TYPE 360: FLOOR HEATING AND HYPOCAUST

(Mode 1 or 2 - FLOOR HEATING or HYPOCAUST)

Karel Fort
Weiherweg 19
8604 Volketswil, Switzerland

Purpose:

This model describes the transient thermal performance of hypocaust/murocaust and floor heating panels. It enables a simulation of a wide range of systems with different geometrical designs. The model is based on the method of finite differences.

Short History of Development and Validation

The model was first developed and validated for water floor heating systems [1]. Validation was based on measurements made in a laboratory cabin. First validation was made for the floor heating only (disconnected from the room).

Later, as this model was implemented into TRNSYS (former TYPE60), validation together with TYPE19 - Single Zone Model was undertaken in the same laboratory cabin [2].

In a further development the model has been extended for parallel flow of air through hypocaust/murocaust type storage. Validation was made in a laboratory test rig [3] on two different types of hypocaust design. Version 1.0 (July 94) has been documented in a detailed report: 'Description of the physical model and its implementation into TRNSYS'. The most of this is as well in this description, together with some improvements and slight changes.

In a further up-date the physical model remained the same. However, several improvements concerning the implementation were made, making it for the user easier and reducing the chances of errors. With Version 3.0 (October 96) the name of the type has been also changed from TYPE 60 to TYPE 360.

Latest update of this routine was in April 2001. Changes were included because of some iteration problems if more than one unit was used. Also the alfa convective correlations are changed [4]. They are now the same which are used in TYPE 80.

For several reasons alfa,fluid was changed. It is now water speed depending.

Table of contents

1 Introduction

The model was developed to describe the transient thermal behaviour of hypocaust and murocaust (vertical positioned hypocaust) type air driven heat storage. Its main purpose is to enable thermal simulations of the performance of buildings containing air collector (window type or opaque type) connected to a hypocaust storage. The aim was to enable a simulation of a wide range of hypocaust types with different geometrical designs. Further the model shall be used to develop calculations and design guidelines for hypocaust systems.

Method

This here described model is based on a model developed earlier for water floor heating systems [1] which was implemented into TRNSYS [2]. The utilised algorithms have been redefined and adapted to the particular physical conditions of an air driven system. **The TYPE 360 can now be used for floor heating systems as well as for hypocaust systems.**

The model was validated with transient measurements made in laboratory test rig on two typical hypocaust systems. The integration into TRNSYS building simulation is possible either within TYPE 19 (single zone) or within TYPE 56 (multizone building).

We thank the Bundesamt für Energiewirtschaft for the financial support of this project as well as the Swiss Federal Institute of Testing of Materials (EMPA) for giving us the opportunity to use their facilities for the measurements. Stefan Holst (ZAE Bayern now TRANSSOLAR) made a valuable contribution for the difficult link of TYPE 360 to TYPE 56 and the elimination of some bugs. Peter Vogelsanger supplied the example 2 of TYPE 360 linked to TYPE 56.
The latest update and example (exampl56 for TYPE 360 / TYPE 56) has been made in liason with A. Knirsch/TRANSSOLAR.

The following chapters describes the physical model and describes how to use TYPE 360 within TRNSYS. This description is based on TRNSYS 14.2.

2 Physical Model

2.1 General description

The TYPE 360 models the transient thermal behaviour of a heat storage in a floor, in a wall, or in a section of it. The energy transporting media moving through channels or pipes can be air or fluid (called hereafter: fluid). The model describes the heat conduction and the storage of a 3-dimensional element. Besides the application for heating panels it can also be used for modelling cooling ceilings or wall panels.

The geometry of the hypocaust / floor heating has to be defined in a form of a finite difference mesh for a typical cross section prior the simulation. An simple graphical program is supplied to check the geometrical input.

The INPUTS to the model are

- the temperature of the incoming air / fluid
- the flow rate of the fluid
- thermal behaviour conditions of the surrounding rooms

The OUTUTS of the model are

- the outlet temperature of the fluid (as well the mass flow; the model was developed for applications where an accurate knowledge of the outlet temperature for the fluid is required)
- the heat flux on the surfaces
- the surface temperatures
- the energy stored in the panel

Fig. 1 shows a typical implementation of a hypocaust within a building.

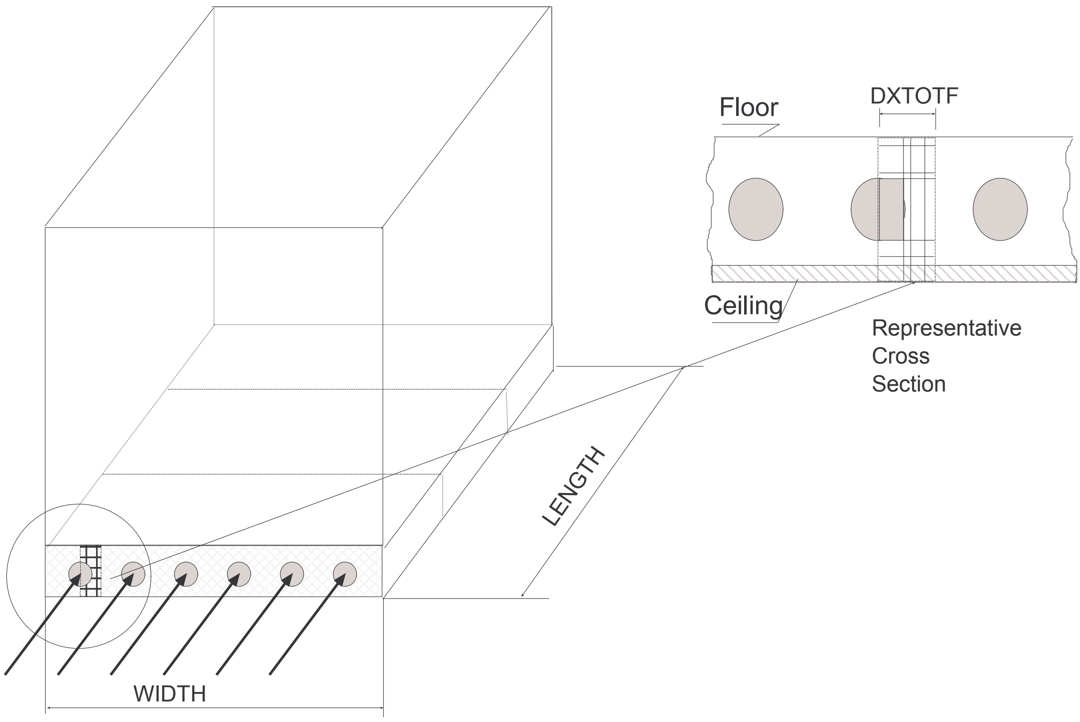


Fig. 1: Typical implementation of a hypocaust within a building.

In the model, the panel is devided into segments. A representative cross section is selected for the segment taking account of the symmetries. For each representative section the heat conduction is calculated using the Finite Difference Method (FDM). As the result from the model the temperature profile of the representative section is obtained as well as the heat flux in the surface and the fluid temperature at the exit of the segment.
The fluid at the outlet of one segment is the inlet temperature of the next segment. In this manner the temperature change of the fluid is taken into consideration.

2.2 Finite Difference Method

The heat conduction, heat storage and heat transfer on the boundaries is a complex 3-dimensional problem. In this model, a simplified 2-dimensional solution of the heat conduction in a representative cross section was coupled to the moving fluid.

2.3 Symmetries

First of all, the representative cross section has to be defined. The user has to decide on possible simplifications and where symmetries can be assumed and utilised. In order to reduce the calculation effort, **it is important** to set a symmetry. It also means that at this plane of the cross section the conduction problem will be considered as adiabatic (no heat flux across the symmetry plane). Fig. 2 shows an example of symmetries to a hypocaust storage system.

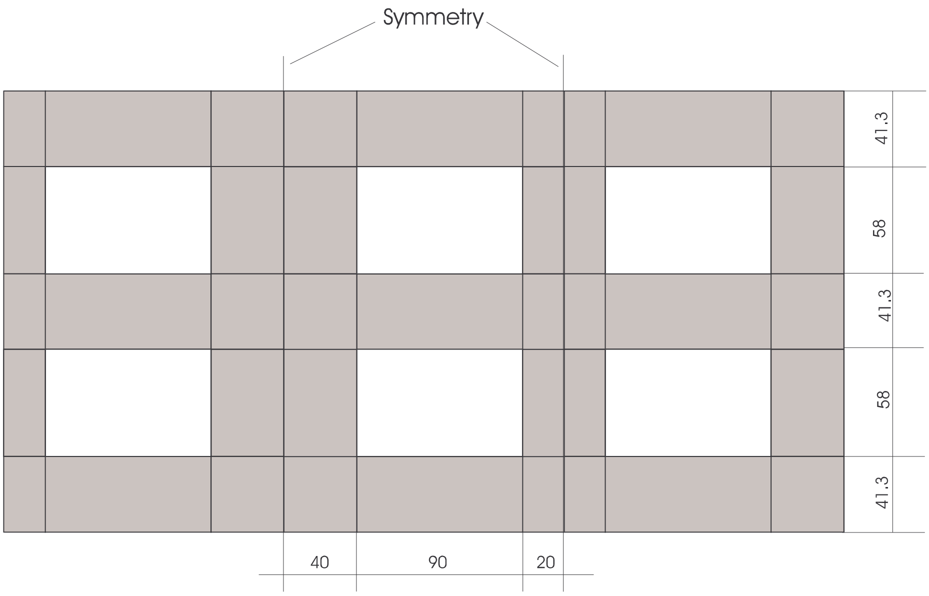


Fig.2: Definition of the representative cross section and symmetries

Hence the representative cross section to be modelled is only a small portion of the entire cross section of the hypocaust.

2.4 Nodes

Now, the representative cross section is divided into a number of rectangular nodes. Fig 3 shows a typical mesh of nodes for the given hypocaust.

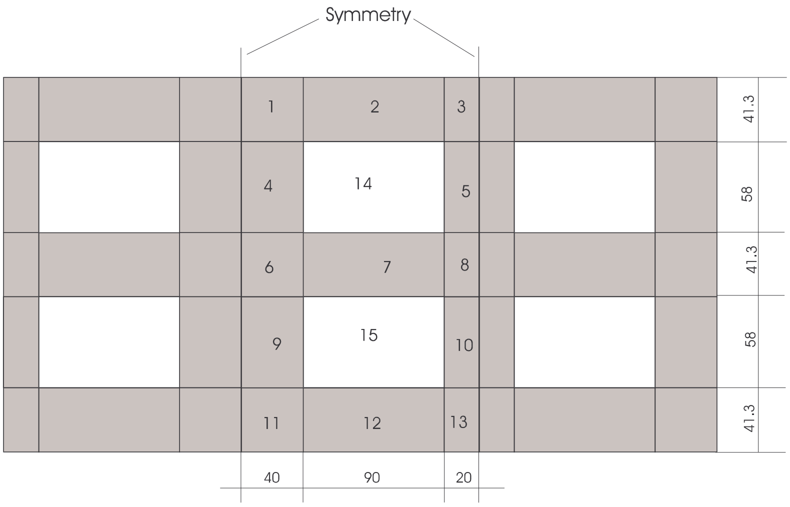


Fig. 3: Division of the representative cross section into nodes

The definition of the geometry of the nodes, the definition of the neighbourhood of the nodes and the materials described in chapter 4.

2.5 Energy balance

For each node the transient energy balance is written as shown in the following example of the solid node 4 next to fluid 14:

$$\frac{dT_4}{dt} \cdot Dx_4 \cdot Dy_4 \cdot c_p \cdot \rho = \left[\frac{T_{fl,in} + T_{14}}{2} - T_4 \right] \cdot R_{4,14} - (T_4 - T_1) \cdot R_{4,1} - (T_4 - T_6) \cdot R_{4,6}$$

Where:

$$R_{4,14} = \frac{Dy_4}{\frac{Dx_4}{2 \cdot \lambda_4} + \frac{1}{\alpha_{fluid}} + \frac{wallthickness}{\lambda_{Wall}}}$$

$$R_{4,1} = \frac{Dy_4}{\frac{Dx_4}{2 \cdot \lambda_4} + \frac{Dy_1}{2 \cdot \lambda_1}}$$

In order to avoid numerical oscillations the CRANK-NICHOLSON form of this equation has been used:

$$B = T_4^{(k+1)} \cdot (1 + a \cdot R_{4,i}) - a \cdot T_{14}^{(k+1)} \cdot R_{4,14} - a \cdot T_1^{(k+1)} \cdot R_{4,1} - a \cdot T_6^{(k+1)} \cdot R_{4,6}$$

$$B = T_4^{k} \cdot (1 + a \cdot R_{4,i}) + a \cdot T_{fl}^{k} \cdot R_{4,14} + a \cdot T_1^{k} \cdot R_{4,1} - a \cdot T_6^{k} \cdot R_{4,6}$$

$$a = \frac{dt}{2 \cdot Dx_4 \cdot Dy_4 \cdot cp \cdot \rho}$$

Fluid nodes

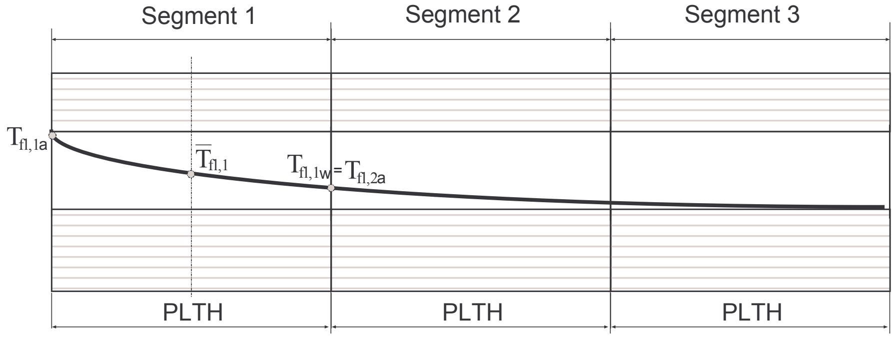


Fig. 4: Placement of the fluid nodes

For the fluid node 14 within the first segment of the hypocaust, the energy balance can be written as follows:

$$(T_{fl,\alpha} - T_{fl,w}) \cdot V \cdot c_{p,fl} \cdot \rho_{fl} = (T_{fl} - T_i) \cdot R_{fl,i}$$

Where

$$R_{4,14} = \frac{Plth \cdot Dy_4 \cdot Corrfactor}{\frac{Dx_4}{2 \cdot \lambda_4} + \frac{1}{\alpha_{fluid}} + \frac{wallthickness}{\lambda_{Wall}}}$$

Note that $T_{fl,in}$ is considered as known: for the first segment it is the fluid inlet temperature, for the following segments it is the outlet temperature of the previous segments. The solution of the equation is not written down here.
Alfa,Fluid is now calcualated depending on the water speed w.

$$\alpha_{fluid} = 2040 \cdot (1 + 0.015 \cdot t_{fluid,av}) \cdot \frac{w^{0.87}}{(da_{pipe} - 2 \cdot wallthick)^{0.13}}$$

with:

$$w = \frac{mdot}{\rho_W \cdot A \cdot 3600} \qquad A = (da_{pipe} - 2 \cdot wallthick)^2 \cdot \frac{\pi}{4}$$

and
da,pipe = pipe outside diameter
t_fluid,av = average fluid temperature

2.6 Boundary conditions

2.6.1 Boundary conditions on the room surfaces

There are two surfaces of the panel with two boundary conditions to the rooms. The panel can be situated horizontal as a floor respectively ceiling or vertical as a wall. The placement (vertical / horizontal) is defined by means of parameter NWALL (see TYPE description).

The boundary condition defines the heat transfer between the surface of the hypocaust and the room. Because of the two alternative TYPES for rooms (TYPE 19 and TYPE 56) who handle the heat transfer in different ways, there is a difference in the INPUTs and PARAMETERs.

2.6.2 TYPE 19

Radiation gains Q_{absi} absorbed by the hypocaust surface i (due to solar radiation, lightening equipment and peoples) are taken into account by means of the equivalent temperature:

$$T_{eq,TYPE19,i} = T_{air} + \frac{Q_{abs,i} + \alpha_{rad,i,j} \cdot (T_{s,j} - T_{s,i})}{\alpha_{conv,i}}$$

Where:

T_{air}	= Air temperature in the zone
α_{conv}	= Convective heat transfer coefficient (calculated within TYPE 360)
α_{rad}	= Radiation heat transfer coefficient
T_s	= Surface temperature

The heat flux on the surface is then calculated as:

$$Q_{floor} = A_{floor} \cdot \alpha_{conv,floor} \cdot (T_{Floor} - T_{eq,TYPE19,i})$$

Using TYPE 19, the equivalent temperature T_{eq} can easily be obtained as an optional OUTPUT of this TYPE.

2.6.3 TYPE 56

Using TYPE 56, the equivalent temperature has to be calculated as:

$$T_{eq,TYPE56,i} = Tstar + \frac{Q_{absi}}{A_i \cdot \alpha_{comb,i}}$$

$$\alpha_{comb,i} = \frac{Qcomb}{(T_{s,i} - Tstar) \cdot A_i}$$

Where:

Q_{absi}	= solar and internal radiative gains to surface (NT21 Output of TYPE56)
α_{eq}	= equivalent (combined) heat transfer coefficient (calculated within this TYPE)
Q_{comb}	= combined convective and radiative heat flux (NT19 Output of TYPE 56)
T_{star}	= Star network temperature of the Zone:(NT23 Output TYPE 56)
T_s	= Surface temperature: (NT17 Output TYPE 56)

The heat flux on the surface is then calculated as:

$$Q_{floor} = A_{floor} \cdot \alpha_{comb,floor} \cdot (T_{Floor} - T_{eq,TYPE56,i})$$

Convective heat transfer:
For horizontal surfaces the convective heat transfer coefficient for heating is calculated by:

$$\alpha_{conv,floor} = 2 \cdot (T_{Floor} - T_{air})^{0.31}$$

$$\alpha_{conv,ceiling} = 1.08 \cdot (T_{Floor} - T_{air})^{0.31}$$

and for cooling the heat transfer is calculated:

$$\alpha_{conv,floor} = 1.08 \cdot (T_{Floor} - T_{air})^{0.31}$$

$$\alpha_{conv,ceiling} = 2 \cdot (T_{Floor} - T_{air})^{0.31}$$

If PAR(9)=NWALL is set to –1 or –2 6 additional INPUTs have to be specified. This results in the following equations:

$$\alpha_{conv,floor} = Kfloor \cdot (T_{Floor} - T_{air})^{EFloor}$$

$$\alpha_{conv,ceiling} = KCeiling \cdot (T_{Ceiling} - T_{air})^{ECeiling}$$

For vertical surfaces the convective heat transfer coefficient is calculated as:

$$\alpha_{conv} = 1.5 \cdot (T_s - T_{air})^{0.25}$$

and

$$\alpha_{conv,vertical} = KVertical \cdot (T_{Wallsurface} - T_{air})^{EVertical}$$

respectively.
The user is able to use coefficients which can be found in literature or by measurements. Internally, the heat transfer coefficient is set to 1 W/m²K, if the temperature difference gets to small.

2.7 Initial condition:

The initial condition of the simulation is determined when the TYPE is called for the first time. The model calculates the steady state solution for the conditions of the first hour of the simulation. The temperature distribution of the steady-state condition (and hence the energy stored with this temperature distribution) is used as the starting point for the simulation.

The user has therefore to bear in mind that the selection of the data (i.e. boundary conditions of the room, inlet temperature of fluid etc.) of the first hour determines strongly the starting condition of the hypocaust / floor.
A volume flow rate > 0 has to be defined for the initial condition and as well by PAR (12) the initial start temperature of the panel.

2.8 Stability and accuracy problems

Internal time steps
The TYPE uses an internal time step which is usually shorter than the TRNSYS time step (typically 0.25 h - 1h) The internal time step is a parameter to be specified by the user,. The criteria for its choice is the type of input data and the purpose of the simulation **in general the recommended internal time step is 600 to 900 seconds**.

The model checks the possibility of numerical oscillations and reduces the length of the time step if necessary. This can occur after the flow has been switched off. In such a case, the time original value once the danger of oscillation is averted - usually in the next hour.
The accuracy of the model is a little influenced by the time step. Too small time steps (smaller than 10 sec) can however lead to round up errors.

2.8.1 Number of segments

The optimum choice of the number of segments causes is more difficult for the user. The maximum of the segments is set to 20 (which can be modified by the user in the file type360_mod.for - see appendix C). The accuracy increases with number of segments. However, form certain number of segments the gained in accuracy is marginal and not adequate to the increased computing time. Fig. 5 shows the outlet temperature for different number of segments.

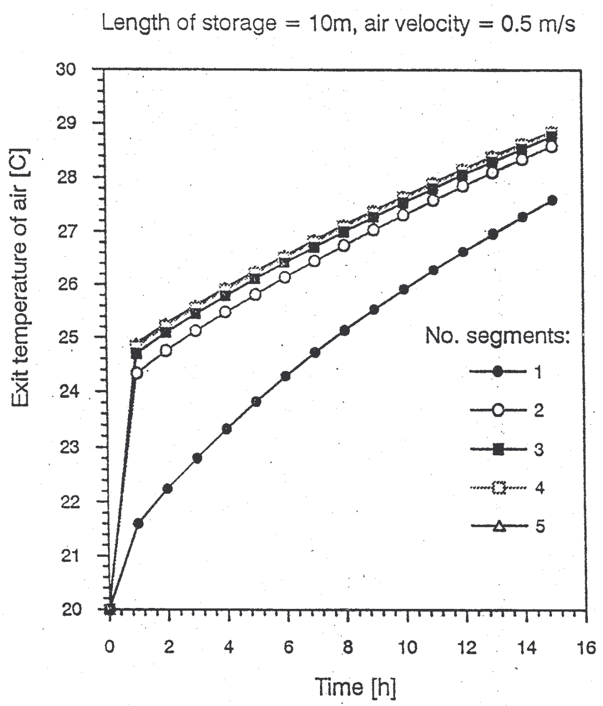


Fig. 5: Influence of the number of segments on the accuracy

It has to be recognised, that the main parameters (i.e. air velocity, length of the storage, thermal resistance between the fluid and the surface) determine the number of segments required for the desired accuracy.

2.8.2 Recommendations for the choice of accuracy influencing parameters

2.8.2.1 General

No universal recommendations can be given. it remains within the responsibility of the user to determine the optimum combination of parameters. A sensitivity study shall be made prior to fix the parameters for simulation.

2.8.2.2 Time step

For TRNSYS time step 0.25, 0.5 or 1 h. TYPE 360 time step 600 to 900 seconds.

2.8.2.3 Number of segments:

For typical residential buildings with Length of the storage of 6 - 8 m and forced air flow of 0.5 - 1 m/s the recommended number of segments are 2 - 3. For lower air velocities of longer channels the number of segments shall be increased.

2.8.2.4 Number of nodes

Small nodes shall be used in the region of high temperature gradient (close to the channel wall).

3 Implementation into TRNSYS

This chapter describes how the user shall utilise the model within the TRNSYS program. The following necessary steps has to be performed:

1. Create the geometrical data file of your hypocaust / floor heating which represents your wall / floor
2. Crate your building file (*.bui) including the wall/floor heating systems.
   Define NTYPEs 17 19 21 23
3. Create your TRNSYS Input file (*.dck) and connect the TYPE 56 OUTPUTs to TYPE 360 resp. TYPE 360 OUTPUTs to TYPE 56.

3.1 Definition of the geometry

This section describes how the distribution of nodes, the definition of neighbours and the material constants are made.

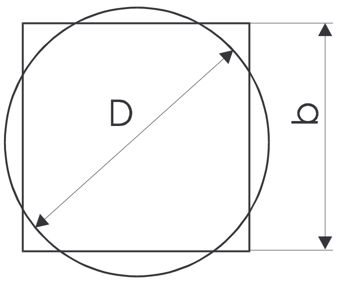
- Make a sketch of the representative section
- Decide on simplifications. The most common simplification, which has to be made is the model of a non-rectangular channel. The problem is the discrepancy between the model and the reality in cross sectional flow area and circumference (respectively heat transfer area)

model flow area = real flow area

In order to satisfy also the heat transfer conditions a correction factor has been introduced:

correction factor = circumference real / circumference model

Thus for the common case of a circular channels:



$$\frac{\pi \cdot D^2}{4} = b^2 \Rightarrow Corrfactor = \frac{\sqrt{\pi}}{2} = 0.886$$

Fig. 6: Calculation of the correction factor

- Distribute nodes - Definition of mesh Draw now the mesh of nodes over the simplified representative area using the following rules:
  - the maximum number of nodes is 50
  - use smaller nodes in the vicinity of a channel
  - make sure that all the fluid nodes have the same flow i.e. have the same cross sectional area
  - it is recommended to use a mesh with the same width of all does in a column and the same height of all nodes in a row

Then, number the nodes using the following rule:
  - the fluid nodes have to be the last nodes
  - for horizontal panels: neighbours above the channel should have lower numbers than neighbours below the channel

- Create a file with all required information (recommended *.fbh). This file contains:
  - the definition of the relative location of all nodes
  - the dimensions of the nodes
  - the material constants

This file has to be assigned in the TRNSYS input file (*.dck) to a logical unit number, e.g.

ASSIGN geofile.fbh 56
(File name logical unit number)

For more details see the example.

In the following, the definition of the records will be explained. The format is not any more necessary with this version of TYPE 360 because with PAR (13) it can be chosen if the geometrical file should be read in formatted or not. But it is recommended to keep the structure of the file like it is displayed. The only difference is a blank between the values.

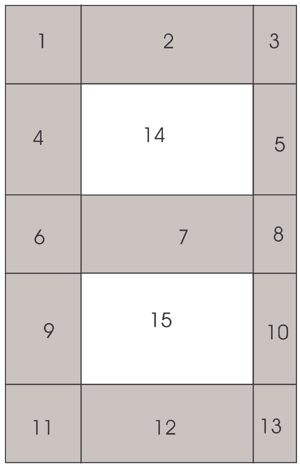


Fig. 7: Example of geometry and mesh of nodes

Record 1 (= Line 1)
Number of nodes N / Last solid node / Wall thickness of tube *(optional:Format: I4 / I4 / F10.0)*
20 19 0.0010

Definition of all nodes: Record No: (3*N + 1)
Each node is defined by means of three records:
First record per node ; number of the node

First record per node: number of the node
1 (Format I2)

Second record per node: Definition of neighbours
Note following rules:

neighbour is upper surface	100
neighbour is lower surface	101
neighbour is horizontal direction	**-** sign
neighbour is vertical direction	**+** sign

up to 7 neighbours can be defined
the last number is the number of neighbours
Format 8I3

Example: the definition of neighbours of node 1:
100 4 -2 0 0 0 0 3
Example: the definition of neighbours of node 7
14 -6 15 -8 0 0 0 4

Example: the definition of neighbours of node 15
7 -9 12-10 0 0 0 4

Third record per node: Definition of nodes; material, dimensions
Density*specific heat [J/m³ K] [1) 2)], conductivity [W/mK] [3)], dx [m], dy [m]
(optional: Format 4F10.0)
Example for Node 7
2108400.0 2.10 0.09 0.0413

Example for Node 15 (here: last node)
1196.0 0.2 0.0900 0.058

Notes:
1) Because the density and the specific heat of the air varies with temperature, both values shall be taken at the average expected temperatures of the air.
2) Based on the product of density and specific heat of the fluid the model will treat the fluid as gas air or as water.
3) In the special case of the fluid node, the second number is for the conductivity of the tube wall

Example of the entire file for the hypocaust shown in figure 7:

```
  15   13  0.0
 1
100  4 -2  0  0  0  0  3
2108400.0     2.10      0.04      0.0413
 2
100 -1 14 -3  0  0  0  4
2108400.0     2.10      0.09      0.0413
 3
100 -2  5  0  0  0  0  3
2108400.0     2.10      0.02      0.0413
 4
  1  6-14  0  0  0  0  3
2108400.0     2.10      0.04      0.0580
 5
  3-14  8  0  0  0  0  3
2108400.0     2.10      0.02      0.0580
 6
  4  9 -7  0  0  0  0  3
2108400.0     2.10      0.04      0.0413
 7
 14 -6 15 -8  0  0  0  4
2108400.0     2.10      0.09      0.0413
 8
  5 -7 10  0  0  0  0  3
2108400.0     2.10      0.02      0.0413
 9
  6 11-15  0  0  0  0  3
2108400.0     2.10      0.04      0.0580
10
  8-15 13  0  0  0  0  3
2108400.0     2.10      0.02      0.0580
11
  9101-12  0  0  0  0  3
2108400.0     2.10      0.04      0.0413
12
 15-11101-13  0  0  0  4
2108400.0     2.10      0.09      0.0413
13
 10-12101  0  0  0  0  3
2108400.0     2.10      0.02      0.0413
14
  2 -4  7 -5  0  0  0  4
   1196.0     0.2       0.0900    0.058
15
  7 -9 12-10  0  0  0  4
   1196.0     0.2       0.0900    0.058
```

3.2 Check of the geometry

In section 4.1 the geometry of the representative cross section has been defined by means of the above described file. This procedure is susceptible to errors. Therefore, a stand-alone program „GEOVIEW.EXE“ is attached. By means of this program the user can check weather the input of the geometry has been made correctly. The program „geoview.exe“ is described in appendix A.

3.3 Coupling to the building

Within TRNSYS two different TYPES for building simulation are available, either TYPE 19 - single zone or TYPE 56 multizone building.
These two TYPES model the internal heat transfer within the two zones (rooms) in a different way. Therefore, the coupling of the TYPE 360 requires an exchange of different variables between TYPE 19 or TYPE 56.

3.3.1 Linking TYPE 360 to TYPE 19

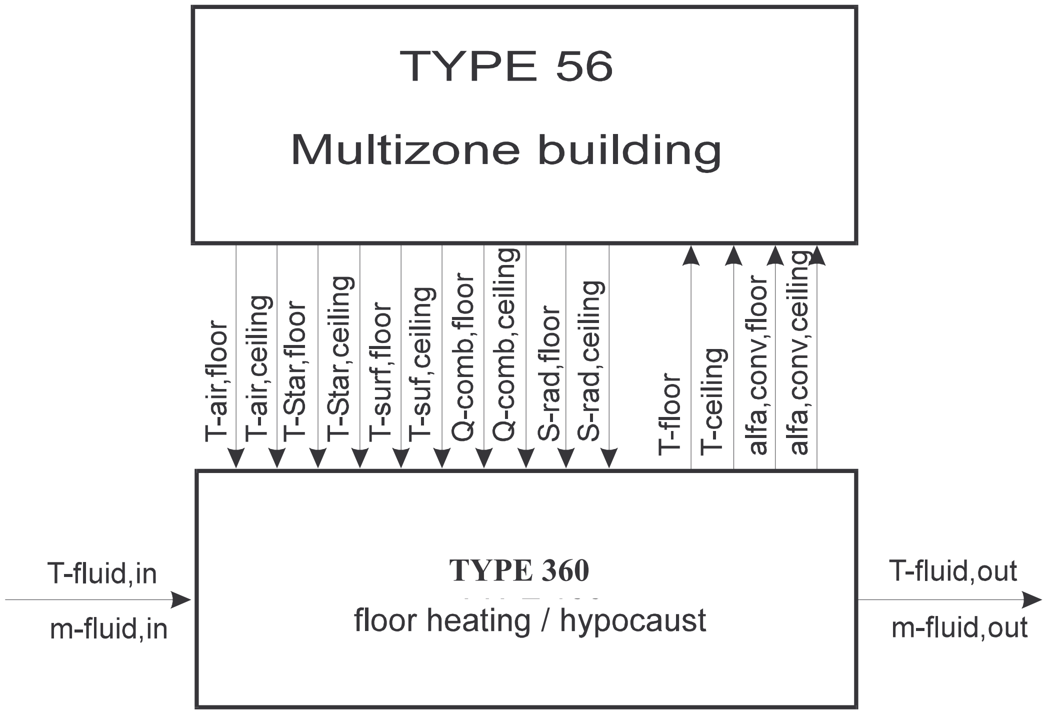
The TYPE has to be implemented as conduction for a „NON-ASHRAE Wall“. This means a wall is not modelled by using transfer functions. The energy conduction at the inside surface of this wall must be provided as an input to the TYPE 19 (For a detailed description see TRNSYS manual).
Specification of a „NON-ASHRAE Wall“ in TYPE 19:

Parameter	Description:	
1	Surface number	1 - 15
2	Surface Type	Specify 4
3	Surface Area	Area [m²]
4	Reflectance of inner surface for solar	[-]
5	inside convection coefficient	[kJ/m²K]

Inputs:

1	Energy transfer into zone at the inside surface of the wall [kJ/h]	

3.3.2 Linking TYPE 360 to TYPE 56



Linking TYPE 360 to the TYPE 56 a (heating) wall with boundary condition have to be defined. This wall get the surface temperature calculated by TYPE 360. The *wall type* has to be defined as a RESISTANCE with an HFRONT as an INPUT and an fictive HBACK =0.0001.
By specifying the necessary NTYPES within TYPE 56 *.BUI file (see below) the correct information can be connected to the TYPE 360 (For more details see also exampl56).

The necessary NTYPES are:

TYPE 56 - OUTPUTS ⇒ NTYPE:	17	= T_{wall}
	19	= $Q_{comb,s,i}$
	21	= $Q_{,absi}$
	23	= T_{star}

3.3.3 TYPE Description

PARAMETER NO.			DESCRIPTION
1	Mode	-	1 for Floor Heating 2 for Hypocaust
2	Width	-	Total width of panel [m] (see Fig 1 & 2) Important Note: It is essential, that in case of a Hypocaust the WIDTH of the panel (Par (2)) is equal to the product of the width of the cross section of the Hypocaust (DXTOTF, defined in the Geometry file) multiplied by a whole (integer!) number (NU_SECTIONS). Otherwise the simulation stops with warning: 'WARNING HYPOCAUST DEFINITION: NU_SECTIONS*DXTOTF IS NOT EQUAL WIDTH'.

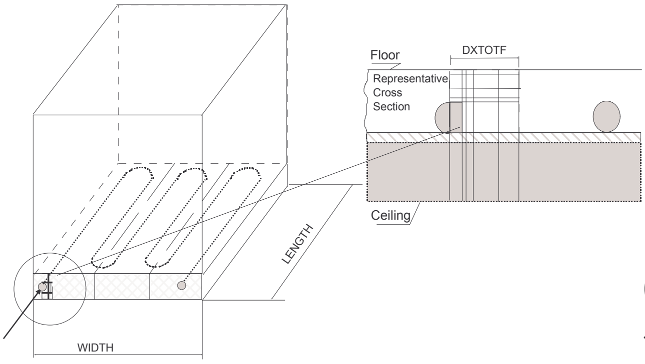


Fig. 1: Floor heating

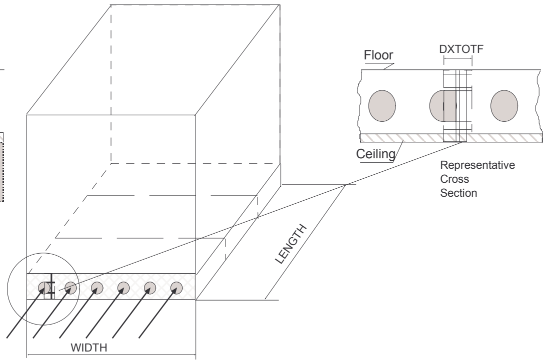


Fig. 2: Hypocaust

3	NSEG	-	Number of segments (in Fig. 1 & 2 is NSEG = 3). Maximum number of segments is set to fifty (50). However, this value can be modified in the file "TYPE360_mod.for".
4	LENGTH	-	Total length of panel [m] (see Fig 1&2)
5	CORR_FACT	-	Correction factor 0.886 for round channel, 1 for rectangular channel
6	DT	-	Internal time step [sec] This TYPE uses an internal time step which is usually shorter than the TRNSYS time step. In general a time step of 900 sec is used.
7	LU	-	Logical Unit No. to which the geometry file is assigned

8	NTYPE	-	to which type is TYPE 360 connected ? TYPE 360 is compatible to both important TRNSYS models of building: TYPE 56 and TYPE19. Because these two models use different algorithms the transfer of variables between the panel and the zone is for both cases different (see separate paragraphs). The user specifies in the parameter list of TYPE360 to which zone is the model connected: Par (8) = 19 connection to TYPE19 Single Zone Par (8) = 56 connection to TYPE56 Multi Zone (No equations are needed for connection to TYPE56 with the Version 3.0)
9	NWALL	-	horizontal or vertical application = 1 Vertical (Murocaust if MODE 2 is chosen) = 2 Horizontal (Floor heating , Hypocaust) If NWALL is set to a negative value (-1 or –2) 6 additional INPUTs has to be specified.
10	NTR_MON	-	Shall all segments be monitored ? For NTR_MON>0 gives average temperatures, and heat fluxes for all segments as optional outputs (see also output list)
11	ROUGHNES	-	roughness [m] Roughness of the channel is utilized for the calculation of pressure drop of air flow within the Hypocaust channel. In case of floor heating no pressure drop calculation is made.
12	TPANINI	-	Initial temperature of panel [C] The temperature of the panel at the begin of the simulation is assumed to be homogeneous for the entire panel
13	NFORMAT	-	Formatted/Unformatted read: = 1= Formatted read 0= Unformatted read

if imode=2 (hypocaust mode)

14	dhydr	-	hydraulic diameter $d_{hydr} = \frac{2 \cdot a \cdot b}{a + b}$ necessary for hypocaust systems

if connected TYPE19 Par (8) = 19

INPUT NUMBER		DESCRIPTION
1	Tfalfa	Inlet temperature of the fluid [C]
2	FlowMass	Mass flow of the fluid for the entire panel [kg/h]
3	Tair_floor	Air temperature for the side Floor [C]
4	Tair_ceiling	Air temperature for the side Ceiling [C]
5	Teq_floor	Equivalent temperature for floor [C]
6	Teq_ceiling	Equivalent temperature for ceiling [C]
7	not used	enter any value
8	not used	enter any value
9	not used	enter any value
10	not used	enter any value
11	not used	enter any value
12	not used	enter any value
13	DENSITY	Fluid density [kg/m^3]

6 Optional INPUTs (can be used if NWALL =-1 or NWALL =-2)

14	K_floor	coeff. for calculating Alfa,conv (see section 2.6.3) Floor Surface
15	E_floor	exponential coeff. for calculating Alfa,conv Floor Surface
16	K_Ceiling	coeff. for calculating Alfa,conv (see section 2.6.3) Ceiling Surface
17	E_Ceiling	exponential coeff. for calculating Alfa,conv
18	K_Vertical	coeff. for calculating Alfa,conv (see section 2.6.3) Vertical surface
19	E_Vertical	exponential coeff. for calculating Alfa,conv Vertical surface

if connected to TYPE56 Par (8) = 56

INPUT NUMBER		DESCRIPTION
1	Tfalfa	Inlet temperature of the fluid [C]
2	FlowMass	Mass flow of the fluid for the entire panel [kg/h]
3	Tair_floor	Air temperature for the side Floor [C]
4	Tair_ceiling	Air temperature for the side Ceiling [C]
5	Tstar_floor	Star node temperature of zone (NYPE 23) [C]
6	Tstar_ceiling	Star node temperature of zone (NYPE 23) [C]
7	Tsurf_floor	Surface temperature for floor (NYPE 17) [C]
8	Tsurf_ceiling	Surface temperature for ceiling (NYPE 17) [C]
9	Qcomb_floor	Combined heat flux for floor (NYPE 19) [kJ/h]
10	Qcomb_ceiling	Combined heat flux for ceiling (NYPE 19) [kJ/h]
11	Qabsi_floor	solar and internal radiative gains on floor (NYPE 21) [kJ/h]
12	Qabsi_ceiling	solar and internal radiative gains on ceiling (NYPE 21) [kJ/h]
13	DENSITY	Fluid density [kg/m^3]

6 Optional INPUTs (can be used if NWALL =-1 or NWALL =-2)

14	K_floor	coeff. for calculating Alfa,conv (see section 2.6.3) Floor Surface
15	E_floor	exponential coeff. for calculating Alfa,conv Floor Surface
16	K_Ceiling	coeff. for calculating Alfa,conv (see section 2.6.3) Ceiling Surface
17	E_Ceiling	exponential coeff. for calculating Alfa,conv
18	K_Vertical	coeff. for calculating Alfa,conv (see section 2.6.3) Vertical surface
19	E_Vertical	exponential coeff. for calculating Alfa,conv Vertical surface

OUTPUT NUMBER		DESCRIPTION
1	Tout	Outlet temperature of the fluid [C]
2	Mass Flow	Mass flow of the fluid for the entire panel [kg/h]
3	Taverage	Average temperature of the panel [C]
4	T_floor	Average mean temperature of the floor [C]
5	T_ceiling	Average mean temperature of the ceiling [C]
6	Qfluid	Heat input into the panel from the fluid [kJ/h]
7	Q_floor	Heat output through the floor [kJ/h]
8	Q_ceiling	Heat output through the ceiling [kJ/h]
9	Estored	Energy stored in panel at end of timestep [kJ]
10	Error	Relative error of the heat balance [%] =(Diff.energybalance)*100/(Qfluid+Qfloor+Qceil)
11	Alfa_conv_floor	Convective heat transfer coefficient on the floor surface [kJ/hm ^{2}K]
12	Alfa_conv_ceil	Convective heat transfer coefficient on the ceiling surface [kJ/hm ^{2}K]
13	Dp	Pressure loss in air channel [Pascal]
14	Velocity	Velocity in channel/tube [m/s]
15	A_channel	Cross sectional area of (half) channel as defined in Geometry file [m ^{2}]
16	Area	surface area of hypocaust /floor [m ^{2}]
17	alpha_fl	heat transfer coeff. of fluid to inner channel wall [W/m²/K]

Optional Outputs when Par(10)=NTR_MON>0:

18	Tfloor_segment $_1$	Floor temperature of segment 1 [C]
19	Tceil_segment $_1$	Ceiling temperature of segment 1 [C]
20	q_floor_segment $_1$	Specific heat flux on the floor surface [W/m ^{2}]
21	q_ceil_segment $_1$	Specific heat flux on the ceiling surface [W/m ^{2}]
22	Taverage_ segment $_1$	Average panel temperature of segment 1 [C]
23	Tfluid_av	Average fluid temperature of segment 1 [C]

6*(ISEG-1)+18	$Tfloor\_segment_{iseg}$	Floor temperature of segment ISEG [C]
6*(ISEG-1)+19	$Tceil\_ segment_{iseg}$	Ceiling temperature of segment ISEG [C]
6*(ISEG-1)+20	$q\_floor\_segment_{iseg}$	Specific heat flux on floor surface ISEG [W/m^2]
6*(ISEG-1)+21	$q\_ceil\_segment_{iseg}$	Specific heat flux on ceiling surface ISEG [W/m^2]
6*(ISEG-1)+22	$Taver\_ segment_{iseg}$	Average panel temperature of segment ISEG [C]

Apendix A GEOVIEW.EXE

Instruction on how to use the program geoview.exe

1) Copy the file geoview.exe on the hard disc.

2) Copy the file KPM150.DLL into the windows directory (i.e. C:\WINDOWS)

3) - start the program as a usual windows program
- load your geometry file with „Datei/öffnen“
- the geometry will be displayed if now error is enclosed
- if correction of the geometry is required, load the corrected file again
- the program stops if a format error is detected. In this case you have to check your file and start again....

NOTE: With the release of this version of TYPE 360, it is possible to use either formatted or unformatted geometrical files (*.fbh). But the program geoview can only read formatted files. So, please make sure that the file you create is doubled checked if you like to use geoview.

Apendix B TYPE360_mod.for

Due to all the changes the include file Type360.inc has been replaced by the file

Type360_mod.for

This is a new feature in FORTRAN 90 for an better program style. Nevertheless you can also modify the maximum number of segments (NSEG) within this file. Also the number of nodes (NNOD) of the geometry file (*.fbh) can now be modified as well as the number of Fluid nodes (NNFL; (this is only possible for imode=2 hypocaust systems). The maximum number of Nodes is limited to 99.

Don’t forget to rebuild your DLL afterwards!

Apendix C LITERATURE

References:

[1] K.Fort:
'Dynamisches Verhalten von Fussbodenheizungen'
Diss. Nr. 8893, Eidgenössische Technische Hochschule Zürich, 1989

[2] K.Fort ,P.Gruber, P.Hartmann, A.Weber
'Optimaler Betrieb von Fussbodenheizungen'
Forschungsbericht im Auftrag des Bundesamtes für Energiewirtschaft, Bern, 1993

[3] K. Fort, W.Gygli
'TRNSYS-Model TYPE60 for Hypocaust Thermal Storage and Floor Heating'

[4] Glück, "Wärmetechnisches Raummodel", C.F. Müller Verlag, Karlsruhe, pp 66- 67, 1997

The type is available at:

TRANSSOLAR Energietechnik GmbH
Technologien zur rationellen Energienutzung und Solartechnik
Curiestraße 2
D-70563 Stuttgart
Fax: +49/ 711 / 67976 - 11
e-mail: software@transsolar.com
http://www.transsolar.com