Tutorial

Geometrical modelling functions: getting started with TRISCO

TRC Tutorial a

physibel

TRISCO

1. Introduction

building_corner_a.trc

This first tutorial deals with the object modelling functions in TRISCO. The overall modelling philosophy via a coordinate system and blocks will be illustrated for a simple room corner. Thereafter, the obtained model will be used to show how to quickly combine and adjust existing models via block transformations.

Key elements:

- Creating a geometrical model from scratch
- Coordinate system and blocks
- Block transformation functions

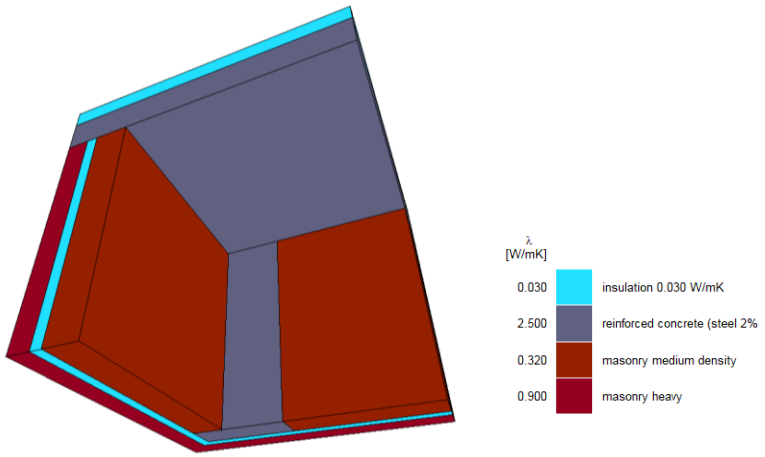
2. Case study data

The example concerns a simplified three-dimensional room corner consisting of:

- insulated cavity wall
  - outer leaf: λ = 0.9 W/(m.K), d = 10 cm
  - insulation: λ = 0.030 W/(m.K), d = 5 cm
  - inner leaf: λ = 0.32 W/(m.K), d = 15 cm
- insulated roof
  - a concrete roof slab: λ = 2.5 W/(m.K), d = 15 cm
  - insulated at the upper side: λ = 0.030 W/(m.K), d = 5 cm
- a concrete column in the corner of the inner leaf : λ = 2.5 W/(m.K), section 45 cm x 15 cm

The boundary conditions imposed are:

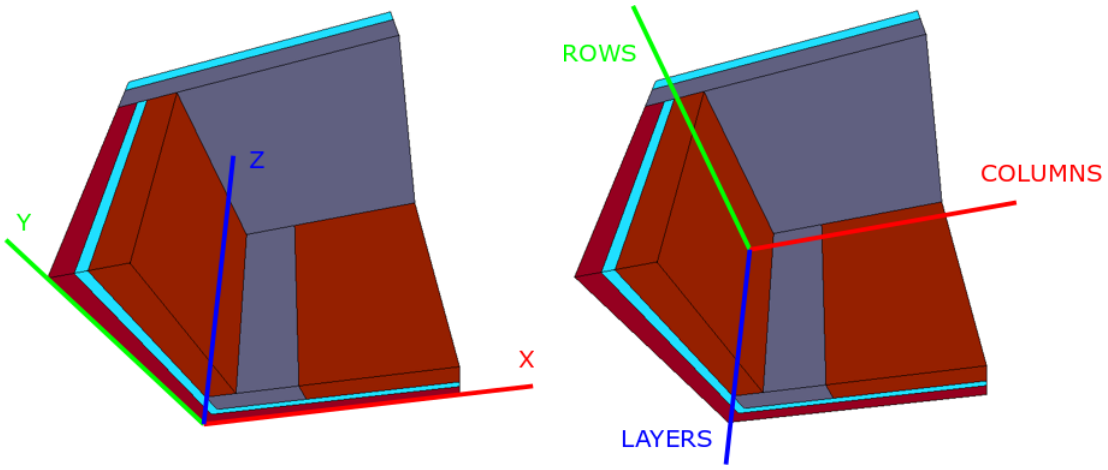
- Indoors: temperature θ_i = 20 °C, wall surface heat transfer coefficient = 5 W/m^{2}K
- Outdoors: temperature θ_e = 0 °C, wall surface heat transfer coefficient = 25 W/m^{2}K



3. Data preparation

Step 1: Choose a coordinate system

- By default TRISCO will use the classic concept of a Cartesian coordinate system XYZ to define the position of geometrical blocks. However, an alternative system defining geometrical blocks via rows, columns and layers as in a 3D matrix is also available. At any time it is possible to switch between both systems (*Settings → Show RCL* and *Settings → Show XYZ).*
  Note: the Z axis should point upwards to the sky (left figure), and layers should be numbered downwards (right figure), to get a normal natural behaviour of the rotate function (*Image → Rotate*).

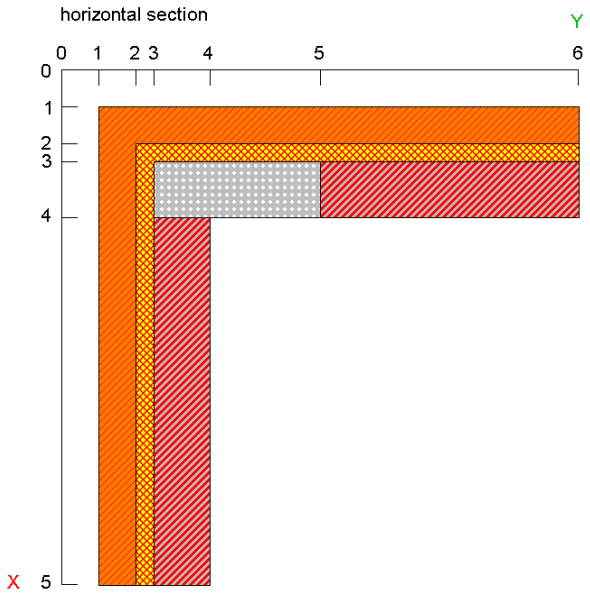


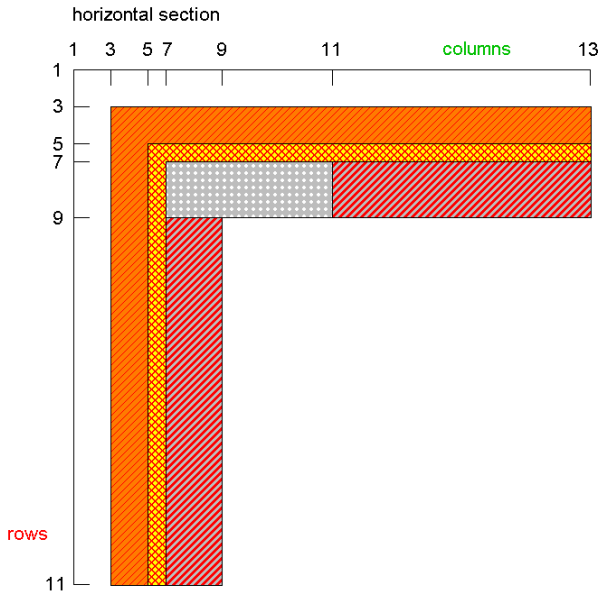
Step 2: Drawings of relevant sections

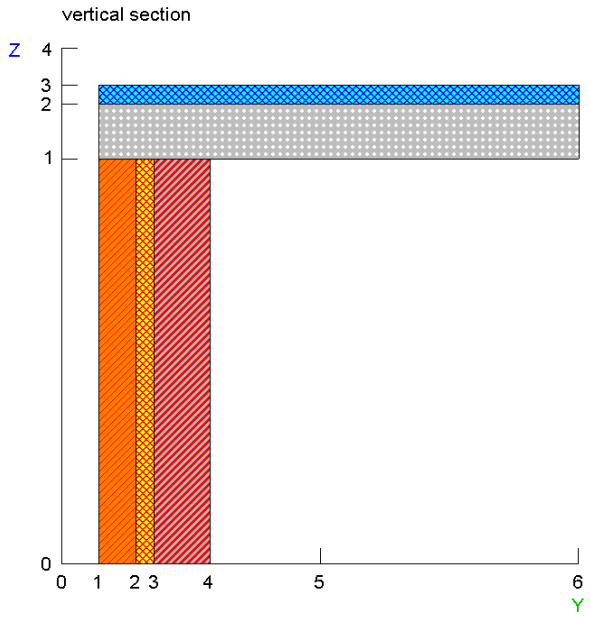
- It is useful to make a (hand-drawn) sketch of all relevant horizontal and vertical sections of the object.

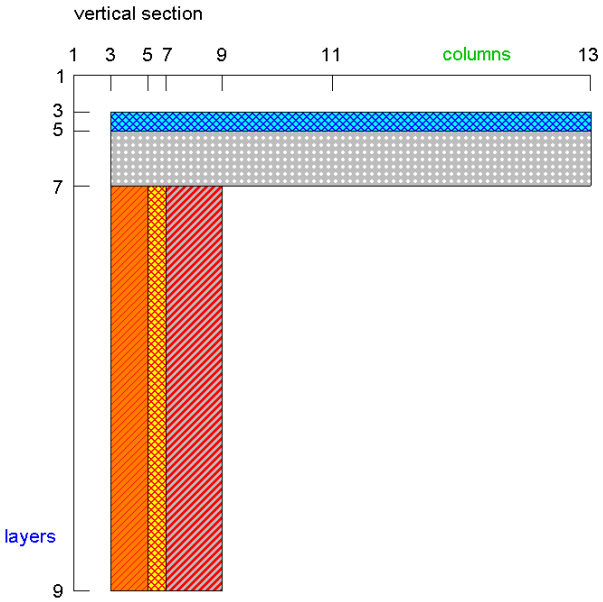
Step 3: Creation of the minium grid

- Mark all relevant coordinates on the section drawings to define the minimum grid of the object model.
- The minimum grid must contain all material boundaries and boundary condition boundaries. At the outside of the object additional grid planes may be necessary to define boundary conditions.
- The minimum grid has to be created using the following rules:
  - Each material border is a grid plane of the minimum grid.
  - On each axis the position of the minimum grid planes have to be marked and numbered.
  - In case of the XYZ coordinate system, a numbering with base 0 is used: 0,1,2,3, ...
  - If a material border coincides with the first or last grid plane, an adiabatic boundary condition (i.e. without any heat transfer) will be assumed.
  - An additional grid plane is necessary if there is a non-adiabatic boundary condition at such material border.
- For our example see the cross section drawings on the next page:
  - At grid border X = 5 (row 11), Y = 6 (column 13) and Z = 0 (layer 9) an adiabatic boundary condition is assumed.
  - Grid border X = 0 (row 1), Y = 0 (column 1) and Z = 4 (layer 1) are added in order to allow an outer boundary condition in X = 1 (row 3), Y = 1 (column 3) and Z = 3 (layer 3).









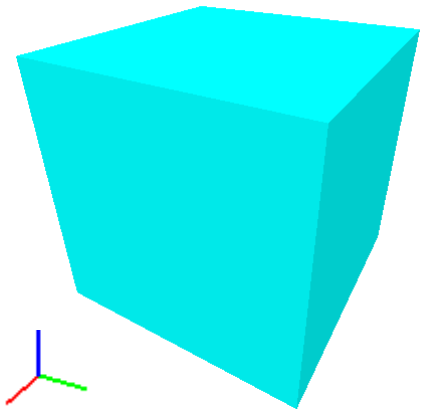
Coordinate system (XYZ) (default)

Coordinate system (RCL)

4. Entering the data

Step 4: Open a new data file

- Open a new file using the command *File* → *New* to start the data input. A default block (cube with side 1 m) with colour number 5 is shown as starting data.



Grid

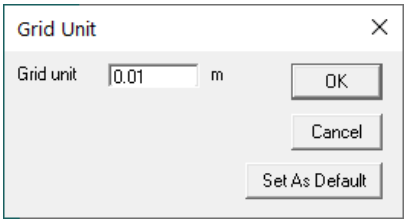
No.	X [m]	Y [m]	Z [m]
Sum	1.000	1.000	1.000
0-1	1.000	1.000	1.000

Blocks

No.		Col.	Xmin	Xmax	Ymin	Ymax	Zmin	Zmax
1		5	0	1	0	1	0	1

Step 5: Enter the minimum

- The minimum grid are dimensions needed to create the object.
  This data is entered in the **Grid window**.
  Make this window active by clicking on its title bar using the left mouse button, or by selecting the menu command *Edit* → *Grid*.
- Enter the scale of the drawing: choose the command *Grid* → *Grid Unit...* and enter 0.01 m to switch the measurements in the **Grid window** to centimeters.



- Answer the prompt "Transform grid widths ?" with Yes.
  Enter the first grid mesh width in the Grid window by typing its value (i.e. 10) and pressing <Enter>.

Grid

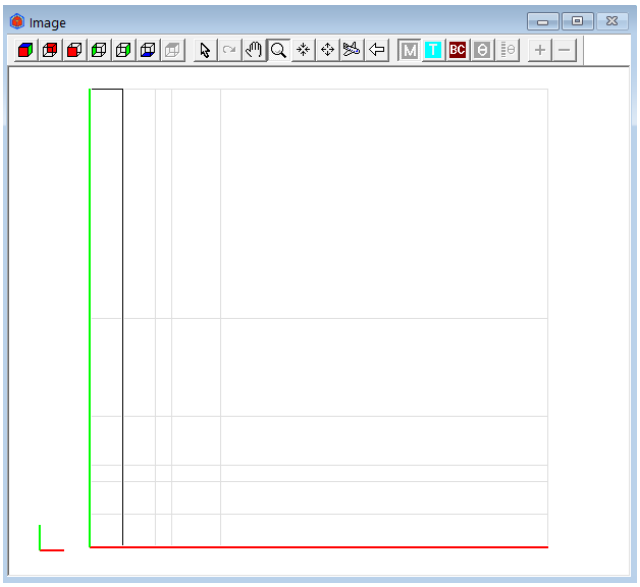
No.	X [cm]	Y [cm]	Z [cm]
Sum	10.000	100.000	100.000
0-1	10.000	100.000	100.000

Grid

No.	X [cm]	Y [cm]	Z [cm]
Sum	11.000	100.000	100.000
0-1	10.000	100.000	100.000
1-2	1.000		

- Add a new grid cell using the command *Edit* → *Insert After* or press <Insert>.
  By default the new mesh width equals 1 (cm).
  Change the inserted mesh width into 10 (i.e. the thickness of the outer leaf of the cavity wall).

Insert new mesh widths by repeating the command *Edit → Insert After* and fill in the right mesh widths. The resulting minimum grid is shown below (right).
To see the minimum grid in the XY plane, the view can be changed to Zmax (*Projection → 6: Zmax*).



Grid

No.	X [cm]	Y [cm]	Z [cm]
Sum	140.000	140.000	**130.000**
0-1	10.000	10.000	100.000
1-2	10.000	10.000	15.000
2-3	5.000	5.000	5.000
3-4	15.000	15.000	10.000
4-5	100.000	30.000	
5-6		70.000	

- It is a good pratice to save the (intermediate) data.
  Choose the command *File → Save As...* and enter a file name.
  Now the blocks can be entered using the grid coordinates as border specifications.

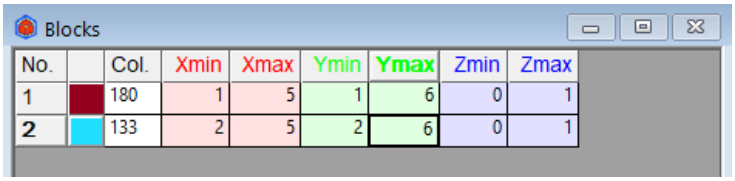
Step 6: Adding material properties and boundary conditions to the minium grid

- Materials and boundary conditions can now be defined via blocks in reference to the minimum grid. This block data have to be entered in the **Blocks window.**
- Each block contains a colour reference and border coordinates. The colour reference will be used to identify the thermal property or boundary condition.
  The border coordinates are expressed in grid numbers.
  Note: The user can overwrite colour properties in a project file and/or use predefined colours from the Colour database (see 'How to modify the Colour database?' (*Help → How to's...*). To see all predefined colours select in the menu command *Colours → List defined*).
- Change the colour reference number and coordinates of the default block, as in the figure below.

Blocks

No.		Col.	Xmin	Xmax	Ymin	Ymax	Zmin	Zmax
1		180	1	5	1	6	0	1

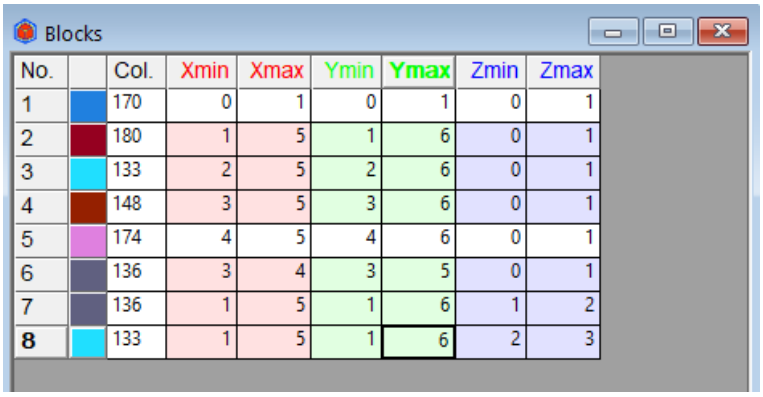
- The **Image window** shows a perspective view of the created block. The viewpoint can be changed using the commands in the Zoom submenu and mouse dragging within the **Image window**.
- Enter a new block by selecting *Edit → Insert After*.
- Enter the values as in the figure below.



Blocks

No.		Col.	Xmin	Xmax	Ymin	Ymax	Zmin	Zmax
1		180	1	5	1	6	0	1
2		133	2	5	2	6	0	1

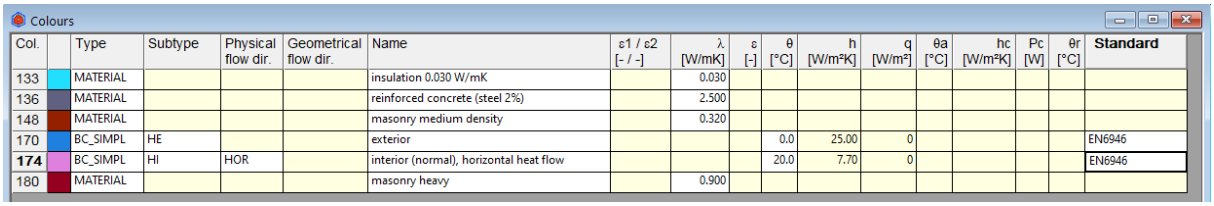
- This second block overwrites partially the first block.
  The superposition principle is crucial in the (optimal) definition of blocks: a block with higher sequence number overwrites the common parts of blocks with lower sequence numbers.
- Insert new blocks using *Edit → Insert After* or *Edit → Insert Before*, and adjust the colour and coordinate values as in the list below.



Blocks

No.		Col.	Xmin	Xmax	Ymin	Ymax	Zmin	Zmax
1		170	0	1	0	1	0	1
2		180	1	5	1	6	0	1
3		133	2	5	2	6	0	1
4		148	3	5	3	6	0	1
5		174	4	5	4	6	0	1
6		136	3	4	3	5	0	1
7		136	1	5	1	6	1	2
8		133	1	5	1	6	2	3

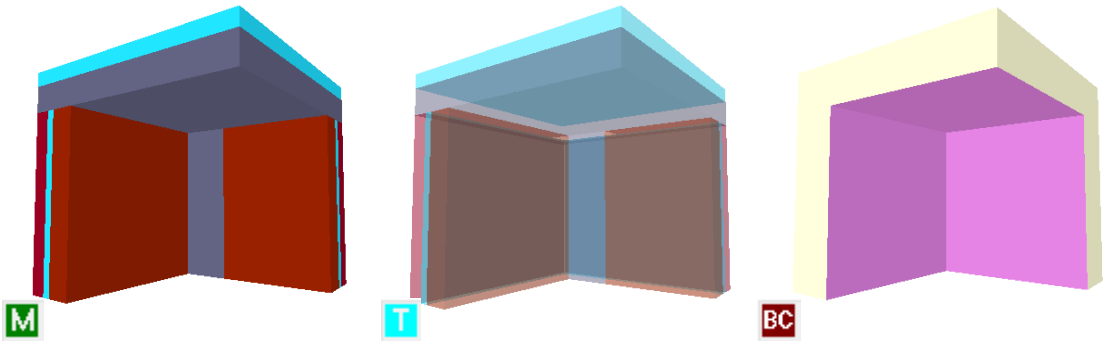
- A block is hollow when its colour corresponds to a boundary condition type in the **Colours window**. In this example colour 174 and 170 have a colour type BC_SIMPL, which stands for a (simplified) boundary condition.
- In this example we make use of the predefined colours in the Colour database (*File → Open Colour database...*) corresponding to following colour types:



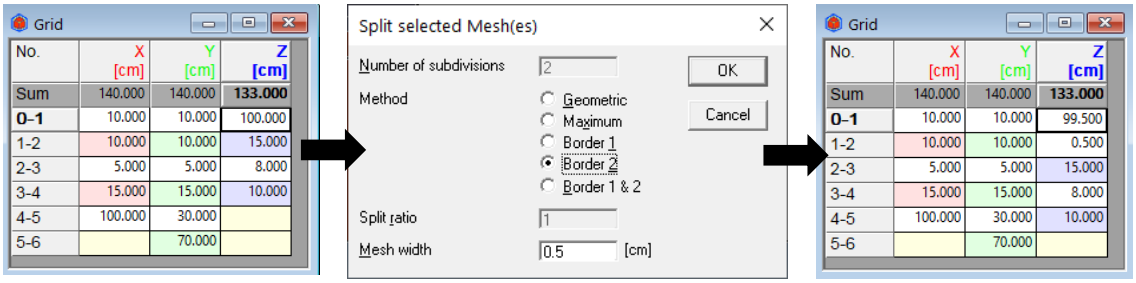
Colours

Col.		Type	Subtype	Physical flow dir.	Geometrical flow dir.	Name	ε1 / ε2 [- / -]	λ [W/mK]	ε [-]	θ [°C]	h [W/m²K]	q [W/m²]	θa [°C]	hc [W/m²K]	Pc [W]	θr [°C]	Standard
133		MATERIAL				insulation 0.030 W/mK		0.030									
136		MATERIAL				reinforced concrete (steel 2%)		2.500									
148		MATERIAL				masonry medium density		0.320									
170		BC_SIMPL	HE			exterior				0.0	25.00	0					EN6946
174		BC_SIMPL	HI	HOR		interior (normal), horizontal heat flow				20.0	7.70	0					EN6946
180		MATERIAL				masonry heavy		0.900									

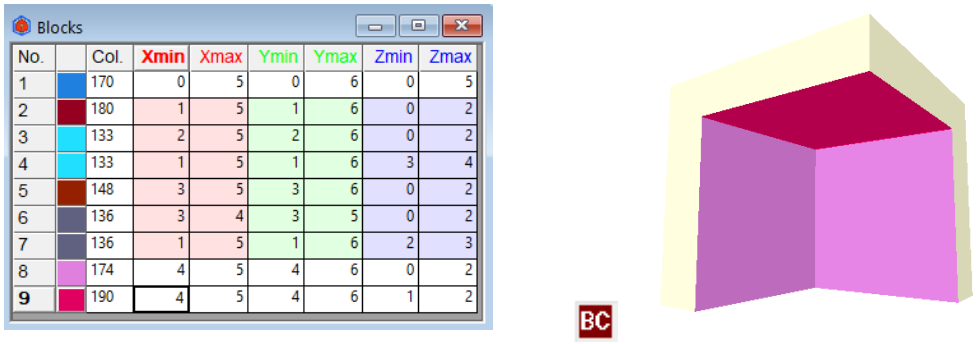
- For the Type MATERIAL a thermal conductivity and name are defined.
- For the Type BC_SIMPL following elements are defined:
  - Subtype
  - Physical flow direction
  - Geometrical flow direction
  - Name
  - Temperature
  - Global heat transfer coefficient
  - Standard
- It is possible to adjust the data in the **Colour window**; this will only be stored in the current TRISCO project file and does not affect the Colour database. Changing the fields in the first 4 columns and the last column in the **Colour window** is done by activating the cell with the cursor and pressing <Enter>.
- Now the final object is visible in the **Image window**



- It is possible to display the object transparently by selecting *Image → Show Transparent.*
- The contact surfaces of the boundary conditions on the object can be made visible by selecting the command *Image → Show BCs.*
- As illustatrated in the above (right) picture, one global surface coefficient (7.7 W/m².K) has been defined in the current configuration. To be fully in line with EN ISO 10211 the inner boundary condition should be split between a heat transfer coefficient for horizontal heat transfer (7.7 W/m².K) and vertical upwards heat transfer (10 W/m².K).
- To add the vertical upward heat tranfer coefficient to the object:
  - Select the grid mesh in Z from 0 to 1 (grid mesh Z 0-1) in the **Grid window**
  - Select *Grid → Split...* or press the shortcut key <Ctrl+I>
  - "Method": Select "Border 2"
  - "Mesh width": Define a small distance (e.g. 0.5 cm)
  - Add in the **Blocks window** a new block with colour 190 which is the predefined colour for upward heat flow (h=10 W/m².K) and coordinates X:4-5, Y:4-6 and Z:1-2.



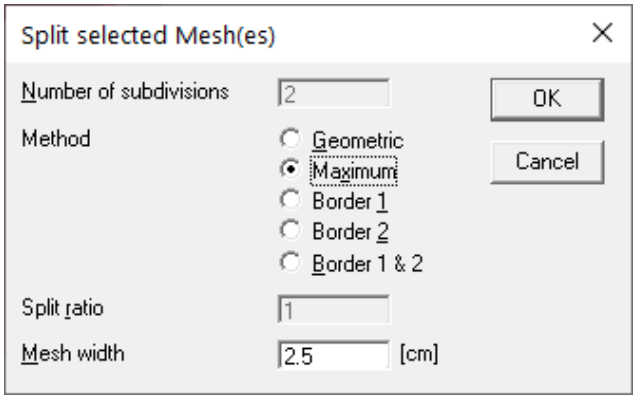
- In the Image window two different boundary conditions are visualised.



- Restore the previous view by selecting *Image → Show Materials.*
- Now all basic data are ready for a thermal calculation.
  However the grid meshes should be refined to get reliable results.

Step 7: Refine the grid

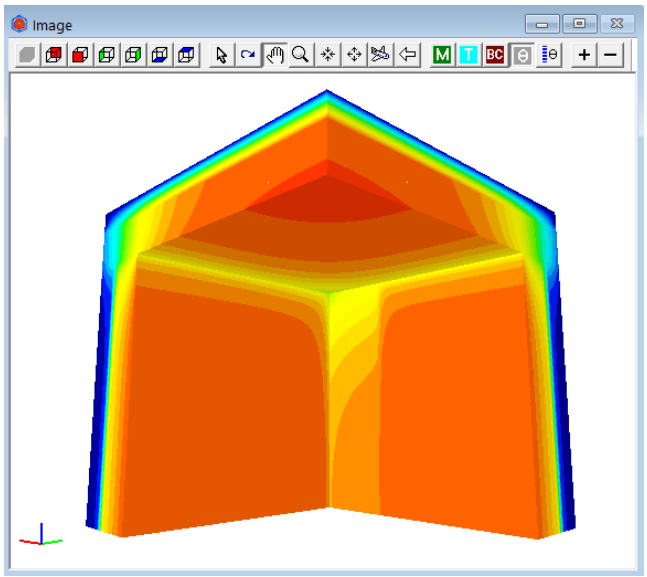
- All grid meshes are split into smaller meshes of width = 2.5 cm.
- Select all meshes using the command *Grid → Select All (3 x).* The selected meshes in the **Grid window** are coloured (light or dark) turquoise.



- Then open the Split dialog box using the command *Grid → Split...* (or press shortcut key <Ctrl+I>) and set the parameters as in the figure above right.
- These settings cause that all selected meshes are split into equal subparts of width 2.5 cm. (In case an original selected mesh is not a multiple of the new mesh width, also one smaller mesh is created containing the remainder of the division.)
- To go back to the minimum grid: *Grid → Create Minimum Grid...*
- Other mesh functions (e.g. *Grid → Auto Split...*) can be used to obtain a more balanced grid (with less calculation nodes and yet a good solution quality): building_corner_b.trc

5. System calculation

- Start the calculation: *Calc → Calc System.*
- The calculated temperatures are shown in the **Image window**.



- Move the mouse cursor over the object in the **Image window** to see (in the status bar, at the bottom of the screen) the temperature at the mouse cursor position. (This requires the **Image window** is active.)

- Detailed output results can be explored in the **Graphic Output window** and **Text Output window**.

6. Object cut operation using automatic block arrangement

building_corner_c.trc

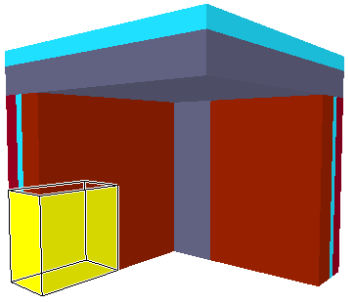
- By default the block arrangement in TRISCO is using a superposition principle as illustrated in the previous examples.
- When a part of an object must be cut, automatic block arrangement with juxtaposition can be used to remove all overlapping parts in the object.
- First mark out the cutout region with a new block (using a new colour) over the object.
    - Add grid meshes in X and Z to obtain the **Grid window** depicted below (left)
    - Insert a new block for the cutout region: see block no. 10 with colour no. 7 in the picture below middle. The picture below right shows the corresponding 3D image.

Grid

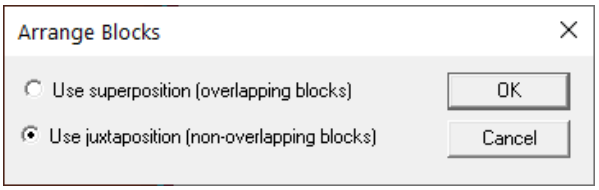
No.	X [cm]	Y [cm]	Z [cm]
Sum	140.000	140.000	133.000
0-1	10.000	10.000	50.000
1-2	10.000	10.000	49.500
2-3	5.000	5.000	0.500
3-4	15.000	15.000	15.000
4-5	50.000	30.000	8.000
5-6	50.000	70.000	10.000

Blocks

No.	Col.	Xmin	Xmax	Ymin	Ymax	Zmin	Zmax
1	170	0	6	0	6	0	6
2	180	1	6	1	6	0	3
3	133	2	6	2	6	0	3
4	133	1	6	1	6	4	5
5	148	3	6	3	6	0	3
6	136	3	4	3	5	0	3
7	136	1	6	1	6	3	4
8	174	4	6	4	6	0	3
9	190	4	6	4	6	2	3
10	7	5	6	1	4	0	1



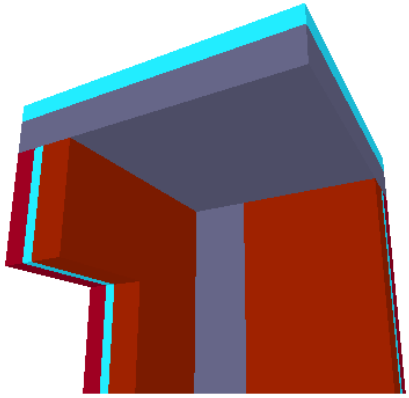
- Use the command *Blocks → Arrange Blocks...* and select "Use juxtaposition" to remove all superposition of blocks (in particular under the new block).



- The blocks are remodelled in the **Blocks window** (with 18 blocks, all adjacent to each other without any superposition), but resulting in the same 3D object (i.e. the **Image window** does not alter).
- Position the active cell in the **Blocks window** in block no. 1 (which was added with colour 7) and delete this block using *Edit → Delete* to clear the wanted cutout. The figures below show the result.

Blocks

No.	Col.	Xmin	Xmax	Ymin	Ymax	Zmin	Zmax
1	7	5	6	1	4	0	1
2	133	1	6	1	6	4	5
3	133	2	3	2	6	0	3
4	133	3	5	2	3	0	3
5	133	5	6	2	3	1	3
6	136	1	6	1	6	3	4
7	136	3	4	3	5	0	3
8	148	4	6	3	4	1	3
9	148	3	4	5	6	0	3
10	148	4	5	3	4	0	1
11	170	0	6	0	1	0	6
12	170	0	1	1	6	0	6
13	170	1	6	1	6	5	6
14	174	4	6	4	6	0	2
15	180	1	2	1	6	0	3
16	180	2	5	1	2	0	3
17	180	5	6	1	2	1	3
18	190	4	6	4	6	2	3

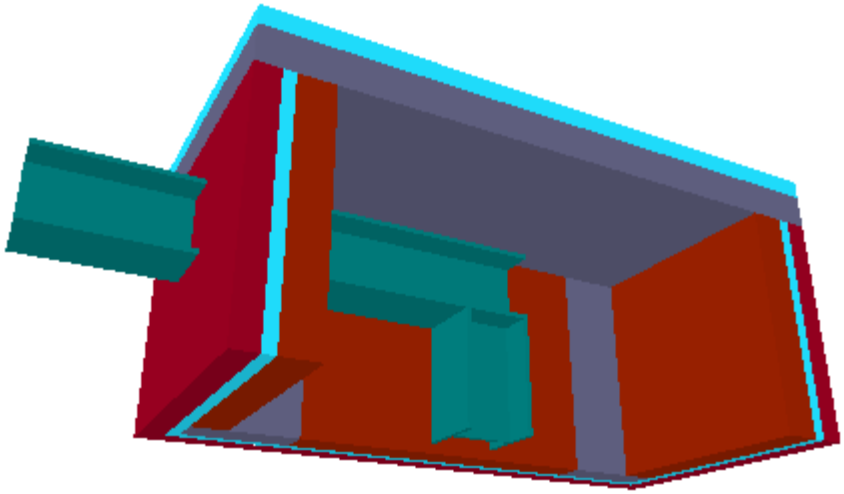


- The space of the cutout is now void, neither containing any boundary condition (and consequently adiabatic). Use the command *View → BCs* to view the boundary conditions active on the object surfaces.

7. Block transformations

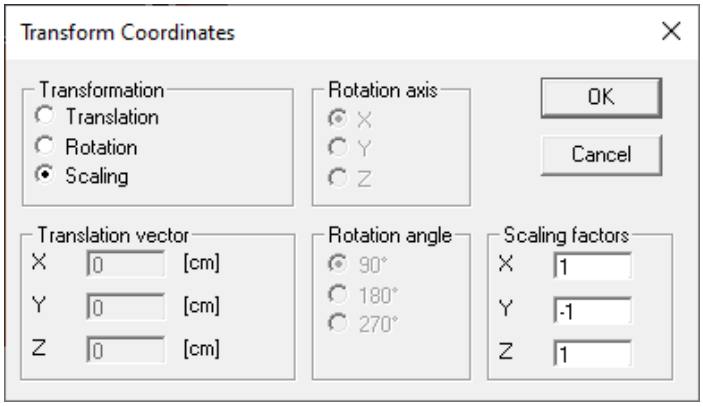
building_corner_d.trc
steelbeam.trc

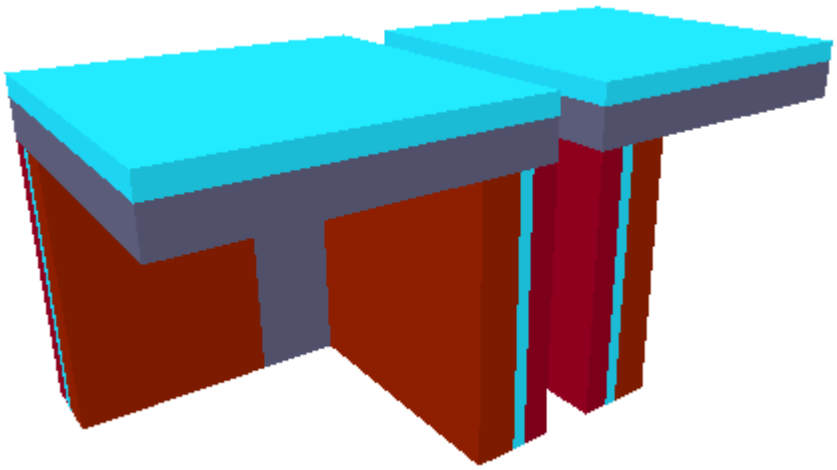
- Starting from the first example in this tutorial (building_corner_a.trc) and another simple object detail (steelbeam.trc) a composite object is constructed to show the possibilities of block transformations and file insertion.



Step 1: Copy and mirror functions

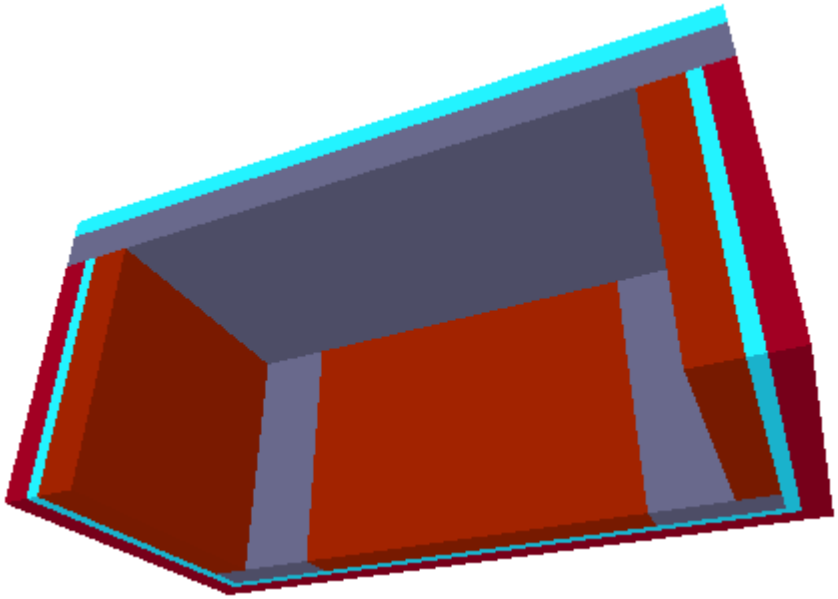
- Open the file building_corner_a.trc from *File → Open...*
- Save under a new file name Compose.trc via *File → Save As...*
- Make the **Blocks window** active by clicking with the left mouse button in this window
- Select all blocks (*Edit → Select All* or via shortcut key <Ctrl+A>).
- Copy the selected blocks (*Blocks → Copy*).
- The copied blocks (no. 10-18) remain selected (marked in turquoise).
- Mirror the selected blocks against the coordinate face XZ:
  Blocks → Transform Coordinates... (or press <Ctrl+F>) with the following parameters:
  - "Transformation": select "scaling".
  - "Scaling factors": define X = 1, Y = -1, Z = 1.
  - This coordinate scaling corresponds to the wanted mirror operation.





Note the grid is changed into a minimum grid (only grid planes through object block borders). The transition to a minimum grid is done automatically after each block operation that affects the grid (*Blocks → Transform Coordinates..., Blocks → Insert File...*).

– Translate the selected blocks to the maximum Y position.
  Blocks → Transform Coordinates... (<Ctrl+F>)
    - "Transformation": select "Translation".
    - "Translation vector": define X = 0, Y = 280 cm, Z = 0.



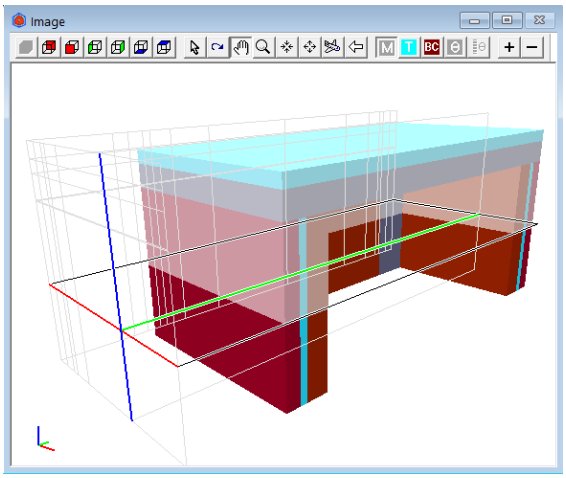
– End the current block selection: *Edit → Deselect All*.

Step 2: Import the steel beam

- The steel beam will be loaded from a separate file which was prepared in advance.
- Set a new coordinate origin as anchor point for the file insertion.
    - Split the grid mesh X 4-5:
    - Set the current grid cell on this grid mesh in the **Grid window**.
    - Then do *Grid → Split...* with the default parameters (Number of subdivisions = 2, Method = geometric, Split ratio = 1).
- Split the grid mesh Z 0-1 into two equal parts using the same method.
- Insert a new mesh before Y 0-1:
    - Set the current grid cell on the mesh Y 0-1.
      Edit → Insert Before.
      Type the new mesh width: 50 (cm).
- Load the **Coordinates window** (with absolute coordinates of grid borders):
  Edit → Coordinates.
- Set the new origin on the X axis:
  Set the current grid cell on coordinate X 5 (= 90.000).
  Execute *Coord → Set Origin.*
  All coordinates on the X axis are moved so that X 5 becomes the new origin (= 0.000).
- Set the new origin on the Z axis equal to Z 1 (= 49.750).
- The **Image window** shows the new origin: i.e. the point where X axis (in red), Y axis (in green) and Z axis (in blue) cross.

Grid

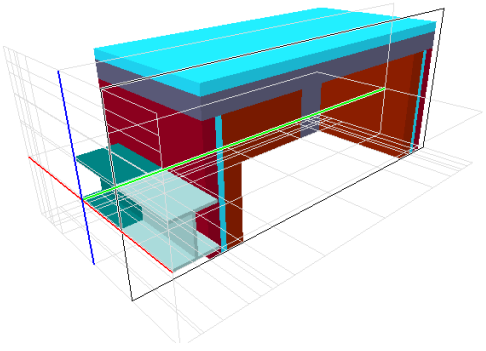
No.	X [cm]	Y [cm]	Z [cm]
Sum	140.000	**330.000**	133.000
0-1	10.000	50.000	49.750
1-2	10.000	10.000	49.750
2-3	5.000	10.000	0.500
3-4	15.000	5.000	15.000
4-5	50.000	15.000	8.000
5-6	50.000	30.000	10.000
6-7		70.000	
7-8		70.000	
8-9		30.000	
9-10		15.000	
10-11		5.000	
11-12		10.000	
12-13		10.000	



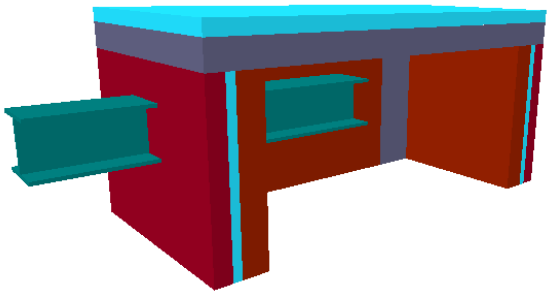
Coordinates

No.	X [cm]	Y [cm]	Z [cm]
0	-90.000	0.000	-49.750
1	-80.000	50.000	0.000
2	-70.000	60.000	49.750
3	-65.000	70.000	50.250
4	-50.000	75.000	65.250
5	0.000	90.000	73.250
6	50.000	120.000	83.250
7		190.000	
8		260.000	
9		290.000	
10		305.000	
11		310.000	
12		320.000	
13		330.000	

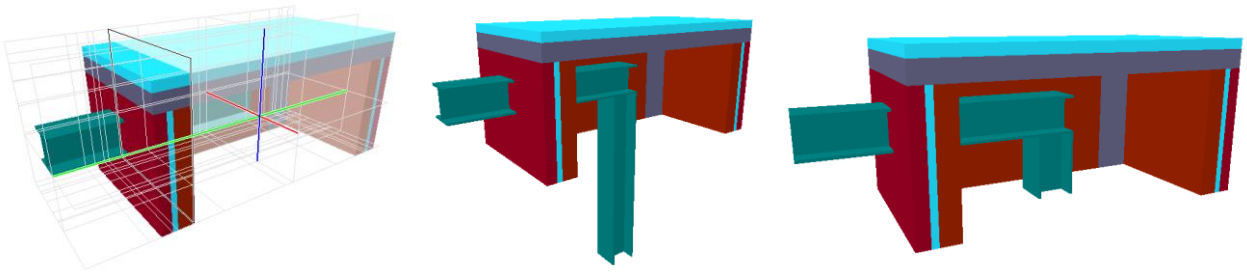
- Import the file steelbeam.trc using *Blocks → Insert File...*
- The blocks in this file are inserted into the **Blocks window** after the current position (here the end of the list of blocks). The inserted blocks are translated to the current origin. The inserted blocks are selected (marked in turquoise). The thermal properties related to colours of the inserted blocks that are not yet present in the current project, are loaded from the inserted file into the **Colours window** (see colour 13).



- Transform the beam into the desired position:
    - Rotate the beam around the current Z axis (in blue).
      Blocks → Transform Coordinates...
      "Transformation": select "rotation".
      "Rotation axis": select "Z".
      "Rotation angle": select "90°".
    - Adjust the length (= 100 cm) of the horizontal beam to be 200 cm.
      Blocks → Transform Coordinates...
      "Transformation": select "scaling".
      "Scaling factors": define X = 1, Y = 2, Z = 1.

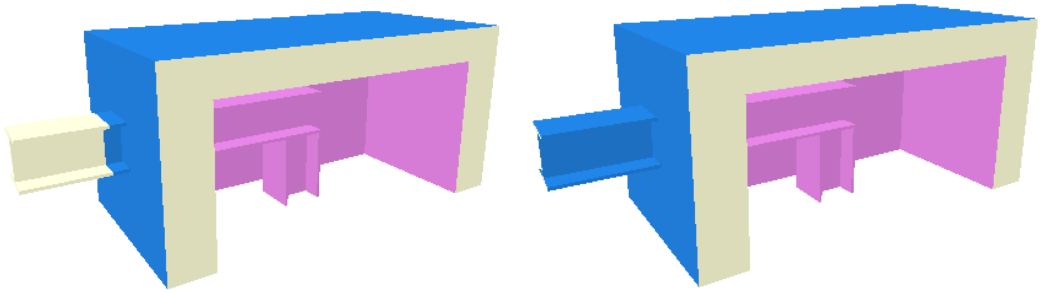


- The vertical beam is set as follows:
    - Move the coordinate origin on the Y axis onto coordinate Y 8 (= 200.00) using *Coord → Set Origin*.
    - Copy the selected blocks (corresponding to the horizontal beam).
      Blocks → Copy.
    - Rotate the selected blocks (which form a copy of the horizontal beam) around the (new) X axis over 90° using *Blocks → Transform Coordinates...*
    - Adjust the length (= 200 cm) of the vertical beam to be 50 cm using coordinate scaling.
      Blocks → Transform Coordinates...
      Transformation = scaling.
      Scaling factors: scale X = 1, scale Y = 1, scale Z = 0.25.
    - End the current block selection: *Edit → Deselect All*.

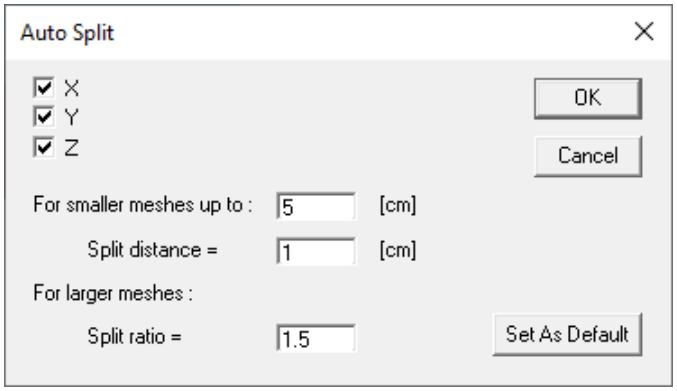


Step 3: System calculation

- Adjust the outside boundary condition block:
- Change Ymin (= 1) of block no. 1 (colour 2) in the Blocks window into Ymin = 0 to cover the exterior part of the beam with outer boundary conditions (colour 170).



- Refine the grid.
  Here a simple grid refinement is used starting from the current minimum grid:
  - Activate Grid window (*Edit → Grid*)
  - Refine grid via automatic split function:
    Grid → Auto Split... with following parameters.



- Solve the system: *Calc → Calc System*.
- The system contains 268 995 nodes and is solved in a few seconds.