EnergyPlus Articles from the
Building Energy Simulation User News

01/01/2003 through 12/31/2003

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

January 2004

Table of Contents

TEACHING MATERIALS: SLAB.EXE

When using the slab program, as outlined in Lecture 24 of the EnergyPlus teaching materials, am I correct in thinking that I don't need to place a layer of earth in the floor construction once the ground and outside face temperatures have been determined? For past models, I have used 1m of earth as part of the floor construction though I have never been too sure that this is realistic.

Answer:

You are correct. There should be no extra layer of earth in the slab floor construction. The slab program is reporting the temperature of the ground which is in contact with the outside surface of the floor slab.

DAYLIGHTING

I am running an EnergyPlus Model against a validated DOE-2.1E model. For the daylight component of the simulation, I have hourly illuminance values for EnergyPlus that are four to ten times greater than what I am seeing with DOE-2.1E.   Can this be correct? I am using the same weather file and the same WINDOW 5.1 file for both simulations. I have checked over my input and am fairly sure that it is correct. Any suggestions?

Answer

The problem is in the DOE-2 input. The DOE-2 and EnergyPlus daylighting reference points appear to be in the same place at first glance. In both programs they are relative to the zone origin and the values are equivalent in the two input files. However, the DOE-2 input has zone origins of 0,0,0 and the EnergyPlus input does not. It is a very subtle difference between the two runs -- it took two sets of eyes to find the problem at this end.

Since your DOE-2 file is essentially set up in global coordinates (with all zone origins at 0,0,0), you will need to reposition the lighting reference points in DOE-2 to be consistent. There is a handy drawing tool (DrawBDL at www.drawbdl.com) available for DOE-2 which shows the daylighting reference points and revealed the problem.

Here's another option. In EnergyPlus, the daylighting reference points are shown in the DXF drawing file. To create this file, add "REPORT,Surfaces,DXF;" to your IDF file. If you do not have software which will view a DXF format drawing file, VoloView Express is available for free at

http://usa.autodesk.com/adsk/servlet/index?siteID=123112&id=2753004

For an example, see ExampleFiles\DaylightingDeviceTubular.idf. It uses two reference points in each daylighting zone. The first reference point is shown as a red circle, the second as a blue circle.

REPORTING SENSIBLE HEATING AND COOLING LOADS

I have built a simulation and when I run it I get both zone sensible heating and cooling rates during the same hours. I am using NormalDamperHeatingAction, but when I look at my zone output, I have mass flow rates above the minimum specified fraction. I then look at my "Total water heating coil rate" and find that I am still transferring heat into the air stream, which seems very odd. Has anyone run into to similar problems, and is there a remedy? Thanks.

Answer

If you are reporting hourly, you are getting an average result over several HVAC time steps. If you report results at the "detailed" frequency, so that you can see data for every HVAC time step, then this may help explain what is happening.

CANADIAN WEATHER FILES (CWEC)

Does anyone know of a source of free TMY2 weather files for Canadian locations?

Answer

The Canadian equivalent to TMY2 is called CWEC (Canadian Weather for Energy Calculations) and is in WYEC2 format. Hourly files for 55 locations are available from U. S. National Climatic Data Center (http://www.ncdc.gov/ or http://www.fedworld.gov/) or the Information Archive at Environment Canada (http://www.ec.gc.ca/). We heard through the "grapevine" that the Environment Canada website is being revamped and the weather files may not be easy to find. Until the website is finalized, you may direct questions to climate.services@ec.gc.ca.

The Quick Fix

If you download and install EE4 (http://www.ee4.com/) you get 44 CWEC files in a spreadsheet with the city for each weather file name.

ERROR IN THE ACH CALCULATION

When I used COMIS and obtained the infiltration volume (m^3) and ACH for each zone, I checked to see if the ACH was the same as dividing the volume output by each zone's volume. My output time step is in hours so I figured that the volume output represented all the exterior air that had entered the zone during that hour. What I discovered was that my own calculated ACH was exactly six times smaller than EnergyPlus's ACH. I figured it had something to do with my HVAC time steps per hour being six. Is there an error with Energy Plus's ACH or am I going about this the wrong way.

Answer

You are right; there is a bug in EnergyPlus. The time step values of "Infiltration-Air Change Rate" are OK. However, for hourly reporting, the sum of the time step values is being used instead of the average. We have fixed this and thank you for finding the problem!

RADIATOR SIMULATION

I am trying to simulate a building with a boiler that serves three zones heated with water radiators made of aluminum. I am using the "LOW TEMPERATURE RADIANT SYSTEM:HYDRONIC" component. To do this I have to calculate a fictitious hydronic tubing length that produces energy and power similar to that of the real radiators. Is this correct or is there another way to simulate such a plant?

Answer

The component "LOW TEMPERATURE RADIANT SYSTEM:HYDRONIC" models hyrdonic tubing that is embedded in a surface, such as a heated floor slab. For radiators, the best available model is BASEBOARD HEATER:WATER:CONVECTIVE. This models fin-tube coils, which heat primarily by natural convection, and is currently used to model all types of free-standing or surface mounted hot water "radiators."

BUILDING GEOMETRY MODIFICATION

We are simulating a large building with more than 900 walls and windows. If we want to make a change in the geometry of any building component, like walls or windows, we have to modify the 12 coordinates of the corresponding component every time. For example, if we want to make a change in a window, we have to define, in an IDF file, new XYZ coordinates as four points, each corresponding to the position of the component. This is a tedious process, particularly when we have a large number of windows and walls. Is there a simpler method?

Answer

You could try the IFCtoIDF translator that converts the building geometry in CAD drawings saved in IFC format into IDF format (see http://www.eere.energy.gov/buildings/energyplus/ifc.html). This is still a beta product, but it would allow you to draw your building and make changes in CAD and then more easily transfer these changes to EnergyPlus. You would need to do some manual editing of the IDF to change Construction types of the envelope elements. The IFCtoIDF utility is included as part of the EnergyPlus download and can be installed by checking the IFCtoIDF checkbox during installation. Note that you should then download the update of this utility from the EnergyPlus Support website at the URL below.

http://groups.yahoo.com/group/EnergyPlus_Support/files/Updates/Windows/IFCtoIDF_Update_6-10-03.zip

DAYLIGHT SAVINGS TIME

Does anyone know what actually happens when Daylight Savings Time (DST) is turned on in EnergyPlus? A simple test that compared annual runs both with and without DST showed no difference in the hourly results. I used normal EnergyPlus weather files and the object "DaylightSavingPeriod, 1st Sunday in April, Last Sunday in October." Do the output time stamps shift with DST or do they stay in Standard Time? And what happens to those messy hours at the point of change?

Answer:

When active, DST will shift all scheduled items by one hour. Reporting is always in Standard Time. We ran a quick experiment with the PurchAir example file, added the DST object and ran from April 15 to May 15. What we found is that the DST is not active unless the RUN PERIOD field for DST says "Yes."

```
RunPeriod,
  4,  !- Begin Month
  15,  !- Begin Day Of Month
  5,  !- End Month
  15,  !- End Day Of Month
  UseWeatherFile,  !- Day Of Week For Start Day
  Yes,  !- Use WeatherFile Holidays/Special Days
  Yes,  !- Use WeatherFile DaylightSavingPeriod <-- MUST BE YES
  No,  !- Apply Weekend Holiday Rule
  Yes,  !- Use WeatherFile Rain Indicators
  Yes;  !- Use WeatherFile Snow Indicators
```

DWG / IFC / IDF

I have the latest version of Architectural DeskTop 3.3 (ADT), which supports IFC. BSPro COM-Server is installed and authorized correctly, too. I created a 3-D CAD file and did an IFC export. It gave me an error message but created an IFC file and then ran the IFCtoIDF utility; it created the IDF file too.

Then, on running the simulation from the EnergyPlus launch it terminated. The errors quoted were:

```
 ** Severe  ** No items found for Required Object=ZONE
 ** Warning ** Note -- Some missing fields have been filled with defaults.
               See the audit output file for details.
 ** Severe  ** Other miscellaneous errors found in input
 **  Fatal  ** Out of "range" values and/or blank required fields.
 ************* EnergyPlus Terminated--Fatal Error Detected. 1
               Warning; 2 Severe Errors
```

Is there a specific way to build an AutoCAD model in order for it to export correctly as an IFC? And can I take a regular DWG file of a building and export it?

Answer

There are two questions to be answered here.

1. An update to the IFCtoIDF utility is available on the EnergyPlus Support site at
   http://groups.yahoo.com/group/EnergyPlus_Support/ under Files/Updates.
   This update is intended to address the problems with Spaces/Zones in ADT.

2. The problem with opening a DWG and trying to export it to IFC is that the DWG may not contain the object-based representation of entities like Walls, Windows, etc. that is necessary for mapping to the IFC data model. The drawing must be created in ADT.

SURFACE GEOMETRY AND DIMENSIONS

When inputting surface geometry, should we refer to dimensions from axis to axis or should we refer to the dimensions of the inner surface of the construction element? We have noticed that zone volume is calculated as if the inputted dimensions are those of the inner surfaces; how, then, are differences in thickness of the envelope's construction elements handled?

Answer

The choice of interior, exterior, or center-line (axis) for surface dimensions is up to the user. For most wall constructions, the slight change in zone volume or wall surface areas will have very little impact on the results. Use whatever dimension is most conveniently available from the building plans. For very thick walls, center-line (axis) dimensions should probably be used to avoid overestimating the heat transfer through exterior walls. Differences in the thickness of construction elements is only used to determine the heat transfer characteristics of the surface. For shadowing and other geometric calculations, all surfaces are planes with no thickness.

TIME STEPS

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

Answer:
The Time step In Hour object specifies the "basic" time step for the simulation. This is used in the Heat Balance calculation as the driving time step. When the HVAC portion of the simulation begins its solution for the current time step, it uses the basic time step as its maximum but then can reduce the time step, as necessary, to reach the solution. The technical details of the approach are explained in the Engineering Documentation, under "Integrated Solution Manager." Users can see the actual HVAC time step used if they select the "detailed" option on an HVAC report variable (e.g. Zone/Sys Air Temp). To contrast, the "Zone" variables will only be reported on the Heat Balance time step (e.g. Mean Air Temperature). Four time steps per hour are recommended in EnergyPlus for non-HVAC and six for HVAC calculations.

Question:
Does this mean that using a time step of one will cause EnergyPlus to diverge or sacrifice the accuracy?

Answer:
Actually, four or six time steps per hour are *necessary* for the HVAC calculations. The heat balance simulation will give reasonable results at one, four or six time steps per hour. Note that the Time Step value must be evenly divisible into 60 – therefore is limited to: 1,2,3,4,5,6,10,12,15,30,60

OPERATIVE TEMPERATURE PREDICTION

Is it possible in Energyplus to predict the operative temperature at a particular location within a space? I need the mean radiant temperature (MRT) and mean air temperature (MAT) at user-specified locations within the space.

Answer:
EnergyPlus does not calculate exact view factors, but you do have partial control over the MRT calculation for thermal comfort. You can choose "Zone Averaged" or "Surface Weighted" MRT. With Surface Weighted MRT, you specify a single surface in the zone which will be given a view factor of 0.5 to the person, and the other half of the MRT is the zone average. For more details, please see p. 146 of the Input Output Reference.

ENERGY AND ELECTRICITY OUTPUTS

I am running three simple heat pumps serving three simple classrooms. I know the RDD file very well but I don't know which report variable to choose in order to find the electricity demand by the heat pumps. For the cooling and heating energy supplied for the room air, I use the zone/sys sensible heating and cooling report. Is this correct? Another thing : I noticed that the report meter: Electricity:Facility gives me a value much bigger than the sum of : zone/sys sensible heating + sensible cooling + lightning. can you tell me why ?

Answer:
To get the electric power of the heat pump, you must request the electric power output variables for each of its components: DX cooling coil (DX Cooling Coil Electric Power), DX heating coil (DX Heating Coil Electric Power), and fan (Fan Electric Power). Zone/sys sensible heating/cooling are the net heating/cooling delivered to the zone. To see the gross load on the coils, see the DX heating/cooling coil output variables.

Heating and Cooling Systems

Does EnergyPlus support heating or cooling systems other than air conditioning systems? I am thinking of water systems like underfloor heating, active wall/ceiling systems and radiator systems.

Answer:
Yes, it can do all of these. In addition to forced air systems using DX and chilled water cooling, EnergyPlus can model hot water radiators (BASEBOARD HEATER:Water:Convective), heated and cooled surfaces (LOW TEMP RADIANT SYSTEM:HYDRONIC, LOW TEMP RADIANT SYSTEM:ELECTRIC) and gas or electric radiant heaters (HIGH TEMP RADIANT SYSTEM).

PURCHASED HOT WATER

We use the UnitHeater example and we would like to know if we can control the temperature of inlet and outlet of the water in the Purchased:Hot Water. We think that the temperature of the outlet water is controlled by the loop temperature setpoint schedule but we don't know for sure.

Answer
Yes, the outlet temperature from PURCHASED:HOT WATER will be the loop setpoint if the capacity is sufficient. The inlet temperature is determined by the coil loads and the loop flow rate.

Question
What kind of buildings use this type of system of heating?

Answer
This type of system is typically used by buildings in an urban area with district heating available. It is also an option for modelers who want to look just at plant loads without simulating a boiler (or chiller, PURCHASED:CHILLED WATER).

COOLING AND HEATING CAPACITY LIMIT

I would like to limit the cooling and heating capacity of a system. On the one hand, when applying an electric baseboard heater the nominal capacity can be limited. On the other hand, in the case of a DX cooling coil, the rated capacity refers to design conditions but the effective capacity at a time step (output variable) can be beyond the rated one. What system allows specifying a cooling capacity limit that cannot be exceeded in the simulation run?

Answer
The only way to limit cooling capacity is to use a chilled water system. A fan coil unit would probably be the easiest. Use purchased chilled water to serve the chilled water supply loop, because it has a fixed capacity limit. When the limit is reached, the chilled water temperature will rise above setpoint and limit the total cooling that can be delivered to the space. There is some lag in the chilled water loop, so you may see some small variations in cooling capacity when the system is at the limit.

DOE-2 TO ENERGYPLUS CONVERTER

Question
I have some DOE-2 files that I would like to convert and run in EnergyPlus. Is there a software package that would allow me to do this.

Answer
Read about the DOE-2 to EnergyPlus translator in *gettingstarted.pdf* under *creating input files*, *doe2translator* (pg 43 of 53). Note that the DOE-2 to EnergyPlus translator does not convert HVAC systems/equipment.

LOAD AND ENERGY BALANCE

1) What is the best/easiest way to find out a building's hourly total cooling load using EnergyPlus?

2a) How can I verify that the cooling plant is providing enough (or not enough) energy to meet the building load on an hourly basis?

2b) Also, I pulled out some outputs from EnergyPlus and tried to do a balance check but it didn't match.

Answer
1) The easiest way to determine the sensible load requirements to maintain the specified thermostat setpoints is to use the purchased air simulation with no capacity limits and include outside air if applicable. This will also provide a rough approximation of latent load requirements.

2a) If the thermostat setpoints are being met, then the cooling plant is providing enough cooling; this is the only available measure of performance. Use the report:table:bin feature to see the distribution of temperatures in the zones; you can then determine whether the setpoints are being met.

2b) It is very difficult to do a simplified energy balance calculation which will agree with the simulation result. To balance the building envelope, you must reduce the problem to a steady-state condition by defining a constant temperature day with no sunshine. Balancing loads within the HVAC simulation is easier since each time step is a quasi-steady-state simulation, but there are still many details which must be accounted for.

MEANING OF OUTPUT VARIABLES

Can anybody explain to me the meaning of these output variables:

Layer Rel Humidity[ ]. Units
This is the Relative Humidity inside the construction at each layer interface. Relative Humidity is explained in just about every Thermodynamic text book, ASHRAE Handbook of fundamentals, and any Psychrometric book. It is unit-less and runs normally between 0 and 1, where anything greater than 1.0 there would be condensation. This value can be reported along with the humidity ratio to help the user determine if there was or could be condensation in the wall construction.

Outside Surface Mass Flux[kg/s/m$_2$]
The Surface Mass Flux would be the amount [kg] of moisture that would enter or leave that material at the outside face.

USE A BATCH COMMAND TO QUEUE ENERGYPLUS SIMULATIONS

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.

OPTIMIZE ENERGYPLUS RUNS

Also, given a fixed building design, I'm trying to optimize the orientation that would result in the lowest energy consumption (maximum self shadow from the building). How can I do this in EnergyPlus?

Answer
There are a couple of options:
1) GenOpt (SimulationResearch.lbl.gov/) from LBNL can be used to drive EnergyPlus for optimizing building rotation.

2) Also, the Deringer Group has developed a tool called EZPlus-Parm, available for free download at their web site: www.deringergroup.com/Software/EPlusTools.htm EzPlus-Parm is a standalone Windows tool intended primarily for EnergyPlus parametric analysis. It was designed to simplify the running of multiple parametric EnergyPlus simulations.

RUNNING COMIS WITH HVAC

I know that COMIS is not susposed to give reliable results when the HVAC system is running but is this only true when mechanical ventilation is present where outside air is being brought in. If my HVAC system just recirculates the inside air such that the conditioning system is not pressurizing or depressurizing the zone, will the COMIS results be accurate.

Answer
The COMIS simulation calculates the pressures and corresponding flows based on the interior and outdoor environmental conditions from the previous timestep, i.e., it does not know anything about pressures or flows that may result from the HVAC operation. So, theoretically, if your system is simply recirculating inside air perfectly without any pressurization effects, this should not impact the COMIS predicted flows and pressures. So, if your HVAC system is not pressurizing or depressuring the zone, the COMIS results should be OK. Building Energy

MOISTURE STORAGE AND PASSIVE HUMIDITY CONTROL

I am trying to model passive humidity control using building materials as moisture and thermal flywheels. Most of the rooms are internal zones, with mass walls such that moisture and thermal effects from adjacent zones will be minimized. I'd like to be able to look at design days and hopefully at longer term simulations as well. Is EnergyPlus capable of this?

Answer:
Yes, EnergyPlus can model this using the EMPD (effective mean penetration depth) method. Since material property data for moisture modeling is limited, you should run sensitivities on this and qualify your results accordingly.

PID CONTROLLED VALVE

If a constant speed pump is used in the plant, the flow rate is controlled by mixing valve. How to model a PID controlled mixing valve in EnergyPlus?

Answer
EnergyPlus controllers are not modeled as proportional or PI or PID. In the case of a mixing valve it will balance perfectly to the flow requests from the various branches.

REPORT VARIABLE *SUMHAT [J]*

Will you explain the variable *SUMHAT [J]* that appears in the report variable data dictionary.

Answer:
The short answer is twofold:

SUMHAT is a "developer"/debug variable that a developer might use to determine if the heat balance calculation is going properly, but is not labeled as such. It's not really intended for users. It is the sum of product of heat trans coefficients, surface area and inside surface temperature. Here is a somewhat longer explanation:

SUMHAT is an internal variable, used during the calculation of the zone air heat balance. It is calculated during the inside surface heat balances.

SUMHAT applies to a particular zone and is the sum of the product of *h*, *A*, and *T* for each surface:

- *h* is the convection coefficient for a particular surface (this can vary with temperature difference, orientation, etc.).
- *A* is the area for a particular surface.
- *T* is the temperature of the surface (_C). Note that *T* is not the temperature difference between the surface and the zone air because EnergyPlus must solve for the zone air temperature.

So, for each zone, during the inside surface heat balance, EnergyPlus sums up *hAT* for each surface in that zone and stores this value in the variable *SUMHAT*. The number reported is in Joules [J].

VARIABLE SPEED PUMP EQUATION

Please verify the equation listed in the Input-Output reference manual for Pump:VariableSpeed, page 291. The last Part Load Ratio (PLR) term in the equation is to the fourth power. Should the last term in the equation be to the third power?

Answer:
Yes, the last term should be to the third power. Here's the equation from the code:

```
FracFullLoadPower
= PumpEquip(PumpNum)%PartLoadCoef(1) &
+ PumpEquip(PumpNum)%PartLoadCoef(2) *
PartLoadRatio &
+ PumpEquip(PumpNum)%PartLoadCoef(3) *
PartLoadRatio**2 &
+ PumpEquip(PumpNum)%PartLoadCoef(4) * PartLoadRatio**3
```

STEADY STATE TEMPERATURE – PART I

In the Control Types Schedules, say for the thermostat of a VAV with reheat, you are using discrete numbers 0 through 4. When the Day Schedules are created, I assume each whole number in the Day Schedule corresponds to a type of control such as SINGLE COOLING SETPOINT or DUAL SETPOINT WITH DEADBAND and is used to switch between the different types of control based on occupancy. How is the correlation made between the whole numbers used in the Day Schedule and the type of control?

Answer: (see pp. 324ff in the Input Output Reference for more details)
0 – Uncontrolled (No specification or default)
1 – Single Heating Setpoint
2 – Single Cooling Setpoint
3 – Single Heating/Cooling Setpoint
4 – Dual Setpoint (Heating and Cooling) with deadband

STEADY STATE TEMPERATURE – PART II

I would like to simulate the temperature in a room with an internal load of 1000 W, the external temperature at constant 22°C. I do not apply a weather file but would like to estimate the steady state temperature in that room. This requires a run period of several weeks. Which objects do I need to define to get the required result. So far I can model only one day.

Answer:
What you will need is to artificially prepare a weather file with the appropriate outdoor conditions. Fortunately, this is fairly easy with the EnergyPlus WeatherConverter program (folder preprocess\weatherconverter). Generate a .csv file of your appropriate location. Take that file into Excel for easy editing, and replace the dry bulb temperature with your 22°C temps. You will need to determine an appropriate dew point as well. Delete the rows beyond where you want the simulation to proceed (or not). Now direct the Weatherconverter program to convert a .csv file to the normal .epw file and then use this as your weather file for the simulation.

INDEX