversions based on user inputs. Because of this, all the ‘SimpleGas’ auxiliary kernels require a common set of input parameters, although some do require additional arguments beyond the base set of arguments.

Since most of those input arguments are common to all other ‘SimpleGas’ auxiliary kernels, you can put those arguments in the ‘GlobalParams’ block of you input file (see example below). However, some of the arguments in the derived ‘SimpleGas’ auxiliary kernels may be specific to a particular gas species or physical property.

Units for specific inputs are provided as strings. Errors will be reported if user provides units that are not currently supported.

[GlobalParams]

```
# You may have the arguments below in GlobalParams
temperature = temp_var      # in K
pressure = press_var        # in kPa
micro_porosity = eps_p
macro_porosity = eps_b
velocity = vel_y
vel_length_unit = "cm"
vel_time_unit = "min"
execute_on = 'initial timestep_end'

# Arguments in the below list can also go in GlobalParams
#       or defined locally in other auxiliary kernels if desired
#       NOTE: it is usually recommended to provide these locally
characteristic_length = dh   #hydraulic diameter
char_length_unit = "cm"
ref_diffusivity = 0.2        #diffusivity of H2O in air (@STP)
diff_length_unit = "cm"
diff_time_unit = "s"
ref_diff_temp = 273          # in K

# This factor is a power applied to the micro_porosity to
#       estimate an effective pore diffusion rate. It ranges
#       from 1 to 2, and defaults to 1.4.
effective_diffusivity_factor = 1.4 #Default
```

[]

How is this different from [GasPropertiesBase](#)?

The original interface for calculating gas properties ([GasPropertiesBase](#)) is more difficult to use and more strict on units. Thus, the purpose of this auxiliary system was to simplify the user interface and allow for more flexibility on units. The downside is that these auxiliary kernels using the 'SimpleGas' approach will always make the assumption of an ideal gas and will always assume that your gas is a dilute system that is primarily made up of standard air (i.e., mostly N₂, O₂, and H₂O, with other gas species being on order of ppm to 1% by volume). These assumptions allow us to use simpler approximations for gas properties, like viscosity, density, and molecular diffusivity, and free up the strict unit convention to allow users to give inputs and receive outputs in units they need.

Currently Supported Units

length = "m", "cm", "mm"

time = "hr", "min", "s"

(More will be added as needed)

Computation

For convenience, several standard calculations are presented below. These are the calculations used for gas density, gas viscosity, molecular diffusivity, and other calculations.

$$\rho = \frac{P}{287.058 \cdot T} \quad \mu = 1.458 \times 10^{-5} \cdot \frac{T^{1.5}}{(110.4 + T)} \quad D_m = D_{ref} \cdot \exp\left(-887.5 \left[\frac{1}{T} - \frac{1}{T_{ref}}\right]\right)$$

P = gas pressure (in kPa)

T = gas temperature (in K)

ρ = gas density

μ = gas viscosity

D_{ref} = molecular diffusivity of a gas species of interest at a reference temperature

T_{ref} = reference temperature for molecular diffusivity

D_m = molecular diffusivity of a gas species of interest at system gas temperature

$$Re = \frac{\rho v d}{\mu} \quad Sc = \frac{\mu}{\rho D_m} \quad Pr = \frac{c_p \mu}{K_g}$$

Re = Reynolds number

v = average gas velocity

d = characteristic length (or hydraulic diameter)

Sc = Schmidt number

Pr = Prandtl number

c_p = isobaric heat capacity (see [SimpleGasIsobaricHeatCapacity](#))

K_g = thermal conductivity (see [SimpleGasThermalConductivity](#))

Usage → None, this object should not be invoked in the input files.

SimpleGasMonolithMassTransCoef

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the film mass transfer coefficient for a gas species in a monolith system. That calculation is based on relationships with the Sherwood number and is a function of both the Reynolds and Schmidt numbers. Users must include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Mass Transfer Rate (length / time)} \Rightarrow k_m = \frac{Sh \cdot D_{eff}}{d_h}$$

D_{eff} = Effective diffusivity in the micropores

$$= (\varepsilon_w)^f \cdot D_m$$

ε_w = porosity of the microscale (i.e., washcoat)

f = effective diffusivity factor (default = 1.4)

D_m = molecular diffusivity of the species (see [SimpleGasPropertiesBase](#))

Sh = Sherwood number in a monolith geometry

$$= 0.3 + \left\{ 0.62 Re^{1/2} Sc^{1/3} \cdot \left[1 + (0.4/Sc)^{2/3} \right]^{-1/4} \right\} \cdot \left\{ 1 + (Re/282000)^{5/8} \right\}^{4/5}$$

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the mass transfer coefficient of O₂ gas into the washcoat. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User provides O₂ diffusivity reference in cm²/s and outputs the mass transfer term in cm/min.

```
[./km_O2_calc]
  type = SimpleGasMonolithMassTransCoef
  variable = km_O2

  #NOTE: we skipped the other parameters set here (assuming they
  #      got set in GlobalParams)

  ref_diffusivity = 0.561  #diffusivity of O2 in air (@473K)
  diff_length_unit = "cm"
  diff_time_unit = "s"
  ref_diff_temp = 473      # in K

  output_length_unit = "cm"
  output_time_unit = "min"

[../]
```

[SimpleGasMonolithHeatTransCoef](#)

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the heat transfer coefficient in a monolith system at the interface between the open channels and the washcoat/monolith walls. That calculation is based on relationships with the Nusselt number and is a function of both the Reynolds and Prandtl numbers. Users must include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Heat Transfer Rate (energy / time / length}^2 \text{ / temperature)} \Rightarrow h_s = \frac{Nu \cdot K_g}{d_h}$$

K_g = thermal conductivity of gas (see [SimpleGasThermalConductivity](#))

d_h = hydraulic diameter

Nu = Nusselt number in a monolith geometry

$$= 0.35 \cdot Re^{1/2} Pr^{-1/6}$$

Pr = Prandtl number (see [SimpleGasPropertiesBase](#))

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the heat transfer coefficient into the washcoat. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User requests output in J/min/cm²/K.

```
[./hs_calc]
  type = SimpleGasMonolithHeatTransCoef
  variable = hs

  #NOTE: we skipped the other parameters set here (assuming they
  #      got set in GlobalParams). Here only the important params
  #      are being set.
  pressure = 101.35
  temperature = Tf
  characteristic_length = dh
  char_length_unit = "cm"

  output_length_unit = "cm"
  output_time_unit = "min"
  output_energy_unit = "J"

[../]
```

[SimpleGasSphericalMassTransCoef](#)

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the film mass transfer coefficient for a gas species in a fixed-bed system, assuming the particles are spherical in shape. That calculation is based on relationships with the Sherwood number and is a function of both the Reynolds and Schmidt numbers. Users must include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Mass Transfer Rate (length / time)} \Rightarrow k_m = \frac{Sh \cdot D_{eff}}{d_h}$$

D_{eff} = Effective diffusivity in the micropores

$$= (\varepsilon_w)^f \cdot D_m$$

ε_w = porosity of the microscale (i.e., spherical particles)

f = effective diffusivity factor (default = 1.4)

D_m = molecular diffusivity of the species (see [SimpleGasPropertiesBase](#))

Sh = Sherwood number in a monolith geometry

$$= 2 + \{0.4Re^{1/2} + 0.06Re^{0.67}\} \cdot Sc^{0.4}$$

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the mass transfer coefficient of O₂ gas into the particles. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User provides O₂ diffusivity reference in cm²/s and outputs the mass transfer term in cm/min.

```
[./km_O2_calc]
type = SimpleGasSphericalMassTransCoef
variable = km_O2

#NOTE: we skipped the other parameters set here (assuming they
#      got set in GlobalParams)

ref_diffusivity = 0.561  #diffusivity of O2 in air (@473K)
diff_length_unit = "cm"
diff_time_unit = "s"
ref_diff_temp = 473      # in K

output_length_unit = "cm"
output_time_unit = "min"

[../]
```

SimpleGasSphericalHeatTransCoef

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the heat transfer coefficient in a fixed-bed system at the interface between the surface of spherical particles and the bulk fluid phase. That calculation is based on relationships with the Nusselt number and is a function of both the Reynolds and Prandtl numbers. Users must include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Heat Transfer Rate (energy / time / length}^2 \text{ / temperature)} \Rightarrow h_s = \frac{Nu \cdot K_g}{d_h}$$

K_g = thermal conductivity of gas (see [SimpleGasThermalConductivity](#))

d_h = hydraulic diameter

Nu = Nusselt number in a monolith geometry

$$= 2 + 0.4 \cdot Re^{1/2} Pr^{1/3}$$

Pr = Prandtl number (see [SimpleGasPropertiesBase](#))

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the heat transfer coefficient into the particles. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User requests output in J/min/cm²/K.

```
[./hs_calc]
```

```
type = SimpleGasSphericalHeatTransCoef  
variable = hs
```

```
#NOTE: we skipped the other parameters set here (assuming they  
#      got set in GlobalParams). Here only the important params  
#      are being set.
```

```
pressure = 101.35  
temperature = Tf  
characteristic_length = 0.2  
char_length_unit = "cm"
```

```
output_length_unit = "cm"  
output_time_unit = "min"  
output_energy_unit = "J"
```

```
[../]
```

SimpleGasCylinderWallHeatTransCoef

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the heat transfer coefficient at the walls of a cylindrical reactor. This will primarily be used to evaluate the heat transfer at a physical boundary of the domain. That calculation is based on relationships with the Nusselt number and is a function of both the Reynolds and Prandtl numbers. Users must include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Heat Transfer Rate (energy / time / length}^2 \text{ / temperature)} \Rightarrow h_s = \frac{Nu \cdot K_g}{d_h}$$

K_g = thermal conductivity of gas (see [SimpleGasThermalConductivity](#))

d_h = hydraulic diameter

Nu = Nusselt number in a monolith geometry

$$= 0.27 \cdot Re^{4/5} Pr^{1/3}$$

Pr = Prandtl number (see [SimpleGasPropertiesBase](#))

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the heat transfer coefficient at the walls of the domain (assuming a cylindrical reactor geometry). All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User requests output in J/min/cm²/K.

NOTE: The characteristic length for this kernel is the diameter of the reactor (not the hydraulic diameter of a monolith or a particle size.)

```
[./hw_calc]
```

```
type = SimpleGasCylinderWallHeatTransCoef  
variable = hw
```

```
#NOTE: we skipped the other parameters set here (assuming they  
#      got set in GlobalParams). Here only the important params  
#      are being set.  
pressure = 101.35  
temperature = Tf  
characteristic_length = 2  
char_length_unit = "cm"
```

```

output_length_unit = "cm"
output_time_unit = "min"
output_energy_unit = "J"
[../]

```

SimpleGasEffectivePoreDiffusivity

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the effective pore diffusivity (or applied pore diffusivity) of a gas species in the microscale dimension. The difference from this diffusivity and [SimpleGasPoreDiffusivity](#) is that this kernel calculates a variable that would be used directly into the Microscale kernel material balance ([MicroscaleVariableDiffusion](#), etc), where as [SimpleGasPoreDiffusivity](#) is just a calculation of the true pore diffusivity (D_p). Users should specify the volume basis of their microscale material balance to get the correct effective term calculated by setting the 'per_solids_volume' Boolean on input.

Users must also include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

Effective Pore Diffusivity (length² / time) = D_{eff}

If 'per_solids_volume' == true

$$D_{eff} = \varepsilon_w D_p$$

else

$$D_{eff} = \varepsilon_w (1 - \varepsilon_b) D_p$$

$$D_p = (\varepsilon_w)^f \cdot D_m$$

ε_w = porosity of the microscale (i.e., washcoat or particles)

f = effective diffusivity factor (default = 1.4)

ε_b = porosity of the macroscale (i.e., volume solids per total volume)

D_m = molecular diffusivity of the species (see [SimpleGasPropertiesBase](#))

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the effective diffusivity of O₂ gas through the pore space of the microscale to use in Microscale kernels. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User provides O₂ diffusivity reference in cm²/s and outputs the diffusion term in cm²/min.

```
[./De_O2_calc]
    type = SimpleGasEffectivePoreDiffusivity
    variable = De_O2

    #NOTE: we skipped the other parameters set here (assuming they
    #       got set in GlobalParams)

    ref_diffusivity = 0.561  #diffusivity of O2 in air (@473K)
    diff_length_unit = "cm"
    diff_time_unit = "s"
    ref_diff_temp = 473      # in K

    per_solids_volume = false
    output_length_unit = "cm"
    output_time_unit = "min"

[../]
```

SimpleGasPoreDiffusivity

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the pore diffusivity of a gas species in the microscale dimension. Users must include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Pore Diffusivity (length}^2 \text{ / time)} = (\varepsilon_w)^f \cdot D_m$$

ε_w = porosity of the microscale (i.e., washcoat or particles)

f = effective diffusivity factor (default = 1.4)

D_m = molecular diffusivity of the species (see [SimpleGasPropertiesBase](#))

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the diffusivity of O₂ gas through the pore space of the microscale. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User provides O₂ diffusivity reference in cm²/s and outputs the diffusion term in cm²/min.

```
[./D_O2_calc]
    type = SimpleGasPoreDiffusivity
    variable = D_O2

    #NOTE: we skipped the other parameters set here (assuming they
```



```

#      got set in GlobalParams)

ref_diffusivity = 0.561  #diffusivity of O2 in air (@473K)
diff_length_unit = "cm"
diff_time_unit = "s"
ref_diff_temp = 473      # in K

output_length_unit = "cm"
output_time_unit = "min"

[../]

```

SimpleGasEffectiveKnudsenDiffusivity

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the effective Knudsen diffusivity (or applied Knudsen diffusivity) of a gas species in the microscale dimension. The Knudsen diffusivity generally only applies for microscale physics in situations where the pore-space molecules diffuse through are very, very narrow pores (i.e., not mesoporous materials). The total diffusivity is then a non-linear combination of pore diffusivity and a Knudsen diffusion correction. The difference from this diffusivity and [SimpleGasKnudsenDiffusivity](#) is that this kernel calculates a variable that would be used directly into the Microscale kernel material balance ([MicroscaleVariableDiffusion](#), etc), where as [SimpleGasKnudsenDiffusivity](#) is just a calculation of the true Knudsen diffusivity. Users must include the unit basis they want for the calculated value at the end. Users should specify the volume basis of their microscale material balance to get the correct effective term calculated by setting the 'per_solid_volume' Boolean on input.

NOTE: The 'characteristic_length' for this auxiliary kernel must be average pore radius.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Effective Knudsen Diffusivity (length}^2 \text{ / time)} = D_{k,eff}$$

If 'per_solid_volume' == true

$$D_{k,eff} = \varepsilon_w [D_k^{-1} + D_p^{-1}]^{-1}$$

else

$$D_{k,eff} = \varepsilon_w (1 - \varepsilon_b) [D_k^{-1} + D_p^{-1}]^{-1}$$

$$D_k = 9700 r_p \sqrt{T / MW}$$

r_p = pore radius (in cm)

T = temperature (in K)

MW = molecular weight (in g/mol)

$$D_p = (\varepsilon_w)^f \cdot D_m$$

ε_w = porosity of the microscale (i.e., washcoat or particles)

f = effective diffusivity factor (default = 1.4)

D_m = molecular diffusivity of the species (see [SimpleGasPropertiesBase](#))

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the Knudsen diffusivity of O₂ gas through the pore space of the microscale. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User provides O₂ diffusivity reference in cm²/s and outputs the diffusion term in cm²/min.

```
[./Dk_O2_calc]
  type = SimpleGasEffectiveKnudsenDiffusivity
  variable = Dk_O2

  #NOTE: we skipped the other parameters set here (assuming they
  #      got set in GlobalParams)

  ref_diffusivity = 0.561  #diffusivity of O2 in air (@473K)
  diff_length_unit = "cm"
  diff_time_unit = "s"
  ref_diff_temp = 473      # in K

  characteristic_length = 2E-6 # average pore radius
  char_length_unit = "mm"
  molar_weight = 32           # g/mol
  per_solids_volume = false

  output_length_unit = "cm"
  output_time_unit = "min"

[../]
```

[SimpleGasKnudsenDiffusivity](#)

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the Knudsen diffusivity of a gas species in the microscale dimension. The Knudsen diffusivity generally only applies for microscale physics in situations where the pore-space molecules diffuse through are very, very narrow pores (i.e., not mesoporous materials). The total diffusivity is then a non-linear combination of pore diffusivity

and a Knudsen diffusion correction. Users must include the unit basis they want for the calculated value at the end.

NOTE: The 'characteristic_length' for this auxiliary kernel must be average pore radius.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Knudsen Diffusivity (length}^2 \text{ / time)} = [D_k^{-1} + D_p^{-1}]^{-1}$$

$$D_k = 9700 r_p \sqrt{T/MW}$$

r_p = pore radius (in cm)

T = temperature (in K)

MW = molecular weight (in g/mol)

$$D_p = (\varepsilon_w)^f \cdot D_m$$

ε_w = porosity of the microscale (i.e., washcoat or particles)

f = effective diffusivity factor (default = 1.4)

D_m = molecular diffusivity of the species (see [SimpleGasPropertiesBase](#))

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the Knudsen diffusivity of O₂ gas through the pore space of the microscale to use in Microscale kernels. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User provides O₂ diffusivity reference in cm²/s and outputs the diffusion term in cm²/min.

```
[./Dk_O2_calc]
```

```
type = SimpleGasKnudsenDiffusivity
```

```
variable = Dk_O2
```

```
#NOTE: we skipped the other parameters set here (assuming they
```

```
#      got set in GlobalParams)
```

```
ref_diffusivity = 0.561 #diffusivity of O2 in air (@473K)
```

```
diff_length_unit = "cm"
```

```
diff_time_unit = "s"
```

```
ref_diff_temp = 473 # in K
```

```
characteristic_length = 2E-6 # average pore radius
```

```
char_length_unit = "mm"
```

```

molar_weight = 32          # g/mol

output_length_unit = "cm"
output_time_unit = "min"

[../]

```

SimpleGasDispersion

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the dispersion coefficient of a gas species in the macroscale dimension. When using this kernel, users must provide the diameter of the reactor for the 'characteristic_length'. The calculation of dispersion is based on an empirical relationship with average linear velocity, column/reactor diameter, and the dimensionless numbers for Reynolds and Schmidt. Users must include the unit basis they want for the calculated value at the end.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Dispersion Coefficient (length}^2 \text{ / time)} = v \cdot d \cdot \left(0.5 + \frac{20}{Re \cdot Sc}\right)$$

d = reactor diameter (i.e., characteristic_length)

v = average linear velocity

Re = Reynolds number

Sc = Schmidt number

(see [SimpleGasPropertiesBase](#) for other calculation information)

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the dispersion coefficient of O₂ gas through the macroscale of the domain. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User provides O₂ diffusion reference in cm²/s and outputs the dispersion term in cm²/min.

```

[../D_O2_calc]
type = SimpleGasDispersion
variable = D_O2

#NOTE: we skipped the other parameters set here (assuming they
#      got set in GlobalParams)

ref_diffusivity = 0.561  #diffusivity of O2 in air (@473K)
diff_length_unit = "cm"

```

```

diff_time_unit = "s"
ref_diff_temp = 473    # in K

characteristic_length = 2 # diameter of reactor
char_length_unit = "cm"

output_length_unit = "cm"
output_time_unit = "min"

[../]

```

SimpleGasDensity

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the density of the gas phase assuming it behaves as an ideal gas and is primarily standard air (i.e., roughly 79 % N₂ and 21 % O₂).

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Density (mass / length}^3\text{)} = \frac{P}{R_g T}$$

P = gas pressure (kPa)

T = gas temperature (K)

R_g = Standard Air gas constant = 287.058 kPa*cm³/g/K

NOTE: This calculation gives g/cm³ for density (but users can request different units on output)

(see [SimpleGasPropertiesBase](#) for other calculation information)

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the density of air for variable 'rho' based on fluid temperature 'Tf'. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User sets the 'output' unit arguments to get density back in kg/cm³.

```

[../rho_calc]
  type = SimpleGasDensity
  variable = rho

  #NOTE: we skipped the other parameters set here (assuming they
  #      got set in GlobalParams). Most relevant parameters are
  #      being set below.
  pressure = 101.35
  temperature = Tf

```

```

        output_length_unit = "cm"
        output_mass_unit = "kg"
[../]

```

SimpleGasViscosity

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the viscosity of the gas phase assuming it behaves as an ideal gas and is primarily standard air (i.e., roughly 79 % N₂ and 21 % O₂).

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Viscosity (mass / length / time)} = 1.458 \times 10^{-5} \cdot \frac{T^{1.5}}{(110.4+T)}$$

T = gas temperature (K)

NOTE: This calculation gives g/cm/s for viscosity (users can request different units on output)

(see [SimpleGasPropertiesBase](#) for other calculation information)

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the density of air for variable 'mu' based on fluid temperature 'Tf'. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User sets the 'output' unit arguments to get density back in kg/cm/min.

```

[../mu_calc]
    type = SimpleGasViscosity
    variable = mu

    #NOTE: we skipped the other parameters set here (assuming they
    #      got set in GlobalParams). Most relevant parameters are
    #      being set below.
    pressure = 101.35
    temperature = Tf

    output_length_unit = "cm"
    output_mass_unit = "kg"
    output_time_unit = "min"
[../]

```

SimpleGasIsobaricHeatCapacity

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the heat capacity (at constant pressure) of the gas phase assuming it behaves as an ideal gas and is primarily made up of standard air (i.e., roughly 79 % N₂ and 21 % O₂). This is an empirical formulation based on data for standard air heat capacity as a function of temperature and pressure. These approximations are good between 1 to 20 bar of pressure and 0 to 1600 °C.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Heat Capacity (energy/ mass / temperature)} = 1 + c_{p,max} \frac{K_c T^{2.5}}{1 + K_c T^{2.5}}$$

$c_{p,max}$ = maximum heat capacity = 0.3 kJ/kg/K

K_c = heat capacity coefficient = 2.68588E-08 K^{-2.5}

T = gas temperature (K)

(see [SimpleGasPropertiesBase](#) for other calculation information)

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the heat capacity of air for variable 'cp' based on fluid temperature 'Tf'. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User sets the 'output' unit arguments to get heat capacity back in J/kg/K.

```
[./cp_calc]
  type = SimpleGasIsobaricHeatCapacity
  variable = cp

  #NOTE: we skipped the other parameters set here (assuming they
  #       got set in GlobalParams). Most relevant parameters are
  #       being set below.
  pressure = 101.35
  temperature = Tf

  output_energy_unit = "J"
  output_mass_unit = "kg"

[../]
```

SimpleGasThermalConductivity

Inheritance → [SimpleGasPropertiesBase](#)

Notes → This kernel calculates the thermal conductivity of the gas phase assuming it behaves as an ideal gas and is primarily made up of standard air (i.e., roughly 79 % N₂ and 21 % O₂). This is

an empirical formulation based on data for standard air to calculate thermal conductivity as a function of the gas temperature.

(See [SimpleGasPropertiesBase](#) for additional notes and calculations)

Computation

$$\text{Conductivity (energy / time / length / temperature)} = 2.66 \times 10^{-4} \cdot T^{0.805}$$

T = gas temperature (K)

(see [SimpleGasPropertiesBase](#) for other calculation information)

Usage

Recall that there are arguments for this pushed to GlobalParams (See [SimpleGasPropertiesBase](#))

Example below calculates the thermal conductivity of air for variable 'Kg' based on fluid temperature 'Tf'. All common parameters were set in 'GlobalParams' (see [SimpleGasPropertiesBase](#)) and parameters unique to this kernel are set here. User sets the 'output' unit arguments to get conductivity back in J/min/cm/K.

```
[./Kg_calc]
  type = SimpleGasThermalConductivity
  variable = Kg

  #NOTE: we skipped the other parameters set here (assuming they
  #      got set in GlobalParams). Most relevant parameters are
  #      being set below.
  pressure = 101.35
  temperature = Tf

  output_energy_unit = "J"
  output_length_unit = "cm"
  output_time_unit = "min"
[./]
```

SolidsVolumeFraction

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is to be used to calculate the solids volume ratio for the system given the voids volume ratio as an input variable/parameter. This is a specific ratio that you may use in other mass balances to convert units from a solids volume basis to a total volume basis. The units for the solids volume fraction is volume of solids per total system volume.

Computation

$$\text{Solids volume fraction (volume of solids / total volume)} = (1 - \varepsilon_b)$$

ε_b = porosity of the macroscale (i.e., bulk bed porosity)

Usage

Couple with the variable for porosity to formulate an auxiliary variable and corresponding calculation for the solids to total volume ratio. In this example, 'non_pore' is the variable representing the solids volume ratio. The porosity is represented by 'eps_b'.

```
[AuxKernels]
  [./non_pore_calc]
    type = SolidsVolumeFraction
    variable = non_pore
    porosity = eps_b
    execute_on = 'initial timestep_end'
  [../]
[]
```

SphericalAreaVolumeRatio

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is to be used to calculate the area to volume ratio for spherical particles. By default, this will return the area of the particles per the volume of the particles, however, user's may request to return the area of the particles per total volume. This can be useful depending on how your mass balances are formulated. Units on output will follow same unit basis on input (i.e., if user provides cm as units for particle diameter, then the area to volume ratio will come out as cm^{-1}).

Computation

Area-volume Ratio of spheres (surface area / volume of spheres) = G_a

If (want area per total volume)

$$G_a = (1 - \varepsilon_b) \cdot 6/d_p$$

If (want area per particle volume) - Default

$$G_a = 6/d_p$$

ε_b = porosity of the macroscale (i.e., bulk bed porosity)

d_p = particle diameter

Usage

Couple with the variable for porosity to formulate an auxiliary variable and corresponding calculation for the ratio of spherical area to volume. In this example, user is requesting to get output in terms of total volume, thus, user must provide porosity and set the 'per_solid_volume' option to 'false'.

```
[AuxKernels]
  [./Ga_calc]
```

```

type = SphericalAreaVolumeRatio
variable = Ga
particle_diameter = 1
porosity = eps_b
per_solids_volume = false
execute_on = 'initial timestep_end'
[../]
[]

```

VoidsVolumeFraction

Inheritance → AuxKernel (i.e., the MOOSE base class for the auxiliary system)

Notes → This kernel is to be used to calculate the voids volume ratio for a packed-bed system given the bulk packing density of the bed and an approximate single particle density (calculated from average particle size and mass). This is a specific ratio that is also known as the bulk bed porosity and is used in many other kernels. Typically, you may just give this value in other kernels as a constant, but this auxiliary kernel will estimate it for you and check to see if the calculated value is reasonable for a packed-bed system. Warnings will be output if the calculated value is deemed unreasonable, and by default, it will override 'bad' values with theoretical maximums or minimums for spherical packing. User's are responsible for making sure that the units in the system work out. If you give bulk packing density in g/cm³, then you must give particle mass in g and particle size in cm.

Computation

$$\text{Voids volume fraction (volume of bulk voids / total volume)} = \left(1 - \frac{\rho_b}{\rho_p}\right)$$

ρ_b = bulk bed packing density (in mass / total volume)

ρ_p = average particle density (in mass of particle / volume of particle)

$$= \frac{6m}{\pi d_p^3}$$

m = average mass of a particle

d_p = average diameter of a particle

Theoretical Max and Min

Max voids fraction = 0.66

Min voids fraction = 0.26

(based on mathematical limitations of sphere packing)

Users may override these limits if desired.

Usage

Users provide bulk packing density, particle mass, and particle diameter. Optionally, users may set the 'override_limits' Boolean to allow for the calculation to produce voids fractions that go beyond the mathematical limits.

```
[AuxKernels]
  [./pore_calc]
    type = VoidsVolumeFraction
    variable = pore
    particle_diameter = 1
    particle_mass = 5
    packing_density = 1
    override_limits = false
    execute_on = 'initial timestep_end'
  [../]
[]
```

Materials

The materials system in MOOSE is generally used as a way to setup parameters in a simulation for each subdomain in the system. In CATS, we accomplish this same functionality through the [Auxiliary](#) system, which has the added advantage of allowing us to seamlessly change parameters into variables without significant changes to the code base. However, since CATS uses the built-in MOOSE modules for Navier-Stokes modeling, and the Navier-Stokes modules use materials for parameters such as density and viscosity, we must provide some basic custom materials to interface with those older codes.

INSFluid

Inheritance → Material (i.e., the MOOSE base class for the material system)

Notes → This kernel is to be used specifically in conjunction with the incompressible Navier-Stokes module to provide an interface between our calculated fluid properties from the auxiliary system to the required material properties of density and viscosity. There are no calculations performed by this system. It simply creates the appropriate material property objects and sets their values to those dictated by our auxiliary variables.

Computation

Density (in kg/m^3) is calculated as in [GasDensity](#).

Viscosity (in kg/m/s) is calculated as in [GasViscosity](#).

(**Note:** Optionally, you can simply replace the required variables in the input value with specified constants for simplicity.)

Usage

This object is to be used in conjunction with the built-in Navier-Stokes incompressible flow module in MOOSE. In the example below, the density is being set to another variable in the system named “rho” and the viscosity is being set to a constant value.

Note: All material objects require the user to specify which “blocks” (i.e., subdomains) the material properties apply to. Whenever a material is declared, all blocks in the mesh must invoke a material object regardless of whether or not that subdomain needs that property.

```
[Materials]
  [./ins_material]
    type = INSFluid
    block = 'washcoat channel'
    density = rho
    viscosity = 1.81E-5
  [./]
[]
```

Utilities

Utilities are codes or subroutines that are not actually a part of the MOOSE framework set of kernels. In CATS, they are used for setting up other MOOSE kernels or performing parameter estimations used in MOOSE simulations, generally through the [Auxiliary](#) system.

Egret

Purpose → Egret is the set of subroutines that actually perform most of the calculations associated with all the [GasPropertiesBase](#) calculations in the [Auxiliary](#) system. Those calculations include, but are not limited to:

- [GasSpecHeat](#)
- [GasSpeciesDiffusion](#)
- [GasSpeciesMassTransCoef](#)
- [GasViscosity](#)

Many of the other Auxiliary kernels for gas properties also use calculations from Egret as a part of their own calculations and computations.

Error

Purpose → This is a helper file for [Egret](#) that is used to provide specific error messages upon failure. User does not need to interface with this subroutine.

Macaw

Purpose → This is a helper file for [Egret](#) that is used to provide a templated dense matrix object used for vector calculations and/or data storage of diffusion tensors.