

UC Berkeley

HVAC Systems

Title

Energy performance of underfloor air distribution systems part V: energy plus development

Permalink

<https://escholarship.org/uc/item/62g683x7>

Author

Buhl, Fred

Publication Date

2007-01-18

Peer reviewed

ENERGY PERFORMANCE OF UNDERFLOOR AIR DISTRIBUTION (UFAD) SYSTEMS PART V: ENERGYPLUS DEVELOPMENT

Fred Buhl

Lawrence Berkeley National Laboratory

Commission Project Manager: Norman Bourassa

PIER Buildings Program

CEC Contract 500-01-035

I VARIABLE SPEED FAN TERMINAL UNIT: ENERGYPLUS DOCUMENTATION

I.1 SINGLE DUCT:VAV:REHEAT:VS FAN – INPUT/OUTPUT REFERENCE

The VAV terminal unit with variable-speed (VS) fan and reheat coil is an air system terminal unit consisting of a variable speed fan in series with a heating coil. These units are usually employed in underfloor air distribution (UFAD) systems where the air is supplied at low static pressure through an underfloor plenum. The fan is used to control the flow of conditioned air that enters the space. When the fan is off the plenum pressure drives the minimum air flow through the terminal unit. At maximum cooling the fan runs at its maximum speed. At full heating the fan runs at its heating maximum – usually less than the cooling maximum flow rate. Thus this unit has two separate maximum flow rates – one for heating and one for cooling.

For cooling, control is maintained simply by varying the fan speed. For heating, the unit first tries to meet the heating load by varying the heating coil output while keeping the air flow at minimum (fan off). If this is not adequate the fan turns on and operates in variable flow mode up to the heating maximum flow rate.

This unit is modeled in EnergyPlus as a compound component – a variable speed fan and a heating coil in series in the air stream. The unit is blow through – the fan is upstream of the heating coil.

Field: Name of unit

A unique user assigned name for a particular VS fan VAV reheat terminal unit. Any reference to this unit by another object will use this name.

Field: Unit availability schedule

The name of the schedule (ref: Schedule) that denotes whether the unit can run during a given hour. A schedule value greater than 0 (usually 1 is used) indicates that the unit can be on during the hour. A value less than or equal to 0 (usually 0 is used) denotes that the unit must be off for the hour.

Field: Maximum cooling air volume flow rate

The maximum volumetric air flow rate through the unit in cubic meters per second when the thermostat is calling for cooling. Normally this is the same as the unit's fan maximum volumetric flow rate.

Field: Maximum heating air volume flow rate

The maximum volumetric air flow rate through the unit in cubic meters per second when the thermostat is calling for heating.

Field: Zone Minimum Air Flow Fraction

The minimum flow rate to the zone while the system is operating, specified as a fraction of the maximum air flow rate. For this unit this is the flow rate when the fan is off.

Field: Unit supply air inlet node

The name of the HVAC system node that is the air inlet node for the terminal unit. This is also the air inlet node for the unit's fan.

Field: Unit supply air outlet node

The name of the HVAC system node that is the air outlet node of the unit. This same node will be the unit heating coil's air outlet node. This node is also a zone inlet node.

Field: Heating coil air inlet node

The name of the HVAC system node that is the air inlet node of the unit's heating coil. The same node is the outlet node of the unit's fan. This is an internal node of this compound component.

Field: Hot water control node

The name of the HVAC system node which regulates the flow of hot water through the unit's hot water heating coil. This should be the same node as the water inlet node of the hot water coil. For gas or electric coils this field should be left blank.

Field: Fan object

The type of fan in the terminal unit. At this time the only type of fan allowed is *FAN:SIMPLE:VariableVolume*.

Field: Fan name

The name of the particular fan object in this terminal unit.

Field: Heating coil object

The type of heating coil in the terminal unit. The choices are:

COIL:Water:SimpleHeating

COIL:Electric:Heating

COIL:Gas:Heating.

Field: Heating coil name

The name of the heating coil object contained in this terminal unit.

Field: Max hot water flow

The maximum hot water volumetric flow rate in m³/sec through the unit's heating coil. If the heating coil is not a hot water coil this field should be left blank.

Field: Min hot water flow

The minimum hot water volumetric flow rate in m³/sec through the unit's heating coil. If the heating coil is not a hot water coil this field should be left blank.

Field: Heating Convergence Tolerance

The control tolerance for the unit heating output. The unit is controlled by matching the unit output to the zone demand. The model must be numerically inverted to obtain a specified output. The convergence tolerance is the error tolerance used to terminate the numerical inversion procedure. Basically this is the fraction:

$$\frac{|Q_{unit,out} - Q_{zone\ load}|}{Q_{zone\ load}} \leq ConvergenceTolerance$$

The default is 0.001.

Table I: Full IDD specification for single duct VAV reheat with VS fan

SINGLE DUCT:VAV:REHEAT:VS FAN,	
A1,	\field Name of unit \required-field
A2,	\field System availability schedule \required-field \type object-list \object-list ScheduleNames
N1,	\field Maximum cooling air volume flow rate \required-field \type real \units m3/s \autosizable \Minimum 0.0
N2,	\field Maximum heating air volume flow rate \required-field \type real \units m3/s \autosizable \Minimum 0.0
N3,	\field Zone Minimum Air Flow Fraction \required-field \note fraction of cooling air flow rate
A3,	\field Unit supply air inlet node \note same as fan inlet node \alpha \required-field
A4,	\field Unit supply air outlet node \note same as heating coil air outlet node \note same as zone inlet node \alpha \required field
A5,	\field heating coil air inlet node \note same as fan outlet node \alpha \required field
A6,	\field Hot water control node \alpha \note same as hot water coil water inlet node
A7,	\field Fan object \required field \type choice \key FAN:SIMPLE:VariableVolume
A8,	\field Fan name \required field \type object-list \object-list FansVAV
A9,	\field Heating coil object \type choice \key COIL:Water:SimpleHeating \key COIL:Electric:Heating \key COIL:Gas:Heating
A10,	\field Heating coil name \type object-list \object-list HeatingCoilName
N4,	\field Max hot water flow \type real \units m3/s \autosizable \ip-units gal/min
N5,	\field Min hot water flow \type real \units m3/s \minimum 0.0 \default 0.0 \ip-units gal/min
N6;	\field Heating Convergence Tolerance \type real \Minimum> 0.0 \Default 0.001

Table 2: Example IDF inputs for single duct VAV reheat with VS fan

SINGLE DUCT:VAV:REHEAT:VS FAN,	
SPACE2-1 VAV Reheat,	!- Name of System
ReheatCoilAvailSched,	!- System Availability schedule
autosize,	!- Maximum cooling air volume flow rate
autosize,	!- Maximum heating air volume flow rate
0.05,	!- Zone Minimum Air Flow Fraction
SPACE2-1 ATU In Node,	!- Unit supply air inlet node
SPACE2-1 In Node,	!- Unit supply air outlet node
SPACE2-1 Zone Coil Air In Node,	!- heating coil air inlet node
SPACE2-1 Zone Coil Water In Node,	!- Hot water control node
FAN:SIMPLE:VariableVolume,	!- Fan object
SPACE2-1 Zone Fan,	!- Fan name
COIL:Water:SimpleHeating,	!- Heating coil object
SPACE2-1 Zone Coil,	!- Heating coil name
autosize,	!- Max hot water flow
0.0,	!- Min hot water flow
0.001;	!- Heating Convergence Tolerance
COIL:Water:SimpleHeating,	
SPACE2-1 Zone Coil,	!- Coil Name
ReheatCoilAvailSched,	!- Available Schedule
autosize,	!- UA of the Coil {W/K}
autosize,	!- Max Water Flow Rate of Coil {m3/s}
SPACE2-1 Zone Coil Water In Node,	!- Coil_Water_Inlet_Node
SPACE2-1 Zone Coil Water Out Node,	!- Coil_Water_Outlet_Node
SPACE2-1 Zone Coil Air In Node,	!- Coil_Air_Inlet_Node
SPACE2-1 In Node;	!- Coil_Air_Outlet_Node
FAN:SIMPLE:VariableVolume,	
SPACE2-1 Zone Fan,	!- Fan Name
FanAvailSched,	!- Available Schedule
0.7,	!- Fan Total Efficiency
125.0,	!- Delta Pressure {Pa}
autosize,	!- Max Flow Rate {m3/s}
0.0,	!- Min Flow Rate {m3/s}
0.9,	!- Motor Efficiency
1.0,	!- Motor In Airstream Fraction
0.00153028,	!- FanCoefficient 1
0.00520806,	!- FanCoefficient 2
1.1086242,	!- FanCoefficient 3
-.11635563,	!- FanCoefficient 4
0.000,	!- FanCoefficient 5
SPACE2-1 ATU In Node,	!- Fan_Inlet_Node
SPACE2-1 Zone Coil Air In Node;	!- Fan_Outlet_Node

1.2 VARIABLE SPEED FAN TERMINAL UNIT - ENGINEERING REFERENCE

1.2.1 OVERVIEW

The VS fan VAV terminal unit (object name: Single Duct:VAV:Reheat:VSFan) is a typical VAV reheat unit with the addition of a variable-speed blower fan to assist in moving supply air from the plenum to the conditioned zone. It is typically used with under-floor air distribution systems (UFAD) where the supply air is sent at low static pressure through an under-floor supply plenum. The fan has two maximum flow rate settings: one for cooling and one for heating. The cooling maximum is usually the actual fan maximum while the heating maximum is a lesser flow rate. The fan is upstream of the heating coil (this is a blow-through unit). The heating coil can be hot-water, electric or gas. Cooling control is obtained by varying the supply air flow rate from the cooling maximum to the minimum flow rate. Heating control is established by varying both the heating coil output (or hot water flow rate for hot water coils) and the supply air flow rate. Note that for this unit the minimum supply air flow rate is the flow rate when the fan is off.

1.2.2 MODEL

The VS fan VAV terminal unit is modeled as a compound component consisting of two sub-components: a fan and a heating coil. In terms of EnergyPlus objects the fan is a *FAN:SIMPLE:VariableVolume* object and the heating coil is a *COIL:Water:SimpleHeating*, *COIL:Electric:Heating* or a *COIL:Gas:Heating*. The terminal unit is a forward model: its inputs are defined by the state of its inlets: namely its air inlet and its hot water inlet, if it has a hot water coil. The outputs of the model are the conditions of the outlet air stream: flow rate, temperature and humidity ratio. The terminal unit data and simulation are encapsulated in the module *SingleDuct*. The main simulation routine for the unit within the module is *SimVAVVS*.

1.2.3 INPUTS AND DATA

The user describes the terminal unit by inputting the name and type of the heating coil and the name and type of the fan. The user must also specify the connectivity of the component by naming the inlet air node; the air node connecting the fan and heating coil (fan outlet, coil inlet); the unit air outlet node (same as the zone inlet node); and hot water inlet node (if any). Maximum flow rates need to be specified (although these can be autosized): maximum cooling and heating air flow rates and the maximum hot water flow rate (if there is a hot water coil). Minimum flow rates are specified by giving a minimum flow fraction for the air flow and a volumetric flow rate minimum for the hot water. For the units with hot water coils the relative convergence tolerance for the hot water flow rate also needs to be input (or allowed to default).

All input data for the VS fan VAV terminal units is stored in the array *Sys*.

1.2.4 CALCULATION

Given the needed inputs, the output is calculated in subroutine *CalcVAVVS*. The temperature and humidity of the supply air stream are taken from the inlet air node. The inlet air flow rate and the hot water flow rate are passed in as parameters. If the coil is electric or gas the coil heating power is passed instead of the hot water flow rate. Then

1. The fan is simulated (call *SimulateFanComponents*). If the fan is off the fan outlet conditions are set to the inlet conditions.
2. The heating coil is simulated (call *SimulateWaterCoilComponents* if the coil is a hot water coil; call *SimulateHeatingCoilComponents* if the coil is gas or electric).

Finally the sensible load met by the terminal unit is calculated and passed back to the calling routine:

$$\dot{Q}_{out} = \dot{m}_{air} c_{p,air} (T_{air,out} - T_{air,zone})$$

Note that data is never explicitly passed between the sub-components. This is all handled automatically by the node connections and the data stored on the nodes.

1.2.5 SIMULATION AND CONTROL

From the result of the zone simulation we have the heating/cooling demand on the terminal unit $\dot{Q}_{tot}$. For the given inlet conditions *CalcVAVVS* will give us the terminal unit heating/cooling output. We need to vary the air or hot water flow rate or the heating coil power (for gas or electric coils) to make the unit output match the demand. To do this we need to numerically invert *CalcVAVVS*: given the output, we want one of the inputs – the air or hot water flow rate or the heating coil power. The numerical inversion is carried out by calling subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero

of a function (the *residual* function) of a single independent variable. In this case the residual function calculates $(\dot{Q}_{tot} - \dot{Q}_{out}) / \dot{Q}_{tot}$. *SolveRegulaFalsi* varies either the air mass flow rate, the hot water mass flow rate or the heating coil power to zero the residual.

The unit is simulated in the following sequence.

- 1) Decide whether the unit is on or off. The unit is off if: a) it is scheduled off; b) the inlet air mass flow rate is zero; or c) the zone thermostat is in the deadband
- 2) If the unit is off, call *CalcVAVVS* with flow rates set to their minimum flows and return.
- 3) If the unit is on, we need to establish the boundaries of 4 conditioning regions: a) active cooling with fan on; b) active heating with fan on; c) active heating with fan off; d) passive cooling with fan off. The heating/cooling demand will fall into one of these regions. Once the correct region is determined, we will know which model input to vary for control and thus how to invert the calculation.
 - a) To establish the boundaries of region a) we call *CalcVAVVS* twice: once with the supply air flow rate set to the cooling maximum, once with the cooling air flow rate set to the minimum. In both cases the heating coil output is at the minimum and the fan is on. Call the 2 cooling outputs $\dot{Q}_{cool,max,fanon}$ and $\dot{Q}_{cool,min,fanon}$. Remembering that EnergyPlus convention is that cooling loads are negative, then if $\dot{Q}_{tot} < \dot{Q}_{cool,max,fanon}$ the terminal unit can not meet the demand. Set the air mass flow rate to the cooling maximum and call *CalcVAVV* again. This concludes the simulation. If $\dot{Q}_{cool,max,fanon} < \dot{Q}_{tot} < \dot{Q}_{cool,min,fanon}$ the cooling demand is in the active cooling region. We hold the heating at the minimum, allow the supply air flow to vary between the cooling maximum and the minimum with the fan on, and call *SolveRegulaFalsi* to obtain the supply air flow rate that will produce the unit sensible cooling output that matches the demand. This concludes the simulation.
 - b) To establish the boundaries of region b) call *CalcVAVVS* twice: once with the supply air flow rate set to the heating maximum, once with the supply air flow rate set to the minimum. In both calls, if the heating coil is a hot water coil, the hot water flow rate is at the maximum. For electric and gas coils, the heating power is set to the maximum at maximum supply air flow and to zero at the minimum supply air flow. In both calls the fan is set to be on. Call the 2 heating outputs returned from the two calls to *CalcVAVVS* $\dot{Q}_{heat,max,fanon}$ and $\dot{Q}_{heat,min,fanon}$. If $\dot{Q}_{heat,max,fanon} < \dot{Q}_{tot}$ the terminal unit can not meet the load. Set the air flow rate to the heating maximum and the hot water flow rate or heating coil power to the maximum and call *CalcVAVVS* again. This concludes the simulation for this case. If $\dot{Q}_{heat,min,fanon} < \dot{Q}_{tot} < \dot{Q}_{heat,max,fanon}$ the heating demand is in the active heating, fan on region. For a hot water coil we call *SolveRegulaFalsi* with the supply air flow rate as the input that is varied and the hot water flow rate set to the maximum. For electric and gas coils the coil power and the supply air flow rate are both varied together from their minimum to maximum in a call to *SolveRegulaFalsi*. The call to *SolveRegulaFalsi* concludes the simulation for this case.
 - c) This region only applies to terminal units with a hot water coil. To establish the boundaries of region c) the fan is set to off, the supply air flow rate is set to minimum flow and *CalcVAVVS* is called twice: once with the hot water flow at maximum and once with the hot water flow at minimum. Call the two heating outputs $\dot{Q}_{heat,max,fanoff}$ and $\dot{Q}_{heat,min,fanoff}$. If $\dot{Q}_{tot}$ is between these values, the supply air flow rate is set to its minimum, the fan is set to off, and in the call to *SolveRegulaFalsi* the hot water flow rate is varied to meet the load. This concludes the simulation for this case.

If the cooling demand does not fall into cases a) – c), the unit is assumed to be in the passive cooling state: heating is off or at the minimum, the fan is off, and the minimum supply air flow is delivered to the zone.

Note that the terminal unit output is never explicitly passed to another routine. Instead the output is saved as the outlet conditions on the terminal unit outlet air node. The node data is accessed when the terminal unit output is needed elsewhere in the program (in *SimZoneAirLoopEquipment* for instance).

1.2.6 REFERENCES

No relevant references.

2 DUCT: ENERGYPLUS DOCUMENTATION

2.1 DUCT – INPUT/OUTPUT REFERENCE

Duct is a component for air systems that is a direct analogue for Pipe in the fluid loops. It is used when it is necessary (due to the HVAC system configuration) to have a branch that has no functional components. This case most often arises for a bypass branch. Since every branch must have at least one component, the Duct component is used for this situation. The duct is a pass-through component; all of its inlet conditions are passed through unchanged to its outlet.

Field: Duct Name

This is the unique name for this component. Any reference to this component (in a Branch List, for instance) will refer to it by this name.

Field: Inlet Node Name

The name of the component's air inlet node.

Field: Outlet Node Name

The name of the component's air outlet node.

Table 3: IDD definition of the Duct component.

```
DUCT,
  \memo Passes inlet node state variables to outlet node state variables
A1, \field DuctName
A2, \field Inlet Node Name
A3; \field Outlet Node Name
```

Table 4: Example input for Duct.

```
DUCT,
VAV Sys 1 Bypass Duct,           !- DuctName
VAV Sys 1 Bypass Duct Inlet Node, !- Inlet Node Name
VAV Sys 1 Bypass Duct Outlet Node; !- Outlet Node Name
```

There are no outputs for Duct.

3 SETPOINT MANAGER:RETURN AIR BYPASS FLOW: ENERGYPLUS DOCUMENTATION

3.1 SET POINT MANAGER:RETURN AIR BYPASS FLOW – INPUT/OUTPUT REFERENCE

This manager is user in conjunction with a return air bypass configuration. This type of air system is basically a standard single duct system with the addition of a by pass duct that bypasses return air around the main system components – in particular, the central cooling coil. This allows the central cooling coil to sufficiently dehumidify the mixed air; the bypassed air is then added to the supply air stream to bring the supply air temperature up to the desire temperature set point. This scheme is very useful in situations where an higher than normal supply air temperature is used – for instance, in underfloor air distribution systems.

This manager relies on the program to figure out the system configuration and to extract the needed data from the system nodes. All the user needs to input is the name of the air system and a schedule giving the desired supply air temperature set point. No node names are required. The manager will establish a bypass air flow rate that upon mixing with the non-bypassed air will yield the temperature given in the schedule.

Field: Name

A unique, user assigned name for an instance of an Return Air Bypass Flow set point manager. Anywhere in the input that this set point manager is used, it is referred to by this name.

Field: Control variable

The type of variable that will be controlled. The choices are: TEMP (temperature), HUMRAT (humidity ratio), or FLOW (mass flow rate). For this type of set point manager this input should be FLOW.

Field: AIR PRIMARY LOOP name

The name of the Air Primary Loop (the central air system) which will use this set point manager to set its return air bypass flow rate.

Field: Schedule Name

The name of a schedule whose values are temperatures in °C. The schedule value for the hour is the set point for this type of set point manager. The set point is assumed to be at the air handler outlet.

Table 5: IDD definition for the Set Point Manager:Return Air Bypass Flow object

```
SET POINT MANAGER:RETURN AIR BYPASS FLOW,  
  \min-fields 4  
  \memo This set point manager determines the required  
  \memo mass flow rate through a return air bypass duct  
  \memo to meet the specified temperature setpoint  
A1, \field Name  
  \required-field  
  \reference SetPointManagers  
A2, \field Control variable:  
  \type choice  
  \key FLOW  
A3, \field name of AIR PRIMARY LOOP for which manager will be employed  
  \required-field  
  \type object-list  
  \object-list AirPrimaryLoops  
A4 ; \field Schedule Name; scheduled temperature setpoint  
  \type object-list  
  \object-list ScheduleNames
```

Table 6: Example input for Set Point Manager with Return Air Bypass Flow.

```
SET POINT MANAGER:RETURN AIR BYPASS FLOW,
RETURN AIR BYPASS Manager 1,
FLOW,
VAV Sys 1,
Seasonal Reset Supply Air Temp Sch;
```

3.2 SET POINT MANAGER:RETURN AIR BYPASS FLOW - ENGINEERING REFERENCE

3.2.1 RETURN AIR BYPASS FLOW

This set point manager sets the air flow rate in a bypass duct such that when the bypassed and non-bypassed air are mixed the resultant air stream will be at the user-specified set point temperature.

The user specifies the desired setpoint temperature T_{set} through a input temperature schedule.

This temperature is modified to account for any potential fan heat gain:

$$T_{set,mod} = T_{set} - (T_{loop,out} - T_{mixer,out})$$

Here $T_{loop,out}$ is the temperature at the air loop outlet node and $T_{mixer,out}$ is the temperature at the outlet node of the bypass – nonbypass air mixer. Depending on the system configuration these may be the same node. Then

$$\dot{m}_{bypass} = (\dot{m}_{tot} T_{set,mod} - \dot{m}_{nonbypass} T_{nonbypass}) / T_{bypass}$$

where $\dot{m}_{tot}$ is the total supply air flowrate in kg/s sent to the zones, $\dot{m}_{nonbypass}$ is the nonbypassed air flowrate (the conditioned air), $T_{nonbypass}$ is the corresponding temperature just before mixing with the bypassed air, and T_{bypass} is the temperature of the bypassed (unconditioned) air. The resulting $\dot{m}_{bypass}$ is the mass flow rate setpoint for the bypass air branch.

4 UCSD UFAD INTERIOR MODEL CONTROLS: ENERGYPLUS DOCUMENTATION

4.1 UCSD UFAD INTERIOR MODEL CONTROLS – INPUT/OUTPUT REFERENCE

This model is applicable to interior spaces that are served by an underfloor air distribution system. The dominant sources of heat gain should be from people, equipment, and other localized sources located in the occupied part of the room. The model should be used with caution in zones which have large heat gains or losses through exterior walls or windows or which have considerable direct solar gain. The model predicts two temperatures in the room:

- An occupied subzone temperature (T_{OC}), representing the temperature in the region between the floor and the boundary of the upper subzone..
- An upper subzone temperature (T_{MX}) essential for overall energy budget calculations and for modeling comfort effects of the upper layer temperature.

The following fields are used to define an instance of the ‘UCSD UFAD Interior Model Controls’ object.

Field: Zone Name

This field provides the unique name of a zone described elsewhere in the file. A single instance of the 'UCSD UFAD Interior Model Controls' object is needed for each zone that is to be modeled using this method.

FIELD: GAIN DISTRIBUTION SCHEDULE

This field specifies the unique name of schedule defined elsewhere in the input file. The schedule values are the fractions of the convective portion of the internal gains in the occupied subzone that remain in the occupied subzone. The remainder of the convective portion of the internal gains in the occupied subzone enters the plumes and is carried to the upper subzone. The types of internal gains that are assumed to be located in the occupied subzone are:

- people
- task lights
- electric equipment
- gas equipment
- hot water equipment
- steam equipment
- other equipment
- baseboard heat

Types of internal gains that are assumed to be in the upper subzone are:

- general lights
- tubular daylighting devices
- high temperature radiant heaters

The schedule values should be between 0 and 1. A value of 1 means that all the convection gains from equipment, task lights and people are dispersed in the lower occupied subzone. Conversely a value of 0 puts all the lower subzone convective gains into the plumes rising into the upper well-mixed subzone.

Field: Number of plumes per occupant

This field specifies number of plumes per occupant. Plumes are associated with localized sources of convective heat gain from people and equipment. For example, a value of 2 would be used if each occupant has a computer that generates a separate plume that does not merge with the plume from the occupant in the lower, occupied, subzone.

Field: Number of diffusers per plume

The zone is served by diffusers mounted in floor panels located in the raised floor. This input specifies the average number of diffusers per plume. It can be estimated by dividing the total number of diffusers in the zone by the design occupancy level (number of people).

Field: Effective area of diffuser

This is the design air flow opening area in square meters of a single diffuser. The default value depends on the diffuser type. For swirl diffusers it is .011 m², for variable area diffusers .035 m², for DV diffusers .00548 m², and for linear bar grilles .062 m².

Field: Angle between diffuser slots and the vertical

This input describes the angle at which air emerges from the diffusers. It should be the angle in degrees between the vertical and the diffuser slots. The defaults depend on the diffuser type: for swirl diffusers it is 28 degrees, for variable area diffusers 45 degrees, for DV diffusers 72.9 degrees, and for linear bar grilles 15 degrees.

Field: Height of heat sources

The average height in meters of the occupied subzone heat sources (people, workstations) above the floor. But this should take into account the finite size of the heat sources: this input is the location of the point source that would produce the plume area at the actual heat source. A good input choice would be -.26 meters.

Field: Thermostat height

This field is the height in meters of the thermostat/temperature control sensor above the floor. The default is 1.1 meters.

Field: Comfort Height

The height in meters above the floor at which air temperature is calculated for comfort purposes. The air temperature at this height is used in calculating the available measures of comfort: Fanger, Pierce or KSU. The default is 1.1 meters.

Field: Temp. Difference Threshold for Reporting

This field specifies a minimum temperature difference between the upper subzone and the occupied subzone that will be used to trigger whether or not the UFAD auxiliary outputs will be calculated. These outputs are *UF Transition Height* and *UF Average Temp Gradient*. They are set to zero when the temperature difference is less than the threshold and the output *UF Zone Is Mixed* is set to 1.

The value should be greater than or equal to zero and is in units of degrees Centigrade. The default value is 0.4 degrees C.

Field: Diffuser type

The choices for this alpha field are *SWIRL* | *VARIABLE AREA* | *DISPLACEMENT*. The swirl and displacement diffusers are fixed area diffusers. The variable area diffusers change their effective area as a function of air flow rate and maintain an approximately constant exit velocity.

Field: Transition height

An optional field to allow the user to specify the transition height (meters above floor) rather than have the program calculate it.

Table 7: IDD definition of UCSD UFAD Interior Model Controls

```
UCSD UFAD INTERIOR MODEL CONTROLS,
\min-fields 12
A1 , \field Zone Name
    \note Name of Zone being described. Any existing zone name
    \required-field
    \type object-list
    \object-list ZoneNames
A2 , \field Gain Distribution Schedule
    \note fraction of the occupied subzone gains that remain in the subzone;
    \note i.e., don't go into plumes.
    \note 0<= Accepted Value <= 1.
    \note In this model 1 means all convective gains in the lower layer.
    \required-field
    \type object-list
    \object-list ScheduleNames
N1 , \field Number of plumes per occupant
    \note Effective number of separate plumes per occupant in the occupied zone.
    \note Plumes that merge together in the occupied zone count as one.
    \required-field
    \type real
    \minimum> 0.0
    \default 1.0
N2 , \field Number of diffusers per plume
    \type real
    \minimum> 0.0
    \autosizable
    \default autosize
N3 , \field Design effective area of diffuser
    \type real
    \units m2
    \minimum> 0.0
    \autosizable
    \default autosize
N4 , \field angle between diffuser slots and the vertical
    \type real
    \units deg
    \minimum 0.0
    \maximum 90.
    \autosizable
    \default autosize
N5 , \field Height of heat sources
    \note average height of the internal heat sources (people, workstations) above the floor
    \type real
    \units m
    \minimum -2.0
    \default .61
N6 , \field Thermostat height
    \note Height of thermostat/temperature control sensor above floor
    \units m
    \type real
    \minimum> 0.0
    \default 1.1
N7 , \field Comfort Height
    \note Height at which air temperature is calculated for comfort purposes
    \units m
    \type real
    \minimum> 0.0
    \default 1.1
N8 , \field Temp. Difference Threshold for Reporting
    \note Minimum temperature difference between predicted upper and lower layer
    \note temperatures above which DV auxilliary outputs are calculated.
    \note These outputs are 'DV Transition Height', 'DV Fraction Min Recommended Flow Rate'
    \note 'DV Average Temp Gradient' and 'DV Maximum Temp Gradient'. They
    \note are set to negative values when the temperature difference is less than the
    \note threshold and the output 'DV Zone Is Mixed' is set to 1
    \units deltaC
    \required-field
    \type real
    \minimum 0.0
```

```

\default 0.4
A3 , \field Diffuser type
\required-field
\type choice
\key SWIRL
\key VARIABLE AREA
\key DISPLACEMENT
\default SWIRL
N9; \field Transition height
\note user-specified height above floor of boundary between occupied and upper subzones
\type real
\units m
\minimum> 0.0
\autosizable
\default autosize

```

Table 8: Example input UCSD UFAD Interior Model Controls

```

ROOMAIR MODEL,
SPACE5-1 RoomAir Model,  !- Room-Air Model Name
SPACE5-1,                !- Zone Name
UCSD UFAD INTERIOR,      !- Room-Air Modeling Type
DIRECT;                  !- Air Temperature Coupling Strategy

UCSD UFAD INTERIOR MODEL CONTROLS,
SPACE5-1,                !- Zone Name
GainDistSched,           !- Gain Distribution Schedule
2.0,                     !- Number of plumes per occupant
0.5,                     !- Number of diffusers per plume
autosize,                 !- Design effective area of diffuser {m2}
autosize,                 !- angle between diffuser slots and the vertical {deg}
-.26,                    !- Height of heat sources {m}
1.4,                     !- Thermostat height {m}
1.0,                     !- Comfort Height {m}
0.001,                   !- Temp. Difference Threshold for Reporting {deltaC}
SWIRL;                   !- Diffuser type
1.3;                     !- Transition height {m}

```

4.2 UCSD UFAD INTERIOR ZONE MODEL – ENGINEERING REFERENCE

4.2.1 OVERVIEW

The UCSD UFAD Interior Zone Model provides a simple model for heat transfer and nonuniform vertical temperature profile for interior zones of a UFAD system. These zones are expected to be dominated by internal loads, a portion of which (such as occupants and workstations) will act to generate plumes. The plumes act to potentially create room air stratification, depending on the type & number of diffusers, the amount and type of load, and the system flowrate. In order to better model this situation the fully-mixed room air approximation that is currently used in most whole building analysis tools is extended to a two node approach, with the purpose of obtaining a first order precision model for vertical temperature profiles for the interior zones of UFAD systems. The use of 2 nodes allows for greatly improved prediction of thermal comfort and overall building energy performance for the increasingly popular UFAD systems.

The UCSD UFAD Interior Zone Model is one of the non-uniform zone models provided through the Room Air Manager in EnergyPlus. The intent is to provide a selection of useful non-uniform zone air models to enable the evaluation of air-conditioning techniques that use stratified or partially stratified room air. Such techniques include displacement ventilation (DV) and underfloor air distribution (UFAD) systems. The methodology can also include natural displacement ventilation and also wind-driven cross-ventilation (CV).

4.2.2 UNDERFLOOR AIR DISTRIBUTION SYSTEMS

UFAD systems represent, in terms of room air stratification, an intermediate condition between a well-mixed zone and displacement ventilation. Air is supplied through an underfloor plenum at low pressure through diffusers in the raised floor. The diffusers can be of various types: e.g., swirl, variable-area, displacement, and produce more or less mixing in the zone. UFAD systems are promoted as saving energy due to: higher supply air temperature; low static pressure; cooler conditions in the occupied subzone than in the upper subzone; and sweeping of some portion of the convective load (from ceiling lights, for instance) into the return air without interaction with the occupied region of the zone.

Modeling a UFAD system is quite complex and involves considerably more than just a non-uniform zone model. The zones' coupling to the supply and return plenums must be modeled accurately (particularly radiative transfer from a warm ceiling to a cool floor and on into the supply plenum by conduction). The supply plenum must be accurately modeled, giving a good estimate of the supply air temperature and conduction heat transfer between supply & return plenums through the slab. The HVAC system must be modeled including return air bypass and various types of fan powered terminal units.

The UCSD UFAD interior zone model is similar to the UCSD DV model. The most obvious difference is that the UFAD model has no separate near-floor subzone. Like the UCSD DV model it is a plume equation based multi-layer model (2 layers in this case). The zone is modeled as being divided into 2 well separated subzones which we denote as "occupied" and "upper". Each subzone is treated as having a single temperature. The boundary between the 2 subzones moves up & down each time step as a function of zone loads and supply air flow rate. Thus at each HVAC time step, the height of the boundary above the floor must be calculated, portions of surfaces assigned to each subzone, and a separate convective heat balance performed on each subzone.

4.2.3 MODEL DESCRIPTION

The UFAD interior zone model is based upon non-dimensional analysis of the system and using the non-dimensional description to make a comparison between full-scale UCB test chamber data & small-scale UCSD salt tank measurements.

In order to do the non-dimensional comparisons, we need to define two dimensionless parameters. One is Γ , and the other is ϕ . Lin & Linden [Lin & Linden, 2005] showed that in a UFAD system, the buoyancy flux of the heat source (B) and the momentum flux of the cooling jets (M) are the controlling parameters on the stratification. Since $[B] = L^4 T^{-3}$ and $[M] = L^4 T^{-2}$, we can have a length scale as $M^{3/4} / B^{1/2}$.

4.2.3.1 Definition of Γ for the single-plume single-diffuser basic model

We observed, in our small-scale experiments, that the total room height does not affect the interface position, or the height of the occupied zone. In other words, H might not be the critical length scale for the stratification. Therefore, we started to use $\sqrt{A}$ as the reference length. Then Γ is defined as

$$\Gamma = \frac{M^{3/4}}{B^{1/2} A^{1/2}} = \frac{(Q^2 / A)^{3/4}}{B^{1/2} A^{1/2}} = \frac{Q^{3/2}}{A^{5/4} B^{1/2}} \quad (0.1)$$

Definition for multi-diffuser and multi-source cases

We only considered single-diffuser, single-source cases in above analysis. Suppose there are n equal diffusers and m equal heat sources in a UFAD room. We shall divide the number of diffusers up into a number of separate heat sources so that each subsection with $n' = n/m$ diffusers per heat source will have the same stratification as other subsections. Further, the air flow and the heat load into the subsection Q' and B' will be $Q' = Q / m$ $B' = B / m$ respectively, where Q' and B' are the total air flow and the total

heat load for the entire UFAD space. Then the momentum flux each diffuser per heat source carries is $M_d = (\frac{1}{n'} Q')^2 / A$. (0.1) will be modified as

$$\Gamma = \frac{(n' M_d)^{3/4}}{B^{1/2} \sqrt{n' A}} = \frac{Q^{3/2}}{(n' A)^{5/4} B^{1/2}} = \frac{Q^{3/2}}{m(n' A)^{5/4} B^{1/2}} \quad (0.2)$$

Full-scale cases

Because B is the buoyancy flux of the heat sources and M is the momentum flux of the cooling jets, in a real full-scale room, we shall consider the total room net heat load (plume heat input, minus the room losses) and the total net flow rate coming from the diffusers (input room air flow, minus the room leakage). Further, if the diffuser is swirl type, the vertical momentum flux should be used.

$$\Gamma = \frac{(Q \cos \theta)^{3/2}}{m(n' A)^{5/4} (0.0281W)^{1/2}} \quad (0.3)$$

where, Q is the net flow rate coming out from all diffusers (m^3/s); W is the total net heat load (kW); A is the effective area of each diffuser (m^2); n' is the number of diffusers per heat source; θ is the angle between the diffuser slots and the vertical direction and m is the number of heat sources

4.2.3.2 Definition of Φ

In our theoretical model, two-layer stratification forms at steady state, provided that each diffuser carries the same momentum flux, and each heat source has the same heat load. We could define a dimensionless parameter Φ , which indicates the strength of stratification.

Small-scale cases

In our salt-water tank experiments, fluid density ρ is measured. Define that

$$\phi = \frac{\rho_u - \rho_l}{\rho_u - \rho_o} \quad (0.4)$$

where, ρ_u and ρ_l are the fluid density of the upper layer and lower layer, separately; and ρ_o is the source density at the diffusers.

Therefore, $\rho_l = \rho_o$ gives $\phi = 1$, which means the largest stratification (displacement ventilation case); $\rho_l = \rho_u$ leads to $\phi = 0$, in which case there is no stratification (mixed ventilation case).

4.2.3.3 Full-scale cases

Similarly, we can define ϕ for full-scale cases by using temperature.

$$\phi = \frac{T_r - T_{oz}}{T_r - T_s} \quad (0.5)$$

where T_r , T_{oz} , and T_s are the return air temperature, the occupied zone temperature and the supply temperature, respectively (K). Again $\phi = 1$ occurs in displacement ventilation; while $\phi = 0$ happens in mixed ventilation.

4.2.3.4 Comparisons between full-scale UCB data and small-scale UCSD data

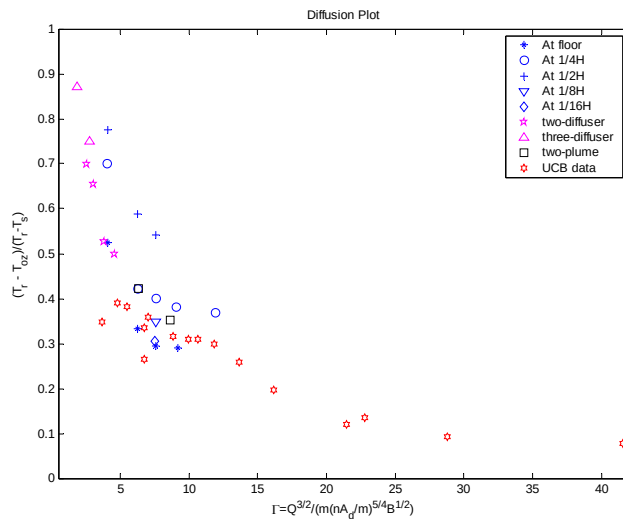
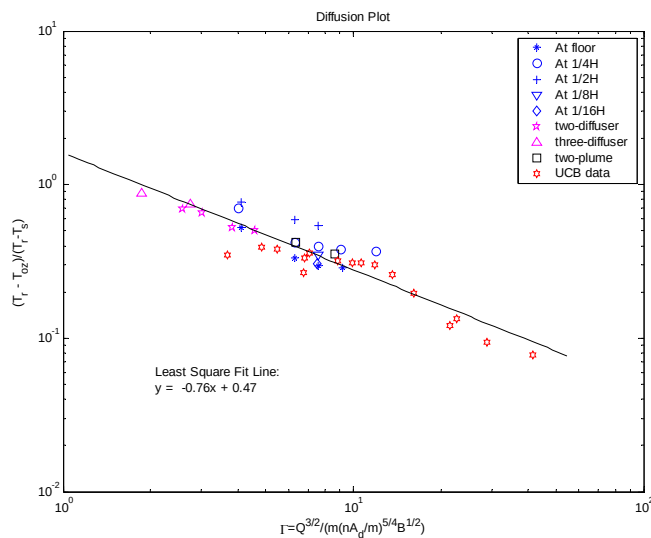


Figure 1: Data comparisons in the non-dimensional regular Γ - ϕ plot



(b)

Figure 2: Data comparisons in the non-dimensional log-log Γ - ϕ plot

Figure 1 and Figure 2 show the comparisons between UCB's data and the UCSD salt tank data in the Γ - ϕ plot. As seen in the figure, the full-scale and small-scale data are on the same trend curve. This provides the evidence that the salt-tank experiments have included most characteristics of a UFAD system. Note that big Γ (>20) of UCB's experiments all have large DDR (from 1.19 to 2.18). The largest DDR (2.18) even gives a negative $(T_r - T_{oz})/(T_r - T_s)$, which is NOT shown in the figure.)

We could work out the occupied zone temperature by using the least-square fitting line suggested in Figure 2. Hence the interface height is needed to determine a entire two-layer stratification. Figure 3 shows the dimensionless interface height $(h/\sqrt{n'A})$ of the UCSD small-scale experiments plotted against

Γ . Note that for the experiments with elevated heat source, the interface heights have been modified by $h' = h - \frac{1}{2}h_s$ where h_s is the vertical position of the elevated heat source. All data then are located along a line in Figure 3. Since the salt-tank experiments are concluded to represent important characteristics of a full-scale UFAD room, Figure 3 provides some guidelines for estimate the interface position in a real UFAD room.

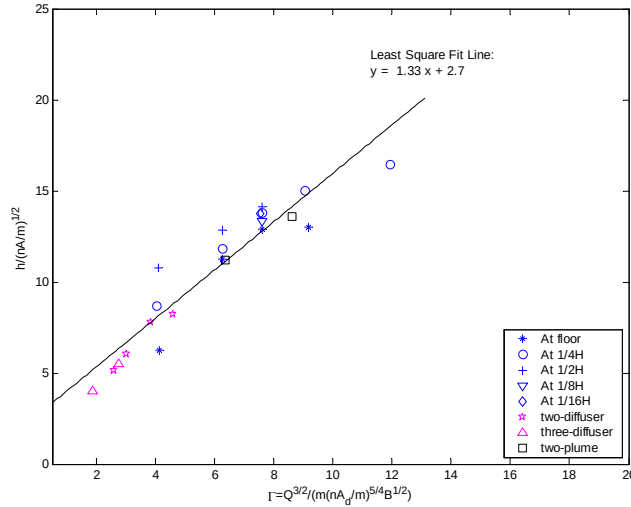


Figure 3: Non-dimensional interface height of small-scale experiments.

Formulas for EnergyPlus based on the dimensionless parameter Γ

If we have input including the supply temperature T_s (K); the number of diffusers n ; the number of heat sources m ; the vertical position of the heat sources h_s (m); the heat load W (kW); the effective area of a diffuser A (m^2); and the total supply air flow rate Q (m^3/s) then the output will be

$$T_r = \frac{0.0281W}{Qg} T_s + T_s \quad (0.6)$$

$$T_{oz} = T_r - 1.6\Gamma^{-0.76}(T_r - T_s) \quad (0.7)$$

$$h = \sqrt{\frac{n}{m}} A(7.43\ln(\Gamma) - 1.35) + \frac{1}{2}h_s \quad (0.8)$$

where T_r is the return temperature (K); T_{oz} is the occupied subzone temperature (K); h is the interface height (m); and Γ is defined above.

Implementation

The implementation closely follows the procedure described in the displacement ventilation zone model. The model predicts two temperatures that characterize the two main levels in the stratification of the room:

1. an occupied subzone temperature T_{oc} representing the temperature of the occupied region;

2. an upper level temperature T_{mx} representing the temperature of the upper, mixed region and the outflow temperature.

We will use (0.8) to calculate the interface height and do a heat balance calculation on each subzone. Γ is given by (0.3). The supply air flow rate $\dot{V}$ is obtained by summing all the air flows entering the zone: supply air, infiltration, ventilation, and inter-zone flow. The heat gain $\dot{Q}$ is estimated by summing all the convective internal gains located in the occupied subzone – task lights, people, equipment – and dividing this power equally among the n plumes. With these assumptions we can describe the implementation.

The UCSD UFI model is controlled by the subroutine *ManageUCSDUFModels* which is called from the *RoomAirModelManager*. The *RoomAirModelManager* selects which zone model will be used for each zone.

The calculation is done in subroutine *CalcUCSDUI*. First we calculate the convective heat gain going into the upper and lower regions.

$$\begin{aligned}\dot{Q}_{ocz} &= \dot{Q}_{oc,conv} + \dot{Q}_{tl,conv} + \dot{Q}_{eleq,conv} + \dot{Q}_{gaseq,conv} + \dot{Q}_{otheq,conv} + \dot{Q}_{hw,conv} \\ &\quad + \dot{Q}_{stmeq,conv} + \dot{Q}_{bb,conv} + \dot{Q}_{waterheater,conv} + \dot{Q}_{fuelcell,conv} \\ \dot{Q}_{mxz} &= \dot{Q}_{gl,conv} + \dot{Q}_{ltp} + \dot{Q}_{htrd,conv} \\ \dot{Q}_{tot,conv} &= \dot{Q}_{ocz} + \dot{Q}_{mxz}\end{aligned}$$

Next we sum up the inlet air flows in the form of MCP (mass flow rate times the air specific heat capacity) and MCPT (mass flow rate times C_p times air temperature).

$$\begin{aligned}MCP_{zone} &= MCP_i + MCP_{vent} + MCP_{mix} \\ MCP_{sys} &= \sum_{inlets} \dot{m}_i C_{p,i} \\ \dot{m}_{systot} &= \sum_{inlets} \dot{m}_i \\ MCP_{tot} &= MCP_{zone} + MCP_{sys} \\ MCPT_{zone} &= MCPT_i + MCPT_{vent} + MCPT_{mix} \\ MCPT_{sys} &= \sum_{inlets} \dot{m}_i C_{p,i} T_i \\ MCPT_{tot} &= MCPT_{zone} + MCPT_{sys}\end{aligned}$$

The number of plumes per occupant $N_{plumesperspers}$ is a user input. The total number of plumes in the zone is:

$$N_{plumes} = N_{occ} \cdot N_{plumespersperson}$$

Using this we calculate the total power in the plumes and the power per plume.

$$\dot{Q}_{plumes} = \dot{Q}_{ocz}$$

$$\dot{Q}_{perplume} = \dot{Q}_{plumes} / N_{plumes}$$

The number of diffusers per plumes is also a user input. To obtain the number of diffusers in the zone:

$$N_{diffusers} = N_{plumes} \cdot N_{diffusersperplume}$$

The area A_{diff} is also a user input. For swirl diffusers and for displacement diffusers this area is used as input. For the variable area diffusers, though, we calculate the area. We assume 400 ft/min velocity at the diffuser and a design flow rate per diffuser is 150 cfm (.0708 m³/s). The design area of the diffuser is 150 ft³/min / 400 ft/min = .575 ft² = .035 m². Then the variable area each time step is

$$A_{diff} = \dot{m}_{systot} / (.0708 \cdot N_{diffusers})$$

We now calculate the height fraction Fr_{hb} (height of boundary layer divided by the total zone height).

$$\Gamma = \frac{(\dot{m}_{systot} \cdot \cos(\theta_{throw}))^{3/2}}{N_{plumes} (N_{diffusersperplume} \cdot A_{diff})^{5/4} (0.0281 \cdot 0.001 \cdot \dot{Q}_{plumes})^{1/2}}$$

$$Fr_{hb} = \frac{(N_{diffusersperplume} \cdot A_{diff})^{1/2} (7.43 \ln(\Gamma) - 1.35 + 0.5H_s)}{H_{ceil}}$$

where θ_{throw} is a user input: the angle between the diffuser slots and vertical; and H_s is the source height above the floor (m).

Next we iterate over the following 2 steps.

4.2.3.5 Iterative procedure

1. Call subroutine *HcUCSDUF* to calculate a convective heat transfer coefficient for each surface in the zone, an effective air temperature for each surface, and HA_{mx} , HAT_{mx} , HA_{oc} , HAT_{oc} . Here HA is $\sum_{surfaces} h_{c,i} \cdot A_i$ for a region and HAT is $\sum_{surfaces} h_{c,i} \cdot A_i \cdot T_i$ for a region. The sum is over all the surfaces bounding the region; $h_{c,i}$ is the convective heat transfer coefficient for surface i , A_i is the area of surface i , and T_i is the surface temperature of surface i .
2. Calculate the two subzone temperatures: T_{oc} and T_{mx} .

The h_c 's calculated in step 1 depend on the subzone temperatures. In turn the subzone temperatures depend on the HA and HAT 's calculated in step 1. Hence the need for iteration

Next we describe each steps 1 and 2 in more detail.

Step 1

Subroutine *HcUCSDUF* is quite straightforward. It loops through all the surfaces in each zone and decides whether the surface is located in the upper, mixed subzone or the lower, occupied subzone, or if the surface is in both subzones. If entirely in one subzone the subzone temperature is stored in the surface effective temperature variable *TempEffBulkAir(SurfNum)* and h_c for the surface is calculated by a call to subroutine *CalcDetailedHcInForDVModel*. This routine uses the "detailed" natural convection coefficient

calculation that depends on surface tilt and $\Delta T^{1/3}$. This calculation is appropriate for situations with low air velocity.

For surfaces that bound 2 subzones, the subroutine calculates h_c for each subzone and then averages them, weighting by the amount of surface in each subzone.

During the surface loop, once the h_c for a surface is calculated, the appropriate subzone HA and HAT sums are incremented. If a surface is in 2 subzones the HA and HAT for each subzone are incremented based on the area of the surface in each subzone.

Step 2

The calculation of subzone temperatures follows the method used in the **ZoneTempPredictorCorrector** module and described in the section **Basis for the System and Zone Integration**. Namely a third order finite difference expansion of the temperature time derivative is used in updating the subzone temperatures. Otherwise the subzone temperatures are obtained straightforwardly by solving an energy balance equation for each subzone.

$$T_{oc} = (C_{air,oc} \cdot (3 \cdot T_{-1,oc} - (3/2) \cdot T_{-2,oc} + (1/3) \cdot T_{-3,oc}) + \dot{Q}_{ocz} \cdot Fr_{gains} + HAT_{oc} + MCP_{tot}) / ((11/6) \cdot C_{air,oc} + HA_{oc} + MCP_{tot})$$

$$T_{mx} = (C_{air,mx} \cdot (3 \cdot T_{-1,mx} - (3/2) \cdot T_{-2,mx} + (1/3) \cdot T_{-3,mx}) + \dot{Q}_{ocz} \cdot (1 - Fr_{gains}) + \dot{Q}_{mxz} + HAT_{mx} + T_{oc} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,mx} + HA_{mx} + MCP_{tot})$$

Here $C_{air,oc}$ and $C_{air,mx}$ are the heat capacities of the air volume in each subzone. $C_{air,mx}$ is calculated by

$$R_{air,mx} = V_{mx} \cdot (\Delta z_{mx} / z_{ceil}) \cdot \rho_{air,mx} \cdot c_{p,air,mx} \cdot Mul_{cap} / (\Delta t_z \cdot 3600)$$

$$C_{air,mx} = R_{air,mx} \cdot \Delta t_z / \Delta t_{sys}$$

The gains fraction Fr_{gains} is a user input via a schedule. It is the fraction of the convective gains in the occupied subzone that remain in that subzone.

The other subzone air heat capacities are calculated in the same manner.

Mixed calculation

The above iterative procedure assumed that the UFAD nonuniform zone model was appropriate: i.e., conditions were favorable temperature stratification in the zone. Now that this calculation is complete and the subzone temperatures and depths calculated, we check to see if this assumption was justified. If not, zone conditions must be recalculated assuming a well-mixed zone.

If $T_{mx} < T_{oc}$ or $MCP_{tot} \leq 0$ or $H_{fr} \cdot H_{ceil} < \Delta z_{occ,min}$ then the following mixed calculation will replace the UFAD interior zone calculation.

Note: $\Delta z_{occ,min}$ is the minimum thickness of occupied subzone. It is set to 0.2 meters.

The mixed calculation iteratively calculates surface convection coefficients and room temperature just like the displacement ventilation calculation described above. In the mixed case however, only one zone temperature T_{avg} is calculated. The 3 subzone temperatures are then set equal to T_{avg} .

First, Fr_{hb} is set equal to zero.

Then the code iterates over these steps.

Calculate T_{avg} using

$$T_{avg} = (C_{air,z} \cdot (3 \cdot T_{-1,z} - (3/2) \cdot T_{-2,z} + (1/3) \cdot T_{-3,z}) + \dot{Q}_{tot,conv} + HAT_{oc} + HAT_{mx} + MCPT_{tot}) / ((11/6) \cdot C_{air,z} + HA_{oc} + HA_{mx} + MCP_{tot})$$

$$T_{mx} = T_{avg}$$

$$T_{oc} = T_{avg}$$

Call $HcUCSDUF$ to calculate the h_c 's.

Repeat step 1

Final calculations

The UFAD interior zone calculation finishes by calculating some report variables.

We define heights:

$$H_{trans} = Fr_{hb} \cdot H_{ceil}$$

$$H_{mxavg} = (H_{ceil} + H_{trans}) / 2$$

$$H_{ocavg} = H_{trans} / 2$$

Using the user defined comfort height we calculate the comfort temperature.

If mixing:

$$T_{comf} = T_{avg}$$

If UFAD:

If $H_{comf} < H_{ocavg}$

$$T_{comf} = T_{occ}$$

Else if $H_{comf} \geq H_{ocavg}$ and $H_{comf} < H_{mxavg}$

$$T_{comf} = (T_{oc}(H_{mxavg} - H_{comf}) + T_{mx}(H_{comf} - H_{ocavg}) / (H_{mxavg} - H_{ocavg}))$$

Else if $H_{comf} \geq H_{mxavg}$ and $H_{comf} < H_{ceil}$

$$T_{comf} = T_{mx}$$

Using the user defined thermostat height we calculate the temperature at the thermostat.

If mixing:

$$T_{stat} = T_{avg}$$

If UFAD:

If $H_{stat} < H_{ocavg}$

$$T_{stat} = T_{occ}$$

Else if $H_{stat} \geq H_{ocavg}$ and $H_{stat} < H_{mxavg}$

$$T_{stat} = (T_{oc}(H_{mxavg} - H_{stat}) + T_{mx}(H_{stat} - H_{ocavg}) / (H_{mxavg} - H_{ocavg}))$$

Else if $H_{stat} \geq H_{mxavg}$ and $H_{stat} < H_{ceil}$

$$T_{stat} = T_{mx}$$

The average temperature gradient is:

If $H_{mxavg} - H_{ocavg} > 0.1$

$$GradT_{avg} = (T_{mx} - T_{occ}) / (H_{mxavg} - H_{ocavg})$$

else $GradT_{avg} = 0.0$

Finally, the zone node temperature is set to T_{mx} .

Other variables that are reported out are Γ and ϕ .

$$\phi = (T_{mx} - T_{occ}) / (T_{mx} - T_{sup})$$

where T_{sup} is the zone supply air temperature.

4.2.4 REFERENCES

Lin, Y.J. and P.F. Linden. 2005. A model for an under floor air distribution system. Energy&Building. Vol. 37, pp. 399-409.

5 UCSD UFAD EXTERIOR MODEL CONTROLS: ENERGYPLUS DOCUMENTATION

5.1 UCSD UFAD EXTERIOR MODEL CONTROLS – INPUT/OUTPUT REFERENCE

This model is applicable to exterior spaces that are served by an underfloor air distribution system. The dominant sources of heat gain should be from people, equipment, and other localized sources located in the occupied part of the room, as well as convective gain coming from a warm window. The model predicts two temperatures in the room:

- An occupied subzone temperature (T_{OC}), representing the temperature in the region between the floor and the boundary of the upper subzone..
- An upper subzone temperature (T_{MX}) essential for overall energy budget calculations and for modeling comfort effects of the upper layer temperature.

The following fields are used to define an instance of the ‘UCSD UFAD Exterior Model Controls’ object.

Field: Zone Name

This field provides the unique name of a zone described elsewhere in the file. A single instance of the ‘UCSD UFAD Interior Model Controls’ object is needed for each zone that is to be modeled using this method.

FIELD: GAIN DISTRIBUTION SCHEDULE

This field specifies the unique name of schedule defined elsewhere in the input file. The schedule values are the fractions of the convective portion of the internal gains in the occupied subzone that remain in the occupied subzone. The remainder of the convective portion of the internal gains in the occupied subzone enters the plumes and is carried to the upper subzone. The types of internal gains that are assumed to be located in the occupied subzone are:

- people
- task lights
- electric equipment
- gas equipment
- hot water equipment
- steam equipment
- other equipment
- baseboard heat

Types of internal gains that are assumed to be in the upper subzone are:

- general lights
- tubular daylighting devices
- high temperature radiant heaters

The schedule values should be between 0 and 1. A value of 1 means that all the convection gains from equipment, task lights and people are dispersed in the lower occupied subzone. Conversely a value of 0 puts all the lower subzone convective gains into the plumes rising into the upper well-mixed subzone.

Field: Number of diffusers

The zone is served by diffusers mounted in floor panels located in the raised floor. This input specifies the total number of diffusers in the zone. For linear bar grille diffusers, the length of an individual diffuser is assumed to be 1.219 meters (4 feet).

Field: Design effective area of 1 diffuser

This is the design air flow opening area in square meters of a single diffuser. The default value depends on the diffuser type. For swirl diffusers it is .011 m², for variable area diffusers .035 m², for DV diffusers .00548 m², and for linear bar grilles .062 m².

Field: Angle between diffuser slots and the vertical

This input describes the angle at which air emerges from the diffusers. It should be the angle in degrees between the vertical and the diffuser slots. The defaults depend on the diffuser type: for swirl diffusers it is 28 degrees, for variable area diffusers 45 degrees, for DV diffusers 72.9 degrees, and for linear bar grilles 15 degrees.

Field: Position of plume vertex relative to floor

The average height in meters of the occupied subzone heat sources (people, workstations, window plume) above the floor. But this should take into account the finite size of the heat sources: this input is the location of the point source that would produce the plume area at the actual heat source. A good input choice would be -.5 meters.

Field: Thermostat height

This field is the height in meters of the thermostat/temperature control sensor above the floor. The default is 1.1 meters.

Field: Comfort Height

The height in meters above the floor at which air temperature is calculated for comfort purposes. The air temperature at this height is used in calculating the available measures of comfort: Fanger, Pierce or KSU. The default is 1.1 meters.

Field: Temp. Difference Threshold for Reporting

This field specifies a minimum temperature difference between the upper subzone and the occupied subzone that will be used to trigger whether or not the UFAD auxiliary outputs will be calculated. These outputs are *UF Transition Height* and *UF Average Temp Gradient*. They are set to zero when the temperature difference is less than the threshold and the output *UF Zone Is Mixed* is set to 1.

The value should be greater than or equal to zero and is in units of degrees Centigrade. The default value is 0.4 degrees C.

Field: Diffuser type

The choices for this alpha field are *SWIRL* | *VARIABLE AREA* | *DISPLACEMENT*. The swirl and displacement diffusers are fixed area diffusers. The variable area diffusers change their effective area as a function of air flow rate and maintain an approximately constant exit velocity.

Field: Transition height

An optional field to allow the user to specify the transition height (meters above floor) rather than have the program calculate it.

Table 9: IDD definition of UCSD UFAD Exterior Model Controls

UCSD UFAD EXTERIOR MODEL CONTROLS,	
\min-fields 11	
A1 ,	\field Zone Name \note Name of Zone being described. Any existing zone name \required-field \type object-list \object-list ZoneNames
A2 ,	\field Gain Distribution Schedule \note fraction of the occupied subzone gains that remain in the subzone; \note i.e., don't go into plumes. \note 0<= Accepted Value <= 1. \note In this model 1 means all convective gains in the lower layer. \required-field \type object-list \object-list ScheduleNames
N1 ,	\field Number of diffusers \note assume one plume \required-field \type real \minimum> 0.0
N2 ,	\field Design effective area of 1 diffuser \type real \units m2 \minimum> 0.0 \autosizable \default autosize
N3 ,	\field angle between diffuser slots and the vertical \type real \units deg \minimum 0.0 \maximum 90. \autosizable \default autosize
N4 ,	\field Position of plume vertex relative to floor \type real \units m \minimum -2.0 \default -0.5
N5 ,	\field Thermostat height \note Height of thermostat/temperature control sensor above floor \units m \type real \minimum> 0.0 \default 1.1
N6 ,	\field Comfort Height \note Height at which air temperature is calculated for comfort purposes \units m \type real \minimum> 0.0 \default 1.1
N7 ,	\field Temp. Difference Threshold for Reporting \note Minimum temperature difference between predicted upper and lower layer \note temperatures above which DV auxilliary outputs are calculated. \note These outputs are 'DV Transition Height', 'DV Fraction Min Recommended Flow Rate' \note 'DV Average Temp Gradient' and 'DV Maximum Temp Gradient'. They \note are set to negative values when the temperature difference is less than the \note threshold and the output 'DV Zone Is Mixed' is set to 1 \units deltaC \required-field \type real \minimum 0.0 \default 0.4
A3 ,	\field Diffuser type \required-field \type choice \key SWIRL \key VARIABLE AREA \key DISPLACEMENT \key LINEAR BAR GRILLE

```

\default LINEAR BAR GRILLE
N8 ; \field Transition height
\note user-specified height above floor of boundary between occupied and upper subzones
\type real
\units m
\minimum> 0.0
\autosizable
\default autosize

```

Table 10: Example input for UCSD UFAD Exterior Model Controls

```

UCSD UFAD EXTERIOR MODEL CONTROLS,
SPACE2-1,           !- Zone Name
GainDistSched,      !- Gain Distribution Schedule
3,                  !- Number of diffusers
autosize,            !- Design effective area of 1 diffuser {m2}
autosize,            !- angle between diffuser slots and the vertical {deg}
-0.5,               !- Position of plume vertex relative to floor {m}
1.4,                !- Thermostat height {m}
1.0,                !- Comfort Height {m}
0.001,              !- Temp. Difference Threshold for Reporting {deltaC}
LINEAR BAR GRILLE,  !- Diffuser type
1.3;                !- Transition height

```

5.2 UCSD UFAD EXTERIOR ZONE MODEL – ENGINEERING REFERENCE

5.2.1 OVERVIEW

The UCSD UFAD Exterior Zone Model provides a simple model for heat transfer and a nonuniform vertical temperature profile for exterior zones of a UFAD system. These zones are expected to be dominated by internal loads, a portion of which (such as occupants and workstations) will act to generate plumes, and by window solar and conduction heat gains. The solar radiation penetrating the room is not expected to generate plumes. However, a window plume is likely to be generated in sunny conditions, particularly if an interior blind is deployed. Thus the exterior UFAD zone will have potentially have plumes from people and equipment and plumes arising from the windows. The plumes act to potentially create room air stratification, depending on the type & number of diffusers, the amount and type of load, and the system flowrate. In order to better model this situation the fully-mixed room air approximation that is currently used in most whole building analysis tools is extended to a two node approach, with the purpose of obtaining a first order precision model for vertical temperature profiles for the exterior zones of UFAD systems. The use of 2 nodes allows for greatly improved prediction of thermal comfort and overall building energy performance for the increasingly popular UFAD systems.

The UCSD UFAD Exterior Zone Model is one of the non-uniform zone models provided through the Room Air Manager in EnergyPlus. The intent is to provide a selection of useful non-uniform zone air models to enable the evaluation of air-conditioning techniques that use stratified or partially stratified room air. Such techniques include displacement ventilation (DV) and underfloor air distribution (UFAD) systems. The methodology can also include natural displacement ventilation and also wind-driven cross-ventilation (CV).

5.2.2 UNDERFLOOR AIR DISTRIBUTION SYSTEMS

UFAD systems represent, in terms of room air stratification, an intermediate condition between a well-mixed zone and displacement ventilation. Air is supplied through an underfloor plenum at low pressure through diffusers in the raised floor. The diffusers can be of various types: e.g., swirl, variable-area, displacement, and produce more or less mixing in the zone. UFAD systems are promoted as saving energy due to: higher supply air temperature; low static pressure; cooler conditions in the occupied subzone than in the upper subzone; and sweeping of some portion of the convective load (from ceiling lights, for instance) into the return air without interaction with the occupied region of the zone.

Modeling a UFAD system is quite complex and involves considerably more than just a non-uniform zone model. The zones' coupling to the supply and return plenums must be modeled accurately (particularly radiative transfer from a warm ceiling to a cool floor and on into the supply plenum by conduction). The supply plenum must be accurately modeled, giving a good estimate of the supply air temperature and conduction heat transfer between supply & return plenums through the slab. The HVAC system must be modeled including return air bypass and various types of fan powered terminal units.

The UCSD UFAD exterior zone model is similar to the UCSD interior zone model. The most obvious difference is that the exterior UFAD has 2 different types of plume sources: people & equipment and windows. Like the UCSD UFAD interior model it is a plume equation based multi-layer model (2 layers in this case). The zone is modeled as being divided into 2 well separated subzones which we denote as "occupied" and "upper". Each subzone is treated as having a single temperature. The boundary between the 2 subzones moves up & down each time step as a function of zone loads and supply air flow rate. Thus at each HVAC time step, the height of the boundary above the floor must be calculated, portions of surfaces assigned to each subzone, and a separate convective heat balance performed on each subzone.

5.2.3 MODEL DESCRIPTION

As in the interior zone case, we define 2 dimensionless parameters: Γ and ϕ . The definitions of the 2 parameters are the same as in the previous section (equations (0.1), (0.2), (0.3), (0.4), and (0.5)). As in the previous case, the experimental data can be plotted versus Γ and lines fitted to the data give the following formulas for occupied subzone temperature and interface height.

$$T_{oz} = T_r - 1.4\Gamma^{-0.6}(T_r - T_s) \quad (0.9)$$

$$h = \sqrt{\frac{n}{m}} A(11.03 \ln(\Gamma) - 10.73) + \frac{1}{2} h_s \quad (0.10)$$

where T_{oz} is the occupied subzone temperature (K); T_r is the return temperature (K); Γ is the dimensionless height parameter defined above; T_s is the supply temperature (K); h is the interface height (m); n is the number of diffusers; m is the number of heat sources; A is the effective area of a diffuser (m^2); and h_s is the vertical position of the heat sources (m). The formula for T_r is the same as in the previous section.

5.2.4 IMPLEMENTATION

The implementation closely follows the procedure described in the UFAD interior zone model. The model predicts two temperatures that characterize the two main levels in the stratification of the room:

1. an occupied subzone temperature T_{oc} representing the temperature of the occupied region;
2. an upper level temperature T_{mx} representing the temperature of the upper, mixed region and the outflow temperature.

We will use (0.10) to calculate the interface height and do a heat balance calculation on each subzone. Γ is given by (0.3). The supply air flow rate $\dot{V}$ is obtained by summing all the air flows entering the zone: supply air, infiltration, ventilation, and inter-zone flow. The heat gain $\dot{Q}$ is estimated by summing all the convective internal gains located in the occupied subzone – task lights, people, equipment – and adding to this the convective gain coming from the window surface. With these assumptions we can describe the implementation.

The UCSD UFE model is controlled by the subroutine *ManageUCSDUFModels* which is called from the *RoomAirModelManager*. The *RoomAirModelManager* selects which zone model will be used for each zone.

The calculation is done in subroutine *CalcUCSDUEI*. First we calculate the convective heat gain going into the upper and lower regions.

$$\begin{aligned}\dot{Q}_{ocz} &= \dot{Q}_{oc,conv} + \dot{Q}_{tl,conv} + \dot{Q}_{eleq,conv} + \dot{Q}_{gaseq,conv} + \dot{Q}_{otheq,conv} + \dot{Q}_{hw,conv} \\ &\quad + \dot{Q}_{stmeq,conv} + \dot{Q}_{bb,conv} + \dot{Q}_{waterheater,conv} + \dot{Q}_{fuelcell,conv} \\ \dot{Q}_{mxz} &= \dot{Q}_{gl,conv} + \dot{Q}_{ltp} + \dot{Q}_{htrd,conv} \\ \dot{Q}_{tot,conv} &= \dot{Q}_{ocz} + \dot{Q}_{mxz}\end{aligned}$$

Next we sum up the inlet air flows in the form of MCP (mass flow rate times the air specific heat capacity) and MCPT (mass flow rate times C_p times air temperature).

$$\begin{aligned}MCP_{zone} &= MCP_i + MCP_{vent} + MCP_{mix} \\ MCP_{sys} &= \sum_{inlets} \dot{m}_i C_{p,i} \\ \dot{m}_{systot} &= \sum_{inlets} \dot{m}_i \\ MCP_{tot} &= MCP_{zone} + MCP_{sys} \\ MCPT_{zone} &= MCPT_i + MCPT_{vent} + MCPT_{mix} \\ MCPT_{sys} &= \sum_{inlets} \dot{m}_i C_{p,i} T_i \\ MCPT_{tot} &= MCPT_{zone} + MCPT_{sys}\end{aligned}$$

For exterior zone model, we assume one plume: $N_{plumes} = 1$. The number of diffusers in the zone

$N_{diffusers}$ is a user input.

The area A_{diff} is also a user input. For swirl diffusers, linear bar grilles, and displacement diffusers this area is used as input. For the variable area diffusers, though, we calculate the area. We assume 400 ft/min velocity at the diffuser and a design flow rate per diffuser is 150 cfm (.0708 m³/s). The design area of the diffuser is 150 ft³/min / 400 ft/min = .575 ft² = .035 m². Then the variable area each time step is

$$A_{diff} = \dot{m}_{systot} / (.0708 N_{diffusers})$$

We now make an initial estimate of the convective gain from the windows.

$$\dot{Q}_{win,conv} = \sum_{i=1, n_{win}} h_{c,i} A_{win,i} T_{winsurf,i} - h_{c,i} A_{win,i} T_{mat}$$

Then

$$\dot{Q}_{plumes} = \dot{Q}_{ocz} + \dot{Q}_{win,conv}$$

We now calculate the height fraction Fr_{hb} (height of boundary layer divided by the total zone height).

$$\Gamma = \frac{(\dot{m}_{systot} \cos(\theta_{throw}))^{3/2}}{(N_{diffusers} A_{diff})^{5/4} (0.0281 \dot{Q}_{plumes})^{1/2}}$$

$$Fr_{hb} = \frac{(N_{diffusers} A_{diff})^{1/2} (11.03 \ln(\Gamma) - 10.73 + 0.5 H_s)}{H_{ceil}}$$

where θ_{throw} is a user input: the angle between the diffuser slots and vertical; and H_s is the source height above the floor (m).

Next we iterate over the following 2 steps.

5.2.4.1 Iterative procedure

1. Call subroutine *HcUCSDUF* to calculate a convective heat transfer coefficient for each surface in the zone, an effective air temperature for each surface, and HA_{mx} , HAT_{mx} , HA_{oc} , HAT_{oc} ,

$HA_{mx,win}$, $HAT_{mx,win}$, $HA_{oc,win}$, and $HAT_{oc,win}$. Here HA is $\sum_{surfaces} h_{c,i} \cdot A_i$ for a region and HAT is

$\sum_{surfaces} h_{c,i} \cdot A_i \cdot T_i$ for a region. The sum is over all the surfaces bounding the region; $h_{c,i}$ is the

convective heat transfer coefficient for surface i , A_i is the area of surface i , and T_i is the surface temperature of surface i . Variables with the *win* subscript are summed over window surfaces only. Then the convective gain from the window is recalculated:

$$\dot{Q}_{win,conv} = HAT_{mx,win} + HAT_{oc,win} - HA_{mx,win} T_{mx} - HA_{oc,win} T_{oc}$$

and the power in the plume is recalculated: $\dot{Q}_{plumes} = \dot{Q}_{ocz} + \dot{Q}_{win,conv}$.

2. Calculate the two subzone temperatures: T_{oc} and T_{mx} .

The h_c 's calculated in step 1 depend on the subzone temperatures. In turn the subzone temperatures depend on the HA and HAT 's calculated in step 1. Hence the need for iteration

Next we describe each steps 1 and 3 in more detail.

Step 1

Subroutine *HcUCSDUF* is quite straightforward. It loops through all the surfaces in each zone and decides whether the surface is located in the upper, mixed subzone or the lower, occupied subzone, or if the surface is in both subzones. If entirely in one subzone the subzone temperature is stored in the surface effective temperature variable *TempEffBulkAir(SurfNum)* and h_c for the surface is calculated by a call to subroutine *CalcDetailedHcInForDVModel*. This routine uses the “detailed” natural convection coefficient calculation that depends on surface tilt and $\Delta T^{1/3}$. This calculation is appropriate for situations with low air velocity.

For surfaces that bound 2 subzones, the subroutine calculates h_c for each subzone and then averages them, weighting by the amount of surface in each subzone.

During the surface loop, once the h_c for a surface is calculated, the appropriate subzone HA and HAT sums are incremented. If a surface is in 2 subzones the HA and HAT for each subzone are incremented based on the area of the surface in each subzone. The subroutine calculates a separate HA and HAT for the windows for use in calculating the window convective gain.

Step 2

The calculation of subzone temperatures follows the method used in the **ZoneTempPredictorCorrector** module and described in the section **Basis for the System and Zone Integration**. Namely a third order finite difference expansion of the temperature time derivative is used in updating the subzone temperatures. Otherwise the subzone temperatures are obtained straightforwardly by solving an energy balance equation for each subzone.

$$T_{oc} = (C_{air,oc} \cdot (3 \cdot T_{-1,oc} - (3/2) \cdot T_{-2,oc} + (1/3) \cdot T_{-3,oc}) + \dot{Q}_{ocz} \cdot Fr_{gains} + HAT_{oc} + MCPT_{tot}) / ((11/6) \cdot C_{air,oc} + HA_{oc} + MCP_{tot})$$

$$T_{mx} = (C_{air,mx} \cdot (3 \cdot T_{-1,mx} - (3/2) \cdot T_{-2,mx} + (1/3) \cdot T_{-3,mx}) + \dot{Q}_{ocz} \cdot (1 - Fr_{gains}) + \dot{Q}_{mxz} + HAT_{mx} + T_{oc} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,mx} + HA_{mx} + MCP_{tot})$$

Here $C_{air,oc}$ and $C_{air,mx}$ are the heat capacities of the air volume in each subzone. $C_{air,mx}$ is calculated by

$$R_{air,mx} = V_{mx} \cdot (\Delta z_{mx} / z_{ceil}) \cdot \rho_{air,mx} \cdot c_{p,air,mx} \cdot Mul_{cap} / (\Delta t_z \cdot 3600)$$

$$C_{air,mx} = R_{air,mx} \cdot \Delta t_z / \Delta t_{sys}$$

The gains fraction Fr_{gains} is a user input via a schedule. It is the fraction of the convective gains in the occupied subzone that remain in that subzone.

The other subzone air heat capacities are calculated in the same manner.

Mixed calculation

The above iterative procedure assumed that the UFAD zone model was applicable: i.e., conditions were favorable temperature stratification in the zone. Now that this calculation is complete and the subzone temperatures and depths calculated, we check to see if this assumption was justified. If not, zone conditions must be recalculated assuming a well-mixed zone.

If $T_{mx} < T_{oc}$ or $MCP_{tot} \leq 0$ or $H_{fr} \cdot H_{ceil} < \Delta z_{occ,min}$ then the following mixed calculation will replace the UFAD exterior zone calculation.

Note: $\Delta z_{occ,min}$ is the minimum thickness of occupied subzone. It is set to 0.2 meters.

The mixed calculation iteratively calculates surface convection coefficients and room temperature just like the displacement ventilation calculation described above. In the mixed case however, only one zone temperature T_{avg} is calculated. The 2 subzone temperatures are then set equal to T_{avg} .

First, Fr_{hb} is set equal to zero.

Then the code iterates over these steps.

Calculate T_{avg} using

$$T_{avg} = (C_{air,z} \cdot (3 \cdot T_{-1,z} - (3/2) \cdot T_{-2,z} + (1/3) \cdot T_{-3,z}) + \dot{Q}_{tot,conv} + HAT_{oc} + HAT_{mx} + MCPT_{tot}) / ((11/6) \cdot C_{air,z} + HA_{oc} + HA_{mx} + MCP_{tot})$$

$$T_{mx} = T_{avg}$$

$$T_{oc} = T_{avg}$$

Call $HcUCSDUF$ to calculate the h_c 's.

Repeat step 1

Final calculations

The UFAD exterior zone calculation finishes by calculating some report variables.

We define heights:

$$H_{trans} = Fr_{hb} \cdot H_{ceil}$$

$$H_{mxavg} = (H_{ceil} + H_{trans}) / 2$$

$$H_{ocavg} = H_{trans} / 2$$

Using the user defined comfort height we calculate the comfort temperature.

If mixing:

$$T_{comf} = T_{avg}$$

If UFAD:

$$\text{If } H_{comf} < H_{ocavg}$$

$$T_{comf} = T_{occ}$$

$$\text{Else if } H_{comf} \geq H_{ocavg} \text{ and } H_{comf} < H_{mxavg}$$

$$T_{comf} = (T_{oc}(H_{mxavg} - H_{comf}) + T_{mx}(H_{comf} - H_{ocavg}) / (H_{mxavg} - H_{ocavg}))$$

Else if $H_{comf} \geq H_{mxavg}$ and $H_{comf} < H_{ceil}$

$$T_{comf} = T_{mx}$$

Using the user defined thermostat height we calculate the temperature at the thermostat.

If mixing:

$$T_{stat} = T_{avg}$$

If UFAD:

If $H_{stat} < H_{ocavg}$

$$T_{stat} = T_{occ}$$

Else if $H_{stat} \geq H_{ocavg}$ and $H_{stat} < H_{mxavg}$

$$T_{stat} = (T_{oc}(H_{mxavg} - H_{stat}) + T_{mx}(H_{stat} - H_{ocavg}) / (H_{mxavg} - H_{ocavg}))$$

Else if $H_{stat} \geq H_{mxavg}$ and $H_{stat} < H_{ceil}$

$$T_{stat} = T_{mx}$$

The average temperature gradient is:

If $H_{mxavg} - H_{ocavg} > 0.1$

$$GradT_{avg} = (T_{mx} - T_{occ}) / (H_{mxavg} - H_{ocavg})$$

else $GradT_{avg} = 0.0$

Finally, the zone node temperature is set to T_{mx} .

Other variables that are reported out are Γ and ϕ .

$$\phi = (T_{mx} - T_{occ}) / (T_{mx} - T_{sup})$$

where T_{sup} is the zone supply air temperature.

5.2.5 REFERENCES

Qing Liu. 2006. The Fluid Dynamics of an Underfloor Air Distribution System. A PhD dissertation in Engineering Sciences (Systems Science) at UC San Diego.

6 SPECIAL VERSION OF ENERGYPLUS FOR MODELING YORK TEST ROOM WITH SOLAR ARRAY

A special version of EnergyPlus 1.2 (called here V1.2x) has been created that allows simulation of the York test room using the solar array, which is a bank of high-temperature lamps acting as an artificial sun. V1.2x is the same as the regular version except that it replaces the window heat transfer and solar gain calculation with measured values of glass temperature, solar radiation transmitted into zone, and, for inside opaque surfaces, ratio of solar radiation incident on surface to solar radiation transmitted into zone. When an interior blind is in place, the program also uses measured values of the slat temperature. V1.2x is based on the 5/20/04 StarTeam version of EnergyPlus.

There are several restrictions on using V1.2x (in the following the term “solar” refers to the short-wave radiation from the solar array):

There can only be one window. It must be an exterior window, which means that OutsideFaceEnvironment = ExteriorEnvironment for the window’s base surface. This is the window exposed to the solar array. (The York test room also has an interior window that faces the conference room, but this window will be covered by opaque panels and so will be input as an interior wall.

If the window has a shading device it must be an interior blind. EnergyPlus allows the slat angle to be input, but for any particular measurement run and corresponding EnergyPlus calculation, the slat angle should be fixed (as opposed to allowing it to vary during the calculation, which is possible in EnergyPlus).

The window input, as entered using Material:WindowGlass (and Material:WindowGas for double glazing), should match the actual window in terms of number of glass layers and inside surface emissivity. Otherwise the glass layer input properties are ignored. Neither a window frame nor a window divider should be entered.

If a blind is present, its input, as entered using Material:WindowBlind, should match the actual blind. This is necessary since the program needs accurate blind information to (1) calculate the long-wave radiation transfer from the glass/blind combination to the rest of the room, which depends on glass emissivity and slat properties (angle, width, slat-to-slat distance and emissivity); and (2) calculate the natural convective air flow in the channel between the glass and blind, which depends on glass/blind separation, slat properties (angle, width and slat-to-slat distance) and the blind’s top-, bottom-, left- and right-side opening data.

Only the window-related output variables shown in the sample inputs, below, are guaranteed to be meaningful for V1.2x. In particular, “Zone Window Heat Gain” and “Zone Window Heat Loss” are not meaningful and should not be printed. A new output variable, “Opaque Surface Inside Face Short-Wave Radiation Absorbed,” is described below.

The EnergyPlus modules that were changed for V1.2x are:

- Energy+.idd (to add the MeasuredUFADWindowData and MeasuredUFADSolarFraction objects)
- SurfaceGeometry.f90
- DataSurfaces.f90
- DataHeatBalSurface.f90

- HeatBalanceSurfaceManager.f90
- WindowManager.f90
- DataHeatBalance.f90
- HeatBalanceManager.f90
- SolarShading.f90

Sample inputs with bare glass and with interior blind are given below.

The measured values of transmitted solar, glass temperature and slat temperature are read into EnergyPlus 1.2x using schedules. A new input object, MeasuredUFADWindowData, has been created that references the schedules.

Another new input object, MeasuredUFADSolarFraction, allows input of the measured value of the ratio of solar incident on a surface to the transmitted solar.

6.1 DESCRIPTION OF NEW OBJECTS

6.1.1 MEASUREDUFADWINDOWDATA

At most one of these objects is allowed per input.

Field: Name of Window to which This Measured Data Applies

This is the name of the window, i.e., the name of a Surface:HeatTransfer:Sub object with Surface Type = Window.

Field: Name of Schedule of Measured Inside Surface Glass Temperatures

This is the name of a schedule whose values are measured inside surface glass temperatures in deg C. “Inside surface” here means the glass surface in contact with the zone air. This schedule name is a required input.

It is recommended that glass temperature be measured at several roughly equally separated points covering the entire glass surface and the results averaged. Of course, the glass temperature probes need to be well shielded from the solar radiation.

If the glass temperatures do not vary much with time it is sufficient to use an hourly schedule. Otherwise, a sub-hourly schedule should be considered.

Note that EnergyPlus does not interpolate hourly schedule values if a heat balance simulation time step less than an hour is used. In other words, if an hourly schedule is used the schedule value for a given hour will be used for the entire hour no matter what the simulation time step is.

Field: Name of Schedule of Measured Transmitted Solar Values

This is the name of a schedule whose values are measured transmitted solar values in W. This schedule name is a required input. Of course, the measured glass temperatures and measured solar values should be for the same measurement run. The V1.2x calculations will be meaningless if non-corresponding values of glass temperature and transmitted solar are used.

If the solar values do not vary much with time it is sufficient to use an hourly schedule. Otherwise, a sub-hourly schedule should be considered.

Field: Name of Schedule of Measured Blind Slat Temperatures

This is the name of a schedule whose values are measured blind slat temperatures in deg C. Assuming that the slats are metal and therefore of high conductivity, it does not matter on which side of the slat the temperature probes are placed, although placing them on the side away from the incident solar radiation will produce better solar shielding. As with the glass, slat temperatures should be measured at several equally separated points covering the blind area and the results averaged.

This schedule name is required if the window has a blind.

If the slat temperatures do not vary much with time it is probably sufficient to use an hourly schedule. Otherwise, a sub-hourly schedule should be considered.

Table 11: MeasuredUFADWindowData as it appears in the Energy+.idd file for V1.2x

```
MeasuredUFADWindowData,  
  \min-fields 4  
  \memo Used to enter measured window data for UCB/LBNL UFAD project  
  A1, \field Name of Window to which This Measured Data Applies  
      \note Must be the name of a Surface:HeatTransfer:Sub  
      \with Surface Type = WINDOW.  
      \note This object can only be used with exterior windows.  
      \required-field  
      \type alpha  
  A2, \field Name of Schedule of Measured Inside Surface Glass Temperatures (C)  
      \required-field  
      \type alpha  
  A3, \field Name of Schedule of Measured Transmitted Solar Values (W)  
      \required-field  
      \type alpha  
  A4; \field Name of Schedule of Measured Blind Slat Temperatures (C)  
      \note Used only if an interior blind is present on the window  
      \type alpha
```

6.1.2 MEASUREDUFADSOLARFRACTION

There should be one of these objects for each wall, floor, ceiling and internal mass surface in the zone. This object is not used for the window.

Field: Name of Surface (wall, floor, ceiling, internal mass)

This is the name of the surface to which the following ratio applies. Applicable surfaces are:

- Surface:HeatTransfer with Surface Type = FLOOR, WALL, CEILING or ROOF, or
- Surface:HeatTransfer:InternalMass

One or more Surface:HeatTransfer:InternalMass can be used to represent furniture.

Field: Ratio of Solar Incident on Inside of Surface to Zone Transmitted Solar

This is ratio of the solar incident on the inside of the surface to the total solar transmitted into the zone. Non-intuitively, the sum of these ratios should be greater than 1.0. This is because of inter-reflection of the transmitted solar among the zone surfaces.

If MeasuredUFADSolarFraction is not specified for a surface, this ratio will default to 0.0.

Table 12: MeasuredUFADSolarFraction as it appears in the Energy+.idd file for V1.2x

```
MeasuredUFADSolarFraction,  
  A1, \field Name of Surface (wall, floor, ceiling, internal mass)  
      \note Surface should be in zone containing window for which
```

	\MeasuredUFADWindowData applies
	\required-field
	\type alpha
N1;	\field Ratio of Solar Incident on Inside of Surface to Zone Transmitted Solar
mass	\note Sum of these fractions for all surfaces in zone, including any internal
	\note surfaces, should be > 1.0 due to inter-reflection of transmitted solar
	\note inside the zone.
	\type real
	\minimum 0.0
	\maximum 1.2
	\default 0.0

6.1.3 NEW OUTPUT VARIABLE

A new output variable has been added for V1.2x: “Opaque Surface Inside Face Short-Wave Radiation Absorbed.” This has units W/m² and is the sum of the solar radiation and short-wave radiation from the lights absorbed at the inside face of the opaque surfaces in the zone. Its value is:

$$[(\text{Measured transmitted solar}) * (\text{MeasuredUFADSolarFraction for surface}) + (\text{Short-wave radiation from lights incident on surface})] * (\text{Solar absorptivity of surface}) / (\text{Area of surface})$$