

Part 3:

Tutorial

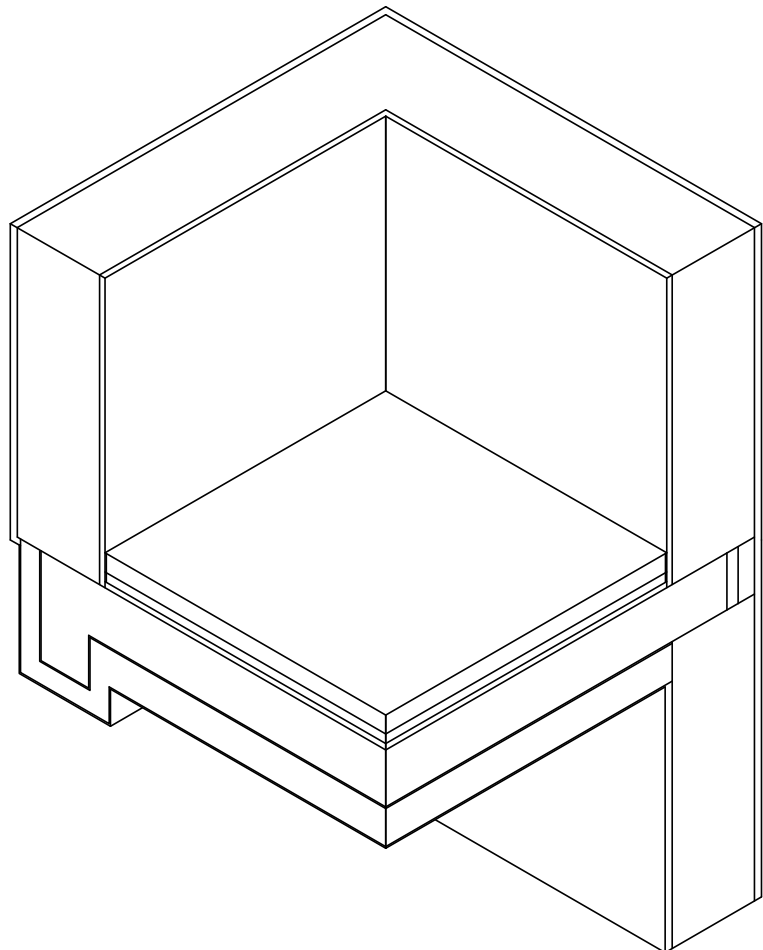
3.1. General Overview

The program branch operations explained in the last part of this manual, **Using WAEBRU**, shall be illustrated here with step-by-step instructions for analysing a demonstration object. The procedures of input and evaluation can be followed as described to generate and evaluate two- and three-dimensional input models equivalent to the examples contained on the demonstration disks (**TUTOR2** and **TUTOR3**). These completely calculated model files, which can be perused before working with WAEBRU directly, also serve as an object of comparison and aid towards successful completion of the model as described in the following.

3.1.1.

Object of Analysis

The demonstration object to be modelled and evaluated encompasses the corner region of a masonry and concrete construction over an exterior space (e.g. carport). As illustrated in the axiometric rendering below, the masonry bearing wall is bonded with the concrete floor slab along the side which flanks the exterior space underneath, and supported by a girder spanning an opening in the other direction.



demonstration object

The standard masonry wall construction is a total of 35.5 cm thick, composed of (from the exterior):

2.5 cm	stucco
30.0 cm	masonry

2.0 cm interior plaster

In the region flanking the floor slab, the masonry wall is substituted by:

2.5 cm stucco
 6.0 cm perimeter brick
 4.0 cm perimeter insulation
 (floor slab)

The reinforced concrete girder supporting the other side of masonry is encased in 7.0 cm of standard insulation.

The floor construction over the exterior space is built up to a total thickness of 41.5 cm with:

0.5 cm exterior plaster
 12.0 cm insulation
 0.5 cm adhesive mortar
 17.5 cm reinforced concrete
 2.0 cm sand cushion
 3.0 cm isolating underlayment*
 1.0 cm embedded heating assembly (concrete)*
 5.0 cm concrete topping*

*- expansion strip along outer perimeter (0.4 cm)

Despite extensive exterior insulation, such a prominent corner situation could prove to be a critical thermal bridge. A potential heat source has also been included to simulate an electrical floor heating assembly in the concrete topping. The efficiency of such a heating alternative in this exposed location can thus be assessed as well.

3.1.2.

Stages of Analysis

The most critical cross-section of the object under consideration is through the wall/floor connection at the girder. This section shall be modelled and evaluated first in the two-dimensional mode of WAEBRU. The results obtained here would be valid for the region of the thermal bridge in which heat flow can be expected to occur strictly parallel to the section plane, i.e. at least approximately 1 m from the corner. Instructions for this application are provided in the next chapter, **Two-dimensional Analysis**.

In order to obtain a picture of the temperature distribution in the apparent thermal bridge at the outer corner of the construction, a three-dimensional model of the situation must be generated for evaluation. The procedures of this stage are treated in the chapter on **Three-dimensional Analysis**.

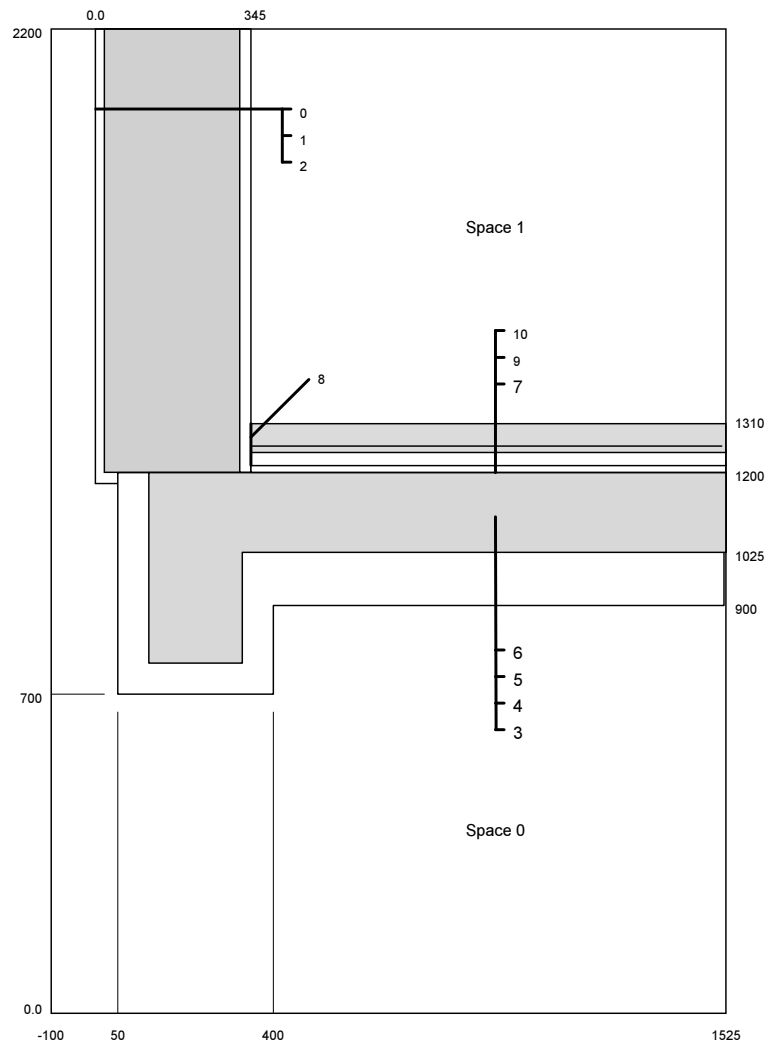
To facilitate input in three-dimensional mode, the previously generated two-dimensional model can serve as the first layer (component group) and reference basis for further component group input. The transformation of an object from two to three dimensions by means of the input branch of WAEBRU is outlined in the section on **Generating the Model - Input** in Chapter 3.3.

3.2. Two-dimensional Analysis

3.2.1. Modelling the Object

Before actually working with WAEBRU, a sketch of the object should be prepared in which relevant data is summarised for model input. Such a sketch of the two-dimensional demonstration object would include the information shown below: dimensions, reference coordinates, and material properties.

model preparation



	material	λ [Wm ⁻¹ K ⁻¹]
0	stucco	0.800
1	masonry	0.164
2	interior plaster	0.700
3	exterior plaster	0.800
4	insulation	0.041
5	adhesive mortar	0.271
6	reinf. concrete	2.200
7	sand	1.000
8	expansion strip	0.150
9	isol. underlayment	0.040
10	concrete topping	1.200

	space	α [Wm-2K-1]
0	exterior	25
1	interior	6

Since negative coordinate values are allowed, the positioning of the graph origin ($x=0$, $y=0$) is generally arbitrary. The reference coordinates shown here, however, are the ones consistent with the geometric input data as related in the next section.

3.2.2.

Generating the Model - Input

The program is called up with the run command **waebru**, whereupon the file survey screen appears for input of a filename for the new project, e.g. **tutor2**. Upon pressing **Enter**, a message appears, informing the user that this file does not yet exist, and asking if the input branch should be executed.

Confirmation with **y** ('yes') leads to the project description screen for entering text information identifying the current project, e.g.

PROJECT DESCRIPTION

TUTORIAL - 2-dimensional:

Reinforced concrete slab with masonry bearing wall on girder over carport (+ exterior insulation and electric heating cable assembly embedded in concrete floor topping).

Note: Whatever text is typed here will be included at the head of program printouts (model documentation and numeric results). This screen can be accessed from the input branch at any time by pressing **Alt-T**; input here can be edited using the usual text field functions (see **List of Key Functions**, Appendix C).

Confirm with **F9** or **Enter** from the last line to move on to the Main Screen.

The first element to be entered is the exterior space. The boundaries of element 1 shall demarcate the full extent of the model with

```
X_1:  -100
X_2:  1525
Y_1:    0
Y_2:  2200
```

Enter the element type: s ('space').

To specify a transfer coefficient for the suggested identification number 0, press **Alt-M** and enter

```
ID#:0   $\alpha$ :25  Name:Ext.transfer coeff.
```

Finally, the space numbers must be defined in the last input fields:

```
Space#:0  sub-#:0
```

Once these numbers are confirmed, element 1 is complete and the screen is ready for input of element 2 (see status fields of upper screen; mode should be **Add new element**).

Now it would be worthwhile to re-scale the control graph for a better view of the component during input of successive elements: press **Alt-V**, activate the option **Full Scale**, and confirm the suggested graph ranges with **F9** to re-draw.

Note: The view menu (**Alt-V**) can also be used to select the object level displayed in the name field above the control graph. The standard setting is aggregate display. To specify a name for the aggregate (entire model), press **Alt-N** and move the cursor to the corresponding field for text input. To name the current component or group, activate the view menu option **Component Display** or **Group Display** first; then move the cursor into the name/status fields with **Alt-N** for name input.

Before setting out to build up the material component, some thought should be given to the sequence of element input so as to best utilise the advantages of *element overlapping*. To illustrate this principle, the demonstration example described here has been organised into construction parts of predominantly parallel material layers. This simplifies coordinate input substantially.

The first part of the aggregate to be entered is the wall structure. An "onion skin" sequence begins at the outermost layer (stucco) with element 2 -

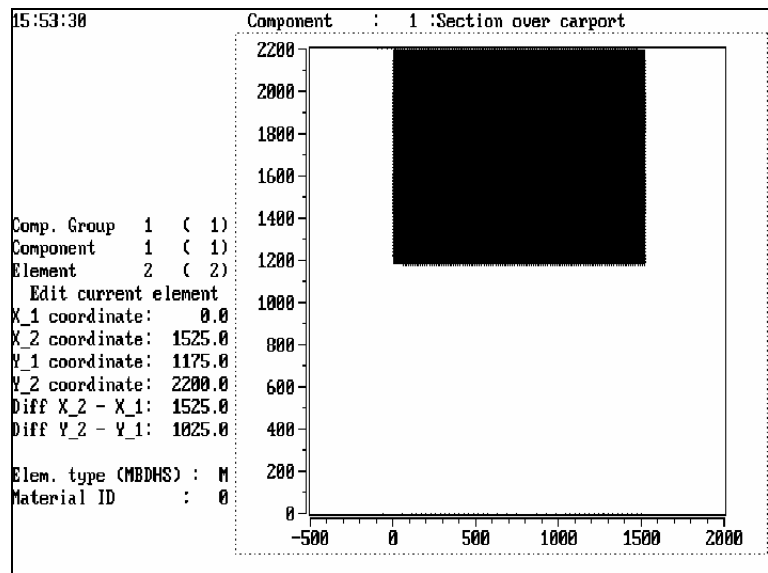
```
X_1:   0
X_2: 1525
Y_1: 1175
Y_2: 2200
```

Specify the element type with **M** ('material') and then press **Alt-M** to enter the first building material:

```
ID#: 0    λ: 0.8    Name: Stucco
```

The control graph should now look like this:

input - element 2



Note: The display here shows the Main Screen in edit mode with element 2 confirmed. It is advisable to switch from add/insert to edit mode (**edit current element**) with **Alt-I** whenever it is necessary to page back elements for correction. Otherwise, a new element is generated with every confirmed correction (**Insert new element**). To avoid a proliferation of unnecessary elements, "proof read" the component sequence at least once, with **PgUp** and **PgDn**, before filing.

The next element is the masonry wall, which overlaps - and thereby displaces - all but a thin strip of stucco (25 mm to the left and bottom):

element 3 -

```
X_1:  25
X_2: 1525
Y_1: 1200
Y_2: 2200

elem. type :M
material ID:1
```

Press **Alt-M** -

```
ID#:1    λ:0.164    Name:Masonry wall
```

- and **Enter**.

The interior plaster, element 4, delineates the right side of the masonry. Here, only one coordinate of the set still visible from element 3 needs to be adjusted to leave 200 mm of masonry thickness:

element 4 -

```
X_1:  325
X_2: 1525
Y_1: 1200
Y_2: 2200

elem. type :M
material ID:2
```

Press Alt-M -

ID#:2 λ :0.7 Name:Interior plaster

- and Enter.

The next element is the "kernel" of this part of the aggregate: the interior space overlapping all but the total wall thickness (with extended strip of stucco) of the material entered thus far.

element 5 -

X_1: 345

X_2: 1525

Y_1: 1200

Y_2: 2200

elem. type:S

coeff. ID :1

Press Alt-M -

ID#:1 α :6 Name:Int.transfer coeff.

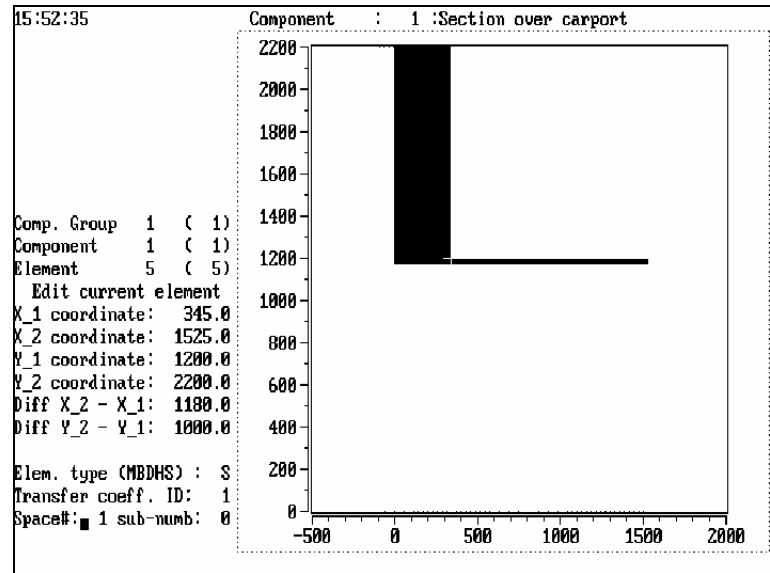
- and **Enter** the last input fields:

Space#: 1 sub-#: 0

to complete the element.

The control graph should now display the following figure of the standard masonry construction:

input - element 5



Note: The illustration above shows only the silhouette of the component in its current state.

The second part of the construction to be entered is the insulated girder, followed by the concrete floor slab with its exterior insulation. The final part of input covers the interior floor construction over the slab. These sequences are covered in the section on **Completing the Component** further on in this chapter.

Completing the Tables

With the over-all sequence of input in mind, the table of building materials can be completed in one uninterrupted input series. Once complete, the list of available material properties can be accessed with **F2** to facilitate material identification and input.

For the material table to be accessible, an element of type **m** must be currently displayed in the Main Screen. Switch to edit mode with **Alt-I** and page back to element 4 with **PgDn**. Now press **F2** to have a look at the material table generated thus far.

At this point, the table already contains three entries, numbered 0 through 2. Press **Esc** to return to the display of the current building material element (without making a selection).

To add the next material (3) to the list, press **Alt-A** and specify:

ID#: 3 λ : 0.8 Name: Exterior plaster

- **Enter**.

Material number 4 is subsequently appended and numbered automatically with the same procedure again:

Alt-A -

ID#:4 λ :0.041 Name:Insulation

- Enter.

These steps can be repeated as often as necessary to enter all the materials required for the model at hand. For the demonstration example, specify the remaining building materials as follows:

ID#:5	λ : <u>0.271</u>	Name: <u>Adhesive mortar</u>
ID#:6	λ : <u>2.2</u>	Name: <u>Reinf. concrete</u>
ID#:7	λ : <u>1</u>	Name: <u>Sand cushion</u>
ID#:8	λ : <u>0.15</u>	Name: <u>Expansion strip</u>
ID#:9	λ : <u>0.040</u>	Name: <u>Isol. underlayment</u>
ID#:10	λ : <u>1.2</u>	Name: <u>Concrete topping</u>

Any of the elements of type **M** can be subsequently identified using the material table: press **F2**, move cursor to mark desired line, and press **Enter** to select and return to Main Screen. To have a printout of the complete tables at hand as a reference during input, press **Alt-A** for the file menu and execute the option **Tables**.

The data associated with a particular ID number can be edited by pressing **Alt-M** from the Main Screen. This key function always accesses the material number that is specified for the element currently displayed. Individual entries can be deleted from the table screen directly (mark line and press **Alt-D**).

Note: The method described above for completing the materials table can also be used to generate the other types of tables (space and reinforcing bar) for later selection. The key function **Alt-A** automatically accesses the table appropriate for the currently displayed element type (**M/H**, **B/D**, or **S**).

Completing the Component

To move on to input of the remaining elements, page up to the last element generated thus far (in this case, element 5). If **PgUp** is pressed again from here while in edit mode, a message appears, informing the user that there are no more elements in this component, and asking if the mode should be switched to add/insert. Confirm a mode switch by answering with **I**. The screen should then appear ready for input of element 6, with the mode status line reading **Add new element**.

The next sequence of four overlapping elements describes the girder part of the aggregate:

elem. 6, ext. plaster -

X_1: 45
X_2: 405
Y_1: 695
Y_2: 1200

elem. type :M
material ID:3

elem. 7, insulation -

X_1: 50
X_2: 400
Y_1: 700
Y_2: 1200

elem. type :M
material ID:4

elem. 8, adh. mortar -

X_1: 120
 X_2: 330
 Y_1: 770
 Y_2: 1200

elem. type :M
 material ID:5

elem. 9, reinf. concrete -

X_1: 125
 X_2: 325
 Y_1: 775
 Y_2: 1200

elem. type :M
 material ID:6

Elements 10 through 13 describe the same sequence of materials for the insulated concrete floor slab construction connected to the girder:

elem. 10, ext. plaster -

X_1: 400
 X_2: 1525
 Y_1: 895
 Y_2: 1200

elem. type :M
 material ID:3

elem. 11, insulation -

X_1: 400
 X_2: 1525
 Y_1: 900
 Y_2: 1200

elem. type :M
 material ID:4

elem. 12, adh. mortar -

X_1: 400
 X_2: 1525
 Y_1: 1020
 Y_2: 1200

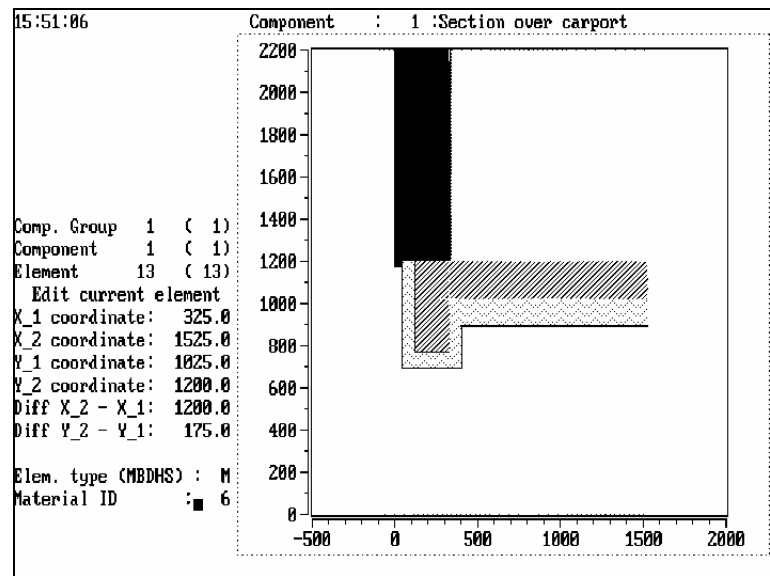
elem. type :M
 material ID:5

elem. 13, reinf. concrete

X_1: 325
 X_2: 1525
 Y_1: 1025
 Y_2: 1200

elem. type :M
 material ID:6

input - element 13



The interior flooring over the concrete slab is built up with the last five elements of the aggregate:

elem. 14, sand -

X_1: 345
 X_2: 1525
 Y_1: 1200
 Y_2: 1310

elem. 15, exp. strip -

X_1: 345
 X_2: 1525
 Y_1: 1220
 Y_2: 1310

```
elem. type :M
material ID:7
```

```
elem. type :M
material ID:8
```

```
elem. 16, isol. layer -
```

```
X_1: 349
X_2: 1525
Y_1: 1220
Y_2: 1310
```

```
elem. type :M
material ID:9
```

```
elem. 17, concrete (h.s.) -
```

```
X_1: 349
X_2: 1525
Y_1: 1250
Y_2: 1310
```

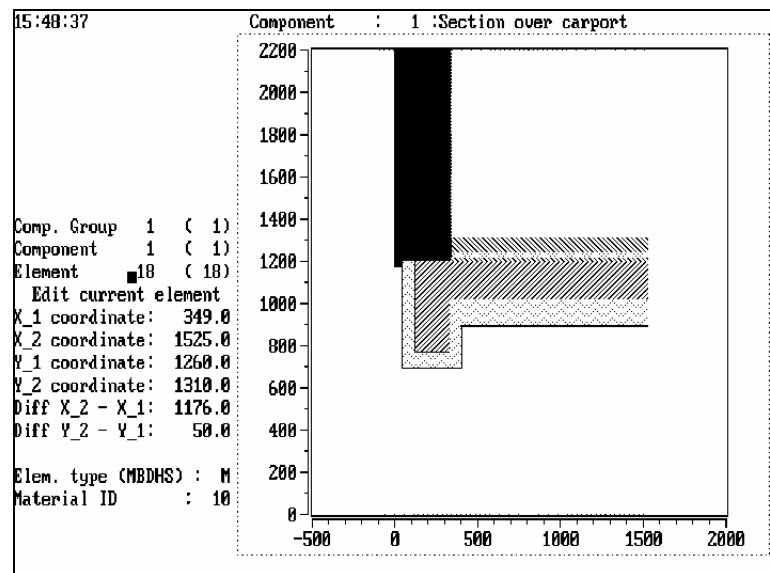
```
elem. type :H
material ID:10
```

```
elem. 18, concrete topping -
```

```
X_1: 349
X_2: 1525
Y_1: 1260
Y_2: 1310
```

```
elem. type :M
material ID:10
```

input - complete aggregate



Aggregate File

The geometric and material data of the model must now be saved in a construction file. This is done via the file menu, which is called up from the Main Screen with **Alt-F**.

Upon activating the option **Write aggregate**, the file survey screen appears for input of a *filename*. The model name (e.g. **tutor2**) given at the start of input is shown here automatically. It can be changed if desired, or simply confirmed as is with **Enter**.

After confirming the message **Write onto this file?**, the operation is executed and the aggregate data saved in the form of a construction file.

The file menu also contains options for hard-copy documentation of the current input model.

3.2.3.**Generating the Grid**

The construction file of a model must be accompanied by a grid file for calculation in the next branch of WAEBRU. To call up the gridding screen (with grid menu) from the Main Screen, press **Alt-G**.

Based on standard parameters, a grid file of the current model is generated by activating the following sequence of grid menu operations:

1. **Min. grid - generate**
2. **Fine grid - generate**
3. **Write grid** (+ confirmation of *filename*)

Note: A graphic documentation of the grid screen display can be obtained with **Alt-P**.

Press **Esc** to return to the Main Screen.

The input branch is finished when the two model files (construction and grid) have been completed satisfactorily.

Press **F10** to move on to the Main Menu.

3.2.4.**Obtaining Base Solutions - Calculation**

The demonstration object contains two spaces (interior and exterior environments) and one heat source. To calculate a complete set of base solutions:

- Activate the first selection option, **All cases**, from the Main Menu. Space number 1 and the heat source are then shown to be **selected** in the case status field of the Main Menu screen; the exterior space, number 0, is identified as **superposable**.
- Start the calculation branch by activating the line **Begin calculation**. Convergence behaviour of the iterative calculation process is displayed in the form of running status messages.
- As soon as the calculation of base solutions has been terminated successfully, the Main Menu screen appears again. The cases selected for calculation are now identified as **calculated** in the case status field -

```

2 cases/base solutions:
1 - 1 calculated ( 708 iter.)
2 - 0 superposable.

1 heat source case(s):
3 - 1 calculated ( 707 iter.)

```

3.2.5.**Obtaining Results - Evaluation**

Activate the output option **Evaluation** from the Main Menu to start the evaluation branch of WAEBRU.

Evaluation Conditions

The case status window appears first, with active input fields for specifying evaluation conditions. Enter:

```
TEMP.      Space number(s)
  20.00 - 1 calculated ( 708 iter.)
  -10.00 - 0 superposable.

POWER      Heat source(s)
   0.00 - 0 calculated ( 707 iter.)
```

The heat source is thus "turned off" for the first round of evaluation.

As soon as all cases have been assigned conditions, an evaluation menu appears, listing the options available under the conditions given (in this case, also **Heat flow**).

Numeric Results

Activate the option **Conductance matrix** under **Printouts** of the evaluation menu. The matrix appears on the screen, with a message asking if this data should be sent to the printer. Answer with **y** to obtain the following output:

(program information - date, time, etc.)

(project description)

Number of balanced cells: 1489

Conductances(2-dimensional)[W/(mK)]

Space#	0:	1:
	-----	0.91917
	0.91930	-----

Error analysis:

Space#:	Closeup-error [W/(mK)]	Conduct.sum [W/(mK)]	Linear closeup-error
0:	1.36495E-04	0.9192	1.48498E-04
1:	-1.36495E-04	0.9193	-1.48476E-04

Note: The conductance matrix results shown above are only "borderline acceptable" in terms of precision. For some purposes, a re-calculation of the base solutions with a more stringent termination condition may be called for at this point, before continuing with evaluation. For a demonstration object, however, the precision of these results will suffice.

The table of heat source distribution factors is printed out separately (after confirmation again with **y**):

(program information - date, time, etc.)

(project description)

Number of balanced cells: 1489

Heat source distribution factors

Space#	HS 0:
0:	0.08246
1:	0.91752

Error analysis

HS #	Closeup-error [1]
0:	2.15173E-05

These results imply that the heat source, i.e. the simulated electric heating assembly meant to heat space 1, would lose approximately 8% of its power to the exterior (space 0) - without flooring over the concrete topping.

Note: Just as the conductance matrix, the heat source distribution factors are condition-independent, i.e. valid for any temperatures and source powers of evaluation.

The option **Evaluation results** is for obtaining numeric results which are dependant on the evaluation conditions (temperature and power). After a short waiting period, during which the temperature distribution in the component is calculated by the program, the following results are sent directly to the printer (without screen preview):

(program information - date, time, etc.)

(project description)

Number of balanced cells: 1489

Bound.cond.	Sp.temp.	min.temp.	max.temp.	dewpoint
Space: 0	-10.00	-9.98	-9.39	
Space: 1	20.00	15.28	18.49	74.22 %

Heat sources	Power [W/m]
HS : 0	0.00

Weights for coldest surface point of each space

Space#	g1	g0	h0
0	0.000828	0.999172	0.000122
1	0.842556	0.157444	0.064879

Coordinates of coldest surface point of each space

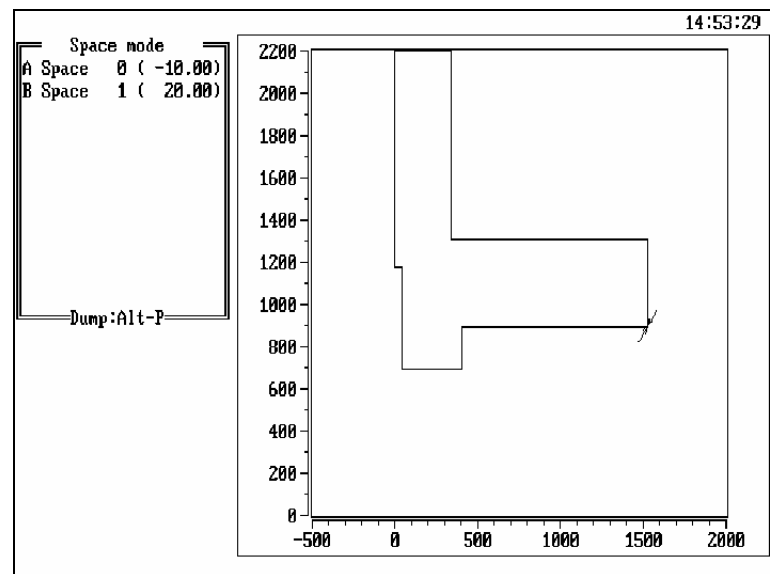
Space#	x	y
0	45.0000	695.0000
1	345.0000	1310.0000

Note: Though weighting factors as a whole are not dependant on boundary conditions, they are included here in association with the coldest surface points (which are condition-dependant).

Graphic Results

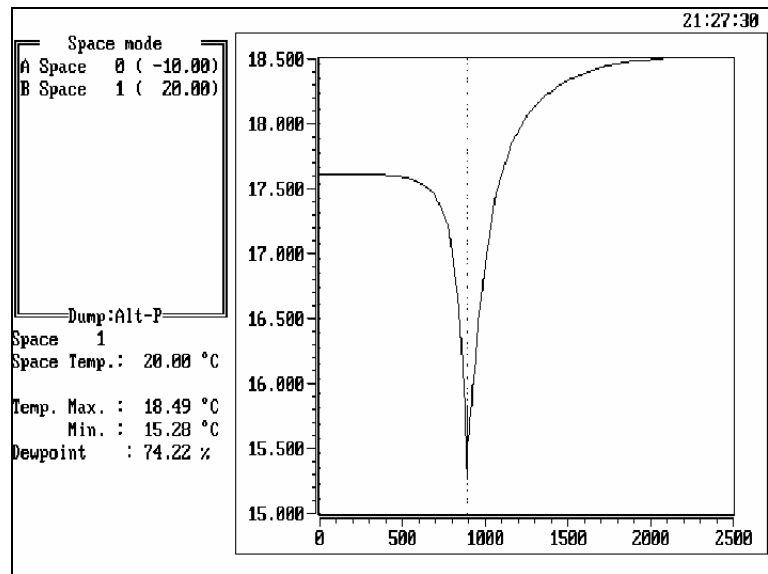
To plot the temperature along the surface of a space, activate the menu option **Surface temperature**. The first screen which then appears is for selecting the surface along which the curve shall be plotted.

surface selection



Move the cursor to select space 1 (the interior), and press **Enter**. Before generating the temperature curve on the screen, the program asks if this shall be written onto a plotfile; confirm with **y** so that this graph will be available for the output branch of WAEBRU.

surface temperature



Note: The dotted vertical line at 900 mm marks the floor/wall corner of the section - quite apparently the coldest surface point of space 1.

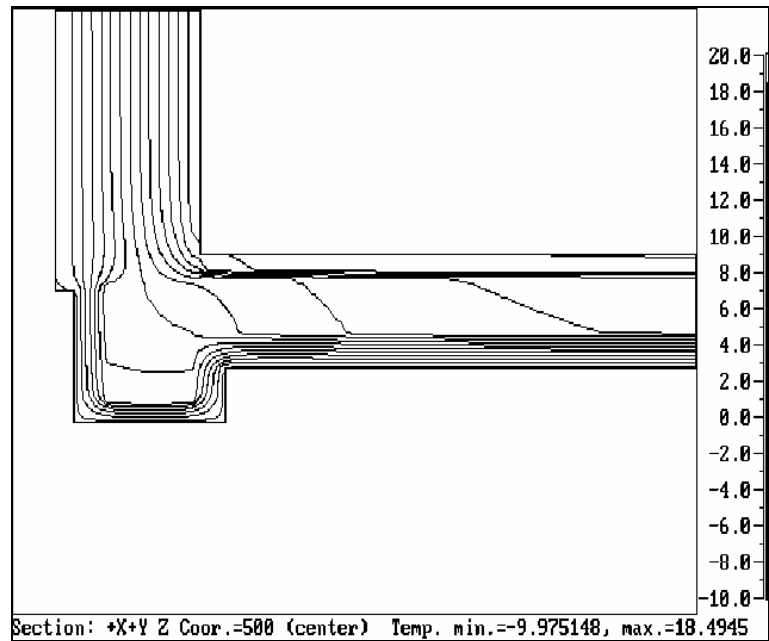
Next, a rendering of the temperature distribution in the form of isotherms shall be generated using the menu option **Isotherms**. Enter the following range and interval specifications:

```
Eval. range: -9.60 : 18.40
Interval    : 2
No. of int.: 0
```

Press **F9** or **Enter** from the last input field to generate the graph (also confirm the plotfile option with **y**).

The input window can be temporarily removed from the screen for a better view with **Alt-V**; press any key to bring it back.

isotherms



(Isotherm screen shown here without colour legend to the right.)

If a graph dump of the display is desired, press **Alt-P** (input window is automatically removed from the printed image). Press **Esc** to return to the evaluation menu.

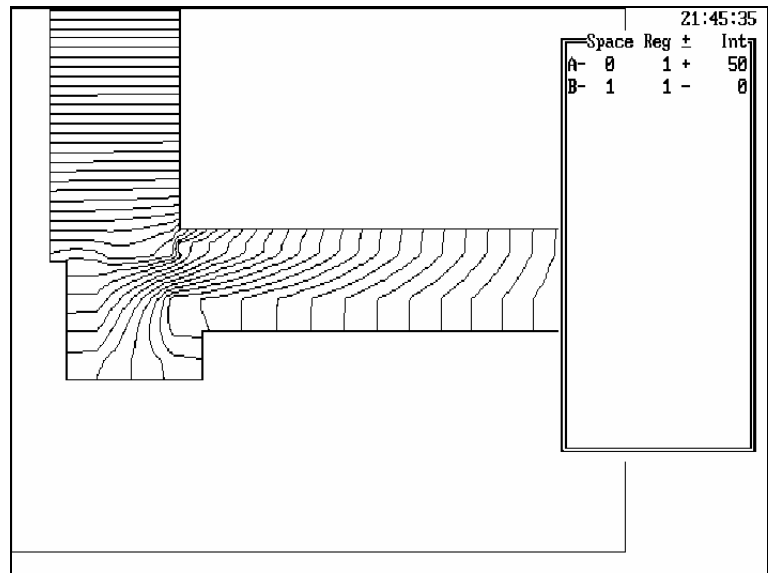
Note: The information reflected in the isotherms can also be viewed in the coarser form of a pseudo-colour graph, which is generated on the screen simply by activating the option **False color graph**. Graph dump documentation, however, is not possible, since the colour areas would print black.

Since the heat source has been "turned off" for this evaluation, and is thus treated as a regular material element, the option of generating a heat flow diagram is also available.

Upon activating the line **Heat flow**, a screen with interval specifications appears. To activate an input field, move the cursor to mark the line and press **Enter**. The field is automatically de-activated again after an integer has been entered. Press **F9** to generate the graph.

The diagram below was drawn with 50 intervals for heat flowing into space 0 (positive flow), and 0 intervals for the negative flow out of space 1.

heat flow diagram



Note: Since this model contains only two spaces, the heat flowing out of space 1 is the same as the heat entering space 0. Setting both intervals to 50 would therefore result in the same lines being drawn twice.

Comparing Results under Different Conditions

The same model shall now be evaluated under the condition of an active heat source element. The electric heating assembly embedded in the floor is assumed to have a power output of 60 W/m². In the two-dimensional simulation, which is modelled as a layer of 1 m thickness, this output corresponds to 60 Watts per meter length of heating element in the section. The total power to be assigned to the heat source here is therefore:

$$(60 \text{ W/m}^2) \times (1.175 \text{ m}) \rightarrow 70.5 \text{ W/m}$$

To re-define the conditions of evaluation, activate the menu option **Boundary conditions**. Move the cursor to the power input field (the temperature values can be left as is) and enter the new power value [W/m]:

```

TEMP.      Space number(s)
          20.00 - 1 calculated ( 708 iter.)
          -10.00 - 0 superposable.

POWER      Heat source(s)
          70.5 - 0 calculated ( 707 iter.)

```

Numeric Results

Since the conductance matrix and heat source distribution factors remain the same for any given evaluation conditions, these need not be printed out again.

The Evaluation results printout, however, does show new data:

(program information - date, time, etc.)
(project description)

Number of balanced cells: 1489

Bound.cond.	Sp.temp.	min.temp.	max.temp.	dewpoint
Space: 0	-10.00	<u>-9.97</u>	<u>-9.18</u>	
Space: 1	20.00	<u>17.61</u>	<u>27.80</u>	<u>86.14</u> %

Heat sources	Power [W/m]
HS : 0	<u>70.50</u>

Weights for coldest surface point of each space

Space#	g1	g0	h0
0	0.000828	0.999172	0.000122
1	<u>0.920457</u>	<u>0.079543</u>	<u>0.000014</u>

Coordinates of coldest surface point of each space

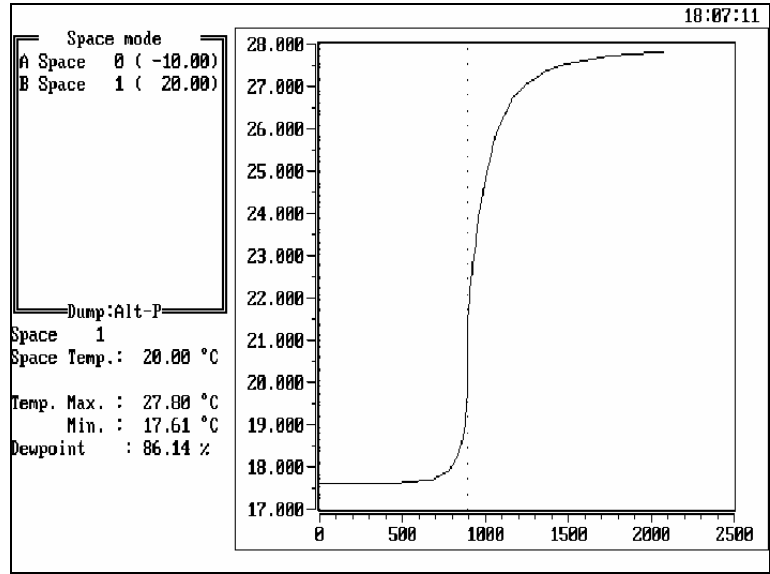
Space#	x	y
0	45.0000	695.0000
1	<u>345.0000</u>	<u>2166.2500</u>

The coldest interior surface point is no longer at the floor/wall corner. The temperature on the floor surface is also considerably warmer than without the heat source (nearly 8° higher).

Graphic Results

Now the temperature curve shall be plotted for the same surface as before (space 1). Note that the heat source has eliminated the temperature drop at the thermal bridge located in the corner:

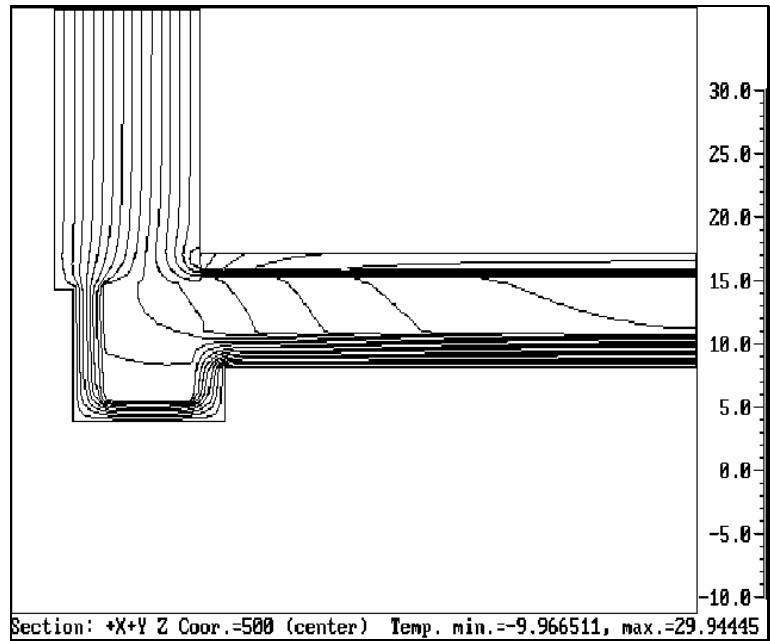
surface temperature



An isotherm diagram shall also be generated for comparison:

Eval. range: -9.60 : 18.40
 Interval : 2
 No. of int.: 0

isotherms



This time the evaluation range is noticeably broader. As could be expected, the isotherm lines, drawn at the same temperature interval as before, are highly concentrated in the insulation.

Press **Esc** to return to the evaluation menu.

Note: Various possibilities for manipulating screen display and graph dump images are available; for a summary of these functions, see the section on **Evaluation Branch** in Appendix C, **List of Key Functions**.

Return to the Main Menu with **F10**.

3.2.6.

Plotting Graphic Results - Output

High resolution prints or plots (depending on output device - see **Configuration Data**) can be produced by activating the Main menu option **Graphic output**. This executes the output branch of WAEBRU.

Assuming that all the graphs evaluated in the last section have been written onto the model plotfile, the output selection screen will look something like this:

```
File: (filename).2GR
SqNr  Description
 14: SURFACE TEMPERATURE (Space 1; T= 20)
 21: ISOTHERMS (+X+Y Z= 500)
 27: HEAT FLOW
 41: SURFACE TEMPERATURE (Space 1; T= 20)
 48: ISOTHERMS (+X+Y Z= 500)
Selected: 48
Enter key (?SVPDE) :_
```

The available graphs are listed in the order that they were generated; the (sequence) numbers have been assigned by the program and serve as graph identification.

Enter **S** at the prompt to select a particular graph. Then enter a sequence number - from the list displayed - in the subsequent input field. This graph number is now shown to be selected.

To preview the selected graph on the screen before plotting, type **V** at the prompt.

Type **P** to plot the final output. The selected graph is displayed briefly before being sent to the assigned output device (**WAEBRU.INI** file).

Enter **E**, or press **F10**, to return to the Main Menu.

Press **F10** to end WAEBRU.

3.3. Three-dimensional Analysis

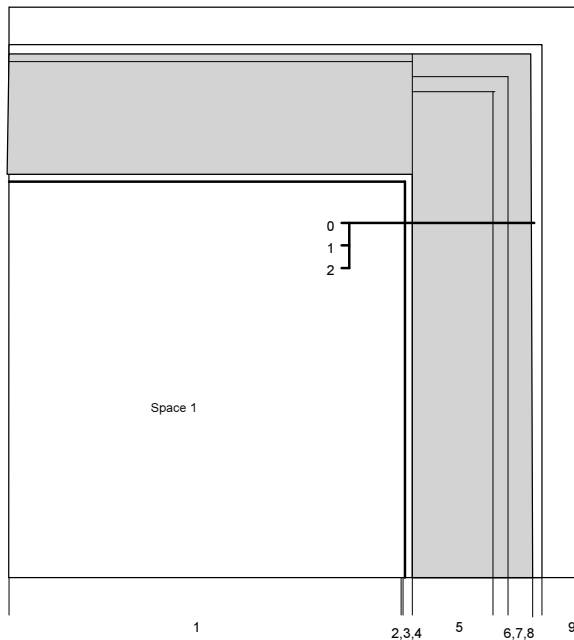
3.3.1. Modelling the Object

Three-dimensional objects are entered as a series of component groups, each describing a two-dimensional section of the object for a specified thickness. A sketch of the model should therefore be made as an orientation aid for input of layers in the z direction.

Such an orientation sketch should show the object in orthogonal sections, including reference coordinates consistent in all three directions. For the three-dimensional demonstration object, the sketch would show the geometry as follows:

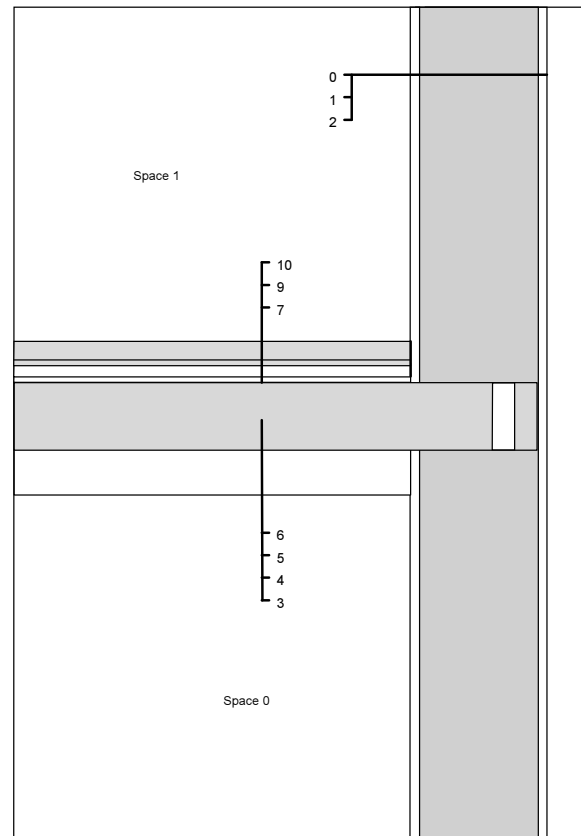
x/y section - see sketch of 2-dimensional model

x/z section



layers:	thickness:
1	1175
2	1
3	4
4	20
5	200
6	40
7	60
8	25
9	100

z/y section



The reference list of material properties for the two-dimensional section modelled in the last chapter covers nearly all of the materials required here. The only new material is that of the perimeter brick, for which $\lambda = 0.500$.

3.3.2. Generating the Model - Input

As already mentioned, the three-dimensional input model of the demonstration object need not be generated "from scratch". A section of the first layer in the z direction already exists in the form of the two-dimensional model, including nearly complete tables. These files can be transformed and used as the basis for the model to be generated in this chapter.

Transforming an Existing 2-Dimensional Model

In order to load two-dimensional files for editing in three-dimensional mode, the input branch of WAEBRU (**waebinp**) must be executed separately from the rest of the program package. Assuming the 2D model generated in the last chapter was named **tutor2**, the run command would look like this:

```
waebinp /2 tutor2 /3
```

The project description screen appears first, with the text as it was entered for the two-dimensional files. This can be appropriately edited here, e.g.

TUTORIAL - 3-dimensional:

```
Corner construction over carport. Masonry bearing wall on girder
over carport and bonded with concrete floor slab (+ext. insul.,
perimeter insul., elect. heating assembly in floor topping).
```

Press **F9** to save the changes and move on to the Main Screen.

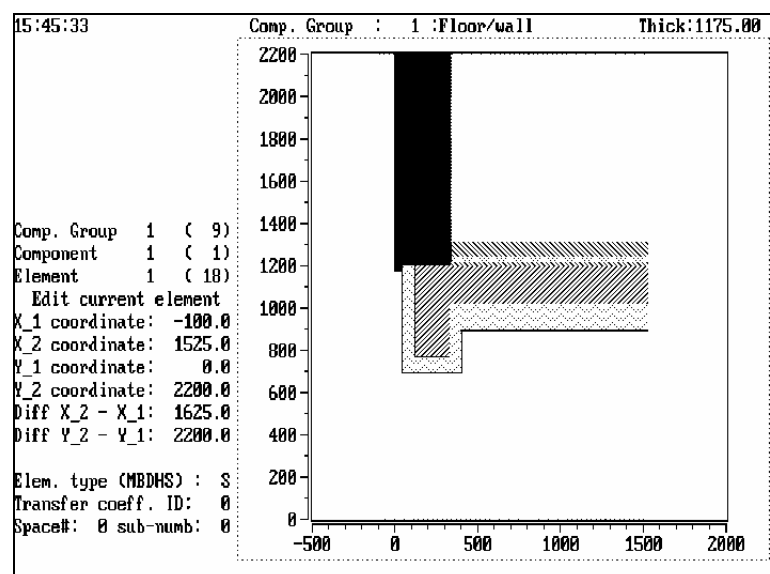
To attain the appropriate display level, call up the view menu with **Alt-V** and activate the option **Group Display**.

The current section, shown as a component group, should be named at this point and assigned a thickness. Press **Alt-N**, move the cursor to the name/status fields above the control graph, and enter:

```
Comp.Group: 1 :Floor/wall      Thick.:1175
```

After re-scaling (**Alt-V** → **Full Scale** → **F9**), the Main Screen should look like this:

input - group 1



Input can be continued from here as described below. Before leaving the input branch, however, the model must be explicitly saved in 3D files which are accessible for the rest of the program package in three-dimensional mode.

To secure the files at this point, press **Alt-F** for the file menu and activate the option **Write aggregate**. Specify a new filename (e.g. **tutor3**) in the subsequent file survey screen and confirm to return to the Main Screen. The input branch of WAEBRU can be ended now with **F10**.

The formerly two-dimensional model is now transformed and ready for completion. To run the entire program package for continued work on this model, type the standard command and follow the usual procedures through file survey. An alternate possibility is the following command chain, which will load **tutor3** for operation in three-dimensional mode automatically:

```
waebru /3 tutor3
```

(Substitute **tutor3** with appropriate filename.)

Completing the Model in Three Dimensions

The aggregate will be built up by copying and editing component groups in succession. Starting with group 1, floor/wall section, displayed as illustrated above (use view menu), press **F4** to establish and move into the next component group.

Group 2 is, of course, still empty at this point. Press **Alt-C** to call up the copy menu, activate the option **Group (of comp.)**, and specify **Component group 1**. The geometric and material data of all the elements in group 1 is hereby copied into the currently displayed group number (2).

Now edit the name and thickness of this group by pressing **Alt-N** and moving the cursor to the name fields over the control graph:

```
Comp.Group: 2 :Floor/stucco           Thick.:1
```

The section through the stucco (layer 2 of the model) is simpler than the previous one, i.e. composed of fewer material elements. The first stage of editing is to page to those copied elements which are not needed for this section and simply delete them, one by one, with **Alt-D**. All remaining elements are re-numbered automatically.

Starting at the "top" of group 2, that is, from the last element (18), page down in order to display and then delete the following elements:

element #	material (ID#)
10	ext. plaster (3)
6	ext. plaster (3)

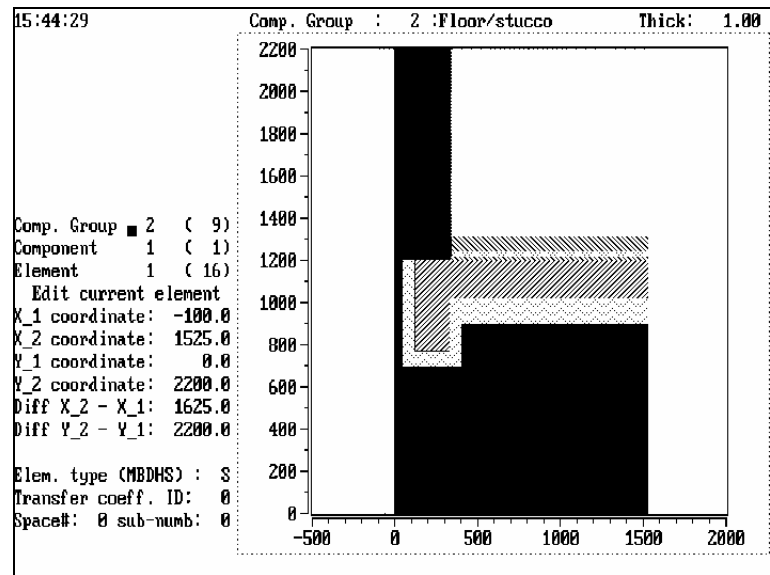
The second stage of component group editing is to make any necessary adjustments in the input values of the remaining elements.

For this layer of the model, only the size of one element needs to be changed. Page to element 2 - stucco, move the cursor to input of \mathbf{Y}_1 , and enter:

\mathbf{X}_1 :	0
\mathbf{X}_2 :	1525
\mathbf{Y}_1 :	<u>0</u>
\mathbf{Y}_2 :	2200

Press **F9** to confirm the change and re-draw the component group. It should look like this (at element 1):

input - group 2



Now the same copy and editing procedure shall be repeated to establish the next layer of the model, component group 3:

- Press **F4**, then **Alt-C** for the copy menu. Activate the option **Group (of comp.)** and specify **Component group 2**.
- Press **Alt-N** and edit the name fields:
Comp.Group: 3 :Exp.strip layer Thick.:4
- Starting at the top of the group, page down and delete:

element #	material (ID#)
16	concrete topping (10)
15 (heat source)	concrete topping (10)
14	isol. underlayment (9)
10	adhesive mortar (5)
9	insulation (4)
8	reinf. concrete (6)
7	adhesive mortar (5)
6	insulation (4)

- Edit element 6 (formerly 11) - reinf. concrete:

X_1: 125
X_2: 1525
Y_1: 1025
Y_2: 1200

The cross-section for this and several of the following model layers shows the perimeter brick as well as the insulation along the edge of the floor slab. For this part of the construction, two new elements must be inserted into the component group "under" the element of reinforced concrete.

Switch to add/insert mode by pressing **Alt-I** (mode status should read **Insert new element**).

Edit the input data still visible from the former element 6 (reinf. concrete):

```

X_1: 85
X_2: 1525
Y_1: 1025
Y_2: 1200

```

```

material ID: 4
           (insulation)

```

The new element 6 - insulation - is generated upon confirmation (**F9** or **Enter** from the last field), and the reinforced concrete element is now number 7.

The perimeter brick element shall be inserted "under" the insulation:

```

X_1: 25
X_2: 1525
Y_1: 1025
Y_2: 1200

```

```

material ID: 11

```

Upon entering the material ID number 11, a message appears informing the user that no material exists in the table for that number. Press **Alt-M** and enter:

```

ID#:11    λ:0.5    Name:Perimeter brick

```

Switch back to edit mode with **Alt-I** and page through elements 5 through 8 to make certain that the sequence corresponds to the following (delete any superfluous elements which may have been generated unintentionally in insert mode):

element #	material (ID#)
5 (space)	int. transfer coeff. (1)
6	perimeter brick (11)
7	insulation (4)
8	reinf. concrete (6)

- Component group 3 is now complete. Press **F4** to establish the next group, 4.

The rest of the layers of the demonstration object require editing of copied groups only with respect to name and thickness, followed by a successive deletion of elements:

group 4 -

- **Alt-C** → Group (of comp.) → Component group 3 (copied)

- **Alt-N** →

```

Comp.Group: 4 :Int.plaster      Thick.:20

```

- Delete (**Alt-D**):

element #	material (ID#)
10	expansion strip (8)
9	sand cushion (9)
5 (space)	int. transfer coeff. (1)

- **F4**

group 5 -

- **Alt-C** → Group (of comp.) → Component group 4 (copied)
- **Alt-N** →
Comp.Group: 5 :Masonry layer Thick.:200
- Delete (Alt-D):

element #	material (ID#)
4	interior plaster (2)

- F4

group 6 -

- **Alt-C** → Group (of comp.) → Component group 5 (copied)
- **Alt-N** →
Comp.Group: 6 :Slab perim.insul. Thick.:40
- Delete (Alt-D):

element #	material (ID#)
6	reinf. concrete (6)

- F4

group 7 -

- **Alt-C** → Group (of comp.) → Component group 6 (copied)
- **Alt-N** →
Comp.Group: 7 :Perimeter brick Thick.:60
- Delete (Alt-D):

element #	material (ID#)
5	insulation (4)

- F4

group 8 -

- **Alt-C** → Group (of comp.) → Component group 7 (copied)
- **Alt-N** →
Comp.Group: 8 :Stucco layer Thick.:25
- Delete (Alt-D):

element #	material (ID#)
4	perimeter brick (11)
3	masonry (1)

- F4

group 9 -

- **Alt-C** → **Group (of comp.)** → **Component group 8** (copied)
- **Alt-N** →
 Comp.Group: 9 :Ext.space Thick.:100
- **Delete (Alt-D):**

element #	material (ID#)
2	stucco (0)

The aggregate is now complete; the eight component groups can be paged through using **F3** and **F4**. Save the aggregate data as a construction file in the usual manner:

- Press **Alt-F** for the file menu, activate the option **Write aggregate**, and confirm a file name.
- For aggregate documentation, use the options for **Data printouts**.

Generating Grid

A grid file for the demonstration object shall be generated based on standard parameters in the same manner as for the two-dimensional model. Call up the gridding screen with **Alt-G**, and perform the standard sequence of operations through the grid menu:

1. **Min. grid - generate**
2. **Fine grid - generate**
3. **Write grid** (+ confirmation of *filename*)

Note: A graphic documentation of the grid screen display can be obtained with **Alt-P**.

Press **Esc** to return to the Main Screen.

The three-dimensional model is now complete for the calculation branch of WAEBRU.

Press **F10** to access the Main Menu screen.

3.3.3.**Calculation**

The base solutions for the three-dimensional model are obtained in the same manner as for the two-dimensional section:

- Select **All cases** from the Main Menu.
- **Begin calculation** (Main Menu) - calculation on this model can be expected to run much longer than for the two-dimensional solutions.
- Upon completion, the case status field will read:

```
2 cases/base solutions:
1 - 1 calculated ( 706 iter.)
2 - 0 superposable.
```

```

1 heat source case(s):
3 - 1 calculated ( 686 iter.)

```

3.3.4.

Evaluation and Output

Start the **Evaluation** branch of WAEBRU, and enter evaluation conditions. The power to be entered here must be the total output of the three-dimensional heat source element [W]. Since this element simulates an area of electric heating assembly with 60 W/m² output, the total power to be assigned here is:

$$(60 \text{ W/m}^2) \times (1.175 \text{ m})^2 \rightarrow \mathbf{82.8 \text{ W}}$$

Therefore, for results comparable to the two-dimensional analysis, boundary condition input must be:

```

TEMP.      Space number(s)
  20.00 - 1 calculated ( 706 iter.)
 -10.00 - 0 superposable.

POWER      Heat source(s)
  82.8 - 0 calculated ( 686 iter.)

```

Numeric Results

Activate the **Conductance matrix** option for the following condition-independent results:

(program information - date, time, etc.)
(project description)

Number of balanced cells: 66474

Conductances [W/K]

```

Space#  0:      1:
0:  -----  1.91661
1:  1.91685  -----

```

Error analysis:

Space#:	Closeup-error [W/(mK)]	Conduct.sum [W/(mK)]	Linear closeup-error
0:	2.44856E-04	1.9166	1.27755E-04
1:	-2.44856E-04	1.9169	-1.27739E-04

(Here again, the conductance matrix would not be symmetrical enough for some purposes; in which case, the base solutions would have to be further calculated.)

Heat source distribution factors

```

Space#  HS 0:
0:      0.11148
1:      0.88849

```

Error analysis

HS #	Closeup-error [1]
0:	3.6800E-05

These results imply that in the corner, the heat source would lose approximately 11% of its output to the exterior space (compared to 8% for the floor regions modelled two-dimensionally).

The **Evaluation results** option provides:

(program information - date, time, etc.)

(project description)

Number of balanced cells: 66474

Bound.cond.	Sp.temp.	min.temp.	max.temp.	dewpoint
Space: 0	-10.00	10.00	-6.58	
Space: 1	20.00	14.04	27.41	68.54 %

Heat sources	Power [W]
HS : 0	82.8

Weights for coldest surface point of each space

Space#	g1	g0	h0
0	0.000000	0.999999	0.000000
1	0.705595	0.294405	0.034722

Coordinates of coldest surface point of each space

Space#	x	y	z
0	0.0000	0.0000	1525.0000
1	345.0000	1310.0000	1180.0000

Graphic Results

The temperature along all three interior space edges - which come together at the thermally critical corner - shall now be plotted using the option **Temp. along an edge** from the evaluation menu.

The coordinate location and extent of each stretch to be evaluated must be specified in a selection screen. A reference list for coordinate selection can be called up from each input field with **F2**; for example -

coordinate selection - x

Y= 1310.000	Grid coordinates :X=
Z= 1180.000	A 295.000
from X= 0.000	B layer center
to X= 0.000	C 315.000
	D layer center
	E 325.000 mat.
	F layer center
	G 330.000 mat.
	H layer center
	I 340.000
	J layer center
	K 345.000 mat. Space
	L layer center
	M 349.000 mat.
	N layer center
	O 354.000
	P layer center
	Q 364.000
	R layer center
	S 374.500
	T layer center
	U 385.000

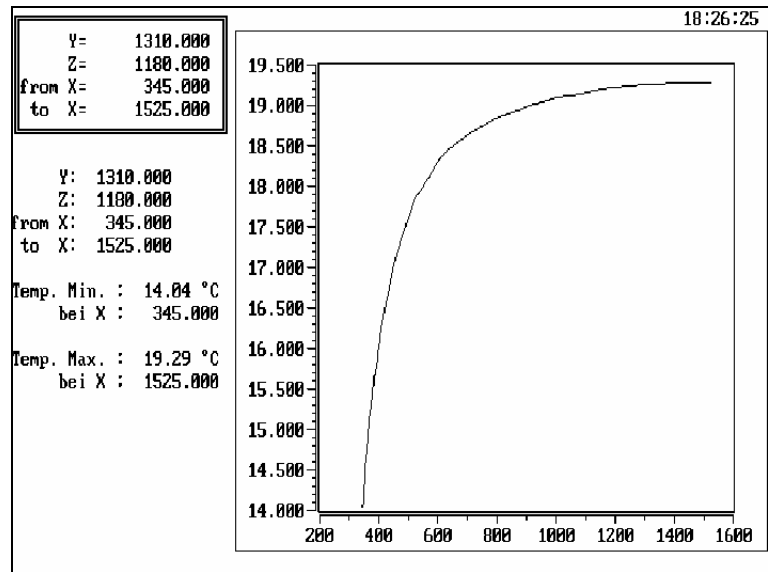
To use such a list for input, simply mark the line corresponding to the desired coordinate and press **Enter** (or press **Esc** to close the list without making a selection).

The first temperature curve shall follow the surface from the corner along the back interior edge of the floor - a stretch in the x direction (from/to). Make certain that the reference axis is x (if not, press **Alt-X**), and enter:

	<i>coordinate</i>	<i>listed plane type:</i>
Y=	<u>1310.000</u>	(space)
Z=	<u>1180.000</u>	(mat.space)
from X=	<u>345.000</u>	(mat.space)
to X=	<u>1525.000</u>	(inf.)

The corner of the room is located at x= 345 mm on the following graph:

temperature along edge - x

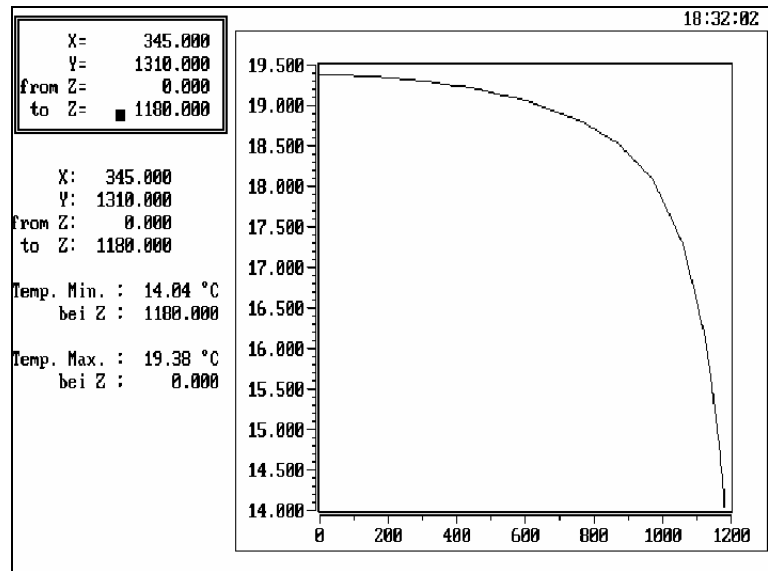


Next, a temperature curve shall be plotted for the edge of the floor surface parallel to the z direction. Press **Alt-Z** and enter:

	<i>coordinate</i>	<i>listed plane type:</i>
X=	<u>345.000</u>	(mat.space)
Y=	<u>1310.000</u>	(space)
from Z=	<u>0.000</u>	(inf.)
to Z=	<u>1180.000</u>	(mat.space)

In this graph (F9), the corner "at the back" of the model is located at z= 1180 mm:

temperature along edge - z



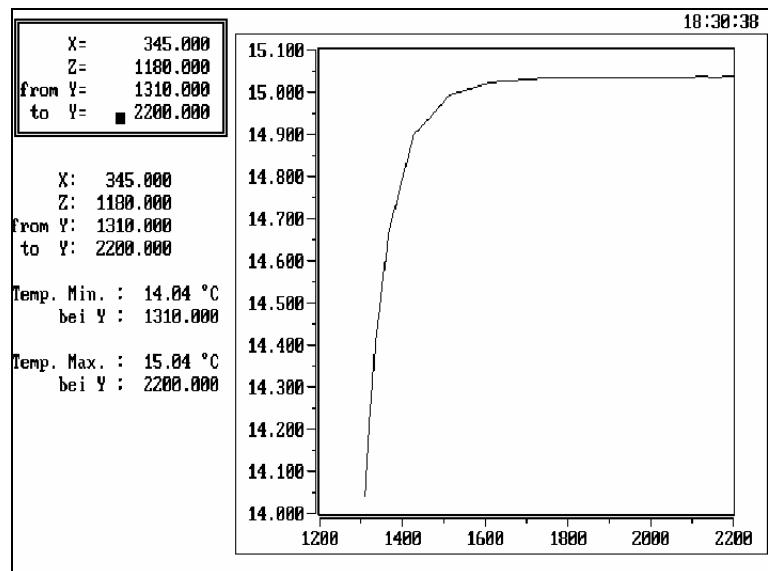
The maximum temperature here - over the girder - is slightly lower than along the other horizontal edge.

Finally, the vertical stretch along the interior corner of the surface shall also be plotted by pressing **Alt-Y** and entering:

	<i>coordinate</i>	<i>listed plane type:</i>
X=	<u>345.000</u>	(mat.space)
Z=	<u>1180.000</u>	(mat.space)
from Y=	<u>1310.000</u>	(space)
to Y=	<u>2200.000</u>	(inf.)

(F9)

temperature along edge - y



As could be expected given the floor heating, the temperature along this vertical edge is lower than for the two previous edges along the floor.

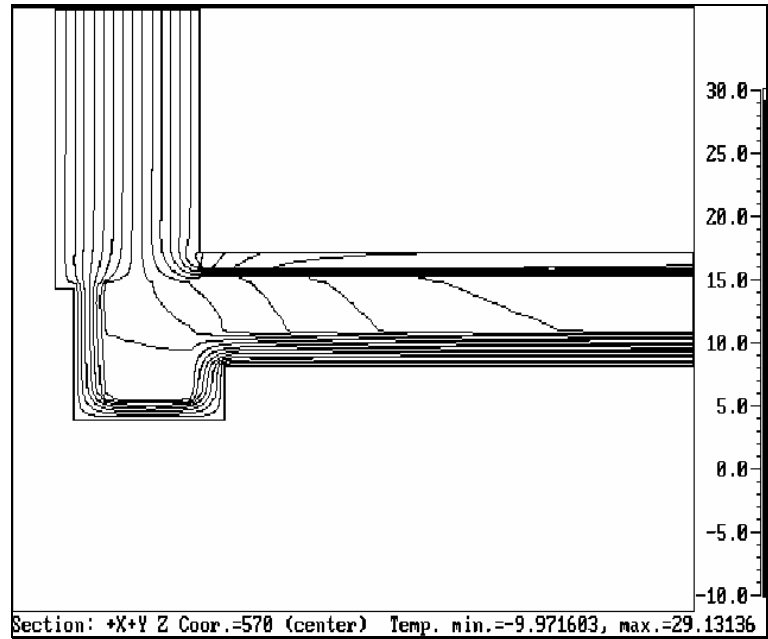
A picture of the overall temperature distribution within the component can be obtained by generating isotherm diagrams for characteristic cross-sections of the model. For this object, these are the x/y and z/y sections (see model sketches).

Activate the menu option **Isotherms**. To evaluate an x/y section, specify the reference coordinate z by pressing Alt-Z. Then select a coordinate position (along the z axis) for the x/y plane: e.g. (570) **layer center**, **Enter**.

To generate a graph comparable to the isotherms plotted for the two-dimensional model, enter:

```
Eval. range: -9.00 : 29.00  
Interval   : 2  
No. of int.: 0
```


isotherms - z



Press **Esc** to return to the evaluation menu.

For another diagram, this time showing the isotherms at the interior wall surface over the girder, activate the **Isotherms** option again. The reference coordinate must be x (**Alt-X**) for a z/y section plane. The orientation of the graph to be generated is shown at the top of the selection list: **+Y+Z**.

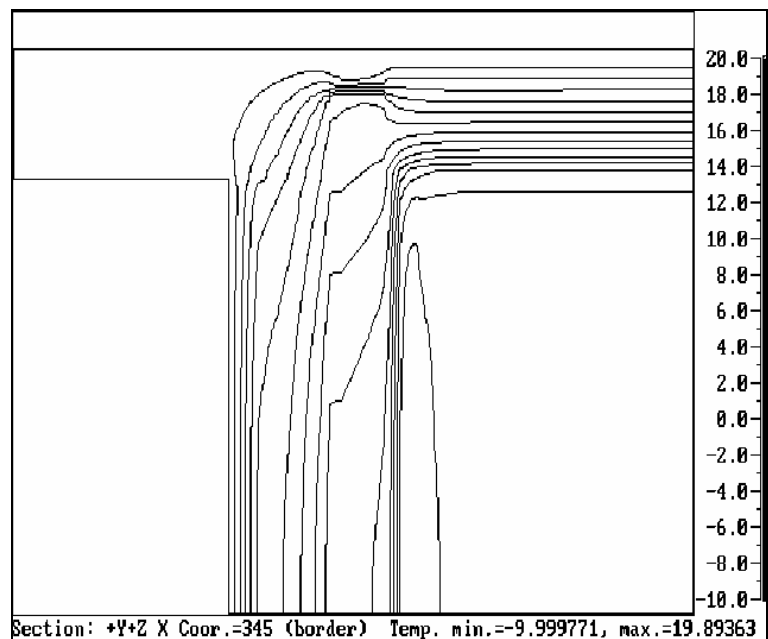
Select **x= 345 mat.space**, and enter:

```

Eval. range: -9.00 : 19.00
Interval    : 2
No. of int.: 0

```

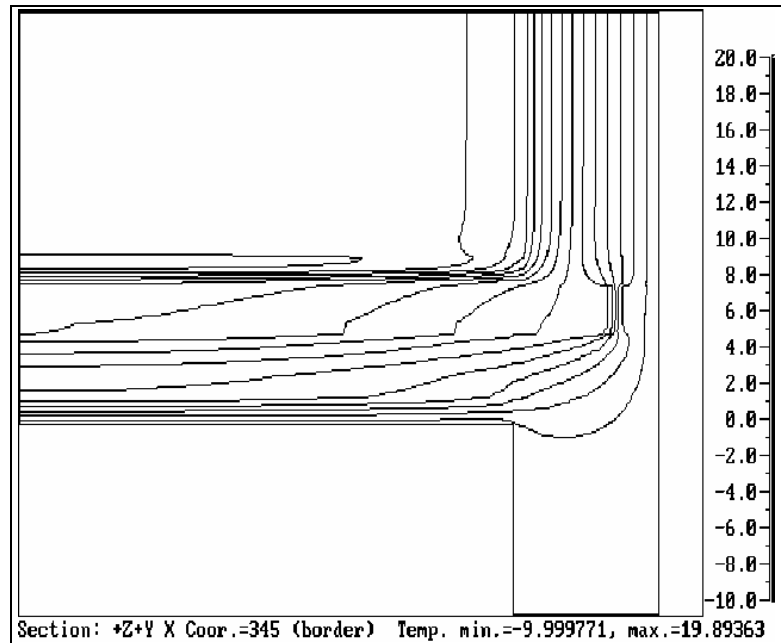
isotherms - x (y/z)



Due to its "unnatural" orientation, this last graph is somewhat confusing at first glance. It would, of course, be more desirable to have the y axis oriented vertically and the z axis horizontally. To obtain a graph with the desired orientation, repeat the **Isotherm** option with reference axis x, and press **F4** to invert the plane axes *before* selecting a coordinate from the list. The plane orientation is then identified at the top of the list as: **+Z+Y**.

With the same isotherm line specifications, the graph will now look like this:

isotherms - x (z/y)



Press **F10** to end the evaluation menu and return to the Main Menu.

The output branch of WAEBRU is executed with the Main Menu option **Graphic output**, in the same manner as for the two-dimensional model. If all the graphic results described in this chapter have been written onto the model plotfile, the output screen will contain at least five graphs:

```
File: (filename).3GR
SqNr  Description
  7: TEMP. ALONG EDGE (X= 345  Z=1180)
 14: TEMP. ALONG EDGE (X= 345  Y=1310)
 21: TEMP. ALONG EDGE (Y=1310  Z=1180)
 28: ISOTHERMS (+X+Y Z= 570)
 35: ISOTHERMS (+Y+Z X= 810)

Selected: 35
Enter key (?SVPDE) :_
```

For operations, see output section of the 2D tutorial.