Part 2:   **Using WAEBRU**

2.1	**General Information**

Although the program package WAEBRU is composed of several modules (program branches), it can be operated as though it were one single program. This is made possible by the execution routine, `WAEBRU.BAT`, which independently ensures proper sequencing through the chain of programs while thoroughly managing the various files generated by individual branches of WAEBRU.

For further information about the names of programs, procedures, and files connected with this program package, please see the chapter on installation (1.1.4) in the first section of this book. The significance of three file types specific to WAEBRU shall be briefly summarised here.

- The file `WAEBRU.HID` contains a letter head with user identification as well as information about the license agreement. This file is necessary for proper distribution and must therefore be included. Changes to this file make the evaluation branch unusable and are therefore strictly prohibited.

- The file `WAEBRU.ERR` contains help and other messages displayed in WAEBRU. This file must also remain unchanged and accessible for the programs in the package. In the event that this file is missing, error messages are displayed in code and on-line help is not available.

- The contents of the initialisation file `WAEBRU.INI`, on the other hand, will generally be changed and adapted to the specific needs and hardware of the user upon installation of the WAEBRU package. The purpose of this file is more thoroughly explained in Appendix A, **Configuration Data**.

In order to reduce the potential of problems arising during operation, it is necessary to check the hard disk regularly and make sure that enough memory for the files generated by WAEBRU is available. As a "rule of thumb", the memory available on the hard disk should be about twice as large as the RAM. This means that for a 5 MB computer (i.e. the minimum RAM recommended for WAEBRU), at least 10 MB must be available in an appropriate directory on the hard disk.

To avoid difficulties due to capacity restrictions, it is advisable to regularly remove completed project files from the hard disk and store them on external disks. Files common to a single project (model) all carry the same primary file name, which is defined by the user. From here on this shall be referred to as *filename*. Thus model data can be simply stored by using the command `copy`, e.g. to disk drive A:

$$\texttt{copy } \textit{filename}\texttt{.* a:}$$

WAEBRU consists largely of typical dialogue programs, which allow interactive operation. The means and possibilities of graphic display are used extensively in the input, evaluation, and output branches. Only the calculation branch is solely a batch program, which runs independently (i.e. without further input from the user) upon execution. The next section on files generated by WAEBRU is followed by an explanation of the methods and rules of operation generally applicable to the entire program package.

2.1.1   Files generated by WAEBRU

Every program of the package generates files specific to WAEBRU and/or reads from the same. Although the management of these files is completely taken care of by WAEBRU, it could be useful to know more about the specific purpose of each of these file types. In the following, a complete run-through of a typical program sequence is sketched with a brief description of all the associated files.

input files

Application of the program package WAEBRU begins with execution of the input branch, in which a geometric model of the object to be analysed (including the spaces adjacent to this object) is generated. The results of the input procedure are stored in two files under the same primary name, but with different extensions. The extensions of the two file types generated in the input branch are **.2BT** and **.2RA** for two-dimensional cases, or **.3BT** and **.3RA** for three-dimensional cases[1].

The "building construction file", *filename*.**2BT**, contains all the information about the structure entered except for grid data. The chosen grid model is stored in a separate "grid file": *filename*.**2RA**. If previously generated versions of such files already exist, these can be loaded and edited in the input branch.

results files

The calculation branch accesses both the construction and the grid files upon execution. Then a mathematical model is generated and base solutions for the cases chosen are calculated. This data is stored in "results files" numbering as many as the cases selected: *filename*.**2**xx (here *xx* stands for the number of the associated calculation case). An auxiliary file, *filename*.**2EV**, is also generated containing the results of the estimation procedure for the optimal relaxation factor - an advantage when continuing calculation of a terminated case.

Further auxiliary files generated by the calculation branch are *filename*.**2AC** and *filename*.**2PA** (which are of no particular significance to the user), as well as *filename*.**2GE**. The latter is the "history file", which contains information about the cases selected and the status of calculation (e.g. terminated, aborted, etc.). This information is also used to access the appropriate preliminary results files when a calculation procedure which has been interrupted is continued at a later point in time.

output files

The evaluation branch utilises all the files described above while generating its own auxiliary file, *filename*.**RAN** (two dimensions only), as well as, if necessary, an additional results file (*filename*.**2**xx). The actual evaluation results are only stored if "plot files" are generated for subsequent use in the graphic output branch.

Such plot files, *filename*.**2GR**, contain the information necessary for producing high quality graphs on an output device (plotter or printer) in the last branch of WAEBRU.

temporary files

---

[1] For the following, only the two-dimensional extensions shall be explicitly mentioned. The descriptions are, of course , also valid for three-dimensional cases.

Furthermore, the individual programs of WAEBRU also generate temporary files, which are automatically deleted upon successful completion of the program. The user is generally unaware of these files under normal conditions of operation. Temporary files of the calculation branch carry the extensions `.TMP` and `.DAT`. The evaluation branch generates temporary files with a standard file name: `WAEBOUT.TMP`, `WAEBOUT.BER`, and `WAEBOUT.FLG`. Finally, files with varying names - but no extensions - are generated by the output branch.

Since irregularities in conditions of operation can never be completely ruled out, it is advisable to check occasionally for the existence of temporary files on the hard disk (with the DOS command `dir` for `*.TMP`, `*.DAT`, etc.) and delete them (`del`) if necessary.

2.1.2 General Operations

Input Fields and Menus

Most data input is done in input screens. These are usually coupled with a graphic control display. Each such screen includes input fields for entering either numeric or text data. Certain input restrictions apply, depending on the type of data called for.

Numeric fields only allow the input of numbers within a given range. To confirm a new number entered in a field, either press **Enter** or simply leave the field using the cursor keys. It is not necessary to explicitly delete any numbers still visible from a previous entry.

Text fields allow the input of any (reasonable) string of characters using standard keyboard text functions. As opposed to numeric fields, previous entries are not automatically deleted - any undesired data still visible must be deleted by the user. For further information pertaining to text field editing functions, please see **List of Key Functions**, Appendix B.

The various execution branches of WAEBRU are activated by means of selection menus. These menus are characterised by a double-line frame surrounding a list of options. Use the cursor keys (↑,↓) or type the first character of the line describing the operation to be selected, and then press **Enter** to execute. Most menu operations can also be directly executed by pressing **Alt** together with the first character of the desired line. To exit a program branch or menu, press **Esc**.

File Survey

As described in 2.1.1, WAEBRU operates with various types of files. In order to facilitate the selection of existing files, a "file survey" function is provided. It can be called up either from the main menu or by pressing **Alt-D**. In certain situations, the file survey screen automatically appears.

Initially the cursor is in the first line of the header field of this screen. To select one of the available files shown in the list below the header, move the cursor into the list by pressing **Tab** and mark the file name using the cursor keys and confirming with **Enter**. The file name then appears in the first line of the header field of the screen. Pressing **Enter** again causes the list to be reduced to the one file selected. Finally, the user is asked to confirm that this is the file to be used (read from or written onto), upon which the file survey is then automatically closed.

The contents of the file list displayed in the survey screen depend on the entries in the header field. In the first line, file name entries consistent with DOS conventions (including wild cards) are possible. The extension is determined by the dimensions of operation (2 or 3), which can be defined in the second line of the header. If a question mark (**?**) is entered here, all files consistent with the name entered, two- as well as three-dimensional, are listed in the field below.

Thus, for example, entering "*****" in the first line and "**?**" in the second causes a complete list of all available files of any dimension to appear for selection. Lists which are too long for the screen space provided can be scrolled using the **PgUp** and **PgDn** keys.

The list field can be left without making a selection by pressing **Esc**. To subsequently exit the file survey without reading from or writing onto a file, press **Esc** again.

Lists

WAEBRU operates with various "lists", or tables (e.g. building material tables, space lists, etc.), which can be called up for use during the appropriate operations. These lists appear on the screen in a form similar to the menus and allow a survey as well as line selection in the same manner as the file list described above.

As opposed to the options listed in a menu, the extent and content of data in these lists and tables is entered and thus defined by the user. List entries can therefore be editted and even deleted (caution!).

Operating System

While running WAEBRU, situations may arise in which it could be helpful to use commands available to the operating system. This can be done from most sections of WAEBRU without leaving the program by pressing **Alt-0** (zero). A system prompt then appears for entering a DOS command. To return to and continue with WAEBRU, enter the command **exit**.

Help and Messages

For support during use of WAEBRU, on-line help is provided in many sections of the program package by pressing **F1**. A text window then appears in which the operations currently applicable are briefly described and explained. Press any key to leave the help function and continue with the program.

Messages with information pertaining to current operations (i.e. status, input criteria, errors, etc.) are displayed in the bottom line of the screen. Error and cautionary messages are furthermore accompanied by an audible warning signal.

If errors causing the program to abort occur (possibly due to improper installation, insufficient hardware capacities, etc.), an error message followed by "`Hit any key to return to system`" appears. In such a case, make a note of the error message and the sequence of operations which led to it, and contact software support for further help.

2.2	**Getting Started**

Before anything can be analysed, the model which is to be considered should be prepared in a form appropriate for evaluation with WAEBRU. An assumed building structure must be defined with respect to

- the relevant thermal properties of its materials and

- the dimensions of and geometric relationships between these materials in space.

Guidelines governing the modelling of the object to be evaluated are explained in the first section of this chapter. Once a model has been sketched, work with WAEBRU can begin as described in the following sections.

2.2.1 Modelling the Object

In the geometric model used by WAEBRU, a single region or volume in space which is defined as "filled" with one homogeneous material is referred to as an *element*. WAEBRU distinguishes between five different principal types of material, categorised as:

- building material (M)

- reinforcing bars, axial (B)

- reinforcing bars, diagonal (D)

- heat source (H)

- space (S)

The *element type* is defined by the material category assigned to it. The specific designations and properties of individual materials to be considered are entered and documented in lists, or *tables*, for each of the above mentioned categories.

Building materials are defined by name (e.g. "concrete masonry unit") and thermal conductivity (*lambda*, $Wm^{-1}K^{-1}$), and are listed by assigned number in the *Materials Table*.

Since heat sources are similarly defined by building material, the *Materials Table* is also used for specifying this element type (a heat source to which no power is assigned in the evaluation stage is treated just as any other non-emanating building material).

Steel reinforcing bars are specified by name, thermal conductivity, and diameter (mm) in the *Steel Table*. This list is referenced for both types of reinforcing bar elements, i.e. axial (running parallel to the coordinate axes) as well as diagonal (at a slope of 45°). The distinction in types here is solely geometric.

In the *Space Table*, relevant characteristics of "air-filled" elements, i.e. spaces, are defined. The air is characterised in this list by the surface heat transfer coefficient (*alpha*, $Wm^{-2}K^{-1}$) appropriate to a designated situation (e.g. "exterior", "interior/along glass", etc.). A single space must be assigned one, unique temperature condition (in the evaluation branch). It may, however, be divided into sub-spaces to account for different surface situations within the space. The sub-spaces can then be identified with different transfer coefficient numbers.

The contents of all tables are user-defined, that is, documented and available only to the extent that they are relevant for a particular application. When working with WAEBRU, the user must enter the above-mentioned specifications as required for the concrete model to be generated and analysed.

The necessary information about material/space properties can be found in standard literature on this topic. This information should be at hand before actually sitting down to work with the program.

As with any numeric method of analysis, the application of WAEBRU with respect to a concrete object requires a considerable abstraction of reality. For dependable results, building components must therefore be abstracted to conform to the geometric principles governing the numeric analysis performed by WAEBRU.

First and foremost, WAEBRU operates only within an orthogonal, Cartesian coordinate system. This limitation is due to the method of analysis, which, briefly put, first divides the model into a network of numerous cells ("gridding") to be mathematically evaluated (see also **Method of Evaluation**, Chapter 1.2). The mathematical description of these cells is a system of equations. The finer the grid, the more exact the evaluation results. However, a finer grid also, logically, increases the number of cells to be calculated, that is, the number of equations to be solved. This quantity is limited by the particular computer hardware configuration of the user (see **Hardware Requirements**, 1.1.3). For orthogonally subdivided cells, a relatively simple system of equations suffices to describe each cell completely, thereby making it possible to evaluate a very finely gridded structure with WAEBRU using a personal computer.

Thus, the first abstraction necessary in modelling a building component is to approximate the geometry of its elements within a strictly right-angled system. It is advisable to prepare a simple, dimensioned sketch of the object to be analysed before beginning actual work with the program. Generally it can be said that simplifications made in the process of abstraction should be based on thermally relevant criteria in order to ensure that the model is thermally equivalent to the "real thing". Though there are no hard and fast rules on developing the ideal model (each concrete object must be treated individually), certain principles and guidelines should be adhered to.

Size

To minimise calculation time, the spatial extent of the region to be analysed should be defined as narrowly as possible. Although entire buildings can also be evaluated by WAEBRU, most practical applications will not be so global in nature. Usually only spatially limited problem zones need be scrutinised. Such a problem zone is typically a structural detail or region which could prove thermally critical.

The dimension of the problem at hand will determine if the model can be limited to a single *building component* or must be extended to include one or more *component groups*. The term *aggregate* denotes the entire model as it is ultimately entered and documented in the input branch of WAEBRU (Chapter 2.3).

Since thermal bridges are characterised by two- and three-dimensional heat flow patterns, regions of a building structure in which heat is expected to flow one-dimensionally (i.e. perpendicular to material surfaces) are generally of little interest in using WAEBRU.
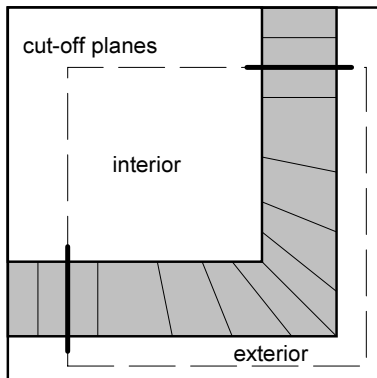
Therefore such regions may be eliminated from the model to be considered without affecting any relevant results. The outer boundary of the region to be analysed is defined by *cut-off planes*. These planes are idealised as a non-conducting surface and should therefore be placed parallel to the direction of heat flow in a region where one-dimensional heat flow can be assumed.

Just where to optimally draw the cut-off planes - the nearest regions of a problem zone where heat can be expected to flow reasonably normal to planar component surfaces - is initially an educated guess, which may, of course, be subject to revision upon evaluation.

One can also draw upon qualitative characteristics inherent in the geometry of a given object to limit the region to be modelled. Particularly axes of local symmetry in a building component, which can be safely assumed to coincide with planes parallel to heat flow, should be used to cut down the size of a model wherever possible.

reinforced c.m.u. wall - symmetry and cut-off planes
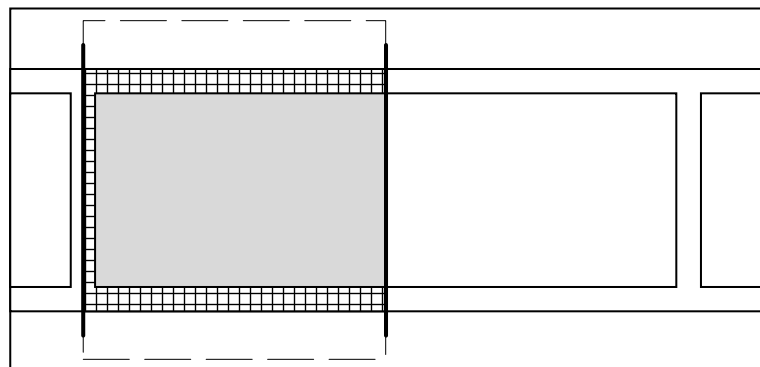
Geometric Principle
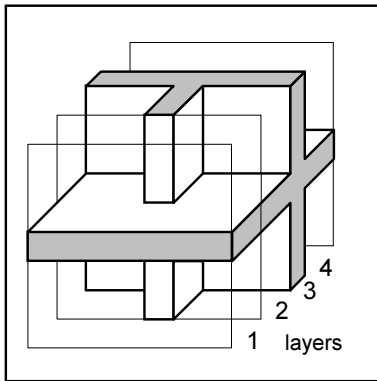
Three-dimensional objects are modelled as a "sandwich" of parallel layers. Each layer is composed of one or more building components and denoted as a *component group*. The elements of the components are entered to form a two-dimensional description (x and y axes), or section, normal to the direction chosen for layer depth (z axis).

The thickness of a particular layer is defined by the depth for which a given section is a valid description of the structure. In other words, a separate layer (component group) must be specified for every change in cross-section when working through the layers of a particular model.

This should be taken into consideration from the very beginning in the form of orientation sketches showing the number of layers (x/y sections) which will be needed in the z direction.

Objects for which a two-dimensional analysis will suffice are modelled as an x-y section and evaluated per meter in the z direction. A dimensioned orientation sketch should include the origin which will be used for x and y coordinate input (an origin position along a component surface is usually the clearest solution).

Abstracting the Geometry
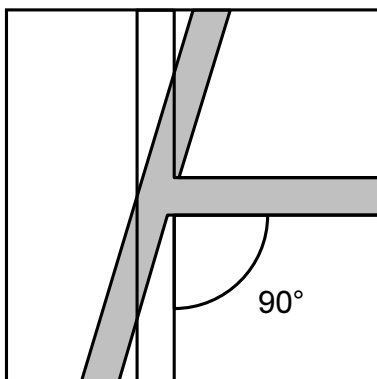
Non-right angles in the elements of a building structure should not be approximated by "stepping" contours orthogonally in such a way as to severely increase their complexity. A more complicated contour generally corresponds to an increase in the surface area of the volume delineated, thereby influencing a key factor in determining heat flow patterns. Therefore, non-orthogonal geometries should be simplified by simple "straightening" whenever possible, that is, whenever the quali-tative effect on heat flow can be expected to be negligible.

Since the placement of diagonal reinforcing steel in concrete could pose theoretical difficulty in orthogonal abstraction, a special element type (*reinforcing bar, diagonal*) is accom-modated for by WAEBRU to approximate this specific case conveniently.

Scope and Scale

The scale of evaluation should be kept in mind when preparing a preliminary model sketch. This means keeping the level of detail consistent with the overall dimensions of the region to be analysed.

Though WAEBRU requires dimensional data to be entered in millimeters, the total size of a particular model could also run up to tens of meters. Over detailing larger spatial structures can cause substantial increases in fine gridding, and therefore in calculation time, without the benefit of increasing the precision of relevant information provided by the evaluation branch (temperature distributions and heat flow patterns).

Finally, any thermally irrelevant elements of a building structure can and should be eliminated from the model. Such are elements which do not primarily influence heat conduction through a building component (e.g. thermally disconnected exterior elements such as ventilated siding, handrails, fixtures, etc.).

2.2.2   **Execution Commands**

The primary command for starting the program is simply `waebru`. If this is all that is typed at the prompt, then the program initially assumes that a two-dimensional case is to be evaluated. The working dimensions can then be changed if necessary in the file survey screen as described in section 2.1.3.

It is, however, also possible to specify the desired working dimensions within the run command by typing the program name followed by a space and then a slash with the number of dimensions:

<div align="center">

`waebru /2` or `waebru /3`

</div>

The file survey screen then appears with the appropriate list automatically.

In the event that a file to be worked on already exists and is known to the user, the file survey screen can be skipped entirely by appending the file name (without an extension) to the start command:

<div align="center">

`waebru /3` *filename*

</div>

When the model existing under this name is complete, the Main Screen appears immediately. If only a construction file - but no grid file - exists, the message

`Grid file does not exist! Run input/grid-generation branch? (Y/N):`

appears. Type `y` to get into the main screen of the input branch or `n` for the file survey screen.

waebinp

In certain (rare) situations, it may be useful to execute only the input branch of WAEBRU. Such a situation is, for example, given when a three-dimensional model which is to be generated can be based on one or more previously generated two-dimensional models. Time spent on input work can then be substantially reduced by loading existing construction files (two- or three-dimensional) into the program and editing them as layers, that is, as component groups of the final three-dimensional structure.

Loading more than one file from the start and/or redefining the working dimensions of a file is only possible when executing the input branch separately. This is done by typing `waebinp` followed by a space and then appending a dimensional parameter together with each filename of that parameter, the next dimension and file name(s), and so on. Adding a dimension parameter at the end of the line specifies the working dimensions assumed upon execution of the input branch. For example:

<div align="center">

`waebinp /2` *filename1 filename2* `/3`

</div>

loads the two-dimensional models, *filename1* and *filename2,* for work in the three-dimensional input branch. The two-dimensional sections then appear as layers (of a yet to be specified thickness) in the order that they are loaded, and can then be editted and completed for three-dimensional evaluation.

Note: The graphic output branch of WAEBRU can also be separately executed (by typing `waebout`).

2.2.3

```
>>>     Input     <<<
Input / Edit / Grid

===  Selection  ===
All cases
1- All spaces

2-All heat sources
Space case
Heat source case

<<< Calculation >>>
Parameters of iter.
Begin calculation

***     Output    ***
Evaluation
Graphic output

+++     Misc.     +++
Continue case calc.


Read file

Dump screen contents
```

Main Menu

The Main Menu is the central option list with which the user controls the sequence of program branches in operation. Thus model files which have already been generated can be loaded and either editted in the input branch, (re-)calculated, or simply evaluated under different conditions. The screen of the Main Menu appears directly when a file name associated with a completed model has been selected in the file survey screen (or already specified in the execution command) as well as upon completion of a new model in the input branch.

The Main Menu screen can also be called up from many sections of WAEBRU by pressing **F10**. To leave the program from this screen, simply press **F10** again.

As shown in the figure to the left, the Main Menu is divided into five sections: **Input, Selection, Calculation, Output,** and **Misc.**.

Input

The **Input** section contains only a single option: **Input/Edit/Grid**, which brings the user into the input branch of WAEBRU when selected (**Alt-I**). The operation of this program branch is explained in Chapter 2.3, **Input Branch**.

Selection

Given a model which has already been generated and gridded, adjacent spaces for which base solutions are to be calculated must first be selected before actual calculation can begin. These calculation cases are specified using the options of the **Selection** section of the Main Menu.

If the line **All cases** is selected (**Alt-A**), a list of all space and heat source cases in the model appears in the status field (left half) of the screen. The numbers correspond to the room or heat source numbers as defined in the input branch, whereby space `"0"` is typically associated with the exterior space.

The options **All spaces** (**Alt-1**) and **All heat sources** (**Alt-2**) automatically list all the cases of the respective type. Each selected case (space or heat source) for which a base solution is calculated can then be treated separately when assigning boundary conditions in the evaluation branch.

If more than one space shall be assigned the same conditions (in other words, if fewer base solutions are necessary), then it is possible to reduce calculation time by selecting the particular cases needed individually. This is done by activating either one of the options **Space case** (**Alt-S**) or **Heat source** (**Alt-H**) and entering the space or heat source number which shall be defined for calculation at the prompt.

The case selection procedure can be repeated as often as necessary, whereby each case is added to the list of the status field and declared as **selected** (see also the section on **Selecting Cases** in Chapter 2.4, **Calculation Branch**). Press **Esc** to return to the Main Menu.

Calculation

The first option in the **Calculation** section is **Parameters of iter.**, which allows the user to define termination conditions for the iterative calculation process. Upon activating this option (**Alt-P**), a box with two input fields appears next to the Main Menu.

The first parameter, termination condition (dT), defines the point at which the case can be deemed solved. Reducing the standard value of 10^{-6} leads to more precise results, but also increases calculation time. Entering a larger value here should generally be avoided.

The second parameter, max. number of iterations, is initially set at 4000, i.e. calculation is terminated automatically after 4000 iterative steps have been completed. If the termination condition has not been satisfied up to this point, the calculation of this base solution is declared as **terminated** in the status field of the Main Menu screen, and cannot be evaluated yet. To complete calculation of such a case, it is necessary to re-start calculation, in which case the program then resumes iteration starting with step number 4001, and continues calculating up to 4000 further steps.

If a case is so large or complex that premature termination is foreseeable, the user can avoid this interruption by increasing the parameter **max. no. of iterations** before the first calculation run (see also the section on **Defining the Parameters** in Chapter 2.4).

The option **Begin calculation** (**Alt-B**) starts execution of the calculation branch of WAEBRU, which is, as previously described, a batch program that runs automatically until reaching termination and can only be aborted by pressing **Ctrl-C**. When calculation of a base solution has been completed successfully, the case is declared as **calculated** (with the number of iterations performed) in the status field of the Main Menu screen, and is thus ready for evaluation in the next branch of WAEBRU.

Output

Once all selected cases have been calculated, the option **Evaluation** in the **Output** section of the Main Menu can be activated (**Alt-E**). The operation of this branch of WAEBRU is explained in Chapter 2.5, **Evaluation Branch**.

Finally, given plot files generated in the evaluation branch, the user can choose to have these files displayed, printed or plotted by selecting the option **Graphic output** (**Alt-G**) - see also Chapter 2.6, **Graphic Output**.

Misc.

If upon evaluation of thermal conductance the base solutions prove to be too imprecise, individual cases can be more accurately calculated by selecting the option `Continue case calculation` (**Alt-C**) in the `Misc.` section of the Main Menu, and subsequently entering the number(s) of the case(s) for which further iterations should be performed. The status of previously calculated cases is then automatically changed to `terminated`.

Before continuing calculation, the parameters of iteration must first be adjusted from the Main Menu to suit more precise conditions as described above. Then calculation can be resumed by activating `Begin calculation` again (**Alt-B**). The continuation is based on the preliminary results achieved before being redefined as terminated.

The option `Read file` (**Alt-R**) calls up the file survey screen, thus allowing the user to continue work on a different model without leaving WAEBRU (see the section on **File Survey** in Chapter 2.1, **General Information**).

Activating the last option, `Dump screen contents` (**Alt-D**), causes an image of the currently visible screen information - Main Menu and case status field - to be sent to an output device. This can be used to document the number of iterations which were required in the process of calculating a complicated model for evaluation.

2.3	Input branch

Once the program package WAEBRU is started, there are two routes which lead to the input branch:

- from the file survey screen - either by entering a new file name or by selecting an incomplete model (for which no grid file exists yet) from the list.

- from the Main Menu - by activating the option `Input/Edit/Grid` (a model file must already be loaded).

Project Description

The first screen which then appears contains a text field for entering and editing a short project description to identify the model under evaluation. The information supplied here by the user is printed at the head of numeric output (model documentation and evaluation results).

Each line of this text field is closed and confirmed by pressing **Enter**. Upon entering the last line, the status of the entire field is confirmed and the project description screen is automatically left. This screen is also closed by pressing either **F9** (to confirm any changes made) or **Esc** (if no changes shall be saved).

The Main Screen of the input branch appears next. To re-enter the project description screen for editing any time while working in input, press **Alt-T** (for 'text').

2.3.1 Main Screen

The input branch of WAEBRU is organised such that all operations needed for generating a model can be performed from a main screen (which appears automatically upon leaving the initial text screen). As illustrated below, the Main Screen shows a scaled graphic control field surrounded on three sides by editable input and status fields.

Main Screen
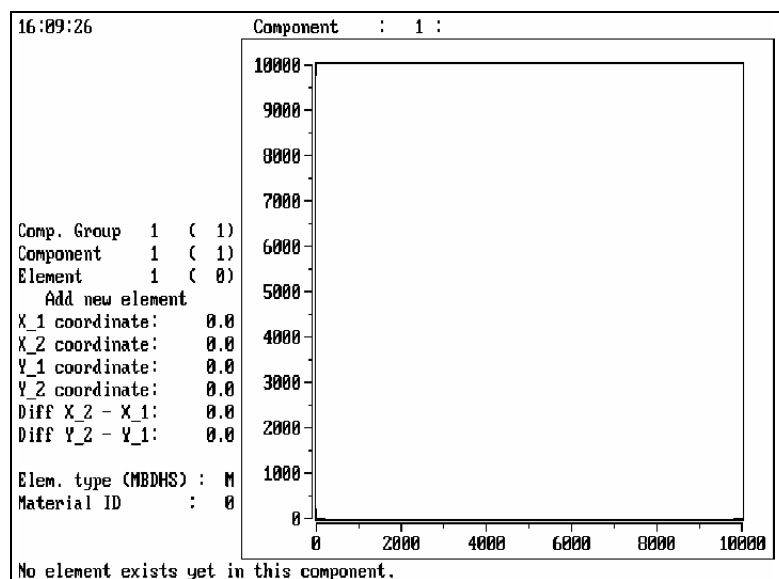
Status Fields

The left side of the Main Screen contains two different regions of data, separated by the mode status line (located approximately in the middle).

The upper region displays general status, i.e. the numbers of the current component group, component, and element. These status fields can furthermore be activated and changed, thus providing an alternate means for the user to move directly to a particular element of the model. The upper region also includes the line at the top of the screen, in which the current building component, group, or aggregate (see **View Menu** below) is named and group thickness (3-dim. only) specified. To move the cursor into and edit the fields of the status/name region, press **Alt-N** (for 'name'). Use the cursor keys or **Enter** to move through the fields and subsequently return to the input region below.

Input Fields

The lower input region is further divided into two blocks corresponding to the two primary aspects of actual model input (geometry and material). The first block contains a column of numeric input fields for specifying the size and position of elements within an x/y coordinate system.

The geometry (points) of the elements are displayed in a tentative form in the control graph as they are entered, that is, parallel to the input process. Rectangular elements are initially delineated by a dotted line connecting markers located at opposite cross corners; reinforcing bars are shown as a single dotted line between two markers (illustrated in the next section on **Generating the Model with Elements**).

In the next block, the material with which the currently displayed element is "filled" is defined by type and number. The first field (**Elem. type**) requires input of one of five possible characters to specify a type:

- **M** for a building material

- **B** for an axial reinforcing bar

- **D** for a diagonal reinforcing bar (45°)

- **H** for a heat source

- **S** for a space (room)

The second field requests input of an identification number corresponding to an entry in the materials list appropriate to the element type specified. For building materials and heat sources, this source list for thermal properties is the materials table. Reinforcing bars of both types are based on a separate table of steel rebars. A selection table for spaces provides surface transfer coefficients.

The procedure for accessing and working with material lists is described in detail in the next section of this chapter, **Generating the Model with Elements**, and then summarised in the section on **Tables**.

When an element has been specified as either a heat source or a space, a third line appears in this block requiring the user to define a heat source number or space and sub-space numbers. These numbers correspond to the cases for which base solutions can later be calculated, and should therefore be defined accordingly (starting with 0 for exterior space). The sub-space function allows alternate spatial combinations to be evaluated under the same conditions. Heat sources are numbered independently; there is no particular convention for the nature of number 0.

The cursor keys as well as **Enter** can be used to move between the fields of the input region. Pressing **Enter** in the last line causes confirmation of the entire current element and leads automatically to the next element. Alternately, **F9** can be used to confirm an element and thereby update the graph. The confirmed element is then displayed as a rectangle filled with a colour corresponding to its thermal conductivity (range). Space elements remain empty (black).

Modes of Operation

There are principally two different modes of operation available for building up and/or correcting a model:

- **Add/Insert new element**
- **Edit current element**

When starting work on a new model, the initial setting is **Add new element** (displayed in the mode status line). If a previously confirmed element needs to be corrected, press **Alt-I** (for 'invert') to switch into edit mode before making any changes in the element displayed. The mode status line then reads **Edit current element**, and no additional element is generated. New data which has just been typed into a field can be cancelled by pressing **Esc**; the previously valid input is then automatically restored.

The **PgUp** and **PgDn** keys are used to scroll through the existing elements of a component when editing. Switching back to add/insert mode (by pressing **Alt-I** again) while a previously confirmed element is displayed causes a new element to be generated and inserted ahead of the existing element. The mode status line then reads **Insert new element**.

When in edit mode, it is possible to delete the entire element currently displayed by pressing **Alt-D** (for 'delete'). A message appears asking the user if this element shall be deleted; entering **y** ('yes') here causes the element to be definitively deleted. All remaining elements in the component are re-numbered automatically.

The bottom line of the screen is reserved for warnings and questions to guide the user during input. A message appears whenever invalid data is entered or the active input mode is inappropriate (i.e. not currently applicable).

View Menu

Certain contents of the Main Screen display (status/name field as well as control graph) can be adjusted by the user by calling up the view menu with **Alt-V**.

The first block of this menu lists options for defining whether the current component, component group, or the complete aggregate is to be displayed in the control graph and named in the top line of the screen. This is of particular importance when generating a three-dimensional model, since layers are defined as component groups of a thickness which must be specified in this field. `Group Display` (**Alt-G**) is therefore the option generally to be chosen when working in three dimensions.

```
Component Display
Group Display
Aggregate Display

Full Scale
Specify Scale
```

When generating a two-dimensional model, `Aggregate Display` (**Alt-A**) is the standard, since it shows all existing components of the model in progress. This option is inappropriate for three-dimensional work, however, since the resulting simultaneous display of all model layers in the control graph is fairly useless.

The second block of the view menu offers two different options for scaling the control graph display: `Full  Scale` or `Specify Scale` (user-defined). When either of these options is activated (**Alt-F** or **Alt-S**), a numeric input window appears, in which x and y coordinate ranges for the graph can be defined.

The full scale option suggests ranges which correspond to a maximised view of the component in the Main Screen graph. The suggested values can be confirmed as is or editted. Once a set of ranges for scaling has been confirmed (with **Enter** from the last line or with **F9**), the menu is closed and the graph rescaled automatically.

2.3.2   **Generating the Model with Elements**

The model of a building component is, generally put, built up with rectangular elements defined by beginning and end values for x and y coordinates. As briefly sketched in Chapter 1.3 in the first part of this manual, a convenient input principle inherent to WAEBRU - element overlapping - can facilitate coordinate input considerably if employed properly.
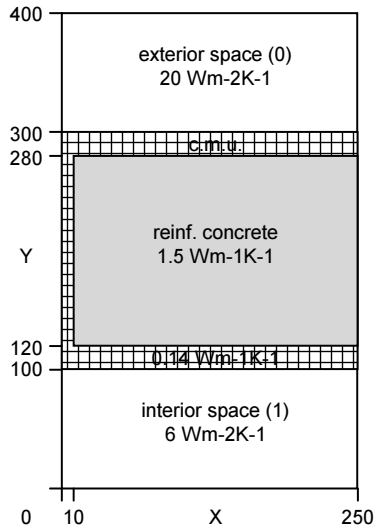
An example

The input operations presented in the last section, together with the principle of element overlapping, are best described by means of a simple example: the thermal bridge in a reinforced concrete masonry unit wall.

The scope of the model can be reduced to the region between two neighbouring axes of symmetry - as illustrated in the section on **Modelling the Object**, Chapter 2.2. The region to be modelled is depicted in the dimensioned sketch to the left, which also includes the necessary information on the thermal properties of all element materials.

reinforced c.m.u. wall section

The total c.m.u. wall thickness is 20 cm (200 mm); the cell walls are assumed to be 20 mm thick. A look at the control graph shows that the default scaling is obviously inappropriate for such dimensions. We can change the graph scale before starting input by calling up the View Menu (**Alt-V**) and tentatively defining an x range from 0 mm to 300 mm as well as a y range from 0 mm to 400 mm (slightly oversized to include some space to both sides).

An alternate route to attaining a reasonable graph scale is by entering an exterior space element extending over the entire model region first, and then activating the full scale option of the view menu. This first element is thus defined in the x direction from cut-off plane to cut-off plane by entering `X_1: 0` mm, and either `X_2: 250` mm or `Diff X_2 - X_1: 250` mm. As the example shows, the second coordinate can be entered either in absolute terms or relative to the first coordinate by defining a differential (negative values are also allowed, but are more likely to cause confusion). In the y direction, the element shall extend from 0 to 400 mm (`Y_1: 0`, and either `Y_2: 400` or `Diff Y_2 - Y_1: 400`).

Main Screen - 1
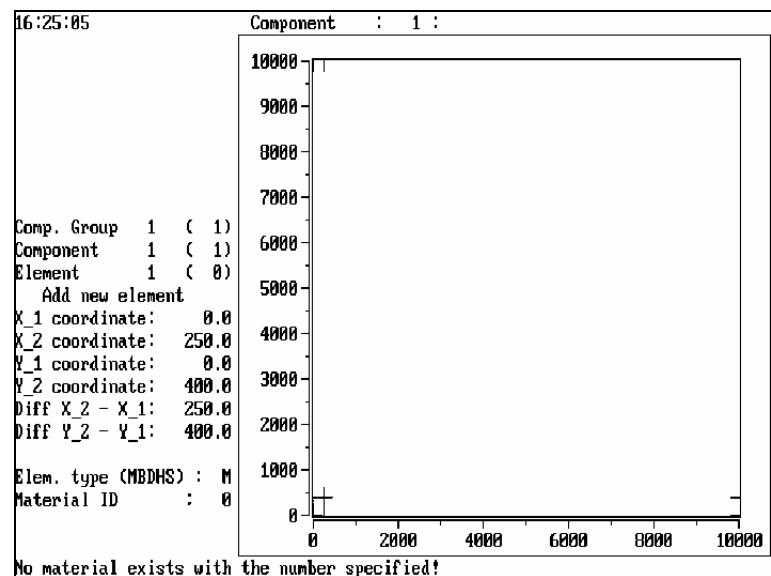
The element type is entered as **s** (space), whereupon a message appears informing the user that no transfer coefficient with this number (in this case, **0**) exists yet: an entry in the appropriate materials list must be made. To enter the first "material" in the transfer coefficients table, press **Alt-M**. A line then appears at the bottom of the screen for assigning a surface transfer coefficient (α: **20**) and a name (e.g. **exterior**) to the number **0** - confirm each field with **Enter**. Revisions to this table entry (or to any material currently in use) can be made at any time by pressing **Alt-M** to access the input line for the currently displayed material number.

Further entries in the various materials lists can either be entered as needed in the course of model input (with **Alt-M**, as above) or appended to the currently accessed list for later use. To add a material which is not to be used for the number currently displayed, press **Alt-A** (for 'append').

The table entries added in this manner are automatically numbered and included in the selection list for the respective element type. Material numbers can then be conveniently selected for subsequent elements of the same type by calling up the complete list with **F2**.

When generating a complex model requiring extensive materials lists, the method of systematically completing tables "in one sitting" can help avoid mistakes and save time in the long run. To illustrate this principle, the transfer coefficients table shall be completed at this point by pressing **Alt-A** and entering the data for transfer coefficient **1**: α: **6** $\rightarrow$ **Enter** $\rightarrow$ **interior** $\rightarrow$ **Enter**.

Input of the first element is completed by entering space (case) and sub-space numbers in the last line of the input field:

space # **0** (exterior), and sub-space # **0**.

Upon confirmation of the last input field, the display status moves on to element 2 automatically, while the coordinate values entered for the previous element still remain visible. Re-scaling the control graph now (**Alt-V** $\rightarrow$ **Full scale** $\rightarrow$ **F9**) provides an appropriate display for continuing work.

Main Screen - 2

```
16:19:00                    Component    :   1 :
                                     400

                                     350

                                     300

Comp. Group   1   ( 1)               250
Component     1   ( 1)
Element       2   ( 1)
   Add new element                   200
X_1 coordinate:      0.0
X_2 coordinate:    250.0
Y_1 coordinate:      0.0
Y_2 coordinate:    400.0               150
Diff X_2 - X_1:    250.0
Diff Y_2 - Y_1:    400.0               100

                                      50
Elem. type (MBDHS) :  S
Transfer coeff. ID:   0                 0
Space#:  0 sub-numb:  0                   0    50   100   150   200   250
```
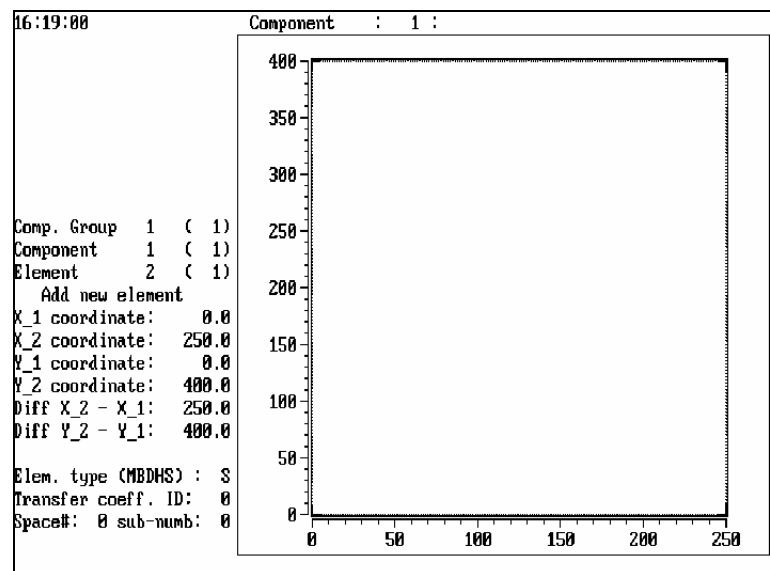
Element 2, the concrete masonry unit, is entered overlapping - thus displacing - almost the entire region defined by the exterior space (only a minimal region must remain along the component surface for valid evaluation results). Building up the model in a sequence analogous to "peeling an onion" simplifies coordinate input considerably, since the coordinates of the previously entered element are initially displayed and can be used as a reference or even re-entered if still valid.

In the case of the concrete masonry, which also extends from cut-off plane to cut-off plane in the x direction, the values for **X_1** and **X_2** can be left as they are. Only **Y_2** needs to be changed by moving the cursor to either **Y_2:** or **Diff Y_2 - Y_1:** and entering **300** mm.

Now the element type must be specified as **M** (building material), whereupon the message that no material list exists yet appears again. The same procedure used for producing the transfer coefficients table can be used here to establish a materials table.

- First enter the data for material number 0 (currently in use) by pressing **Alt-M** →(line appears along bottom of screen) **Mat.No.: 0** , thermal conductivity **lambda: 0.14** → **Enter** → **Name: precast concrete** → **Enter**.

- Then define the next material table entry for later use by pressing **Alt-A** → **Mat. No.: 1** → **lambda: 1.5** → **Enter** → **Name: reinforced concrete** → **Enter**.

Element 2 is now complete and can be confirmed either with **F9** or by pressing **Enter** in the last input field. The updated control graph now shows the element rectangle filled with a colour corresponding to the thermal conductivity of precast concrete (the colour ranges used by WAEBRU in this display are defined in the file **WAEBRU.INI**; see Appendix B, **Configuration Data**).

Main Screen - 3
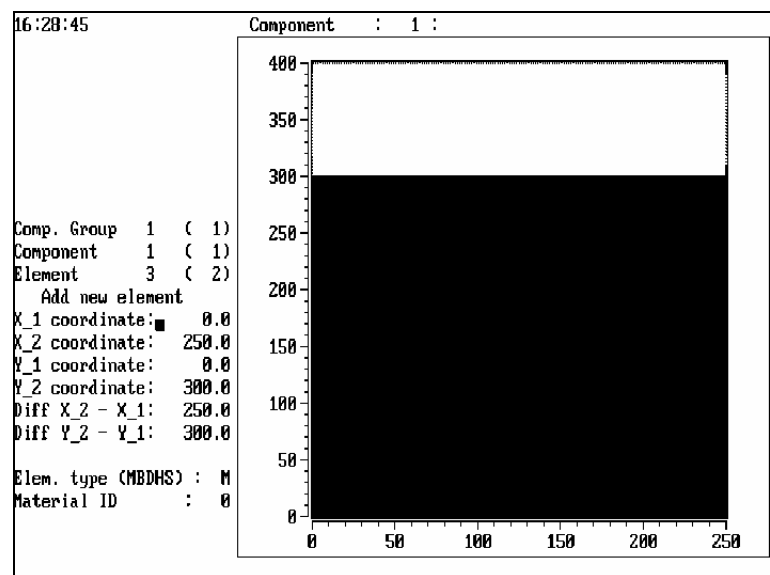
The next element in the "onion skin" sequence is the interior space. Again, most of the coordinates displayed from the previous element (**X_1, X_2,** as well as **Y_1**) still hold. Solely **Y_2** needs to be changed to account for the wall thickness by subtracting 200 mm from the displayed value of $300 \rightarrow$ **Y_2: 100**.

Once the element type has been specified as space (**s**), a transfer coefficient number can be either entered directly (**1**) or selected from the previously completed table. This table is called up by pressing **F2**. The line describing transfer coefficient 1 - **interior** - is marked using the cursor keys and then selected by pressing **Enter**.

Element 3 corresponds to the first (and only) room of this model and shall therefore be assigned the space number **1**. Since this example is very simple in nature, a single sub-space (# **0**) suffices here as well.

Upon confirmation of element 3, the control graph shows a wall section flanked above and below by two strips of black space (exterior and interior). Now the inner structure of the wall can be drawn by displacing precast concrete with one final rectangular element: the reinforced concrete which fills the hollow cell of the masonry unit.

Main Screen - 4

```
16:36:24              Component    :   1 :

                                  400 ┐
                                  350 ┤
                                  300 ┤
Comp. Group   1  ( 1)             250 ┤
Component     1  ( 1)
Element       4  ( 4)
    Edit current element          200 ┤
X_1 coordinate:    10.0
X_2 coordinate:   250.0           150 ┤
Y_1 coordinate:   120.0
Y_2 coordinate:   280.0           100 ┤
Diff X_2 - X_1:   240.0
Diff Y_2 - Y_1:   160.0            50 ┤

Elem. type (MBDHS) : M
Material ID        :  1             0 ┘
                                      └┬────┬────┬────┬────┬────┬
                                       0   50  100  150  200  250
```
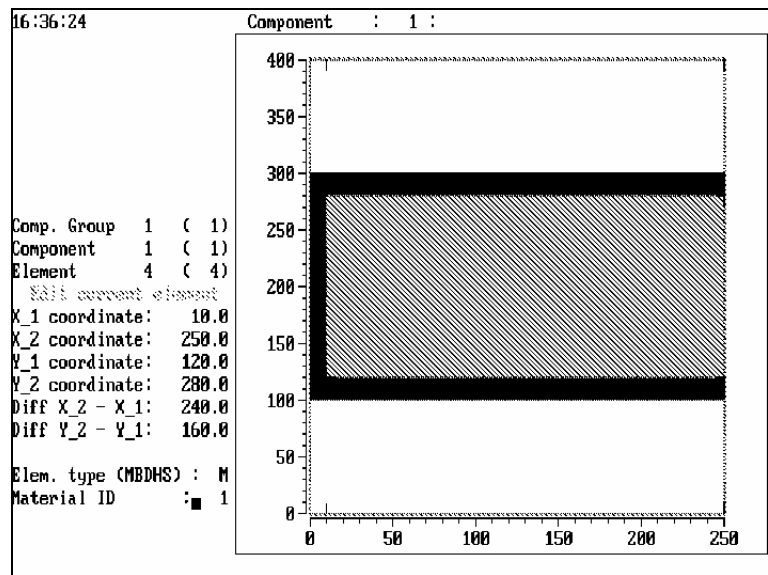
The geometry of element 4 is thus defined by the following coordinates:

> **X_1: 10** (new value)
>
> **X_2: 250**
>
> **Y_1: 120** (new)
>
> **Y_2: 280** (new)

Element type is again **M**; the material number selected is **1** for **reinforced concrete**.

The model of the thermal bridge is now complete. Generated in this sequence it contains only four elements. The same geometric model could also be generated without using the principle of element overlapping, that is, by building up the wall structure with a series of rectangles defining each cell wall of the concrete masonry unit (leading to 6 elements overall for this example). Such a method would, of course, be noticeably more complicated and prone to error in coordinate input than the "onion" sequence described above.

Once the model has been completed, the data must be written as an *aggregate* and saved in a construction file (*filename*.**2BT**). This is done by means of an option in the File Menu (**Alt-F**), which is described in a separate section later in this chapter.

Characteristics of other Element Types

heat source

The example above consists of only two element types: space and (building) material. The element type 'heat source' is treated in the same manner as building material in the input branch, i.e. it must be assigned a number from the materials table.

The difference between an element defined as a heat source (**H**) and one defined simply as material (**M**) lies in the realm of the evaluation branch, where the conditions under which the model is to be evaluated are prescribed. These conditions also include the power assumed to emanate from the heat source(s) of the model for which base solutions have been calculated. A heat source can thus also be "turned off" for the purpose of comparison between different sets of conditions (see Chapter 2.5, **Evaluation Branch**).

reinforcing bar

Since the placement of steel reinforcement in a building structure presents a special thermal problem, two different element types are provided by WAEBRU for approximating this situation in a model.

Normal reinforcing bars (axial) are those which run parallel to one of the three coordinate axes. Diagonally placed reinforcing bars constitute a separate element type (for the purpose of calculation), and must be approximated for the model as running at a slope of 45° to one of the coordinate axes. Elements of both steel reinforcement types are therefore drawn as a line rather than a rectangle, and are assigned material numbers from the same rebar table (which lists diameters with thermal conductivities).

A principal difference lies in the treatment of reinforcement in two and three dimensional modelling. When generating a two-dimensional model, reinforcing bars running parallel to one another in a plane normal to the x/y section are drawn once - in the convention of a section - and assigned an interval (i.e. distance between bar center lines) in the z direction. WAEBRU automatically models a thermally equivalent element analogous to a continuous, thin sheet of metal based on the thermal conductivity, diameter, and interval of the input rebars.

In a three-dimensional model, the reinforcement specified in a section represents a single steel bar assumed to lie in the center of the layer in which it is drawn (i.e. halfway through the defined thickness of the component group in the z direction). For models in which the steel reinforcement is a critical factor, recurring reinforcing bars must therefore be accounted for in additional layers of a thickness equal to the distance between bar center lines.

Finally, it should be noted that the input principle of element overlapping does not apply to diagonal reinforcing bars, **D.** This element type is the only exception and should therefore be entered last during model input. Axial reinforcing bars (**B**), on the other hand, can be treated in the same manner with respect to displacement by overlapping as other previously described element types.

2.3.3   Building Components and Component Groups

Models are structured as an aggregate of hierarchical input units, also referred to as object levels in this manual. Including the overall unit of *aggregate*, WAEBRU operates with four such object input levels, summarised in the following diagram.

identification	level	composed of:
number +	elements ←	(coordinates, materials)
	↓	
number +	component ←	(elements . . .)
	↓	
number	group ←	(components . . .)
	↓	
filename	**aggregate** ←	(component groups . . .)
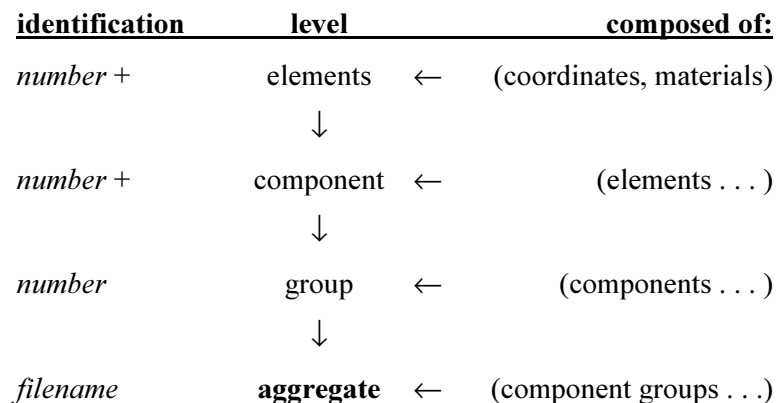
components

Depending on the nature of the object to be analysed, it could be of advantage to structure model input by sub-dividing it into several building components. These can be manipulated separately (see **Manipulation Menus** below) and subsequently combined in component groups.

After the input of a component has been completed, the next one is begun by pressing **Ctrl-PgDn** to display an 'empty' component of the next number. Once more than one building component has been generated within a particular component group, **Ctrl-PgUp** and **Ctrl-PgDn** can be used to page through these components.

groups

As explained in the section on **Modelling the Object**, component groups constitute the layers of a three-dimensional model. These can also be manipulated as a unit with manipulation menus.

The function keys **F3** and **F4** serve to move the display through component groups. The input of each additional component group (model layer) is started by pressing **F4** to move on from the last existing group into a subsequent new - empty - component group (see also **Insert/Delete Menu**).

display

The **View Menu** (**Alt-V**) provides the primary set of options for controlling which level (component, group, or aggregate) is displayed for input and editing in the name field and control graph (see **Main Screen** above):

- **Component display** means that only the building component currently in progress is shown in the control graph.

- **Group display** shows all building components of the current component group (this is the standard working level for generating a three-dimensional model).

- **Aggregate display** allows the entire model (aggregate) to appear simultaneously in the control graph (this level is used when working in two dimensions only).

Elements, components, and groups are numbered as they are entered. This information (current and total numbers) is displayed in the status fields of the Main Menu, which can be accessed (**Alt-N**) and changed, thus providing an additional method for directly paging to a particular previously input element of the model.

2.3.4   Manipulation Menus

Three further operations menus are provided in the input branch of WAEBRU for manipulating various structural levels (element - component - group) already generated for a model in progress: the Copy Menu, the Reposition Menu, and the Insert/Delete Menu.

Copy Menu

The menu used for copying individual elements, components, or entire component groups is accessed by pressing **Alt-C** (for 'copy'). Such operations can be particularly useful when generating a three-dimensional model, since a considerable amount of input work can be saved by copying a component group from one layer to the next and editing as needed in succession.

```
- Copy:
      Element
      Component
      Group (of comp.)
```

The Copy Menu offers three options for the level of an object to be copied: **Element** (**Alt-E**), **Component** (**Alt-C**), or **Group (of comp.)** (**Alt-G**). Upon activating one of these options, a window appears requiring the identification number(s) of the object to be copied. For example, individual elements are identified with a total of three numbers: number of the element within a given component (number) of a particular group (number).

An object is copied as a complete unit of the level specified, and can therefore be inserted into an object of a higher level , but not appended to an object of the same level. In other words, a component can be enlarged by inserting copied elements, but not by adding a copied component. The same is true of component groups - copied components can be inserted into a group, but copied component groups constitute a separate unit and remain complete as copied.

No special consideration needs to be taken if an object is copied into an empty component group. If elements, however, already exist in the current input group, the copied object (of whatever level) is automatically inserted ahead of the currently displayed object (of the respective level). This means that elements copied into an existing component are placed "under" the element currently displayed with respect to the overlapping sequence. The same principle is valid for inserting copied components into a group: these are also placed ahead of and (if overlapping) displaced by the currently displayed component.

Reposition Menu

```
     Move
     Rotate

- Mirror along axis:
        Horizontal
        Vertical
```

Objects can also be manipulated through geometric repositioning by means of a menu which is called up by pressing **Alt-R** (for 'reposition'). The object upon which the operations of this menu are performed is defined by the level currently displayed in the name field of the Main Screen (component, group, or aggregate). Such manipulations can be useful when generating a model in which a particular component appears repeatedly in different locations. A prototype of the component needs to be entered only once and copied (**Alt-C**). The copies can then be moved, rotated, or mirrored as needed for the final aggregate.

The options of this menu require further specifications to be entered in an input window which appears when an option is activated. Once the necessary data has been confirmed with **Enter**, the input window is closed automatically and the object is shown moved, rotated, or mirrored in the Main Screen. All repositioning operations can be cancelled at any time during execution by pressing **Esc**.

move

The first option in the reposition menu is **Move** (**Alt-M**). Upon activating this option, a window appears requiring numeric input to specify the move to be made:

```
        X from:
           to  :
        Y from:
           to  :
    X distance:
    Y distance:

            MOVE
```

Here the repositioning of the object can be defined either by beginning and end points or by a displacement vector (x/y distances), i.e. either four coordinates (first four lines) or two axial distances (last two lines) must be typed in the appropriate fields and confirmed with **Enter**.

rotate

The second option is **Rotate** (**Alt-R**), which leads to the following input window:

```
┌────────────────────────┐
│  Point of rot.:X=       │
│                 Y=      │
│  Degrees of rot.:       │
│  (-90° - clockwise   )  │
│  ( 90° - counter-cw )   │
│  (180° - . . .       )  │
└────────────────────────┘
```

The coordinates of the point about which the object shall be rotated are entered in the x and y fields. As suggested by the text in the window, the third field requires the input of one of three possible degrees of rotation: **-90** for clockwise rotation, **90** for counter clockwise rotation, or **180** for inversion about the point of rotation. Upon confirmation with **Enter**, the object appears rotated by the amount (in degrees) specified here.

mirror

The last two options of the reposition menu are for re-drawing an object by reflecting it about either a **horizontal** (**Alt-H**) or a **vertical** (**Alt-V**) mirror axis. Depending on which of these two options has been selected, one of two possible input windows appears in which either a y coordinate (position of horizontal axis) or an x coordinate (position of vertical axis) must be specified. When the numeric input of this one field has been confirmed with **Enter**, the object appears reflected about the specified axis.

Insert/Delete Menu

As described earlier in this chapter, a building component or component group can be added to a model by scrolling through to the 'end' of an aggregate and entering it as the new last component/group.

```
┌────────────────────────┐
│  Insert  component      │
│  Delete  component      │
│                         │
│  Insert  group          │
│  Delete  group          │
│                         │
│  Delete  aggregate      │
└────────────────────────┘
```

To enter a component at a different location in the overlapping sequence or to insert a new layer (component group) in a three-dimensional model, the insert/delete menu must be accessed by pressing **Alt-X**. This menu includes insert as well as delete options for building components and component groups.

The component/group ahead of which a new component/group is to be inserted must be currently displayed in the Main Screen (before calling up the menu). The same applies to a component/group which is to be deleted through a menu operation. Once an option has been selected, the menu is closed and the operation executed. All subsequent components/groups are re-numbered automatically. In the case of an insertion, an empty component/group - ready for input - appears in the Main Screen display.

A delete option is also available to erase the entire aggregate generated thus far. However it is applicable only to an aggregate which has not been saved (written) as a construction file (see **File Menu** further on in this chapter).

2.3.5   **Tables**

The contents and operation of tables used by WAEBRU (selection lists for materials) have been described in connection with various other aspects of the input branch. For the purpose of a coherent reference, the principles governing the use of tables, including relevant functions, are summarised as a separate topic in this section.

Materials (more specifically, the thermal properties of materials) are assigned to elements by number from three different tables in WAEBRU. Which table is used depends on element type:

Table / Data		Element Type
(building) materials	$\rightarrow$	material (**M**)
- λ, identification		heat source (**H**)
(steel) rebars	$\rightarrow$	axial reinforcing bar (**B**)
- λ, diameter, ident.		diagonal reinforcing bar (**D**)
(surface) transfer coefficients	$\rightarrow$	space (**S**)
- α, identification		

The contents of all tables must be entered by the user for the model currently being generated. Two functions are available for establishing the entries in tables:

- To enter or edit a currently used material (i.e. of the number assigned to the currently displayed element), press **Alt-M**.

- To enter an additional material (i.e. of the number following the last entry in the list) to the table which is appropriate to the current element type, press **Alt-A**.

Established tables are accessed for material selection by pressing **F2**, whereupon the table appropriate to the currently assigned element type appears on the screen in place of the control graph. A material can then be selected in the same manner as in any list (by using the cursor keys to mark and pressing **Enter**, thereby also closing the list). To return to the Main Screen without making a selection, press **Esc**.

Table Screen

```
16:30:46                    Component    :   1 :
                        ┌──────────────────────────────────────────┐
                        │   ID   Lambda  Name                       │
                        │                                           │
                        │ A-  0     .140  precast concrete          │
                        │ B-  1    1.500  reinforced concrete        │
                        │                                           │
                        │                                           │
                        │                                           │
Comp. Group   1  (  1)  │                                           │
Component     1  (  1)  │                                           │
Element       4  (  4)  │                                           │
   Edit current element │                                           │
X_1 coordinate:    10.0 │                                           │
X_2 coordinate:   250.0 │                                           │
Y_1 coordinate:   120.0 │                                           │
Y_2 coordinate:   280.0 │                                           │
Diff X_2 - X_1:   240.0 │                                           │
Diff Y_2 - Y_1:   160.0 │                                           │
                        │                                           │
Elem. type (MBDHS) :  M │                                           │
Material ID       :   1 │                                           │
                        └──────────────────────────────────────────┘
```

2.3.6

```
┌──────────────────────────────┐
│    Read file                 │
│    Write aggregate           │
│                              │
│ --- Data printouts ---        │
│    Geometric data             │
│    Tables                     │
└──────────────────────────────┘
```

File Menu

Once a model has been completed, or whenever input work must be interrupted, the data defining geometry and materials must be saved in a construction file. This is done by means of an option in the file menu, which is accessed from the Main Screen by pressing **Alt-F**.

Activating the option `Write aggregate` (**Alt-W**) leads to the file survey screen, in which the *filename* specified at the start of model input is suggested in the first line. Aggregate data is written onto a construction file with the name displayed here upon confirmation with **Enter**. The filename associated with a model can be changed at this point, if desired, before pressing **Enter** to execute file generation.

The option `Read file` (**Alt-R**) allows the user to continue input work with a different model - or to start input of a new model - without leaving the input branch. Activating this option also leads to the file survey screen for further specification (see the section on **File Survey**, Chapter 2.1).

The last two options of the file menu are for printing out a documentation of the aggregate data (construction file) as described in the next section.

Model Documentation

All input data recorded in the construction file of a model (not grid data) can be documented in hard copy form by means of printout options in the file menu (**Alt-F**). The option `Geometric data` (**Alt-G**) provides a comprehensive printout of all element coordinates, types, and materials, headed by the project description as entered in the text field during input.

The first page includes a quantitative summary of the aggregate structure according to component groups, components, and elements. Subsequent pages show all element data listed by level (component/group names) and number.

Since the documentation of geometric data can be quite extensive (especially for three-dimensional models), a separate option is provided to document material data alone: **Tables** (**Alt-T**). This option prints the information contained in the various material lists used for the model in progress. A copy of the materials, rebars, and transfer coefficients tables can be useful to have at hand when continuing work on a larger model.

2.3.8

Generating the Grid

As already described in the first part of this manual, the principle behind the calculation method of WAEBRU lies in balancing the thermal characterisations of orthogonally subdivided regions. Each subdivision, or *cell*, is thus geometrically bound by orthogonal *grid planes*; these must be defined such that each resulting cell contains only a single building material.

The coarsest grid structure which satisfies the requirement of homogeneous cells is referred to as the **minimal grid**. This can be derived directly from the geometric data of a model.

Since a minimal grid is generally too coarse a subdivision to provide dependably accurate results by numerical methods, a further - finer - gridding of the model must be performed for calculation. WAEBRU offers two different methods for generating such auxiliary grid planes:

- automatically - by using the **fine grid** generated by the program according to pre-defined parameters.

- manually - by directly specifying individual **supplement grid** planes to be included in the gridding structure.

Once a grid is complete for calculation, it must be recorded in a grid file for use in the calculation branch. All operations leading to the generation of this grid file are performed through the grid menu (**Alt-G**), which is accessed from the Main Screen of the input branch upon completing model input.

Grid Menu

When the grid menu is called up with **Alt-G**, the Main Screen display is also altered to accommodate the information needed for grid generation. The actual menu appears in place of the status fields in the upper left area of the screen. The element input fields below are replaced by input fields specifying fine grid parameters.

A schematic representation of the aggregate (line drawing without materials) is initially displayed in the control graph. When stages of gridding have been completed, this display area shows the grid as generated thus far (rendered two- or three-dimensionally, depending on the model).

Gridding Screen - 0

```
16:43:57              Aggregate    :
                      400
Read Grid
Write Grid
                      350
Min.grid - generate
Supplement grid       300
Change parameters
Fine grid - generate  250
1-delete min. grid
2-delete fine grid
3-delete grid lines   200
 0 - Print grid data

Fine grid from:       150
    Space bound. : Y
    Lambda quot. : N  100
        Lim. : 10.000
    Rebar bound. : N   50
    Supp. at mat.: N
Init.thickn. :  5.00    0
Coarsening   : 2.000      0   50  100  150  200  250
Max. thickn. : 100.00
    Ignore XYZ  :
```

read/write grid

The first two options listed in the grid menu are analogous to the read and write options of the file menu. If an appropriate grid file has already been generated at an earlier point in time, the gridding data from this file can be loaded by selecting the option **Read grid** (**Alt-R**), which calls up the file survey screen for *filename* selection.

The option **Write grid** must be executed (**Alt-W**) after gridding is finished before leaving the grid menu. This operation also leads directly to the file survey screen, in which a *filename* for the grid file to be generated is confirmed (see also the section on **File Survey**, Chapter 2.1).
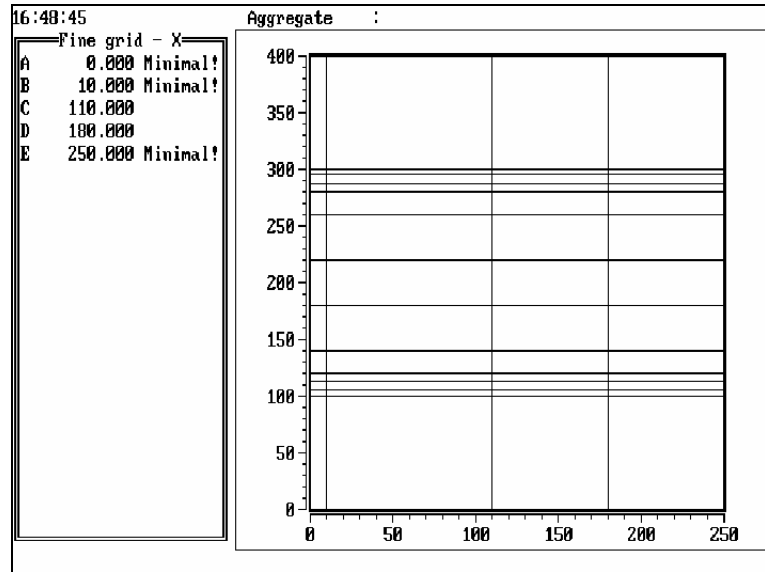
gridding options

The second block of the grid menu contains four options relevant to grid generation and three options for deleting parts of an existing grid. The stages of generating minimal, fine, and supplement grids are described in detail in the following sections.

A complete grid thus generated can be editted by means of selective deletion. If the option **1-delete min. grid** is chosen and entered, both the minimal grid and the automatically generated fine grid are erased from the existing grid structure (any supplement grid planes which have been added manually still remain). The option **2-delete fine grid** causes only the fine grid - but not the minimal grid - to be erased from the current grid structure.

The most important option for editing an existing fine grid is **3-delete grid lines**. Upon selecting this option and pressing **Enter**, a list of all grid planes for a given reference axis appears to the left of the graph. Such a selection list is available for every axis direction pertinent to the model; to change lists, press either **Alt-X**, **Alt-Y**, or **Alt-Z** respectively.

Border planes of the fine grid are identified by coordinate position in the x, y, or z direction listed. An individual plane is simply selected for deletion either by marking it using the cursor keys or by typing the letter of the desired line (scroll the list with **PgUp** and **PgDn**); upon pressing **Enter**, the marked line (border plane) is erased.

Gridding Screen - list

```
16:40:45              Aggregate    :
┌──Fine grid - X──┐   400 ┐
│A    0.000 Minimal!│
│B   10.000 Minimal!│
│C  110.000        │   350 ┤
│D  180.000        │
│E  250.000 Minimal!│   300 ┤
│                 │
│                 │   250 ┤
│                 │
│                 │   200 ┤
│                 │
│                 │   150 ┤
│                 │
│                 │   100 ┤
│                 │
│                 │    50 ┤
│                 │
│                 │     0 ┘
└─────────────────┘       0   50  100  150  200  250
```

The delete function applies to fine grid lines only; planes identified in the list as belonging to the minimal or supplement grids (**!**) cannot be deleted from the grid file here. The deletion list is closed by pressing **Esc** to return to the grid menu.

grid documentation

The grid menu also offers the possibility of printing a copy of the fine grid parameters applied and/or a comprehensive list of the coordinates of all grid borders. This is done by means of the last option, **@ - Print grid data**, but should be used sparingly, since grid data tends to be very extensive (particularly for three-dimensional models).

A graphic documentation for comparing grids can be obtained by printing the contents of the screen using **Alt-P**. This can be used to document the various stages of gridding from the initial line graph of the aggregate to the final fine grid, including supplement grid planes. Two-dimensional grids are drawn in x/y section; three-dimensional grids are rendered axono-metrically.

Minimal Grid Generation

The first stage of gridding is the generation of a minimal grid based solely on the geometric data of the model. This is done automatically by activating the option **Min. grid - generate** (**Alt-M**). The element borders are hereby re-defined and drawn as planes extending over the entire region of the model in the control graph.

The minimal grid thus generated forms the basis for all further gridding, but it does not constitute a complete grid structure. A fine grid in some form must be generated and stored in a grid file before moving on to the calculation branch of WAEBRU.

In the event that a minimal grid would suffice for calculation (e.g. for preliminary evaluation of a model), it must be re-generated as a fine grid based on the appropriate parameters (see the section on **Fine Grid Parameters**).

Gridding Screen - 1

```
16:45:10              Aggregate    :
┌─────────────────┐  400┤┌──────────────────────────┐
│Read Grid        │     ││                          │
│Write Grid       │     ││                          │
│                 │  350┤│                          │
│Min.grid - generate│   ││                          │
│Supplement grid  │     ││                          │
│Change parameters│  300┤│──────────────────────────│
│Fine grid - generate│  ││──────────────────────────│
│1-delete min. grid│    ││                          │
│2-delete fine grid│ 250┤│                          │
│3-delete grid lines│   ││                          │
│ 0 - Print grid data│  ││                          │
└─────────────────┘  200┤│                          │
                        ││                          │
Fine grid from:      150┤│                          │
     Space bound. : Y    ││──────────────────────────│
     Lambda quot. : N    ││                          │
          Lim. : 10.000  ││──────────────────────────│
     Rebar bound. : N 100┤│                          │
     Supp. at mat.: N    ││                          │
Init.thickn. :   5.00  50┤│                          │
Coarsening   : 2.000     ││                          │
Max. thickn. : 100.00    ││                          │
      Ignore XYZ :     0 ┤└──────────────────────────┘
Minimal grid composed of  0    50   100  150  200  250
                         2*5*1 cells.
```
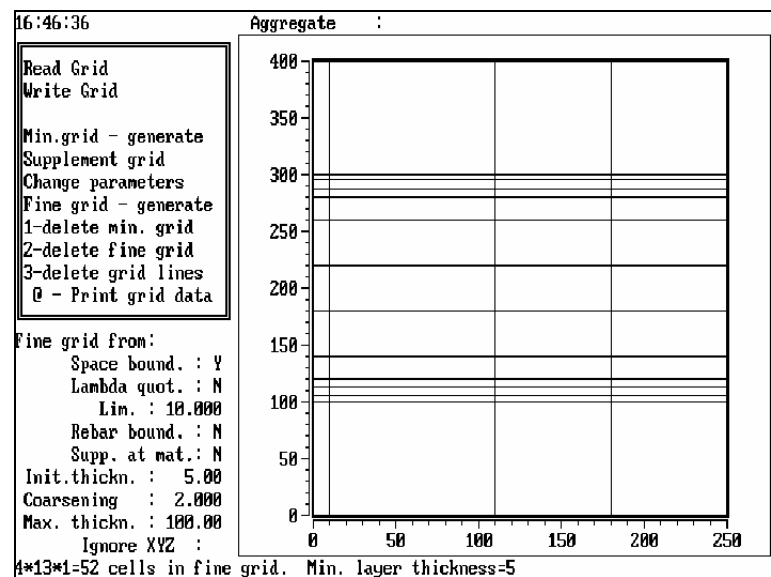
Fine Grid Generation

The next stage of gridding is the automatic generation of a fine grid. If the standard settings for the gridding parameters are acceptable (which is generally the case), this stage is performed by simply activating the option **Fine grid - generate** (**Alt-F**) in the grid menu.

Once computation of the fine grid is complete, the total number of cells in the resulting grid structure ($n_x \cdot n_y \cdot n_z$) as well as the minimum layer thickness (distance between two border planes) are displayed in the bottom line of the screen. Two-dimensional grids always have only one layer in the z direction ($n_z = 1$).

The total number of grid cells displayed here provides a preliminary indicator for the ultimate size of the calculation model, but it also includes a significant number of cells located outside of the material building components of the model. Since such cells are not ultimately calculated, a grid cell number which is somewhat higher than the maximum possible number of calculation cells allowed by the hardware capacity may still be acceptable.

Gridding Screen - 2

Fine Grid Parameters

If the standard fine grid should prove to be inadequately structured (too fine for fast calculation or too coarse for precision), the various parameters governing the process of automatic grid generation can be manipulated by means of the option **Change parameters** (**Alt-C**), which activates the input fields below the grid menu.

The fine grid is generated according to the same principle in all three coordinate directions. Starting at a border plane of the minimal grid, first a layer of a pre-defined *initial thickness* is produced. The initial thickness is then multiplied by a *coarsening* factor to obtain the thickness of the second grid layer to be generated next to the first.

This procedure is repeated until the boundaries of the building component have been reached, whereby the coarsening of the sequential layers is only performed up to the specified *maximum thickness*.

The fine gridding procedure described above can be started at either or both of two border plane types:

- planes defining the outer boundaries of spaces (adjacent to building components) → the parameter `Space bound.:` (`y` or `n`).

- planes separating cells filled with different materials for which the ratio of respective thermal conductivities exceeds a specified maximum → the parameters `Lambda quot.:` (`y` or `n`), and `Lim.:` (numeric input).

The standard setting is for grid planes to be generated starting at space boundaries, but not at lambda quotient borders (`Space bound.: Y, Lambda quot.: N`)

A fine grid equivalent to the minimal grid is generated if neither one of these starting plane types is chosen (by entering `n` in both respective parameter input fields) and the `Max. thickness` specified is appropriately large.

The parameters `Init. thickness, Coarsening,` and `Max. thickness` can be freely defined, the only restriction being that the maximum thickness must, of course, be larger than the initial thickness.

In addition to the grid planes generated from a starting plane, individual planes can also be supplemented automatically along all building material boundaries and/or reinforcing bar planes at a distance equal to the initial thickness, without further repetition using the coarsening factor. The parameters for specifying this supplement option are `Rebar bound.:` (`y` or `n`) and `Supp. at mat.:` (`y` or `n`). The final parameter, `Ignore XYZ`:, is rarely implemented. It allows the user to suppress grid planes in one or more axial directions by specifying reference direction(s) to be eliminated from the fine grid.

Parameter input is confirmed and closed by pressing `F9` (or `Enter` from the last line). Any changes made can also be cancelled by pressing `Esc` to return to the grid menu.

Supplement Grid Planes

A further means of manipulating the fine grid is through augmentation by a supplement grid which has been generated with the grid menu option `Supplement Grid` (`Alt-S`).

Upon activating this option, a border plane list - similar to the deletion list - appears showing all grid planes of the currently listed coordinate direction generated thus far. The reference coordinate can be changed with `Alt-X`, `Alt-Y`, or `Alt-Z` respectively. Border planes inherent to the minimal grid as well as any already existing supplement grid planes are separately identified (`!`) to the right of the position coordinate.

Gridding Screen - supp. grid

Grid planes are individually specified by entering the position (coordinate of the reference axis) of the planes one by one in the input field located under the list.   By pressing **Enter** after each coordinate input, the supplement planes are inserted in the list and marked with `Supp.!`.

Supplement grid planes can also be individually deleted from the list here in the same manner as with the previously described option `3 - delete grid lines`, i.e. by using the cursor keys to mark a line to be deleted and then pressing **Enter** (the list can be scrolled with **PgUp** and **PgDn**).

```
16:50:02
╔═Fine grid      - X═╗
║A      0.000 Minimal!║
║B     10.000 Minimal!║
║C    110.000         ║
║D    180.000         ║
║E    250.000 Minimal!║
║                     ║
║                     ║
║                     ║
║                     ║
║                     ║
║                     ║
║                     ║
║                     ║
╚═════════════════════╝
Suppl.: ▪    0.000
```

All the supplement grid planes of a model remain recorded in a separate file by WAEBRU until the input branch is left. This allows the user to edit - or even completely delete and re-generate - the fine grid without affecting any defined supplement grid planes. Once a fine grid has been established, the user is asked to specify whether or not the supplement grid planes (if existing) shall be included in the final version of the grid file for calculation. Any decision here can be revised by entering the grid menu (**Alt-G**) from the Main Screen again and then pressing **Esc**, whereupon the question about the supplement grid file is repeated.