EnergyPlus Articles from the
Building Energy Simulation User News

01/01/2006 through 12/31/2006

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

January 2007

Table of Contents

COMPACT MODULE

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 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.

Is it possible to use the compact module and the general module to configure the HVAC system and central plants together?

Answer
In some cases this can be done with some care to match node names and component names. It is possible to use COMPACT HVAC ZONES and then add air loops and plant loops using the general HVAC objects. And it is possible to use COMPACT HVAC ZONES and SYSTEMS and then add general plant loops. However, it is not possible to do the reverse such as general zones and systems served by a COMPACT HVAC PLANT.

To mix the two types of input, one must study the expidf file and learn what objects are created for each COMPAC HVAC object and to see what the automatically generated node names and component names are. Then general HVAC objects can be added to match.

An easier approach may be to use the COMPACT HVAC objects that most closely describe the desired system, and then edit the expidf file as needed.

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 <u>only</u> 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.

SET POINT NOT MET

I'm simulating a building that has high internal gains; it requires cooling year round. It also requires humidity control. I installed a terminal reheat coil and a boiler. In the design file (eio) the message states that the design hot water flow is "Design Boiler Water Flow Rate [m^3/s], 0.31956E-02" which is approximately 3.1956 kg/s. However, the spreadsheet states that the demand bypass flow is never less than 3.17 kg/s. I don't understand how this design flow rate is possible; the gas consumption is almost equal to the chiller electric consumption. Am I doing something wrong?

Answer
The sizing routines will size the reheat coils to be large enough to raise the full flow rate from the cooling supply temperature to the heating supply temperature. For a system like this, the reheat coils will tend to be significantly oversized and the entire hot water plant will also be oversized. We would suggest lowering the design heating supply temperature in the zone sizing objects for zones that will not require heating.

CURVE-FIT VALIDATION

Components, i.e., chillers, fans, etc., are defined according to a performance curve-fit policy. Curves, however, are often based on real working conditions and real equipment capacities. Are EnergyPlus users supposed to use the same coefficients as we did previously, after the equipment capacity has been changed? Should we use the curves and parameters in the example files if we cannot get the data from the equipment producers?

Answer
Keep in mind that there are at least two possible "policies" for curve-fits. One option is to use curves that are normalized with respect to capacity (or efficiency) at a single operating point. The other is to use curves that directly fit the capacity/efficiency data without normalization. It is not unreasonable practice to use normalized curves for components that have a different nominal capacity or efficiency than the one used to produce the curves. Clearly it is better to use performance curves that apply to the unit being modeled, but this isn't always practical. On the other hand, direct performance curves (that aren't normalized) are basically limited to the original unit and cannot be used to model components with a different size or nominal performance.

Developers have different opinions on which is the best way to go. The normalized-curve group wants curves to be more general so that one can easily adjust equipment capacities when doing a simulation study. The direct-curve group is more wary of the pitfalls of abusing curves and want to force the user to use curves that match the component. I am not sure if it is true across the board, but curves tend to be of the normalized type in EnergyPlus. So it is not wrong to re-use such curves for different capacities. However, do not assume that the example files that are shipped with EnergyPlus represent best modeling practice; it is better to think of them as test and demonstration files.

Question
Another question is whether the object SINGLE DUCT:VAV REHEAT can deal with heating condition alone in a zone without an additional heating device (such as a baseboard heater), especially in a cold area?

Answer
The VAV reheat terminal unit is capable of meeting the heating load in a zone without additional heating equipment.

Question
So can I assume that the "normalization" means a ratio of a real-condition parameter to a rated-point one? Then how about curves for Electric Chiller, WindAC,Direct-fired absorption Chiller?
Are they called normalized curves?

Answer
We are not aware of any EnergyPlus HVAC equipment curves that are <u>not</u> normalized.

ZONE LOAD PREDICTION

I am confused about the Zone Load Prediction. According to the Engineering Documentation, Zone Load is predicted with Eqn. 3, neglecting the air capacitance. However, in the ZoneTempPredictorCorrector.f90\SUBROUTINE PredictSystemLoads,
TempDepZnLd(ZoneNum) = (11.0/6.0) * AirCap + TempDepCoef
TempIndZnLd(ZoneNum) = TempHistoryTerm + TempIndCoef
Is TempDepZnLd(ZoneNum) = Predicted Zone Load? Or have I misunderstood?

Answer
Eqn. 3 is shown without zone air capacitance and shows the quasi steady state solution that is used to derive Eqn. 4. Then Eqn. 4 is substituted into the unsteady Eqn. 2 to derive the equation for the unsteady solution on the zone air. Later on you will see that the Cz dT/dt is replaced with a Taylor Series expansion and the fully derived equations that are used in Predictor Corrector are Eqns. 8-11. I am not sure why you got stuck on Eqn. 3, but if you look at the form of the algorithm and the equations that I mentioned above you should start seeing the similarities.

COORDINATE SYSTEMS

What's the difference between relative and world coordinates?

Answer
EnergyPlus Documentation is searchable. Open EPlusMainMenu.idf and press the "search" button. Searching for "world coordinates" yields this article from the Input Output Reference.

Field: CoordinateSystem
Vertices can be specified in two ways: using "Absolute"/"World" coordinates, or a relative coordinate specification. Relative coordinates allow flexibility of rapid change to observe changes in building results due to orientation and position. "World" coordinates will facilitate use within a CADD system structure. Relative coordinates make use of both Building and Zone North Axis values as well as Zone Origin values to locate the surface in 3D coordinate space. World coordinates do not use these values. Typically, all zone origin values for "World" coordinates will be (0,0,0) but Building and Zone North Axis values may be used in certain instances (namely the Daylighting Coordinate Location entries).

In other words, with World Coordinates, you specify each coordinate for the surfaces as a stand-alone, absolute entry. With Relative coordinates, you specify coordinates in the Zone with respect to the Zone Origin Coordinates. And with relative coordinates, you can use building and Zone North Axes to change the placement of your building. windows

BASEBOARDS

The zones in my office building simulation have three types of equipment: a low temperature hydronic radiant system, a direct air component, and a baseboard heater. Sometimes, during the winter, the baseboard heater heats the zone even when the zone temperature is above the zone setpoint. I tried to track the problem down by simplifying the model to be only one zone (to be served with constant 18°C air via a DIRECT AIR component and an electric baseboard heater). However, the simple model still has the same problem, with the baseboard heater heating (at low level) even when the zone temperature is above the zone setpoint temperature!

Answer
It appears that the baseboard control logic does not recognize that the zone is in the thermostat deadband. "Zone/Sys Sensible Load Predicted" is zero. The 18°C DIRECT AIR cools the zone, and the baseboard turns on to offset the cooling from the ventilation air, seeking to provide a net load of zero to the zone. A Change Request has been posted to fix this problem. You can accomplish the equivalent simulation using SINGLE DUCT:CONST VOLUME:REHEAT instead of DIRECT AIR plus baseboard. The SINGLE DUCT:CONST VOLUME:REHEAT control logic does not have the same problem. When the zone is above the heating setpoint, the reheat coil is off.

DESIGN-DAY

How do I decide the maximum dry-bulb temperature in the DesignDay object? Is it the highest dry-bulb temperature in summer and the lowest one in winter? And what about the"Day of Month" object? Is it the corresponding date?

Answer
Using extreme high and low temperatures is an option, but this usually results in equipment that is larger than necessary. The best solution is to use one of the predefined DesignDay objects from the *.ddy file (included with each weather file). The temperatures in these objects are from ASHRAE and are based on a statistical analysis of weather data over 20-30 years. Jan 21 and Jul 21 are typically used, but you must also consider solar angles at other times of the year and it may be necessary to add spring or fall design-days.

SOLAR ARRAY WITH BOILER FOR DHW

I am attempting to simulate the following system to provide hot water for a building.

System:
A solar array and a gas-fired boiler combine to provide the DHW load. The solar array provides as much as it can and the rest of the load is picked up by the boiler.

Can anyone suggest how to emulate this kind of system? Is there an object in EnergyPlus that acts as a buffer vessel for hot water and can be heated using two disparate loops?

Answer
The example file "SolarCollectorFlatPlateWater.idf" illustrates a solar hot water system with DHW load, solar collectors, and storage tank (WATER HEATER:MIXED). You can connect a boiler, or another auxiliary WATER HEATER:MIXED (perhaps configured as instantaneous water heater), in series with the storage tank.

GENERATING INPUT FOR SOLAR COLLECTOR

I am interested in simulating the performance of solar panels distributed by a German manufacturer (Vitosol). I do have catalog data from the distributor. The catalog stipulates that ...

"The thermal loss coefficient and optical efficiency combine to form the collector efficiency curve which can be calculated on the basis of the following formula: $n = n0 - k1 . (dt/Eg) - k2 . (dt^2/Eg)$."

However, EnergyPlus's idf requires:

(a) Test Fluid
(b) Test Volumetric Flow Rate $\{m^3/s\}$
(c) Test Correlation Type
(d) Coefficient 1 of Efficiency Equation (Y-Intercept) {dimensionless}
(e) Coefficient 2 of Efficiency Equation (1st Order) $\{W/m^2\text{-}K\}$
(f) Coefficient 3 of Efficiency Equation (2nd Order) $\{W/m^2\text{-}K^2\}$
(g) Coefficient 2 of Incident Angle Modifier (1st Order)
(h) Coefficient 3 of Incident Angle Modifier (2nd Order)

As a result, I am unsure of the values I should instantiate in my input file. I would appear to have direct input for (d) only as the solar circuits have a mixture of glycol in order to provide frost protection.

Does anyone have advice?

Answer
(a) Test Fluid
The only option for EnergyPlus right now is WATER. If the collector is a solar hot water panel, then most likely it was tested using water as the test fluid, unless the manufacturer specified otherwise. We cannot yet model a glycol mixture in the solar loop (or any loop in EnergyPlus), so you must assume that it is water.

(b) Test Volumetric Flow Rate $\{m^3/s\}$
This is the flow rate condition under which the performance of the collector was tested by the manufacturer. The efficiency coefficients (d)-(f) are calculated for this specific flow rate. The flow rate should be indicated somewhere by the manufacturer, maybe called "nominal flow rate" or "recommended flow rate"? If not, try to ask the manufacturer.

(c) Test Correlation Type

This indicates whether the collector efficiency curve is correlated according to the INLET fluid temperature (Tin), the OUTLET fluid temperature (Tout), or the AVERAGE of inlet and outlet (Tavg). So basically you have to figure out if the "dt" in the manufacturer's equation is:

dt = Tin-Tair   or   dt = Tout-Tair   or   dt = Tavg-Tair

In the United States, the INLET correlation is the norm, but this may be different in Europe (thus the ability to choose).

 (d) - (f)
The efficiency equation that you describe from the manufacturer can be found in the EnergyPlus Engineering Reference, and appears as follows:

$n = c0 + c1*(Tin-Tair)/Isolar + c2*(Tin-Tair)^2/Isolar$

The manufacturer's coefficients are therefore related in this way:

(d) Coefficient 1 of Efficiency Equation (Y-Intercept) {dimensionless} c0 = n0

(e) Coefficient 2 of Efficiency Equation (1st Order) {W/m^2-K} c1 = -k1

(f) Coefficient 3 of Efficiency Equation (2nd Order) {W/m^2-K2} c2 = -k2

(g) - (h)
These are usually provided by the manufacturer.

COMIS AND CP VALUES

I have a problem about the CP values (using COMIS air flow model). When I change the model plan area density and expoent of wind velocity profile, on COMIS SITE WIND CONDITIONS, the CP values in the *.cif file are not updated. Does anyone have the same problem or I am missing something?

Answer
The site wind conditions are used to modify the wind speed hitting the building, but they do not change the CP values. The Plan Area Density was originally added for use by the Cp Calculation routine to calculate automatically the CP but that routine has been deleted from COMIS quite a few years ago. Consequently, the PAD input no has no effect on the computations. EnergyPlus does have a limited ability to calculate CPs for low-rise buildings based on an ASHRAE paper (choose Wind Pressure Coefficients as SURFACE-AVERAGE CALCULATION), but that also does not modify the CP values as a function of local wind conditions.

ENERGYPLUS VALIDATION

Is there any comparison between EnergyPlus and DOE 2 with regard to the accuracy of the predicted loads and energy consumption? I'm curious about the benefits (in terms of accuracy) obtained from simultaneously solving the building loads and HVAC systems.

Answer
Standard 140 and other validation results, comparing EnergyPlus to other programs, is on the EnergyPlus web site: http://www.eere.energy.gov/buildings/energyplus/testing.html

CONVECTION FROM WAFFLED CEILING SOFFITS

I am trying to implement a waffled, exposed concrete, soffit in EnergyPlus. I would like to see what effect the additional surface area might have on the performance of its thermal mass. What would be the best way to implement this?

Answer
If this mass is completely enclosed within a given zone, or separating two zones at similar temperatures, then describe it as internal mass and vary the area vs. thickness (Surface:HeatTransfer:InternalMass). If this is an exterior surface or separating zones at different temperatures, then the best we can suggest is to modify the convection coefficients on the waffled side (ConvectionCoefficients).

MULTIPLE CONTROL OF PUMPS

I can simulate multiple plant control units by using "COOLING/HEATING LOAD RANGE BASED OPERATION." However, I don't know how to simulate multiple pump control units.

Answer
In EnergyPlus version 1.2.3.031, there are two ways to model multiple pumps in a plant loop:

1. Use a single PUMP:VARIABLE SPEED in the supply side inlet branch, and let it respond to the flow demands of the other pieces of equipment.

2. Use branch pumps, or multiple pump objects on individual equipment branches. See ExampleFiles\5ZoneBranchSupplyPumps.idf

SCRIPTS, BATCH FILES

I'm trying to find a way to automatically do lots of runs in EnergyPlus. I need to do a single parametric sensitivity analysis and, therefore, need to vary one parameter at a time, save the file, run it and check the results. I had a look at the macros but I'm not sure if they are appropriate as it seems to me that each file needs to be specified anyway and saves should be done manually. I thought I could use the ##def to define all my variables in some files and then use the ##if and/or ##include to substitute each one in the "basic" .idf file but I couldn't find a way to save the file each time before running. So now I have a collection of input files - each of them with only one different variable! I opened a macro example and saw that a macro command should be included inside the .idf file so that it calls the corresponding .imf file. However, I need to use the same .idf file many times, changing one of the variables each time. By doing this I would need to save the files manually if I decide to use a macro command. So, the only way to do a parametric sensitivity analysis is to write a script in order to manipulate the .idf text files – right?

Answer
It is not necessary to save the files manually. The imf macro system is designed to do exactly what you want. You can run thousands of parametric runs all from a single imf input file. Remember that the epmidf output file is saved for future reference, and it shows the final resulting input for EnergyPlus after all macro processing is completed. The key is to use an output file naming scheme that prevents the outputs from overwriting each other.

One option for doing this is to use the Group capability in EP-Launch (see the Getting Started document). Make your master input file a *.imf file (which can then have ##includes of other idf or imf files). With the EP-Launch group simulations, there is an optional counter for running the same file multiple times. A file called "COUNTER.INC" is written prior to each simulation which sets a macro variable called "Counter." The imf input file can then have ##IF blocks that set other variables based on the current value of counter[].

Another option is to write your own batch files which call runeplus.bat. Again, the approach to use is to have the batch file write a short file such as "parameters.inc" (can be any extension you like) with ##set1 statements in it to set the desired parameters for a single run. Then in the master imf file ##include parameters.inc at the top of the file. Then the rest of the file can have ##IF block that are controlled by the variables which were set in parameters.inc.

We know this is rather confusing at first, but it turns out to be quite simple and powerful once you get past the initial learning curve.

From the March-to-August 2003 User News …
Question
I have to run quite a lot of (large and time consuming) simulations. Is it possible to use a batch file to queue the IDF files?

Answer
Yes, you can run EnergyPlus with batch files. There is a batch file included, called "RunEplus.bat" in the main EnergyPlus folder. Just write another batch file which calls RunEPlus.bat multiple times. (Note, use the "call" command to execute RunEPlus so that the main batch file will wait until it is complete). For more information about RunEplus.bat, see the Getting Started document, pp. 14-18.

COMIS AND NODES

I have a one zone model and use COMIS for natural ventilation. I need to determine mass flow through two openings on opposite sides of the zone and I need to define two external nodes at each of the two windows through which the air flows. But can I define a node in the middle of the zone for use only in COMIS? I can't find it in the manual? Only thing I can find is the defining of nodes for all types of mechanical ventilation. Or do i use 'Air mass flow from fromzone to tozone' for this?

Answer
The nodes used in the HVAC system inputs are not the same nodes used in the COMIS simulation. "Air Flow from FromZone to ToZone through COMIS Link" (and the reverse "Air Flow from ToZone to FromZone through COMIS Link") are what you want to report. Note that large openings can have bidirectional flow (for example, air can flow out the top of a large open window and flow in at the bottom) so you need to report both directions.

The Air Mass Flow FromZone to ToZone report variable describes the air flow across a surface (from an external node to a zone node).

The external node you describe is required (or not) based on the input for the COMIS Simulation object.

From the IO Reference manual:

COMIS Simulation object

Field: Wind Pressure Coefficients

Determines whether the wind pressure coefficients are input or calculated. The choices are INPUT or SURFACE-AVERAGE CALCULATION.

If INPUT, you must enter a COMIS CP Array object, one or more COMIS External Node objects and one or more COMIS CP Values objects.
The second choice, SURFACE-AVERAGE CALCULATION, should only be used for rectangular buildings. In this case surface-average wind pressure coefficients vs. wind direction are calculated by the program for the four vertical facades and the roof based on your entries for "Building Type," "Azimuth Angle of Long Axis of Building," and "Ratio of Building Width Along Short Axis to Width Along Long Axis" (see description of the following three fields). With this choice you do not have to enter any of the following objects: COMIS CP Array, COMIS External Node and COMIS CP Values. The calculated wind pressure coefficients are shown in the eplusout.cif file (see the example COMIS input file, next page).

Use this method when the wind pressure coefficients field is set to INPUT:

```
COMIS EXTERNAL NODE,
  SFacade,          !- Name
  1.0;              !- Outside Pollutant Concentration Factor {dimensionless}

COMIS SURFACE DATA,
  Surface_1,        !- Name of Associated EnergyPlus Surface
  CR-1,             !- Name of Air Flow Crack or Opening Type
  SFacade,          !- External Node Name
  1;                !- Crack Actual Value or Window Open Factor for
Ventilation {dimensionless}
```

Use this method when the wind pressure coefficients field is set to SURFACE-AVERAGE CALCULATION:

```
COMIS SURFACE DATA,
  Surface_1,          !- Name of Associated EnergyPlus Surface
  CR-1,               !- Name of Air Flow Crack or Opening Type
  ,                   !- External Node Name
  1;                  !- Crack Actual Value or Window Open Factor for
Ventilation {dimensionless}
```

For COMIS inputs you do not specify an internal node in a zone, use the zone name in the COMIS Zone Data objects and a surface name in the COMIS Surface Data object.

So try outputting the Air Mass Flow FromZone to ToZone and Air Mass Flow ToZone to FromZone report variables and see what happens.

BYPASS PIPES

I am modeling a huge building and am only simulating only the building side (and not the plant).
I understand that I need to use only purchased chilled water supply and purchased hot water supply instead of chiller and boiler, respectively. Now coming to my question. Do I really need to use bypass pipes for each purchased chilled water and purchased hot water.

Answer
Bypass pipes should not be necessary if the pump is intermittent, but it never hurts to have one even if it is never used.

I am puzzled that you say for "each purchased chilled water and purchased hot water". You only need one purchased chilled water and one purchased hot water to supply the entire building. One hot water loop supplying all hot water coils with a single purchased hot water supply object. Likewise for the chilled water side. For an example, see the hot water loop in TermReheat.idf

ATTACHED SHADING SURFACE

attached shading surface
I have an attached shading surface with some openings. However, I don't know how to describe these openings geometrically. I've considered using the field " Fraction of Shading Surface That Is Glazed" in shading the surface reflectance object without geometrically describing the openings. Then, to model the openings, I would use a window glass with very high transmittance and very low reflectance. Would this be correct?

Answer
"Fraction of Shading Surface That Is Glazed" in shading surface reflectance only affects the way the shading surface *reflects* sunlight, it does not alter the transmittance of the shading surface.

Shading surfaces are opaque, unless the the transmittance schedule (field "TransSchedShadowSurf") has values greater than 0.0. The easiest option is to set the shading surface transmittance schedule to AreaOfHoles divided by TotalAreaOfShade.

INTERNAL GAIN FROM LIGHTS

I need to set the design level of lights in order to calculate internal gain. According to daylighting illuminance, some fraction of the lights are switched on or not. Does EnergyPlus consider that situation while calculating the internal gains of lighting? I know there is a relation between the lighting consumption and daylighting illuminance, but is there also any relation between internal gain of lighting and daylighting?

Answer
Yes. When the power consumption of LIGHTS is reduced by daylighting controls, the internal heat gain is also reduced. The following report variables can be used to verify this: Ltg Power Multiplier from Daylighting, Lights Total Heat Gain, Lights Electric Consumption, Zone Total Internal Total Heat Gain.

OUTSIDE AIR NODE

I am using room AC packaged units with outside air mixers, fans, DX heaingt and cooling coils and electric heaters. If I use a common outside air node (NOUTSIDE) for all the room AC units and the outside air mixers, I get lots of Warning and Severe messages. If I use a different outside air node for each room unit (e.g., NOUTSIDE1,

NOUTSIDE2, etc.), then I get no errors.
Why can't I use a common outside air node?

Answer
Each outside air inlet node must be unique. Why? Because the outside controller, or in this case the window AC unit, sets the mass flow rate on the outside air inlet node. Then the mixer component, which is passive, takes the mass flow present on the outside air inlet node and mixes it with the flow from the return air node. HVAC nodes in EnergyPlus are single point inlets or outlets to a piece of equipment.

REPORT – MAXIMUM HEATING RATE VALUES

I have noticed that maximum heating rate values obtained with Report:Table:Monthly are very different(much larger)than the ones that can be obtained from the report of heating rates in Report Variable. This happens when using Purchased Heating as well as with a Heat Pump System.
Am I overlooking something or doing something wrong ?

Answer
Max values in table reports are computed for each timestep. Our guess is that the report variables are being reported hourly, so you are only getting the maximum rate averaged over one hour in that view. Set the REPORT VARIABLE reporting frequency to TIMESTEP (for Zone timestep variables) or DETAILED (for HVAC timestep variables), and the max values should match what is reported in the table report.

CONTROL NODE SETUP

I have a fixed flow system feeding two zones. On the air loop there's a cooling coil and a supply fan followed by a steam humidifier. Then I have two separate reheat coils serving the two zones. In these zones I want to control both the temperature and the humidity.

I have a CONTROLLER:SIMPLE for each of the coils (one cooling, two reheats) and SET POINT MANAGER:SINGLE ZONE REHEAT for the reheats and SET POINT MANAGER:SINGLE ZONE COOLING for the cooling coil.

I'm able to get control of the RH or the temperature in the zones but not both at the same time.

Could my control nodes be the source of the problem?

Answer
Since you have two reheat coils, I assume that they are part of a terminal unit such as SINGLE DUCT:CONST VOLUME:REHEAT or SINGLE DUCT:VAV:REHEAT. These components have the reheat controls built into the terminal unit and do not require a set point manager or a CONTROLLER:SIMPLE object. SET POINT MANAGER:SINGLE ZONE REHEAT is used to control coils in the main branch of an air loop, not in terminal units.

To control dehumidification, the cooling coil requires both a temperature set point manager and a Set Point Manager:Single Zone Max Hum to override when humidity control is required.

If this system is serving two zones, SET POINT MANAGER:SINGLE ZONE COOLING will
not work, because it will control the cooling coil for only the single zone to which it is attached. SET POINT MANAGER:WARMEST may be better for this application.

WINDOW HEAT GAINS

I'm studying window heat gains. I want to put gains and losses together like this:

```
                    [(losses)*(-1)] + gains = window heat transfer
```

Then I want to separate the solar portion from this so that I can plot a window breakdown graph. To do so I assume I subtract the solar portion from this calculated value; is that correct? Or by doing so will I be subtracting the solar portion twice? I ask this because when I looked at the documentation I saw that the window heat gains are composed of several components, including transmitted solar energy. However, when I checked the heat losses it says *"absolute value of heat flow when the total heat flow is negative (see the definition of heat gains above)."* Does that mean that the transmitted solar portion is zero or different in this case and the balance is negative? I ask this question because I'm afraid I might be subtracting the solar portion twice.

Answer
The components of Window Heat Gain and Window Heat Loss are the same. The net heat flow of the window is computed, then positive values are reported as Gain and negative values are reported as Loss.

CONTROLLERS

I am getting max iterations on a loop and I'm trying to go through the file iteratively to locate the source of the problem (node connections, correct flow rates, etc.). A brief description of the system I am trying to emulate:

I have a fixed flow system. The AHU takes full fresh air before entering a cooling coil, a heating coil and a supply fan (no reheat). The temperature leaving the supply fan is varied according to zone temperature.

This is what I have done so far:
I have two Controller:simple objects which both use the same control node (the supply fan outlet node). I have set this at 18C. However, I would prefer to let EnergyPlus decide what the setpoint should be.

Could this be the source of my problems?

Answer
EnergyPlus controllers all have a finite control tolerance (specified in the controller object), so the control is not perfect. Our guess is that the heating and cooling coil controllers are battling each other. If we understand your application correctly, SETPOINT MANAGER:SINGLE ZONE HEATING and SETPOINT MANAGER:SINGLE ZONE COOLING can be used to control your system.

Move the heating coil to be after the fan and place the SETPOINT MANAGER:SINGLE ZONE HEATING setpoint on the heating coil outlet node. Place the SETPOINT MANAGER:SINGLE ZONE COOLING setpoint on the fan outlet node and use a SETPOINT MANAGER:MIXED AIR to place a setopint minus fan heat correction on the cooling coil outlet node.

Question
Just so I know for future systems, do I have to use the SETPOINT MANAGER:MIXED AIR even though the system is set up to take 100% fresh air?

Answer
From the Engineering Reference Manual under Setpoint Managers:

MIXED AIR
The mixed air setpoint manager takes an already established setpoint (usually the supply air outlet node setpoint temperature), subtracts the supply fan heat gain, and applies the result as the setpoint temperature at the mixed air node (or any other node the user specifies).

So yes, if you want to control equipment to a setpoint and include fan heat, you will need to use the Setpoint Manager:Mixed Air object.

MINIMUM ENERGY USE

I want to predict the lowest amount of energy use that a building would require (in essence, a benchmark) in order to control adequate levels of thermal comfort in a space throughout a typical year. Would it be best to use purchased air or should I use a dummy HVAC system?

Answer
You should use an HVAC system that includes a full blown outdoor air system with economizer. Purchased air won't capture the free cooling. Because purchased air/heating/cooling is perfectly controlled and infinitely available, it's not the best way to accomplish your goal.

ASHRAE Standard 90.1-2004 Appendix G contains standard HVAC system configurations for various commercial building types along with the definitions of various parts of the systems. It is intended for evaluating 'above code' evaluations such as green building rating systems.

Also the 30% Guide for Small Offices from ASHRAE, IES, USGBC shows how to take small offices 30% beyond 90.1-2001 (or ~ 25% beyond 90.1-2004).

EXTRACT VENTILATION SYSTEM

I want to simulate a building with an extract ventilation system only, there is no supply air. I did this initially using ZONE EXHAUST FAN objects for each zone. Now I want to have a duct system connecting multiple zones, with a single extract fan. Can I set up an AIR PRIMARY LOOP without any supply ?

Answer
No; every AIR PRIMARY LOOP is a loop, with return air, outside air, mixed air, supply air. Also, the air flow must be balanced. Thus, whenever you exhaust air, it must be balanced by inlet outside air which is part of the supply air. So to use an AIR PRIMARY LOOP, you would need to set up the usual air loop but ensure there is no return air by setting the minimum outside air flow rate to the maximum in the OA Controller object.

PLANT LOOP -- AUTOSIZING

When autosizing a plant loop, do you have to autosize everything on that loop and everything connected to that loop or can you just autosize specific things like flow rates, etc. The reason I ask is that I tried autosizing flowrates in a plant loop and I encountered numerous errors. Then I tried autosizing everything on the plant loop and got even more errors. What should be my next course of action?

Answer
The Plant Loops and associated components are generally able to mix and match autosizing with hard sizes. In the Input Output Reference, there is a section that discusses this issue. Search for "Mixing User-Specified and Autosized Inputs." If you have not seen that yet, it may help you solve this problem.

WORK EFFICIENCY SCHEDULE

I am evaluating the thermal performance of a multi-family dwelling and I would like to know which value is recommended for the "work efficiency schedule" for summer and winter.

Answer
You should probably refer to the ASHRAE Handbooks and/or the Thermal Comfort Standard (55).

In the EnergyPlus datasets folder, there is a SCHEDULES.idf which contains example schedules for occupants, lighting, and equipment based on ASHRAE standards including multifamily dwellings.

The Building America benchmark also includes information on dwelling energy baselines and schedules, available at http://www.nrel.gov/docs/fy05osti/36429.pdf

UNDERFLOOR HEATING

What is the simplest method to model underfloor heating? I considered using a low temperature radiant system and, if this is a good choice, do I need to make another object. I don't see a link with any other objects (except for the nodes of course).

Answer
Reference ExampleFiles\RadLoHydrHeatCoolAuto.idf. The objects required to define a low temperature radiant floor are:

CONSTRUCTION WITH INTERNAL SOURCE and LOW TEMP RADIANT SYSTEM:HYDRONIC

In addition, the usual zone control objects are required for any HVAC systems (ZONE CONTROL:THERMOSTATIC, CONTROLLED ZONE EQUIP CONFIGURATION, ZONE EQUIPMENT LIST) along with plant loops and branches to connect it to the boiler.

LOW TEMP RADIANT SYSTEM:HYDRONIC requires specification of the "Hydronic tubing inside diameter" and the tubing length and water flow rate in order to autosize.

Andy Tindale, of DesignBuilder offers this alternative:
You could use a low temperature radiant System for modeling underfloor heating, but it isn't exactly simple. At DesignBuilder we have been experimenting with the high temperature radiant system which has a simpler definition. You can set up the high temperature radiant system to emit all of its radiant heat into the floor surface and this seems to give believable results. Of course, the heat should really be injected within the floor construction, but provided the underfloor heating system you are trying to model has coils fairly close to the surface this method should provide good results.

The next version of DesignBuilder will have an option for high-temperature radiant systems using this mechanism. You can get a working prerelease from the DesignBuilder website www.designbuilder.co.uk/downloads .

WINDOW-5

I am using WINDOW-5 to create a window. If I don't create exactly the same size window in .idf then EnergyPlus issues a warning. Is this simply a warning or will it affect the results as well.
How does EnergyPlus handle size variations?

Answer
It depends on the window details. Some windows will scale well, others will not. See discussion in Input Output Reference titled "Importing Windows from WINDOW-5" beginning on p. 180 (pdf p. 216), especially the top of p. 180 (pdf p. 218).

Question
Thanks for the suggestion. I guess I am not still clear how EnergyPlus adjusts the dimensions. The text pdf p. 218 states "*If there is one glazing system, the glazing system height and width from the* WINDOW-5 *data file are not used.*" Let's say my window in the idf file (actual size in my building) is 1000 mm x 1600 mm but in WINDOW-5 I had modeled it as Fixed (picture) window 1200 mm x 1500 mm. Will EnegyPlus adjust these dimensions with reasonable accuracy?

Answer
In the Input/Output Reference Manual, beginning on p. 184 (pdf p. 220), there is a description of the contents of the WINDOW-5 Data file. The quantities used in EnergyPlus are shown in bold.

The glazing system properties are all for center of glass and do not change with the window dimensions. The window subsurface dimensions in the EnergyPlus input file are the glass dimensions, and the glazing properties from the WINDOW-5 data file are applied to this glass area (less any dividers, frames are outside the subsurface dimensions).

The only quantities that are dependent on actual window dimensions are the average frame and divider characteristics. If the frame is uniform on all sides, then the average properties will be the same no matter what the window dimensions are.

So, for a window with a uniform frame, EnergyPlus will adjust the dimensions just fine (using the geometry in the EnergyPlus surface input object). Strictly speaking, however, the WINDOW-5 model results for center of glass performance will vary with size of the product (because of how convection is modeled inside the gap). Although this subtle feature leads to including a disclaimer/warning on the dimension mismatch, most would agree that it can be safely ignored for annual energy modeling. Matching the height dimension is more important than matching width. You could play with sizes in WINDOW-5 to see the magnitude of the impact on window performance levels.

BACKWARD COMPATABILITY

Is EnergyPlus v.1.3.0 backward compatible? Can I run a v.1.2.3 file in v.1.3.0?

Answer
For the most part, the answer is yes. There are many changes that require you to transition to v.1.3.0. EP-Launch will now check the selected file (if there is a version object) and you can launch the Transition program from the EP-Launch "File" menu. You can also view the ObjectStatus spreadsheet in the root folder of the installed version. This file has detailed explanations of changes to the objects.

This is particularly important for existing input files with COMIS objects – they are both internally and externally transitioned but will not be internally transitioned in the next release. If you have both COMIS and ADS objects in your input file – please contact email support (EnergyPlus-Support@gard.com) for assistance.

HUMAN INTERVENTION IN ZONAL MIXING

Are there any objects in EnergyPlus that account for the human intervention of closing/opening doors when temperatures in a room are too hot/cold? I have a situation in a simulation that I would like to emulate.

I'm modeling several atrium spaces that contain hot air which could be circulated around the open corridor spaces. If the spaces connected to these corridors are too cold, the occupants often open these doors until the temperatures have risen within the space. For instance, the corridor will be 25C while the room may only be 16C.

Are there any objects in EnergyPlus to cover this situation?

Answer
The Mixing object might be usable in this context. It has to be used judiciously, though, since there is no overall air balance being performed. However, in your case if the ultimate source of air is a large zone (atrium) this could work adequately. But you won't see lowered temperatures in the corridor and atrium, just higher temperatures in the offices. There's a delta T trigger that could mimic the opening/closing of doors. You'd have to estimate the flow rate.

BUILDING ORIENTATION

I want to simulate a building with a long axis rotated for 349 degrees from the north axis. I gave this value to the field 'North Axis' in 'Simulation Parameters: Building'. If I open the .dxf, no rotation occurred and I always get the following warning:

```
** Warning ** World Coordinate System selected.  Any non-zero Building/Zone
North Axes or non-zero Zone Origins are ignored.
   **   ~~~   ** These will be used in Daylighting:Detailed calculations but
not in normal geometry inputs.
```

(The same warning appears for every rotation angle.)

Answer
You need to switch your "SurfaceGeometry" from WorldCoordinates to RelativeCoordinates to have it take the origins, etc into account. Since you were originally in WorldCoordinates, you are probably okay to just do that but you should check Zone Origins and make sure they are all "0". In relative coordinate systems, windows and doors are relative to the zone origins.

OVERSIZING IN A HEATING COIL

I have a problem with a system. I have three zones which are supplied with fresh air (heated to 20C) and three electric heaters. The electric heaters take priority and, as a result, the heated fresh air picks up any additional heat. The problem is with an autosized system which manages the temperatures in the zones fine, the problem is in the plant loop.

The AHU consists of full FA setup with a supply fan and a heating coil. The heating coil is connected to a constant volume system (with bypass pipes) and a boiler. When I autosize the system, it is set to an overly large level. The heating coil never requires these flowrates so the vast majority of hot water flows through the bypass pipes (less than 1% flows through the heating coil). This means that the pump adds all the heat and the boiler is never required.

Can anyone point out why the flowrate through the heating coil would be so hugely oversized leading to zero input from a boiler.

My sizing parameters are 1,4;

Answer

The problem is due to autosizing a system which is providing neutral fresh air in a zone where the heating is to be met by baseboard heat. Both the air handler and the baseboard are being sized to meet the entire heating load.

In this case, the ZONE SIZING heating supply temp is specified as 20C and the zone heating setpoint is 20C, so the heating supply air flow rate for the air loop is being calculated as 2258.9 m3/s(!) (See component sizing reports in the eio output file).

In the ZONE SIZING objects, make these changes, and I think you will get a more reasonable result:

```
flow/zone,                !- heating design air flow method
<same value as outside air flow>; !- heating design air flow rate {m³/s}
```

HUMIDITY CONTROL

I have a single zone, single duct,constant volume, reheat system with a cooling coil, followed by a fan, followed by a humidifier on the air loop and a reheat coil just before the zone. In winter the temperature and RH are at their set levels but in the summer the RH levels are too high. (I need the RH to be 50% ± 5% year round as the zone is a gallery storage area). I have SET POINT MANAGER:SINGLE ZONE MAX HUM with the control node set to the fan outlet and SET POINT MANAGER:SINGLE ZONE MIN HUM with the control node set to the humidifier outlet. I also have SINGLE ZONE COOLING with the control node set to the fan outlet node.

Answer

Your cooling coil may not have sufficient capacity to meet the dehumidification load. If the coil was autosized, it was sized to meet the thermostat set point. A future release of EnergyPlus will allow the coil to be autosized to meet both the thermostat and humidistat set points. You may need to manually increase the cooling coil capacity.

If you are using a Controller:Simple object to control the cooling coil, the control variable should be set to TEMPandHUMRAT.

Report the set point at the fan outlet node and compare this value with the fan outlet node temperature.

```
Report Variable,
    Fan Outlet Node Name,       !- Key_Value
    System Node Setpoint Temp, !- Variable_Name
    Timestep;                   !- Reporting_Frequency

Report Variable,
    Fan Outlet Node Name,       !- Key_Value
    System Node Temp,           !- Variable_Name
    Timestep;                   !- Reporting_Frequency
```

When the actual temperature is above the set point, if the coil is at its maximum capacity you will need to add a report variable for your specific coil's capacity; you can find this report variable in the rdd file.

When the humidity is above the humidistat set point, the set point temperature calculated by the Set Point Manager:Single Zone Max Hum object will overwrite the set point temperature placed on the node by the Set Point Manager:Single Zone Cooling object.

An alternative method would be to add a Set Point Manager:Mixed Air object and use the cooling coil outlet node as the set point node and the fan outlet node as the reference set point node (leave the Set Point Manager:Single Zone Cooling object set point node name as the fan outlet node). Then move the name of the set point node for the existing Set Point Manager:Single Zone Max Hum object to the cooling coil outlet node. The same guidance given above also applies to this configuration (except you would add report variables for the System Node Setpoint Temp and System Node Temp using the cooling coil outlet node as the key value to help you debug this).

CONDENSER LOOP FLOW RATE

I have three variable flow chillers and a bypass branch in parallel between the condenser demand side splitter and mixer, staged with cooling load range based operation. On the supply side, I have a single variable-speed intermittent condenser water pump and single two-speed cooling tower on the supply side with a bypass in parallel. The problem is that the cooling tower and pump operate at 100% design flow (therefore high energy usage) as though all three chillers were operating, even when only one chiller operates.

I tried removing the bypass branch on the demand side but this generated an error. What is the correct arrangement of bypass branches to simulate the cooling tower and condenser pump flow, varying as the chillers are switched on and off (to model an arrangement where each chiller has it's own condenser water pump)?

Answer

The problem lies with having a single cooling tower on the condenser loop. Once the tower is active, it is requesting its full flow rate. Try making the tower a PASSIVE component on its branch. If that does not work, then you will need to model multiple cooling towers in parallel. When load controls correspond to the chiller load controls, then the towers should come on in sequence as the chillers do, and the condenser loop flow rate should follow.

UNIT VENTILATOR

I get this warning when I use a unit ventilator in one of my simulations:

```
** Severe  ** Error in Node="NODE 14066 IN", ZoneInlet node did not
find an outlet node.
**   ~~~   ** Reference Object=CONTROLLED ZONE EQUIP CONFIGURATION,
Name=14066
```

The unit ventilator seems to be working properly but I would prefer to know exactly why this error is appearing.

Answer

This error messages means that "NODE 14066 IN" is listed as a Zone Inlet node, but there is no component that has the same node name as an outlet node. Thus, this inlet node will have no flow. Perhaps this is an extra zone inlet node and is not intended to be connected to anything?

IFCTOIDF AND IFC HVAC INTERFACE

I am interested in the IFC to IDF file conversion capabilities mentioned on the DOE website. I've downloaded the IFCtoIDF utility and installed the BSPro COM server. I also have Autodesk Revit installed on the machine and am trying to convert simple buildings over to Energy Plus.

1. Am I correct in understanding that the IFCtoIDF utility is one directional?
2. And you cannot convert back to IFC from IDF?
3. Has anyone had success moving information from Revit to EnergyPlus?
4. If not Revit are there other CAD style or IFC editors anyone has had luck using?
5. Does IFCtoIDF only deal with building geometry?
6. The rest of the questions are about the utility "IFC HVAC interface."
    6a. Has anyone used this software?
    6b. Does anyone know how/where to get this software?
    6c. Does it work as described (Convert IFC mechanical information into and back out of
         IDF format)?

Answers (from Rob Hitchcock, LBNL)

IFCtoIDF is a utility that imports building geometry from an IFC data file and exports an EnergyPlus IDF with an energy simulation view (i.e., thermal view) of that geometry. IFCtoIDF has been developed using the BSPro middleware for IFC import. We are currently working with the developer of BSPro to iron out problems in IFCtoIDF in using the latest update of BSPro.

Support for IFCs is currently provided in Autodesk Architectural Desktop through G.E.M. Team Solutions GbR and their partner Inopso (see http://www.inopso.com/).

Since IFCtoIDF is designed to work with the IFC data format, independent from the CAD environment that generates an IFC data file, it is intended to work with any IFC data file from any CAD vendor. In reality, this is problematic since different CAD vendors implement their IFC export in different ways. This issue becomes important when you also have to transform what might be called the "architectural view" created in the CAD environment, into a "thermal view" necessary for energy simulation (e.g., subdividing a floor/ceiling slab covering the entire building footprint into surfaces bounding thermal zones above and below the slab).

With this background in mind, I have appended responses from a colleague to your specific questions below.

Answers (from Dr. Vladimir Bazjanac, LBNL)

Q 1: Am I correct in understanding that the IFCtoIDF utility is one directional?
A: Yes.

Q 2: And you cannot convert back to IFC from IDF??
A: Correct - you cannot rewrite building geometry without an appropriate CAD tool.

Q 3: Has anyone had success moving information from Revit to EnergyPlus?
A: Yes, but only on an experimental basis.

Q 4: If not Revit are there other CAD style or IFC editors anyone has had luck using?
A: You need to use a model based CAD tool that can output a valid IFC2x2 file. ArchiCAD 9 is
       one such example.

Q 5: Does IFCtoIDF only deal with building geometry?
A: Yes. Another tool is under development that will do more (e.g. import construction material
       layers defined in CAD).

Q 6: The rest of the questions are about the utility "IFC HVAC interface" to EnergyPlus.

Q 6a: Has anyone used this software?
A: Only in testing.

Q 6b: Does anyone know how/where to get this software?
A: A beta version based on IDD versions 1.1.0 and 1.1.1 is available from LBNL upon request
        – contact this writer (Dr. Vladimir Bazjanac v_bazjanac@lbl.gov).

Q 6c: Does it work as described (Convert IFC mechanical information into and back out of IDF
        format)?
A: It does in tests performed so far.

SOLID ADSORBENTS

In the object Materialproperty:mosisture:EMPD, there are four fields that need to be filled in. They look like fourth-order polynomial coefficients. I checked I/O and Engineering documentation and did not find any detailed information about these except as follows:

Field: Constants to Define Equilibrium Equation
The next four fields describe the sorption isotherm curve used for building materials.

These four fields are dimensionless coefficients.

Where do I find curves for the common Solid Adsorbents?

Answer
The four fields (dimensionless coefficients) that are used in the object Materialproperty:mosisture:EMPD define a relationship between moisture content and relative humidity. Eq. (35) on page 30 in the Engineering Reference (v.1.3) provides the relationship. MositureMaterials.idf has several materials with EMPD properties. If you cannot find the materials you want to use, you may need to find measured sorption data and curve-fit them based on Eq. (35).

PUMP EFFICIENCY

I defined a PUMP:CONSTANT SPEED with a pipe loop. The pump pressure is 10,000 and the motor efficiency is 0.9. Everything sizes OK and I get the result:

Pump power 34.4 W, shaft power 30.9 W, flow 0.2413 kg/s.

The fraction shaft/pump power agrees with the motor efficiency, but I cannot see where the pump efficiency is defined. Using the relationship Power = Pressure x Volume / Efficiency, it seems a pump efficiency of around 0.78 is used.

Answer
From the SizePump subroutine:

```
! note: we assume pump impeller efficiency is 78% for autosizing
TotalEffic = 0.78 * PumpEquip(PumpNum)%MotorEffic
```

THE SLAB PROGRAM

Could someone describe in detail how to use the slab program? I've read the Auxiliary Program Guide, but I still don't understand all the instructions and I can´t make the program run. Thanks.

Answer
1. Open a DOS command prompt window
```
(Start --Programs --Accessories --Command Prompt)
```

2. Change to the directory where EnergyPlus is installed (modify the commands below if you did not install EnergyPlus in the default install path):

```
C:
CD \EnergyPlusV1-2-3-031\
```

3. Change to the ground preprocessor directory:

```
CD PreProcess\GrndTempCalc
```

4. Run the slab example file:

```
runslab SlabExample USA_IL_Chicago-OHare_TMY2
```

We will include these and more instructions in the Auxiliary Program Guide for the next release.

WEATHER – SKY COVER DEFINED

What is the difference between "opaque sky cover" and "total sky cover" ?

Answer
Total sky cover is the amount of sky covered by clouds in 10ths.

Opaque sky cover is the amount of sky covered by opaque clouds in 10ths.

True, they are similar, but you can have sky cover that is not opaque and they can be different.

The following definitions come from the TMY2/IWEC descriptions. The v.1.3.0 release of EnergyPlus includes them under the field definitions:

Field:Total Sky Cover
This is the value for total sky cover (tenths of coverage; i.e., 1 is 1/10 covered. 10 is total coverage. Total sky cover is defined as the amount of sky dome in tenths covered by clouds or obscuring phenomena at the hour indicated at the time indicated. This is not used unless the field for Horizontal Infrared Radiation Intensity is missing and then it is used along with Opaque Sky Cover to calculate Horizontal Infrared Radiation Intensity. Minimum value is 0; maximum value is 10; missing value is 99.

Field:Opaque Sky Cover
This is the value for opaque sky cover (tenths of coverage; i.e., 1 is 1/10 covered. 10 is total coverage. Opaque sky cover is defined as the amount of sky dome in tenths covered by clouds or obscuring phenomena that prevent observing the sky or higher cloud layers at the time indicated. This is not used unless the field for Horizontal Infrared Radiation Intensity is missing and then it is used along with Total Sky Cover to calculate Horizontal Infrared Radiation Intensity. Minimum value is 0; maximum value is 10; missing value is 99.

PARAMETRIC RUNS

How do I create an .imf input file with ##include COUNTER.INC for running a file repeated times?

Answer
Mike Witte created a new example file; it's in the Examples folder in the yahoo group files area. Go to:
http://groups.yahoo.com/group/EnergyPlus_Support/files/Examples/Parametric-5ZoneWarmest.zip
Description: Example input (imf) file and group (epg) file for a series of parametric runs using the group simulation feature in EP-Launch. Can also be accomplished using batch files in a similar fashion.

Background:
When using ##include, be sure you have the included file (COUNTER.INC) in the correct folder (where you run EnergyPlus) or give the full path for the included file, i.e., ##include C:\\xxx\xxx\COUNTER.INC. Otherwise, EnergyPlus won't be able to find the included file. Also, note that ##include does not work with spaces in the path. If you are not aware of the epmdet output file, this shows the full details of the ep-macro processing. Any macro errors will be in this file. We assume you are using the EP-Launch group feature to do this. Make sure that your master input file has an .imf exension. If it does not, then the ## macro commands will not be processed.

GROUND TEMPERATURES

Can you please tell me how (and where) to get the value of ground temperatures for a location?

Answer
The EPW weather files contain the average monthly ground temperatures for 5 ft (1.33 m) below grade calculated using a simple equation developed by Tom Kusuda at the US National Bureau of Standards (now NIST) back in the early 1970's. The constant deep ground temperature, in the absence of geothermal anomalies, is the same as the average annual air temperature plus 1 C. It is true that the statistics file (.stat), distributed with the weather data, lists such ground temperatures, but these are open field ground temperatures. They are useful as inputs to ground heat exchangers or ponds, but not under the floor of a conditioned building. EnergyPlus has a 3-D heat transfer preprocessor that will calculate monthly average ground temperatures for slabs and basements (most accurate approach). Please read the Input Output/Reference (ground temperatures) and Auxiliary Programs (ground heat transfer) documents for more details and a discussion of reasonable default values to use.

DOUBLE FAÇADE

I need to model double façades. In the example files, I found doublefaçade.idf for a one-story building. However, I need to model an eight-story building and don't know how to model the air passing through the floor and ceiling in the double façade zone. Also, how do I specify steel grating in the double façade?

Answer
If the double-façade space is open from bottom to top, then it could be modeled as one thermal zone (one outside surface) that is eight stories high. The inside wall would be eight surfaces that are stacked on top of each other (assuming that this interacts with eight different interior zones). If you have to divide the double-façade space into multiple thermal zones, then you would use either MIXING or CROSS MIXING for simple specified airflows between the zones. Or a full airflow network could also be described using the COMIS (Air Flow Network in EnergyPlus 1.3.0 and subsequent versions) model instead of the simple mixing and cross mixing.

If the steel grating is meant to simply shade some of the solar gains, then it could be modeled as an interior or exterior shade or blind.

ROOF MODEL

Can a roof spanning over several zones be modeled as one big, rectangular heat transfer surface?

Answer
It needs to be divided into surfaces covering each zone, so break it up by zones.

Question
The roof has sizable overhangs on all sides. According to the EnergyPlus documentation, roof extensions/overhangs do not need to be modeled as separate shading surfaces. If I understand correctly, EnergyPlus determines the portion that extends beyond the walls (using entered wall and roof vertices) to calculate shading effects. Since the roof will be divided into sections covering several zones, will an overhang associated with a particular roof section be limited to shading only the wall(s) of the zone to which it belongs? Or, depending on the Sun's position, can it also shade the walls of other zones?

Answer
No! Surfaces cast shadows only in the direction that they face. A roof faces upward and does not cast shadows downward. If you use FullExterior as the shadowing option (in the BUILDING object), then shadows cross zone boundaries, that is automatic.

For roofs that extend beyond the walls, there are two issues to consider: (1) how much of the roof area transfers heat into the building? and (2) how to model shading impacts.

In the case of a gable roof with an attic, the full roof area transfers heat into the attic zone, and there need to be additional heat transfer surfaces (facing downward) to model the soffit heat transfer.

```
  /    ← ← ← Roof heat transfer surface, sloped, facing upward
 /
/___   ← ← ← Soffit heat transfer surface, horizontal, facing downward
   !
   !
   !
```

In the case where the roof extension does not transfer heat, the extra roof area must be a shading surface. Shading surfaces are mirrored automatically, so they *do* cast shadows in all directions. In this case, extending the roof heat transfer surface area would also overestimate the heat transfer through the roof.

```
   /
  /   ← ← ←  Roof heat transfer surface, sloped, facing upward
 /!
/ !   ← ← ←  Overhang shading surface, sloped, facing up or down, mirrored
  !
```

```
    !
    !


_____ _____      ← ← ← The right-hand portion is the heat transfer
surface
     !                          The left-hand portion is the shading surface
     !
     !    Zone Interior
     !
```

PARAMETRIC RUNS

I am trying to run a very simple macro. I want to generate three files with three different values for the "complex field 4" of a SCHEDULE:COMPACT object, but my counter won't work.

I set a group simulation and used the "user defined location" to set the number of repeat simulations to three. EnergyPlus runs three times but keeps assigning the same value to my variable. I think there's something wrong with my syntax but I don't know how to fix this. I'm using version 1.3.

Answer
For additional guidance, download the example file (Parametric-5ZoneWarmest.zip) in the Files\Example folder on the EnergyPlus support site.

EXHAUST FROM UNCONDITIONED SPACES

I am trying to simulate a "semi open" system where air is extracted from a bathroom that has no direct connection with the primary air loop. My first instinct was to use the "Zone Exhaust Fan" object between the subject zone and an outside node, while making up the extracted volume of air using the "Mixing" object between the unconditioned bathroom and an adjacent zone. How do I define a Node representing the bathroom space (inlet to the exhaust fan)? Is there such thing as an "open-loop" air system in EnergyPlus?

Answer
The simplest approach would be to combine the bathroom space into the same thermal zone as the adjacent space.

If you need to model the bathroom as a separate thermal zone, then what you suggest is the best approach. For the bathroom zone, use a CONTROLLED ZONE EQUIP CONFIGURATION object to define a zone node and an exhaust node and list the exhaust fan as the only piece of equipment in the ZONE EQUIPMENT LIST.
Note that EnergyPlus has no way to do an air balance on this, so the simple zone with exhaust fan will see no impact from the exhaust fan operation other than electric power consumption.

The MIXING object will impact the bathroom zone by providing air from the adjacent zone. But MIXING has no impact on its source zone; you must allow for this by adding INFILTRATION to the source zone or by supplying outside air in the air loop that serves the source zone.

If you are modeling heat recovery, or anything else where the return air flow rate is important, then you will also need an exhaust fan in the adjacent zone (with zero power consumption) to reduce the return air flow from that zone.

POLYGONS

How do I define the vertices of a rectangle when the EnergyPlus documentation states:
   "define all roofs and floors as rectangles regardless of the shape of the zone."

Answer
With the advent of >4 sided polygons, you no longer have to limit yourself to 4-sided/rectangular floors/roofs though it is perfectly correct (reference example file Flr_Rf_8Sides.idf for a floor and roof with 8 sides covering an L-Shaped zone). This is the only example file with >4 sided polygons that comes with EnergyPlus so any of the others will (usually) illustrate making a rectangular floor or roof (ZoneAirCooled.idf, for example).

MINIMUM HUMIDITY CONTROL

I want to control minimum relative humidity in a zone and have added the following to an air handling system generated using COMPACT HVAC objects THERMOSTAT, ZONE:UNITARY, and SYSTEM:UNITARY. This is what I've done:

Humidistat controlling RH in Zone 1 according to a specified RH schedule.
Humidifier located downstream of the AHU "Cooling Coil" and upstream of the "Zone Splitter." The inlet and outlet nodes of the humidifier are "Air Loop Outlet" and "Supply Path Inlet" and both nodes were generated using the COMPACT:HVAC objects.

A "Setpoint Manager: Single Zone Min Hum" object with "AHU 1 Supply Path Inlet" as the SetPoint Node and "ZONE 1 ZONE AIR NODE" as the name of the Control Zone Node. Note that the latter field is called "Name of a Node list (or Node)" in the IDF, while the Input/Output Reference describes it as the "name of the zone node of the humidity control zone." Input was based on the description provided in the Input/Output Reference.

The above modifications made no difference to the results. The minimum RH specified in the humidistat RH schedule has not been met. Inspection of the "eplusout.bnd" indicates that there is no humidifier in the system.

Is it possible to add a component, such as a humidifier, to an air system created using COMPACT:HVAC objects?
How is the signal from the Humidistat used by the "Setpoint Manager" or the "Humidifier"?
I do not see a call/reference to the humidistat from either of these objects.
Do I need to add other objects besides those described above to simulate minimum humidity control?

Answer
Run the base input file that contains the COMPACT HVAC objects.

Copy the resulting .expidf output file to a new *name*.idf. (The expidf file is the resulting input file in which the COMPACT HVAC objects have been replaced with sets of basic EnergyPlus HVAC objects.)

Add the humidifier and related objects to this input file. Note that you must add the humdifier to the BRANCH object for the air loop, and you must change the outlet node name of the preceding component, or the inlet node name of the component after the humidifier in order for this to flow properly.

COORDINATE SYSTEMS

In a relative coordinate system, are the coordinates of surface and sub-surface vertices entered relative to the BUILDING ORIGIN or to the applicable ZONE ORIGIN?

Answer
No Building Origin is ever entered; relative coordinates are relative to the Zone Origin, if input. Also, the relative coordinate system does use both Building and North Axes; see the Input/Output Reference Manual, p. 128 /pdf p. 164).

Question
Is the North axis shown on the DXF file True North or Building North?

Answer
The arrow on the DXF outputs are annotated "True North."

Question
By default, the Blast Translator converts models into the absolute coordinate system. Is there a way of retaining the relative coordinate system when converting BLAST models?

Answer
No

MASS FLOW EQUATION ERROR

Usually the unit of Effective Leakage Area (ELA) is cm^2 in SI. However, in EnergyPlus the AirflowNetwork ELA is listed as m^2/m^2, which is the ratio of the leakage area of the associated surface.And when compared to the equation in the 2005 ASHRAE Handbook of Fundamentals, p. 27.13, there is no figure on the density of air. Is the EnergyPlus equation correct?

Answer
There are two mistakes in object AirflowNetwork:Multizone:Surface Effective Leakage Area in the Input/Output Reference on page 601. The first mistake occurs in the equation where the square root of air density is missed. The correction equation should be:

$$m = ELA*Cd*(2*Air\ Density)^{0.5}*(dP_r)^{(0.5-n)}*(dP)^n$$

The second mistake is the unit of effective leakage area (Field:Effective Leakage Area) on the next page. The correct unit should be m^2, instead of m^2/m^2. The program uses m^2.

There is a detailed description in the NIST publication on page 144 of the *CONTAMW 2.0 User Manual* by W. S. Dols and George Walton, NISTIR-6921, 2002.
http://fire.nist.gov/bfrlpubs/build02/art178.html

We will correct the mistakes in both Input/Output Reference and EnergyPlus .idd. And thank you for bringing this error to our attention.

MAXIMUM HUMIDITY CONTROL

I get this message when I use the "Max Humidity Control" option with the object "Furnace:BlowThru:HeatCool."

```
** Severe  ** Node Connection Error, Node="ZONE 2  ZONE AIR NODE",
Sensor node did not find a matching node of appropriate type.
**   ~~~   ** Reference Object=SET POINT MANAGER:SINGLE ZONE MAX HUM,
Name=ZONE2MAXRH
```

The goal of the controller is met (RH in Zone 2 does not exceed 50%) despite the node connection error! I really would like to know the meaning/implications of the error.

Answer
There are two node names requested in the Set Point Manager:Single Zone Max Hum object. Both of these nodes must be defined elsewhere in the EnergyPlus input.

The Name of the set point Node or Node list (A4): The node name or each node name in the node list should be a valid node in the air loop (this name is used by other air loop objects; for example, the outlet node name of the cooling coil).

The Control zone air node name (A5): This is the zone name where the humidity is to be controlled. This node name must be listed in a Controlled Zone Equip Configuration object.

Inputs for Control Variable (A2) and Schedule Name (A3) will no longer be needed in future versions of EnergyPlus.

If this does not help, check the spelling of all node names referenced by the Set Point Manager:Single Zone Max Hum object and make sure they show up elsewhere in the input and in the appropriate places.

```
SET POINT MANAGER:SINGLE ZONE MAX HUM,
      \min-fields 5
  A1 , \field Name
      \required-field
      \reference SetPointManagers
  A2 , \field Control variable
      \note Deprecated Field.  This field is not used
  A3 , \field Schedule name
      \note Deprecated Field.  This field is not used
  A4 , \field Name of the set point Node or Node List
      \required-field
      \note Node at which humidity ratio set point will be set
  A5 ; \field Control zone air node name
      \required-field
      \note Name of the zone air node for the humidity control zone
```

AUTOSIZING – HEAT RECOVERY LOOP

If I want to autosize the heat recovery loop, which type of loop should be input in the plant-sizing field? I used a water heater to recover the exhaust heat from an IC engine and supply hot water to the HVAC terminals. There is no node-branch connection error in my idf file, and I do get the correct svg file. But the pump in the heat recovery loop does not work; there is no flow getting through the pump. Therefore, the water heater uses the electric heater to keep the water temperature in the allowed range. I have compared my file with the example file "HeatRecoveryPlantLoop.idf." The setpoint manager and the schedules are the same. The only difference is that my file autosizes the heat recovery loop, and the example file does not. So I wonder whether if it is even possible to autosize the heat recovery loop?

Answer
Check the component sizing reports in the eio output file. It's possible that the flow rate in the heat recovery loop is sizing to zero, which explains why there is no flow. There is a known problem related to plant loop autosizing when the component flow rates are not autosize-able. This is part of the problem you are having. You will not be able to autosize the flow rate of the heat recovery loop.

AUTOSIZING – COIL VALUE

Whenever I ask EnergyPlus to autosize the SHR for a coil (DX:CoolingByPassFactorEmpirical) it always returns a value of about 0.64. This is the case whether the room has a high or low sensible to latent load (e.g., meeting room vs. computer room for a given size). What factors or inputs determine the SHR? Should it be changing depending on the ratio of latent to sensible for given room size?

Answer
The EnergyPlus autosizing calculations are currently driven by sensible load requirements (which establish the air volume flow rates) and the user-specified supply air humidity ratios (in the SYSTEM SIZING object in this case). Note that the DX coil SHR is at ARI rated conditions, so the autosized SHR uses the ARI rated entering conditions and the user-specified leaving conditions. You should see a change if you alter the cooling supply humidity ratio in the SYSTEM SIZING object.

FAN COIL, AUXILIARY AIR

Is it possible to configure a primary air loop to supplement a set amount of auxiliary air AND have a Fan Coil unit provide the additional tempering of the zone conditions?

Answer
Yes, that will work. However, make sure you set the heating and cooling priority of the fan coil unit to 2 (in the Zone Equipment List for each controlled zone) so that it runs after the air from the air loop has been supplied, in case that air changes the net load.

PURCHASED AIR LATENT LOAD

I am using purchased air in a building. In the output, when I looked at the sensible and total loads, I realized that, in some zones, the sensible is higher than the total load. Does this mean that I have a negative latent load?

Answer
The purchased air model is very simple. Using the specified supply air temperature, it determines the flow rate required to meet the current zone load. It takes air from the zone, adds outside air if specified (use this with caution), and then provides the calculated flow rate of air at the specified supply temperature and humidity. To determine the total load, the change in enthalpy is used. To determine the sensible load, the change in temperature is used. If the change in humidity is in the opposite direction of the change in temperature, then you will see the condition where sensible is greater than total. Purchased air is an excellent model for calculating sensible loads. The latent loads calculated by purchased air are very approximate and are somewhat arbitrary given that the supply air humidity ratio is specified directly rather than computed as part of a system model.

ONE ZONE, TWO FLOORS

Is it possible for one zone to have two floors? I would like to split the floor of a zone into two parts, one exposed to air and the other adjacent to a lower floor.

Answer
Yes, you can have as many floors (and roofs and walls) as necessary to define the boundaries of the zone. Even if the outside environment is the same, it is sometimes easier to describe certain shapes by breaking them into two parts.

DXF VIEWER == GOOGLE'S SKETCHUP

Google's "SKETCHUP" -- A New Way to View DXF* Output!
A TIP FROM THE ENERGYPLUS TEAM

<u>Brent Griffith (NREL)</u>
Google SketchUp is a really great tool for viewing the DXF output files from EnergyPlus.
The software is free and can be downloaded from <u>http://www.sketchup.com/</u>. After using VoloView for so long, I was blown away by how much better the models look in SketchUP.
Be sure to check out the X-ray view.

<u>Dru Crawley (USDOE)</u>
Not only that, but you can change the windows to transparent and put materials on the rest... make it look like a real building.
*DXF = drawing file in Autocad Format

UPSTREAM COMPONENTS -- MIXER

I want to recool/reheat the outside air (OA) to the specified setpoint temperature (usually dewpoint temperature for dehumidification in summer). I added a cooling coil and a heating coil upstream from the outside air mixer. The outside air flow is as follows: Outside Air → Cooling Coil → Heating Coil → OA mixers. For OA temperature control (inlet node of the OA mixer), I added two simple controllers to control the cooling/heating coil. After the simulation ran I got no severe errors but I was left with an orphaned branch. I checked the values and found that the outside air VolFlowRate was 0, the OA reheat/reheat coil were not sized in the eio file, and the water inlet MassFlowRate was 0.1. According to the Input/Output Reference, adding an upstream component for the mixer is allowed. Is my setup incorrect?

Answer
Sorry, but at the present time EnergyPlus cannot do what you ask. The Controller:Simple will only function in the main air loop; it does not work in the outside air system. Components in the outside air stream must have their own built-in controllers in order to control to a setpoint. Chilled and hot water coils do not have built-in control.

Question
Thanks for the clarification. So which components may be added upstream of the mixer for reheat/recool?

Answer
Right now, the only ones available to you in the outside air system are:

- COIL:Gas:Heating
- COIL:Electric:Heating
- DXSystem:AirLoop

You'll have to check the Input/Output Reference for input details. In each case you need to specify the setpoint node.

WATERHEATER:MIXED

I am using a water heater to recover the heat from an IC engine. In the winter, the water heater will supply hot water to the fan coil unit for heating. But I want the heater to supply domestic water during the summer rather than supply hot water to the fan coil unit. How can set up the water heater for the purpose? And how can I make the heater use the exhausted heat from the engine?

Answer
For this system you'll have to set up two PLANT LOOPS connected together by the Water Heater:Mixed object. One loop, for heat recovery, will add heat to the Water Heater on its source side; the other loop, for hot water, will serve the coil and domestic water. It might help you to study the plant-related input objects in the example file SolarCollectorFlatPlateWater.idf, then swap out your IC engine for the solar panels. On the demand side of the hot water plant loop, use a splitter/mixer to put the hot water coil and the Domestic Hot Water on parallel branches . The water heater object can be used to simulate domestic hot water usage by adding a DOMESTIC HOT WATER object to your plant loop. Make sure to put the DOMESTIC HOT WATER object in a branch that is parallel to your hot water fan coils, not in series.

AIRFLOW VENTILATION OBJECTS

I'm running an idf file with no AirFlow ventilation objects. In the same file I have a ventilation object set to 0 m^3/s (no ventilation). Yet, I get different results, does that make sense?

Answer
If you have any "simple" airflow objects in the original input file, such as INFILTRATION, VENTILATION, MIXING and CROSS MIXING, they may be turned off by the "AirflowNetwork Control" field in the AIRFLOWNETWORK SIMULATION object.

Question
In the original idf there is a "simple" ventilation. And I understand that you are giving me a way to stop the ventilation. But does that means that an idf file with no ventilation object is incorrect or is not equivalent to a building with no ventilation?

Answer
No. The reason that the AIRFLOWNETWORK SIMULATION object has a switch to turn off simple VENTILATION and INFILTRATION is to make it convenient for a user to compare "simple" against AirflowNetwork.

INITIAL CONDITIONS

What are the initial conditions assumed in each EnergyPlus simulation? Can the user define some or all initial conditions? For example, recovery from a temporary loss of HVAC equipment? My search for information on the subject, in both the user documentation and support database, was not successful.

Answer
There is no direct user control over initial conditions. Here is a summary of the initializations which are performed at the beginning of each environment (an environment is RunPeriod or a DesignDay):

1. All building heat transfer related temperatures are initialized to 23C.

2. All HVAC system nodes are initialized to 20C and the outdoor humidity ratio of the first time step of weather data.

3. The first day of the environment is repeated until the changes in zone loads and zone and surface temperatures are less than the tolerances specified in the BUILDING object. If convergence is not reached within the "Maximum Number of Warmup Days" (which defaults to 25 days, see the BUILDING object), then a warning is issued "Loads Initialization did not Converge."

The only way for the user to control initialization would be to customize a weather data file with an initial time period long enough to establish the desired initial conditions and then have the weather data proceed into the period of interest.

SURFACE OUTSIDE TEMPERATURE (ROOF)

I'm modeling a building where the surface outside temperature of the roof reaches 73C at noon.
However, I'm not sure if this temperature is reasonable. The direct solar at noon (10:00-14:00) is about 760-860 W/m^2. Listed below are the settings of the outside layers of the roof in the simulation.

```
MATERIAL:REGULAR,
R006-1, !- Name
Rough, !- Roughness
0.1, !- Thickness {m}
0.17, !- Conductivity {W/m-K}
600, !- Density {kg/m3}
1600, !- Specific Heat {J/kg-K}
0.9, !- Absorptance:Thermal
0.8, !- Absorptance:Solar
0.7; !- Absorptance:Visible
```

I know that two main factors, conductivity and absorptance solar, will affect the surface outside temperature. Are there any other factors I should take into consideration?

Answer
Two other significant factors are thermal absorptance (emissivity) and the exterior convection coefficient (which is driven primarily by wind speed). And, of course, the thermal conductivity of the other layers in the roof construction, the outdoor air temperature and the temperature of the thermal zone on the inside of the roof. Note that 73C is not unreasonable on a hot sunny day.

BLOWTHRU:HEATCOOL

What is difference between these:
FURNACE:BLOWTHRU:HEATCOOL and UNITARYSYS:BLOWTHRU:HEATCOOL?

Answer
To quote Edwin Starr, from his classic song War* . . . *Absolutely Nothing*! Of course, you could also refer to the Input/Output Reference, p. 732 under UnitarySystem: BlowThru:HeatCool, where it states:
The UnitarySystem:BlowThru:HeatCool is the identical model to the Furnace: BlowThru: HeatCool object.
* *http://www.oldielyrics.com/lyrics/edwin_starr/war.html*

FURNITURE MODEL

How do I model the percentage of floor covered by furniture in a zone? I know I can use internal mass, but is there a way to account for the surface of the floor being reduced?

Answer
There isn't a requirement that the floor surface(s) cover the entire area of the zone, so if you want you can just shrink the floor surface to remove area. The entire floor can be broken up into different surface objects and different Constructions used for different portions. If the furniture is in very close thermal contact with the floor (no gaps or air flow underneath), then you could model the presence of furniture by adding additional material layer(s) to the Construction.

HEAT PUMP EQUATIONS -- EQUATIONFIT

What numerical formulas use Capacity Coefficient1-5 and Power Consumption Coefficient1-5?

I could not find the equations in Input/Output Reference.

Answer
You need to look in the Engineering Reference, not the Input/Output Reference. The equations for the EquationFit model for water-to-water Heat Pump are explained in Engineering Reference (p 530, search for "water-to-water heat pump").

AUTOSIZING ERRORS

In order to serve one room, I set up a fan coil unit accompanied by a direct air object that is, in turn, served by an air-to-air heat pump. Since I lack of the performance data on both units, I chose to autosize these two objects at one time. I set the fan coil as the highest priority for air conditioning, and the direct air as an assist. The node-branch connection passed the test; however, the design load for heating coil became 0, and the leaving humidity of the cooling coil was larger than the entering humidity. For the heat pump, the air volume flow rate per watt of rated total heating capacity is out of range. How, can I fix them? Should I set up the parameters for the fan coil unit, and then autosize the heat pump alone? And does the configuration of the zone equipment list set the priorities of the equipment?

Answer
EnergyPlus autosizing is not able to split the load among multiple units serving the same zone.
It will attempt to size both units to meet the entire heating and cooling load. The zone equipment list controls the sequence in which multiple units are simulated to meet the current zone load. As a starting point, you should be able to autosize both units to meet the full load. Then you can decide what portion of the loads should be met by each unit and then specify hard sizes for everything based upon the sizing outputs in the eio file. If the design load for the heating coil is zero, then there is something wrong in one of the sizing objects. Check the thermostat, zone sizing, system sizing, and plant sizing inputs.

STRATIFICATION, DISPLACEMENT VENTILATION

I am attempting to compare various indirect evaporative cooling systems in a large production hall with displacement ventilation. In some configurations, the secondary air flow is extracted near the ceiling and I assume that the effects of stratification would be significant. So far I have considered coupling air flow with multiple zones (stacked) or defining the plenum space or multiple nodes into a single zone. Each possibility seems feasible, but I lack the experience to know which assumptions are necessary to obtain realistic results. The simulation should be on an hourly basis (at least). Can you give me some pointers on how to define such a space?

If you are able to characterize the stratification in the hall by some other means, you could model the implications of that temperature distribution in EnergyPlus using User Defined Room Air models. Various temperature patterns can be entered. EnergyPlus also has a displacement ventilation room air model but it may not be applicable to your space.

STANDARD BUILDING MODELS

I'm trying to find standard/generic building models that can be used for retrofit evaluations. I am working on a project where we are testing various new ventilation technologies in several climate zones. The technologies are to be evaluated in several generic buildings – a hotel, school, hospital and office building – to see the effect of the different climates. Has anyone constructed hypothetical models that fall into these categories?

Answer (from Drury Crawley, USDOE)
You can get simulation inputs for different building types by using the EnergyPlus Example File Generator. It is tied to a version of ASHRAE 90.1 (you input the size). Follow this link
http://www.eere.energy.gov/buildings/energyplus/interfaces_tools.html

USDOE is creating prototypes based on the 2003 CBECS (Joe Huang is updating work he did based on 1995 CBECS). We hope to make those available for use this Fall or early Winter.
(We will announce their availability on the various email lists.)

STANDARD VS DAYLIGHT SAVINGS TIME

I created a weather file with daylight saving time. However, Energy Plus calculated the solar altitude angle for standard time. That doesn't match the weather parameters with solar altitude and irradiation. How do I make EnergyPlus calculate the simulation using daylight saving time?

Answer
The reported times in the csv output files are always in standard time. Daylight savings shifts schedule values by 1 hour when daylight savings is active. Add REPORT VARIABLE "Daylight Saving Time Indicator" to your output to confirm operation of daylight savings. If it is always zero, then check that the daylight savings option in the RunPeriod object is set and that the specifications in the weather file are correct. When the daylight saving time indicator is 1, you should see a shift in scheduled values such as internal loads profiles and thermostat settings.

OTHERSIDECOEFFICIENTS OBJECT, WINDOWS

I was trying to set up the outside surface temperature for a surface that has a window (and the same temperature for the window) and I get the following error. What am I doing wrong?

```
  ** Severe **
GetHTSubSurfaceData: Other side coefficients are not allowed with windows.
Surface=SURF-7:ROOM ** Fatal ** Fatal error discovered in GetSurfaceData, see
previous messages
```

I'm confused because the documentation states that we are allowed to use the "OtherSideCoefficients" object for windows.

Answer
Other side coefficients can <u>only</u> be used on opaque surfaces. They will not work for windows, glass doors, and tubular daylighting devices. We will make this more clear in the documentation.

WEATHER, DESIGN-DAY

I ran several simulations, each time with same input file (purchased air tables) but different weather files (the weather files that come with EnergyPlus installation). The outdoor dry bulb temperatures of design-days are same in each case (each simulation run), no matter what the weather file is. Why are the design-day outdoor dry bulb temperatures the same even if I keep changing the weather file?

Answer
The answer might depend on how the RUN CONTROL object is specified in your input file.

If you have "Do the design-day simulations" = YES, and "Do the weather file simulation" = NO, then the weather file is not being used; only the design-day information/objects in the IDF would be used in that case.

For the input deck in question (PurchAirTables.idf), the Run Control object is set to run the winter and summer design-days, followed by an annual simulation (Jan 1 through Dec 31).

For design-day simulations, the weather information provided in the DesignDay object(s) will always be used, independent of the weather file being passed to the simulation (in.epw). See the "DesignDay" object in the Input/Output Reference for further discussion.

In contrast, the "weather file simulation" (annual simulation in this case), which is run after the two design-days in this particular instance, will use the information in the weather file (in.epw).

And note that the design-days are actually in the input file. Changing the weather file won't affect the design-day input. To do that you need to replace the design-day objects in your input file (because the design-days have been embedded in your input file). If you need the design-days that "go" with the weather files, you will need to download those separately. And as locations change, you would probably notice different sunrise/sunset times.

STORY MULTIPLIER

I am a new user of EnergyPlus and I have defined a simple 5-zone office building level with both glazing and shading on all sides. I want to make this 1-story model into a 10-story model and I was wondering if there is a way of copying this level and multiplying the base story, rather than having to copy and paste it 10 times and replace the Z geometry for all zones manually?

Short Answer
You can use the Zone List and Zone Groups to achieve this. Typically you will need to do another floor for the top story, though. Check out the example file: MultiStory.idf

Longer Answer (from Brent Griffith)
There are few ways you can deal with this.

I use zone multipliers for buildings with 4 or more floors for faster run times. The simplest multiplier is in the Zone object, just multiply each zone by a factor of 10. In EnergyPlus, you can also group zones into a list and multiply that list - usually used as a floor multiplier. See the tips for multistory simulations in the Zone Group object description in the Input/Output Reference Manual.

If you want to see an example input file of a model for 10-story building, go to the new EnergyPlus Example File Generator and request an input deck with 10 floors. (These input files have three floors with a single ground floor, a single top floor, and a middle floor with multipliers.)

If you really want your model to have all the surfaces for 10 floors, then it is easier to use zone relative coordinates; you copy and paste the geometry without having to alter the geometric values in the vertices -- just shift the zone origins. You still have to work out unique names and the other side surfaces, but be aware that you'll end up with some pretty long simulation run times going this way.

BUILDING GEOMETRY

I want to use one of the front ends for EnergyPlus to define building geometry. Which one is best?

Answer
There are a number of tools that will give you accurate geometry. It depends on whether you want a 'full' interface or something to generate input files. DesignBuilder and Hevacomp both are full interfaces and allow you to review output as well. E2AC creates simple building cases and allows you to quickly compare options. EP-Quick gives you a lot of flexibility in defining the geometry but does not include HVAC (yet). EFEN allows you to focus on windows and fenestration--defaulting much of the building description.

If you have CAD, there are two options--IFCtoIDF and Green Building Studio. Also, ESP-r and ECOTECT can export EnergyPlus input files.

There is also our web interface, which consists of rectangular buildings with defaults based on building code and building type (See the EnergyPlus Example File Generator).

More info and links to all of these are available at:

VENTED PV CAVITY

I am modeling a building with integrated photovolatic (BIPV) system for the building façade. The photovoltaic (PV) panels are attached to the exterior wall with an air cavity between them. I want the PV panels to serve as a solar heat barrier but experienced problems with heat gain on the façade. Then, by using "OtherSideConditionsModel," I applied "Vented PV Cavity" as the type of modeling used to determine boundary conditions. I used the "SURFACE:HEAT TRANSFER:EXTERIOR NATURAL VENTED CAVITY" object to describe the condition. However, there was no heat gain reduction for the underlying surface (i.e., the exterior wall), even though the required cooling energy for the corresponding zone is higher. Any advice would be appreciated.

Answer
First describe the building in more detail.

Reply
It's a 3-story building. The west-facing façades on the second and third floors are attached with vented PV panels. The surface material of the exterior wall is "glass." You can see from the files I uploaded the details, highlighted in green, in "construction_material (IDF file).doc."

Answer
I took a look at your files. I didn't succeed at running it because it is for EnergyPlus version 1.2.2.
I would recommend moving to the current EnergyPlus version 1.3.

You have set up the PV modeling correctly. The basic wall construction here is un-insulated heavy concrete. Adding the PV here is increasing the wall's solar absorptivity from 0.7 to 0.97 so this could be why the cooling loads go up. This could also be one of those non-intuitive situations where adding the first bit of insulation can actually increase cooling loads. Remember that the added PV layer will also trap more heat at night and disconnect the underlying surface from cool night sky. You might try adding a different insulating layer to test expectations.

When using the external cavity model, the Construction should describe only the underlying surface and not the air gap and actual outer layer. The models will add these to the outside of the Construction. So in your file, the walls with PV actually have two layers of glass and air gap. The Material:Air object isn't really needed for the SURFACE:HEAT TRANSFER: EXTERIOR NATURAL VENTED CAVITY object (unless you really want double layers of glass and air gaps). Your value for the height scale for buoyancy looks like the one for horizontal orientation; for vertical you might increase this one-quarter or one-half of the PV module height.

Question
Thanks for your advice. And I will convert the files to EnergyPlus 1.3.0 after I solve my problem.

I have tried using other insulating materials instead of the heavy concrete. In addition, I have also tried increasing the height scale for buoyancy, effectiveness for perforations with respect to Wind (Cv) and discharge coefficient for openings with respect to bouyancy (Cd). However, the resulting annual electricity consumption was not improved. The annual cooling electricity is almost the same as the case without vented BIPV. I think attaching the exterior naturally vented BIPV can reduce the annual cooling electricity. Besides, better PV efficiency can be achieved.

Will you review the file I uploaded? I am not sure I set the height scale for buoyancy, Cv and Cd correctly. You can also see my exterior wall construction. Is the "SURFACE:HEAT TRANSFER:EXTERIOR NATURAL VENTED CAVITY" object preferable for vertically mounted BIPV?
I am also interested in "CONSTRUCTION WITH INTERNAL SOURCE" for PVSpandrlPanel. But, will the efficiency be lower by using this model because of higher temperature behind the PV panel?

Answer
We reviewed your files and didn't see a problem.

For vertical BIPV, you have two options in EnergyPlus.

The first, <u>construction with internal source</u> is a more robust model but it is only for cases where there is no ventilation on the back side.

The second, <u>external cavity model</u>, allows modeling the effect of ventilation on the back side, but experimental research is still needed to have a better understanding of the model inputs for Cd, Cv, open area, etc.

The Sandia PV model can respond to the module temperature in both cases, but the changes in efficiency turn out to be pretty small most of the time. You could model both ways and compare to see the potential improvements from ventilation.

And it is difficult to confirm your observation that "resulting annual electricity consumption is not improved." I would need to examine complete input files for both cases (after macro processing please). Just guessing, I would say the vertical walls are not a big part of the cooling load compared to the windows, roof, and internal gains and anything you do to them won't make much difference. I would compare the two runs by plotting the inside face surface temperatures for the wall for at least 24 hours.

PLANT: SEQUENTIAL VS OPTIMAL

For cooling plant equipment, I am using purchased chilled water with a constant COP chiller. When the cooling load is less than 10kW, I want the purchased chilled water; when the demand is larger than 10kW, I want the constant COP chiller to supply additional chilled water (with the purchased chilled water still working). I set the purchased chilled water as the primary equipment in the plant equipment list, with the chiller as secondary. However, the chiller supplied much more chilled water (during the yearly simulation) when the load was larger than 10kW. In most cases, the chiller was working at its rated capacity. Because the flow rate of the chiller is variable, and the load distribution scheme is optimal, I don't know how to control the chiller now.

Answer
If I understand the question correctly, you should set the load distribution option to be SEQUENTIAL, not OPTIMAL. SEQUENTIAL runs the first piece of equipment until it is fully loaded, then activates the next piece of equipment and so on.

PLANT DEMAND ERROR

I am using two fan coil units to serve two zones; they are supplied by purchased chilled water and a constant COP. The output of the design-day simulation shows that the cooling demand of the two fan coil units is less than 25kW, but the plant cooling demand is around 35kW. There are only two fan coil units in the cooling water loop. Are these results correct?

Answer
We reviewed the file you uploaded. In the input file, the only report variable at the zone level is "Zone/Sys Sensible Cooling Rate." This does not include latent cooling load, and it is the net cooling provided to the space. The chilled water coil in the fan coil unit must offset the fan heat and the chillers/purchased cooling must offset pump heat in addition to that. To follow this, you also need to report the cooling coil total load, pump heat that is added to the fluid.

Also, note that "Plant Loop Cooling Demand" represents the cooling required to bring the plant loop to its setpoint. If for any reason, the loop is not holding setpoint, this value will not match the current loads on the loop. It is best to compare the coil load plus pump heat with the chiller evaporative heat transfer rate plus purchased cooling rate.

Question
I checked the coil total cooling load. The sum of the two coils is less than 27kW; however, the plant cooling demand is still 35kW. In the csv file, the sum of the flow rate through the two coils was equal to that of the chiller. And the temperature difference of the coil inlet and outlet node is almost the same of the chiller. I used the formula Q=Cp*Mdot*DeltaT to calculate the heat transferred. The heat transferred from the chiller corresponded to the plant cooling demand; however, the heat transferred in the two coils corresponded to the zone cooling demand. And it is 10kW bigger than the sum of the coil total cooling load from the csv file. It's hard to believe that the pump and fan can add 10kW cooling demand to the loop. Can I just increase the setpoint of the cooling loop to fix this problem? The coil cooling demand and the loop cooling demand was matched before the domestic water object was added; did it influence the loop cooling demand? The domestic water is operating when the coil are supplying chilled air to the zones.

Answer
Well, not exactly. The sum of "Fan Coil Total Cooling Rate[W](Hourly)" is approximately 27kW. But the sum of "Total Water Cooling Coil Rate[W](Hourly)" is approximately 35kW, and it exactly matches the plant loop cooling demand each hour.

What's the difference? See the definition of "Fan Coil Total Cooling Rate" in the Input/Output Reference. It is the cooling delivered to the zone.

"Total Water Cooling Coil Rate" is the coil load. The difference is the outside air load.

And domestic hot water can certainly impact the plant loop. When hot water is "used" it is replaced by "new" water that can be colder. You can set the temperature of this new mains water in different ways.

ATRIUM SIMULATION

I am simulating a building with a huge glass-roofed atrium. On the two opposite sides of the atrium are floors for offices; I need to model the office and atrium zones that are adjacent to each other.

Do I have to define the inter-zone wall and its associated windows twice for two adjacent zones? If I understand EnergyPlus correctly, each zone needs to be defined with a complete set of surfaces (say, 6) surrounding it. True?

Answer

Zones do not necessarily require a complete enclosure. If you want sunlight and heat to be transferred between the atrium zone and the office zones, then you will need inter-zone walls and windows to model that transfer. And yes, the inter-zone walls and windows are each described twice, once for each zone, facing in opposite directions. It is critical to get the facing directions correct, or the window will not see sunlight.

Question

Also, other than CFD, are any modeling tools capable of simulating the temperature stratification effect of the atrium, which is approximately 50 meter high?

Answer

Stratification is a tough problem, but the implications can be modeled in EnergyPlus using user-defined room air models - see the Input/Output Reference Manual for the ROOMAIR TEMPERATURE PATTERN objects. However, these models require that you already know something about the stratification. CFD is probably needed to do a good job of determining the vertical distribution of air temperatures, but a simple way to model atria in EnergyPlus is to assume a vertical gradient (for a 50 meter high space maybe 0.1°C per meter?) and use the ROOMAIR TEMPERATURE PATTERN:CONSTANT GRADIENT object.

Note from Andy Tindale, developer of the DesignBuilder software

You may be interested to know that the ROOMAIR TEMPERATURE PATTERN:CONSTANT GRADIENT object mentioned above has been implemented in the DesignBuilder user interface to EnergyPlus. This allows you to easily model vertical distribution of air temperatures by modulating the temperature gradient during the simulation based on a range of different variables. In DesignBuilder, this data is accessed on the HVAC tab under the Air Temperature Distribution header. For details, go to http://www.designbuilder.co.uk/

REPORTS

Is there a standard output file where I can view calculated surface areas and zone volumes? Alternatively, what report variables can I select to display these parameters in the "eplusout.eso" file?

Answer

For surface areas, you need to specify: **Report, Surfaces, Details;**
The zone volumes are automatically reported. Both will appear in the .eio file, which is a comma separated variable file.

SURFACE GEOMETRY

On my building model, all roofs/floors with more than four sides don't appear in the graphical representation. Is this because information was lost when data was translated from EnergyPlus to CAD?

Answer

By default, they are there. If you use DXF reporting, they appear as thick lines; if you ask for triangulation they will appear as "faces."

Question

I am modeling the floors of a "Plenum Zone" as the respective ceilings of the zones below. In other words, I am using the same vertice coordinates for both surfaces but I am designating their surface types as Ceiling and Floor. EnergyPlus calculates the Tilt angle of the floors as 0.0 instead of 180. On pg 129 of the Input Output Reference, it is stated that any vertex can be chosen as the starting position for horizontal surfaces, as long as the VertexEntry convention is followed. Are there other parameters beside "surface type" and "vertice coordinates" used in coming up with the tilt angle of a surface? Don't quite understand why the floors have tilt angles of 0.0.

Answer

Is it possible that you are inputting them counter-clockwise rather than clockwise or vice versa? EnergyPlus uses the coordinates and calculates the "outward facing normal" angle to determine the tilt angle -- it does not depend on

surface type for this calculation. So, be sure to flip the order of the vertices between the "floor" and "ceiling" versions of each surface. Setting the surface type won't do anything to the geometry.

Thermal Comfort and natural ventilation

I am modeling a space with Natural Ventilation using Airflow Networks and I would like to look at thermal comfort in the spaces. I believe that the thermal comfort model is specified under the PEOPLE command. I selected the Fanger model, but then I am required to input an air velocity schedule that the model uses for its calculations. I want thermal comfort calculated based on the natural ventilation that occurs in the space. I can't schedule the air velocity, because that's what I'm trying to model.

Answer

The EnergyPlus Airflownetwork model only calculates the bulk flows in and out of each zone. It does not calculate local air velocities. For your purpose, it will be necessary to obtain hourly (or time-step if you prefer) output from the airflownetwork simulation, then make your own calculation of local air velocities for each PEOPLE object based on the flow rates, opening sizes and zone volumes. The air velocities can then be fed back into the PEOPLE object using SCHEDULE:FILE:COMMA.

BASEMENT INPUT

Please explain the following input fields for me.

1. "Gravel fill above the floor slab"
Is this gravel fill above the basement floor and alongside the walls? Or, does this refer to gravel fill above the slab covering a totally underground building?

2. "Aspect Ratio"
Should the aspect ratio be based on the entire heat transfer surface of the underground structure or on a single wall? And is the 3-D grid generated for the entire basement including the enclosed space? Or, does the program generate 3-D grids for each wall and floor slab?

Answer
1. Alongside the walls. These three fields go together:

- Field N3: This specifies the width of the gravel fill bed beside the basement wall.
- Field N4: This specifies the depth of the gravel fill above the floor slab.
- Field N5: This specifies the depth of the gravel fill below the floor slab.

2. There is an "A/P ratio" in the EquivSlab object. This is the area-to-perimeter ratio of the floor slab only. I do not know if the 3-D grid subdivides the interior space, but this would expect this to have little impact on the results.

CONDENSING TEMPERATURE CONTROL

I'm new in the field of building energy simulation. I need to investigate how the condensing temperature serves to accurately determine the energy efficiency or COP of air-cooled chillers under part load conditions. How can I model variable lower condenser temperature in EnergyPlus/DesignBuilder? Can anyone point the way?

Answer
I am not sure I understand the question, but here goes: the chiller models use curve-fits to establish efficiency (COP or EIR) as a function of chilled water and condenser temperatures.
The equations are explained in the Input/Output Reference and/or the Engineering Reference Manuals, depending on the particular chiller model.

If you want to see this effect in EnergyPlus output results, you could set up a simulation that is similar to the PlantLoadProfile.idf example file using an air-cooled chiller and a fixed plant load. Then model it with a weather file

or a series of design-days and compare efficiency vs. outdoor
dry bulb.

MOISTURE TRANSFER

I would like to simulate transfer of moisture from the outside surfaces of a basement into the conditioned space.
Should the MTF method be used for this application and not the EMPD algorithm?

Answer
EMPD only models moisture storage and release, so it will not model your example. MTF is intended to model
moisture transfer, however the current implementation is rather limited due to a single set of linear coefficients when
in reality the moisture properties are very nonlinear as conditions change. See the cautions in the Input/Output
Reference and Engineering Reference Manuals in the sections regarding moisture material properties. I do not think
that MTF can deal with soil as the outside environment. So, you probably need to find another tool to model this
moisture transport.

Question
Can I use "Othersidecoeff" boundary conditions when the MTF solution method is selected?

Answer
No. Othersidecoeff has no provision for establishing moisture conditions.

Question
Is there a sample file containing moisture properties data for typical construction materials?

Answer
Please refer to EnergyPlusV1-3-0\DataSets\MoistureMaterials.idf

MODELING ATTIC SPACE

I am simulating a simple residential building. How do I define the attic space, which has a pitched roof? The attic
space is not used but it is naturally ventilated. Does the attic space have to be modeled as a different zone or is there
another way to model it?

Answer
It would be best to model it as a different space. Check out the example files: AirflowNetwork_Simple_House.idf
and AirflowNetwork_Multizone_House.idf. See the geometry (heat transfer surface) portion of the example file
AirFlowNetwork_Simple_House.idf.

ELECTRIC VS. HOT-GAS REHEAT

I am modeling a unitary system (type:Furnace:BlowThru:HeatCool) with the high humidity control activated. First, I
ran the model with a Hot Gas Reheat Coil, then with an electric reheat coil. The rest of the model remained
unchanged. The results from the simulation using the electric reheat coil show that zone temperature and humidity
are tightly controlled. On the other hand, the Hot Gas reheat simulation resulted in significant zone temperature
deviations from the cooling setpoint (zone is cooled as much as 7F lower than setpoint temperature, to maintain
desired max humidity). This, in turn, required a much high system cooling capacity.

Answer
As explained in the documentation, the maximum amount of heat reclaim for the COIL:Desuperheater:Heating is
30% of the total heat rejection. For your simulation, this must not be sufficient (particularly at part-load conditions).

Question

Are there any other aspects of the model that need to be modified depending on which reheat coil type is used?

Answer

Did you compare the maximum heating capacity of your electric reheat coil to the maximum heat provided by your COIL:Desuperheater:Heating object? And, no, there are no other aspects of the model that need to be modified.

Question

Also, are there any Unitary System Objects in EnergyPlus that support a draw-through configuration for cooling and blow-through for heating?

Answer

UNITARYSYSTEM:HEATPUMP:AIRTOAIR and UnitarySystem:HeatPump:WaterToAir currently allow the user to specify both blow-through or draw-through fan arrangements. We intend to add drawthrough configuration to the Unitary HeatCool systems in a future release (Oct 2006 or April 2007 depending on scheduling).

REPORT VARIABLES

In example file "5ZoneAutoDXVAV" I added report variable "System Node Temp" and set all reporting frequencies to one-hour; however, I was unable to see the corresponding fields in the Excel Output file. The ".RDD" file didn't contain the subject variable name. Also, I'm not able to get the usual warning that the report variable is not being reported due to spelling error, etc. Any hints on how to fix this?

Answer

Do you also have a 5zoneAutoDXVAV.rvi file? If so, it will only report the variables that are in that file (if you're using the example file out of the example files folder). If you remove the .rvi file you will get everything reported (up to 255 variables); alternatively, you could add the System Node Temp to the .rvi file. Also, check the .eso file (5ZoneAutoDXVAV.eso) because the first lines show what report variables will be shown in the file along with their "report id" numbers.

SHADED BUILDING

When I defined neighboring buildings that shaded my building I got many warning messages. After I deleted some of the shading surfaces the number of warning messages decreased. Do I have to pay attention to these warning messages?

Answer
There are two things you can do to solve the "too many figures" problem.

1. When describing surrounding buildings, describe only the façades that will cast shadows on your building (do not describe all four sides of a neighboring building if one side is adequate to describe the shading).
2. Add "Diagnostics, DoNotMirrorDetachedShading;" to your input file. This turns off the automatic mirroring of detached shading surfaces. This must be used with caution, because it requires that you be careful about the facing direction of every detached shading surface. Every detached shading surface must face toward the building being modeled, because shadows are only cast in that direction (hemisphere).

OUTPUTS

I need some clarification of EnergyPlus outputs.

1) Report variable: zone/system sensible cooling energy
According to the Input/Output Reference Manual, this refers to the sensible energy supplied by the system to the zone reporting the cooling delivered by the HVAC to the zone. It does not always indicate the operation of the cooling coil. The manual states that if supply air is cooled by outside air it will be reported as a cooling load even if the coils are off.

If I am using Purchased Air with no outside air (just recirculation), and I am defining a ventilation object, is it true that the zone/system cooling energy will be reported only when the coils are on?
I ask the question because I do have air from the outside that might be cooling the zone - as I specified a ventilation object - but this air is not influencing the supply air as it is not being mixed with the return air of the Purchased Air, it is being delivered to the zone directly. Would the zone/system cooling energy be reporting a cooling load with the coils off, even if there is a ventilation object assigned to the zone, if the Outside air is set to "NO" in the purchased air object?

2) Report variable: Zone/sys sensible load to cooling setpoint predicted
According to the documentation this variable reports the predicted sensible load required to meet the setpoint. Again will this variable refer to the loads due to the coils ("on") or all the cooling loads being delivered when I simulate using a Purchased Air object?

3) Report variable: Ventilation loads reports
I noticed that there are new report variables in EnergyPlus 1.3.0 for ventilation loads. I tried to use them with the same settings, Purchased air objects, no outside air specified, but ventilation objects used instead. I got null values for all the variables referring to the ventilation reports. Do these reports only refer to ventilation loads when specified through outside air objects? If I use the ventilation object only - even with mechanical ventilation sets - these variables will report me null values - is that correct?

Answer
The zone heat balance includes the impact of VENTILATION before computing the "Zone/System sensible load to cooling setpoint predicted." The HVAC system (PURCHASED AIR) sees the net load including any increase or decrease in load due to the VENTILATION air. So, in answer to the specific questions:

1) For PURCHASED AIR (with zero outside air flow in the PURCHASED AIR object), zone/system cooling energy will be reported only when the coils are on.

2) Zone/System sensible load to cooling setpoint predicted is the load to be delivered by the HVAC system. VENTILATION objects are not considered to be part of the HVAC system in EnergyPlus. So, for PURCHASED AIR this is the cooling coil sensible load.

3) Correct. The mechanical ventilation report variables only report outside air delivered from an HVAC system, and it only supports systems that are modeled with an AIR PRIMARY LOOP, not zone equipment.

Question
I would like to isolate cooling loads from coils on due to ventilation only - but setting this ventilation separately from my ideal machine. Would this be possible in any way?

Answer
And in order to achieve your goal of determining the load due to ventilation air, three options come to mind:

Do two runs, one with VENTILATION and one without.
Report the Zone/System sensible cooling load, Zone Ventilation Sensible Heat Loss, and Zone Ventilation Sensible Heat Gain on an hourly basis. Then you can analyze the hourly results to determine the benefits and penalties of the ventilation air.
Use COMPACT HVAC and model a unitary system with outside air provided by the HVAC system. Then the mechanical ventilation report variables will sort out the load impact of the outside air (cooling load increase, cooling load decrease, etc.)

OUTPUT – HEAT GAIN

Which output variable can export the heat transferred from the outside condition to the zone through the exterior wall? And which variable can export the heat transferred from other zones?

Answer
There are no report variables that report this. These heat flows all get mixed in with solar gains and internal loads and they cannot be easily separated.
energy outputs

Question
Is it possible to get the energy [J] needed for a HVAC system without introducing the HVAC type?

Answer
No. In EnergyPlus you need some kind of HVAC system to control the zones, or else the zone's thermal conditions will float. The Purchased Air HVAC system is the simplest way to get load information (in Joules) without really describing a specific type of HVAC system. But the *energy* needed to meet loads is a different question. Try the CompactHVAC objects to facilitate adding an HVAC system.

LONG RUN TIMES

I am running a simulation of a heavily glazed, three story building that has 107 zones. The simulation is taking too much time to run. I've tested the debug with two days only (the two design-days) and it took approximately 10 hours to run.

My settings are:
- minimal shading (but I will need to change it to full),
- 25 warm up days (this one can be reduced)
- 6 time steps per hour
- 14 periods of shading calculation (but I'd like to have fewer days, if possible, due to the heavily glazed façade).

How do I reduce the calculation time so I can run a whole year? My next try is going to be fewer time steps and fewer warm-up days, but using full shading.

Answer
Memory might be the issue. You can check this by monitoring memory usage in the Windows task manager.

Don't change the warm-up days. Each environment "warms up" the thermal history of the building by repeating the first day of the environment until the results change within the convergence tolerances specified in the BUILDING object. The maximum number of warm-up days is just an upper limit. The simulation reports the actual number of warm-up days in the eio output file.

If the model has many detached shading surfaces (Surface:Shading:Detached:Fixed and Surface:Shading:Detached:Building), it may help to turn off the automatic mirroring of these shading surfaces:

`DIAGNOSTICS, DoNotMirrorDetachedShading;`

But use this with caution. If DoNotMirrorDetachedShading is specified, then these surfaces must face toward the building being simulated, because surfaces only cast shadows in the direction they face. In general, reducing the total number of surfaces helps speed up the simulation.

WINDOW SHADES

How can I model interior shades that are closed during daytime for all months and exterior blinds that are controlled using "OnIfHighSolarOnWindow" with a defined set point only for summer days. In other words, is it possible to model both interior and movable fixed exterior window shades at the same time?

Answer
The EnergyPlus window shade controls can only model one shading layer. I cannot think of a way to model what you describe.

SOLAR RADIATION

What are the definitions of these solar radiation variables in the EnergyPlus IDD?

- Extraterrestrial Horizontal Radiation
- Extraterrestrial Direct Normal Radiation
- Horizontal Infrared Radiation from Sky
- Global Horizontal Radiation
- Direct Normal Radiation
- Diffuse Horizontal Radiation

Answer
These are defined in the Auxiliary programs document, page 40ff (page 52 of the pdf). However, not all of these are currently used by EnergyPlus. Climate outputs during simulations are shown in the Input/Output Reference, page 47ff, (page 82 of the PDF).

Question
How does EnergyPlus use Horizontal Infrared Radiation from Sky, Direct Normal Radiation, and Diffuse Horizontal Radiation?

Answer
Please refer to the Engineering Reference document "Sky and Surface/Shading Calculations", page 74 (page 106 of the PDF). See if that helps answer your questions.

FLOW RATE PROBLEM

I set a domestic hot water object that is parallel to a fan coil unit; both objects are served by a water heater. I set the flow rate of the domestic water object to be equal to $0.001m^3/s$. I used the autosizing feature of EnergyPlus to size the fan coil unit and found that the maximum flow rate of the plant loop is equal to the amount of the fan coil unit, rather than that of the domestic water in the EIO file. Since the flow rate of the fan coil unit is less than that of the domestic water, I wonder why EnergyPlus doesn't use the flow rate of the domestic water? During the simulation, I used a variable speed pump; the autosizing result of the pump flow rate is equal to that of the fan coil unit. When I changed the pump type to be constant speed, and set the flow rate of the pump equal to that of the domestic water, EnergyPlus showed warnings that the flow resolver is unable to calculate the flow rate correctly. The warning said that the upper limit of the flow rate is equal to the flow rate of the fan coil unit, and the flow rate of the constant speed pump is bigger than that of the fan coil unit. I want to set a bigger flow rate for the plant loop in order to increase the heating demand of the domestic water. Since the temperature difference of the water mixer is around 8C, I can't make the heating demand of the domestic water large enough, based on the flow rate of the fan coil unit. How can I fix these problems?

Answer
The autosizing routines are only looking at the thermal loads on the zone and are not quite smart enough to also factor in the added loads from domestic hot water. You'll have to manually size the pump and plant loop design flow rate. You can find the value by taking the autosizing's flow rate result (reported in the EIO) and then manually add the additional flow rate for the domestic hot water. For a constant flow pump, you will also need a third bypass that is in parallel with the coil and domestic hot water.

ONE LARGE ZONE VS MANY SMALL ZONES

I'm modeling a workshop with a large area and no partitions. However, the temperature in different areas of the space will be very different. Should I partition the workshop into many small zones and compute each temperature gain? Is this possible using Energyplus?

Answer
Yes, it's possible. You'll get some warnings about Zones that don't have six surfaces but these can be ignored. The HVAC system(s) will need to condition each zone separately. This can cause some simulation run time issues if you go with too many separate zones. Also note the 50 zone limit on multizone systems.

A simple way to model the effect of air exchanges among adjacent open plan zones is to use pairs of Mixing objects (if you can guess what the air flow rates might be). Otherwise there is the Airflow Network.

Note that thermal radiation cannot (yet) be exchanged across separate thermal zones (so an inner work area won't be modeled as exposed to a cold or hot window in the adjacent zone).

ZONE MULTIPLIER

I successfully modeled a 3-story building, designating 1 floor as 1 zone (which included a cooling water loop, condenser loop, hot water loop, and primary air loop). Can I use the zone multiplier to expand the 1 zone to, say, 50 zones in a future simulation?

Answer
Yes, you can use the zone multiplier to expand the building. However, all it does is multiply the load sent to the HVAC system. If your original building had 1 air loop, the new building will still have 1 air loop. If everything is auto-sized, then the air loop flow rates, capacities, etc. will increase to meet the larger load.

WEATHER FILES

Which application opens the .epw file so I may read it?

Answer
Use the "WeatherConverter" to convert an epw file to "csv" format, which organizes the data and adds column headers to identify the data. Then open the csv file in a spreadsheet program. Note that EnergyPlus does not use all of the columns of data that are present in the epw files. Read more about weather data in the Auxiliary Programs document. There are also Report Variables in EnergyPlus which report some of the weather data (see the rdd output file).

```
(Start --> Programs --> EnergyPlusV1-3 Programs --> WeatherConverter)
```

AIRFLOW, VENTILATION

I am simulating Natural Ventilation with Airflow Networks and Ventilation Control Mode = Temperature. I want to have only two conditions: closed windows (opening factor = 0) and open windows (opening factor = 1). I have set Limit Value on Multiplier for Modulating Venting Open Factor = 1, and all opening factors equal to 1 as well. However, when i output hourly results, my Opening Factor and Opening Factor Multiplier for Venting Modulation vary between 0 and 1. I tried changing my time-steps and reporting frequency, but this had no effect. Right now, I have my time-step set to 1 and my report frequency set to time-step. I also tried letting all of the fields default to disable the modulation, but this had no impact either.

Answer
What you have done is correct; to make all opening factors either 0 or 1. Output results are also correct at "time-step" report frequency. There are two time-steps used in EnergyPlus: zone time-step and system time-step.
zone time-step is your input from the TIME-STEP IN HOUR object.
system time-step is used internally and is determined by zone temperature difference between previous time-step and current time-step.

If the difference is more than 0.3C, the system time-step is shorter than the zone time-step. The AirflowNetwork model is performed at the system time-step level. If you use "detailed" reprt frequency, you can see opening factors are either 0 or 1.0. When "time-step" report frequency is used, the opening factors are given as an average value over a zone time-step when system time-step < zone time-step. That is why you see opening factors vary between 0 and 1.0.

Question
Also, I am trying to model cross ventilation, but the opposite windows are not always opening at the same times. Is there any way to model ventilation with temperature control and force the windows to open simultaneously?

Answer
The window opening with the temperature control option (ventilation control mode) is determined by the temperature difference between indoor and outdoors only. It is possible that a window is opened in a minute and closed at the next minute, because the minimum system time-step is a minute. The AirflowNetwork model does not consider any opening duration. According to your input file the two opposite windows belong to two different zones. Since the window opening is determined by the temperature difference between a zone and the outdoors, two windows will not open at the same time if two zones have different temperatures. Although you have options to select a ventilation control mode, you only can select one of them, not both, such as constant and temperature control options.

PLENUMS

I have a large building with seven floors, each floor with a plenum. All the return air from the zones enters the plenum before entering the air handling unit in the mechanical room and this happens for every floor. One side of the air handling unit sucks the return air from the plenum and sends fresh air from its other end into the supply duct as supply air to the zones. My questions are these:

Should I model the plenum as a separate zone? I am actually using EP-Quick to build my zones but unfortunately EP-Quick does not have the capability of building plenums.

Should I just ignore the plenum and treat it as return air duct that carries return air from zones to the air handler in mechanical room?

Answer
Two members of the EnergyPlus team weighed in with answers:

For calculating building energy consumption, modeling the plenums as separate zones is generally not very important. And, yes, there are cases where it will be important: a one-story building with a large roof area and considerable outside air, for instance. However, for a multi-story office building it isn't important.

Should you model plenums? It depends largely on the specific goals of the analysis. If you decide to ignore the plenums, you must make the conditioned zones tall enough to include the plenum height so that the total area of exterior wall is accounted for. If you choose to model the plenums, then the plenum is modeled as a thermal zone. With EP-Quick it may be possible to create two buildings - one that is seven stories with a floor-to-floor height of the conditioned space only, and then another which is seven stories with a floor-to-floor height of the plenum space only, and then mix the two together. All of the surface coordinates should be OK except for the z values which could be easily modified to stack thing up correctly. Note that the plenum floors will come out of EP-Quick as multiple zones, but these can be combined into a single zone simply be reassigning the interior environment for the plenum surfaces and deleting the extra interior surfaces that would be created in the plenum. Well, it may not be a real simple task, but it is doable.

SHADING DEVICES AND WINDOW-5 PROGRAM

I want to model controlled interior shades and movable exterior shading devices for windows in a multi-story residential building. I specified window systems with WINDOW-5 and used the WINDOW-5 data file in EnergyPlus. I specified MATERIAL:WINDOWSHADE as the shading device, but I don't know how to integrate that shading device into my WINDOW-5 data.

Answer
For this case, in the WindowShadingControl object, leave "Name of construction with shading" blank and specify the shade material name in "Material Name of Shading Device."

AIR-HANDLING UNIT

I'm trying to control an AHU that consists of:

Fresh Air ⇨ Cooling Coil ⇨ Heating Coil ⇨ Supply Fan ⇨ Zone

The run period takes place over seven days. The AHU is a constant volume system. The cooling coil cools the air to 11.5C. The heating coil heats the air back up to 16C.

Unfortunately, the heating coil does not engage adequately. After numerous attempts, it seems that the heating coil does not *sense* that the air has been cooled to 11.5C. The heating coil is only engaged when the outside air is below 16C (it does not realize that the air has been cooled by the cooling coil).

All additional heating in the zone is supplemented by a baseboard heater system and the zone is maintained at 23C. The system is a full fresh air constant volume system. Regardless of the outside conditions, I require the air to be cooled to 11.5C and heated to 16C before it hits the supply fan. It is supplied directly to the zone after this. Can anyone spot how I may fix this problem?

Answer
The Controllers on the water coils operate sequentially. In this case the cooling coil controller needs to go first, and the HW coil controller second. So change your input as follows:

from:
```
    CONTROLLER LIST,
    AHU1:ControlList, !- Name
    Controller:Simple, !- Controller Type 1
    HC:Controller, !- Controller Name 1
    Controller:Simple, !- Controller Type 2
    CC:Controller; !- Controller Name 2
```

to:
```
    CONTROLLER LIST,
    AHU1:ControlList, !- Name
    Controller:Simple, !- Controller Type 2
    CC:Controller, !- Controller Name 2
    Controller:Simple, !- Controller Type 1  HC:Controller; !- Controller Name 1
```

MULTIPLE AIR-HANDLING UNITS

I'm trying to model a 165,000-ft^2 four-story building that has four large AHU, each with heating and cooling coils. The AHU use hot water and chilled water that is supplied from a central plant. The system is a VAV with reheat at the terminals. I'm not sure how to set it up. Is there an EnergyPlus sample file I can use?

Answer
I don't think there is an example file with more than one AHU (or Air Loop in EnergyPlus); however, you could start with the example file named LgOffVAV.idf. Then replicate the entire set of objects that describe the air loop to have four AHUs instead of one. The single central plant can serve all the various demands by setting the splitter/mixer/branches to include all the coils in the different air loops. But if all the AHUs are all similar (in terms of operation, loads served, recirculation, outdoor air, etc.), then -- depending on your goals -- it may be okay to simplify the energy model to have a just a single AHU. Simpler models run faster.

LOW TEMPERATURE RADIANT SYSTEM HYDRONIC

I want to simulate a low temperature radiant system hydronic using purchased heating and without any other equipment. I only have a plant loop and want to see the energy consumption of the whole system. I know that the heating demand of the building should be 6.4kW (external calculation), so I set the purchased heating to that value. I want to simulate the temperature pattern in the four zones as a result of the calculated heating power. The problem is that EnergyPlus uses the value of the heating load range based operation. If I set the value to 6400W the program turns off the simulation because the demand is higher than the entered value. Is there a chance that the program only uses the purchased heating value even if the demand is higher than that value?

Answer
In any LOAD RANGE BASED object, it is recommended that the last load range value be a very large number. This way the equipment will operate, even if it is not large enough to meet the current (or accumulated load if the loop setpoint is drifting). So, if you set this to be something like 10MW, the system should never shut off due to load range.

In your input file, the PURCHASED:HOT WATER capacity is set to 4000 W. The simulation should meet the hot water loop demand up to 4000W. If the demand exceeds 4000 W, then the loop temperature will fall and the space temperature will fall. When the demand becomes lower, then the temperatures should recover.

INDEX