



Geometric Construction Steps

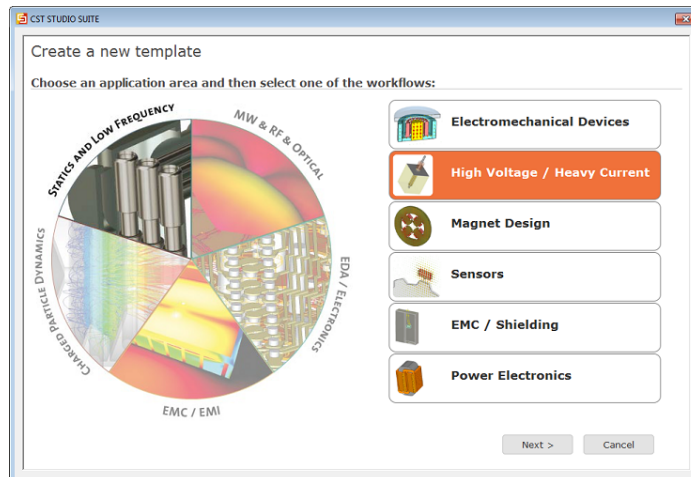
After launching the CST STUDIO SUITE you will enter the start screen showing you a list of recently opened projects and allowing you to specify the application which suits your requirements best. The easiest way to get started is to configure a project template which sets the basic settings that are meaningful for your typical application. Therefore click on the *New Template*  button in the [New Project from Template](#) section.

Next you should chose the application area, which is *Statics and Low Frequency* for the example in this tutorial and then select the workflow *High Voltage / Heavy Current* by double-clicking on the corresponding entry.

Geometric Construction Steps

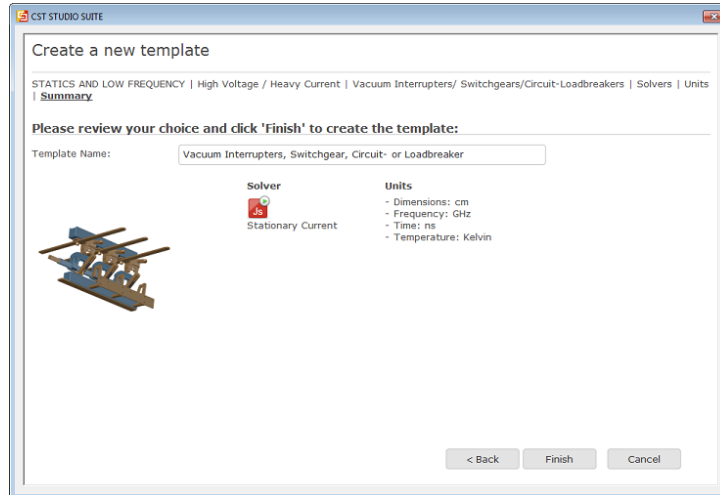
After launching the CST STUDIO SUITE you will enter the start screen showing you a list of recently opened projects and allowing you to specify the application which suits your requirements best. The easiest way to get started is to configure a project template which sets the basic settings that are meaningful for your typical application. Therefore click on the *New Template*  button in the [New Project from Template](#) section.

Next you should chose the application area, which is *Statics and Low Frequency* for the example in this tutorial and then select the workflow *High Voltage / Heavy Current* by double-clicking on the corresponding entry.




For the circuit breaker, please select *High Voltage / Heavy Current > Vacuum Interrupters/Switchgears/Circuit-Loadbreakers > Stationary Current* .

At last you are requested to select the units which fit your application best. The circuit breaker device in this tutorial has dimensions given in cm. Therefore select cm from the *Dimensions* drop-down list if not already set. For the specific application in this tutorial, the other settings can be left unchanged. After clicking the *Next* button, you can give the project template a name and review a summary of your initial settings.




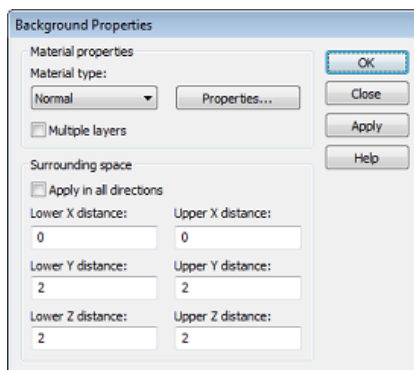
Finally, click the *Finish* button to save the project template and to create a new project with appropriate settings. CST EM STUDIO will be launched automatically due to the choice of the application area Statics and Low Frequency.

Please note: When you click again on *File: New and Recent* you will see that the recently defined template appears below the [Project Templates](#) section. For further projects in the same application area you can simply click on this template entry to launch CST EM STUDIO with useful basic settings. It is not necessary to define a new template each time. You are now able to start the software with reasonable initial settings quickly with just one click on the corresponding template.


Please note: All settings made for a project template can be modified later on during the construction of your model. For example, the units can be modified in the units dialog box ([Home: Settings](#) > [Units](#) ) and the solver type can be selected in the [Home: Simulation](#) > [Setup Solver](#) drop-down list.

Define background properties

Once you have entered the modeling view, the background properties should be adjusted. Because the structure will be defined in a vacuum background, you must define the size of the surrounding empty space. Therefore, select [Modeling: Materials](#) > [Background](#)  to open the *Background Properties* dialog box. The *Material type* has been set to *Normal* already by the project template. Please deactivate the *Apply in all directions* checkbox, enter 0 cm for *Lower X* and *Upper X* distance as well as 2 cm for the *Lower Y*, *Upper Y*, *Lower Z* and *Upper Z* distance. No space is added for X-distances because the circuit breaker needs to touch the boundaries in order to apply the current port sources correctly in a later step, where the current distribution is used as source for a magnetostatic simulation. Now confirm by pressing the *OK* button.



Define the right contact

The first step for the circuit breaker is the construction of the right contact which uses the trace function of a curve. Therefore, define the a line ([Modeling: Curves > Curves > Line](#) ) with

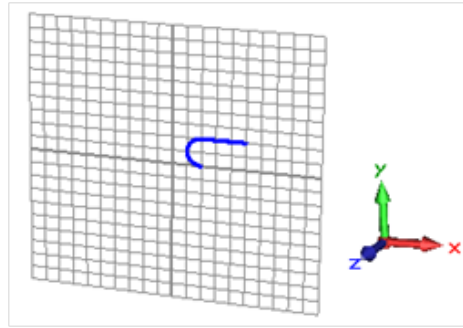
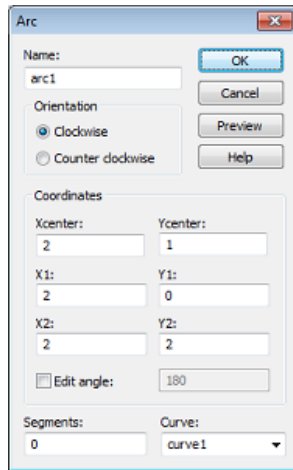
1. first point at $X = 5, Y = 2$ and
2. second point at $X = 2, Y = 2$.


You can do this by clicking at the corresponding coordinates in the main view or by pressing the *Tab* or *Esc* key and enter the coordinates numerically.

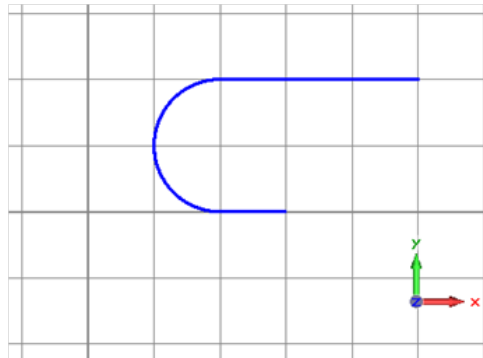
The next part of the curve is an arc. Select [Modeling: Curves > Curves > Arc](#)  and define


1. the arc center at $X = 2, Y = 1$;
2. start point at $X1 = 2, Y1 = 0$;
3. end point at $X2 = 2, Y2 = 2$.
4. deactivate the "Edit angle" field.

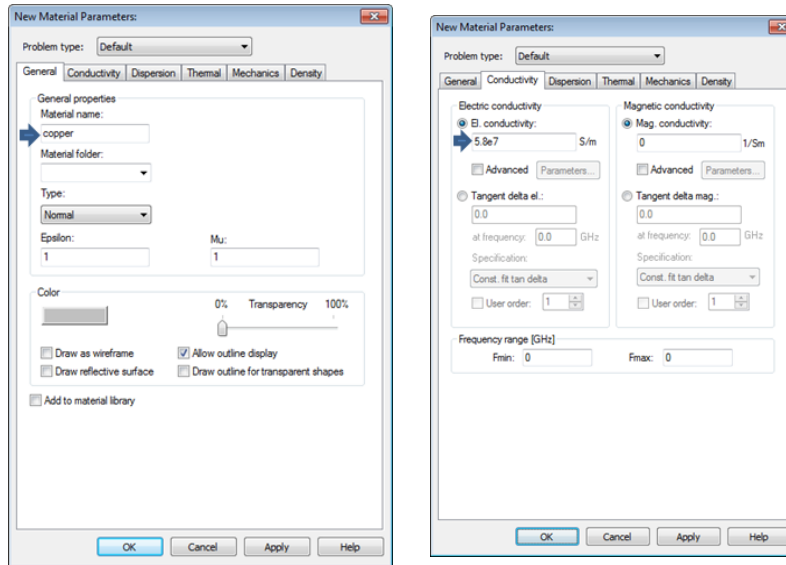
Check your settings against the screenshots below and verify that the arc is oriented correctly (otherwise you can change the arc's orientation in the dialog box). Finally, confirm with *OK*.



The last curve element is another line and can be created similar to the first one. Activate the curve line tool ([Modeling: Curves > Curves > Line](#) ) and enter the points $X = 2, Y = 0$ and $X = 3, Y = 0$. The screenshot below shows the current model status:



Finally, the curve has to be transformed into a solid to model the contact. The right tool for this procedure is the *Trace from Curve* tool. Select [Modeling: Curves > Curve Tools > Trace from Curve](#)  and double-click on any curve element when asked for the curve. Before you confirm the trace creation, a new material must be defined. Select *[New Material...]* from the *Material* drop down list in the trace definition dialog box and define the material copper. This requires changing the *Material Name* to copper, selecting a new color and inputting an electric conductivity of $5.8e7 S/m$ in the *Conductivity* tab.

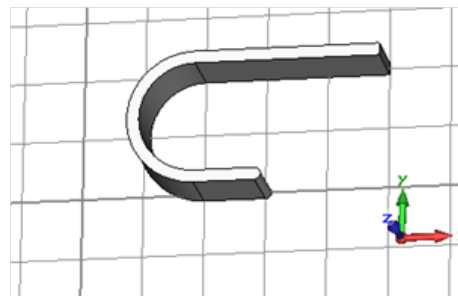
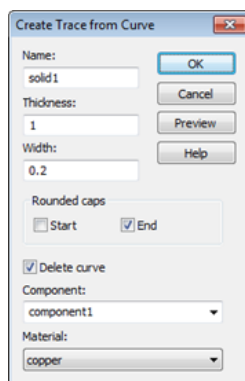


Please note: The defined material copper will be available inside the current project for the further creation of other solids. However, if you want to save this specific material definition for use in other projects, you may check the button *Add to material library*. You will have access to this material database by clicking on *Load from Material Library* in the Materials context menu in the navigation tree.

Pressing the OK button stores the material and closes the dialog box so that you can continue to enter the trace parameters. Back in the trace creation dialog box, please check that the *Material* has changed to copper. Enter *contact spring* in the *Name* field. Furthermore,

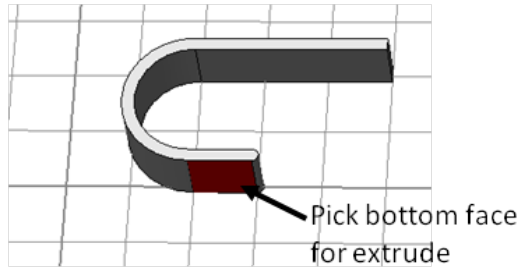
1. define *Thickness* = 1, *Width* = 0.2
2. and *Rounded Caps* at the *End*.



Please note that the end of the curve depends on which part of the curve you chose for the trace action. This fact must be considered in the construction, so you should ensure that the preview of the shape has a rounded cap at the shorter end. If not, it is only necessary to switch off the *Rounded caps* at the *End* and enable them at the *Start* of the curve.



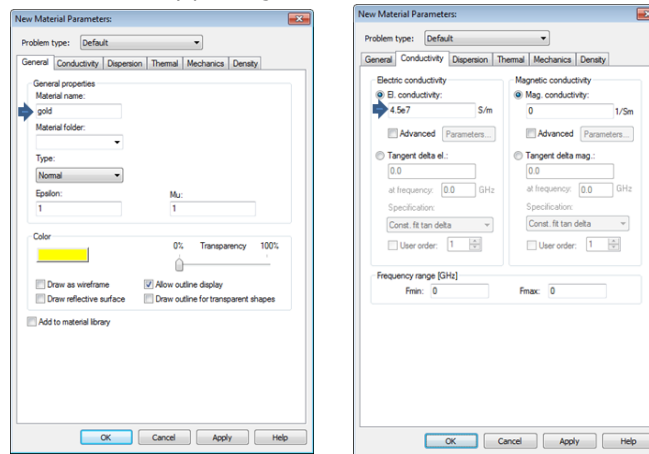
Define the contact pad

To complete the construction of the right contact, the definition of a contact pad is still required. This task is easily done by extruding the bottom face of the spring.

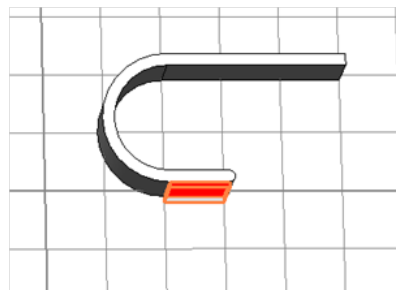
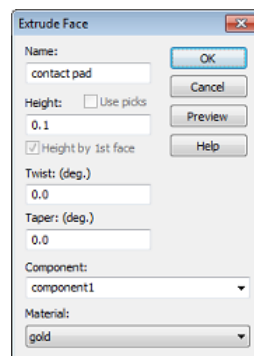


To perform the extrusion, the desired face must be chosen beforehand. Therefore, activate the pick face tool with the shortcut *f* or via **Modeling: Picks > Picks**  and double-click on the bottom face of the spring. Then select **Modeling: Shapes > Extrude**  that opens a dialog box for the extrusion parameters. First, a new material has to be defined because the contact pad is composed of gold. Choose *[New Material...]* from the *Material* drop down list. In the *New Material Parameters* dialog box

1. change the *Material name* to gold,
2. set the *Type* to *Normal*,
3. select a new color,
4. input an *Electric conductivity* of $4.5e7$ S/m in the *Conductivity* Tab,
5. and store the material by pressing the *OK* button.




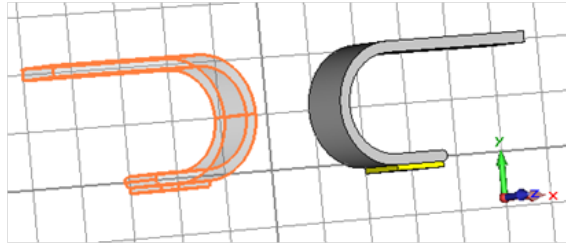
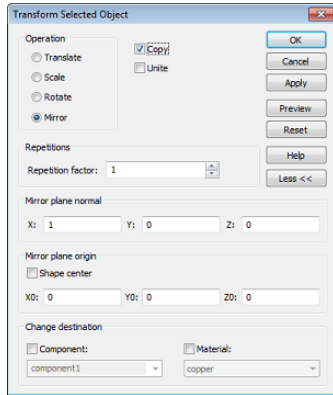
Back in the *Extrude Face* dialog box, enter a name for the solid (*contact pad*) and 0.1 for the *Height*; the other parameters can be left at their default values. Verify your settings by pressing the *Preview* button and finally close the dialog box by pressing the *OK* button.



Generation of the left contact by transformation

Due to the symmetry of the structure, the left contact can be produced by just one mirror operation. Therefore, select *component1* (which contains the contact pad as well as the contact spring) in the

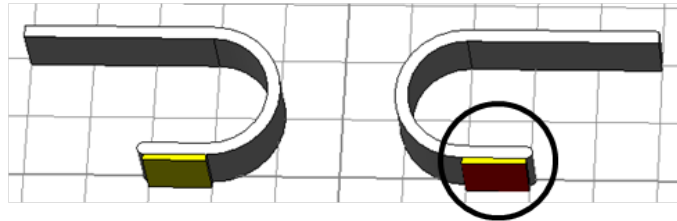
navigation tree and choose *Transform* from the context menu or from the ribbon **Modeling: Tools > Transform** . In order to mirror the structure correctly, select *Mirror* as *Operation*, $X = 1$, $Y = 0$ and $Z = 0$ as the *Mirror plane normal* and activate the *Copy* option:





After confirming with *OK*, two new shapes will appear in the navigation tree named as the transformation originals with *_1* added.

Define the pads of the bridge

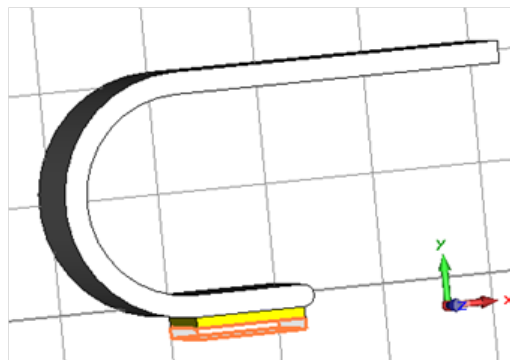
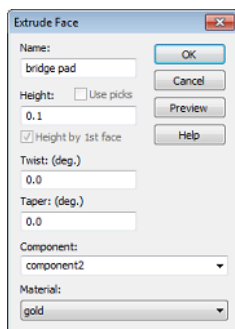
After modeling the contact, the pads of the bridge must be constructed. Therefore, pick the previously defined pad (contact pad) of the contact at the bottom and extrude it again.



First, select **Modeling: Picks > Picks**  (shortcut: *f*) and double-click on the bottom face. Then activate the extrude tool (**Modeling: Shapes > Extrude** ) and enter

1. a *Name* (bridge pad),
2. a value of *0.1* for the *Height* and
3. choose [*New Component...*] from the *Component* drop down list.

In view of the fact that the bridge is physically independent, it is preferable to construct it as a new component; this makes it subsequently easier to manipulate, e.g. for transformations.



Again, check your settings and confirm with *OK*.

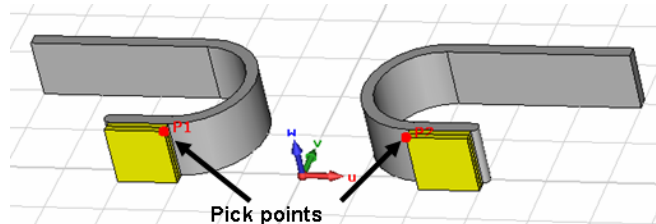
Transform the pad



As previously carried out for the contact, the bridge's pad on the left side can be created with a mirror operation. This requires the newly created pad to be selected by double-clicking on it. Then select *Transform* from the context menu to start the transformation and repeat the previously made steps:




1. Select *Mirror as Operation*,
2. set the *Mirror plane normal* to $X=1, Y=0, Z=0$ and
3. activate the *Copy* option.


Construction of the bridge

The last modeling step is the construction of the bridge. Up to this point, the construction has been performed in the global coordinate system, because no relative construction was needed. For proper positioning, it is advisable to define the bridge using a working coordinate system (WCS) that introduces a local coordinate system. For this tutorial, the WCS will be aligned in the middle of the bridge pads.



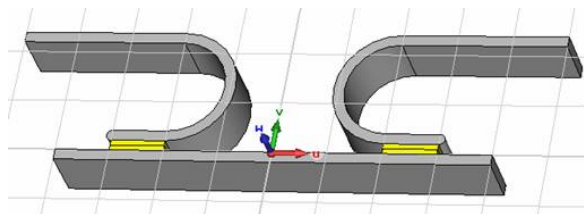
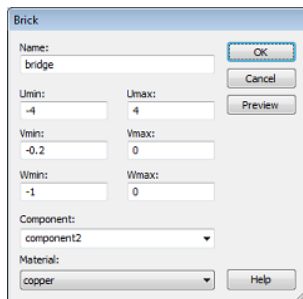
The working coordinate system can be switched on or off by selecting **Modeling: WCS > Local WCS** . It can be aligned with a point (or any other object) with *Align WCS* (shortcut: *w*) which can be found under **Modeling: WCS > Align WCS**  as well. For the bridge construction,

1. activate the WCS (**Modeling: WCS > Local WCS** ) ,
2. pick the two illustrated points (**Modeling: Picks > Picks** , shortcut: *p*),
3. generate a new point with **Modeling: Picks > Pick Point > Mean Last Two Points**
4. and align the WCS to this point (**Modeling: WCS > Align WCS** ).

With the WCS in the right location it is now quite easy to define the bridge. Therefore, select **Modeling: Shapes > Brick**  to activate the *Brick* tool and use the Tab key or double-click in the working plane to enter

1. the first point at $U = 4, V = 0$,
2. the second point at $U = -4, V = -0.2$,
3. and -1 for the height.

In the *Brick* dialog box you only have to enter a proper *Name*, e.g. *bridge*, and choose *copper* from the *Material* drop-down list.



Now, the geometric modeling is complete and the solver settings have to be established.

Solver Settings

Define Boundary Conditions and Symmetry Settings

The calculation domain is bounded by six faces. A boundary condition must be assigned to each of these faces. Open the *Boundary Conditions* dialog box by selecting **Simulation: Settings > Boundaries**. For this example, all boundaries need to be of type *tangential*. This fact is important for the subsequent magnetostatic calculation, because it means that the magnetic flux has only tangential (i.e. no normal) components at the boundaries. For the stationary current problem, the current ports force the boundary conditions locally to be normal, which means that the current field has only normal components at the current ports. Thus activate the *Apply in all directions* checkbox and choose *tangential e, h or j* from the *Type* drop-down list.

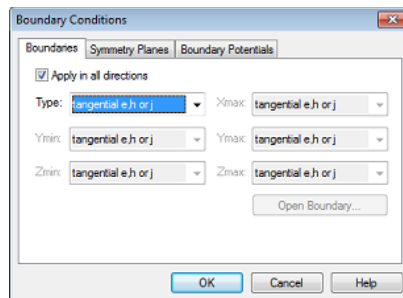
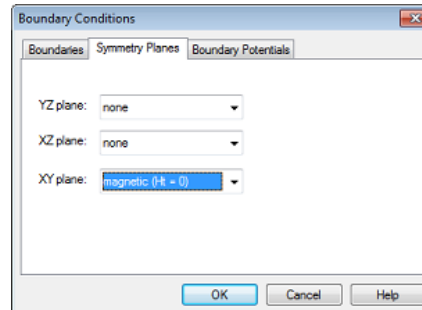
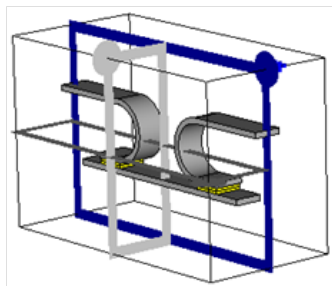


figure 3

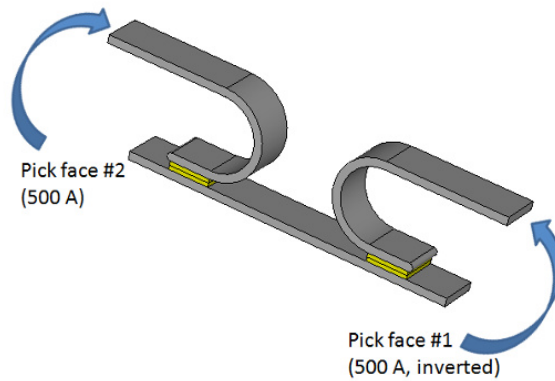
Next, you may define a symmetry condition for the *XY* plane. This will reduce the problem size by a factor of 2, so symmetry conditions should always be defined, if possible. Because currents and electric fields will be tangential and magnetic fields will be normal to the *XY* plane, you should select a magnetic symmetry plane inside the *Symmetry Planes* tab. Finally confirm with *OK*.



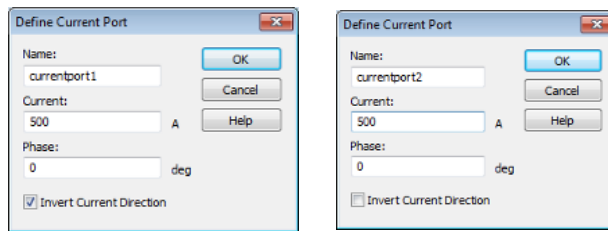
Please note: In particular, for coupled problems between the stationary current solver and the magnetostatic solver, normal and tangential symmetry planes are not recommended. Instead, the electric and magnetic symmetry planes should be used. They correspond to a Perfect Electric Conductor (PEC) for the electric boundary condition and a Perfect Magnetic Conductor (PMC) in case of the magnetic boundary condition.

Definition of the current ports

Before the solver can be started, the driving sources need to be defined. In this tutorial, two current ports on the faces of the contact springs are necessary to feed the structure with a homogeneous distributed current density. Enter the current port definition mode (**Simulation: Sources and Loads > Current Port**) and click on the right face at the upper X-boundary (face#1).



Once the position is defined via the face pick, the dialog opens and the current value of the port is requested. Enter the value *500* for the *Current* of the first port, check the *Invert current direction* box and confirm with *OK*. Then enter the current port definition mode again and pick face #2 at the lower X-boundary in order to define the second port. Enter the same *Current* value of *500* for this port, but keep the current direction unchanged:



Both faces are highlighted with colors corresponding to the current values, when the *Current ports* folder in the navigation tree is selected. The arrows represent the current direction.

Adjust Mesh Properties

Before starting the solver, let's have a look at the mesh used for the calculation. In this tutorial we will employ the tetrahedral solver. Please note that the steps to employ the hexahedral solver are completely identical. To keep this tutorial short, we will restrict the workflow description of the coupled simulation to the tetrahedral solver and refer to the CST EM STUDIO *Workflow & Solver Overview* manual for more details on the hexahedral solver.

By default, the tetrahedral mesh type is selected. So open the *Mesh Properties - Tetrahedral* dialog by clicking **Simulation: Mesh > Global Properties** . The project template should have set the *Cells per max model box edge* values to 10 for the *Model* and 5 for the *Background*. Click *OK* to confirm the *Tetrahedral* mesh settings and close the dialog box. Now activate the mesh view via **Simulation: Mesh > Mesh View** and generate the mesh by clicking **Mesh: Mesh Control > Update** . Apparently, the default mesh is quite coarse:

