EnergyPlus Articles from the *Building Energy Simulation User News*

01/01/2005 through 12/31/2005

Simulation Research Group
MS: 90-3147
Lawrence Berkeley National Laboratory
University of California at Berkeley
Berkeley, CA 94720-0001

January 2006

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Office of Building Technologies, Building Systems and Materials Division of the U.S. Department of Energy, under Contract No. DE-AC02-05CH11231.

Table of Contents

RADIANT SYSTEM FLOW MISMATCH

I defined the Volumetric Flow Rate {m^3/s} to the PLANT LOOP, and the LOW TEMP RADIANT SYSTEM:CONSTANT FLOW, but have autosized both COILs for the central air handler. Here is the error message I received:

```
Flow mismatch in radiant system
--result will be an energy imbalance
-- should not get this error
WaterInjectionRate=-1.15E-005, in Radiant System=Z1SW RADIANT PANEL
```

Answer
Check the autosized flow rate for the cooling coils. Does the sum of those water flow rates equal or exceed the specified flow rate for the plant loop? Mixing hard sizes with autosizing does not always behave as expected. The autosizing algorithms do not have full knowledge of other system component inputs.

INSIDE TEMPERATURE FLUCTUATION

Is it possible to obtain the fluctuation of inside temperature in a time period without specifying an active system? I'm trying to determine if a passive system is maintaining the inside temperature under the comfort level. However, EnergyPlus asks for a constant inside temperature.

Answer
EnergyPlus does a good job of modeling floating temperatures. You should request the report variable "Zone/Sys Air Temp".

Here is a set of report objects that is useful with annual runs. This produces the ABUPS report with extra tables showing a summary of average zone air temperatures, humidity, and Fanger comfort for only those times when people are present.

```
Report:Table:Style,          HTML;
Report:Table:Predefined,     Annual Building Utility Performance Summary;
Report:Table:Monthly,        Occupant Comfort Data Summary, !- Name 3,
                             !- DigitsAfterDecimal People-Number of Occupants,
                             !- VariableOrMeterName01 HoursNonZero,
                             !- AggregationType01 Zone/Sys Air Temp,
                             !- VariableOrMeterName02
                             SumOrAverageDuringHoursShown,
                             !- AggregationType02 Zone Air Relative Humidity,
                             !- VariableOrMeterName03
                             SumOrAverageDuringHoursShown,
                             !- AggregationType03 FANGERPMV,
                             !- VariableOrMeterName04
                             SumOrAverageDuringHoursShown;
                             !- AggregationType04
```

TEMPERATURE STRIATION IN A ZONE

I want to specify a report that lists indoor temperatures of different vertical levels of a zone, i.e., from floor to ceiling. This could be useful, for example, in double facades, to determine stack effect, etc.

Answer
The standard zone model is well-mixed (ROOMAIR MODEL, "Mixing") which means that the zone air temperature is uniform throughout the zone. There are alternate multi-node room air models available, but these are not applicable to double facades. Stack effect modeling for double facades is calculated by looking at the boundary conditions of the zone, not by modeling a temperature gradient within the zone.

CONDENSATION WARNINGS

Can you tell me more about these two warnings? What does the "count = 100" mean? Changing the schedules seems to get rid of them.

```
 ** Warning ** Constant flow radiant system condensation shut-off
occurred: count = 100.

 ** Warning ** Surface temperature below dew-point temperature--
potential for condensation exists
 ** ~~~ ** Flow to the following radiant system will be shut-off to avoid
condensation
 ** ~~~ ** Constant Flow Radiant System Name = Z32INT RADIANT PANEL
 ** ~~~ ** Predicted radiant system surface temperature = 25.52
 ** ~~~ ** Zone dew-point temperature= 25.44
 ** ~~~ ** Occurrence info=COOLING DESIGN CONDITIONS - VGH
AACC RFP, 07/21 03:40 - 03:45
```

Answer
The second warning message indicates that the radiant cooling system you are trying to use is resulting in conditioned surface temperatures that are below the dewpoint temperature of the space. This can either be the result of low surface temperatures or high moisture levels within the space. If the surface temperature drops below the dewpoint temperature, condensation on that surface will result. The current radiant system model is configured to shut the system down to avoid this undesirable situation.

The first warning message indicates that this problem is happening multiple times--in this case at least 100 times. This is used to avoid a huge error file or having the base error message repeated 100+ times.

Changing the schedule (for example, control temperatures) is one way to eliminate these warning messages. We recommend running one of the thermal comfort models to investigate the impact of changed control temperatures on occupant comfort.

INPUT AND OUTPUT WEATHER DATA DIFFERENCES

Why is it that the output weather data called from simulation differs slightly from the input values? Does the weather data (outdoor drybulb, relative humidity, dew point temperature, direct/diffuse radiation, wind speed) change during the simulation process?

Answer
The standard EnergyPlus weather files (epw) contain hourly data. For state variables, such as temperature and humidity, the values in the epw file are the instantaneous measurements at 1:00 am, 2:00 am, and so on. Because EnergyPlus uses zone timesteps of less than one hour, these values are linearly interpolated for each timestep. Weather data values reported from EnergyPlus at the Timestep frequency will match the weather file one timestep per hour on the hour. Values reported at the Hourly frequency represent the average value over the hour, so they will not match the instantaneous value in the weather file. The solar radiation values in the weather file are integrated total values for the hour, so the interpolation must be done differently. The total radiation in [Wh/m^2] is used as an average rate for the hour in [W/m^2], and this average rate is assumed to be the value at the midpoint of the hour. Again, linear interpolation is used to determine the value for each timestep. The reported solar values in the output will match the value in the weather file on the half hour.

BUILDING GEOMETRY

I don't understand this warning in the error file.

```
Program Version,EnergyPlus 1.2.1.022, 2005-1-4 22:20,IDD_Version 1.2.1.022
   ** Warning ** InterZone Surface Azimuths do not match as expected.
   **   ~~~   **   Azimuth=270.0 in Surface=ZN003:WALL003, Zone=EPZONE
   **   ~~~   **   Azimuth=270.0 in Surface=ZN002:WALL003, Zone=WPZONE
   ************* Testing Individual Branch Integrity
   ************* All Branches passed integrity testing
   ************* Testing Individual Supply Air Path Integrity
   ************* All Supply Air Paths passed integrity testing
   ************* Testing Individual Return Air Path Integrity
   ************* All Return Air Paths passed integrity testing
   ************* No node connection errors were found.
```

Answer
Surfaces should face out from a zone and the vertices are entered when looking at the surface from the outside. For help in checking geometry inputs, see "report, surfaces, details"; a listing of all surfaces with azimuth, tilt, area, etc., is produced in the eio output file.

ZONE TEMPERATURES AND UNMET DEMAND

If the unmet demand is zero, then I should have enough capacity in the plant loop. I have a fixed zone cooling setpoint of 23C, but in some of our simulations we see zone air temperatures that are several degrees above our setpoint. If the capacity is there, why would my zone air temperature be off the setpoint?

Answer
If the unmet demand is zero, then the plant has enough capacity to meet the request from the chilled water coil. If the zone setpoint is still not met, then the chilled water coil is not delivering enough cooling. Either the airflow rate is too low, or the supply air temperature is too warm. This could be a control problem, an airflow sizing problem, or a coil sizing problem.

FORCING SYSTEM BEHAVIOR

I'm modeling a single zone with a VAV system and a radiant panel. I control the mixed air temperature, HC water inlet and CC water inlet with the AIR LOOP OUTLET node, which is controlled by a SETPOINT MANAGER to a setpoint of 12.2C. What happens at peak cooling load is that both the radiant panel and the VAV box go to full flow, and EnergyPlus allows the AIR LOOP OUTLET temperature to modulate.

1. Is there any way to lock in the AIR LOOP OUTLET temperature to allow the supply volume to vary?
2. Can the supply temperature and volume both be locked at specific values, so that the radiant panel modulates more fully? Can you suggest appropriate radiant panel schedules for this?
3. In our models we've seen the FANGER PMV produce PPD values of between 6-10% when zone mean air temperatures are relatively high (77F to 79F, the PPD formula is from the Engineering Manual). Does this make sense? The comfort band seems too wide; in most buildings complaints start flowing whenever zone DB temperatures exceed 75F.
4. When we get UNMET DEMAND on our plant loops in these simulations, the values are very high, much more than the total zone loads would ever be. We can't explain the values.
5. With this model the total zone load is the radiant panel cooling rate plus the CC rate, but the Zone/System Cooling Rate values are all over the place (but usually quite low). Why should this be?

Answer

1. If the air loop outlet setpoint is not being met, then the coil capacity or chilled water flow rate or chiller capacity is not adequate. Which cooling coil model are you using? Use of COIL:Water:Cooling or COIL:Water:DetailedFlatCooling. COIL:Water:SimpleCooling does not always deliver adequate capacity under certain conditions (this model will not be available in the next release).
2. It is very difficult to get a hybrid forced air/radiant system to behave as desired with current EnergyPlus control options. In your case, if you want the radiant system to modulate, then perhaps a constant volume system would be better than the VAV. Also, you should set the air system to run as the first cooling priority, so the radiant system can pick up the remaining load. Alternatively, perhaps the VAV system is oversized. If it is sized smaller so that it cannot meet the entire cooling load, then it will open to full flow, thus supplying a constant volume of air.
3. With cool surfaces for radiant slabs, it would be expected that comfort be maintained at higher drybulb temperatures. Check all of the comfort inputs to make sure they are appropriate (clothing, activity level, air velocity).
4. These unmet demands are why #1 is happening. The chiller plant is not able to meet the load, so the chilled water supply temperature rises, and then the coil cannot meet the supply air setpoint. Everything is simultaneous in EnergyPlus, so unmet demands result in unmet setpoints.
5. Zone/System Sensible Heating (and Cooling) do not include radiant slab system loads. (We need to clarify the definition or change the way it is reported.) Also, if there is outside air, that will place a load on the cooling coil that is not included in the load delivered to the zone.

EXTERIOR RADIATIVE HEAT TRANSFER COEFFICIENTS

I was reviewing some of the inputs for output variables and there are three outputs that confuse me:

1. Surface Ext Rad to Air Coeff
2. Surface Ext Rad to Sky Coeff
3. Surface Ext Ground Coeff

The Engineering Manual explains the calculation method thoroughly, but not the coefficients. Are these coefficients used to calculate the impact of radiation in a form similar to the exterior convective heat transfer coefficient? Is it reasonable to define the radiative component of heat transfer as the sum of these values and make the following calculation?

qrad=h_rad*surface_area(T_outside_temp-T_exterior_surface_temp)

Or are the coefficients defined in a different way?

Answer
You're *almost* right. These are indeed linearized coefficients for calculating the radiative heat transfer components; however, they each use a different ΔT. The sky temperature is used with the "Rad to Sky" coefficient, outdoor dry bulb is used as an assumption for the ground temperature with the "Rad to Ground" coefficient, and the "Rad to Air" coefficient also uses outdoor dry bulb unless Other Side Coefficients are being used. I presume that you saw the (rather brief) definitions of these variables in the Input Output Reference (p. 128, pdf p. 158).

In the Engineering Manual, the equations that use these variables are shown in the section "Heat Balance Cases" beginning on p. 53 (pdf p. 91). These variables translate as follows:

Surface Ext Rad To Air Coeff = HA = HExtSurf (which includes both convective HO and radiative HA)

Surface Ext Rad To Sky Coeff = HS = HSky

Surface Ext Rad To Ground Coeff = HG = HGround

The algorithm samples are a bit out of date, because the current code keeps HO and HA separate now instead of combining them first.

SOLAR HEATING SYSTEM

I'm modeling a solar collector heating system with a low temperature radiant floor. I'm using a water heater object to exchange heat from the collector loop to the radiant floor water loop and I'd like to run with both solar energy and gas (as backup energy source). The gas burner should start when the temperature on the second loop (radiant floor) goes below 30C. If I set the water heater setpoint to 30C the temperature inside the water heater tank (collector loop) does not go above 30C! I'd like the water heater storage temperature to go as high as possible, in order to store as much solar energy as possible. I would like to use a water heater and a boiler object in series but I can't make this work.

Answer
The current water heater model can be a bit difficult to work with. I would suggest putting the boiler in parallel with the water heater rather than in series. I know the real system would put them in series, but for the simulation the energy use, this configuration should be equivalent. This way the mass flow does not have to be the same in both components (which may be why you are having trouble with them).

HUMIDITY RATIO

In my simulation model, the Zone heating design supply air humidity ratio doesn't need to be controlled. I set the Field:Zone heating design supply air humidity ratio as blank, but the following error message was given:

```
** Severe  ** Error detected in Object=ZONE SIZING, name=SP1
**   ~~~   ** Field [Zone heating design supply air humidity ratio] is
required but was blank
```

In my model, there is no preheat coil, so I didn't control the preheat design set temperature, and I set this field as blank. The error message was:

```
** Severe  ** Error detected in Object=SYSTEM SIZING, name=VAV SYS 1
**   ~~~   ** Field [Preheat design set temperature] is required but was blank
```

For these two fields, they both can't be set as blank, but they don't need to be controlled in my model, so how do I set them?

Answer
1. The system sizing calculations need some sort of humidity ratio to work with, so the "Central heating design supply air humidity ratio {kg-H_2O/kg-air}" is required. Enter a reasonable value consistent with the specified supply air temperature. Changing this value will have only a minor impact on the sizing results.

2. If there is no preheat coil, the "Preheat design set temperature {C}" field in SYSTEM SIZING is ignored.

SOLAR COLLECTOR LOOP

I was reviewing the solar collector example file and noticed that Column H of the output file (WATER HEATER SOURCE OUTLET NODE:System Node Temp) is not the same as Column J (COLLECTOR LOOP INLET NODE:System Node Temp); does that make sense? I thought they were the same node.

Answer
The "Water Heater Source Outlet Node" and "Collector Loop Inlet Node" are technically different nodes, one on the source side of the loop and the other on the demand side of the loop. The two nodes, however, are directly connected by the interface between the two sides of the loop, so it's natural to expect that they should have the same temperature at all times. The reason that they do not have the same temperature is because it is necessary to simulate the loop with some thermal capacitance in order to achieve stability of the solution. In a real system, this thermal capacitance is the mass of the water in the pipes. This can be input using the "volume of the plant loop" field in the PLANT LOOP object, but most times we autosize that number, which calculates the minimum volume necessary for a stable solution. The jump from "source loop side outlet node" to "demand loop side inlet node" is where we calculate the effects of the thermal capacitance, which results in a temperature difference. If you look closely at any plant loop, you'll see that there is always a temperature difference between the "source loop side outlet node" and the "demand loop side inlet node."

AIR-TO-AIR HEAT RECOVERY WITH ECONOMIZER

I have an air-to-air total heat recovery in the system. I set it up that the system could go into economizer mode when the outdoor condition is within a certain limit. When I ran the simulation using EnergyPlus, an error message was generated; it said: HEAT EXCHANGER:AIR TO AIR:GENERIC "ERV" -- the average air volume flow rate was <50% or >130% of the nominal HX supply air volume flow rate. To my understanding it is because the outside air flow rate is getting higher than the minimum outside air flow rate, which is the same as nominal air flow rate of the ERV, when the system is in economizer mode. Am I right?

Answer
Economizer choice for Controller:Outside Air object should be set to "Bypass" when heat recovery is used. See Heat Exchanger:Air To Air:Generic in the InputOutput Reference for further details.

DUAL SETPOINT EXPLAINED

Will you explain how zone control works if I select a dual setpoint with deadband and both cooling and heating schedules are set.

Answer
Dual setpoint with deadband operates as follows:

Heating
If the zone temperature is predicted to fall below the heating setpoint for the current timestep, then heating is requested from the HVAC system to maintain the zone at the heating setpoint.

Cooling
If the zone temperature is predicted to rise above the cooling setpoint, then cooling is requested from the HVAC system to maintain the cooling setpoint.

Deadband
If the zone temperature is predicted to remain in the deadband between the heating and cooling setpoints, no heating or cooling is requested from the HVAC system. Depending on the system type, the sytem may continue to provide air flow which causes some heating or cooling even when the temperature is in the deadband. For example, a furnace with DX cooling with continuous supply fan operation. The furnace and cooling coil will remain off, but unconditioned mixed air will be supplied to the zone.

Note that the "Zone/Sys Heating/Cooling" report variables indicate the actual impact of the HVAC system on the zone, not the load requested.

LOOPS AND EQUIPMENT IN SERIES

1. Can I use EnergyPlus to simulate a loop with two boilers (or a water heater and a boiler) in series?

2. When I have many single component branches connected in series, where does EnergyPlus actually *see* the order in which the branches are connected? Does the order come from the branch list order? And in the branch list I might also have parallel connected branches (with splitter and mixer); does their position in the branch list count?

Answer
1. Yes, components may be in series on a branch.

2. For complex plant loops, each subloop (supply side or demand side) has an inlet branch (with one or more components in series), then a splitter with one or more branches in parallel (each branch my have one or more components in series), then a mixed and an exit branch. Branches are never in series with each other. Branches are simulated in the order listed in the branch list, so the order must be inlet branch, then parallel branches, then exit branch. The order of the parallel branches should not make any difference.

CHILLER:ELECTRIC INPUT

My question is about the Chiller:electric Object (on page no. 383 of Input/Output Reference Manual), the capacity ratio curve. How do I determine the value of "Ratio of Available Capacity to Nominal Capacity"? Is the value of nominal capacity obtained from manufacturer's data (like the Carrier catalogue)? Also, how do I determine the values of "Temperature entering the condenser" and "Temperature leaving the evaporator"? Are these values in the catalogues too? And what is the difference between these values and those of "TempCondInDesign" and "TempEvapOutDesign"?

Answer
The six "design" values represent the chiller performance at one operating point, usually the published rating point for the chiller. As the water temperatures and part-load change, then the capacity and COP (1/EIR) will change. To define your own performance curves, you will need the manufacturer to provide a set of capacity and COP data for a range of condenser and chilled water temperatures. If you do not have access to such data, then use one of the sets of curves provided in DataSets\PerfCurves.idf

DISPLACEMENT VENTILATION

1. EnergyPlus develops the UCSD Displacement Ventilation Model for the existence of room air temperature difference. However, it requires that the main heat sources be located in the occupied zone, right?

2. Can I model an underfloor air distribution (UFAD) system with EnergyPlus?

Answer
1. No. The user can control how much of the internal gains are added to the occupied zone vs. the upper zone. (See "Gain Distribution Schedule" field in UCSD DISPLACEMENT VENTILATION MODEL CONTROLS).

2. EnergyPlus can model underfloor supply plenums and VAV terminal units with variable-speed fan and reheat coil, which are major components of UFAD systems. Search the Input Output Reference for "UFAD", and see example files 5ZoneSupRetPlen and 5ZoneSupRetPlenVSATU). One major limitation of the EnergyPlus plenum model is that the air temperature is uniform from one end of the plenum to the other. This limitation can be partially overcome by using multiple supply plenums in series, but each sequential plenum air temperature lags by one time zone timestep.

HOW TO USE METEONORM FILES WITH ENERGYPLUS

Over the last few years, a number of users have needed weather data to use with EnergyPlus but cannot find measured data on the EnergyPlus web site. When possible, we have created weather data using the Meteonorm software. Meteonorm extrapolates hourly data from statistical data for a location. Where statistical data aren't available, Meteonorm interpolates from other nearby sites. Generally a statistical approach is a last resort--weather files generated from statistics will not demonstrate the normal hour-to-hour and day-to-day variability seen in measured data.
To help users create a Meteonorm weather file for EnergyPlus, we developed these guidelines:

Step	Directions
1	Start Meteonorm.
2	Click the Site button.
3	Next click the WMO/OMM button and select the continent. (WMO usually means there's a weather station recording hourly data.)
4	In the search site box, enter the first one or two characters of the desired location name and a '*' and click on the >> button.
5	Select the site (if available) from the list and click OK. If there isn't a WMO site available, go to step A.
6	If the location is there, click on the name and Meteonorm will give any warnings about the data. Write down the warnings (in a text file) and note that you used WMO, Station or City data, and the version of Meteonorm used.
7	Click the Format button, select TMY2, and click OK.
8	Click the Hourly Values button, then click the Save button and gave the TMY2 a name when prompted. (Use the ISO 3-letter country abbreviation followed by the city and the format. For example, for Kathmandu, Nepal, this would be: NPL_Kathmandu_MN5.tm2)
9	Convert to EPW using the EnergyPlus WeatherConverter.
10	Post a .ZIP on the EnergyPlus_Support YahooGroup under Files/Meteonorm_Weather_files. The .ZIP should include the .EPW, .DDY, .STAT, and the warnings text file you created (give it a .INFO extension). Save the TMY2 source and the .AUDIT in a separate .ZIP but do not post it to the YahooGroup.

Each .ZIP includes these

- .STAT (EnergyPlus weather data statistics)
- .EPW (EnergyPlus weather file), and
- .INFO (Information about the source data and limitations from Meteonorm).

In all cases, you should review the .stat file for the location before using any of these files to ensure that it represents the climate of the location as you understand it. In many cases, a nearby location with measured data will be more appropriate than one derived from statistics. Use these files at your own risk.

If no WMO data are available, try this:
A. Click the Station button and select the continent again.
B. In the search site box, enter the first one or two characters of the desired location name and a '*' and click on the >> button.
C. Select the site (if available) from the list and click OK. If there isn't a site available, go to step Z. If there is a site, go back to step 6.

Finally as a last resort, try this:
Z. Click the Cities button and select the continent again.
Y. In the search site box, enter the first one or two characters of the desired location name and a '*' and click on the >> button.
X. Select the site (if available) from the list and click OK. If there isn't a site available, you are out of luck. If there is a site, go back to step 6.

NOTE: Quality of data declines exponentially if no WMO or Station data is available. Steps Z-X should be used if when no other data are available.

Please note that Meteotest significantly updated the wind and other calculations [hourly wind direction in TMY2, which we use, was constant ... the update makes them variable]. If you received a file from the EnergyPlus team before early December 2004, we strongly recommend that you download the new file. We are working on getting data for another 200 locations over the next few months; targets include Italy (60+ files), Brazil, Ghana, Kenya, Ethiopia, Nepal, Bangladesh, and China. All except Italy are from the UNEP SWERA project (so we're waiting on the data to become available on the SWERA web site **swera.unep.net/swera/**). As always, if you know of sources of weather data that we might be able to share with the EnergyPlus community, please contact **Dru Crawley.**

COMIS SITE WIND CONDITIONS

From the descriptions in the COMIS User Guide, I don't understand how to calculate the "Plan Area Density" in COMIS Site Wind Condition.

Answer
Plan Area Density = Built Area / Total Reference Area

where:

Built Area = looking from above (plan view), the area that contains buildings or other significant structures within the reference area

Total Reference Area = area surrounding the building; it extends from a radius of 10 times the building height to a radius of 25 times the building height

ASSISTED VENTILATION

I'm using the Direct Air Object given to a dummy zone and then mixed with the zone I need to simulate, adding a separate air supply for ventilation. I also use a completely different air loop for supply the direct air. Within the dummy zone I need to define walls. Is there a way to make the walls into non-heat-transfer surfaces or does the direct air object not care about satisfying zone conditions?

Answer
1. You cannot serve the same zone with two air loops. You can have more than one system serving a zone, but only one can be an AIR PRIMARY LOOP. The other equipment must be zone equipment (window AC, fan coil, etc.).

2. Direct air has no capability to control, so it will deliver the specified flow (unless the air loop supplying it cannot do so) all the times it is available.

3. The presence of ZONE CONTROL:THERMOSTATIC for a given zone determines whether there is a "zone load" that the HVAC equipment will try to meet if it can (but Direct Air cannot do so in any case).

4. The dummy zone does not need a full set of surfaces, it just needs one. You can use Surface:HeatTransfer:InternalMass if you wish. Or use Surface:HeatTransfer and make it adiabatic by setting the OutsideFaceEnvironment to be itself.

CHILLER:ELECTRIC, TEMPERATURE RISE COEFFICIENT

How do I identify the "Temp Rise Coefficient" in the module Chiller:Electric ? Also, how do I get "Required entering condenser air or water temperature to maintain rated capacity" and "Required leaving evaporator water outlet temperature to maintain rated capacity".

Answer
These values can be calculated by looking at a set of performance data for the chiller for a range of evaporator and condenser water temperatures. For example, say an electric chiller has a nominal capacity of 1000W at 6.7C Design Evaporator Outlet and 29.4C Design Condenser Inlet. Find another set of water temperatures at which the same chiller delivers 1000W of capacity (you may have to interpolate to find such a point in the performance data). For example, say the chiller also has a capacity of 1000W at 8C Evaporator Outlet and 32C Condenser Inlet. Then the Temp Rise Coefficient would be (32-29.4)/(8-6.7)=2.6/1.3=2.0.

DETERMINING TOTAL BUILDING ENERGY CONSUMPTION

I am simulating five Zone DX VAV (with gas reheat) building. I want to calculate the energy consumed by the building for the run period. From the variable report, I selected all key values for these variables:

1- Lights-Electric Consumption
2- Heating Coil Gas Consumption
3- DX Cooling coil electric consumption
4- Fan Electric consumption

I added them to calculate the total energy consumption by the building. However, for the same file I requested reporting by ReportMeter for the following variables (all key values were selected):

1- Elecricity
2- Heating gas

I added the result to find the total energy consumption and compared it with the previous results. The total energy consumption value from the second run was higher than the first run. I need help to determine the proper way of calculating the total energy consumption in a building.

Answer
The meter details (*.mtd) output file lists the connections of report variables and meters. The first section lists each metered report variable and the meters to which it is attached. The second section lists each meter and which report variables are listed. Electricity:Facility and Gas:Facility are the total use for the building and HVAC systems.

Question
Does this mean that if I ask for Electricity:Facility and Gas:Facility, I will get the total electricity and gas consumed by the building without having to add the electric and gas consumed by the individual variables?

Answer
Well, the short answer is "yes." However, another method to get these values for a full year is to use

```
Report:Table:Style,
 HTML;          !- ColumnSeparator (can be csv or other options)

Report:Table:Predefined,
 Annual Building Utility Performance Summary; !- ReportName1
```

TEMPERATURE SETPOINTS

I am simulating an insulated single space building with double pane windows using the Purchased Air option. The orientation of the building will change in order to show the effect of daylight in reducing or increasing the heating and cooling loads throughout the year. I would like to run it so that the Purchased Air is turned on according to certain temperature (temperature setpoints). Do the setpoints in EnergyPlus reflect the temperature of the specified space node or from the outdoor environment?

Answer
Purchased air is controlled to meet the setpoints specified in ZONE CONTROL:THERMOSTATIC and related objects. These setpoints specify the desired zone air dry bulb temperature.

SCHEDULE ERRORS

In my model, one of the schedules is set as follows,

```
SCHEDULE,
  COOL-SCHED,        !- Name
  Temperature,        !- ScheduleType
  COOL-SCHED-Week,      !- Name of WEEKSCHEDULE 1
  4,             !- Start Month 1
  1,             !- Start Day 1
  11,            !- End Month 1
  30;            !- End Day 1
```

When I ran the model, the ERR file gave this message:

```
 ** Severe ** Schedule "COOL-SCHED" has missing days in its schedule pointers
```

Should the schedule be specified all year round?

Answer
From p. 51 in the Input/Output Reference:

"Each schedule must cover the entire year or a fatal error will result."

This does not mean that you must have your equipment on the entire year, just that the SCHEDULE must address each day of the year. A replacement for the schedule shown above would be:

```
SCHEDULE,
   COOL-SCHED,        !- Name
   Temperature,        !- ScheduleType
   OFF-Week,
   1,
   1,
   3,
   31,
   COOL-SCHED-Week,      !- Name of WEEKSCHEDULE 1
   4,             !- Start Month 1
   1,             !- Start Day 1
   11,            !- End Month 1
   30,            !- End Day 1
   OFF-Week,
   12,
   1,
   12,
   31;
```

Where, of course, OFF-Week must be defined per usual conventions.

DIFFERENT FAN MODELS

I built a model with an air-cooled chiller; the fan model is the Fan:Simple:Variable Volume, and the fan coefficient values were set as the values of Inlet Vane Dampers in table 20 on page 748 in the Input/Output Reference. The model was run and I obtained the electricity consumption of the fan and the chiller. Then I substituted Fan:Simple:Const Volume instead of Fan:Simple:Variable Volume and ran the model again. And again I obtained the electricity consumption of the fan and the chiller. However, the results of these two cases were almost identical, and the result of the model with the constant air volume fan was even a bit smaller than that of the other model. Shouldn't the model with variable air volume fan consume less energy than the model with constant air volume fan?

Answer
Changing the fan object alone is not sufficient to change a VAV system to a constant volume system. The air terminal units establish the upper limit on airflow through a system and even a constant volume fan can deliver less than full flow (with linear power consumption) in order to maintain mass balance. To do your comparison, you could use either the VAV or CV fan object and simply change the minimum flow fractions on the VAV terminal units to 1.0. This will force the system to function as a constant volume system.

PV AND WYEC FILES

Where can I get detailed explanation of the data fields in the WYEC files? I need the data to do some hand calculations for PV energy.

Answer
EnergyPlus has a various ways of calculating photovoltaic power production using EnergyPlus weather data. It isn't necessary to model an entire building, so developing input files isn't terribly difficult for PV-only modeling. The example file demonstrating PV systems is "GeneratorswithPV.idf." There are three types of models available including a simple user-defined/constant efficiency, the equivalent one-diode model (originating from TRNSYS), and the newer, David King model from Sandia National Labs. Over 100 input data sets are provided for the Sandia model. The incident solar can be modeled to include the effects of other surfaces that might shade, or reflect onto, the panels.

PURCHASED COOLING AS BACKUP COOLING

I am simulating a building that can use cooling directly from condenser loop (cooling tower and so on). But in the summer season there are days the cooling load cannot be matched. So there is need to provide chilled water as backup cooling source. How do I set up purchased cooling as the backup cooling source in EnergyPlus? To clarify,, the cooled water from cooling tower is still the main cooling source. When the cooled water temperature from cooling tower is not low enough, then the purchased cooling is to be mixed with water from the cooling tower to lower the cooled water temperature.

Answer
The current free cooling components assume that the chilled water loop will operate either a chiller or the free cooling heat exchanger but not both. To model your situation, put two cooling coils in series in the main air branch. Connect the first cooling coil to a chilled water loop served by the free cooling heat exchanger. Connect the second cooling coil to a chilled water loop served by a chiller. If the first coil meets the desired supply air setpoint, then the second coil should remain off.

FLUID LOOPS

Does EnergyPlus have a virtual loop for refrigerants? If so, how does the component obtain the fluid properties for simulation? And is there a routine included in every HVAC component specially for the processing of the fluid properties?

Answer
PLANT LOOPS and CONDENSER LOOPS are currently only liquid loops and do not model refrigerants. Specific fluid properties are used by key components, but not by all. REFRIGERANT properties are currently used only by the COIL:WaterToAirHP:Cooling and COIL:WaterToAirHP:Heating.

OUTSIDE MINIMUM FLOW

I have a question about using the object "Controller:outside air." I am not sure how to input the field *minimum outside flow rate* combined with minimum limit field (FIXED or PROPORTIONAL). Assuming the system design maximum airflow rate is 6 m^3/s; with 80 occupants; each zone minimum airflow ratio 0.3 (VAV system); the sum of total maximum airflow rate for all zones is 7 m^3/s (it is bigger than system airflow rate). Set minimum airflow rate for each occupant 8L/s. If I use "fixed minimum," then the minimum outside air flow rate should be: 80*0.008=0.64 m^3/s. The maximum outside flow is 6m^3/s. Is this correct?

If I choose PROPORTIONAL MINIMUM and set the ratio of outside air flow rate to system air flow rate of 0.3, what should be the minimum outside air flow in this field? To my understanding, it should be 7 (sum of total air flow)* 0.3 (zone minimum air flow)* 0.3 (outside airflow to system airflow ratio)=0.63 m^3/s. So should the input be 0.63 into the minimum outside air flow rate field? And is this method correct?

Answer
The minimum outside air flow rate input should reflect the desired minimum OA flow when the system is at its full flow rate. With FIXED MINIMUM, the OA flow rate will equal the "minimum outside air flow rate" multiplied by the current "Minimum Outside Air Schedule Value" (if specified). However, if the total system supply flow rate falls below this value, the OA flow will be reduced to match the supply flow rate. It is up to the user to set the VAV minimum flow rates to deliver enough total flow to meet the desired OA minimum flow.

With PROPORTIONAL MINIMUM, the OA flow rate will equal the "minimum outside air flow rate" multiplied by the current "Minimum Outside Air Schedule Value" (if specified) multiplied by the ratio of the current system supply flow rate divided by the design full system supply flow rate.

Eventually, a clearer explanation of this will be added to the Input/Output Reference.

DIMMER

How do I specify the efficiency of a dimmer in daylighting:detailed. For example, if the efficiency is 70%, how would it be entered in the minimum power output and minimum lighting output?

Answer
If by 70% efficiency, you mean that the minimum power the dimmer will go to is 30%, look at the example file: DElight-Detailed-Comparison.idf. You enter the minimum power and minimum light output in DAYLIGHT:DETAILED in these fields: Minimum input power fraction for continuous dimming control, Minimum light output fraction for continuous dimming control. See the discussion of DAYLIGHT:DETAILED beginning on page 223 of the Input/Output Reference (page 253 of the PDF). These fields are discussed beginning on page 227 (page 257 of the PDF).

COIL MODEL QUESTIONS

I have two questions about the DX Coil Model. For the DX Coil Model -- DX Cooling Coil Model, I'm confused by a sentence in the Engineering Document:°

The total cooling capacity modifier curve (function of flow fraction) is a quadratic curve with the independent variable being the ratio of the actual air flow rate across the cooling coil to the rated air flow rate (i.e., fraction of full load flow).

The "actual air flow rate across the cooling coil" seems to be a mean value in a time step. What does that mean exactly?

Answer
Yes, the air flow rate reported by EnergyPlus is an "average" value for the simulation time step when the DX system's fan cycles on and off to meet the sensible cooling load. This average value is determined by the air flow rate across the cooling coil when the fan is on multiplied by the fraction of the time step the fan operated. So the "actual air flow rate across the cooling coil" referred to above means the air flow rate when the fan was on, NOT the average reported each time step.

Question
How can I define the "the actual air flow rate" from the manufacture's catalogue data in order to get the modifier coefficient?

Answer
Manufacturers provide a rating for air conditioning systems at some nominal air flow rate and some "rated" inlet air condition. At this nominal air flow rate and "rated" inlet air condition, the manufacturer states a gross and/or net cooling capacity. Be careful to use the gross cooling capacity as stated by the manufacturer. If the manufacturer only provides a net cooling capacity, the fan heat must be subtracted to yield a gross cooling capacity (gross cooling capacity is higher than net cooling capacity).

For this example, let us assume 1000 W gross cooling capacity at 2 m^3/s nominal air flow rate and at a "rated" inlet air condition of 26.7 C dry-bulb and 19.4 C wet-bulb when the outdoor temperature is 35 C.

The manufacturer also provides performance data tables at air flow rates different from the nominal air flow rate. Find the "rated" cooling capacity in the chart and note the "rated" inlet air conditions.

So, at indoor conditions of 26.7 C dry-bulb / 19.4 C wet-bulb and 35 C outdoor temperature we have from the performance data table:

```
Air flow (m³/s)
2.0
1.5
1.0
Capacity (W)
1000     *** rating point ***
  850
  657
```

Next this data has to be normalized to 1 at the rating point as follows:

Divide the air flow column by the rated air flow (2.0 in this example) and the capacity column by the rated capacity (1000 W in this example). Since capacity is a quadratic function of air flow I will also show the air flow rate squared in the example below. You get:

```
Normalized Capacity
1.0
0.85
0.675
```

```
Normalized Air Flow
1.0
0.75
0.5
Normalized Air Flow squared
1.0
0.5625
0.25
```

This normalized data set must then be curve fit according to the equation:

```
Normalized Capacity = A1 + B1*Normalized Air Flow + C1 * Normalized Air Flow *
Normalized Air Flow
-- or --
Normalized Capacity = A1 + B1*Normalized Air Flow + C1 * Normalized Air Flow
squared
```

When this is completed you will have calculated the regression correlation coefficients (we used a common spreadsheet) for Normalized Cooling Capacity as a function of air flow ratio (ratio of actual air flow to rated air flow) as:

```
A1 =  0.25
B1 =  0.95
C1 = -0.20
```

The EnergyPlus curve object would then look like this:

```
CURVE:QUADRATIC,
  CapacityAsAFuncitonOfFlow,
  0.25,
  0.95,
 -0.20,
  0.0,
  1.0;


!- Name
!- Coeff1 Constant
!- Coeff x
!- Coeff3 x**2
!- minimum value of x
!- maximum value of x
```

Question
From the formula of PLR, what do "sensible cooling load" and "steady state sensible cooling capacity" mean respectively? How does the program calculate them (is latent cooling load neglected)? Any suggestion and guide will be appreciated.

Answer
The sensible cooling load is the load to be delivered by the AC system to meet the thermostat set point temperature. The sensible cooling capacity is is the capacity of the AC unit when it runs continuously. So if the cooling load is 100 W and the sensible cooling capacity is 200 W, then the PLR is 0.5 and the AC unit only runs 50% of the time. To clarify -- Latent loads are not neglected, but the PLR is calculated based on the sensible portion of the load delivered divided by full-load sensible capacity.

COMIS ✯ WIND CONDITIONS

In the COMIS air flow is it necessary to define the wind condition? And if it is not defined, does the program get wind condition from weather file?

Answer
COMIS gets the wind speed and direction from the weather file. However, to translate that to wind pressures on the building and building openings, the user must input Pressure Coefficients for each facade by wind direction. For simple box geometries, EnergyPus has a feature to calculate Pressure Coefficients automatically using equations developed from wind tunnel tests.

COMIS ✯ INTERNAL AIR VELOCITIES

Does EnergyPlus give internal air velocities when COMIS is used for a naturally-ventilated house?

Answer
COMIS will calculate the amounts of bulk air flow from the outside and from zone to zone. It does not calculate internal air velocities, for which some type of CFD program would be necessary. It is possible to crudely estimate the air velocities immediately in front of a large opening by making a number of simplifying assumptions.

DX COOLING AND HEATING COIL INPUTS

Some of the inputs for DX heating and cooling coils include Capacity, rated COP and rated air volume flow rate. The InputOutputReference document states that these inputs must be at particular rated conditions. What if the specifications I have are taken at different rated conditions ? Is there any way i can adjust for this in EnergyPlus ?

Answer
The rated conditions for the DX Cooling Coil is 26.67 C (80 F) inlet air dry-bulb temperature and 19.44 C (67 F) inlet air wet-bulb temperature with 35 C (95 F) air temperature entering the condenser. The rated inlet air conditions are used to determine the rated air mass flow rate through the DX cooling coil which is, in turn, used to calculate a rated bypass factor (approach) for the coil at the "Rated SHR" specified by the user. If you knew the Rated SHR for the DX cooling coil at this condition (26.67 C Entering DryBulb, 19.44 C EWetBulb, 35 C OutdoorDB) you should be able to enter that value along with the your performance information (Capacity, COP, rated air volume flow rate, and normalized performance curves giving a value of 1.0 at your rating point). There are limitations on rated air volume flow rate per watt of rated total cooling capacity (0.00004027 - 0.00006041 m^3/s or 300 - 450 cfm/ton) so you would want to at least be near the rating point. Since the SHR of a DX heating coil is 1.0, the bypass factor is not required and you should be able to input DX heating coil performance information at your rating point (the only check made on a DX heating coil is the flow per watt test and there are less stringent air volume flow rate per watt of total heating capacity requirements for a DX heating coil).

Since EnergyPlus specifies the rating point, this is the only work-around available when using off-design performance information. Manufacturers typically provide the performance information at the rating point specified above, which is why that particular rating point is used.

WINDOW AND GLASS QUESTIONS

What do P, S-C and G-C mean in DOE-2.1E? For example:

GT-WIN-1 =GLASS-TYPE P=1 S-C=.45 G-C=7.75

And do I input the glass parameters such as solar transmittance, solar reflectance and visible transmittance in the IDF of EnergyPlus according to these data in DOE-2?

Answer
P, S-C and G-C are DOE-2 shortcuts: P is for panes, S-C is for shading coefficient, G-C is for glass conductance (center of glass). They do not translate at all to EnergyPlus inputs because these are overall properties of a complete window construction. In the EnergyPlus Datasets folder, WindowConstructs.idf contains predefined window constructions from the DOE-2.1E window library and lists, in comments, the corresponding glass type code, u-value, shading coefficient, etc. The materials listed are defined in WindowGasMaterials.idf and WindowGlassMaterials.idf

Question
Okay, then what is the relationship between the Glass-Conductance (G-C) in the DOE-2.1E INP file and Conductivity {W/m-K} in the MATERIAL:WINDOWGLASS in the IDF file of EnergyPlus?

Answer

DOE-2
GLASS-CONDUCTANCE (G-C) is used to specify the heat conductance of the total window except for the outside film coefficient. This keyword is part of the GLASS-TYPE command which describes a complete window construction which may be made of one or more panes (layers) of glass. (From the DOE-2.1 Reference Manual, p. III.49)

EnergyPlus
MATERIAL:WINDOWGLASS describes a single layer (pane) of glass. The Conductivity of this layer is just the conductivity of the glass and does not include interior or exterior film (convection) coefficients.

Single-Pane Window
For a single-pane window in EnergyPlus, MATERIAL:WINDOWGLASS-Conductivity would be equivalent to the GLASS-CONDUCTANCE minus the inside film coefficient in DOE-2.1E.

Multi-Pane Wiindow
For multi-pane windows, the DOE-2.1E value is the conductance of all glass panes and all air gaps combined, so it is very different from the EnergyPlus Conductivity.

2-PIPE FAN COIL

Can I create a 2-Pipe Fan Coil Unit with EnergyPlus?

Answer
Yes. Use the 4-pipe fan coil object, and set the availability schedules on the heating coil and cooling coil so that only one can be active at any given time.

DAYLIGHTING

I am interested in the daylighting algorithm described in the Engineering Auxillary Manual and by Winkelmann and Sellkowitz (1985). What are the criteria for selecting the sky conditions (there are four in EnergyPlus and two in DOE-2.1E), and what specific parameters are these conditions a function of?

Answer
The sky type is selected using a sky weighting factor determined by SkyClearness and SkyBrightness:

```
3.0 < SkyClearness < 4.0  -->  interpolate between "clear sky" and "clear turbid
sky"
    SkyWeight = MIN(1.0,(SkyClearness-3)/3)
    ...if SkyWeight = 1, then it's a clear sky

1.2 < SkyClearness < 3.0  -->  interpolate between "clear turbid sky" and
"intermediate sky"
    SkyWeight = (SkyClearness - 1.2)/1.8
    ...if SkyWeight = 1, then it's a clear turbid sky

1.0 < SkyClearness < 1.2  -->  interpolate between "intermediate sky" and
"overcast sky"
    SkyWeight = MIN(1.0, MAX(0.0, (SkyClearness-1)/0.2, (SkyBrightness-
0.05)/0.4))
    ...if SkyWeight = 1, then it's an intermediate sky
    ...if SkyWeight = 0, then it's an overcast sky
```

"Sky Clearness for Daylighting Calculation" and "Sky Brightness for Daylighting Calculation" are output variables that you can report out, if you wish. The final daylighting factors are interpolations/weighted averages (using SkyWeight) between the daylight factors calculated for each discrete sky type.

SkyClearness is calculated using BeamSolarRad and DifSolarRad come from the weather file:

```
      Zeta = 1.041*SunZenith**3
      SkyClearness = ( (DifSolarRad + BeamSolarRad)/(DifSolarRad + 0.0001) + Zeta
)/(1.+Zeta)

SkyBrightness is calculated as follows:

SkyBrightness = (DifSolarRad * 93.73)* AirMass / ExtraDirNormIll(Month)

REAL, PARAMETER, DIMENSION(12) :: ExtraDirNormIll=  &     ! Monthly exterrestrial
direct normal illuminance (lum/m²)
      (/131153.,130613.,128992.,126816.,124731.,123240., &
        122652.,123120.,124576.,126658.,128814.,130471./)
```

Also, the criteria are described quite well in the Engineering Document under the heading "Time-Step Luminance."
The reference is Perez et alia, *"Modeling Daylight Availability and Irradiance Components from Direct and Global Irradiance,"* Solar Energy 44, pp 271-289, 1990.

Here are links to the publications containing the daylighting algorithms in your question:

- LBL-18508, *DAYLIGHTING SIMULATION IN DOE-2,* by F.C. Winkelmann and S. Selkowitz, October 1984, published in Energy and Buildings, 8, 271 (1986) and
- LBL-11353, *DAYLIGHTING CALCULATION IN DOE-2, (Chapter III.2.9 of the DOE-2 ENGINEERS MANUAL),* F.C. Winkelmann. May 1983. Simulation Research Group, January 1983.

PID CONTROLLERS

Is it possible to develop dynamic models for HVAC components for use in EnergyPlus? Are there PID controllers in EnergyPlus?

Answer
EnergyPlus does not explicitly simulate any kind of feedback controller: proportional, PI or PID. In EnergyPlus, what CONTROLLER:SIMPLE and other controllers do is to enable the simulation of idealized control in which HVAC equipment, with no dynamics, closely follows the set-point, subject only to capacity constraints. This approach enables the efficient simulation of energy performance since the time step can be long, relative to the closed loop dynamic response time of the real equipment. This approach does not allow the use of dynamic models of HVAC components in EnergyPlus.

There are a number of simulation environments that allow the explicit modeling of HVAC dynamics and control; one is the SPARK program (a free download from http://simulationresearch.lbl.gov/). A unique attribute of SPARK is that it is in the process of being integrated with EnergyPlus, the aim being to allow selected components and subsystems to be simulated on short time scales within an EnergyPlus simulation. Unfortunately, this capability may not be available soon enough to meet your needs.

TIME STEP CLARIFICATIONS

What is the difference between the time step indicated in the Time Step In Hour object and the variable time step indicated in the HVAC simulation (Engineering Documentation)?

Answer
The Time Step In Hour object specifies the basic time step for the simulation. This is used in the Heat Balance calculation as the driving time step. When the HVAC portion of the simulation begins its solution for the current time step, it uses the basic time step as its maximum but then can reduce the time step, as necessary, to reach the solution.

The technical details of the approach are explained in the Engineering Documentation, under "Integrated Solution Manager."

Users can see the effect/actual HVAC time step used if they select the "detailed" option on an HVAC report variable (e.g. Zone/Sys Air Temp). To contrast, the "Zone" variables will only be reported on the Heat Balance time step (e.g. Mean Air Temperature).

Question
Time steps recommended in EnergyPlus are 4 for non-HVAC and 6 for HVAC calculations. Does this mean that using a time step of 1 will cause EnergyPlus to diverge or sacrifice accuracy?

Answer
Actually, 4 or 6 time steps per hour are necessary for the HVAC calculations. The heat balance simulation will give reasonable results at 1, 4, or 6 Time steps per hour.

Remember that the Time Step value must be evenly divisible into 60 -- therefore is limited to:
1,2,3,4,5,6,10,12,15,20,30,60

Question
Okay, so then how do the Loads and Systems modules handle time step settings?

Answer
This scheme is discussed in detail in the EnergyPlus Engineering Documentation (Integrated Solution Manager and sections following) and the papers that are referenced there. The entire Predictor-Corrector is based on this zone-system connection and needs pages to describe its connection. However, in a nutshell, the system tries to simulate lock step with the heat balance time step. When temperature instabilities are determined, the HVAC systems reduce their simulation time steps to something shorter than the heat balance

time step, currently with a one minute minimum. The systems simulate at this shorter time step until the end of the heat balance time step, then they update and the entire simulation moves forward in time.

For more information, please see the discussion of Time Steps previously published in the *Ask an EnergyPlus Expert* section of the newsletter:

- **Vol. 25, No. 1 January/February 2004,** "Time Step Reduction" page 7
- **Vol. 24, No. 5 September/October 2003,** "Time Steps" page 6

ENERGYPLUS CASE STUDY

I'm looking for any case studies of real buildings done using EnergyPlus?

Answer

Here's a new report from the National Renewable Energy Laboratory (NREL) where EnergyPlus was used in an energy performance evaluation of the Philip Merrill Environmental Center, headquarters of the Chesapeake Bay Foundation, in Annapolis, MD.

Analysis of the Energy Performance of the Chesapeake Bay Foundation's Philip Merrill Environmental Center

B. Griffith, M. Deru, P. Torcellini, and P. Ellis
National Renewable Energy Laboratory, 1617 Cole Boulevard, Golden, Colorado 80401-3393

The Chesapeake Bay Foundation (CBF) is dedicated to restoring and protecting the resources of the Chesapeake Bay, North America's largest estuary and a seriously threatened ecosystem. In 2000, CBF built the 31,000-ft^2 (2,880-m_2) Philip Merrill Environmental Center on a 31-acre (12.5-ha) site in Annapolis, MD, to serve as foundation headquarters. CBF incorporated numerous high performance energy-saving features (including a ground-source heat pump system to heat the building in winter and help cool it in summer) into the building to minimize its environmental effects on the bay. Large, south-facing windows provide additional passive solar heating and reduce lighting loads by allowing more use of natural light (daylighting). Building daylighting is monitored by sensors that automatically dim some lights when daylighting is sufficient to displace electric lighting. Manually and motor-operated windows attempt to use the Chesapeake Bay's breezes for natural ventilation. An energy management system tracks the outdoor temperature and humidity and uses this information to control the building's heating, ventilation, and air-conditioning system. The National Renewable Energy Laboratory (NREL) selected the Merrill Center as a technical case study after being contacted by CBF to provide an unbiased energy performance evaluation. Because the center has attracted much attention in the sustainable design community, an unbiased evaluation was necessary to help designers replicate successes and identify and correct problem areas. This report focuses on the monitoring and analysis of the building's overall energy performance. *(April 2005 • NREL/TP-550-34830)*

Download document (145 pages) http://www.nrel.gov/docs/fy05osti/34830.pdf

HEAT RECOVERY SET UP

Is there any plant equipment able to use the heat recovery loop to recover energy from equipment other than a water heater? In the EnergyPlus heat recovery example file the plant loop, rather than the heat recovery loop, is used to set up the heat recovery loop. Does this mean that the heat recovery object in the plant-condenser loops is unusable? Is there another example file that uses the heat recovery loop to specify the loop?

Answer

In development we are heading toward a general loop structure and not specifically-typed loops. We view loops as a collection of piping that allows fluid to flow through and connect components. Thinking in this manner the heat recovery loop object has been made obsolete and replaced with the more general loop structure connecting components together that can facilitate heat recovery. We have several example files that show this:

HeatRecoveryElectricChiller.idf
HeatRecoveryPlantLoop.idf
HeatRecoverywithStorageTank.idf

Yes, the water heater was used in these examples to allow for storage of the recovered heat, but it is not necessary. You could connect the heat recovered from generators directly to heating coils, for instance. Look in the Engineering Documentation in the "Heat Recovery Loop Systems" section for more information.

NO TEMPERATURE CONTROL

This is my first attempt at an idf and it's not working. I can't control the heating and cooling coil. I'm trying to model a very tightly controlled gallery storage area, which is supposed to have an RH of 50% ± 5%.

Answer
1. SET POINT MANAGER:SINGLE ZONE MAX HUM is placing a set point on node "AHU1 Humidifier Outlet Node," but the cooling coil controller's control node is "AHU1 Fan Outlet Node." You need to place the maximum humidity set point on the same node "AHU1 Fan Outlet Node."

2. SINGLE DUCT:VAV:NOREHEAT will not modulate the air volume flow rate to the zone for heating control; it only controls for cooling. If you want variable volume heating, then use SINGLE DUCT:VAV:REHEAT with Damper Heating Action set to Reverse Action.

We would suggest turning on report variables for "System Node Temp," "System Node Humidity Ratio," "System Node Setpoint Temp," "System Node Humidity Ratio Min," "System Node Humidity Ratio Max," and "System Node VolFlowRate" for key nodes in the system to observe how the system is responding to the control.

DESIGN COOLING/HEATING MASS FLOW

What is the meaning of the design cool/heat mass flow in eplszsz.csv? And can the report frequency of the eplszsz.csv be changed?

Answer
These are the "calculated" results of the zone sizing calculations in EnergyPlus. You can read about these calculations in the Engineering Documentation (see the link to documentation on p. 3). The purpose of the sizing calculation is to obtain for each zone a calculated design flow rate for cooling and a calculated design flow rate for heating. Also, the design heating and cooling loads for each zone. The zsz file shows the design flow sequences. They are useful for plotting to see the load or design flow shapes over a day. You cannot change the report frequency.

LOADS REPORTS ON HEAT TRANSFER SURFACES

I need a report variable that will tell me the heating and cooling loads of the various heat transfer surfaces. (Similar to the LS-B report in DOE-2)

Answer
It is very difficult to separate envelope loads from the total zone load, because a heat balance is performed at each surface which includes incident solar and long-wave radiation. There is no report variable with such a value. Surfaces interact with the zone air heat balance by convection. The ultimate load from the surfaces is (SurfaceInsideTemperature - ZoneAir Temperature) * SurfaceArea * SurfaceInsideConvectionCoefficient, but this includes the solar gains and radiant internal gains which are absorbed by the surfaces. The best way to approximate the contribution from the envelope is to take the total load and subtract off the gains from windows, internal loads, infiltration, and ventilation. The remainder will approximate the impact of the envelope.

PURCHASED:CHILLED WATER

I would like to create a Cooling Coil:Water which is only fed by purchased chilled water. Would you please advise me about the plant loop? I created these objects: purchased:chilled water, branch, branch list, plant equipment list. Do I have to create the whole water path (supply and demand sides, plant loop, etc.)? Which objects are essential?

Answer
With Purchased:Chilled Water, you need all the same components as you would with a chiller. See the 5ZoneAuto example file.

OVERHANG

I compared two (almost) identical rooms, one of which has an overhang. For the report variable I chose "Zone Transmitted Solar" but the amount of solar radiation entering the zone was the same for each room.

Answer
In the BUILDING object, check the Solar Distribution field. The overhang will not work if the field is set to "MinimalShadowing." If it is set to one of the FullExterior options, then turn on REPORT, Surfaces, Details and check the tilt and azimuth of every surface. Also, check the TransSchedShadowSurf field on the shading surface. If this is always 1, the surface transmits all solar and has no effect. This schedule should be 0 for opaque shading surfaces.

AIR-TO-AIR HEAT EXCHANGER INPUT DATA

I would like to simulate the performance of an air-to-air heat exchanger based on manufacturing data. However, the rating conditions from the catalog data are different from those of the ARI standard. Does EnergyPlus mimic the actual performance of the exchanger with the effectivenesses calculated under different rating conditions?

Answer

From the manufacturer's rating point, an air-to-air heat exchanger's effectiveness varies much more with changes in air flow than it does with variations in temperature (or humidity when considering latent effectiveness). The ARI 1060 Standard provides a fixed rating point so that all manufacturers are reporting the same information to consumers. You will notice from manufacturers data that the heating and cooling effectiveness does change, but not dramatically. This change is due to a change in the temperature difference between the supply air and exhaust air streams. So the information you have is probably adequate for simulation purposes as long as the delta T for heating and cooling are similar to the ARI standard.

The ARI 1060 Standard uses a 19.3C temperature difference (0.00408 kg/kg humidity ratio difference for latent effectiveness) for heating and an 11C temperature difference (0.00726 kg/kg humidity ratio difference for latent effectiveness) for cooling. You can plot the heating and cooling effectiveness versus delta T (or delta W for latent effectiveness) for the data you have and then draw a line through these points to extrapolate or interpolate and find the corresponding ARI rating points. You will see that the effectiveness data do not change significantly.

Since the variation in effectiveness is small for different delta T's (or delta W's), EnergyPlus assumes a constant effectiveness in cooling and heating. The cooling effectiveness inputs (100% and 75% air flow) are used when the supply air temperature (usually outdoor air) is greater than the exhaust air temperature (usually zone air). The cooling effectiveness inputs are used for calculating exiting conditions until the supply air temperature falls below the exhaust air temperature, when the heating effectiveness inputs are then used.

In addition, the effectiveness for air-to-air heat exchanger wheels is impacted by leakage through the seals. If you choose to simulate a wheel, use the "net" effectiveness values (~2 % lower than the thermal effectiveness) provided by the manufacturer.

AIR TO AIR HEAT EXCHANGER

Is the air-to-air heat exchanger model in EnergyPlus able to simulate the total heat exchange? And was the module developed for a sensible heat exchanger?

Answer
There are two air-to-air heat exchanger models in EnergyPlus:

object HEAT EXCHANGER:AIR TO AIR:FLAT PLATE or sensible-only heat exchange

object HEAT EXCHANGER:AIR TO AIR:GENERIC for sensible and latent (total) heat exchange

See the I/O Reference and Engineering documents (doc links on p. 9) for more details.

CONDENSER LOOP WITH GLYCOL

I need to simulate a building with a glycol-condenser loop. The loop feeds a chiller (supply side condenser loop). For the demand side, glycol passes through tubes into a cooling tower, and so is chilled by water. Is it possible to change the type of fluid in the condenser loop?

Answer
Yes, it is possible to change fluid types, but it's not necessary for most types of equipment. In EnergyPlus 1.2.2, only water-source heat pumps and the ground loop heat exchanger honor different fluid properties. Most other equipment (including chillers and towers) assumes water properties. In general, the primary energy impact of using glycol instead of water is a change in required flow rates due to different specific heats. This modification can be made by adjusting the pump head accordingly.

ZONE VOLUME CALCULATION

How is the zone volume is calculated in EnergyPlus?

Answer
Basic Algorithm:
Given:
A 3D Polyhedron P described as a set of faces where each face is described by its 3D vertex point list

- P can be subdivided into a set of pyramids, each defined by one face and a common "tip" point p somewhere in P
- The volume of each pyramid is given by: (pyramid base area)*(tip height above base plane) / 3
- The total polygon volume is then the sum of the pyramid volumes.

See W. R. Franklin, "Volume of a Polyhedron"
http://www.ecse.rpi.edu/Homepages/wrf/misc_notes/volume.html

Very accurate for enclosed spaces but not so good if many surfaces are missing. Therefore, we allow users to calculate their own and enter it in the Zone Object.

SOLAR HEAT GAIN ON A FAÇADE

I need to model a large convention center but I'm only interested on the effects that a large louver system would make on the solar heat gain in an adjacent interior space. I'm using Autocad, which doesn't use enclosed zones but uses zones corresponding to different parts of the model, like one for the roof, louvers etc. Can I find the w/m^2 incident on the façade by using EnergyPlus (and not have to worry about having the enclosed zones)?

Answer
Yes. To have EnergyPlus calculate the incident solar flux with shading effects, define the surfaces of interest as one or more exterior Surface:HeatTransfer objects which are connected to a single zone. Also define any exterior shading features. The zone does not need any other surfaces. If the simulation fails due to high temperatures, it may be necessary to add PURCHASED AIR or INFILTRATION to remove excess heat from the zone.

MODELING SHADED FAÇADES

I'm trying to model a shaded façade. It's a south facing façade with external shading that consists of a mesh of 6 x 6cm squares that have a 4cm depth, one meter apart from the outer façade surface.

Answer (from Andy Tindale of DesignBuilder) (www.designbuilder.co.uk)
You can achieve 'Grid shading' by placing 10 x 6cm wide X 60cm high windows side-by-side, each window having 2 sidefins (1 vertical shading device each side of each window) and 10 louvers (horizontal blades evenly spaced).

This is very simple to set up using DesignBuilder - just draw one window of the correct size, clone it nine times and set up the appropriate external shading device.

I have put together a simple example grid shading DesignBuilder file – if you're interested in seeing it please let me know and I will send it to you.

MODELING CURVED FAÇADES

How can I use EnergyPlus to simulate an office building with a large number of curved façades?

Answer (from Andy Tindale of DesignBuilder) (www.designbuilder.co.uk)
It is possible to represent curvature in EnergyPlus; the key word is ***represent***. You need to break the curves up into equivalent planar surfaces, 3 or more vertices; breaking a circular building into 8 or so equivalent surfaces is probably enough. DesignBuilder (www.designbuilder.co.uk) allows you to easily create arcs for curved façades and, thus, complete the EnergyPlus simulations.

NODES AND BRANCHES

Are all of the nodes declared in "node list" or can they be declared along the way, for example, while we do the branches? And do I fill in the branch list or is it automatic?

Answer
Nodes only need to be declared in the objects in which they are used. In some instances, this may require the use of a node list to connect more than one node to an object. For example, a SET POINT MANAGER object, which applies the same setpoint to more than one node, would require a NODE LIST. Also, outside air inlet node must be listed in an OUTSIDE AIR INLET NODE LIST object. Branch lists are very specific groups of branches associated with particular sub-loops.
The user must create branch lists.

GLASS ELEVATOR WITHIN A GLASS ELEVATOR SHAFT

I need help with a tricky modeling problem. I have a glass elevator "cabin" within a 15m high glass elevator shaft which is exposed to sunlight on three sides. I can easily simulate the elevator shaft without the elevator but I don't know how to calculate the shaft with the cabin inside. Also, there might be cooling provided by the air between the outside walls of the cabin and the inside walls of the shaft.

Answer
EnergyPlus does allow interior windows, but they are not considered "Sunexposed" because they do not see outdoor solar radiation directly. If we assume that the gap between the elevator cabin and the glass walls of the shaft is fairly narrow, then there are two approaches that may be applicable:

1. Define two zones, one for the empty portion of the shaft, and another for the elevator cabin including a portion of the surrounding shaft. Define the combined shaft glass and cabin glass as double-pane airflow windows (because you mentioned that the air in the shaft might provide cooling).

2. Define the shaft as a zone with exterior windows, and define the cabin as a second zone. The walls of the cabin would be interzone surfaces with windows (these surfaces and windows are defined in both the shaft and cabin zones, with exterior environment of OtherZoneSurface, referencing the companion surface as an Outside face environment object). Windows on interior walls receive both direct and diffuse sunlight from the exterior zone and transmit it to the interior zone. Note, however that the solar gain entering the second zone is treated as all-diffuse and directionality is lost at that point.

In both of these options, we would recommend setting the Solar Distribution field in the Building object to FullInteriorAndExterior or FullInteriorAndExteriorWithReflections.

This is a very complex problem, and the simulation results will have some uncertainty. We would recommend doing some sensitivity studies, varying uncertain parameters, in order to get a range of results. Also, if you use the thermal comfort reporting, they may not account for the impact of solar radiation incident directly on the person, which could be a significant effect in this situation.

CONNECTED ZONES

I am attempting to model zones without partitions separating them; however, the zones are kept at different conditions. To do this I have modeled them with partitions and also have attached windows to the partitions. The partitions are nearly completely transparent to solar, visible and long wave radiation and have a high conductivity. Is this the best way to model this situation or should I approach the problem differently?

Answer
This is a good start. You should also add some cross mixing to transfer air between the two zones; however, it's difficult to say what flow rate to use.

Note that windows in EnergyPlus are opaque to infrared (long-wave) radiation (see Engineering Reference p. 171, pdf p. 199), so there is no direct radiant exchange between surfaces in different zones even when there is an interior window. This should be a small effect if the windowglass is given a high emissivity so that is will simply exchange with Zone A on one side and conduct through and exchange with Zone B on the other side.

Also, be sure to use FullInteriorAndExterior solar distribution (in the BUILDING object) so that beam solar can be transmitted through the partition windows into the adjacent zone.

SURFACE HEAT BALANCE OF DECOUPLED PV

For the case of decoupled PV, does the surface heat balance consider the shading effect by the PV panels? If a PV is input as decoupled, do we need to include it in the construction of the base surface?

Answer
Yes, shading effects are considered. The incident solar will be reduced if other surfaces in the model will shade the PV panels.

If by "decoupled" you mean you are using the object "Surface:HeatTransfer:ExteriorNaturalVentedCavity" then the model assumes that none of the solar will make it to the underlying surface, it is all incident on the decoupled surface (baffle) that has the panels. And no, you do not include the PV panel in the construction of the base surface. (But if you model the PV as "integrated," then you would include it in the construction, but you would use a "Construction With Internal Source" object.)

MACROS IN ENERGYPLUS

Is it possible to write macros or scripts to run EnergyPlus? And if so, where can I find the documentation?

Answer
Go to the EPlusMainMenu.pdf, search for Macros. The EnergyPlus Auxiliary Programs documentation describes the macros in details. Also, try the GettingStarted manual. EnergyPlus also comes with two "batch" files, one accompanies the EP-Launch and one is a stand-alone (RunEPlus.bat). Both get installed in the "root" folder.

MINIMUM SYSTEM AIR FLOW RATE

The Input/Output reference states that

"If the zone VAV dampers are reverse action and can open to full flow to meet heating demand, the ratio should be set to 1."

However, in the example file 5ZoneAutoDXVAV this value set to 0.3. Is this a different case than what the manual describes? If yes, then when should the ratio be set to 1 and what other fields need to be changed?

Answer
For "reverse-action" damper control, the ratio should be set to 1. The change will affect the sizing of the terminal unit heating coils and the central heating coil. It might affect the overall design flow rates of the zones. Basically putting in 1 allows the sizing algorithms to do a better job of sizing your system when you are using reverse action dampers. The documentation is correct; if the example files are incorrect, we will change them.

LOADS CONVERGENCE PROBLEM

I'm simulating a simple room, but I always get this warning message:

Loads Initialization did not Converge (CheckWarmupConvergence)

What should I check?

Answer
Possible causes to your convergence problem:

- very high thermal mass such as very thick concrete

solution: increase max number of warm-up days in the BUILDING object

- moderate mass and inadequate space conditioning such that the building keeps getting warmer and warmer on successive days

solution: add HVAC, check building thermal properties, check if infiltration is included

- a soil layer modeled below the concrete slab

solution: remove this layer and read about ground temperatures in the Auxiliary Programs document

HUMIDIFIER

One of the required fields in Humidifier is "rated capacity," which is the humidifier's steam production capacity in m^3/sec. In the output file, the water consumption is presented in m^3(timestep); however, this water consumption is equal to [(steam consumption rate)*(timestep in seconds)]. To me, this result can't be the water consumption, it has to be the steam consumption -- right?

Answer
The "rated capacity" should be described as m^3/s of water converted to steam.

USING COMIS AND VENTILATION TOGETHER

I am trying to simulate the natural ventilation effect of a deep room with windows on both sides. In order to facilitate air flow in the room, there is an exhaust fan inside of the room. Can I use the COMIS module as well as ventilation at the same time?

Answer
When the COMIS air flow model is active, the Simple air flow model objects (VENTILATION, INFILTRATION, MIXING, and CROSS-MIXING) are inactive. It is one or the other, but never both types together. To add a constant airflow to a COMIS model, use COMIS AIR FLOW:Crack with a value of zero for "Air mass flow exponent." This should result in a constant air flow through the crack, thus simulating the exhaust fan. You would also need to associate an appropriate node and pressure with this crack to force the direction of flow.

HELP WITH ENERGYPLUS AND DESIGNBUILDER

I'm a new user of EnergyPlus and DesignBuilder. My problems are these:

I need to draw boundary walls. In TRNSYS there is a specific type of wall that allows that. So, if I have to simulate a single zone of a whole building, can I separate it with a boundary wall and specify the boundary temperature?.

Answers from Andy Tindale of DesignBuilder Software

Answer
You can't directly specify a wall boundary temperature in DesignBuilder, but you could set up zones surrounding your zone of interest and condition the temperature in the surrounding zones to the temperature required. Or you can generate the basic IDF data using DesignBuilder and set up temperature boundary conditions for each surface in your zone by working directly with the IDF data using the OtherSideCoeff option. Future versions of DesignBuilder will allow temperature boundary conditions to be set directly.

Question
Is it possible to draw perimeter walls of different thicknesses?

Answer
Yes, you may draw perimeter walls of different thicknesses; however, each block will have perimeter walls of the same thickness. That is, it is not possible to draw a block with one face having one thickness and another face with a different thickness. To change the thickness of the external perimeter wall, type in the wall thickness in the drawing options dialog in the bottom left *before* you start the draw block command. You cannot change the thickness of the block perimeter walls once they have been drawn in the current version of DesignBuilder. Note that the thickness of the walls in the 3-D DesignBuilder model does not affect the thermal properties of the wall. This is governed by the construction associated with the wall surface. But the thickness of the wall will affect the volume of air in the zone and therefore the ventilation heat loss/gain.

Question
How do I account for thermal bridges?

Answer
You can draw thermal bridges, such as concrete lintels and columns, in DesignBuilder by going to the surface level using the draw surface tool. You can draw as many opaque sub-surfaces as you want in this way, but they will all have the same construction (as defined on the constructions tab under the sub-surfaces header).

Question
Can I use Design Builder to generate a file and then modify it in Energy Plus?

Answer
Yes you can use DesignBuilder to create EnergyPlus IDF files and then modify them directly in EnergyPlus. However, you cannot import any changes you make to the IDF file back into DesignBuilder.

GRAPHICAL OUTPUT WITH ENERGYPLUS

I need to show graphs of the output variables. Also, I would like plot a number of different output variables on the same graph.

Answer
If you are using EP-Launch to run, after you have "reported" the variables in the output stream, the CSV files are automatically generated. These can be imported into a spreadsheet program (such as Excel) and you can use the spreadsheet program to create your graphs. This is discussed in depth in the Input Output reference under "Input for Output" as well as in the Output Details manual. See the EnergyPlus documentation links on p. 9 of this newsletter.

Alternatively, other than exporting the data to CSV (which EnergyPlus does automatically), you can also use one of the interfaces which will graph data. Both DesignBuilder and E2AC will graph the data.

Find interface links here: http://www.eere.energy.gov/buildings/energyplus/interfaces_tools.html

Question
I'm simulating a building with two thermal zones and I'm using EnergyPlus with IDF-editor. I want to graphically visualize the outputs for the following:

- air temperature of each zone;
- relative humidity of each zone air;
- sensible energy demand of each zone;
- latent energy demand of each zone.

Answer
The report data dictionary (.rdd) lists all report variables available for a given simulation. All variables you requested will be listed in the file. Add report variables to your input file. As an example, the zone temperature may be output by adding:

```
Report Variable,
  *,                        !- Key_Value
  Zone/Sys Air Temperature,       !- Variable_Name
  hourly;                   !- Reporting_Frequency
```

Use Zone Air Relative Humidity instead of Zone/Sys Air Temperature to provide zone relative humidity information.

Cooling and heating demand may be output by using one of the following as the variable name in the example above:

- Zone/Sys Sensible Load Predicted
- Zone/Sys Sensible Load to Heating Setpoint Predicted
- Zone/Sys Sensible Load to Cooling Setpoint Predicted
- Zone/Sys Moisture Load Rate Predicted

CAD / VERTEX ENTRY

I have a question about vertex entry. In my cad file I have a building divided into two thermal zones by a partition. This partition has a thickness (0,30 m) and perimeter walls with a thickness (0,35 m). The Input/Output Documentation states:

"Surfaces are always specified as being viewed from the outside of the zone to which they belong."

Does this mean that I should insert the coordinates related to the vertices of the external surfaces?
In this way, since the walls have a certain thickness, the coordinates of the partition are different according to what zone I analyze. Can someone help me?

Answer
The statement refers to the order of the vertices, not necessarily their values.

- You can specify the coordinates such that the volume is correct for the zone (as the thickness of the wall may change the volume but not the thermal properties of the surface)

OR

- You can enter the volume of the zone in the Zone object.

Question
How important is the starting corner and the vertex entry for shading surfaces?

Answer
As important as it is for walls, generally.

Question
What does it mean in the Input Output Reference Manual when it states that EnergyPlus automatically mirrors the surface?

Answer
This means that you don't have to be as careful of which way the shading "faces" as only things behind the shading surface can be shaded.

Question
Are the chosen entries SurfaceStartingPostion and VertexEntry valid for any shading surface? Is there a difference between horizontal and tilted surfaces?

Answer
Yes, these are valid and the same as for walls, etc.

Question
And is there difference in the geometrical description between detached and attached ones?

Answer
If you choose "relative" coordinates in your Surface Geometry object, then YES, the attached entries are relative to the zone origin coordinates. But the vertex entries are always "3D".

NODE TYPE DEFINITIONS

How are node types defined?

Answer
Node types are defined during input processing by the components that are connected to them (such as coils, chillers, fans, etc.) If the simulation terminates with errors prior to getting all input, it is possible for nodes to be reported as undefined. Otherwise, undefined nodes indicate nodes that were not used properly.

CONSTANT INDOOR TEMPERATURE

I need to simulate a simple "cubic" building where the indoor temperature must be held constant at 20°C. I tried to enter this in EnergyPlus by defining a "single heating setpoint" or defining a "thermostat" but none of these options gave me the constant 20°C indoor temperature after running the simulation. What would be the best way to enter this data into EnergyPlus?

Answer
You need a thermostat plus an HVAC system to serve the load request from the thermostat. Use the "SINGLE HEATING COOLING SETPOINT" to hold a constant indoor temperature and automatically switch between heating and cooling as needed. The simplest form of HVAC system is "Purchased Air", an ideal system with no real equipment. See ExampleFiles\PurchAirTables.idf

EXTRACTION FANS

I'm trying to model various extraction fans within the bathroom areas of a building. The fans have flow rates of around 0.03m^3/s with no supply air at all. What I want is purely an extraction fan serving a few zones, but only an extraction fan. Any information on whether this is possible with EnergyPlus and any help on how to get it started would be greatly appreciated.

Answer
You want to use ZONE EXHAUST FAN objects. You can have more than one exhaust per zone, but not a single one that serves multiple zones. There will also be changes in CONTROLLED ZONE EQUIP CONFIGURATION and ZONE EQUIPMENT LIST objects for the zone. Check out the example file "TermReheatZoneExh.idf." It sounds like you have no air system conditioning the space so you may want to add MIXING objects to declare exactly where the makeup air is coming from.

INTERNAL MOISTURE GAIN

Is it possible to introduce internal moisture gains (such as from people, plants, etc.) in EnergyPlus?

Answer
YES. Open EPlusMainMenu.idf, click the search button and look for "Internal Gains" (especially people, lights, etc.). There is an "other" option you might use for plants; however, quantification for plants could be problematic.

BAFFLES AND EXTERIOR CONVECTIVE CAVITY

I have an example room with an external baffle outside a wall with a window.

1. Considering that the baffle is opaque, why does the window still transmit a significant amount of solar energy (as if there was no obstruction in front of it)?
2. In the inputs for the baffle, how I specify the distance of the baffle from the wall? Since my baffle is vertical, the "Effective Thickness of Cavity Behind Exterior Baffle" field is not used.
   How can I specify the distance between my baffle and wall?

Answer

1. The baffle is associated with the Surface:HeatTransfer object and uses the "net" area. If that surface also has a sub-surface (a window) then the baffle is only on the part of the surface that surrounds the sub-surface. The window is treated in the usual manner and doesn't have the opaque baffle in front of it.
2. You would still use the Effective Thickness field. This determines the depth or thickness of the cavity regardless of tilt.

INDEX